
STRAP 2021
Structural Analysis Programs
©2021 ATIR Engineering Software Development

STRAP

Disclaimer

The STRAP program has been written by a team of highly qualified engineers and programmers and have been extensively tested. Nevertheless, the authors of the software do not assume responsibility for the validity of the results obtained from the programs or for the accuracy of this documentation

The user must verify his own results

The authors remind the user that the programs are to be used as a tool for structural design and analysis, and that the engineering judgement of the user is the final arbiter in the development of a suitable model and the interpretation of the results.

Special thanks to:

All the people who contributed to this document, the programmers, secretaries, STRAP dealers and users.

Last not least, we want to thank EC Software who wrote the help tool called HELP & MANUAL that was used to create this document.

Table of Contents

Part I

General options

1 General options

1.1 General

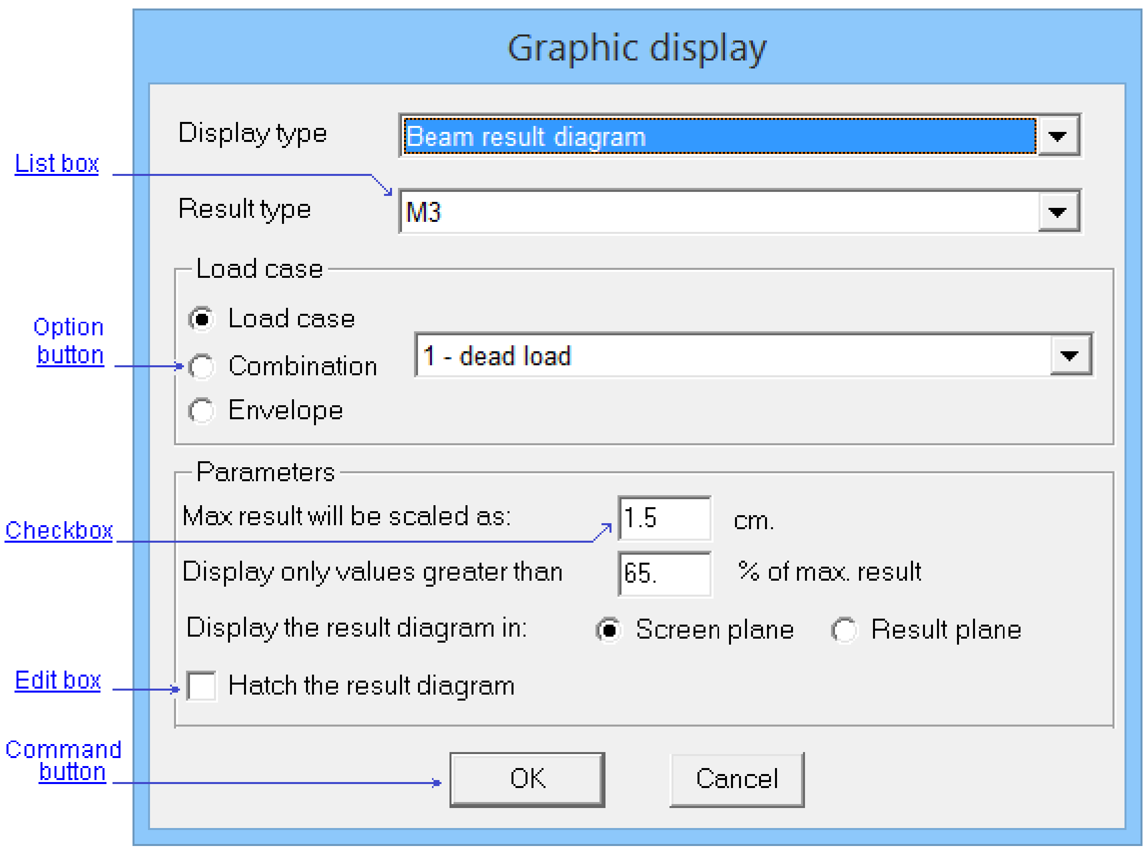
Dialog boxes 18 - an overview of "Windows" dialog box conventions.
Shortcut menus 21 (right-click on object)
Side menus 21
Help 21 - overview of the **Help** option
Step 20 option

1.1.1 Dialog boxes

STRAP uses Windows "Dialog Boxes" to request data and to provide information. Most dialog boxes contain options, each one asking for a different type of information. After all required information is supplied, a "Command Button" is pressed to carry out the command and return the program to the previous menu or dialog box.

Dialog boxes also provide information, warnings and error messages.

A typical "Windows" dialog box contains several types of standard option styles. For example:



Check box

Check boxes represent options that can be turned on or off.

When the option in the check box is turned on, the box is displayed as ☑. In our example, the result diagrams will be hatched.

To revise the option, place the arrow on the check box and click the mouse button.

Command button

Command buttons initiate an immediate action. In the dialog box above,

OK displays the graphic results according to all other options selected in the dialog box.

Cancel closes the dialog box; the option is canceled and all options are returned to the values present when the dialog box was entered

List box

A list box displays a column of available choices. A "drop-down" list box displays only the current option along with an arrow at the end of the line. For example, the "Result type" option in the above dialog box is displayed as:

Element results contour map

When the arrow button is pressed, the list box opens up to:

Element results contour map
Geometry
Results at element centers
Element results contour map
Element results along a line
Displacements

The current option is highlighted.

Option button

Option buttons represent a list of mutually exclusive options, i.e. only one of them can be selected. In our dialog box example, we can select to display the results for either a load case, load combination or an envelope of cases/combinations.

The selected option is highlighted by a black dot in the button ◉. In our example, the program will display the result for a load case.

To select a new option, place the arrow on the option you want and click the mouse button.

Text box

A text box is a rectangle into which you can type information.

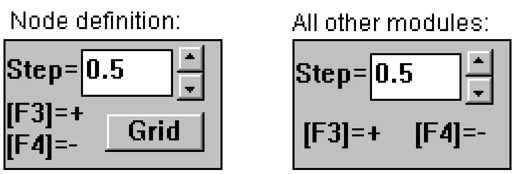
In the dialog box above, the number of contour lines is defined in a text box as 12.

To revise text in a text box:
- use the mouse to move the arrow to the text box and click the left mouse button
- edit the text in the box.

Please note the following ***STRAP*** conventions:
Exponential format
List format

1.1.2 Step

The **Step** is the distance the moves on the screen each time the mouse is moved slightly or an arrow key is pressed. The current **Step** value is continuously displayed at the lower-right corner of the screen.



There are three methods available for revising the Step:

- Move the into the **Step=** text box. Type in the correct value.
- Press to increase the Step or to decrease the Step.
- Click the buttons.

For node definition only:

Click the button to display a grid of dots on the screen; the dots will be spaced at the current ***Step*** interval in both screen directions. The jumps from dot to dot when the mouse is moved or when an arrow key is pressed.

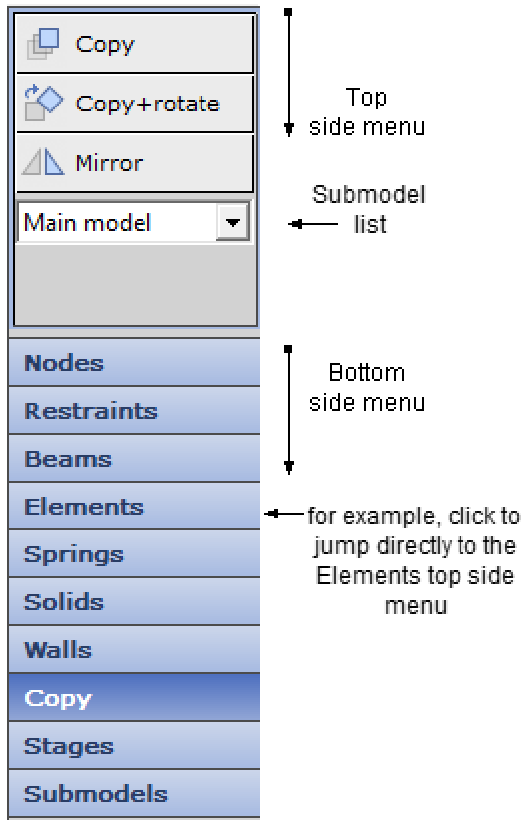
- To display the ***Grid*** automatically every time a node definition option is selected, refer to Setup - Miscellaneous
- The grid is automatically updated every time the Step value is revised.
- If the step value is small relative to the screen dimensions, the density of the dots will be too great. In such cases the program will display the dots with a spacing equal to a multiple of the Step value; several clicks of the arrow key will be required to move from a dot to the adjacent dot.
- A cylindrical grid will be displayed when a cylindrical coordinate system is in effect (note that the mouse can be used for a cylindrical coordinate system only when the Grid is displayed).
- When a working plane not parallel to the screen is in effect and a Grid is displayed, horizontal/vertical motion of the mouse moves the horizontally/vertically on the screen (the moves parallel to the working plane axes when the Grid is not displayed).

1.1.3 Side menus

The main options for any module are found in the "bottom side menu"; the secondary options for each of these are found in the "top side menu".

If submodels have been defined, a list of them is displayed between the two side menus. Selecting a submodel from the list will immediately display the geometry for that submodel.

For example, the 'Copy' options in the geometry module:



1.1.4 Help

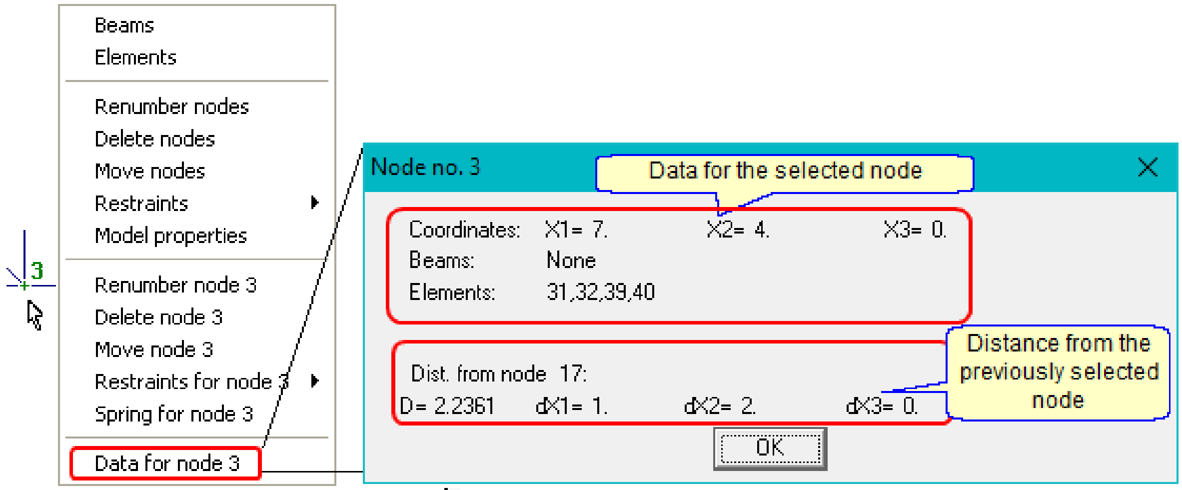
Select **Help** at any time from the menu bar at the top of the screen to display the section of the user's manual pertaining to the current option.

The **Help** is arranged in the standard "Windows" help format; you may jump to any other part of the manual by selecting the **Contents** or **Index** options.

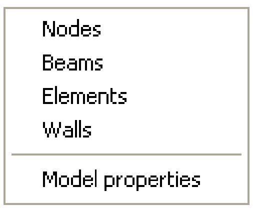
1.1.5 Shortcut menus (right-click)

A Shortcut Menu is a useful pop-up menu that provides options specific to a certain object in the model. ***Right-click*** an object (such as a node or element) to display the Shortcut Menu for that particular object.

- ***Geometry:***
  Right click on nodes, beams or elements. For example, when in Node definition a right-click on a node displays a Shortcut Menu that lets you select all node related options or display data for the selected node.



If the mouse is right-clicked when the cursor is outside the model boundaries, the following menu is displayed:

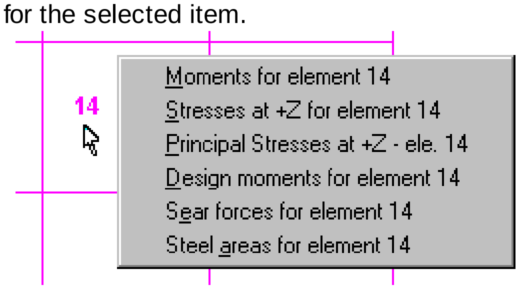


- Model properties
  Weight of displayed elements:
  Center-of-gravity for displayed part:
  - main model is displayed: total weight and center-of gravity are shown for the entire model (including all submodels)
  - submodel is displayed: total weight and center-of gravity are shown only for the current submodel.

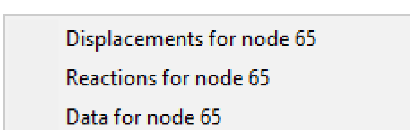
  The calculation includes only elements with **all** nodes displayed on the screen.

- ***Loads:***
  a right-click on a beam displays a Shortcut Menu with beam load definition options, etc.

- ***Results:***
  A right-click on a beam/element displays a Shortcut Menu with options for displaying tabular results for the selected item.



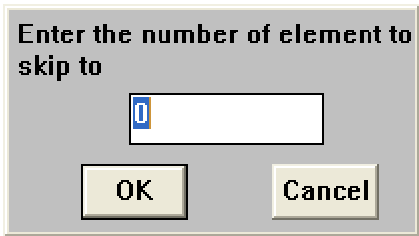
Right-click on a node to display node deflections, reaction and node data (similar to Geometry, above):



1.1.6 Table options

The following toolbar is displayed in all tables:

GoTo



Enter the node/beam/element number; the selected number is then displayed at the top of the table.

Print

Refer to Print tables 92

Copy

Copy the current table to the 'clipboard'; the table may then be pasted into Word, Notepad or Excel documents.

1.1.7 Exponential format

Decimal values may also be entered exponentially. For example:

- **510** may be entered as **5.1E2** or **5.1E+2**
- **0.0037** may be entered as **.37E-2** or **3.7E-3**

Do not leave any blank spaces between the numbers and the letter E.

1.1.8 List format

A series of node or element numbers may be entered in "list" format, where the keywords **TO** and **BY** may be used to simplify the list.

list examples:

1 9 17 20
1 3 TO 6 12 15 18 TO 30
3 TO 11 BY 2 20 TO 24 34

The last example is equivalent to entering:

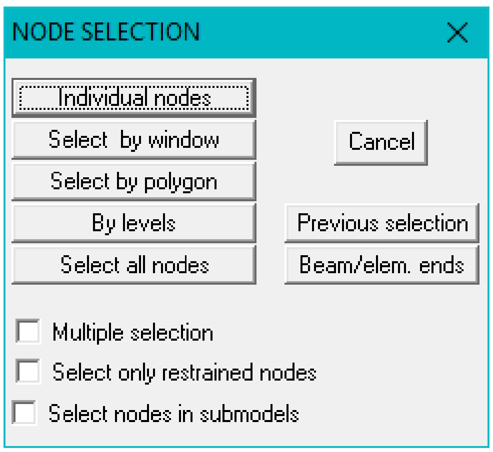
3 5 7 9 11 20 21 22 23 24 34

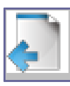
A list can consist of up to 50 items, where " **1 TO 6** " is one item.

1.2 Selection options

1.2.1 Node selection

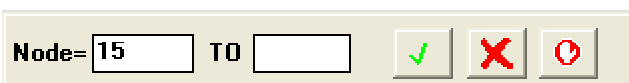
Many options include instructions to select one or more nodes.





Select individual nodes

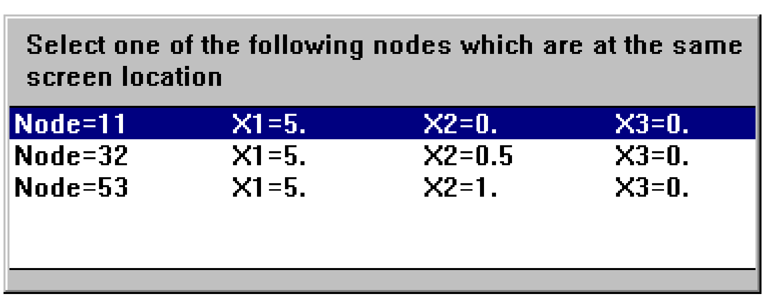
Select individual nodes by moving the cursor alongside each node until it is highlighted by the ■ rectangular blip ; click the mouse. The number of the highlighted node is always displayed at the left-hand side of the Dialog box:



You may also type in the number of the node to be selected, in the form of a "list".

When all the nodes have been selected, press  or click the mouse without moving the cursor.

In space models, more than one node may be at the same **screen** location (the coordinate perpendicular to the screen of these nodes is not identical). In such a case, the program will display a list of nodes and request the user to select one.

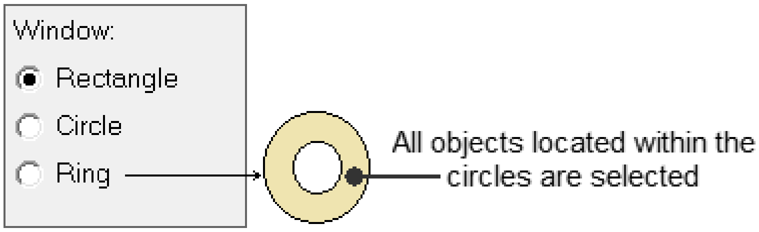


Move the cursor to the line with the correct node and click the mouse.

Select by window

Define a window; the program automatically identifies all nodes located in the window.

The window may be either a rectangle, a circle or a ring.



A rectangle is defined by pointing to its lower-left and upper-right corners with the .

In space models, more than one node may be at the same ***screen*** location and so will be "hidden" from the viewer. In such a case, the program selects ***all*** of the nodes at that location.

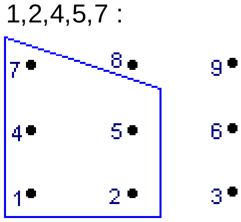
Select by polygon

Define a polygon by pointing to its corners with the . The program automatically identifies all nodes located in the polygon.

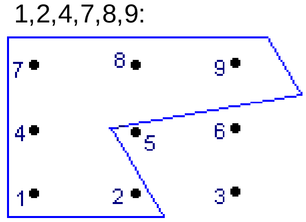
The polygon is constructed as a 'rubber-band' stretched around the defined corners:
- At least three corners must be selected.
- the program automatically connects the last corner defined to the first corner defined.
- click the right mouse button or press [Esc] to delete the previous corner.
- to end the polygon definition, click the mouse without moving the .
- In space models, if more than one node is at the same ***screen*** location, the program selects ***all*** of the nodes at that location.

Examples: select nodes -

1,2,4,5,7 :

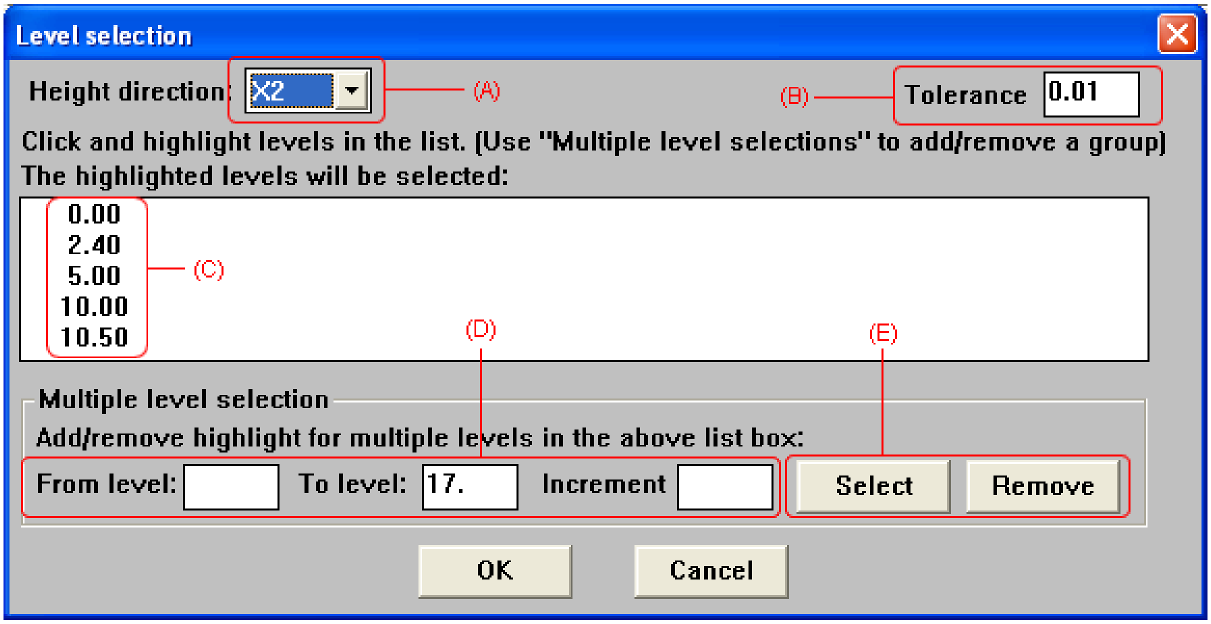


1,2,4,7,8,9:



Select by levels

Select all nodes at selected levels (coordinates):



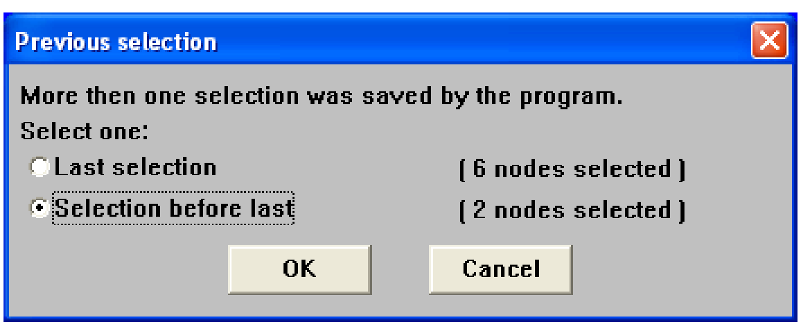
- The program displays a list of the levels parallel to one of the global axes **(A)**. A different global axis may be selected in the Height direction list box **(B)**.
- Select levels by clicking and highlighting them **(A)**.
- Equally spaced levels may be selected by entering the coordinate of the start level, end level and increment in edit boxes **(D)**. Click Select **(E)** to highlight these levels in the list or Remove **(E)** to remove the highlight.
- Click OK; the program will identify all nodes at the selected levels

Note:
- All nodes within the ± tolerance distance **(C)** are selected.

Previous selection

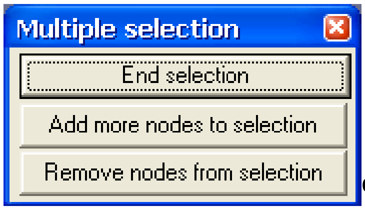
The program remembers the list of selected nodes every time nodes are selected. The nodes from the previous selection are highlighted and the option continues according to **Multiple selection.**

Note:
- the program remembers up to 5 previous selections and you may select any of them. For example:



Multiple selection

Turn on the ☑ checkbox if you want to define several windows, polygons, etc, at the same time. After every window, etc., the following menu is displayed; select:



- Continue without selecting more nodes

- Define another window, polygon, etc. for the same command

- Delete members from the list, window, etc. already defined for this command

Restrained nodes only

☑ only nodes with existing restraints may be selected (does not include nodes with springs or rigid links).

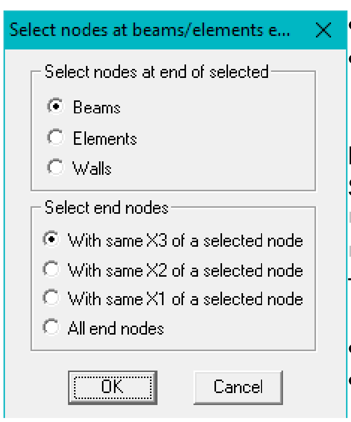
Submodels (results only)

This option is available in results when the Main model is displayed:

☑ the program allows you to select nodes in the main model and the submodels
☐ only nodes defined in the main model may be selected.

Beam/element ends

Select the end nodes of beams/elements/walls:



- Select the beams/elements/walls using the standard selection options.
- If you selected one of the ◉ **With the same Xn of a selected node** , select a single node in the display.

Example: define restraints at the base of columns (all at the same level).
Select -
◉ **Beams**
◉ **With the same X3 of a selected node**
Then -

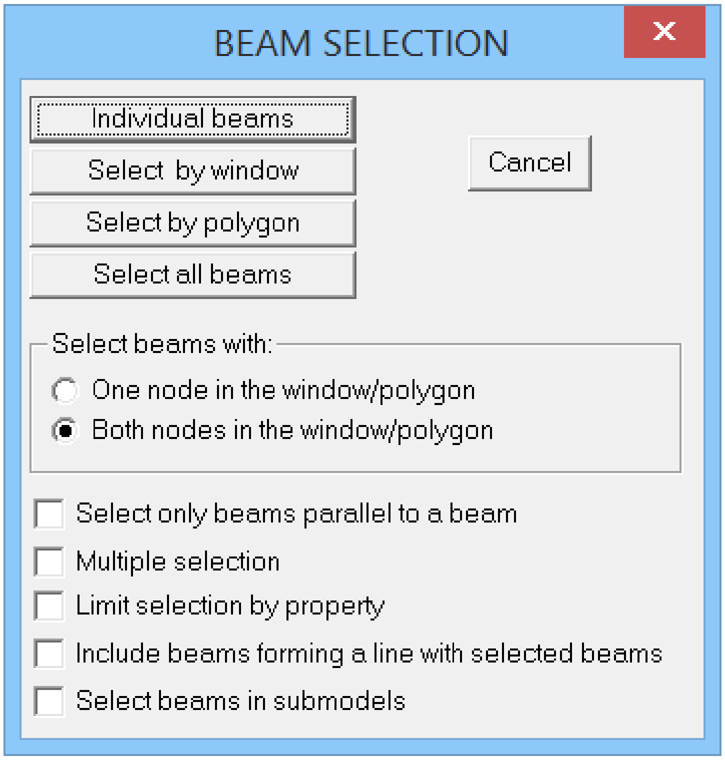
- Select all beams
- select the node at the base of one of the columns

1.2.2 Beam selection

Many options include instructions to select one or more beams.

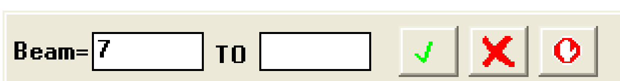
When selecting a beam/element, the beam nearest to the cursor is highlighted with a rectangular blip ■. The number of the highlighted beam is always displayed at the left-hand side of the Dialog box.

You may also type in the number of the beam to be selected.



Individual beams

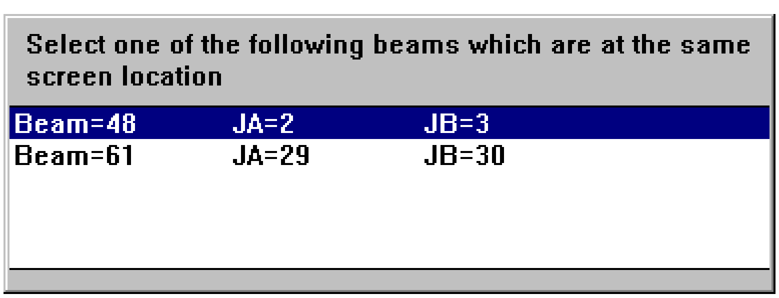
Select a single beam only by moving the alongside the beam until it is highlighted by a rectangular blip ■; click the mouse. The number of the highlighted beam is always displayed at the left-hand side of the Dialog box:



You may also type in the number of the beam to be selected, in the form of a "list".

When all the beams have been selected, press or click the mouse without moving the .

In space models, more than one beam may be at the same screen location (only the coordinate perpendicular to the screen is not identical). In such a case, the program displays a list of beams at that location:

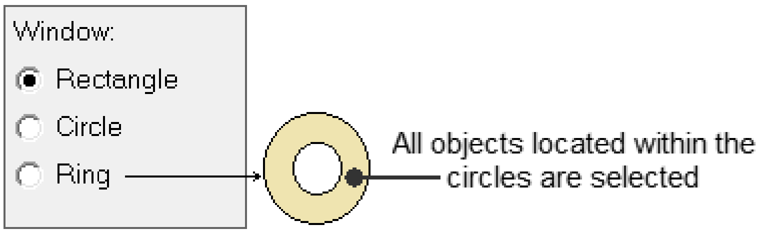


Highlight the line with the correct beam and click the mouse.

Select by window

Define a window; the program automatically identifies all beams with either one or all nodes located in the window (refer to Select beams with 30)

The window may be either a rectangle, a circle or a ring.



A rectangle is defined by pointing to its lower-left and upper-right corners with the .

In space models, more than one beam may be at the same screen location (only the coordinate perpendicular to the screen is not identical). In such a case, the program selects **all** of the beams at that location.

Select by polygon

Define a polygon by pointing to its corners with the . The program automatically identifies all beams with either one or all nodes located in the polygon (refer to Select beams with 30)

The polygon is constructed as a 'rubber-band' stretched around the defined corners:

- At least three corners must be selected.
- the program automatically connects the last corner defined to the first corner defined.
- press [Esc] (right mouse button) to delete the previous corner.
- to end the polygon definition, click the mouse without moving the
- In space models, more than one beam may be at the same screen location (only the coordinate perpendicular to the screen is not identical). In such a case, the program selects ***all*** of the beams at that location.

Select all beams

All beams in the model will be selected.

Note:

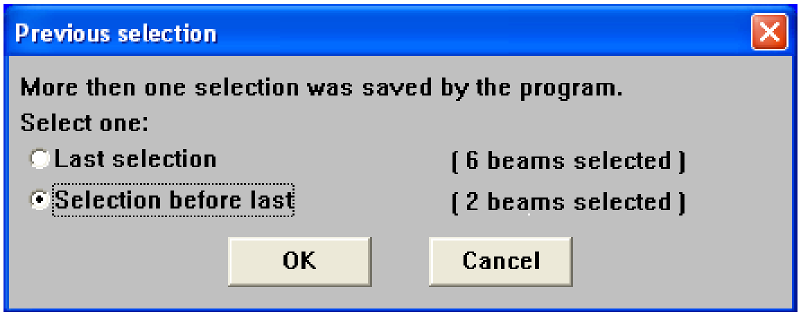
- beams not displayed because of the the **Zoom** option ***are*** selected.
- beams not displayed because of the **Remove beams/elements** option are ***not*** selected.

Previous selection

The program remembers the list of selected beams every time beams are selected. The beams from the previous selection are highlighted and the option continues according to **Multiple selection.**

Note:

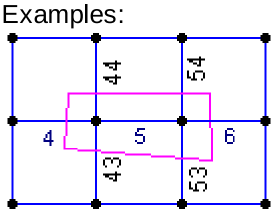
- the program remembers up to 5 previous selections and you may select any of them. For example:

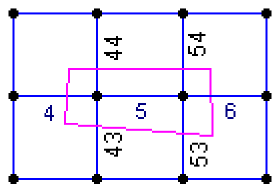


Select beams with

- **One node in the window/polygon**:
  all beams with at least one end node in the window/polygon are selected.
- **Both nodes in the window/polygon**:
  only beams with ***both*** end nodes in the window/polygon are selected.

Examples:





All nodes:
Beam 5 only is selected

One node:
all numbered beams are selected

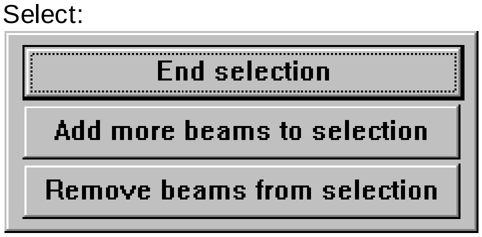
Select only beams parallel to a beam

You may impose a further limit that the beams selected are only those parallel to a specified beam. This option allows you, for example, to define a window around an entire frame but to select only the beams or columns.

Multiple selection

Turn on the checkbox if you want to define several windows, polygons or lists at the same time. After every window, etc., the following menu is displayed:

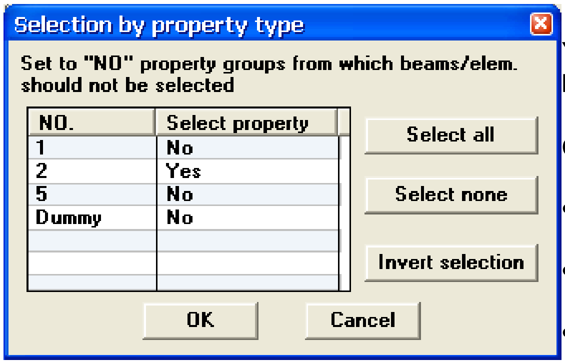
Select:



- Continue without selecting more beams
- Define another window, polygon, etc. for the same command.
- Delete members from the list, window, etc. already defined for this command.

Limit by properties

Further limit the beam selection according to property groups:



Yes: Beams in the property group are selected
No: Beams in the property group are ***not*** selected

Click:

- Select all to set all properties to **Yes**
- Select none to set all properties to **No**
- Invert selection to toggle the selection for all properties

Note:

- Dummy beams may be selected.

Include beams in a line ....

All beams forming a chain on the same line as a selected beam are also selected

Submodels (results only)

This option is available in results when the Main model is displayed:

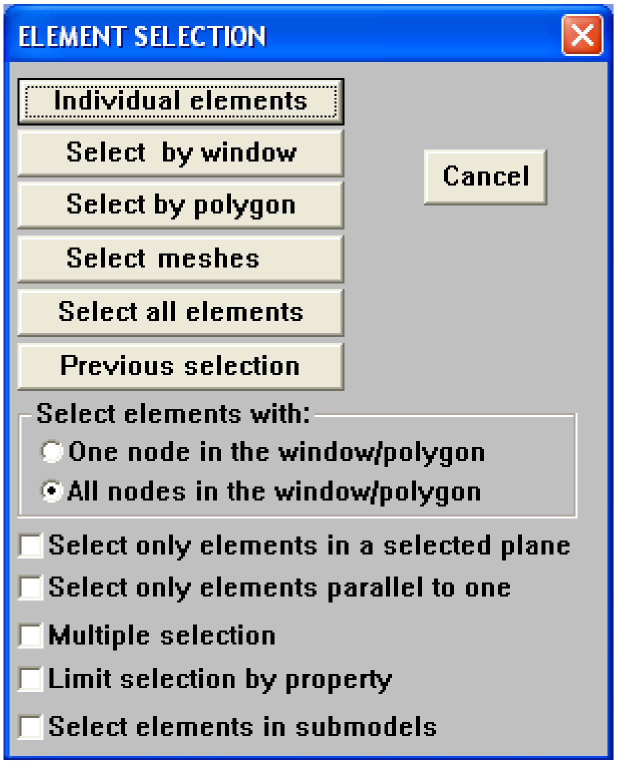
☑ the program allows you to select beams in the main model and the submodels
☐ only beams defined in the main model may be selected.

1.2.3 Element selection

Many options include instructions to select one or more elements.

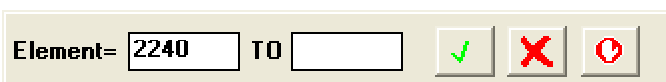
When selecting a element, the element nearest to the is highlighted with a rectangular blip ■. The number of the highlighted element is always displayed at the left-hand side of the Dialog box.

You may also type in the number of the element to be selected.



Individual elements

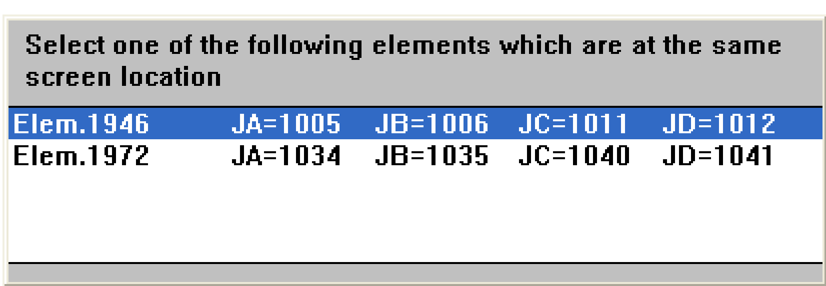
Select a single element only by moving the alongside the element until it is highlighted by a rectangular blip ■; click the mouse. The number of the highlighted element is always displayed at the left-hand side of the Dialog box:



You may also type in the number of the element to be selected, in the form of a "list".

When all the elements have been selected, press or click the mouse without moving the .

In space models, more than one element may be at the same screen location (only the coordinate perpendicular to the screen is not identical). In such a case, the program displays a list of elements at that location and request the user to select one:

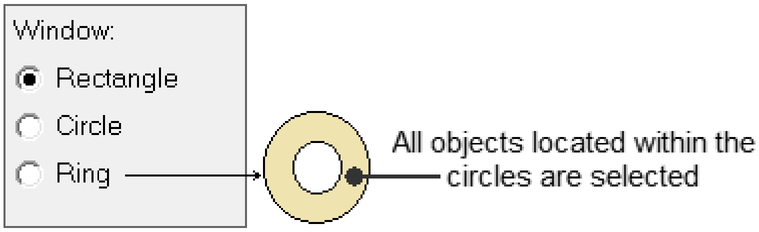


Highlight the line with the correct element and click the mouse.

Select by window

Define a window; the program automatically identifies all elements with either one or all nodes located in the window (refer to Select elements with 34).

The window may be either a rectangle, a circle or a ring.



A rectangle is defined by pointing to its lower-left and upper-right corners with the .

In space models, more than one element may be at the same screen location (only the coordinate perpendicular to the screen is not identical). In such a case, the program selects ***all*** of the elements at that location.

Select by polygon

Define a polygon by pointing to its corners with the . The program automatically identifies all elements with either one or all nodes located in the polygon (refer to Select elements with 34)

The polygon is constructed as a 'rubber-band' stretched around the defined corners:

- At least three corners must be selected.
- the program automatically connects the last corner defined to the first corner defined.
- press [Esc] (right mouse button) to delete the previous corner.
- to end the polygon definition, click the mouse without moving the
- In space models, more than one element may be at the same screen location (only the coordinate perpendicular to the screen is not identical). In such a case, the program selects ***all*** of the elements at that location.

Select element mesh

Select ***all*** elements in a mesh created by the geometry Element - Mesh option. Move the alongside any element in the mesh until it is highlighted by a rectangular blip ■; click the mouse.

Note that all contours of all defined meshes are highlighted with a thick line during the selection.

Select all elements

All elements in the model will be selected.

Note:

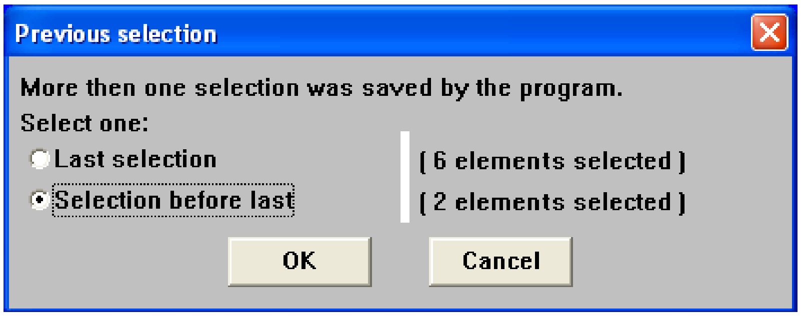
- elements not displayed because of the the **Zoom** option ***are*** selected.
- elements not displayed because of the **Remove beams/elements** option are ***not*** selected.

Previous selection

The program remembers the list of selected elements every time elements are selected. The elements from the previous selection are highlighted and the option continues according to **Multiple selection.**

Note:

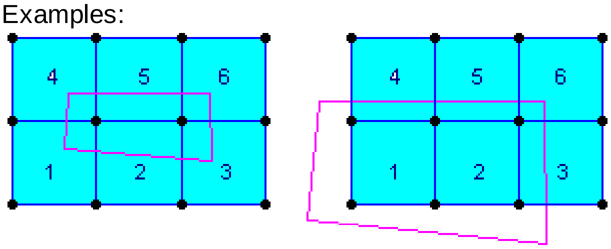
- the program remembers up to 5 previous selections and you may select any of them. For example:



Select elements with:

- **One node in the window/polygon**:
  all elements with at least one corner node in the window/polygon are selected.
- **Both nodes in the window/polygon**:
  only elements with ***all*** corner nodes in the window/polygon are selected.

Examples:



Elements selected:

All nodes: no elements	**All nodes**: elements 1 and 2
One node: all elements	**One node**: all elements

Elements on a plane

You may impose a further limit that the elements selected will be only those lying on a specified plane. This option allows you, for example, to define a window around an entire model but to select only a certain level.

The plane is defined by pointing to an existing element; only elements lying on the same plane as this element are selected.

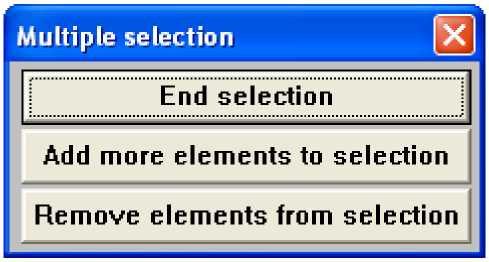
Select only elements parallel to an element

You may impose a further restriction that only the elements parallel to a specified element will be selected. This option allows you, for example, to define a window around several adjacent levels but to select only one of them.

Multiple selection

Turn on the checkbox if you want to define several windows, polygons or lists for the same command. After every window, etc., the following menu is displayed:

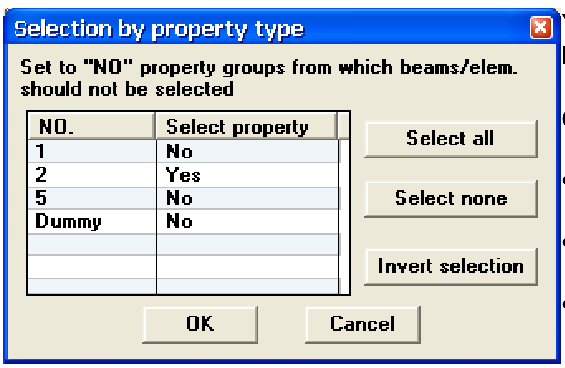
Select:



- Continue without selecting more elements
- Define another window, polygon, etc. for the same command.
- Delete elements from the list, window, etc. already defined for this command.

Limit by properties

Further limit the element selection according to property groups:



Yes: elements in the property group are selected
No: elements in the property group are ***not*** selected

Click:

- Select all to set all properties to **Yes**
- Select none to set all properties to **No**
- Invert selection to toggle the selection for all properties

Note:

- Dummy elements may be selected

Submodels (results only)

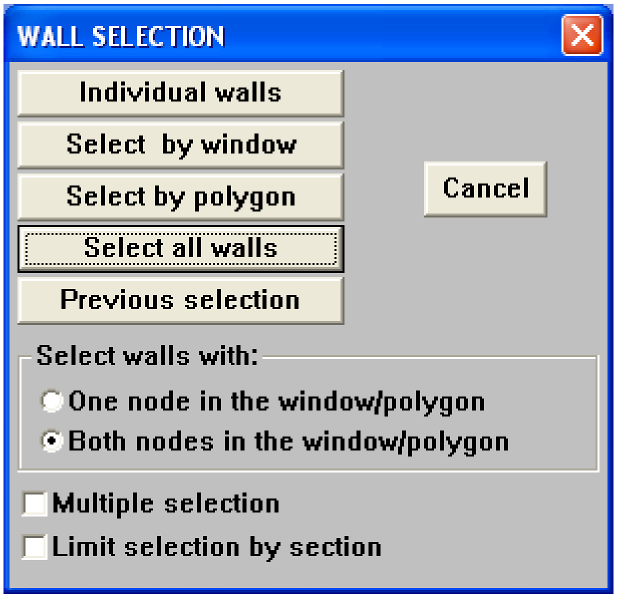
This option is available in results when the Main model is displayed:

☑ the program allows you to select elements in the main model and the submodels
☐ only elements defined in the main model may be selected.

1.2.4 Wall selection

Many options include instructions to select one or more walls.

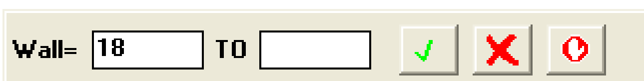
Wall selection is similar to beam selection. The segment nearest to the cursor is highlighted with a rectangular blip ■ (choose any segment to select the wall). The number of the highlighted wall is always displayed at the bottom-left of the screen; you may also type in the number of the wall to be selected.



Individual walls

Select a single wall only by moving the alongside the element until it is highlighted by a rectangular blip ■; click the mouse.

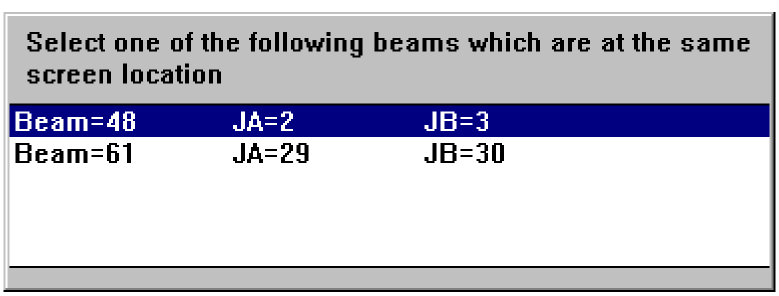
The number of the highlighted wall is always displayed at the left-hand side of the Dialog box:



You may also type in the number of the wall to be selected, in the form of a "list".

When all the walls have been selected, press or click the mouse without moving the .

In space models, more than one wall may be at the same screen location (only the coordinate perpendicular to the screen is not identical). In such a case, the program will display a list of walls at that location and request the user to select one:

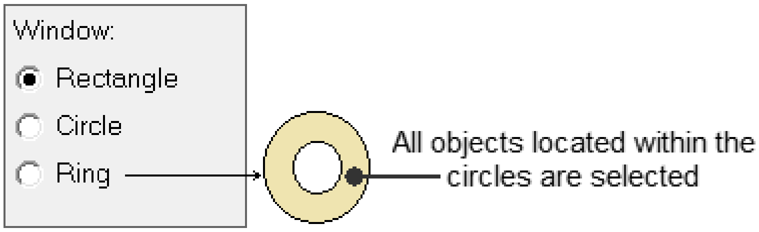


Highlight the line with the correct wall and click the mouse.

Select walls by window

Define a window; the program automatically identifies all walls with either one or both "attachment" nodes located in the window (refer to Select walls with 38)

The window may be either a rectangle, a circle or a ring.



A rectangle is defined by pointing to its lower-left and upper-right corners with the .

In space models, more than one wall may be at the same screen location (only the coordinate perpendicular to the screen is not identical). In such a case, the program selects ***all*** of the walls at that location.

Select walls by polygon

Define a polygon by pointing to its corners with the . The program automatically identifies all walls with either one or both "attachment" nodes located in the polygon (refer to Select walls with 38)

The polygon is constructed as a 'rubber-band' stretched around the defined corners:
- At least three corners must be selected.
- the program automatically connects the last corner defined to the first corner defined.
- press [Esc] (right mouse button) to delete the previous corner.
- to end the polygon definition, click the mouse without moving the
- In space models, more than one wall may be at the same screen location (only the coordinate perpendicular to the screen is not identical). In such a case, the program selects ***all*** of the walls at that location.

Select all walls

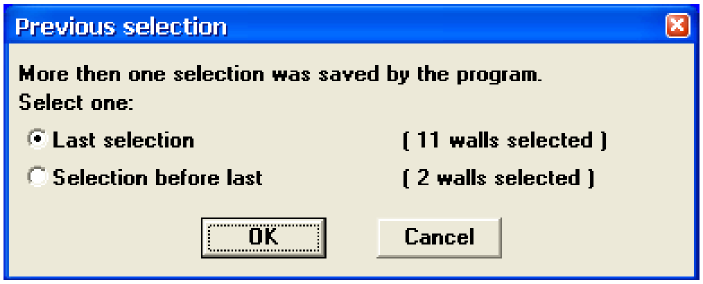
All walls in the model will be selected.

Note:
- walls not displayed because of the the **Zoom** option ***are*** selected.
- walls not displayed because of the **Remove walls** option are ***not*** selected.

Previous selection

The program remembers the list of selected walls every time walls are selected. The walls from the previous selection are highlighted and the option continues according to **Multiple selection.**

Note:
- the program remembers up to 5 previous selections and you may select any of them. For example:

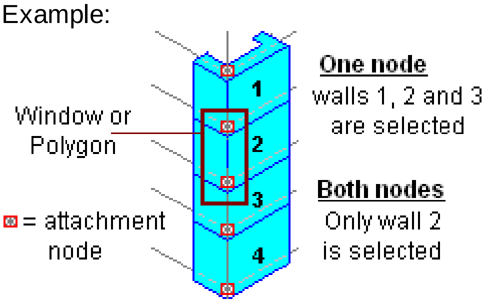


Select walls with:

- **One node in the window/polygon**:
  all walls with at least one end node in the window/polygon are selected.
- **Both nodes in the window/polygon**:
  only walls with **both** end nodes in the window/polygon are selected.

Note that the program only considers the nodes where the wall "reference point" was attached to the model:

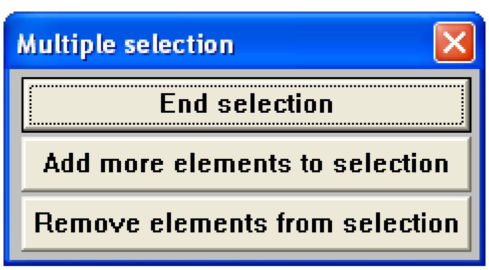
Example:



Multiple selection

Turn on the checkbox if you want to define several windows, polygons or lists for the same command. After every window, etc., the following menu is displayed:

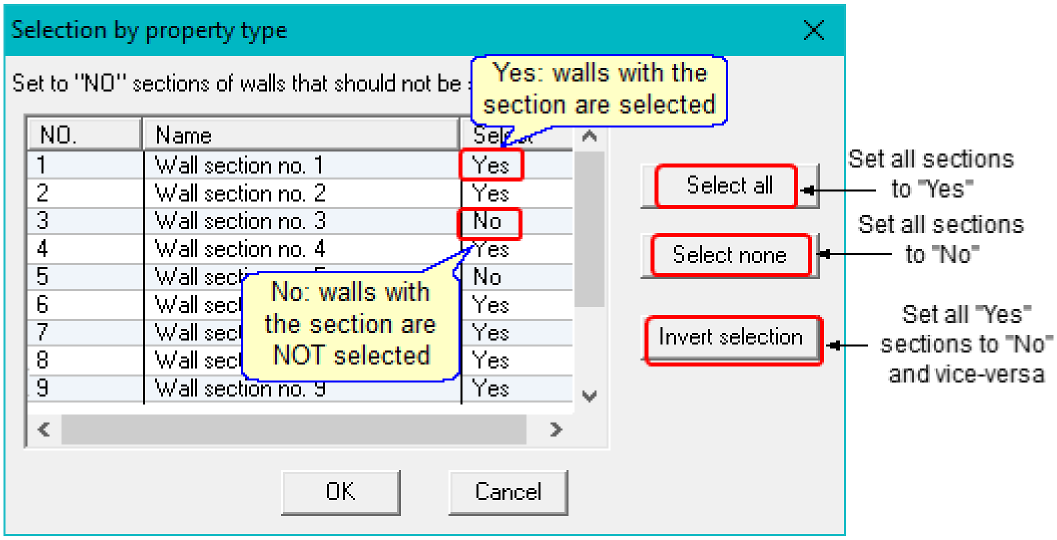
Select:



- Continue without selecting more walls
- Define another window, polygon, etc. for the same command.
- Delete walls from the list, window, etc. already defined for this command.

Limit selection by section

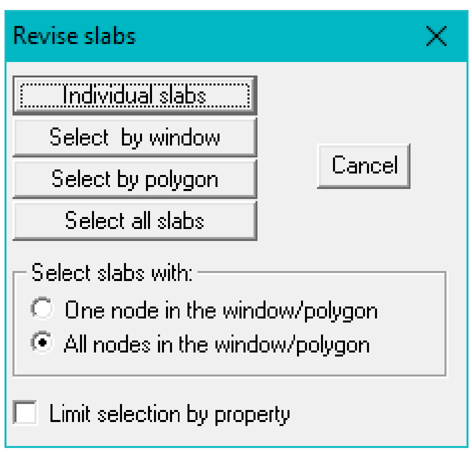
Further limit the wall selection according to section type. Click on any line to toggle the option:



1.2.5 Slab selection

Many options include instructions to select one or more slabs.

Slab selection is similar to beam selection. The contour corners of the slab nearest to the are highlighted with a rectangular blip ■. The number of the highlighted slab is always displayed at the bottom-left of the screen; you may also type in the number of the slab to be selected.



Individual slabs

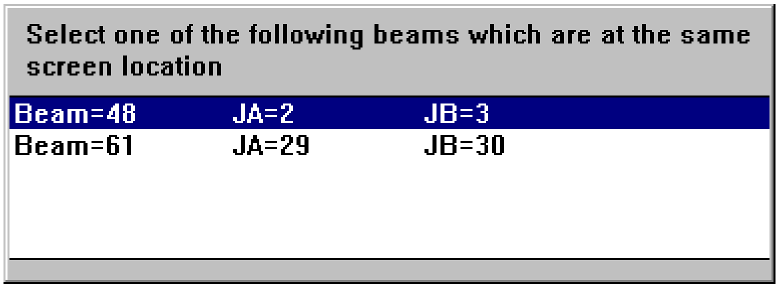
Select a single slab only by moving the alongside the it until it is highlighted by a rectangular blip ■; click the mouse.

The number of the highlighted slab is always displayed at the left-hand side of the Dialog box:

You may also type in the number of the slab to be selected, in the form of a "list".

When all the slabs have been selected, press or click the mouse without moving the .

In space models, more than one slab may be at the same screen location (only the coordinate perpendicular to the screen is not identical). In such a case, the program will display a list of slabs at that location and request the user to select one:

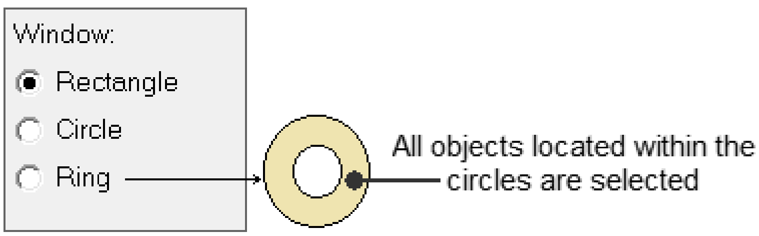


Highlight the line with the correct slab and click the mouse.

Select slabs by window

Define a window; the program automatically identifies all slabs with either one or both "attachment" nodes located in the window (refer to Select slabs with 38)

The window may be either a rectangle, a circle or a ring.



A rectangle is defined by pointing to its lower-left and upper-right corners with the .

In space models, more than one slab may be at the same screen location (only the coordinate perpendicular to the screen is not identical). In such a case, the program selects ***all*** of the slabs at that location.

Select slabs by polygon

Define a polygon by pointing to its corners with the . The program automatically identifies all slabs with either one or both "attachment" nodes located in the polygon (refer to Select slabs with 38)

The polygon is constructed as a 'rubber-band' stretched around the defined corners:
- At least three corners must be selected.
- the program automatically connects the last corner defined to the first corner defined.
- press [Esc] (right mouse button) to delete the previous corner.
- to end the polygon definition, click the mouse without moving the
- In space models, more than one slab may be at the same screen location (only the coordinate perpendicular to the screen is not identical). In such a case, the program selects ***all*** of the slabs at that location.

Select all slabs

All slabs in the model will be selected.

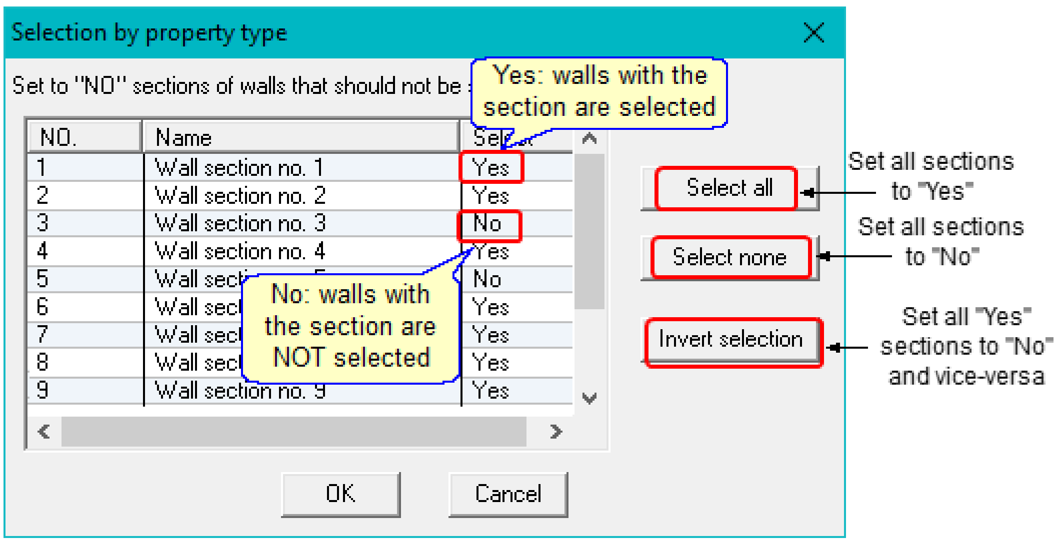
Note:
- slabs not displayed because of the the **Zoom** option ***are*** selected.
- slabs not displayed because of the **Remove slabs** option are ***not*** selected.

Select slabs with:

- **One node in the window/polygon**:
  all slabs with at least one end node in the window/polygon are selected.
- **All nodes in the window/polygon**:
  only slabs with **all** end nodes in the window/polygon are selected.

Limit selection by property

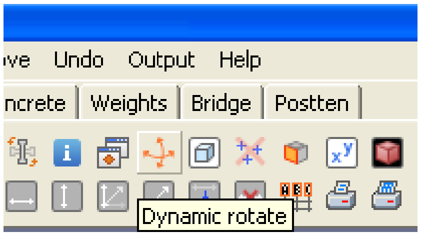
Further limit the slab selection according to section type. Click on any line to toggle the option:



1.3 Toolbar options

Most options in the pulldown menus and the toolbar may also be selected by clicking an icon in the "toolbar".

- Move the arrow adjacent to the icon; a "tool tip" describing the icon function is displayed (after a few seconds); for example:



- click the left mouse button

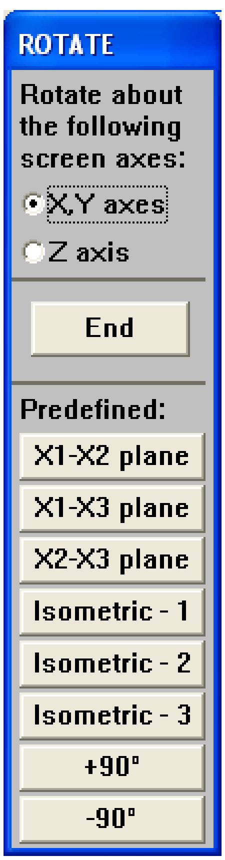
Several options are available only in the toolbar:

 dynamic rotate 42

 isometric view 43

Note:
- you can create new toolbars or customize existing bars. Refer to Setup - toolbars.

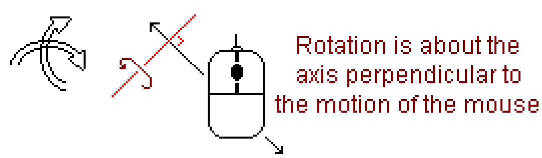
1.3.1 Rotate - dynamic



Rotate - dynamic

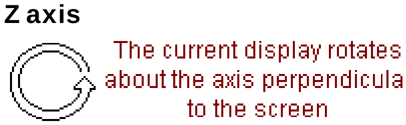
Select one of the options in the side menu and click the left mouse button:

X,Y axes



Note:
You can rotate the model about the X,Y axes by pressing the left mouse button & [Ctrl] while moving the mouse.

Z axis



Note:
You can rotate the model about the Z axis by pressing the left mouse button & [Shift] while moving the mouse.

Rotate - global plane

Click one of these buttons to display the model as projected on one of the three global planes.

Rotate - isometric

Click one of these buttons to rotate the model to one of the three predefined isometric views.

The rotation angles for each view are defined in Setup - Miscellaneous - Isometric

Rotate - 90 degrees

Click one of the buttons to rotate ALL axes by 90 degrees.

Note:
- To rotate the model 'by steps', select Rotate 54 in the toolbar.

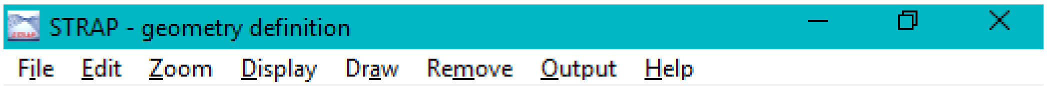
1.3.2 Isometric

Click this button to rotate the model to a predefined isometric view.

The rotation angle is defined in Setup - Miscellaneous - Isometric

1.4 Menu bar options

The graphic display shows all data entered up to the previous command. By default, the program displays the entire model projected on the X1-X2 plane. In order to check the model, it is usually necessary to zoom in on part of it, rotate it, isolate individual planes, add numbering, etc.

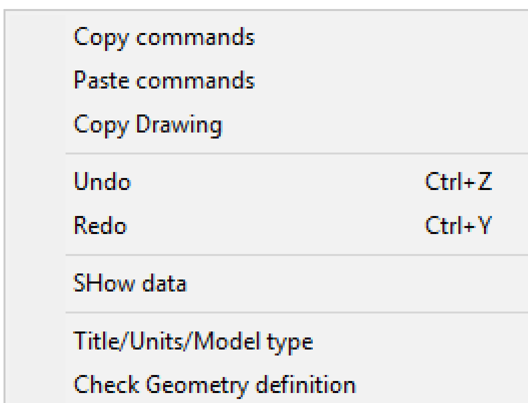


Help is also available for the following topics:

Dialog boxes 18 - an overview of "Windows" dialog box conventions.
Shortcut menus 21 (right-click on object)
Side menus 21
Help 21 - overview of the **Help** option
Step 20 option
Node selection 24
Beam /Element/Wall selection 27

The technique of creating a computer model for a structure is explained in more detail in Getting Started .

1.4.1 Edit



Copy commands

Use this option to copy ***STRAP*** commands to the "Clipboard":

- Enlarge the Command Box so that the entire block of commands to be copied is displayed on the screen.
- Move the cursor to the first command in the block; press the mouse key - ***do not release***.
- Drag the mouse to the last command in the block; all of the commands will be highlighted. Release the mouse key.
- Select the **Copy commands** option in the **Edit** menu; the highlighted commands are copied to the Clipboard.

The commands may then be retrieved by any text editor program with a **Paste** option (or equivalent).

Paste commands

Commands or selected commands located in any ASCII file may be retrieved via the Windows "Clipboard". Refer to Command mode - retrieve a command 1201.

Copy drawing

Copy the current display tto the Windows "clipboard" in "Metafile" format. Refer to Copy display to clipboard 94.

Undo [Ctrl]+[Z]

Clicking the **Undo** option in the menu bar automatically cancels the last definition; the graphic display is immediately redrawn. Note that **Undo** may be pressed repeatedly, but will only cancel commands from the current session.

Redo [Ctrl]+[Y]

Clicking the **Redo** option in the **Edit** pull-down menu automatically cancels the previous **Undo**; the graphic display is immediately redrawn.

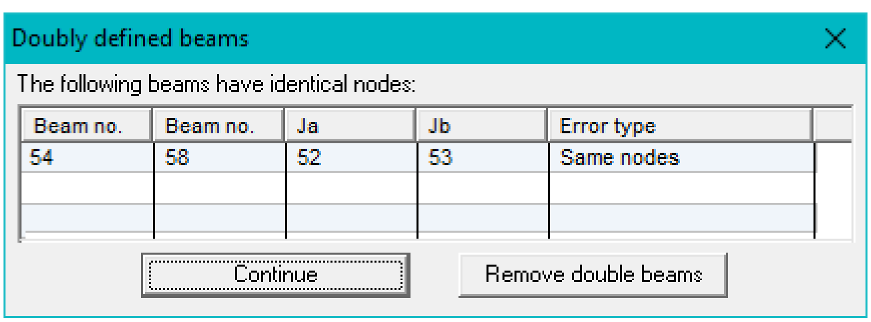
Note:
- **Redo** must follow the **Undo.**
- **Redo** may be pressed repeatedly following a consecutive series of **Undo**.

Title/Units/Model type (geometry only)

Refer to - Geometry wizard.

Check geometry definition (geometry only)

This option checks the legality of the model geometry. Warning messages are displayed when problems are found. For example, two or more beams defined between the same end nodes:



The program checks the following:

Beams:
- two or more beams defined between the same end nodes
- beams with zero length
- beams connected to undefined nodes

- beams on same line (may have different end nodes)

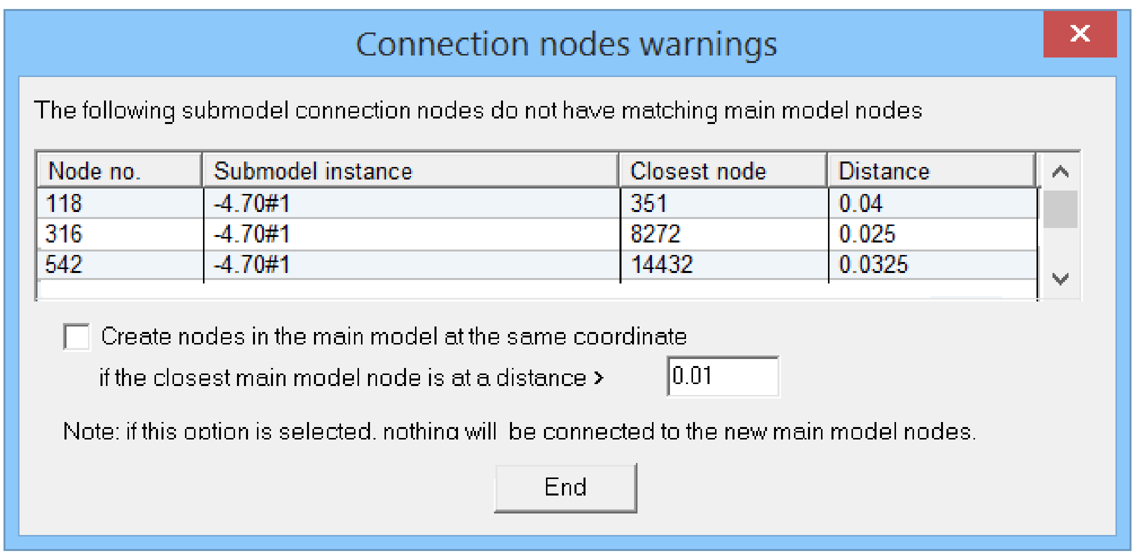
Quad/triangular elements:
- two or more elements defined between the same end nodes
- elements connected to undefined nodes
- elements with zero area
- non-planar elements
- overlapping elements
- elements with common edge without common nodes or element corner on another element edge ("touching elements").
- elements with zero area
- elements where the ratio of the lengths of the shortest side to the longest side is less than 1:15.

Solid elements:
- elements connected to undefined nodes
- elements with zero volume
- end nodes defined in correct order

Walls:
- two or more wall segments defined between the same end nodes

Restraints:
- Support nodes connected to other nodes by Rigid links.

Submodels:
- two submodel instances added to the main model at the same location.
- main model nodes in the area of a submodel not attached to a submodel connection point.
- submodel connection points not attached to a main model node. The program can create new main model nodes at the connection points (note that they will not be connected to anything in the main model):

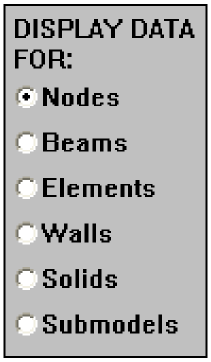


The latter two submodel problems are easily solved by using the Automatic connection points option.

1.4.1.1 Show data

Display all input data for a selected node/beam/element:

- specify the data type option in the box at the right side of the screen:



- highlight the node/beam/element by placing the adjacent to it and click the mouse
- the following data will be displayed:

Nodes:
- Coordinates, restraints, attached beams, elements and walls, springs,
- distance from previous selected node.
- if the node is used:
  - as a JC node for beam section rotation
  - to define a restraint coordinate system
  - to define Grid lines

  (these nodes cannot be deleted)

If a cylindrical coordinate system is currently active, the coordinates displayed are relative to that system.

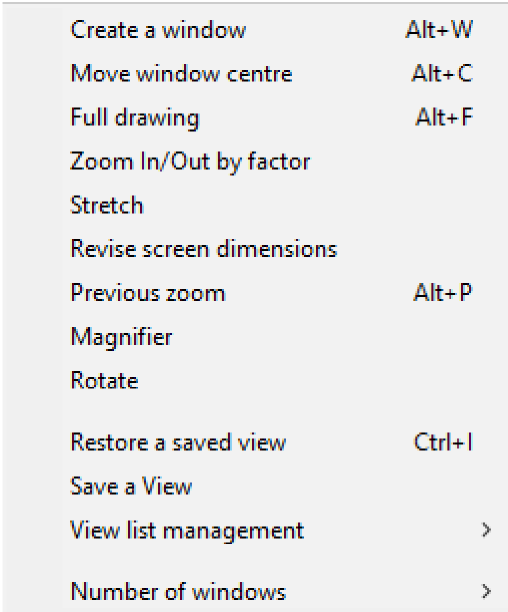
Beams: Length, end nodes, JC node, property group number, section properties/name/ dimensions, material, releases, offsets.

Elements: End nodes, thickness, material, area, releases

Walls: Section drawing, end nodes, area, volume, height, material, centre-of-gravity.

Submodels: Only connection nodes may be selected. The program displays the following data: corresponding main model node, the distance between the connection nodes and the main model node, instance name, connection type,

1.4.2 Zoom



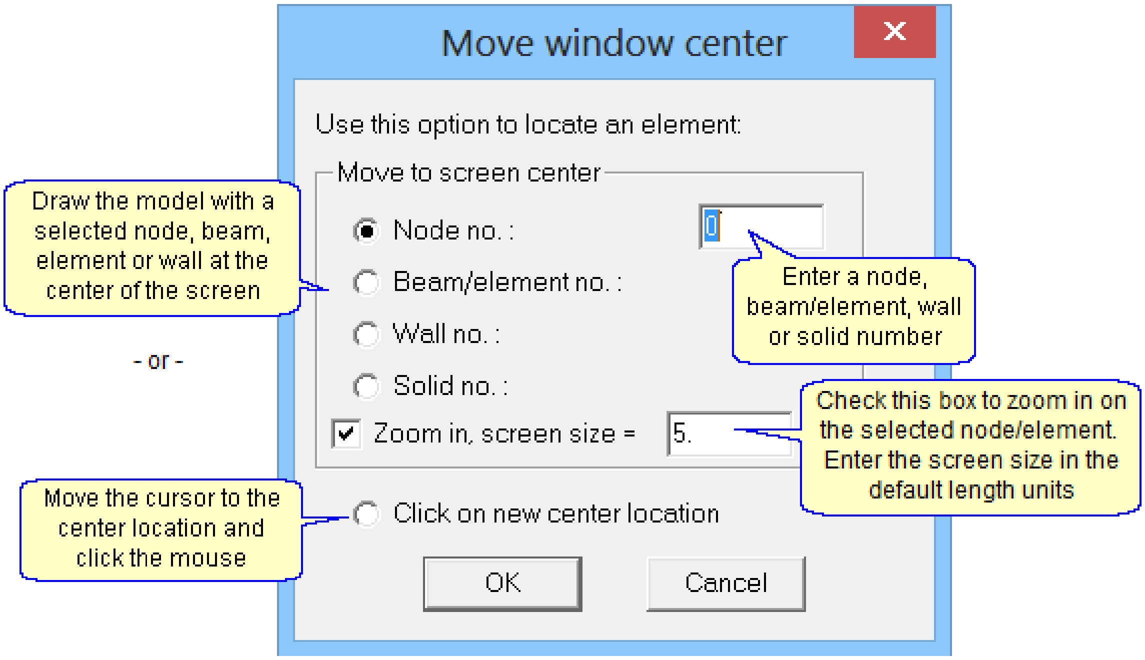
1.4.2.1 Window options

Create a window [Alt]+[W]

Create a window by defining the lower-left and upper-right corners; the contents of the window will be displayed over the entire screen.

Move window centre [Alt]+[C]

Relocate the centre of the current window to a new location. The relocated window is drawn with current options and scale.



Full drawing [Alt]+[F]

Display the entire model according to the current display options. You may also press the [F3] key.

Zoom in/out by factor [Ctrl][+/-]

Zoom in on the current display; the center remains in the same location but the scale is changed. The degree of zoom is defined by entering the percentage scale change (a factor between 0 and 100%). Note that the percentage is the ***ratio between the change in scale to the final scale***.

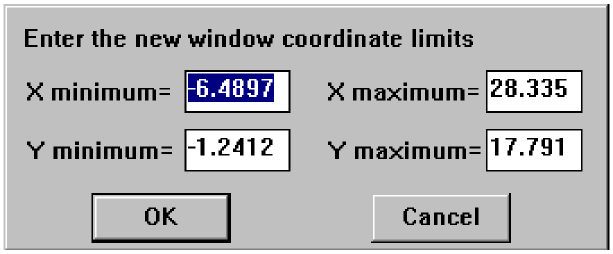
For example, to reduce the scale by half, select 100%.

Stretch

Use this option to stretch the current display over the entire screen. Although the model will be distorted, numbering verification will be facilitated for many models (e.g. tall and narrow) as the nodes/elements will be moved away from each other.

Revise screen dimensions

Create a window or expand the display area by entering the minimum and maximum global coordinates defining the display limits:



Previous zoom [Alt]+[P]

To restore the previous display, i.e. the display before the current zoom.

Note:

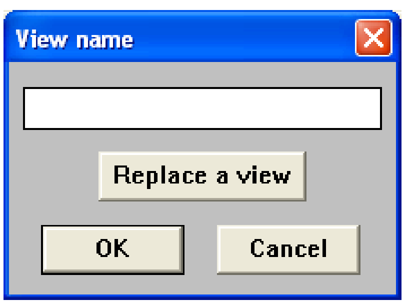
- the program stores up to 20 previous displays
- the stored 'zooms' include those created using the mouse wheel, "Rotate", Dynamic rotate", "Create a window" and "Move window center".
- the zoom will be restored with the ***current*** Numbering and Remove options.

1.4.2.2 View options

Save a view

To save the current display, including:

- zoom, rotate, numbering, remove, etc.
- result display type and parameters; section line locations for finite element graphic results.



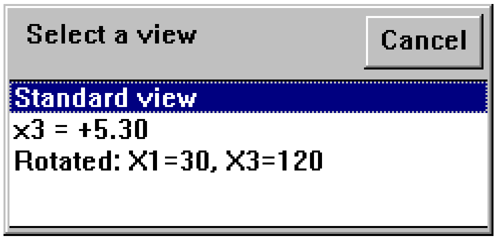
- To save a new view, enter a title for the view and click OK
- To resave an existing view with the new display parameters, click Replace a view and select a view title from the list.

Note:

- up to 500 views may be saved per model.
- 'Views' saved in Results also retain the result type and parameters. For example, save a view when a contour map is displayed; the same contour map is displayed every time the view is recalled when in Results.

Restore a saved view

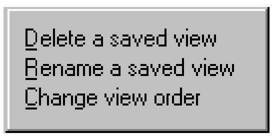
Select a saved "view" from the list; the model will be redrawn immediately.



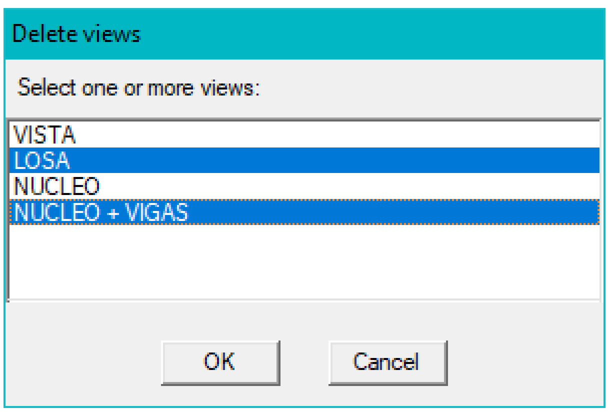
Note that the program automatically saves the initial display (X1-X2 axis, no rotate, zoom or numbering) with the title "Standard view".

View management

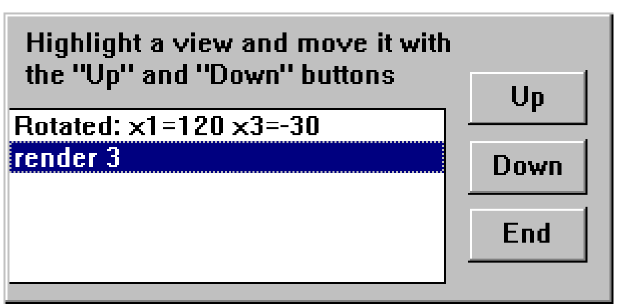
Select one of the following options:



- **Delete a saved view**
  Select one or more saved "views" from the list; they will be deleted from the View list.

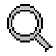


- **Rename a saved view**
  Select a saved "view" from the list; type in a new name.
- **Change view order**
  Change the order of the titles in the View list box:
  - Click and highlight a line in the box; click the Up or Down button to change its position by one line.
  - Click again to continue moving the same line.

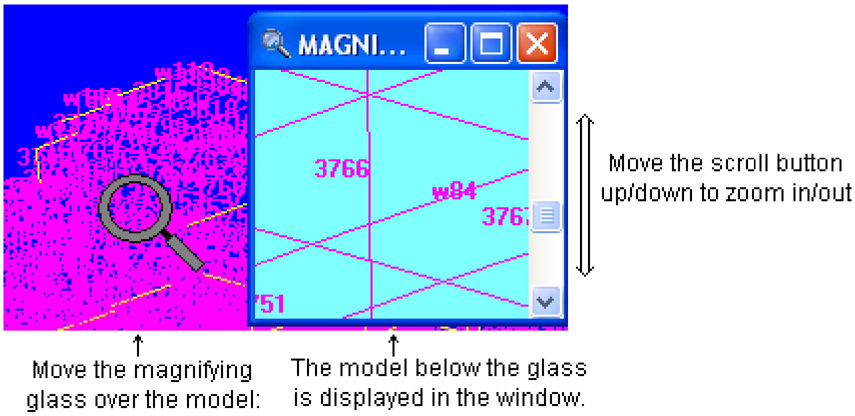


  - repeat for other lines

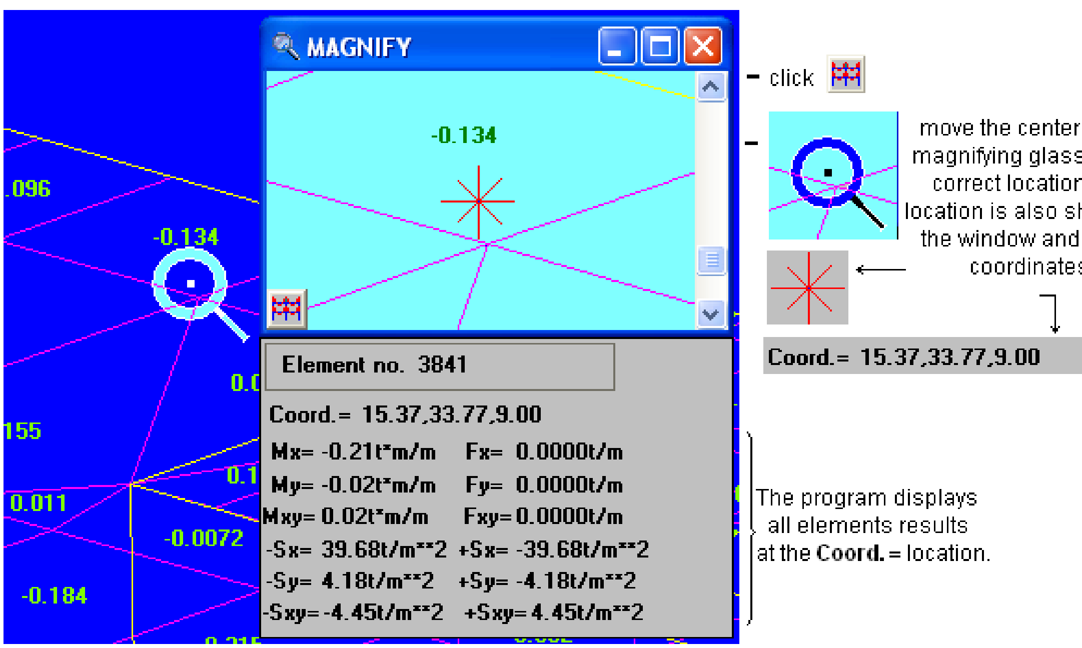
1.4.2.3 Magnifier

Magnify a small portion of the current display without zooming in; the magnified portion is displayed in a new window. Move the  cursor to the correct location. For example:

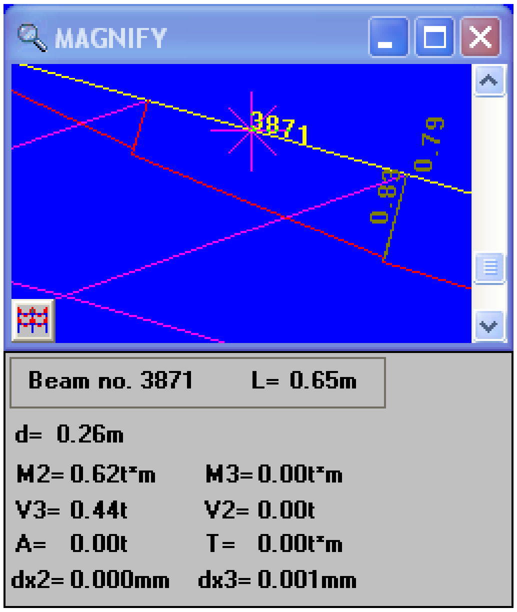
- Geometry, loads:



- Results:
  - Elements:



  - Beams:

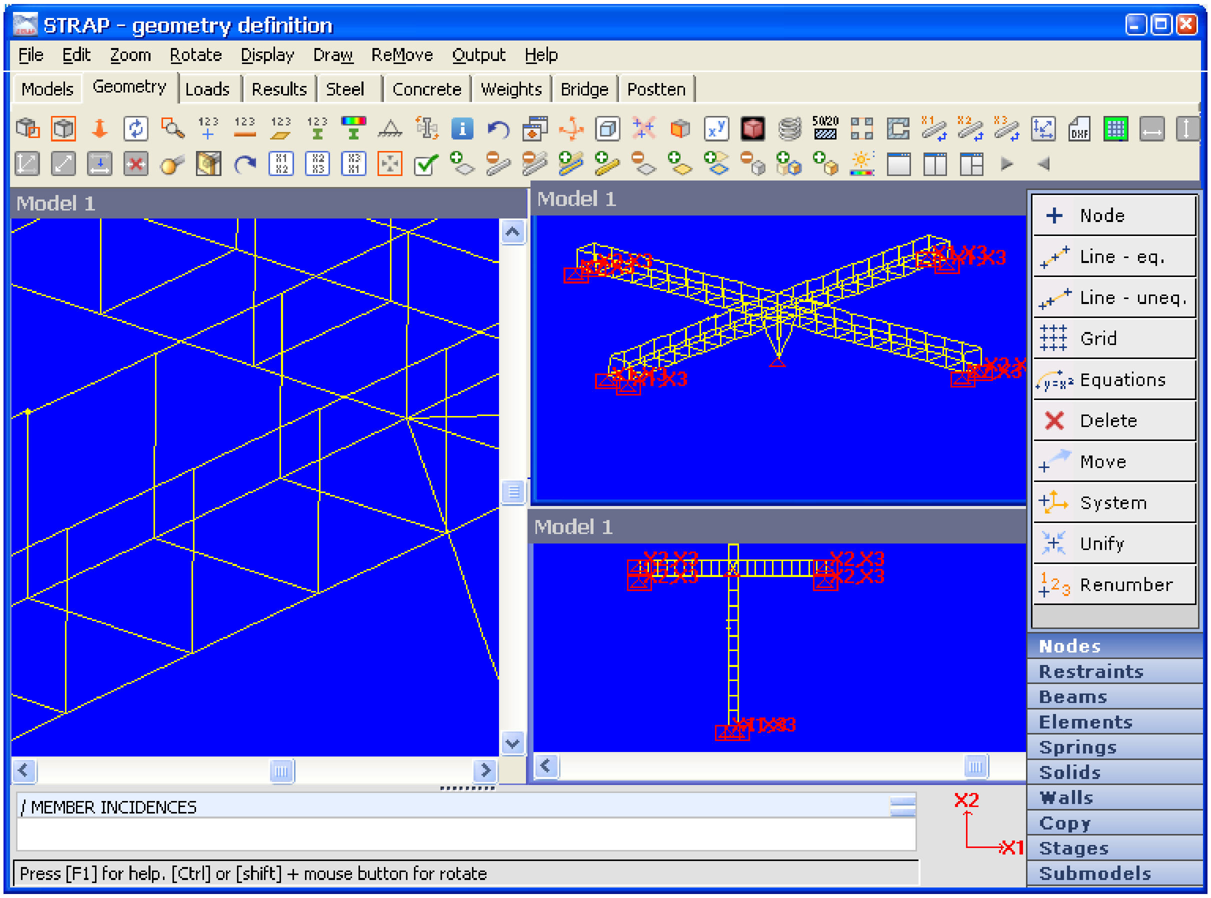


1.4.2.4 Number of windows

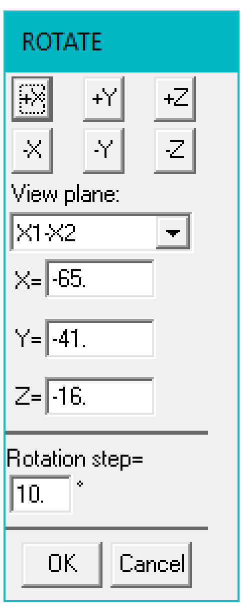
The display screen can be divided into two, three or four display windows, all showing the current model.

- the model is defined in any one of the windows, referred to as the "active" window; double click any part of a window to select it as the active one.
- each window may have different display parameters (rotate, zoom, remove, numbering, etc).
- all windows are updated simultaneously as the model is defined.
- the size of the windows may be modified by dragging the frame.
- the active window may be changed during a definition sequence. For example, when defining a single beam, the two end nodes may be selected in different windows.
- in graphic results, the result type selected is displayed in the active window only. By changing the active window display moment in one window, then shear in another window, etc.

For example: (3 windows)



1.4.2.5 Rotate



Rotation by steps

The model may be rotated in steps about any of the three screen axes.

Each time one of the icons is pressed, the model is rotated by the value displayed at the bottom of the dialog box (default = 10°) from its previous position. The step value may be revised at the bottom of this dialog box.

View plane

Select the initial viewing plane: **X1-X2 , X2-X3** , or **X3-X1**. The default plane is **X1-X2**.

Note that the first axis of the pair will be the horizontal axis on the screen.

Rotation about X/Y/Z axes

Rotate the model about the three axes to view from any direction. Enter the angle of rotation in degrees. A positive angle is measured counter-clockwise about the positive direction of the axis.

Note that the rotation is about the current **View plane** axes:

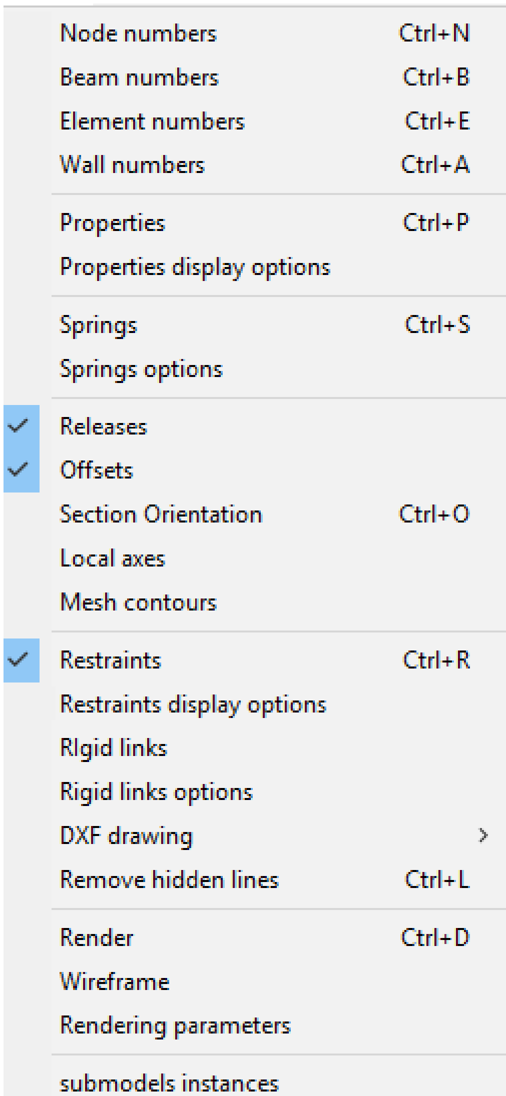
- If the current View plane is X1-X2, then X refers to X1, Y to X2 and Z to X3. If the View plane is revised to X2-X3, then X refers to X2, etc.
- The angles typed in always refer to the unrotated View plane and not to the

current rotated display!

Note:

- for the Dynamic rotate 42 option click in the Icon bar.

1.4.3 Display



Refer also to - Display loads 68

1.4.3.1 Input data

Node/beam/element/wall numbers [Ctrl]+[N]/[B]/[E]/[A]

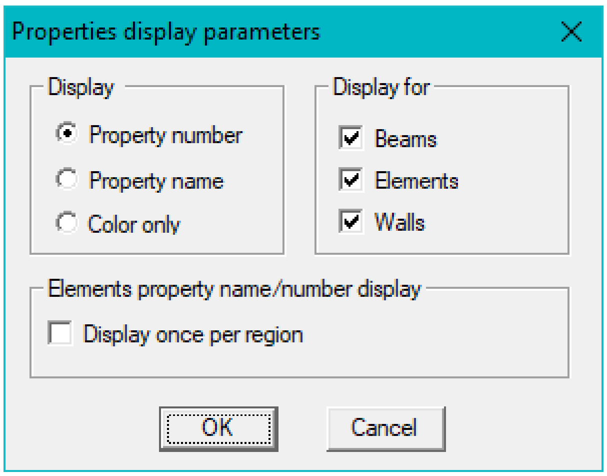
Node, beam, element and/or wall numbers may be added to the current display:

- Move the to the option and click the mouse; a ✓ appears beside the option and the numbering is added to the current display window.

- To remove the numbering select the same option again.

Submodels:
By default the submodel node/beam/element numbers are not shown when the Main model is displayed. There are two methods to display the submodel numbering when the Main model is shown:
- display the submodel, display the numbers, save a view. When the Main model is displayed select **Files** in the menu bar and **Submodel instances** in the menu; select the submodel in the list and the saved view in the listbox in the ***Display*** column.
- display the submodel, display the numbers. Return to the Main model, select **Files** in the menu bar and **Submodel instances** in the menu; select the submodel in the list and ***Current view*** in the listbox in the ***Display*** column.

Properties [Ctrl]+[P]



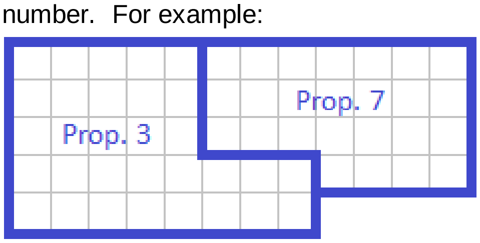
The current property data may be displayed adjacent to beams, elements and/or walls by one of three methods:

Property number :
- beams/elements: the number of the property group is written adjacent to each element
- walls: the wall section number is displayed.

Property name : the property name is written adjacent to the element. For example:
- beams:
  - steel section: **UB127/76**
  - concrete section: **300/650**
- elements: **th=20**
- walls: - not displayed -

Property by colours : each beam/element/wall is drawn with a colour representing the property group

Display once per region
The program display the property number only once in element regions with the same property number. For example:



Springs / Spring options [Ctrl]+[S]

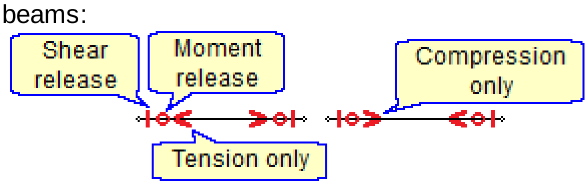
Select this option to display the location of springs and/or the spring constant.

There are three options:
- display a symbol only
- display a symbol with the direction, e.g. **S2**
- display the direction and the value for selected directions (no symbol), e.g. **S2=155**

Select **Spring options** in this menu and select the options and directions.

Releases

- beams:



- slabs, elements: a small circle **o** is displayed at the ends of released edges

Offsets

✓ Display a thick line at the beam ends where rigid offsets have been defined (Figure [a]).

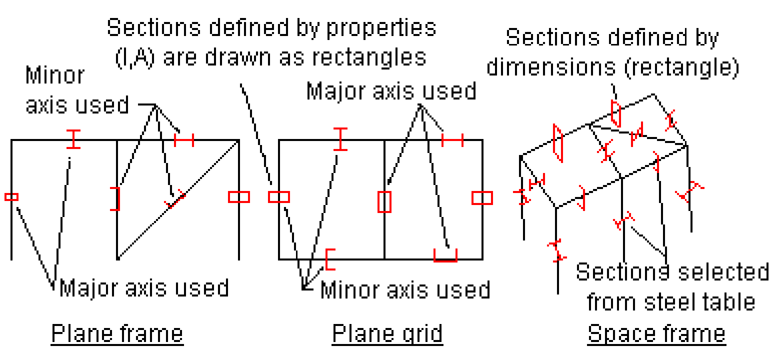
A different style line is drawn if the ☑ **Do not generate offset moments in beam** option is selected (Figure [b]).

Section Orientation [Ctrl]+[O]

Select this option to draw a schematic representation of
- the beam section shape and major/minor axis orientation.
- the orientation of the ribs and holes in elements.
- the initial geometry in cable elements (the sag after the self-weight and initial tension are applied).

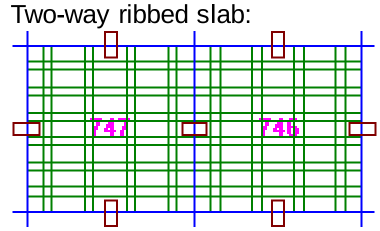
For example:

Beams:



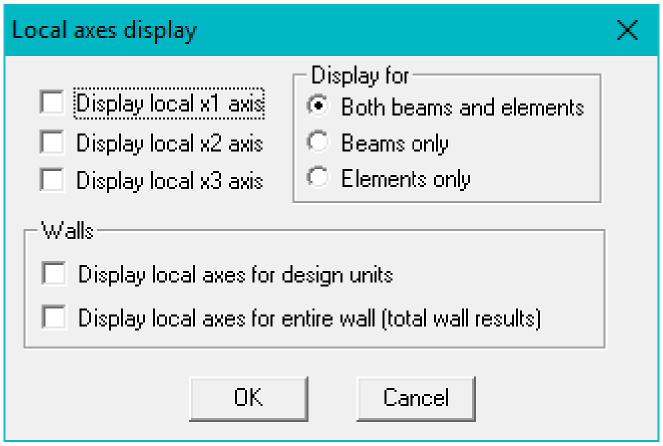
Elements:

Two-way ribbed slab:



Local axes

"Local axes" displays beam, element and wall local axes directions. Note that all axes can be displayed simultaneously.



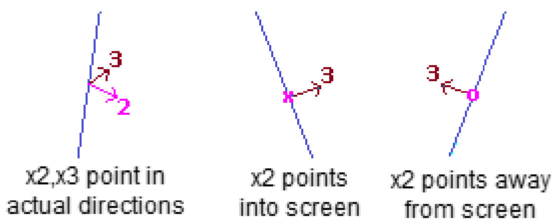
The following conventions are used when displaying the local axes directions:

Beam Elements:

The x1 axis lies on the axis of the beam.
The x2,x3 axes are displayed pointing in their actual direction. If the axis is perpendicular to the screen plane, the axis is displayed with an

- **x** - if the axis points away from the user (into the screen)
- **o** - if the axis points towards the user (out of the screen)

The axes are indicated by a "2" or "3".
Examples:



Quad and Triangular Finite Elements:

The program displays only the x1 and x3 axis directions:

- x1 always lies in the plane of the element. The arrow is drawn from JA in the direction of JB.
- x2 is ***not*** displayed. It always lies in the plane of the element, is perpendicular to x1 and points from JA in the direction of the other corner node(s).
- x3 is always perpendicular to the element. If its positive direction points out of the screen, an "o" is displayed at the centre of the element; if it points into the screen, an "x" is displayed.

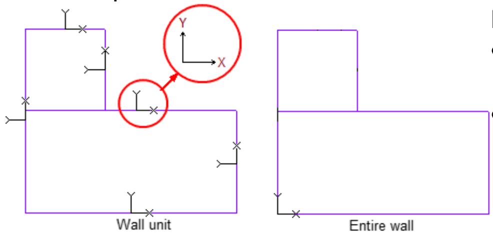
Wall local axes:

Display the design X,Y axis for wall "design units" or the entire wall.

- entire wall: the X axis is the horizontal axis in the wall section definition menu (used for wall 'Total results').
- wall unit: the X axis is parallel to the direction of the longest segment (or combined segment) in the unit.

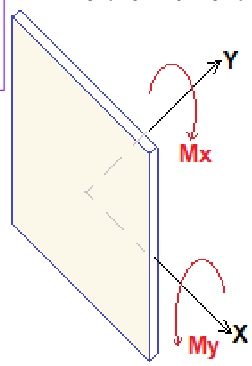
The Y axis is perpendicular to the X axis.

For example:



Note:

- If no wall units are defined, each segment (or a straight line of connected segments) is a design unit.
- **Mx** is the moment in the X direction, i.e. ***about the Y axis***

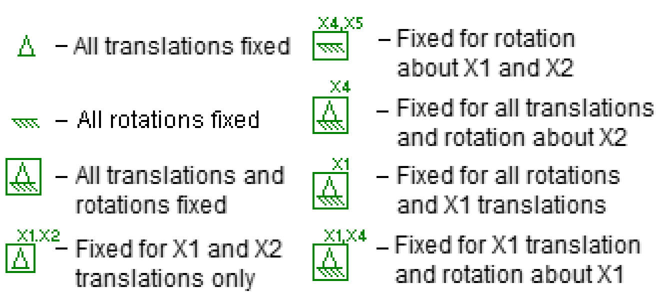


Mesh contours

✓ Display a thick line along the perimeter contour of all meshes defined with the Element - Mesh option.

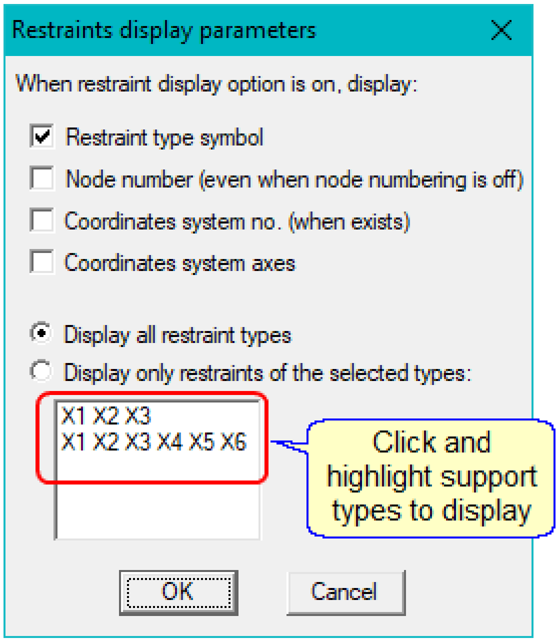
Restraints [Ctrl]+[R]

Restraints in all degrees-of-freedom may be displayed on the model. The restraints are drawn schematically as shown below. Note that X4, X5 and X6 are the rotation degrees-of-freedom about X1, X2 and X3, respectively.

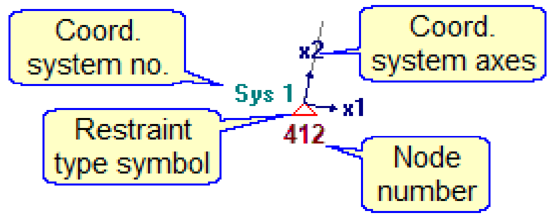


The symbol size can be enlarged in Setup - Miscellaneous.

Restraint display options



Display options:
Select one or more of the following symbols to display:

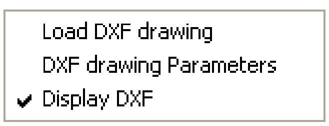


Selected restraints:

Select the support types to be displayed. For example, display only pinned supports.
A list of all defined support types is displayed in the box; select and highlight one or more and click

1.4.3.2 DXF drawing

Add a DXF drawing to the *background display* of the current model. The DXF drawing is ***not*** added to the model. The program allows the ends of all DXF drawing lines to be defined as 'nodes' in the **Node** options.



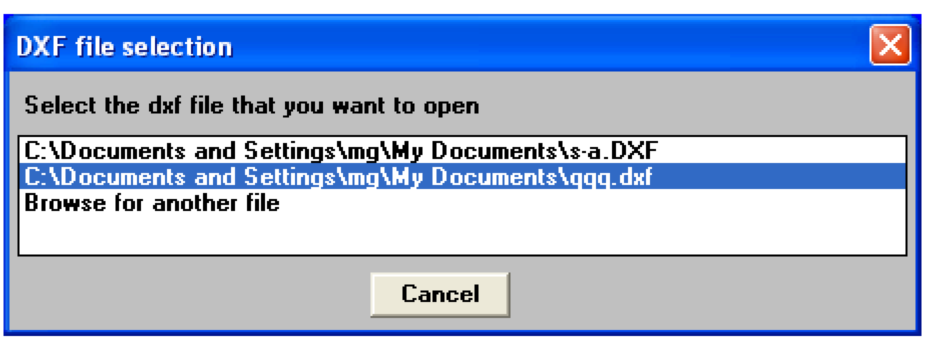
Display DXF

The DXF drawing may be temporarily removed from the background and later restored by clicking on the option.

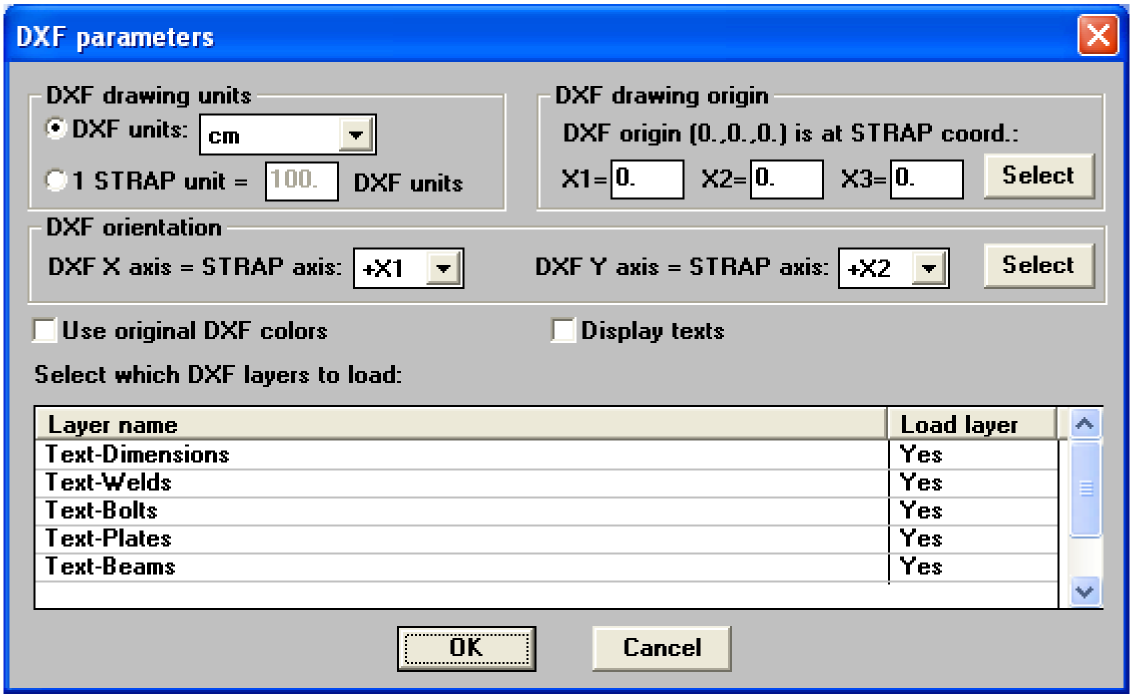
1.4.3.2.1 DXF - parameters

Select the DXF drawing from the list.

The program remembers the DXF drawings previously selected for this model. If more than two have been selected the program ask you which one to use or whether to browse to a different file:



The DXF background drawing parameters are defined when the drawing is loaded into the display and may be revised at any time.



DXF - units

The DXF drawing will be scaled according to the *STRAP* geometry units. Two options are available:

DXF units:
Select a unit from the list box

1 STRAP = x DXF units
Select the ratio between the units (the DXF dimensions will be divided by the value entered here). For example, *STRAP* units are meter, the DXF units are feet, but you want to double the size of the drawing in the background: Enter 3.281/2 = 1.6405

DXF - origin & orientation

Specify:

- the location of the DXF (0,0,0) coordinate on the *STRAP* drawing:

  Specify the global axis coordinates or click Select and select any existing node
- the orientation of the DXF X and Y axes relative to the model axes:
  - Select *STRAP* global axes (Note that the DXF drawing may be inverted by selecting **-Xn** as the global axis), or -
  - Select three nodes to define the directions:
    the first two nodes selected define the orientation of the DXF X axis; the third node defines the general direction of the Y axis

DXF - use original colors

☑ display the DXF lines with the same colors as in the DXF drawing
☐ display all DXF lines with the color selected in Setup - Colors - General.

DXF - display text

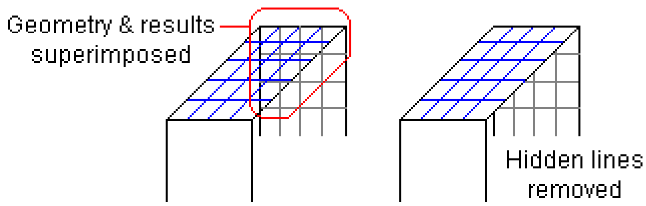
Display the text in the DXF drawing on the model. The text will be ignored when defining nodes.

DXF - layers

Only the selected layers (**Yes**) will be drawn in the background.

1.4.3.3 Hidden lines

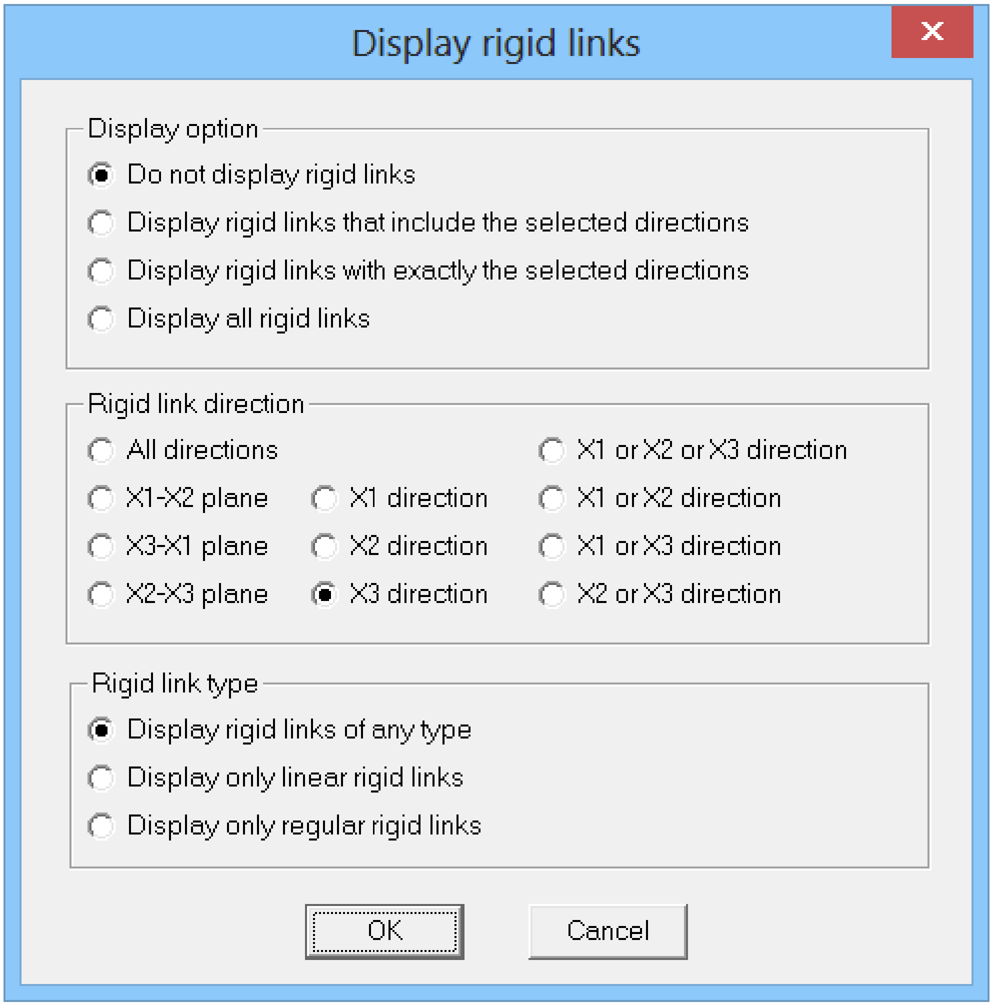
For finite elements only, delete background lines from the display to improve clarity. For example:



Note that displaying large complicated models with hidden lines may be relatively slow.

1.4.3.4 Rigid links

Display groups of nodes connected by "rigid links". Select the type of rigid link (as defined in **Restraints**):



The program displays "**Rnnn**" alongside nodes defined as part of a rigid link group, where "**nnn**" = the lowest node number in the group.

Display option

Specify which of the rigid links selected in the following "Rigid link direction option" to display:

Display rigid links that include the selected directions
Display any rigid link that includes ***any*** of the selected directions. For example, if you select **X2 direction**, the link will be displayed on beams with rigid links in the **X1-X2 plane**, the **X2-X3 plane** and in the **X2 direction**.

Display rigid links with exactly the selected directions
Display only the rigid links with the exact direction selected. Note that if you select any of the **Xn or Xn** options, rigid links in a plane will not be displayed.

Rigid link direction

Select any direction or any combination of directions.

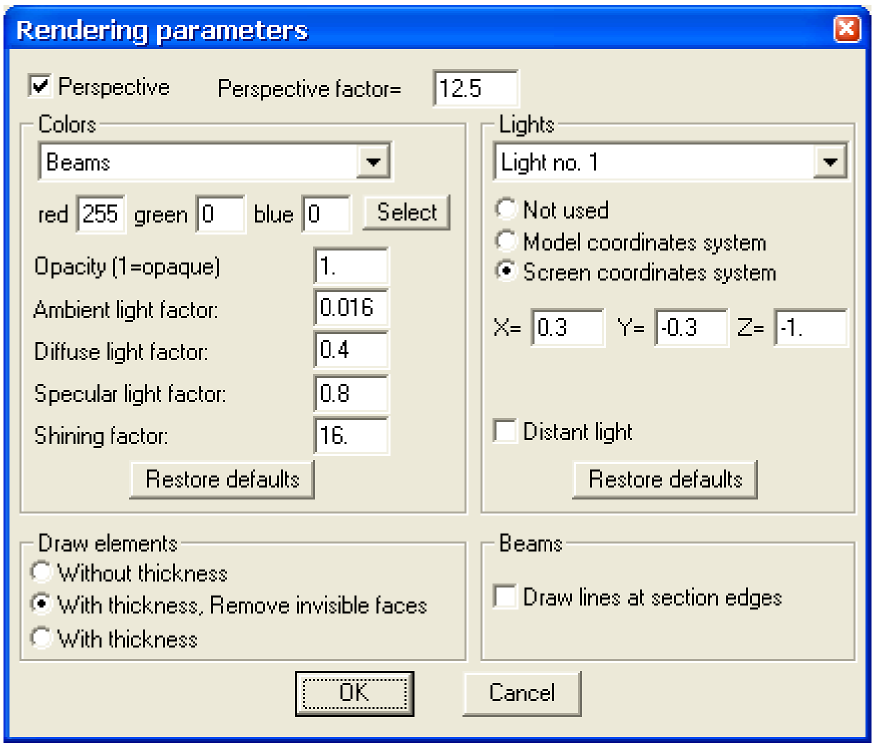
Rigid link type

Display regular rigid links, linear rigid links, or both.

1.4.3.5 Render

The rendering option enables you to display the current model with perspective, to remove hidden lines, to draw the beams and elements with their natural shape and to simulate shading effects caused by a light source.

The rendered model is displayed according to the values specified in the **Rendering parameters** option. The default values in this option have been selected after extensive testing and give good results for most models. The default values may be restored at any time by clicking the Restore defaults button.



Perspective

The **perspective factor** defines the distance of the viewer from the model. The distance is inversely proportional to the factor, i.e. when the factor is increased the viewer moves closer to the model.

- ☑ Display the model with the specified perspective factor.
- ☐ Display the model without perspective, i.e. viewed from an infinite distance (factor = 0)

Colours

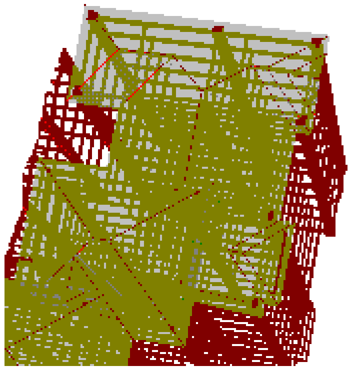
Define the base colour for each of the element types or property groups.

Click the Select button to select the colour from the palette or specify the red/green/blue factors (0-255).

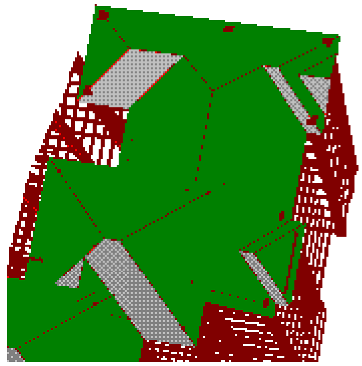
Note:
The colour selected will apply only to the element/property group currently displayed in the first line.

Colours may be specified for each property group or for all beams and elements.

- the property group colours will be used only if **Property by colour** is set to ✓ in the Display menu.
- otherwise the colour specified for **Beams** will be used for all beams in the model and the colour specified for **Elements** will be used for all elements.
- The opacity (transparency) can be defined for each colour, where opacity = 1 indicates zero transparency. Defining a degree of transparency for elements enables the structure behind them to be viewed. For example:



Opacity (1=opaque) 0.4



Opacity (1=opaque) 1.

Lights

Up to six different light sources may be defined. All "used" sources will shine simultaneously on the model.

Select a light source from the list box and specify its parameters:

◉ **Not used**
The light source will not be applied to the model

◉ **Model coordinate system**
The light source will rotate with the model. The origin of the system is defined as follows: X=0 at X1min, Y=0 at X2min, Z=0 at X3min.

◉ **Screen coordinate system**
The light source will remain at a constant screen location when the model is rotated. The origin of the system is defined as follows: X=0, Y=0 at screen centre; Z=0 behind the model.

X/Y/Z = define the coordinates of the light source.
Distant light: the light shines from infinity along the vector defined by **X/Y/Z=.**

Colour factors

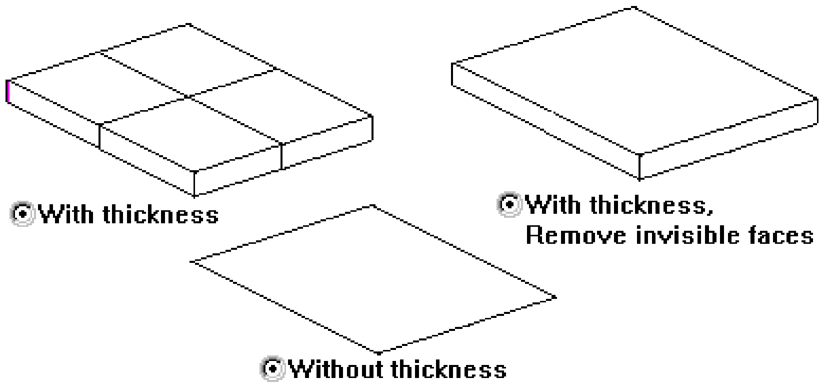
The colour displayed by the program on a surface is the defined base colour for the element type/ property group modified by factors that define the way light from the sources is reflected from the surface. For example, consider a dome built from single material of uniform colour with light shining on it from a single source. To a person viewing the dome the side facing the light source will be perceived as being lighter, i.e,.the dome will not actually appear to have a uniform colour.

There are four factors that modify the base colour:

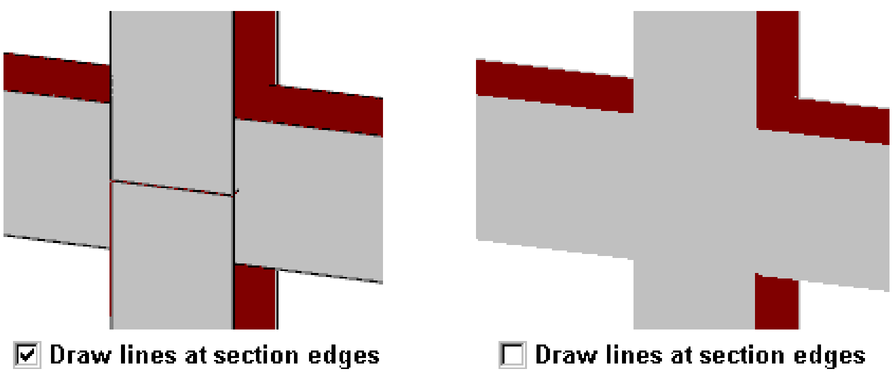
Ambient a light that provides constant illumination on every surface and is not dependent on the location of the light sources. If ambient light is the only light reflected then all surfaces would have the identical colour.

Diffuse light that is a function of the angle between the light source and the surface.
Specular A parameter that defines the reflection properties of the surface. Increasing the value makes the surface look wetter or shinier
Shining Similar to "specular"; decreasing the value makes the surface look wetter or shinier.

Elements



Beam edges



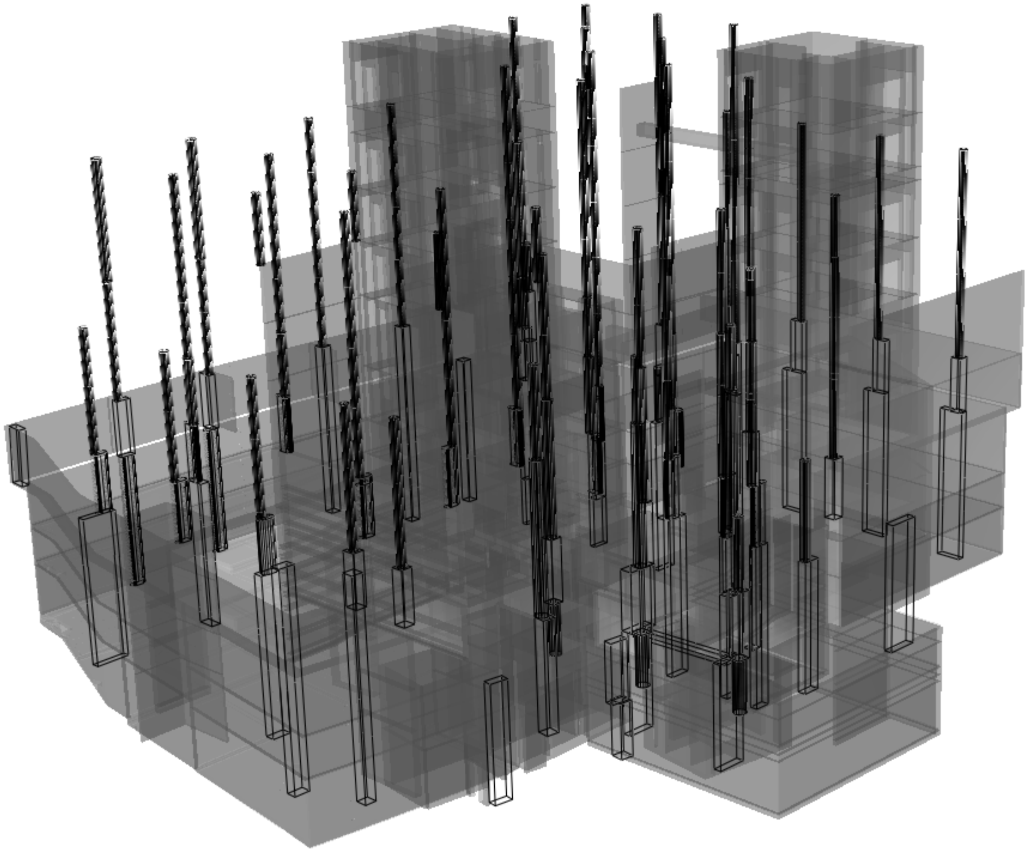
Restore

Restore the default values for all options in this menu.

1.4.3.6 Wireframe

Draw a wireframe image of the model.
- Beams and walls: only the lines forming the sections are drawn:
- Elements: Surfaces are drawn with only the contour lines (individual elements are not displayed). The resulting surfaces are drawn semi-transparent.

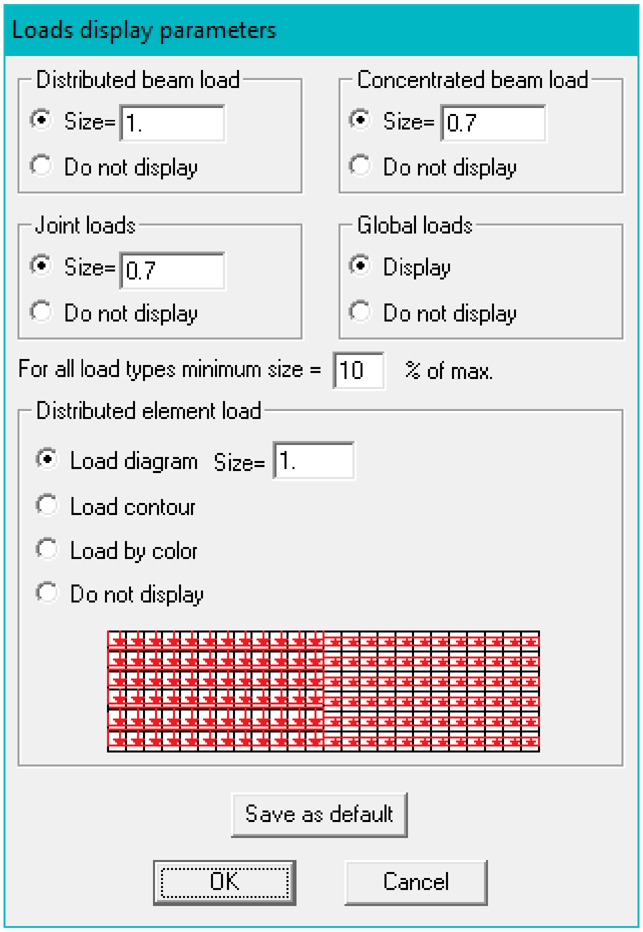
For example:



Note:

- Once "wireframe" is selected, subsequently clicking the render icon will display the wireframe. To restore the rendered drawing to the option, select "Render" in the Display menu.

1.4.3.7 Loads



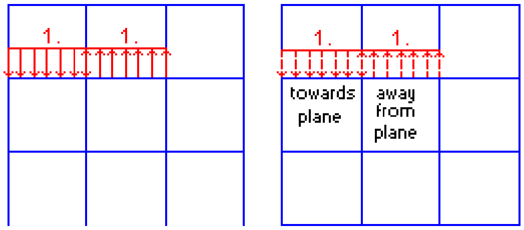
Load arrow length is proportional to the load magnitude. Specify:
- the arrow length for maximum load of each type
- the minimum arrow length for all types

In addition, each of the load types may be removed from the display

The program draws the loads according to the following criteria:

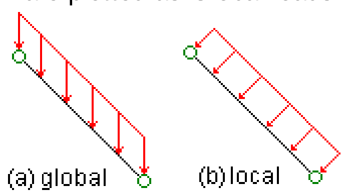
Beam Loads - distributed and concentrated

- The program arranges the loads as follows:
- all loads are displayed separately.
- all loads are displayed in the order of definition.
- concentrated loads are displayed above all other loads.
- loads are always plotted to the left/above the beam; if the sign of the load changes, only the direction of the arrow is reversed (Figure a).
- loads acting in the plane of the screen are displayed with solid lines; loads acting perpendicular to the plane of the screen are displayed with dashed lines (Figure b).



(a) frame (loads in plane) (b) grid (loads perpendicular to plane)

- If the angle between the direction of the load and the plane of the screen is less than 45°, the loads are displayed as if they are acting in the plane of the screen, i.e. with solid lines. Loads with an angle to the plane of the screen greater than 45° are displayed with dashed lines.
- arrow conventions are the same for distributed loads and concentrated loads.
- for Local / Global loads, the arrows always point in the direction of the loads. Global Projected loads are plotted as Global loads.

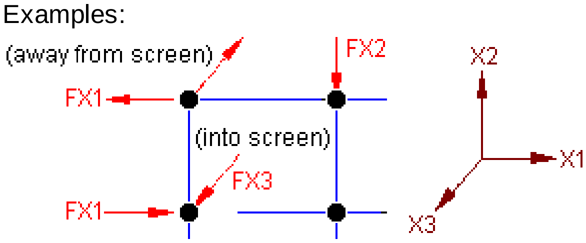


Note:
- The display of loads on elements in rotated models may be confusing. It is recommended that loads be displayed only for elements in planes that are oriented parallel to the screen.

Joint Loads

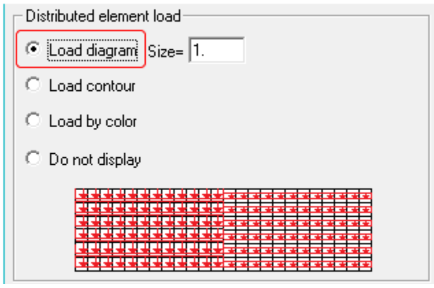
- The ***summation*** of the loads at a node are displayed
- Loads acting perpendicular to the plane of the screen are drawn at a 45° angle. This is convenient for grids but may be confusing for space frames.

Examples:

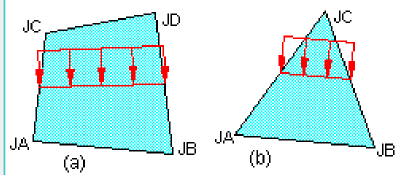


Distributed element loads

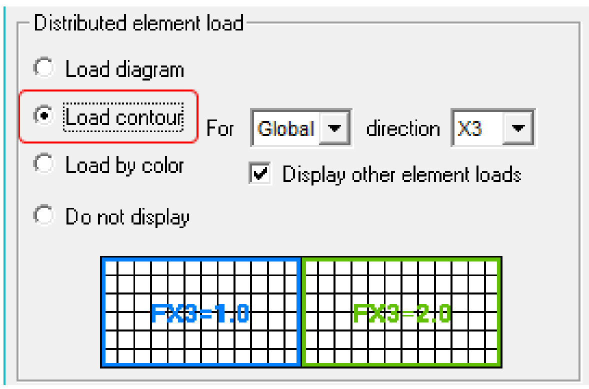
There are three options for displaying distributed element loads:

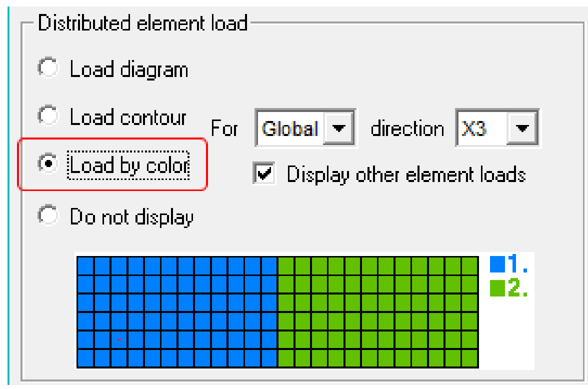


The load is displayed as a linear load along a line through the element centre. For example:



- loads acting perpendicular to the screen plane are displayed with solid lines; loads acting in the plane of the screen are displayed with dashed lines.
- If the angle between the load direction and the screen plane is less than 45°, the loads are displayed with dashed lines. Loads with an angle greater than 45° are displayed with solid lines.

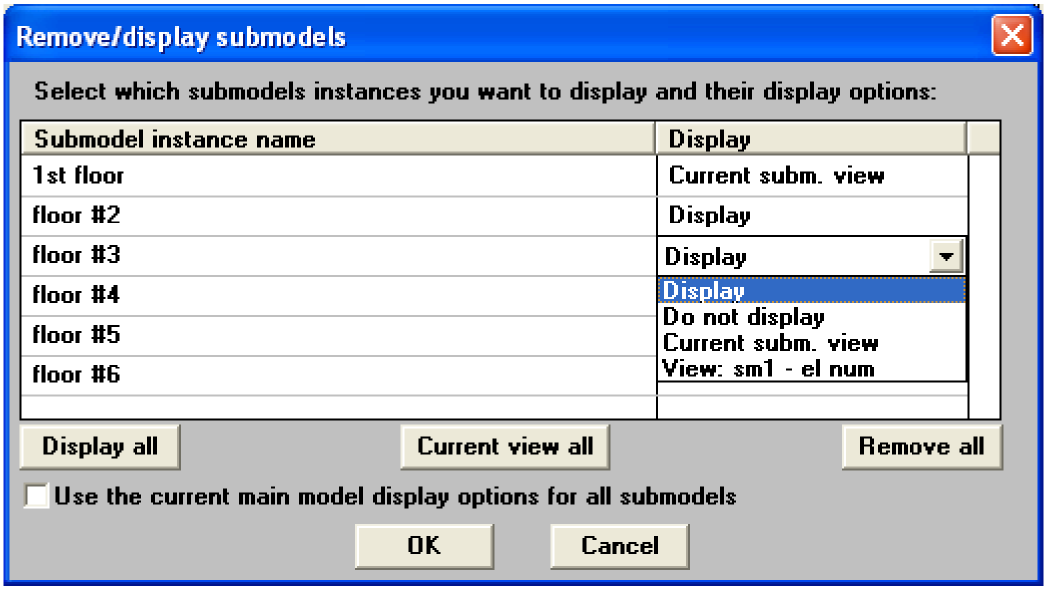




- Loads in only one selected direction can be displayed
- Other element loads (e.g. temperature loads) can be superimposed onto the contour/color display (in the "load diagram" format).

1.4.3.8 Submodel instances

Submodel instances can be added or removed from the ***Main model*** display:

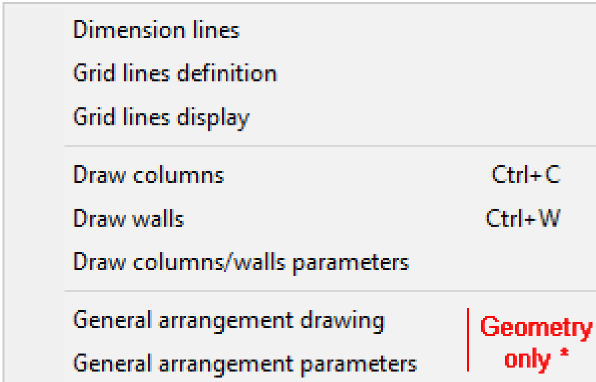


- click and highlight the line with the submodel instance
- click on the ▼ in the **Display** column and select:

Display	-	display the instance (with default minimum display options)
Do not display	-	remove the instance from the main model display
Current subm. view	-	display the instance with the same display options (i.e. numbering) currently specified for the instance.
View: ,,,	-	the program adds to the end of the list all views saved when the submodel was displayed; select one - the instance is shown with the same display options (i.e. numbering) saved in the view.

- To display ***all*** instances with the current ***main model*** display options, set ☑ **Use the current main model display options for all submodels**
- click [Display all] to set the option to **Display** for all instances
- click [Remove all] to set the option to **Do not display** for all instances
- click [Current view all] to set the option to **Current subm. view** for all instances

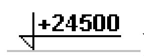
1.4.4 Draw



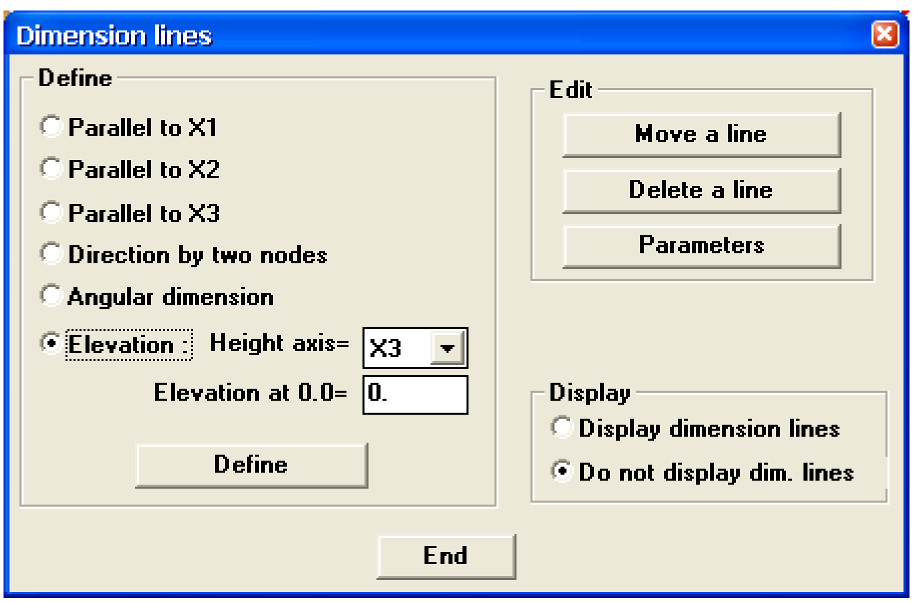
* for "General arrangement drawing" options in the Steel and Concrete modules, refer to:

- Steel - Display options
- Concrete - Display options

1.4.4.1 Dimension lines

Add dimension lines or elevation marks (e.g. ) to the drawing:

- arrow style, text style and number of digits may be specified. Refer to Parameters 74
- dimension lines defined with this option are saved in Views



Select one of the following options:

Parallel to X1/X2/X3

- Select one of these options to plot the dimension line parallel to a global axis.
- click Define
- select the nodes defining the dimension lines using the standard node selection option and then

move the cursor to the dimension line location.
- move the cursor to the line location and click the mouse

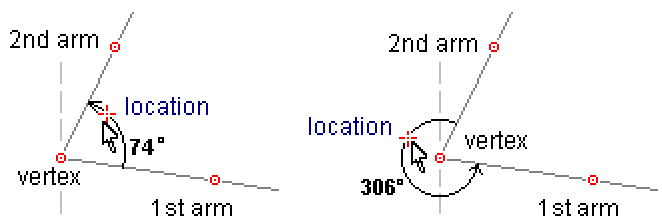
Defined by 2 nodes:

- click Define
- Select two nodes defining the dimension line direction; the dimension line will be drawn parallel to a line connecting two selected nodes.
- select the nodes defining the dimension lines using the standard node selection option and then move the cursor to the dimension line location.
- move the cursor to the line location and click the mouse

Angular dimension

Display the angle between two connecting lines. The lines are defined by selecting nodes - one at the vertex, and one on each of the arms.

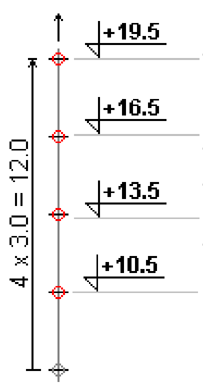
There are two possible angles that can be drawn for each selection. The user must specify the line location. For example:



Note that the arc is drawn through the 'location' point.

Elevation

Display elevation marks along any of the global axes at selected levels. For example:



- Select the model height axis and specify the elevation at the model 0.0 coordinate of that axis
- click Define
- Select the nodes at the levels that require a section mark using the standard node selection options
- move the cursor to the elevation mark location (perpendicular to the height axis) and click the mouse

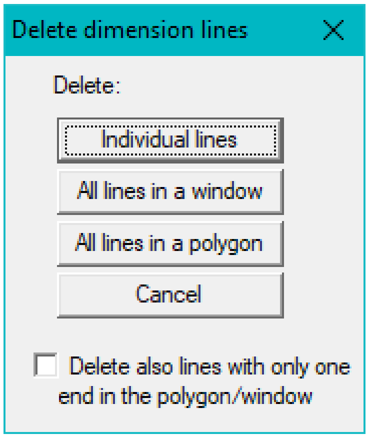
Note that the arrow and text dimensions can be revised in the Parameters 74 option.

Move

Move a dimension line or elevation line from the model
- Highlight a dimension line or elevation line and click the mouse
- Move the to the new location and click the mouse.

Delete

Delete one or more dimension lines or elevation lines from the model



Refer to Beam selection 27 for an explanation of the options.

Display

All dimension lines/elevations defined for this model may be temporarily deleted from the display.

Parameters

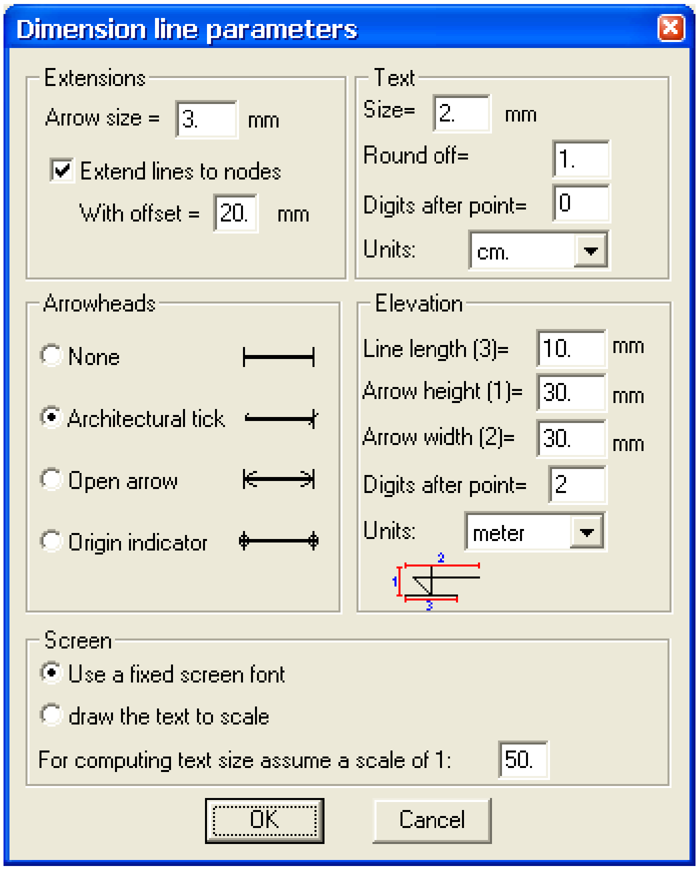
Refer to - Dimension line - parameters 74

Note:

- the dimension and elevation lines will be erased if the display is rotated or a different plane is selected. To retain the lines, save the current View.

1.4.4.1.1 Parameters

Specify the parameters for dimension lines and elevation lines. Note that any changes to the parameters will also revise existing dimension lines.



Extensions

Note:

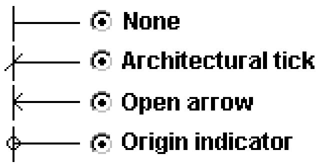
- "arrow size" is revised for the print options only and will remain constant on the screen.
- "arrow size" dimension affects all extension types.

Text

- **Round off**
  Round off all dimensions and elevations to the value specified here
- **Digits after point**
  Specify the number of digits to be displayed after the decimal point. Note that this number of digits will always be displayed, even if the **Round off** value requires more digits.

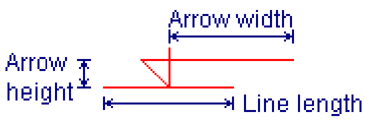
Arrowheads

Select one of the following arrowhead types:



Elevation

Specify the elevation mark parameters and the units for the elevation value:



Screen

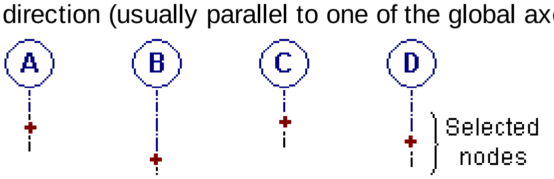
Select the method for displaying the dimension text on the screen:

- **Use a fixed screen font**
  draw the text using the screen font used for all other text, e.g. beam numbers, etc.
- **Draw the text to scale**
  Draw the text graphically according to the scale specified here. This is a preview of the printed output where the text is always printed with the **Size** specified at the top of the dialog box. Note that the text is not actually displayed with the specified size because the scaled drawing is 'squeezed' into the screen.

Note:
- The dimension line text is displayed 'drawn to scale' if ✓**General arrangement drawing** is selected in the **Display** menu; otherwise it is displayed using a fixed screen font.
- when the text is 'drawn to scale' it is a preview of the printed output where the text is always printed with the **Size** selected in the General arrangement parameters option. Note that the text will not actually be displayed with the specified Size because the scaled drawing is 'squeezed' into the screen.
- the text will be printed with the specified Size only if the scale selected when printing is identical to the scale selected in the General arrangement parameters option; otherwise sizes will be modified according to the ratio of the scales. For example, a scale of 1:50 is specified here but a scale of 1:100 is specified when printing: the actual text/arrow sizes will be one-half (50/100) of the sizes selected in this option.

1.4.4.2 Grid lines - define

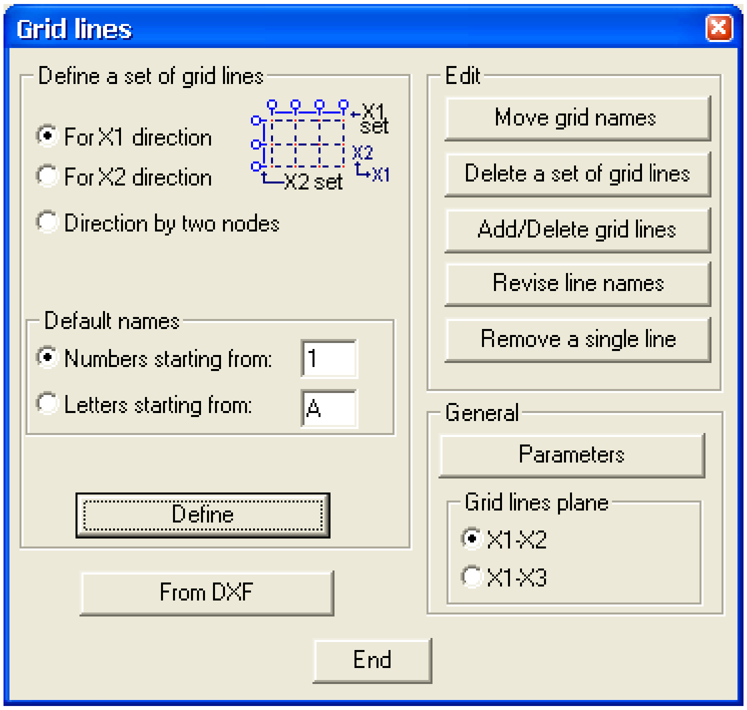
Define "grid lines" for the model; the program draws the grid lines through selected nodes in any direction (usually parallel to one of the global axes). For example:



- The grid lines may be defined only in the geometry module.
- The grid lines may be defined at existing node locations or they may be retrieved from a DXF file (the program creates at least one node for each retrieved grid line)
- The grid lines may then be added to the current display in geometry, loads, steel and concrete design

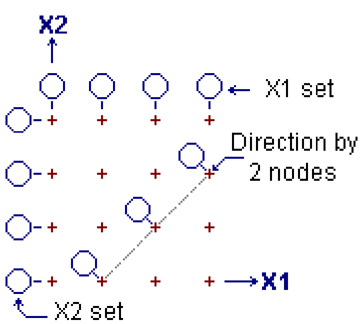
modules; the program displays only those grid lines relevant to the display, i.e. grid lines are not displayed for portions of the model that have been 'removed'.

- Grid lines can be saved in a 'View'.
- Submodels: Grid lines can be defined only for nodes in the Main model. The grid lines may however be added to the submodel display (according to the location of the first instance of the submodel).



Define a set of grid lines

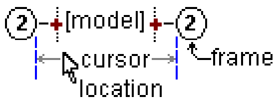
Define a set of grid lines. A set can be parallel to one of the global axes or may be parallel to a line defined by two nodes. The grid line symbols are drawn according to the relevant coordinate of selected nodes.



To define a grid line:

- specify the grid line parameters 79
- select the default name option (numbers or letters)
- select the grid lines global plane (X1-X2 or X1-X3). Note that revising the grid plane erases existing grid lines.
- select the direction and click Define.

- select the nodes using the standard node selection option
- move the to the grid line location and click the mouse.
  note that the side of the frame enclosing the name text closest to the model will be placed at the location, e.g.



- Revise the grid line names, invert the order, etc. Refer to Add/delete lines 78.

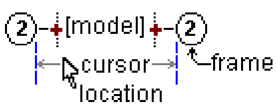
Default names

Default names are assigned automatically to the program when the grid lines are created, but may be revised afterwards.

Select numbers (1,2,3,4 ...) or upper-case letters (A,B,C, ...). The names are assigned in ascending order.

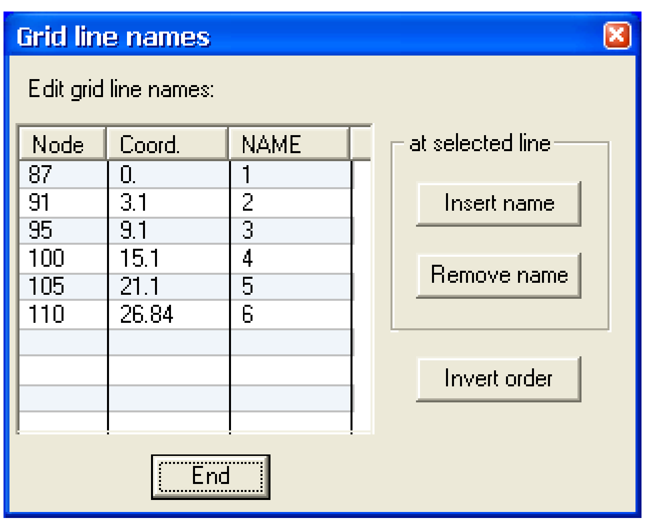
Move grid names / Delete a set

- highlight the grid line (the ■ will appear at the midpoint of the line connecting the two extreme grid marks) and click the mouse
- for **Move,** place the at the new grid line location and click the mouse. Note that the ***side*** of the frame closest to the model will be placed at the location



Add/delete grid lines / Revise names

Revise the names in the grid line:



Click on a name in the right-hand column and enter a new name, or select one of the following options:

Insert name

Insert a name at the current cursor location; the following names will be pushed down. For example, highlight the line with node 91 and click **Insert**; type in a new name (e.g. **2A**) in the empty cell. The list be now be **1,2, 2A, 3,4**

Remove name

Remove a name from the list; the following name will be pushed up and a new name will be automatically written in the last line. For example, highlight the line with node 95 and click **Remove**. The list be now be **1,2, 4, 5, 6**

Invert order

Invert the order of the current names in the list. In the above example, the list will be **6, 5, 4, 3, 2, 1** if this option is selected.

Remove single line

Erase a single grid line from a grid line set:

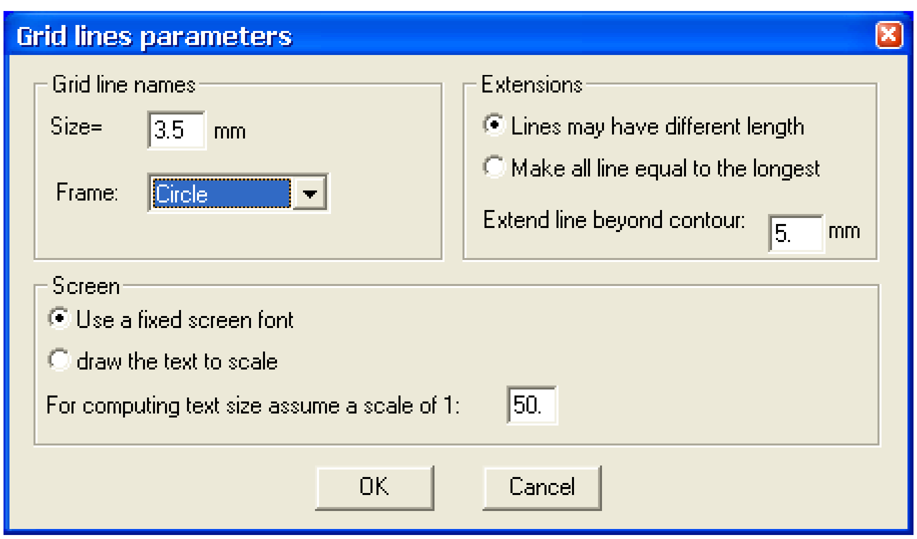
- highlight the grid line (the ■ will appear at the midpoint of the line connecting the two extreme grid marks) and click the mouse
- highlight the relevant line in the table and click OK.

From DXF

Retrieve the grid lines from a DXF file. Refer to Grid lines - DXF 168.

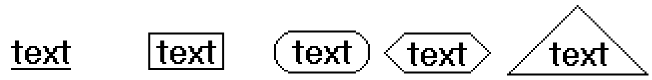
Parameters

Specify the parameters for grid lines. Note that any changes to the parameters will also revise existing grid lines.



Frame

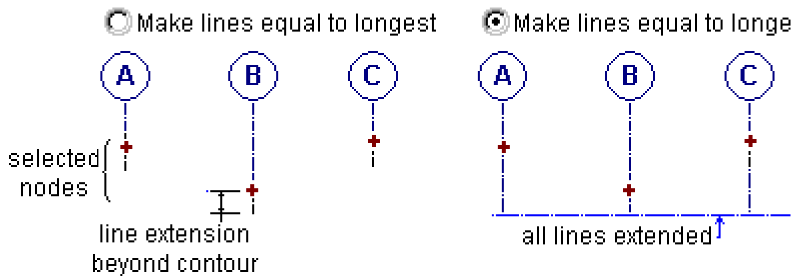
Select one of the following frame types:



Extensions

Grid lines are drawn from the specified location to the selected node and are extended beyond the node by the dimension specified here. If more than one node is selected, all of the lines may be drawn with the same length (equal to the longest).

For example:



Screen

Select the method for displaying the grid line text on the screen:

Use a fixed screen font
draw the text using the screen font used for all other text, e.g. beam numbers, etc.

Draw the text to scale
Draw the text graphically according to the scale specified here. This is a preview of the printed output where the text is always printed with the **Size** specified at the top of the dialog box. Note that the text will not actually be displayed with the specified Size because the scaled drawing is 'squeezed' into the screen.

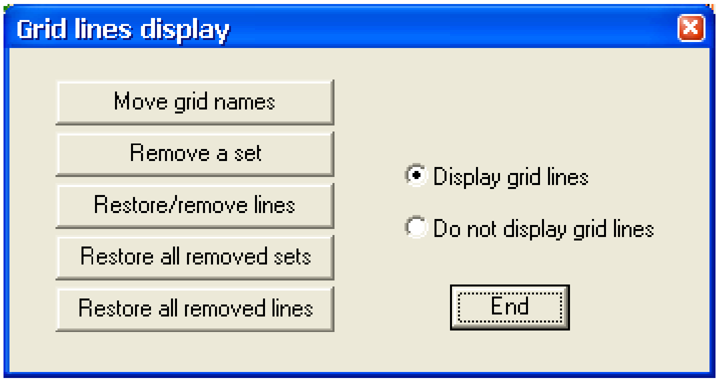
Note:

- The grid line text is displayed 'drawn to scale' if ✓**General arrangement drawing** is selected in the **Display** menu; otherwise it is displayed using a fixed screen font.
- when the text is 'drawn to scale' it is a preview of the printed output where the text is always printed with the **Size** selected in the General arrangement parameters option. Note that the text is not actually displayed with the specified Size because the scaled drawing is 'squeezed' into the screen.
- the text will be printed with the specified Size only if the scale selected when printing is identical to the scale selected in the General arrangement parameters option; otherwise sizes will be modified according to the ratio of the scales. For example, a scale of 1:50 is specified here but a scale of 1:100 is specified when printing: the actual text/frame sizes will be one-half (50/100) of the sizes selected in this option.

1.4.4.3 Grid lines - display

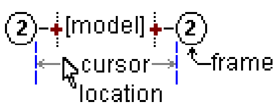
Add the grid lines (***defined in geometry***) to the current display:

- the program does not show grid lines that are not relevant to the current display, i.e. lines on sections of the model that have been 'removed' are not shown.
- Grid line sets or individual grid lines may be removed/restored on the current display; grid line sets may be relocated (the grid lines and their location as defined in geometry are not affected)
- The current grid line display is saved when a 'View' is created.
- if the current display is an existing 'View', then the View must be saved again in order to retain to changes to the Grid lines.



Move grid line / Remove a set

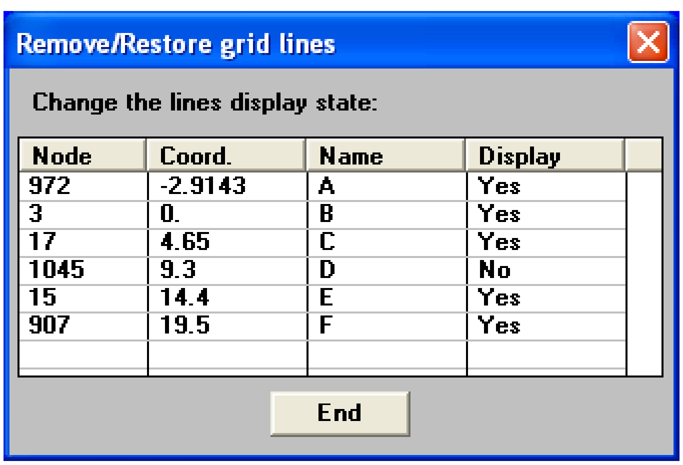
- highlight the grid line (the ■ will appear at the midpoint of the line connecting the two extreme grid marks) and click the mouse
- for **Move,** place the at the new grid line location and click the mouse. Note that the ***side*** of the frame closest to the model will be placed at the location.



Restore / remove lines

Individual lines may be removed from the display and later be restored:

- move the to the grid line set and click the mouse; the program display a list of grid lines in the set:



- move the to the appropriate row and click the mouse to toggle the 'Display' to **Yes/No**.

Restore all removed sets / lines

Restore ***all*** grid lines/sets that were removed from the current display using the 'Remove' options in this dialog box.

Note:

- if the current display is a 'View', then the View must be saved again in order to retain to changes to the Grid lines.

Display / Do not display

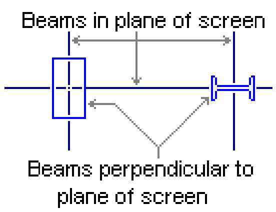
Display grid lines
Select this option to add the grid lines (defined in geometry) to the display, then select the other options in this menu to modify them (remove lines or a set of lines; move names)

Do not display grid lines
Delete the grid lines from the display. The lines may be restored at any time (any modifications must be saved in a 'View' to be retained.

1.4.4.4 Draw columns / walls

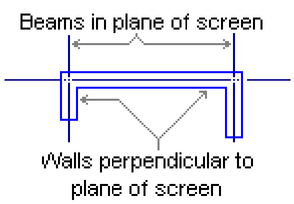
Draw columns

Draw the section of any beam elements that lie ***intersect the plane of the screen*** (not necessarily 'columns'). For example:



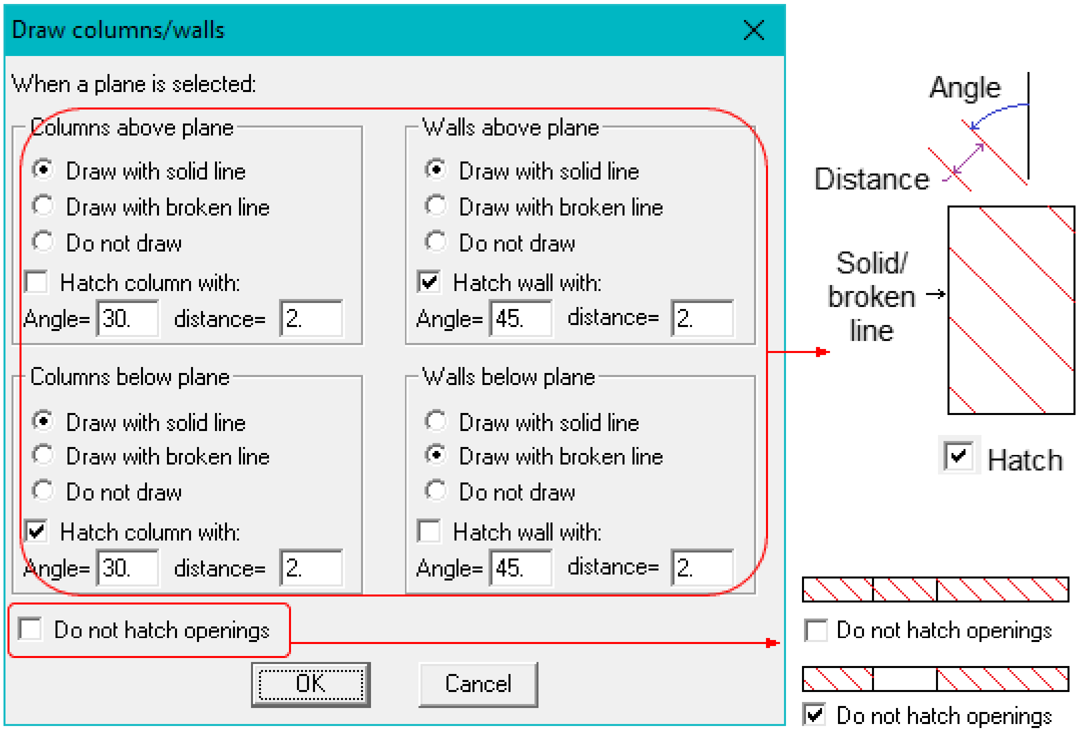
Draw walls

Draw the section of any wall elements that lie ***intersect the plane of the screen***. For example:



Draw column/wall parameters

Different parameters may be selected for the column/wall detail drawn above/below the selected plane:

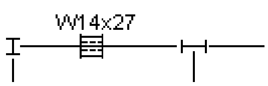


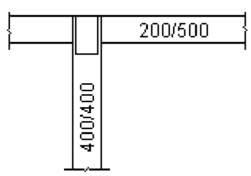
Note:

- parameters selected for any existing model will be used as the default parameters for subsequent new models.

1.4.4.5 General arrangement drawing

Create a "General arrangement" drawing for the model. For example:

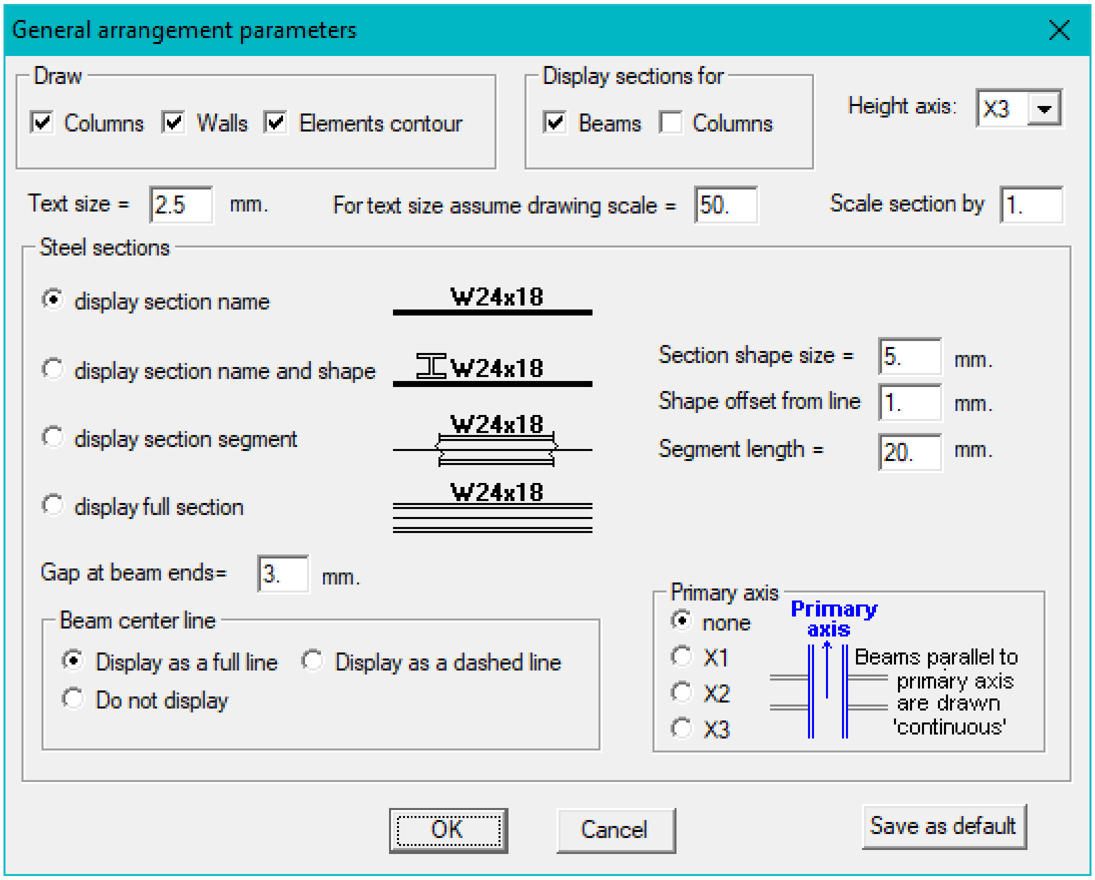
Steel : 

Concrete: 

Note:

- steel and concrete sections are displayed together
- General Arrangement drawings should be requested only when a plane is displayed on the screen. The display will not be accurate for an isometric view.

The drawing parameters are:



General options:

Draw section

Draw beam and/or column section names (steel and concrete)

Draw column

Draw section of columns perpendicular to the plane.

Draw walls

Draw section of walls perpendicular to the plane.

Draw element contour

Draw the contour line surrounding elements in the plane (where there are no beams or walls).

Height axis

The axis perpendicular to the plane.

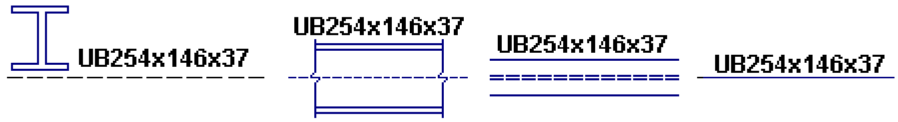
Draw column section name

Draw the column property group names adjacent to the columns.

Steel options:

Section name and shape

The section shape and name may be imposed on the beam line using one of the following four methods:



The name is written only once for a string of identical sections.

- The section by default is drawn according to the same scale as the drawing. Modify **Scale section by** to increase/decrease the size of the sections on the drawing.

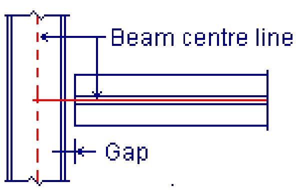
Drawing scale / text size

Specify the drawing scale and the text size. The drawing scale is required to determine the size of the text on the screen display.

Note that the text is printed with this size only if the scale specified when printing is the same as the scale specified in this option; otherwise the text size is modified according to the ratio of the scales. For example, a scale of 1:50 is specified here but a scale of 1:100 is specified when printing: the actual text size will be one-half (50/100) of the size selected in this option.

Beam center line

Specify the beam center line type and the "gap distance":



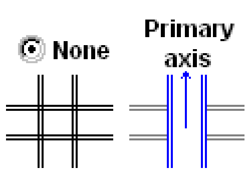
The centre line may be deleted when  **Display full section** is selected.

Primary axis

The program differentiates between primary and secondary beams and terminates the secondary beams at the intersection with the primary beams that are drawn as continuous. By default, beams with releases are always secondary beams

Similarly if a "column" (a member perpendicular to the plane) is drawn at an intersection, then all members on the plane connected to that intersection are terminated.

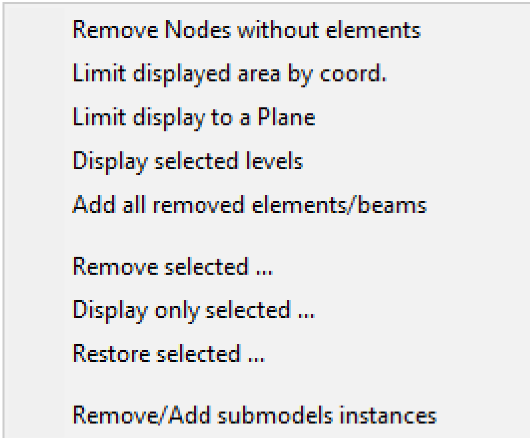
For all other intersections, specify the axis that has continuous members:



If **None** is selected, all beams are terminated at the intersection points.

1.4.5 Remove

For clarity, specified nodes, beams or elements may be temporarily removed from the display:



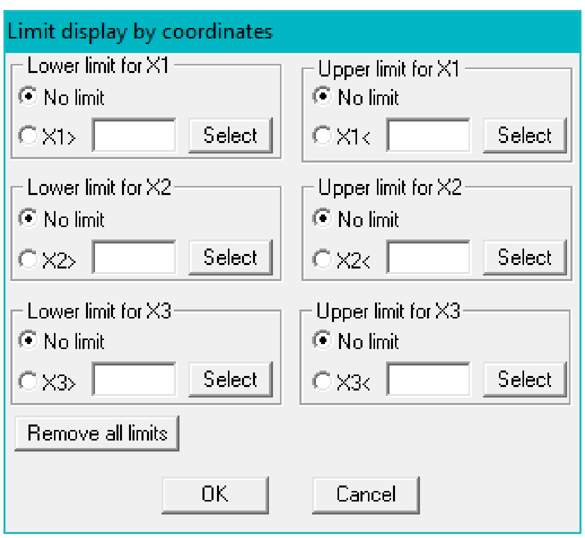
Remove nodes without elements

The default option is **NO**, i.e. nodes which have no beams or elements connected to them are displayed. If, for example, such a node is used as a JC node, revise the option to ✓ to delete it from the display.

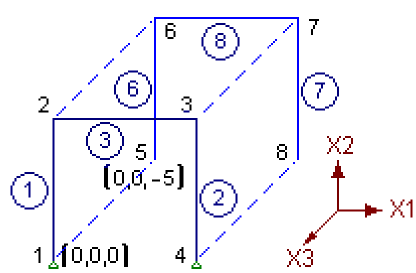
Limit display area by coordinates

This option enables the definition of "slices" through the model. In space structures, use this option to view individual planes.

Limit the minimum/maximum coordinates to be displayed for each global axis; the coordinate value may be typed in or click Select and select an existing node with the same coordinate:



Example:
In the model shown below, the initial display of the X1-X2 plane will superimpose the results of the planes at X3 = 0. and X3 = 5. To view the results on beams at X3 = 5.0 only, define **Lower limit for X3 = 4.9**



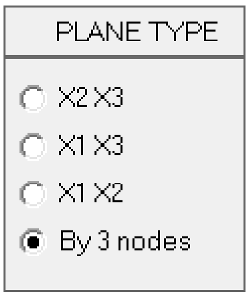
Remove all limits restores all six options to **No limit**

Limit display to a plane

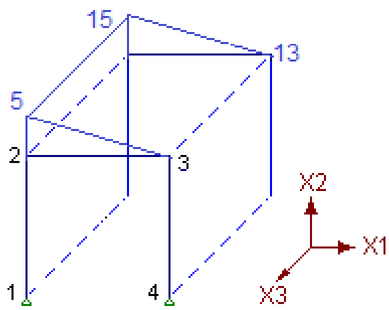
Define a plane parallel to a global plane or in any direction through the model by selecting three nodes.

Only beams located on the plane formed by these three nodes (extending to infinity in all directions) will be displayed.
Note:
- the plane may be defined as a 'slice' extending through the model by specifying a 'tolerance' distance (see below).
- The display is automatically rotated to the selected plane.

Select one of the following options:

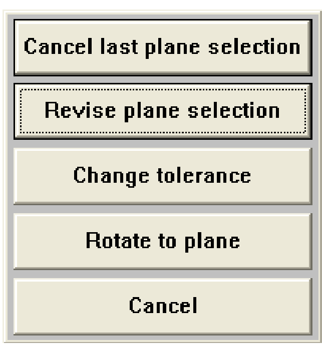


Examples:



- to display the front face - specify **X1-X2** and select any one of nodes 1 to 5.
- to display the sloped roof - specify **By 3 nodes** and select any 3 nodes on the roof,e.g. 3, 5, 15.

If the display has already been limited to a plane and this option is selected again, the following menu is displayed:



- **Change tolerance**
  The program by default displays only nodes located within 0.01 units from the defined plane. Use this option to specify a different tolerance value in order to display nodes that are offset from the defined plane.

- **Rotate to plane**
  Rotate the displayed plane to the screen plane.

Display selected levels

Refer to - Select nodes by level 25.

Add all removed beams/elements/walls/solids

Restore all removed beams/elements/walls to the display.

Remove selected beam/element/wall/solids

Select beams/elements/walls using the standard selection options; the selected items will be deleted from the display

Display only selected beams/elements/walls/solids

Select beams/elements/walls using the standard selection options; only the selected items will be displayed.

Restore selected beams/elements/walls/solids

Restore some of the beams elements that were previously removed.

- the program redraws the model, displaying only the removed beams/elements
- select beams/elements to restore using the standard element selection option

Remove/add submodel instances

Refer to Display - submodel instances 71.

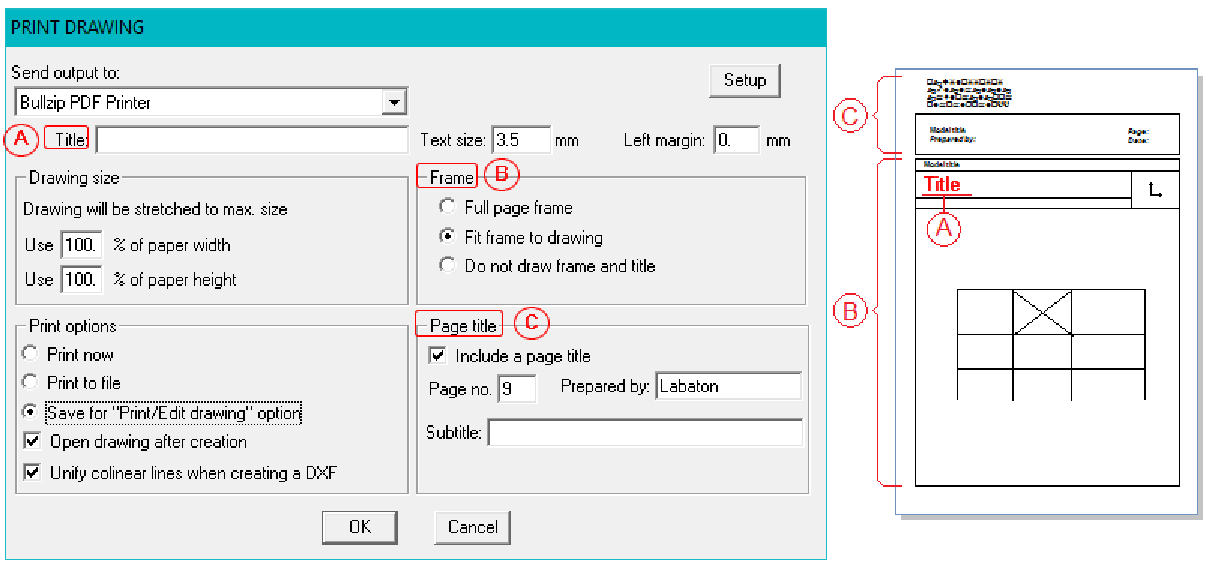
1.5 Print options

Select one of the following options:

1.5.1 Print drawing

Use this option to print the current display directly to the printer or to a file. The printed display will be enclosed in a frame and will include a header.

If the drawing is "rendered", refer to Print rendered drawing 91.



Send output to

Select the output unit, e.g. printer, plotter, etc. The devices must be installed by the "Printers" option in the Windows "Control panel".

Note:
- to create a DXF file, select the Create a DXF file 94 option.

Setup

Specify general information for the output device selected:
- paper size
- graphic resolution
- black & white/colour

Title

Enter the name of a title that will be printed at the top of the drawing.

Text size

Specify the text size in mm. The size is used for beam numbers, node numbers, load and result values, etc.

Left margin

Specify the left margin width. The program recalculates the default scale or the number of pages required whenever a new margin value is entered.

The value is added to the default margin for the printer/page size.

Drawing size

Specify the scale for the drawing. The program initially assumes that the drawing will fit exactly on one page (based on the paper size in the Setup option), calculates the corresponding scale and displays it as the default scale.

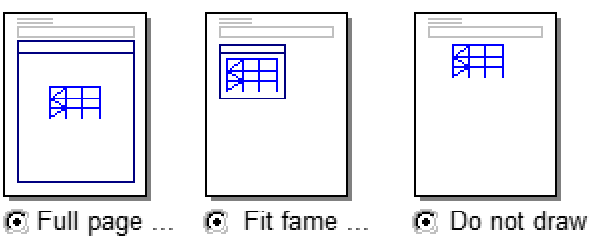
There are two methods for changing the drawing size:

- specify the scale
- specify the percentage of paper width/height to be used. Note that the program always maintains the vertical/horizontal drawing ratio,so it is sufficient to revise only one of the percentages.

If a scale larger than the default scale is specified, the drawing will be automatically printed on several pages which can then be pasted together.

Frame

Specify the location and size of the drawing and its frame:



Do not draw ... is the option that should be selected when saving a drawing for a "report"

Print options

- **Print now**
  Print the current drawing immediately
- **Print to file**
  Send the drawing to a file (not required if **Metafile** is selected in **Send output to**:). The program will prompt for a file name. Note that these drawings ***cannot*** be edited using the **Print/edit saved drawing** option.
- **Save for "Print/edit drawing" option**
  Save the drawing so that it can be edited and printed using the Print/edit saved drawing option.

These drawings can only be printed by *STRAP* and not by other programs or utilities.

☑ **Open drawing after creation**

Jump immediately to the Print/edit saved drawing option after clicking OK and display/edit the new drawing.

☑ **Unify colinear lines when creating a DXF**

If you want to create a DXF drawing in the Print/edit saved drawing option: check this option to create single lines in the DXF file from all series of colinear lines, e.g. short beam segments attached to elements.

Header options

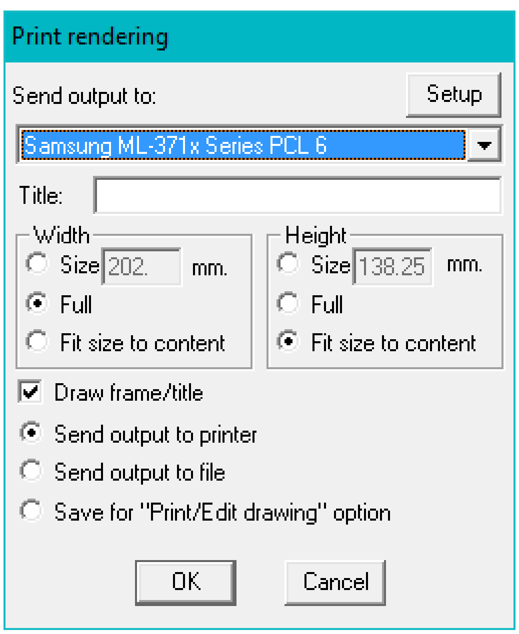
☑ **Include a page title**

Add a header block above the drawing, similar to the header block for tables, with the following information:

- **First page no:** page numbering will be consecutive
- **Prepared by:** designer's name
- **Subtitle:** secondary header

1.5.2 Print rendered drawing

Use this option to print the current rendered display directly to the printer or to a file. The printed display will be enclosed in a frame and will include a header.



Width / height

Select the height and width of the printed drawing. Note that Windows will distort the image if you do not maintain the original proportions.

Select one of the following options for width and/or height:

- **Size** Specify the actual dimension in millimeters
- **Full** The drawing will fit exactly into the page width height.
- **Fit size** The program will automatically calculate the dimension required to maintain the original drawing proportion.

Note:
- to maintain the original proportions, adjust either the width or the height while specifying **Fit size to content** for the other dimension.

Draw frame/title

Remove the frame and title if saving for a "report"; refer to Print drawing 89.

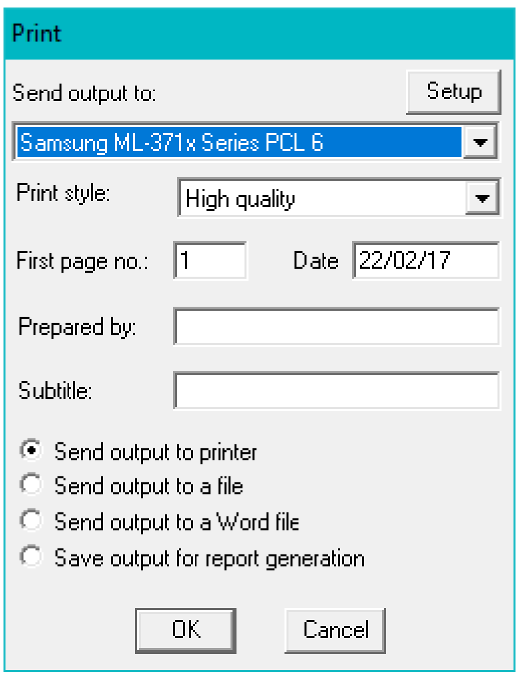
Send output to file/printer ; Save for ....

Send the drawing to the printer, a file or save for the "Print/edit drawing" option.
For "Send output to file":
- The program will prompt for a file name.
- Select "Send output to printer" if **Metafile** is selected in **Send output to**:

Note:
- for all other options, refer to Print drawing 89.

1.5.3 Print tables

The following menu appears when you request to print tables directly to the printer or to a file.



Send output to

Select the output unit, e.g. printer, plotter, etc. The devices must be installed by the "Printers" option in the Windows "Control panel".

Note:
- the printer must be specified even when the output is sent to a Word file or saved for 'report generation' because information on page size, fonts, etc. is required.
- to create a 'delimited file' (data items separated by a delimiter character and not by spaces) suitable

for import by most spreadsheet programs, select **Delimited file** in this option. To specify the delimiter character, refer to STRAP.INI 96.
- Customized delimited files can be generated using the STBatch utility.

Print style

The format for *STRAP* tabular output may be specified by the user.

A list of the "styles" defined using the"Setup" Table print styles option will be displayed; each style contains information on the fonts, margins, lines and spacing to be used when printing the table. Select one of the predefined styles and the tables will be printed in the specified format.

Note:
- "Draft" style prints all data in Courier 10 cpi font without any vertical or horizontal lines. This style cannot be edited.

Setup

Specify general information for the output device selected:
- paper size
- graphic resolution
- black & white/colour

Header options

Define information that will be printed in the header at the top of every printed page:
- **First page no:** page numbering will be consecutive
- **Date:** the date format is specified in the Windows "Control panel"
- **Prepared by:** designer's name
- **Subtitle:** secondary header

Send output to printer/file/etc

Select one of the following:

Send output to printer
Print directly to the printer.

Send output to a file
** This option is currently not functional **

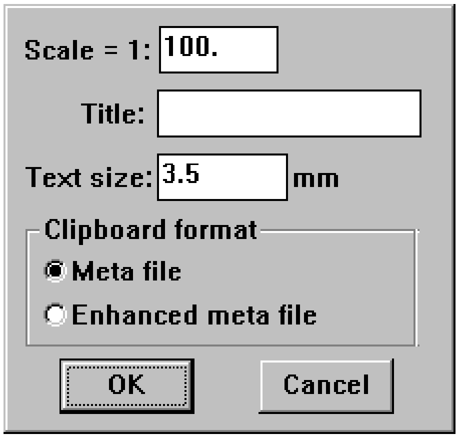
Send output to a Word file
The tables may be saved in a MS Word format file. The file is saved in RTF (Rich Text Format) and may be imported into any word processor or program that recognizes this format. Specify a name for the file. The default extension is .RTF

Save output for report generation
The tables are saved for report generation. Refer also to Saved tables management.

Note:
- a company 'logo' may be printed at the top of every page. Refer to Setup - Print parameters

1.5.4 Copy drawing

Use this option to copy the current display to the Windows "Clipboard". The display will be identical to that produced by Print drawing89, i.e. it will include a frame and a header. The file is transferred to the Clipboard in "Metafile" format.



Clipboard format

Select one of the following metafile formats:

Metafile
WMF format

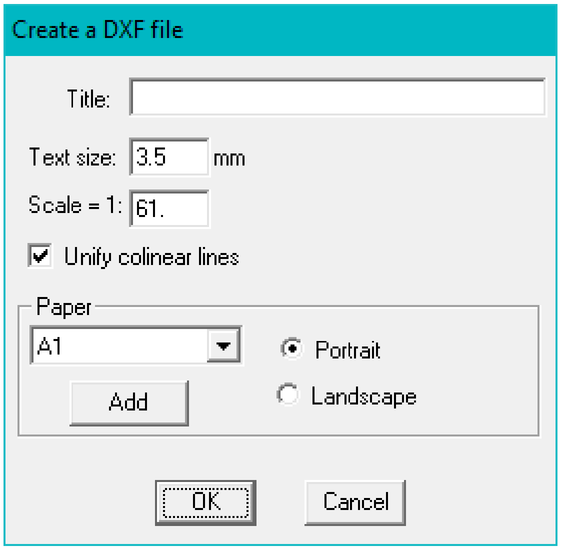
Enhanced Metafile
EMF format

Your graphics program, word processor, etc, may recognize only one of the above formats.

Click the OK button to copy the display. The display may then be retrieved by any Graphics program (Windows) with a **Paste** option (or equivalent).

1.5.5 Create a DXF file

Use this option to create a DXF file that includes the current display. The display will be enclosed in a frame and will include a header.



Title

Enter the name of a title that will be displayed at the top of the drawing.

Text size

Specify the text size in mm. The size is used for beam numbers, node numbers, load and result values, etc.

Scale

Specify the scale for the drawing. The program initially assumes that the drawing will fit exactly on one page (based on the paper size and orientation), calculates the corresponding scale and displays it as the default scale.

Note:

- the scale value may only be decreased as the program cannot divide the drawing to two or more pages (a warning is displayed when OK is clicked).
- the program always creates the drawing in centimeters, i.e. an item 2 meters long drawn at a scale 1:50 will have a length of 4. in the drafting program.

Unify colinear lines

Create a single line in the DXF file from a series of colinear lines, e.g. short beam segments attached to elements.

Paper

Select a paper size from the list and specify the orientation (Portrait or Landscape). Click Add to define a new paper size.

1.5.6 Print order

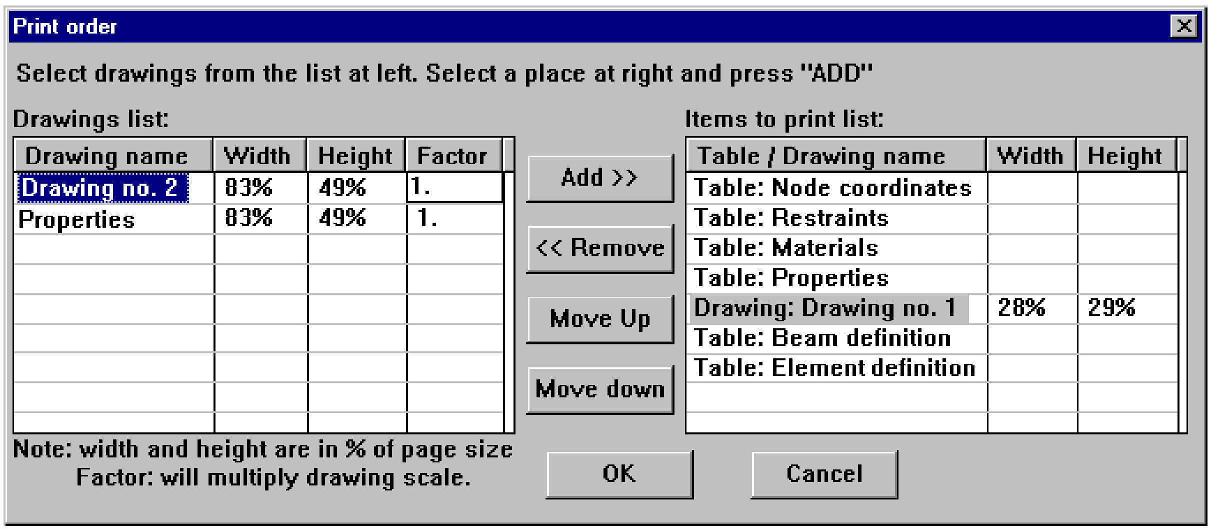
Arrange the order of the printout. Initially, all tables selected in the previous dialog box are displayed in the right list ("Print list"); all drawings are displayed in the left list ("Drawing List") and must be added to the Print List.

To add drawings:

- revise the size of a drawing by entering a "factor" value; the width/height values will be revised.
- Click on a drawing in the Drawing List and highlight it
- Click on the line in the Print List where you want to insert the drawing; click the Add >> button.
- Repeat for additional drawings

To change the order of the Print List:

- click and highlight a table or drawing. Click the Move Up Move down buttons to move the table/ drawing up or down in the list.
- click the << Remove button to remove a drawing from the list



Click OK to start printing.

1.5.7 STRAP.INI - print options

When a user revises a Setup parameter, the program writes the new value in the Windows "Registry". Each user is allocated a separate section in the Registry, so each user can create his own personal set of Setup parameters.

Note that the *STRAP* installation program does not write any Setup parameter values in the "Registry". Default values for *STRAP* program parameters are initially written in the file **STRAP.INI,** located in the program directory. Note that there is only one STRAP.INI file for all users.

The program initially searches for default values in the Registry; if it does not find a value (i.e. the user has not revised the parameter in Setup) it takes the default value from STRAP.INI.

The following options cannot be revised by the program Setup option. Change their default value by editing **STRAP.INI** using any standard line editor program:

[MISC]

SectionUnit=	Default unit for section dimension definition Enter: **mm**, **cm**, **meter**, **inch**, **feet**
GeoLineType=	Graphic results - geometry line type Enter: **SOLID** or **DASHED**
DispLineType=	Graphic results - deflections line type

	Enter: **SOLID** or **DASHED**
DiagramLineType=	Graphic results - result diagram line type Enter: **SOLID** or **DASHED**
Delimiter=	Delimiter character when"**Delimited file**" tabular output is selected. Enter: character ASCII number, e.g. "**,**" (comma) = 44
DelimiterTitle=	Print table headers when "**Delimited file**" tabular output is selected. Enter: **TRUE** or **FALSE**

Part II

STRAP main menu

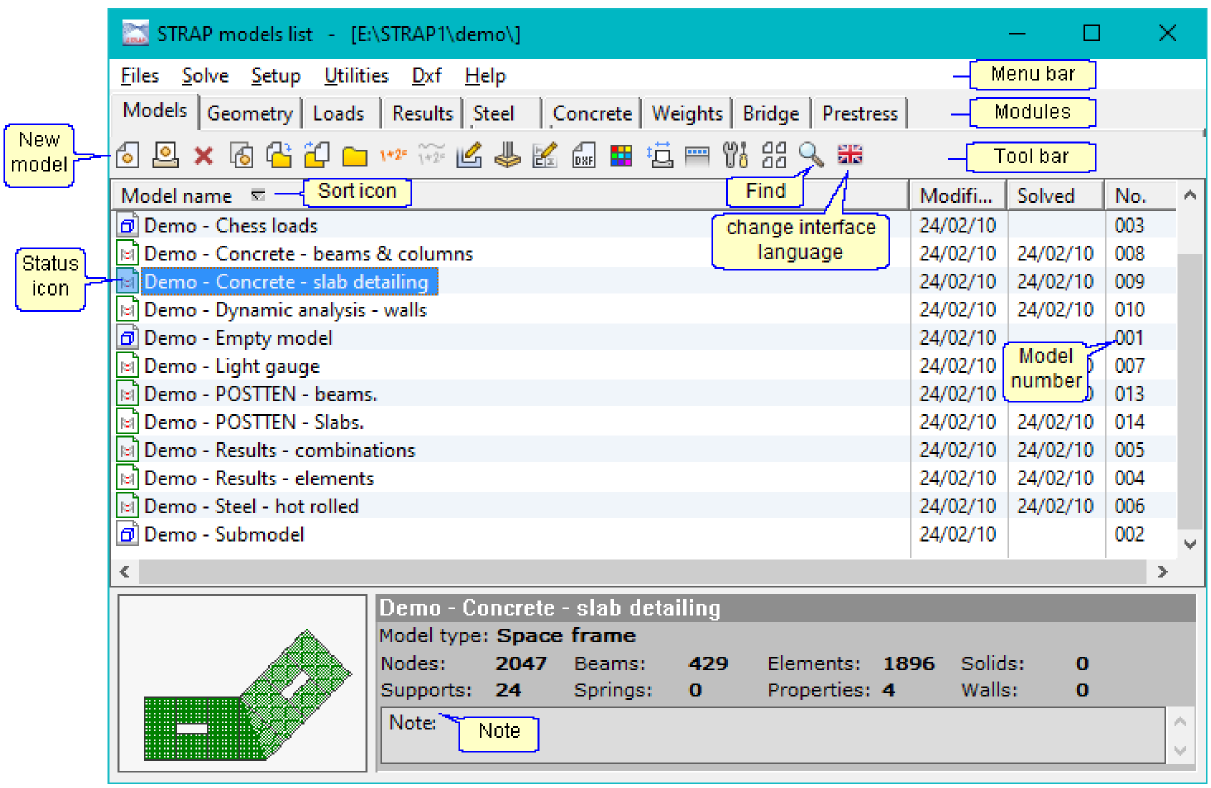
2 STRAP main menu

The program ***Main Menu*** is displayed when the ***STRAP*** icon is selected in the Windows screen or when the Models tab is clicked in any of the program modules:

- the program lists the existing models in the current folder, sorted alphabetically, by date or by model number.
- clicking on any line highlights the model title; the program displays the latest view of the model and its statistics at the bottom of the screen.
- clicking on a tab below the toolbar initiates the relevant option for the highlighted model.
- multiple models may be selected for the delete and copy model options.

Refer to:

Define a new model 101
Amend/rerun an existing model 102
File management 103
Utilities 142
DXF Import/export 149
Setup 121



Status icon

The small icon displays the current status of the model:

only geometry has been defined

geometry and loads have been defined

model has been solved for static loads

model has been solved for dynamic mode shapes (no static loads)

Sort icon

The list of model titles is displayed sorted according to any of the four columns in the table:

- model title
- date revised
- date solved
- model number

To sort the models according to one of the columns, click on the column header; a ▲ is displayed alongside the column header, indicating that the column is sorted in ascending order (e.g. sorted by model title - models starting with 'A' are displayed first; sorted by date - the oldest model is displayed first). Click again to sort in descending order - the symbol in the header is revised to ▼.

Model number

The program stores data for each model in a series of files. All file names for a particular model include a number assigned by the program when the model is created.

For example, if the number displayed is "017", then the model files are:

- **GEOM017.DAT** (geometry)
- **STAT017.DAT** (loads)
- etc.

Note that file management is handled by the program and this number is displayed for information only.

Note

Click on the box and type in any text; this option enables the designer to keep track of revisions, maintain a 'to do' list, etc.

Note:

- the length of text is unlimited but only the first 512 characters may be printed.
- the text is printed beneath the program logo whenever geometry is printed.

Note:

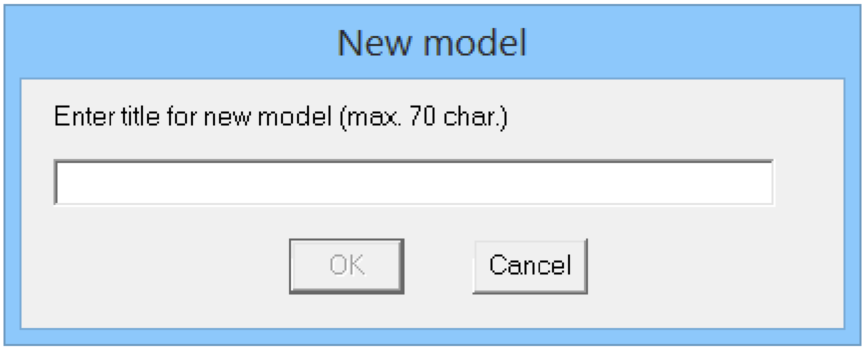
- corrupted model lists may be restored by using the **Utilities** - Recreate a models list option 143.

2.1 Define a new model

- select **Files** in the menu bar
- select **New model** in the menu.

or - click the  icon in the toolbar.

- define the model title:



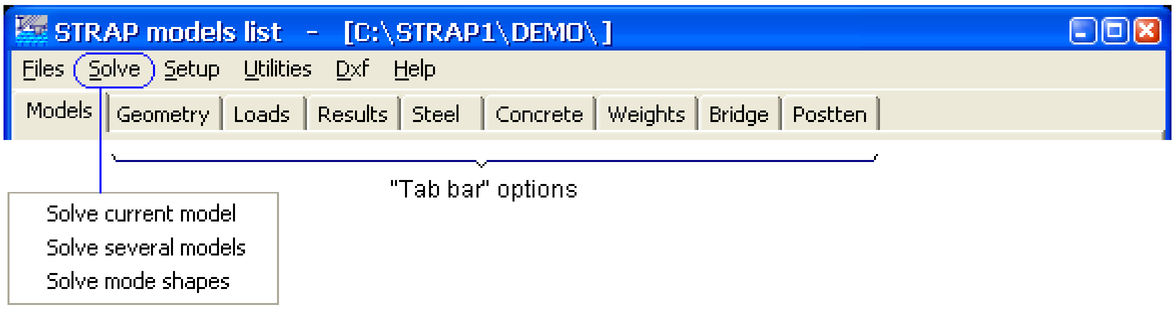
The program then displays the Geometry preliminary menu.

2.2 Revise a model

Revise a model that appears in the list:

To revise the geometry:
- move the cursor to the model name in the list and double-click the mouse

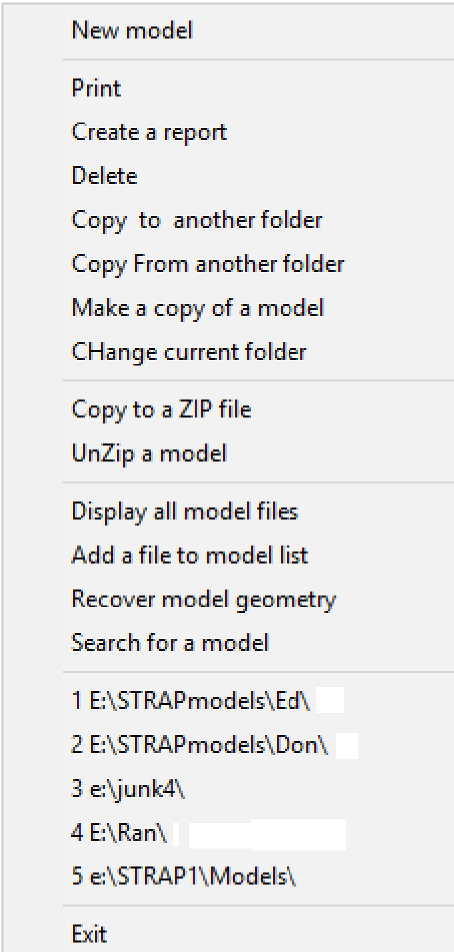
To directly access any other part of the program (loads, results, etc):
- move the cursor to the model name in the list and click the mouse once; the model name is then highlighted.
- click one of the options in the 'tab bar' below the toolbar:



Note:

- click the  icon in the tool bar to display the model list in a different folder
- if the model is in another folder but you don't remember which one, click the  (Search 119) icon to search for the title.

2.3 File options



The "File management" operations should always be carried out by using the options in this menu and *never* by using "Windows Explorer" or any other file management program, the 'Command prompt' DEL, COPY functions, etc.

Previous folders

The last five folders selected in the **Change current folder** option are displayed at the bottom of the File menu. Clicking one one of these lines makes that folder the new 'current folder'.

2.3.1 Print

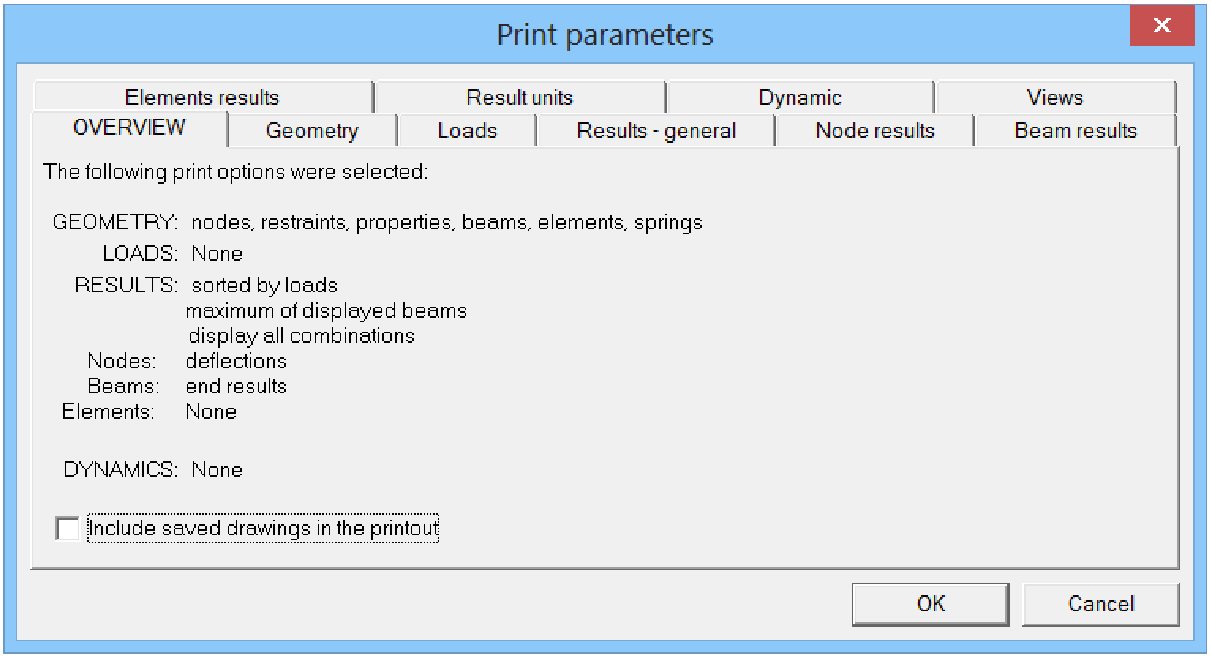
Use this option to print all or selected input, results and saved drawings (in any order) for any model in the list without running the model.

- Specify the tables to be printed by clicking on the **Beam results**, **element results**,...tabs.
- Specify the load cases/combinations to be printed in the **Loads** tab
- Print input/output data for specific elements/nodes only by specifying one or more views in the **Views** tab
- **Include saved drawings** to add drawings created with the **Save for "print/edit drawing"** option
- Dynamic: all defined "**sets**" may be printed at the same time.

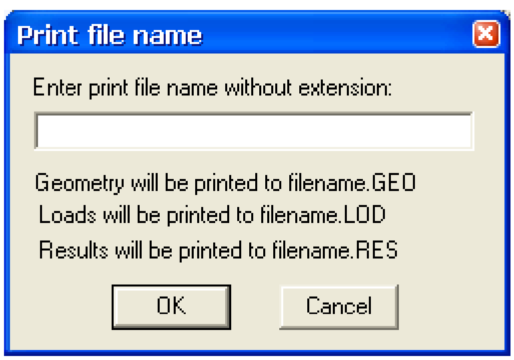
Note:

- ***Create a report*** 105 ***is a more powerful option as it can also include tables and drawings from all STRAP modules (steel, concrete, etc), user-defined text and external objects (graphics, equations, tables, etc)***

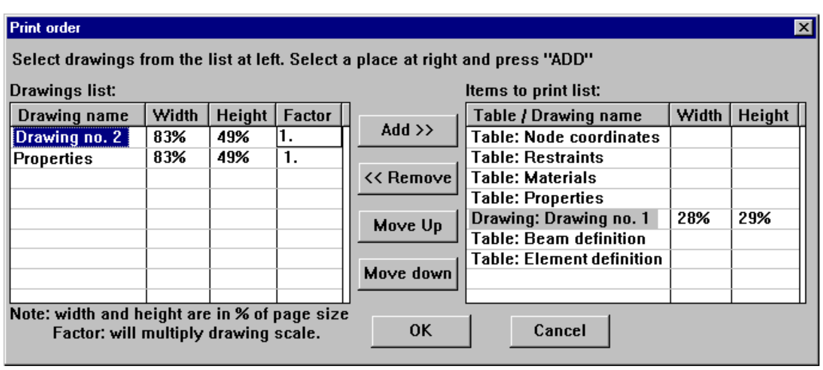
The current selections are displayed in the **Overview** tab. Click OK to start printing.



- Specify the print parameters (printer, page setup, etc)
- If you selected ☑ **Print to file**, the geometry, loads and output are printed in separate files. Enter a file name:



- You may now arrange the order of the tables and instruct the program where to insert the drawings.



Refer to print order for more details

- Click OK to start printing.

2.3.2 Create a report

Create printed "reports" for the highlighted model.

To display a demo video that explains how to create a report:

- *click on to start the video*
- *then click on to enlarge the display.*

A *STRAP* report is an MS Word document (RTF format) that may include:
- tables saved using the **save output for report generation** option (when printing a table)
- drawings saved using the **save for Print/edit drawing** option (when printing a drawing)
- miscellaneous user defined texts.
- graphics, equations, PDF documents and other miscellaneous objects created in other programs.

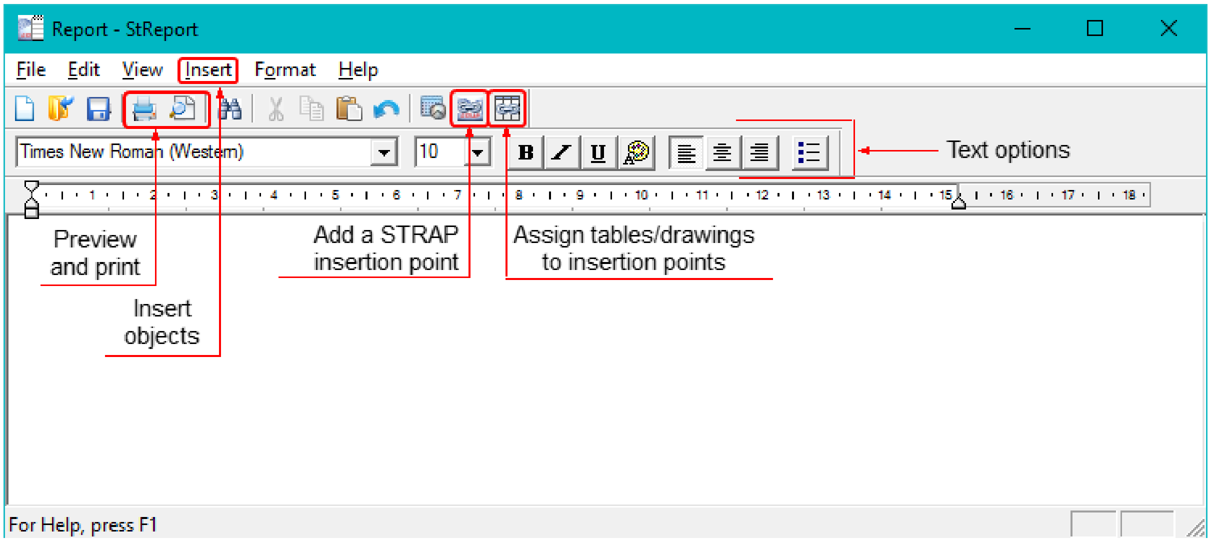
The saved drawings and tables are added to the report at "insertion points":
- saved drawings and tables are assigned by the user to these insertion points.
- the list of drawings & tables assigned to an insertion point is saved with the model and may be updated if the model is revised.

Note:
- A report file created for one model can be used by another model (with different tables & drawings assigned to the insertion points).
- the RTF report file may also be created using MS Word, Wordpad or similar software.

To create the report:
- add STRAP insertion points. Insertion points indicate the locations where the program should insert tables and drawings in the report; tables and drawings must be assigned to these insertion points.
- assign tables/drawings to insertion points
- add miscellaneous texts
- add miscellaneous objects: drawings, equations, PDF documents, etc.
- Preview and print the report

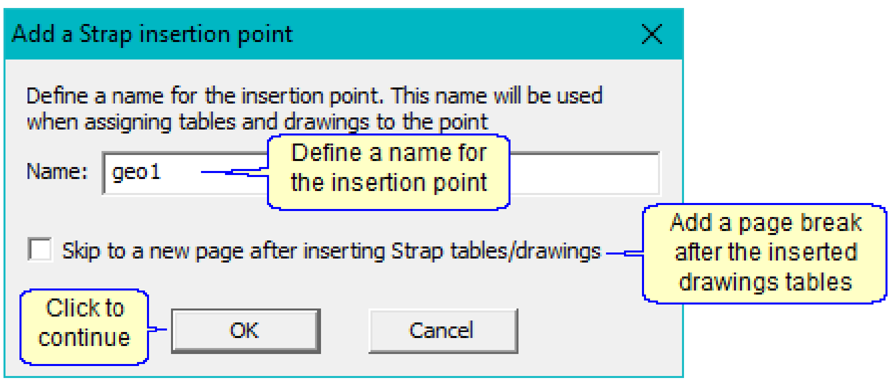
Refer also to Additional notes.

The program initially opens an editing screen with an interface and options similar to "MS Wordpad".

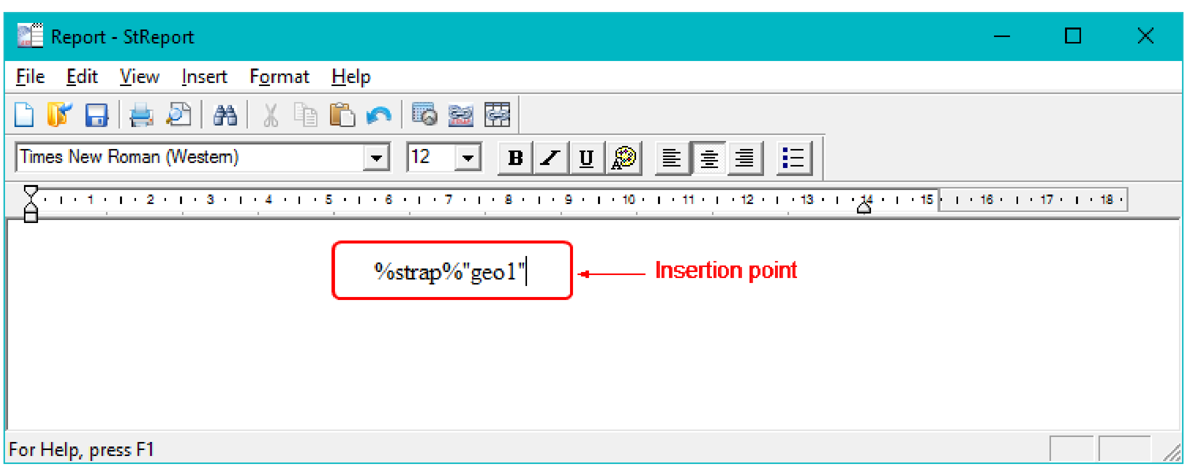


Define an insertion point

- click the icon (or select **Insert - Add Strap insertion point**)



- the insertion point is displayed in the report document:

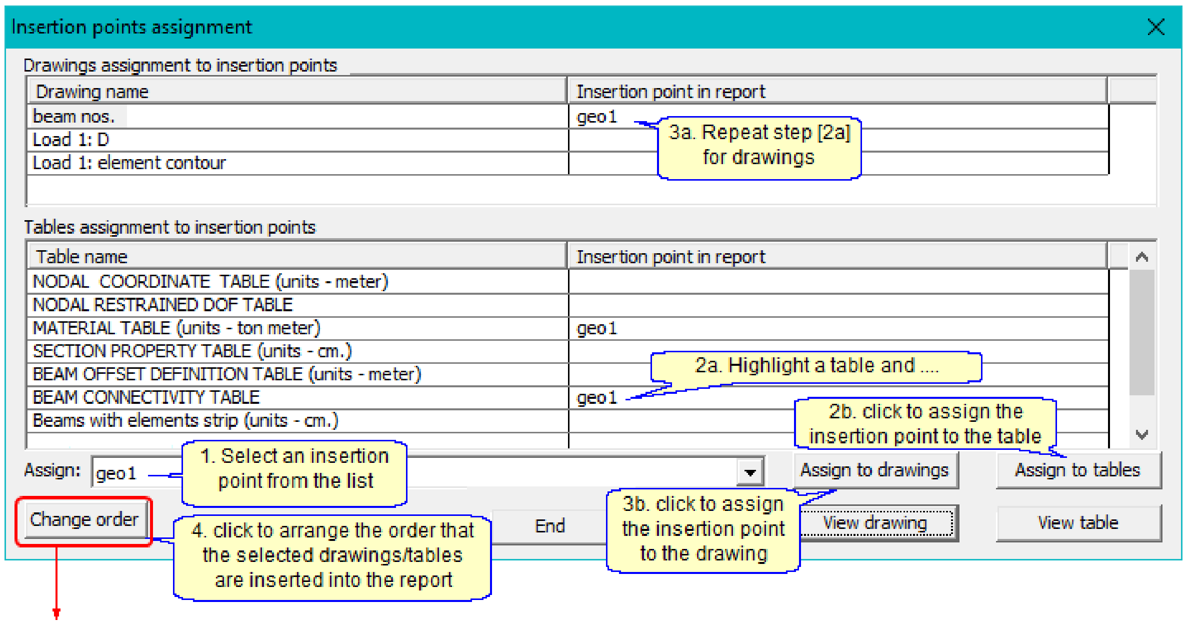


Note:

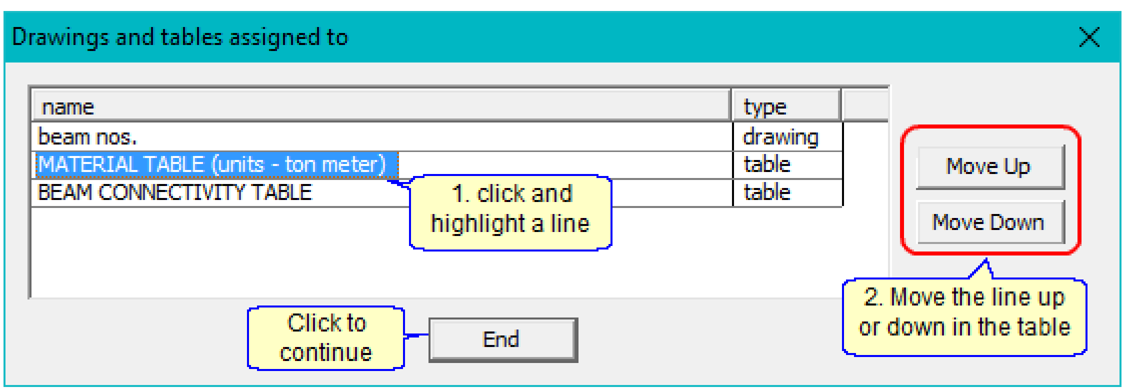
- Insertion points can be typed in manually using the format **%strap%"*name*"**, where ***name*** is the name of the insertion point, e.g. **geo1** in the example above.

Assign tables drawings to an insertion point

- click the icon (or select **Insert - Assign tables/drawings to point**). The program displays a list of of defined insertion points, saved tables and saved drawings:

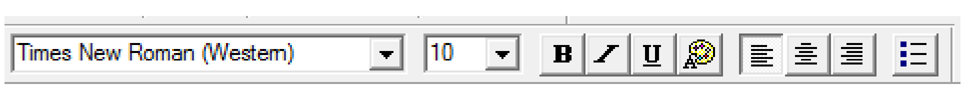


- Arrange the order in which the selected tables/drawings are in inserted into the report (when more than one table/drawing is assigned to an insertion point):

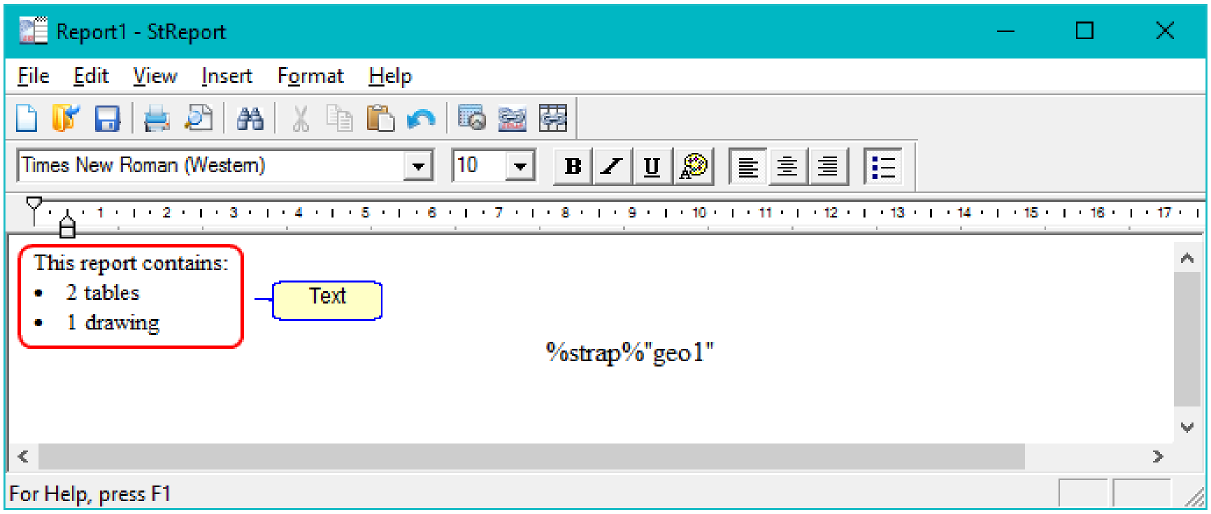


Add miscellaneous texts

Add text anywhere in the report. The standard word processing options are available: fonts, bold, italic, color,bullets, etc:

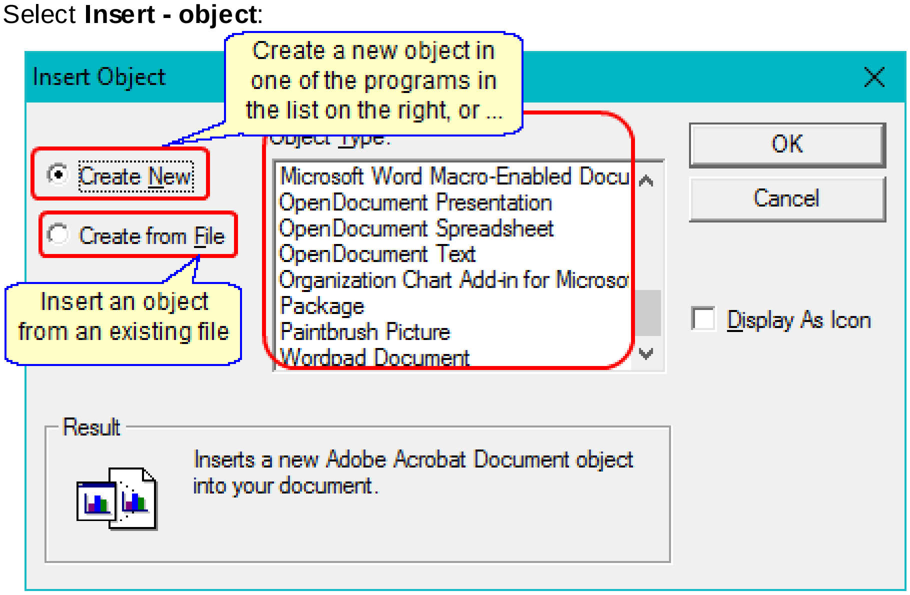


For example:

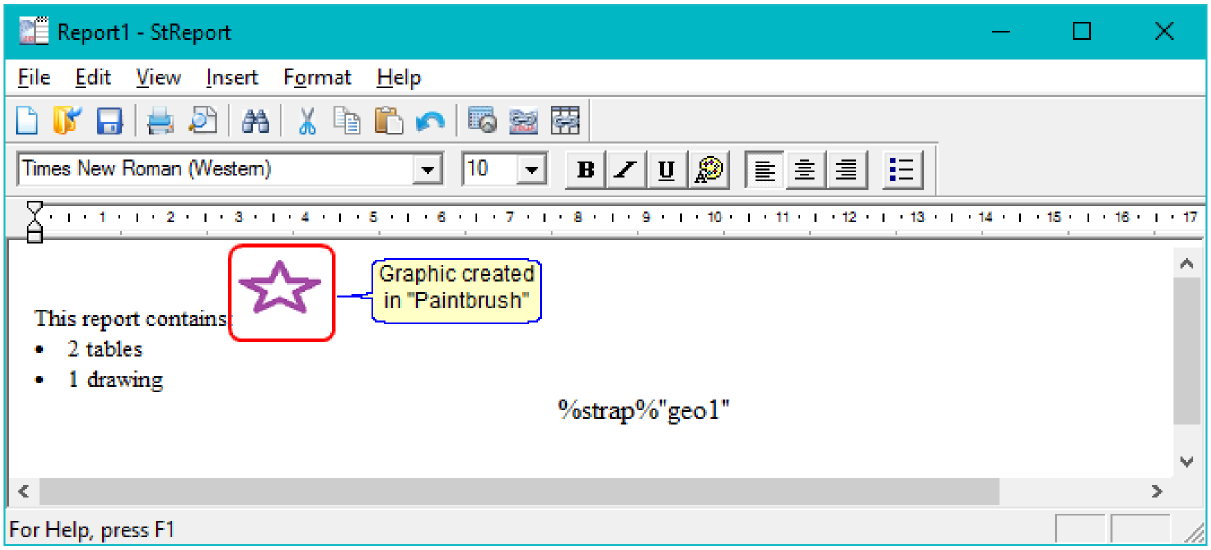


Add miscellaneous objects

Select **Insert - object**:



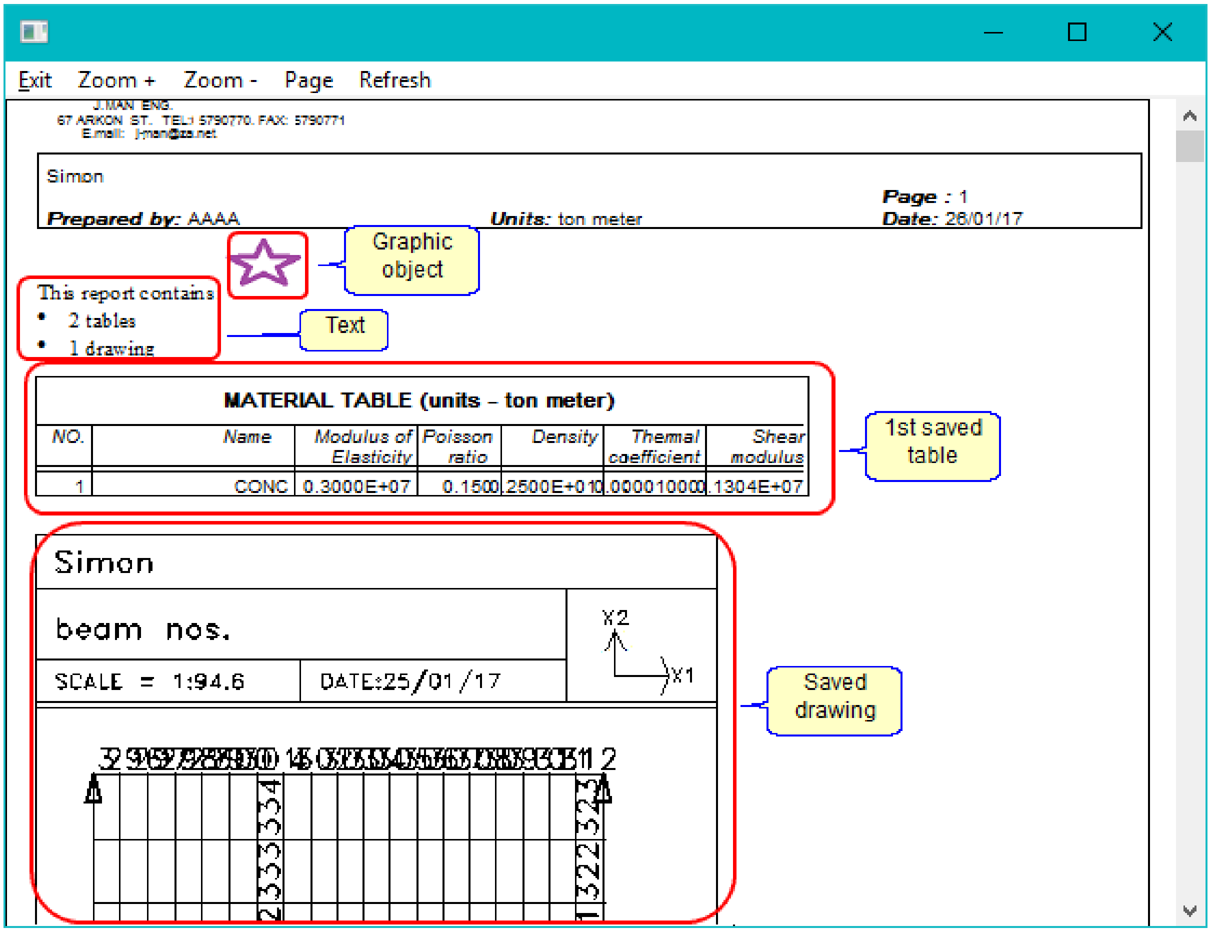
For example:



Preview and print the report

To preview the report:

- click the  icon (or select **File - Print preview**)
- specify the printer (for scaling only).
- the program displays the preview; click **Zoom+** to enlarge. All elements added in the example above are displayed:



Additional notes

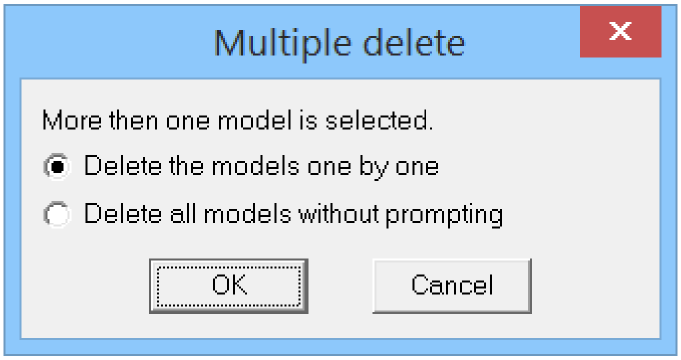
- Report files can be created in any text editor or word processor that can save the file in .RTF format. Insertion points can be typed in manually using the format **%strap%"*name*"**, where ***name*** is the name of the insertion point, e.g. **geo1** in the example above.

2.3.3 Delete a model

This option deletes an entire model from the list or erases selected files only, e.g. loads, results, etc. Note that multiple models may be selected:

To delete models or files:

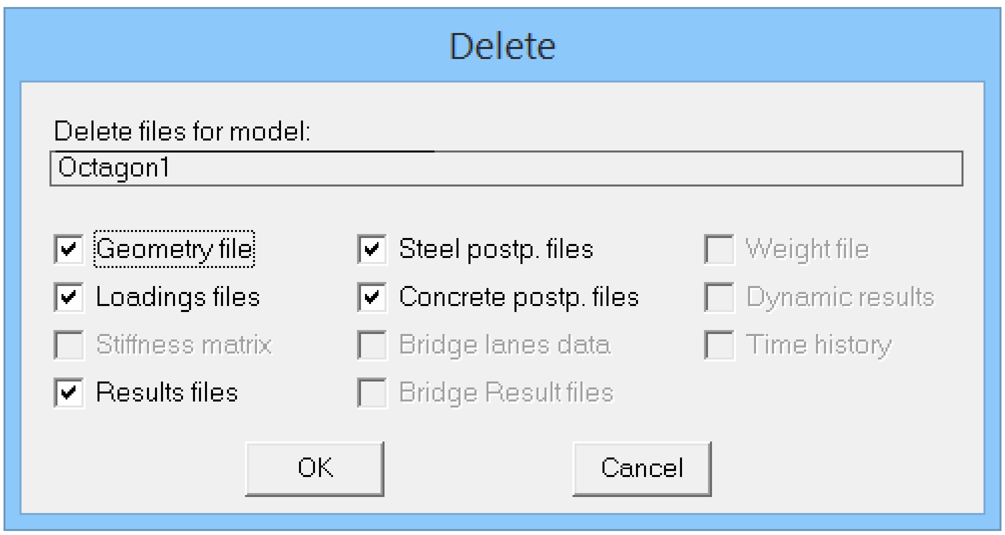
- click and highlight model names in the file list
- if you selected multiple models:



Delete models one-by-one
the following menu is displayed for each model selected

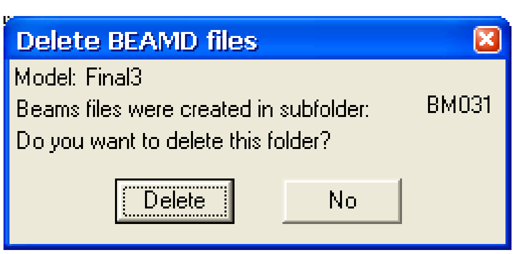
Delete all models without prompting
the following menu is displayed only once and the selection is used for all models

- select the files to be deleted:



- only files that are marked with a ☑ are deleted; click on the file description to toggle the status
- click the OK button.

Note:
- only the relevant files are displayed in the menu
- if **Geometry file** is selected, the program deletes the ***entire model.***
- if **Loading file** is selected, the program also deletes all result files.
- if BEAMD files were created for the model, the files may also be deleted:



To delete a model from another device:

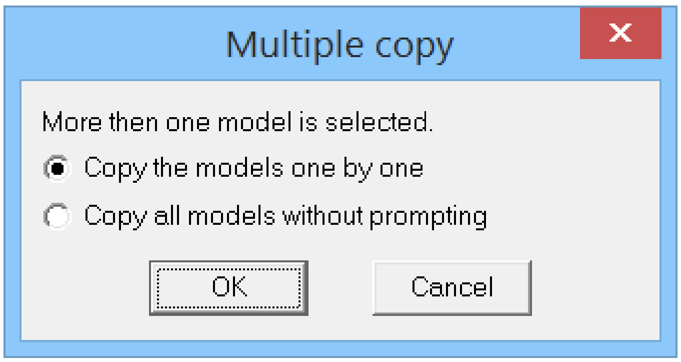
- select **Change current folder** and select the device.
- proceed as explained above.

2.3.4 Copy model to another folder

Use this option to copy a model to the current folder or to backup a model to a different volume. Note that multiple models may be selected.

To copy models or files:

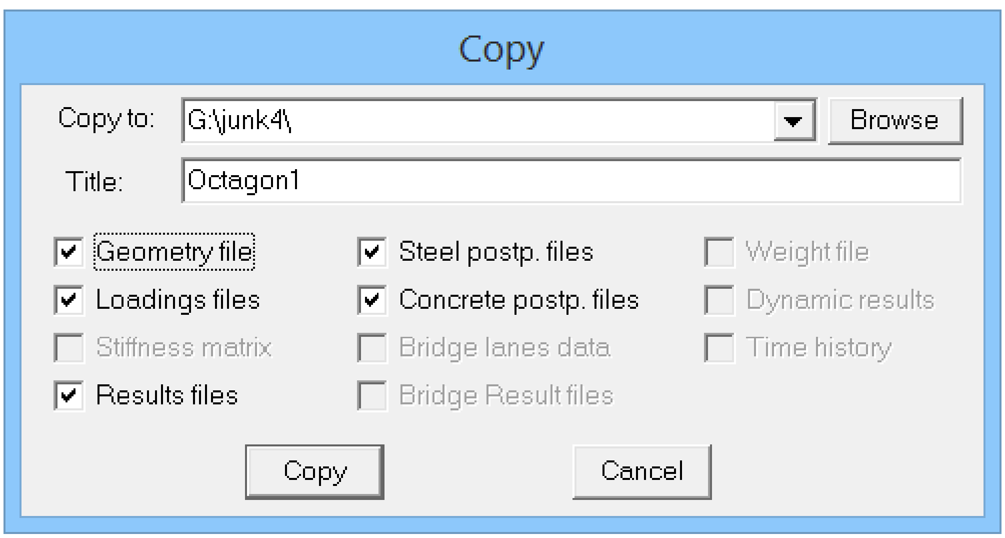
- click and highlight model names in the file list; to select more than one model, press the [Ctrl] key when clicking on the additional models
- if you selected multiple models:



Copy models one-by-one
the following menu is displayed for each model selected

Copy all models without prompting
the following menu is displayed only once and the selection is used for all models

- select the files to be copied:



- type the *drive:\folder* in the **Copy to** box, select a folder using the Browse option, or click the button to choose a recently selected folder.
- Revise the **Title** (optional).
- only files that are marked with a are copied; click on the file description to toggle the status
- click Copy to begin copying.

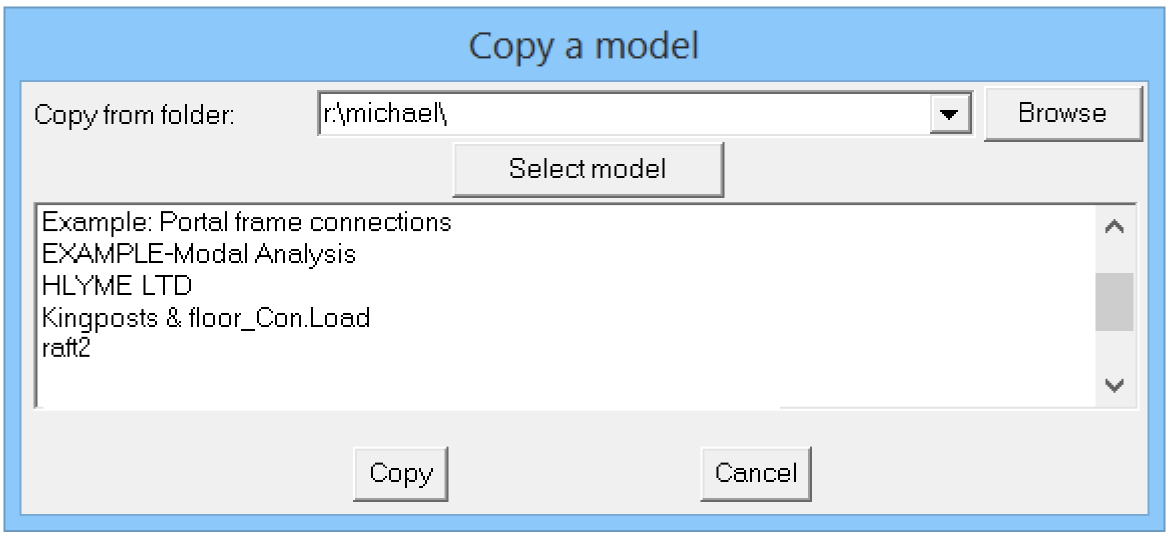
Note:
- the program does not replace an existing model in the destination folder that has the same name; it adds a numerical suffix, e,g. "(1)" to the title of the copied model.
- To create a copy of a model in the current folder, refer to Make a copy of a model 114.

2.3.5 Copy model from another folder

Use this option to copy models from another folder to the current folder or to restore a model from another device. Note that multiple models may be selected:

To copy models or files:

- type the *drive:\folder* in the **Copy from folder** box, select a folder using the Browse option, or click the button to choose a recently selected folder ; press [Enter] or click Select model
- select the model or models to be copied:

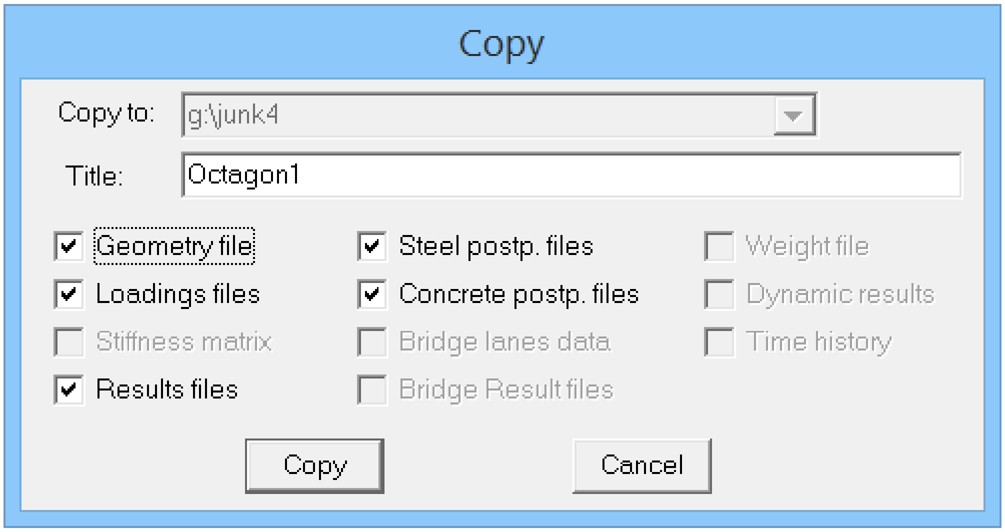


  to select more than one model, press the [Ctrl] key when clicking on the additional models.
- for each model selected, specify the files to be copied from the list displayed; only files that are marked with a ☑ are copied. Click on the file description to toggle the status.
- click the Copy button to begin copying.

Note:
- the program does not replace an existing model in the destination folder that has the same name; it adds a numerical suffix, e,g. "(1)" to the title of the copied model.
- To create a copy of a model in the current folder, refer to Make a copy of a model 114.

2.3.6 Make a copy of a model

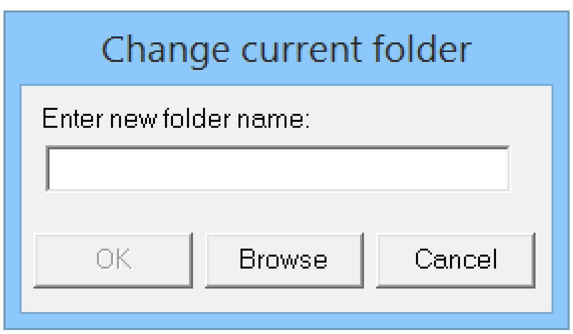
Use this option to create a copy of the highlighted model in the current folder:



- revise the **Title** (optional).
- only files that are marked with a ☑ are copied; click on the file description to change the status.
- click Copy to begin copying.

2.3.7 Change current folder

The current folder is displayed at the top of the screen.



- to select an existing disk folder on any drive type in the name of the path or click the Browse button and select the path in the standard Windows "**Select a folder**" dialog box.
- To create a new folder, click the Browse button and define a new path by clicking the icon ("Create New Folder") in the standard Windows "**Select a folder**" dialog box.

Note

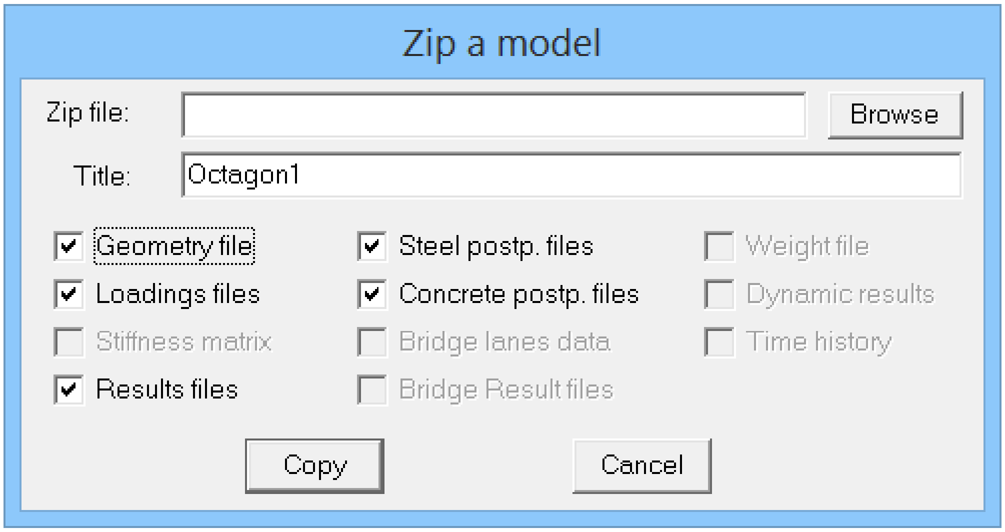
- the last five directories selected in this option are displayed at the bottom of the File menu 103 and may be selected by clicking on them.
- do not select a "Read-only" folder

2.3.8 ZIP

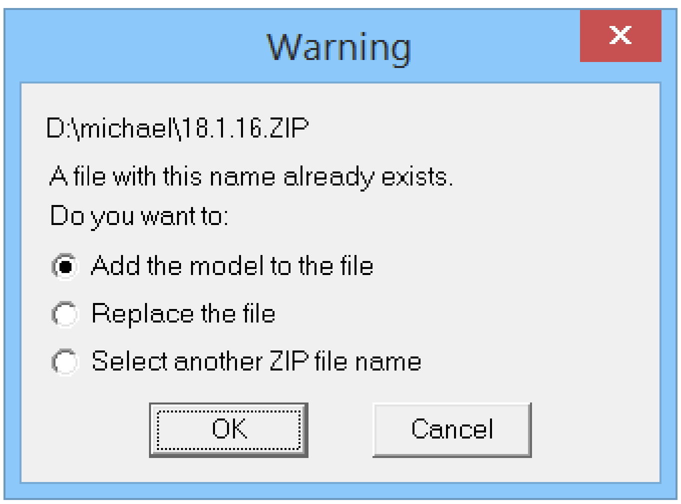
Model files may be condensed to a single ZIP format file and may be imported back into the model list from a ZIP file.

To copy to a ZIP file:

- enter the name of a new ZIP file or select an existing one; specify which of the data files to condense:



- for an existing ZIP file, select one of the following options:



where:

Add the model
the selected models are added to the file

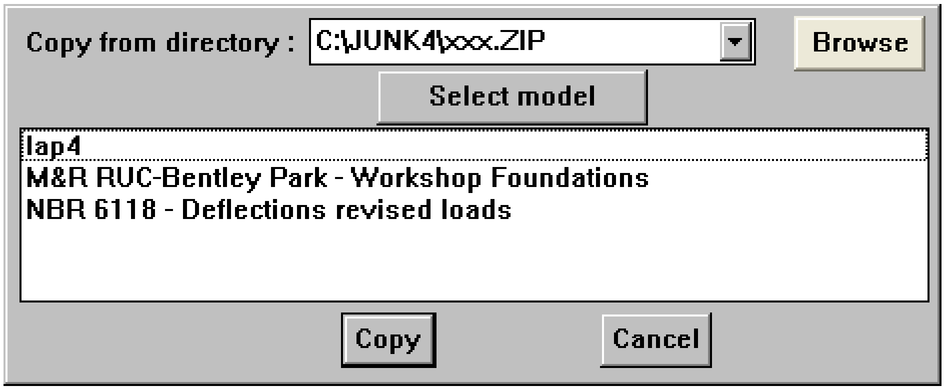
Replace the file
the existing file is erased and a new one is created with the selected models

Select another ZIP file name
Cancel the option and enter a different file name before continuing

2.3.9 unZIP

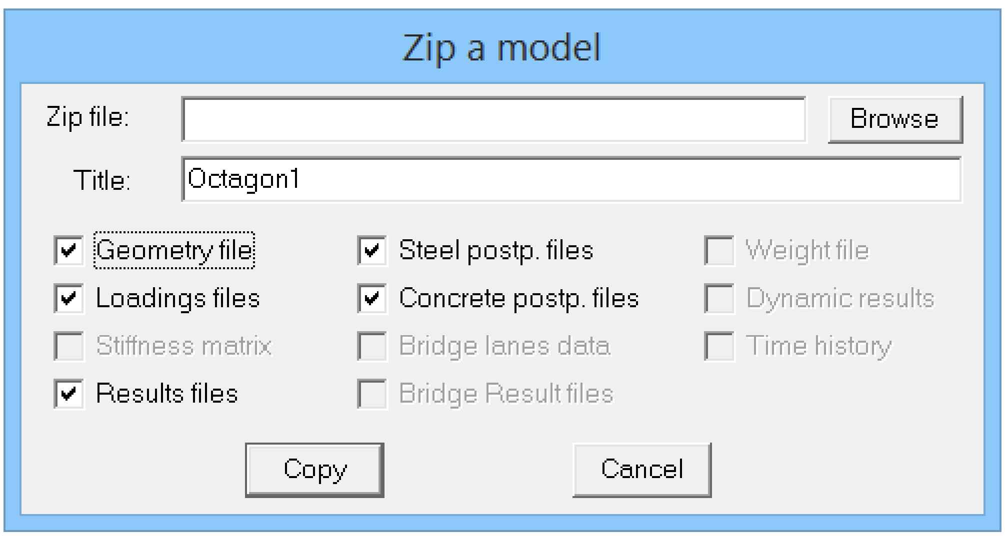
Models may be retrieved from ZIP files; note that the ZIP files do not have to be created by *STRAP* but they must contain the folder file **ZZMODEL.DIR**.

To retrieve a model from a ZIP file:
- select the ZIP file
- click and highlight one or more of the models in the list:



- click 

- select the files to extract:

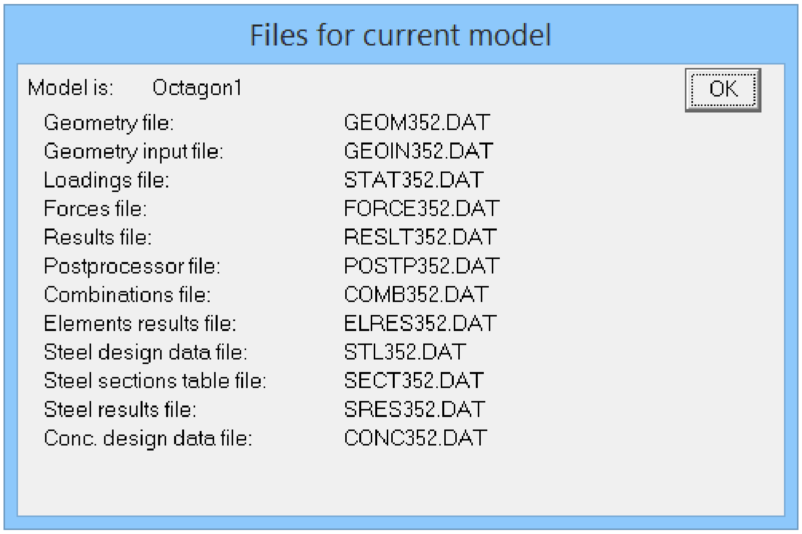


- click 

Note:
- the program does not replace an existing model in the destination folder that has the same name; it adds a numerical suffix, e,g. "(1)" to the title of the unzipped model.

2.3.10 Display all model files

The program automatically assigns file names to all files created for a model. Use this option to display the file names for the highlighted model. For example:



2.3.11 Add a file to list

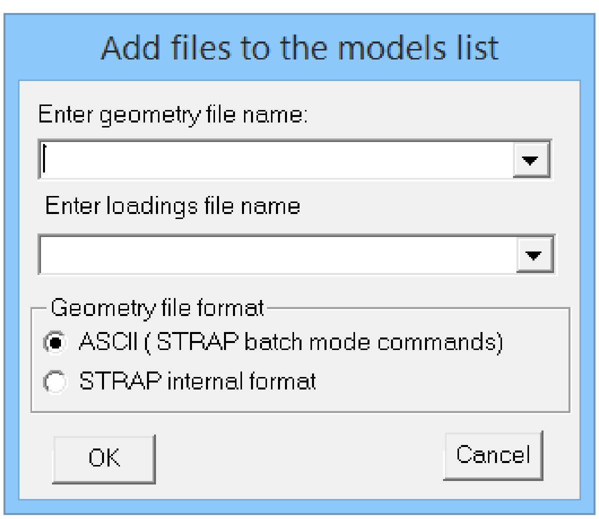
The model list is maintained by the program and displays all models in the current folder, both those created by the user by running *STRAP* in the folder or those copied to the folder using the program "Copy to" and "Copy from" options. Note that the model list is stored in the file ZZMODEL.DIR; this file is found in every working folder and and ZIP file created by the program.

Models whose files are manually copied (e.g. using 'Command Prompt "Copy" command) to the current folder will not be displayed in the model list because the ZZMODEL.DIR file cannot be edited by the user.

This situation may occur in several ways. For example:

- the user defines the model by writing a geometry ASCII file and a loading ASCII file
- the GEOMnnn.DAT file is corrupted and the user wants to recreate the model from the ASCII GEOINnnn.DAT file (a list of geometry commands maintained by the program when the user defines the model)
- the model files are available (e.g. on a backup device) but the ZZMODEL.DIR file is not present or corrupted; the program "Copy to" and "Copy from" options cannot locate the models.

Use this option to add such models to the list.



File names

Type in the name of the geometry and and loading file (optional).

Note:

- a drive name (e.g. G:) or a folder path may be entered.
- click the button to display a list of files in the current folder.

Geometry file format

The geometry file may be in one of two formats:

ASCII (STRAP batch mode commands)
the user created a geometry ASCII file (refer to Batch mode - geometry for more details on file format).

STRAP internal format
a geometry file that was created by *STRAP*, i.e. binary format. This type of file cannot be edited by the user.

Note:

- the files may have any name
- the geometry ASCII file must start with the command REPLACE or ADD (refer to Batch mode - geometry).
- the loading file must start with the command ASCII (refer to Batch mode - loading)

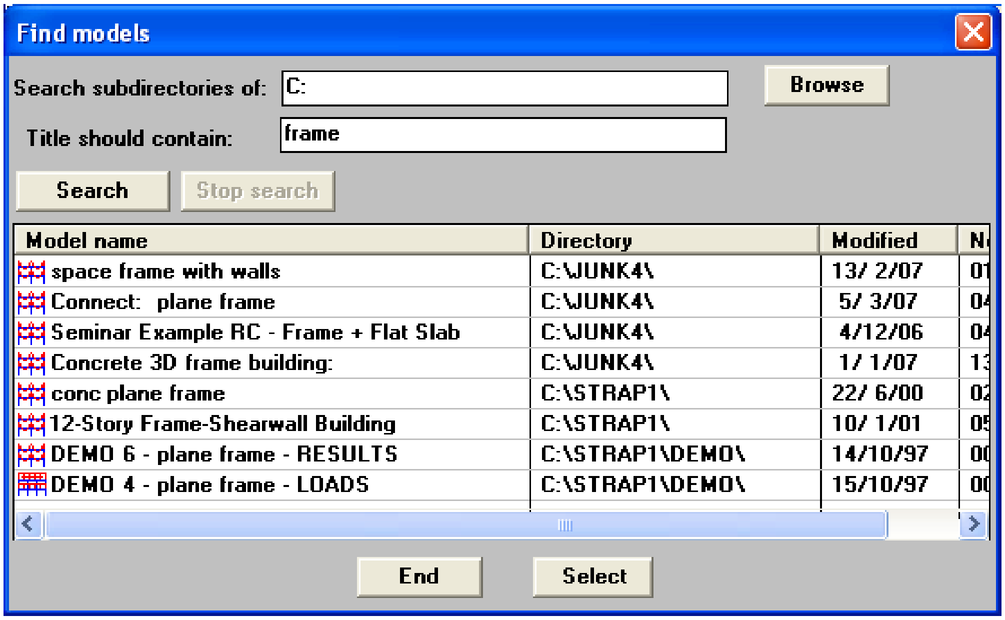
2.3.12 Recover model geometry

The current geometry for each model is stored in a binary file named **GEOMnnn.DAT**, (where "nnn" indicates the model number). However, ***STRAP*** simultaneously creates an ASCII data file for each model named **GEOINnnn.DAT** that contains the geometry data in the form of commands. When the *STRAP* geometry file is corrupted the program may be able to recreate all or part of the file by reading the commands in the GEOINnnn.DAT file.

Always backup the model before recovering the geometry. Please contact your *STRAP* dealer if the recover is not successful.

2.3.13 Search for a model

Find a model anywhere on the disk, etc, by searching for a string in the model titles:



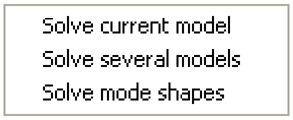
- type in the disk volume to search in the '**Search subdirectories of**' box. For example:
  - if you type **C:**, the program searches the entire disk C:
  - if you type **C:\STRAP1**, the program searches folder **STRAP1** and all its sub-folders.
- type in the text to search for in the **Title should contain** box. The example above searches for frame.
  - the search is not case sensitive
  - if you type in more than one word (separated by spaces), the program displays the titles containing ***all*** of the words (even if they are not consecutive in the title)
  - the program displays all of the models in the search path if this box is left blank
- click Search to start the search
- to revise and edit a model in the list, click and highlight the appropriate line and click Select; the model geometry is then displayed (and its location becomes the new current folder).

Note:

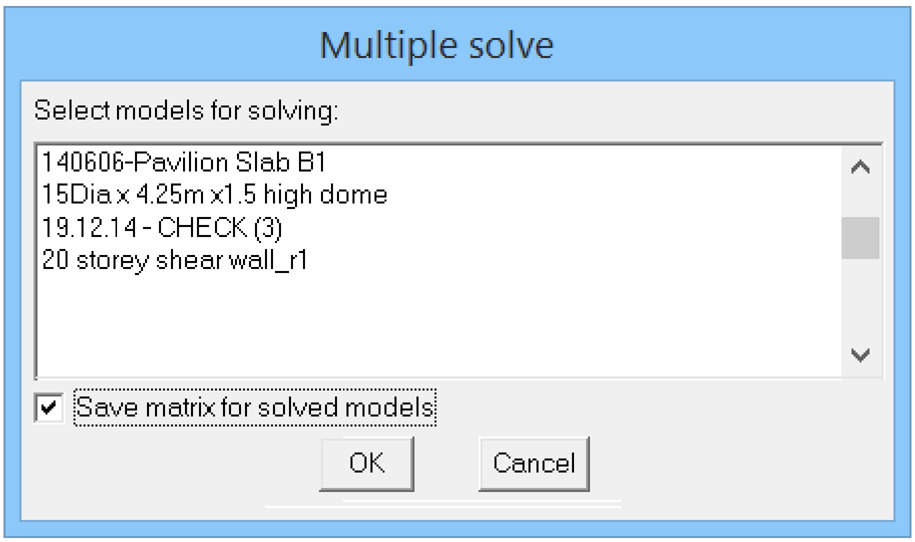
- when you select  the results of the previous search are displayed in the list; if you selected the wrong model previously there is no need to search again.

2.4 Solve

There are two options available:



- **Current model**:
  Start the solution for the highlighted model.
- **Several models**:
  Solve several models in series:
  - click and highlight the models in the current folder to be solved (only models with geometry and loads are displayed):



  - set the ☑ if you want to save the stiffness matrix for ***all*** the selected models.
  - click the OK button to start the solution.
- **Mode shapes**
  Calculate the dynamic mode shapes for the highlighted model (this option is displayed only if dynamic weights have been defined).

Note:
- error messages for all models are written in one file.
- Refer to Solution for a detailed explanation on the solution method.

2.5 Setup

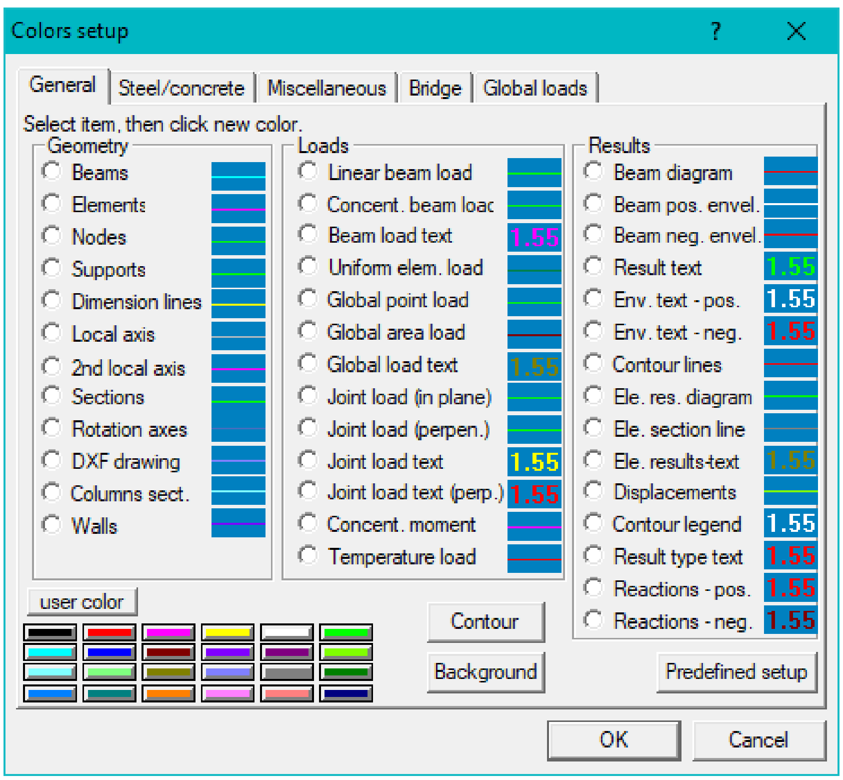
Use this option to specify default values for screen display colours, units, standard material properties, output format, etc.

Refer also to STRAP.INI / Registry.

2.5.1 Colours

Specify the permanent screen colours for:

- graphic lines 121, e.g. beams, elements, various loads, etc.
- text 121 associated with each graphic line
- background 122
- contour map 122 fill colors
- steel postprocessor 122 capacity colors

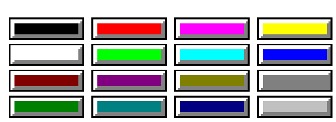


General

Line /text colors - general:

To revise the colour of a geometry/load/result line or text:

- click the ◉ of the relevant item
- select the new colour from the palette.

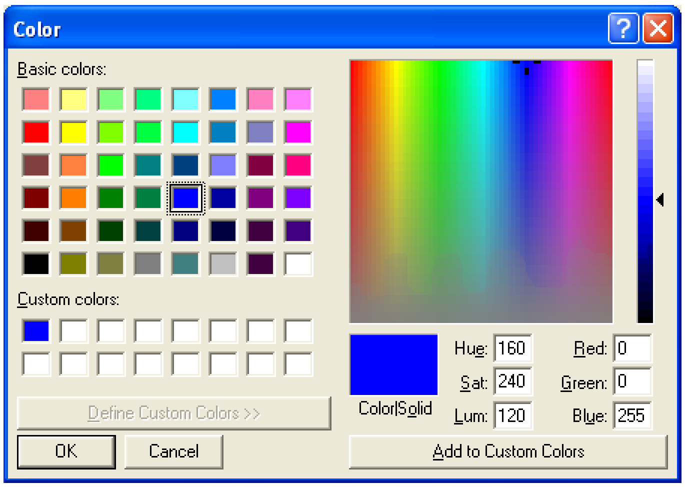


or define a custom user colour

Background:

Revise the background colour:

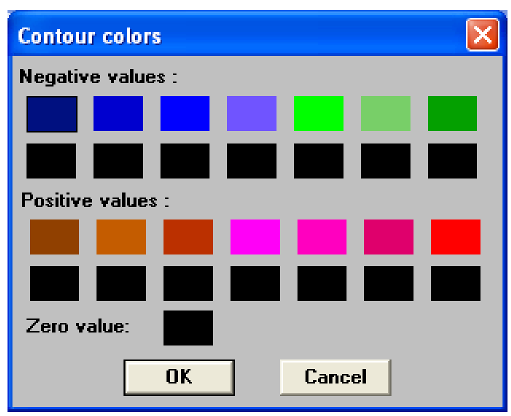
- click the Background button
- select one of the basic colors from the following screen or define a custom color and then click the OK button.



Contour lines:

Revise the contour fill colours:

- click the Contour fill colors button
- the program displays a list of the positive and negative values fill colours (note the colour of the zero value):



- click the colour square to be revised and select a new basic colour as explained in Background 122.

User color:

Any custom colour can be defined (instead of the standard 24 colours). Refer to custom colours in the Background colour option.

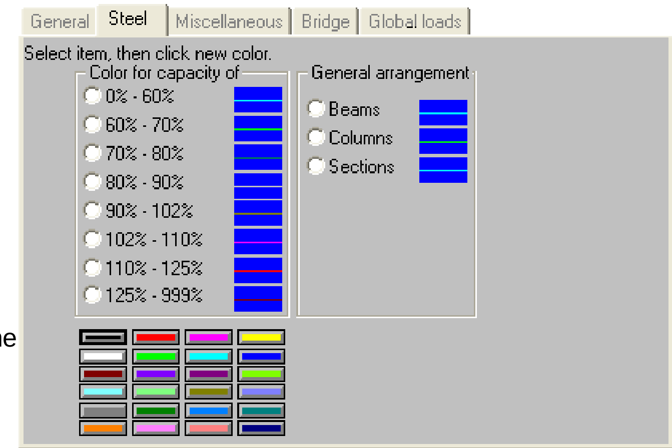
Predefined setup:

Several different colour sets may be defined and saved, e.g.
- a dark background and light coloured lines and text
- a light background with dark coloured lines and text.

Select a predefined set.

Steel/Concrete design

This option allows you to colour code the display of "% of capacity results" in the Steel and concrete design modules; specify a display colour for a range of result percentages:

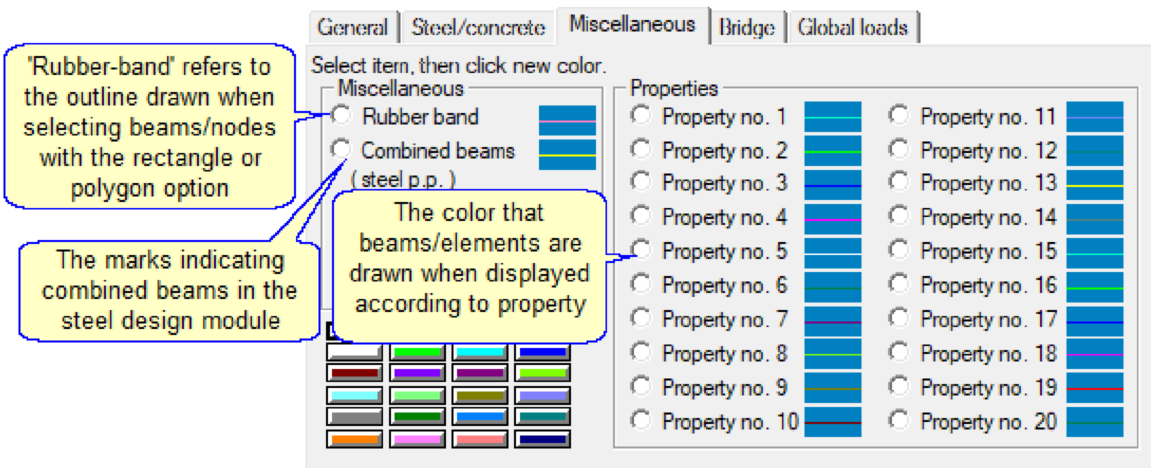


Select a range and then select a colour from the palette below the table.

Note:

- the limits of the ranges may be revised in the design modules.

Miscellaneous



Bridge

Refer to General 121.

Global loads

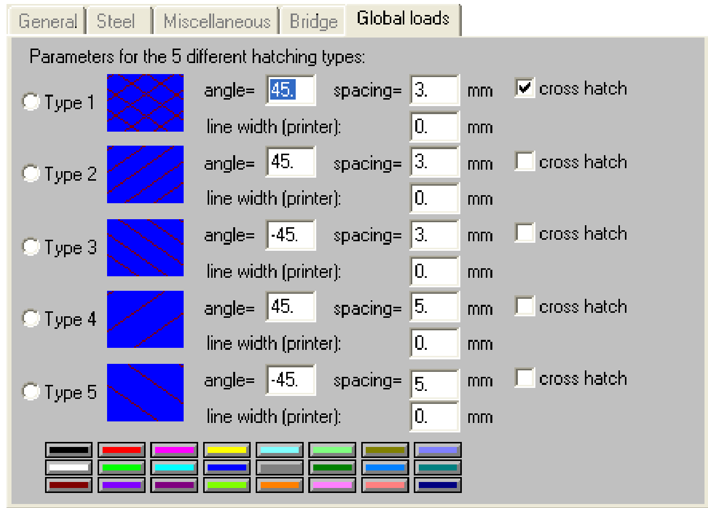
Specify the hatching pattern for global area loads.

The program divides the loads into groups according to magnitude:

- if there are five or less global loads, **Type 1** is used to plot the largest load, etc.
- if there are more than five load,each group is used to plot a range of loads; **Type 1** is used to plot the largest load and **Type 5** is used to plot the smallest.

Define the pattern by specifying

- the angle and the spacing of the lines.
- parallel lines or hatched pattern



2.5.2 Print parameters

The format for ***STRAP*** tabular and graphic output may be specified by the user.

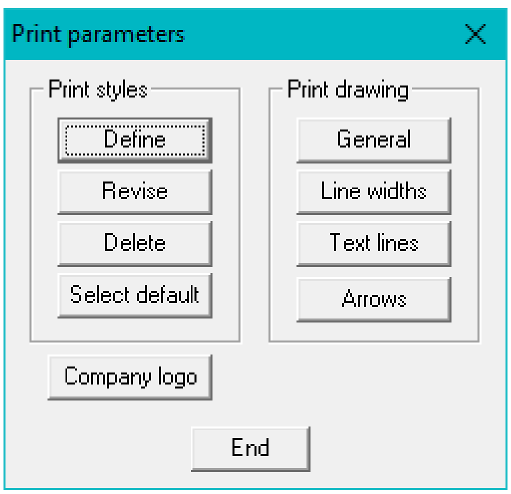
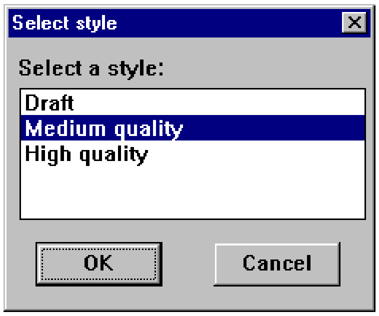


Table print styles

A series of "styles" may be defined using this option; each style contains information on the fonts, margins, lines and spacing to be used when printing the table. When printing tables when in any geometry, loading, result or postprocessor modules, the user may select one of the predefined styles and the tables will be printed in the specified format. Refer to Setup - additional 133 to set similar options for displaying the same tables on the screen.

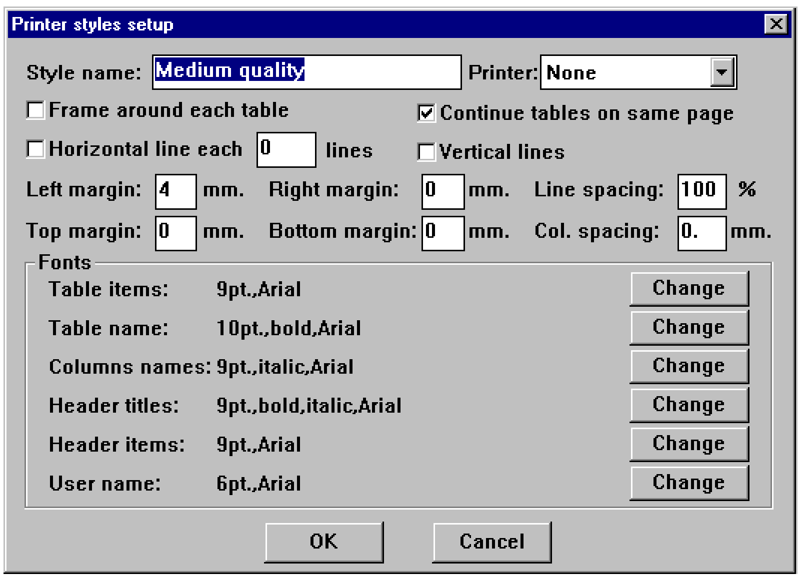
Select one of the defined styles from the displayed list:



Note:

- "Draft" style prints all data in Courier 10 cpi font without any vertical or horizontal lines. This style cannot be edited.

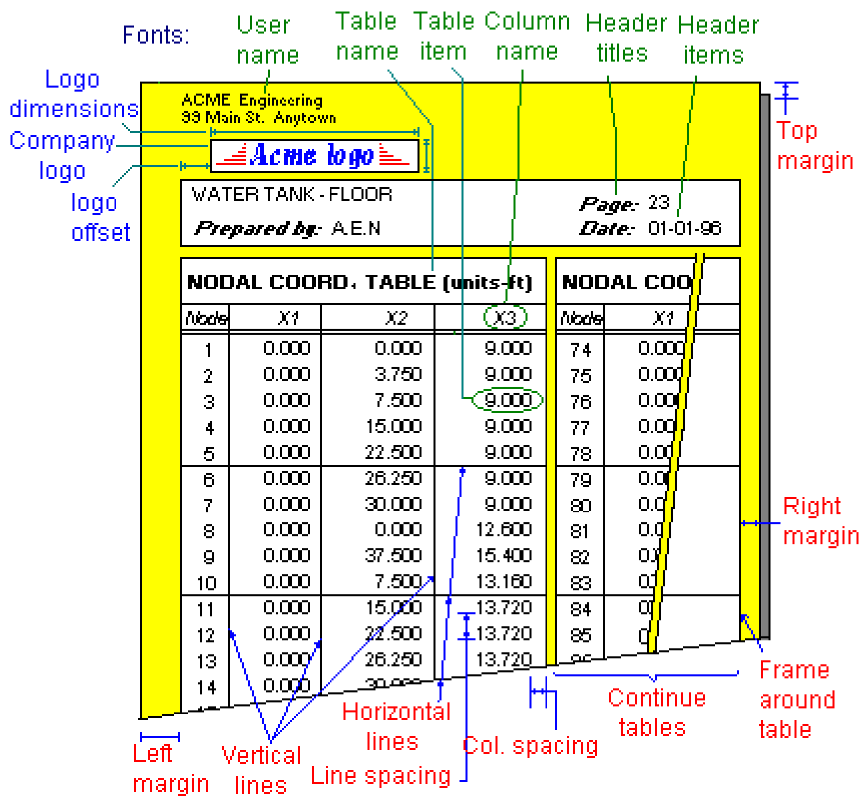
The style items are:



- ***Printer:***

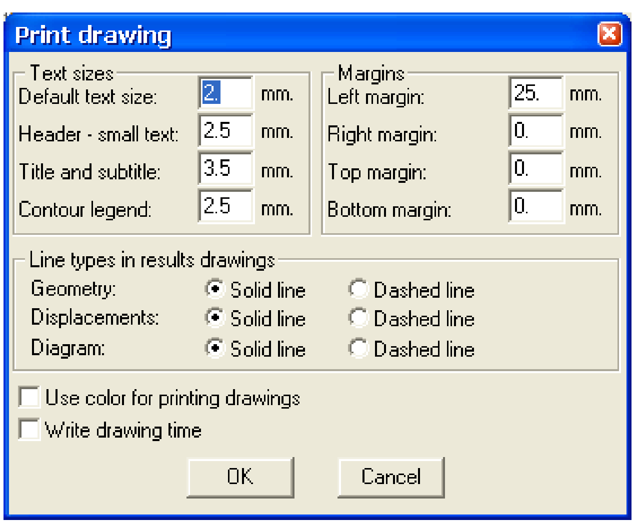
  Select a default printer for the style. The printer may be changed when actually printing the tables.

- ***Style items:***

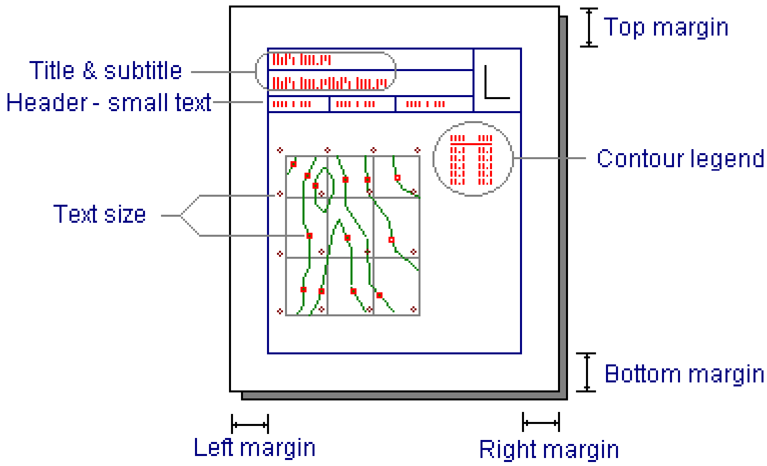


Print drawing - general

Specify default values for "Print drawing" options.



- ***Drawing text/margins***



Note:

- the margins are relative to the maximum print area available for your printer (i.e. margins may be present when zero values are entered in the above menu). Refer to your printer manual.

- ***Drawing - line types***

  Specify the following options for the results drawings:
  - lines types as ◉ **Solid** or ◉ **Dashed**
  - set **Use color** to ☐ to suppress color on color printers (print all graphics in black)
  - set **Write drawing time** to ☑ to write the current time in the header.

Line / text width

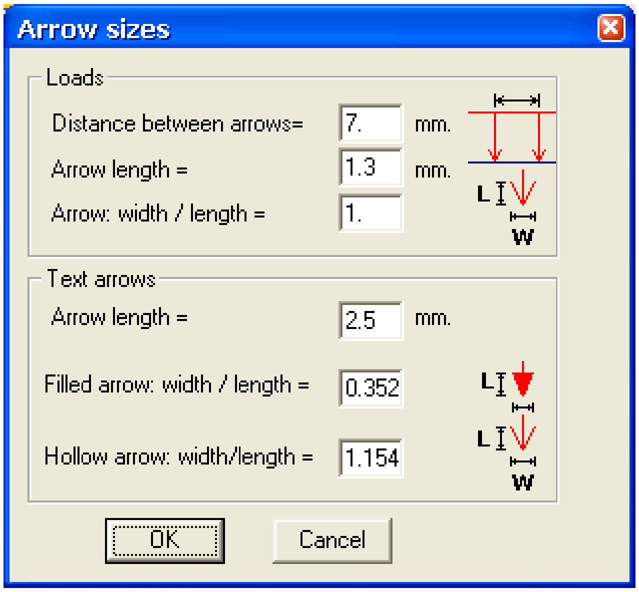
For graphic printing (print drawing and print/edit a saved drawing), define the width of lines associated with each drawing element and text.

Note:

- the line/text widths are defined in millimeters.
- lines defined with zero width are drawn with a width of one dot, e.g. for a printer resolution of 300 dpi, the line width is 1/300 inch.
- geometry line/text widths also apply to the geometry in loads and results.

Arrows

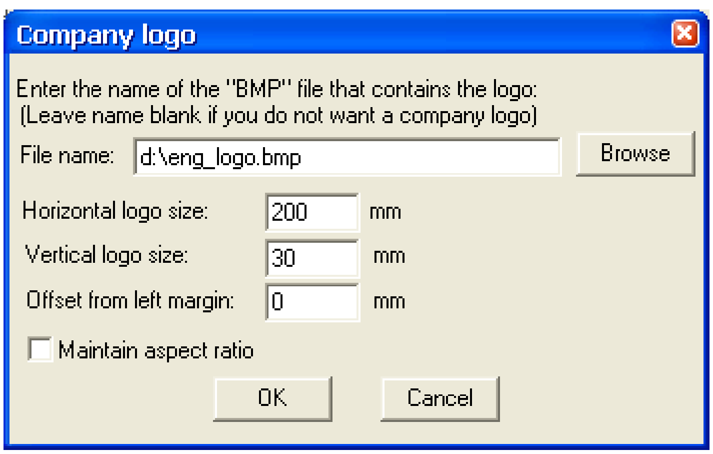
Specify arrow options for loads and text:



The "Text arrow" parameters are used in the **Print/edit a saved drawing** option.

Company logo

Print the company logo at the top of every page of tables (not Print drawing or Print/edit saved drawing).



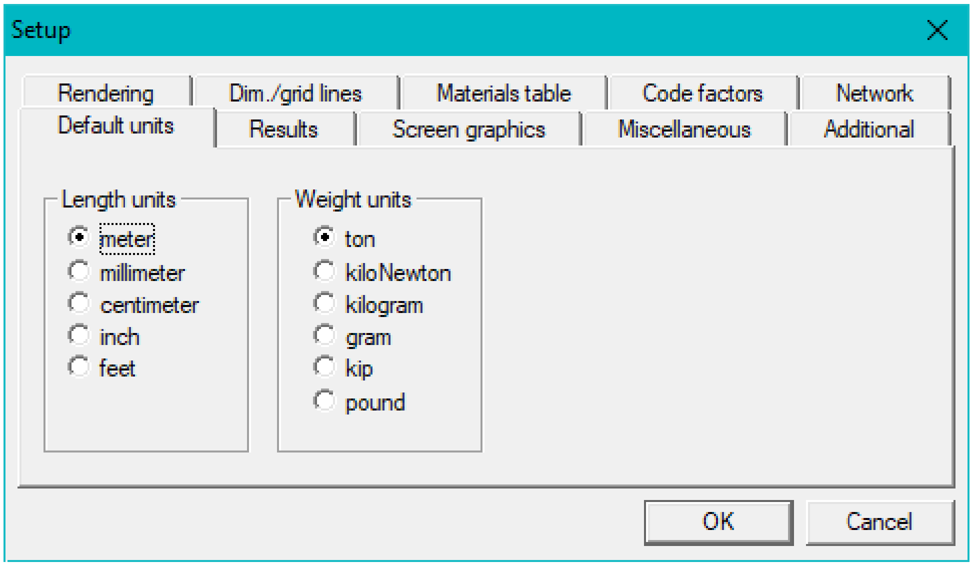
- the logo must be in the standard Windows Bitmap (.BMP) format and must not be removed from the specified path at a later date.
- enter the horizontal and vertical dimensions of the logo:

☐ **Maintain aspect ratio**:	the program distorts the image if the dimensions entered here are not proportional to the bitmap dimensions.
☑ **Maintain aspect ratio**:	the program maintains the drawing proportions by using only one of the dimensions entered here.

- enter the offset from the left margin; the value is measured from the left margin value specified in the Style.

Refer to 'Print styles 125' for an example.

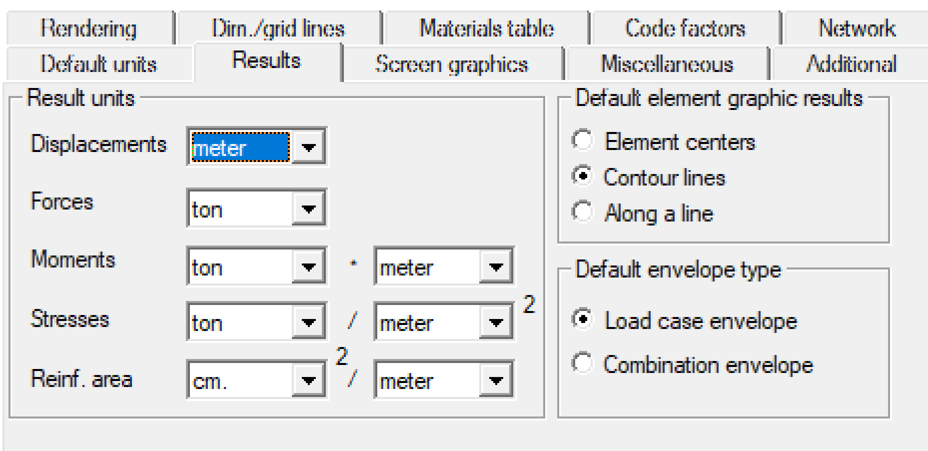
2.5.3 Miscellaneous



Setup - units

Set the default input units for ***all*** new models. The units may revised in the geometry and loads definition module, but the selected units are used only in the current model.

2.5.3.1 Results



Result units

Set the default result units for ***all*** new models. The units may revised in the results module, but the selected units are used only in the current model.

Default element graphic results

Specify the default graphic element result type (highlighted in the list of result types)

Default envelope type

Specify the default envelope type (case or combination) for the result display options.

2.5.3.2 Materials

The properties of 10 materials are permanently stored in the program. Four of these materials are user-defined materials. These 10 properties are displayed when the **Materials** option is selected in beam or element property definition.

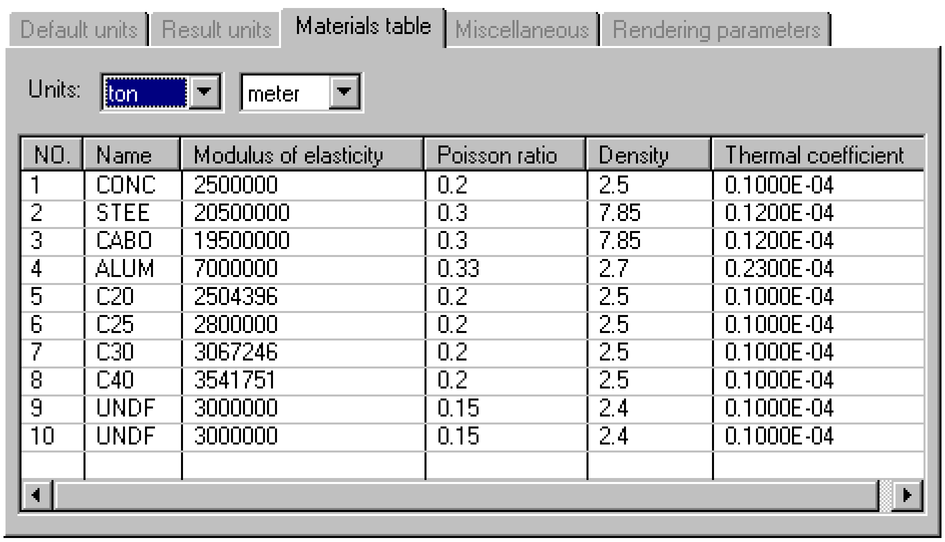
The properties of all 10 materials may be edited. The properties are:

- Modulus of elasticity
- Poisson's ratio
- Density
- Thermal expansion coefficient (1/°Celsius or 1/°Fahrenheit)

Units:

- Specify the units for modulus-of-elasticity and density in the listboxes at the top of the screen.
- The thermal coefficient may be defined according to either unit. However the temperature difference value entered when a temperature load is applied must be according to the same units.

The program displays the material table; select a material and revise/define its properties.

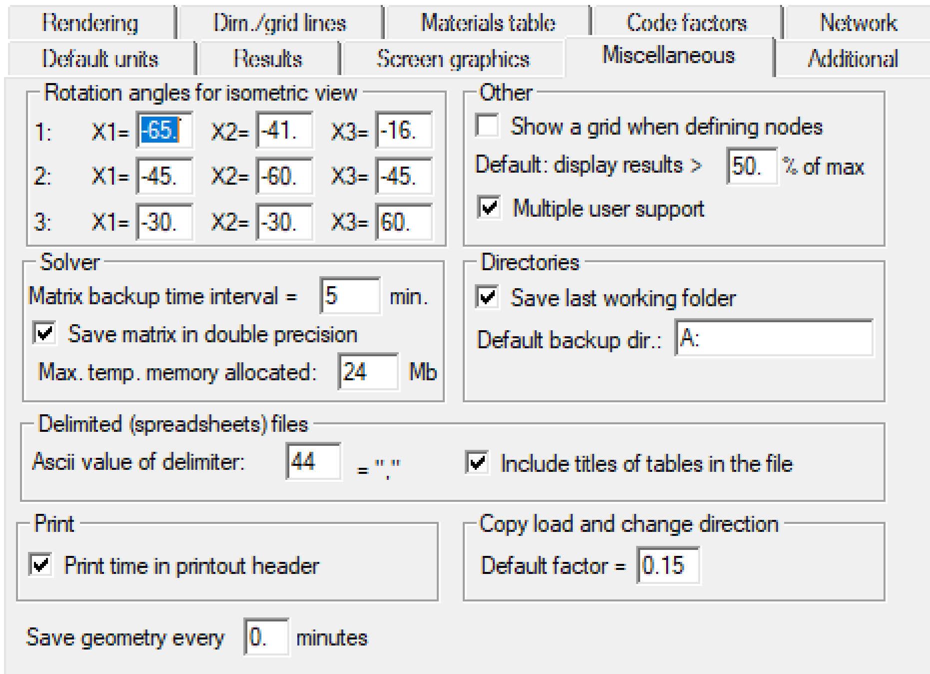


NO.	Name	Modulus of elasticity	Poisson ratio	Density	Thermal coefficient
1	CONC	2500000	0.2	2.5	0.1000E-04
2	STEE	20500000	0.3	7.85	0.1200E-04
3	CABO	19500000	0.3	7.85	0.1200E-04
4	ALUM	7000000	0.33	2.7	0.2300E-04
5	C20	2504396	0.2	2.5	0.1000E-04
6	C25	2800000	0.2	2.5	0.1000E-04
7	C30	3067246	0.2	2.5	0.1000E-04
8	C40	3541751	0.2	2.5	0.1000E-04
9	UNDF	3000000	0.15	2.4	0.1000E-04
10	UNDF	3000000	0.15	2.4	0.1000E-04

Note:

- maximum name length = 4 characters
- properties may be defined in exponential format
- additional user-defined materials may be defined in each model ***for that particular model only***.
- revising the program materials does not affect the material properties of existing models.

2.5.3.3 Miscellaneous



Isometric view

For space models only, specify the angles of the three standard isometric views of the model. The isometric views are displayed by clicking the or icons in the toolbar

Define the angles assuming that the model is displayed on the X1-X2 plane. If the model is displayed on another global plane, the program rotates it to equivalent angles.

Show grid line

For node definition:

☑ the Step "Grid" is displayed automatically when a Node definition option is selected

☐ The "Grid" is not displayed automatically.

Display only results greater than:

For graphic results only, part of the numerical values may be deleted from screen (the entire geometry and result diagram are plotted). All values less than a given fraction of the maximum result are not displayed. Specify the default ratio; the value may be revised when displaying the results.

Example:

Maximum bending moment = 12 kN m and fraction = 0.5 : Only numbers greater than 6 kN m are displayed on the screen.

Multiple user support

☑
- a single user may run more than one copy (instance) of the program simultaneously
- all Setup parameter revisions are written to the computer "Registry", per user

☐
- only one copy (instance) of the program may be started by a single user.
- all Setup parameter revisions are written to the STRAP.INI (one file per installation)

To revise a setup parameter for ***all*** users when ☑ Multiple user support is in effect:

- set this option to ☐
- revise the Setup parameter; it will be written to STRAP.INI
- set this option back to ☑

Refer also to STRAP.INI / Registry.

Backup time

The solved stiffness matrix is automatically copied to the hard disk at an interval specified here by the user. If the solution is interrupted by the user, a power failure, hardware failure, etc. the calculation may be resumed from the location of the last backup.

Double precision matrix

Set the checkbox to ☑ to save the stiffness matrix in double precision format.

Note:

- single precision is the default option and is recommended.
- double precision files are considerably larger
- ***double precision files are required only for models with relatively thick elements supported on springs. Reaction values may be inaccurate for these models if single precision is used; all other result values are accurate with single precision.***

Memory allocation

Specify the memory allocation for the solver program according to the memory in your computer. In general, Increasing the memory allocation speeds up the solution.

Set the maximum temporary allocation to [memory-32]. For example, if you have installed 128 mb memory, allocate 96 mb. However the optimum memory allocation varies according to the resident programs present.

Note:

- this option is not relevant for modern computers and operating systems (XP, Vista, etc).

Save last working folder

The current working folder is displayed at the top of the screen. This option determines the current folder at program startup:

☑ the working folder the last time the program was run.

☐ the program folder

Backup folder

Specify the default backup folder and volume. The folder is used by File management 103 options.

Spreadsheet

- **Delimiter**
  Spreadsheet files are ASCII format text files in which the numerical values are separated by a character called a "delimiter". The default delimiter is a comma (ASCII character 44). A different character may be specified by entering its ASCII number.
- **Title**
  Set the box to ☑ to add the table title rows to the delimited file.

Print options

- **Time**
  Set this option to ☑ to print the time in the printout header

Save geometry every 'n' minutes

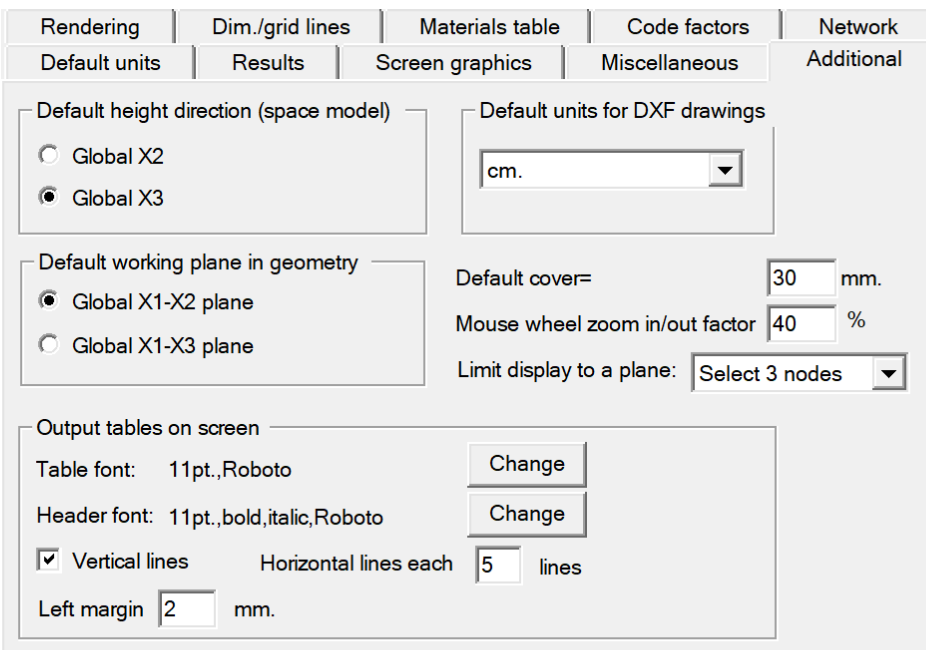
Automatically save the model geometry at the specified interval.

- the actual geometry file (GEOMnnn.DAT) is saved; if there is a power failure and you later return to geometry, the model up to the last save is the one displayed.
- the program also creates a file GEOMnnn.BAK with the current geometry every time you enter the geometry module. To restore this geometry, rename the file to GEOMnnn.DAT ***before*** entering the program ; the existing GEOMnnn.BAK file is deleted every time you enter the geometry module.

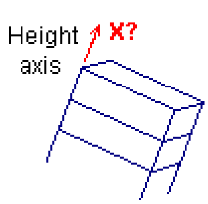
Copy load and change direction

For the option to copy an entire load case and apply it in a different direction; specify the default factor.

2.5.3.4 Additional



Default height axis



Many options ask the user to identify the height axis of the model; the program assumes that columns are parallel to this axis and floor levels are perpendicular.

Specify the default height axis for all space models.

DXF units

For the geometry toolbar option - **Display - Load DXF drawing**, specify the default unit for the DXF drawing.

Default working plane

Select the default working plane ***for space models***:
- this plane is the default global working plane for the geometry **Nodes** option.
- models generated by the Model Wizard option are created on the plane specified here.

Note that all plane models are always defined or created on the X1-X2 plane.

Default cover

Specify the ***gross*** cover for concrete reinforcement (to the center-of-gravity of the bars). This value is used only by the results module (and not by the Concrete design module).

Mouse wheel zoom in/out factor

The program changes the display scale by a set factor when the mouse wheel is rotated. The default value is 10%.

Limit display to a plane

Set the default option for the command "Limit display to a plane".

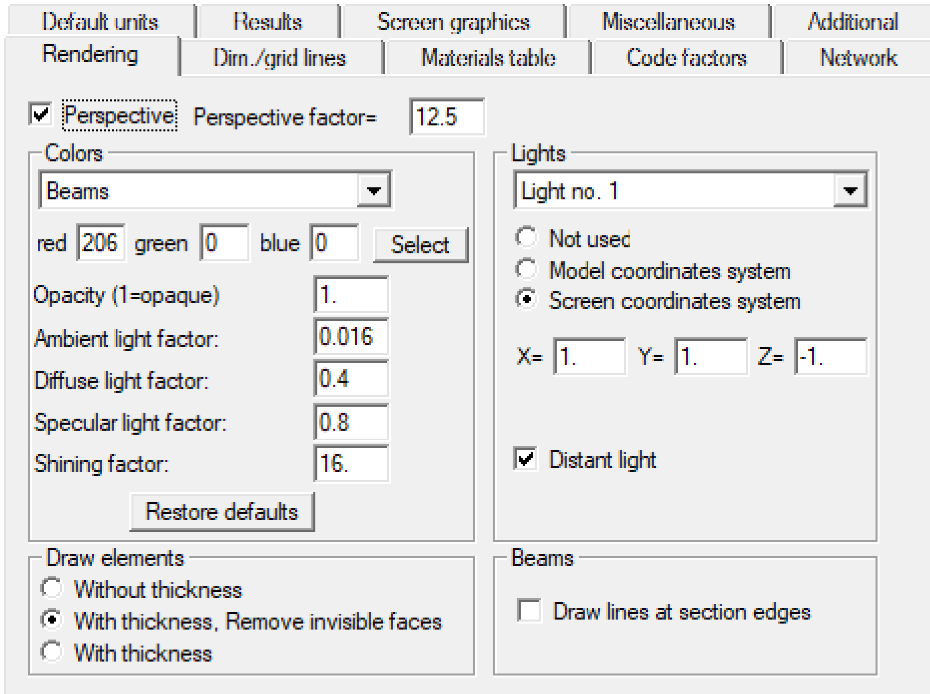
Output tables on screen

Change the font, add vertical and/or horizontal lines or set the left margin for the display of output tables on the screen. Refer to Print parameters 124 to set similar options for printing the same tables.

2.5.3.5 Rendering

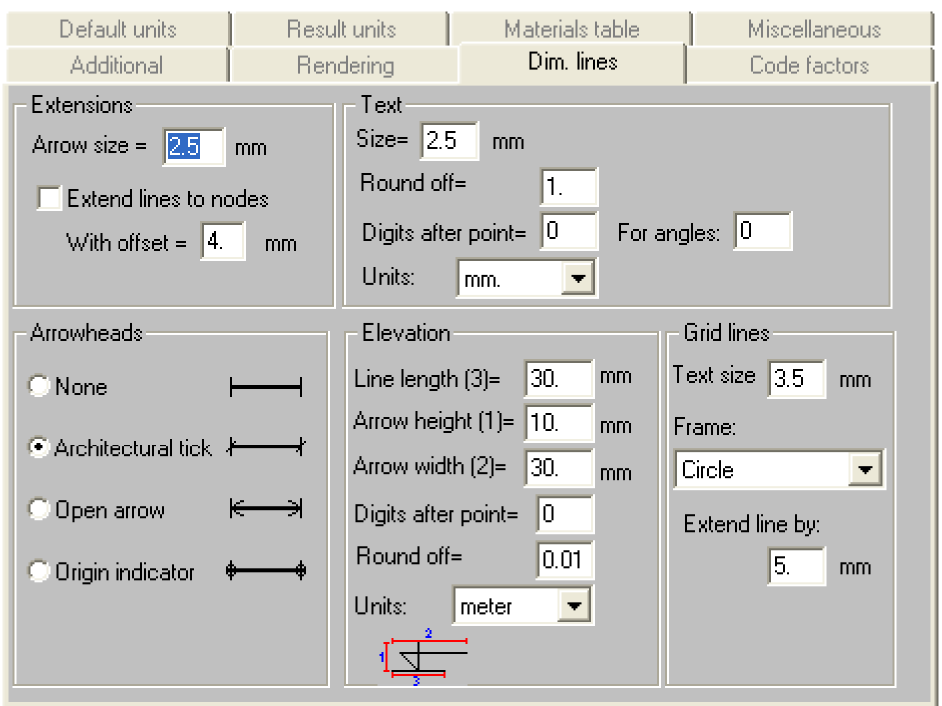
The rendering option enables you to display the current model with perspective, to remove hidden lines, to draw the beams and elements with their natural shape and to simulate shading effects caused by a light source.

The rendered model is displayed according to the values specified in the Rendering parameters option. The default values in this option have been selected after extensive testing and give good results for most models. The default values may be restored at any time by clicking the Restore defaults button.

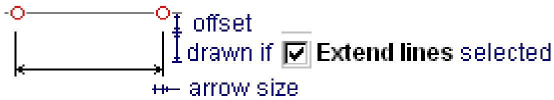


2.5.3.6 Dimension lines

Specify the default parameters for the **Draw - Dimension line** option in all *STRAP* modules



Extensions



Note:

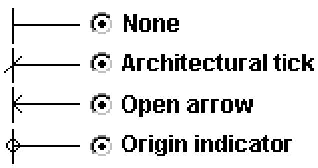
- "arrow size" is revised for the print options only and remains constant on the screen.
- "arrow size" dimension affects all extension types.

Text

- **Round off**
  Round off all dimensions and elevations to the value specified here
- **Digits after point**
  Specify the number of digits to be displayed after the decimal point for dimensions and angles. Note that this number of digits will always be displayed, even if the **Round off** value requires more digits.

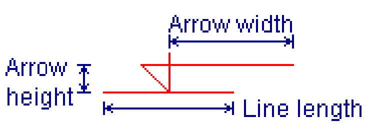
Arrowheads

Select one of the following arrowhead types:



Elevation

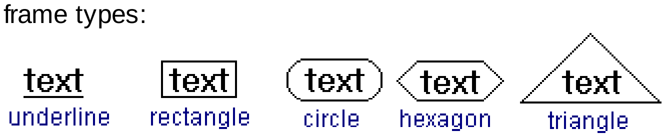
Specify the elevation mark parameters and the units for the elevation value:



Grid lines

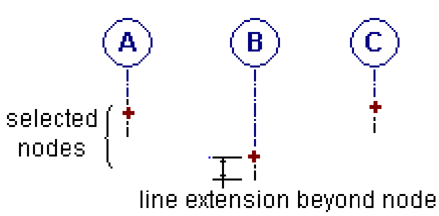
Specify the parameters for grid lines:

- Size:
  Enter the default text size according to the displayed units.
- frame types:



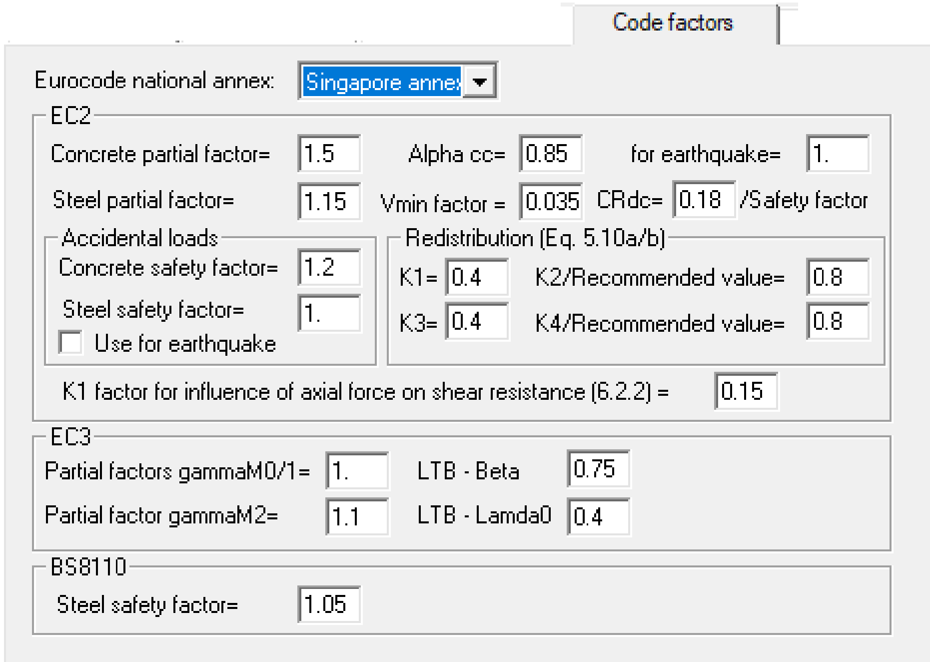
- Extensions

  Grid lines are drawn from the specified location to the selected node and are extended beyond the node by the dimension specified here. For example:



2.5.3.7 Code factors

Define the following Code related factors:



Eurocode national annex:

Select a national annex from the list; the program will revise the EC2/EC3 factors below accordingly. "No annex" uses the recommended values in the codes.

Eurocode 2:

- Concrete safety factor = γc (2.4.2.4)
- Steel safety factor = γs (2.4.2.4)
- Accidental loads safety factor = $\gamma s, \gamma c$ (2.4.2.4)
- Alpha-cc = α_{cc} (3.1.6)

  a different value for earthquake design may be defined (for columns and walls only).
- CRdc/Safety factor: $C_{Rd,c}$= (factor)/γc (6.2.2)
- Vmin factor: v_{min} = (factor) $k^{3/2}$ $f_{ck}^{1/2}$ (6.2.2)
- Redistribution: (5.5)
  - k_1, k_3
  - k_2: factor = value/[1.25(0.6+0.0014/ε_{cu2})]
  - k_4: factor = value/[1.25(0.6+0.0014/ε_{cu2})]
- Accidental loads: specify γc and γs

  check ☑ **Use for earthquake** if earthquake is considered an 'Accidental load' (columns and wails only)

Eurocode 3:

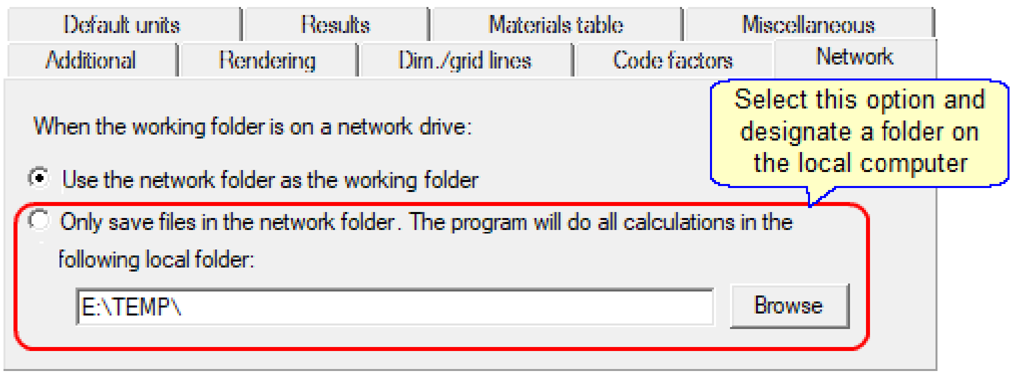
- General safety factor = γ_{Mo}, γ_{M1}
- Axial tension safety factor = γ_{M2}
- LTB - β - refer to 6.3.2.3, eq. 6.57
- LTB - $\lambda_{LT,0}$ - - refer to 6.3.2.3, eq. 6.57
- National Annex - Select **UK Annex** to design according to the BSI - "UK National Annex to EC3"

BS8110:

- Steel safety factor = γ_s

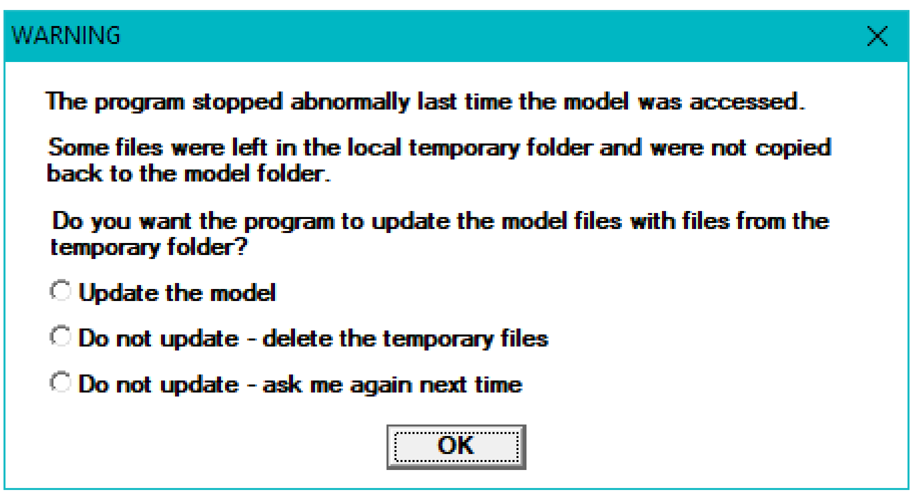
2.5.3.8 Network

Models saved on a network may now be automatically transferred to a local drive for processing. Every time the model is accessed the program automatically copies the files to the local drive, then copies them back to the network drive when the session ends. This reduces the solution time significantly in large models on busy networks.



Note:

- All models are copied to the same local drive; the files in the local drive are erased when the session ends.
- if the program crashes during the session and does not copy the files back to the network drive, the following will be displayed the next time *STRAP* is started:



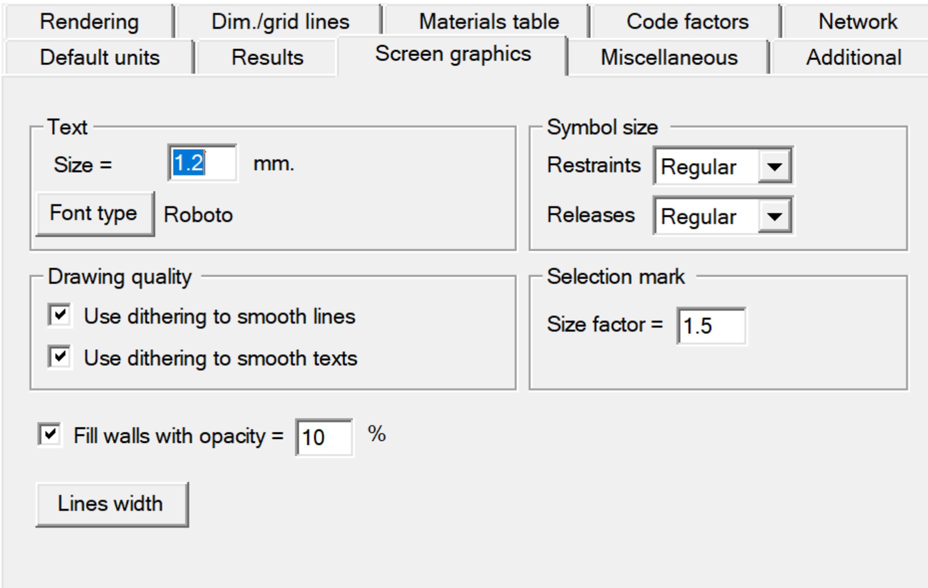
Update the model
Complete the copying of the temporary files back to the network.

Do not update - delete
Delete the temporary files. All changes made in the previous session are lost.

Do not update - ask again
The files are neither deleted nor copied. The same question will be asked in the following session.

2.5.3.9 Screen graphics

Specify default values for the graphic display:



Text

Select the font and the text size used on the graphic display.
Note:

- for certain fonts (e.g. MS Sans Serif), the program will select the the "font size" (8, 10, 12 ...) that is closest to the size (mm) specified here, hence a small change in font size may not cause a corresponding change in the displayed text.
- for other fonts (e.g. Segoe UI), the displayed font size always changes corresponding to the size specified here.

Drawing quality

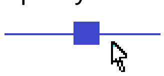
"Dithering" creates smoother lines and text.

Symbol size

Specify the symbol size for restraints - Δ - and releases - o -

Selection mark

Specify the size of the square selection mark by defining a factor for the default size:



Fill walls

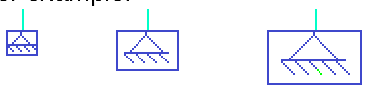
Check the box to display walls with fill color and set the opacity percentage.

Line width

Set the display line width for beams, elements, loads, result types, etc. Values are in 'pixels' (0 = 1 pixel)

Symbol size

Select size for restraints and releases.
For example:

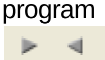


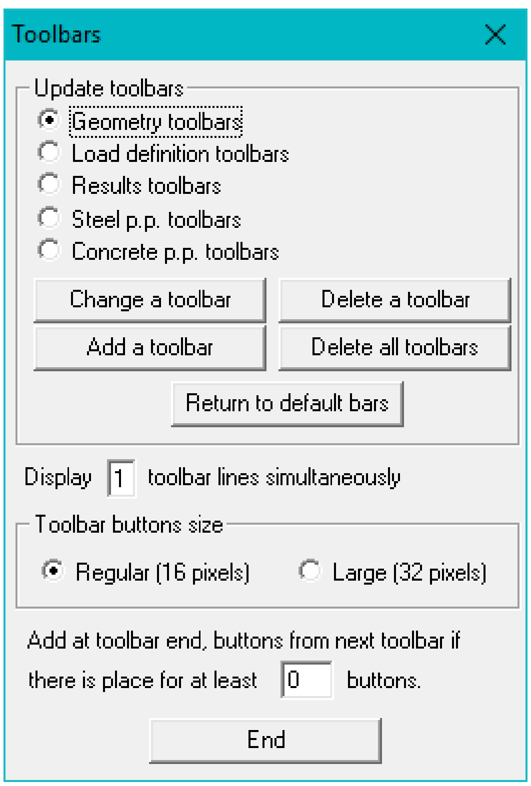
Regular Large Very large

2.5.4 Toolbars

Toolbars are displayed above the graphic display Clicking an icon on the bar calls the option that it represents. For example:

Use this option to customize the toolbars.

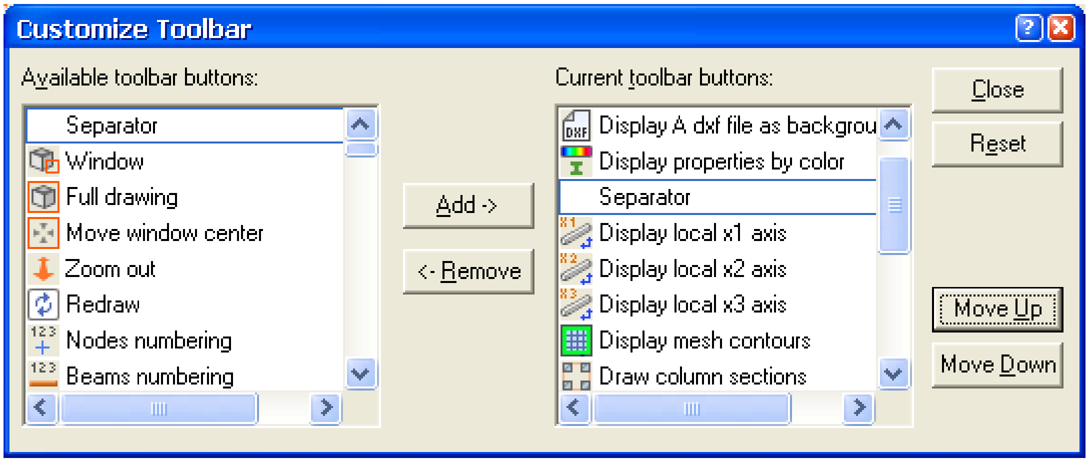
- Different toolbars are available for geometry, loads, results, steel design and concrete design.
- Up to 10 different toolbars may be defined for each module("default" bars are present when the program is installed). The program automatically displays toolbar no. 1 on the screen Clicking the  icons at the end of the bar displays the following/previous bars.
- Alternatively, display more than one row of toolbars by changing the **Display ☐ toolbar lines simultaneously** value.



To revise an toolbar or to add a new toolbar:

- select ◉ one of the modules - geometry, loads, etc.
- Click the [Change a toolbar] or [Add a toolbar] button
- select an toolbar from the list (Bar no. 1 to Bar no. 10)
- the program displays a dialog box showing all of the available icons in a the left list box and the icons

currently in the bar in the right list box:



- add an icon: click the icon in the left box, then click Add->.
- delete an icon: click the icon in the right box, then click <-Remove
- to change the order of the icons, click and highlight an icon in the right box, then click Move Up or Move Down or drag the icon up/down to the correct location.
- click the Reset button to restore the default toolbar.
- click the Close button to continue.

2.5.5 Language

Select an interface language from the list:

Note:

- the program uses the Unicode standard for displaying characters.
- some characters may not display correctly after selecting a different language. Please notify your *STRAP* dealer. To correct temporarily, Revise the "Language and regional settings" in the Windows "Control panel".

2.6 Utilities

The following utility modules are available:

- Footing design
- Create/edit a Steel sections table
- Compute section properties
- Connection design
- Combine results of 2 projects
- Recreate models list
- AutoStrap

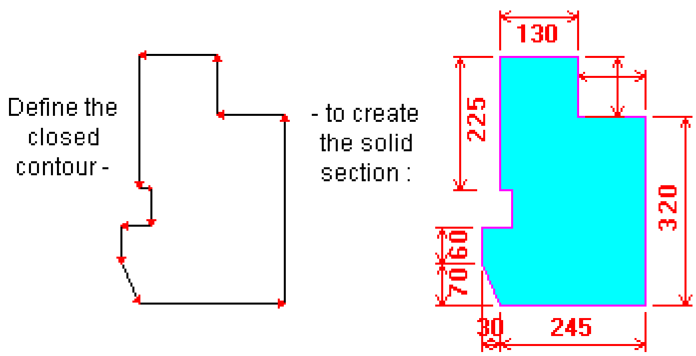
- Convert a STAAD Input file to STRAP
- Create a STAAD Geometry Input File
- Create a StruCAD Neutral File
- Create a SDNF File
- Create an AFES File
- Import a SIX File

Compute section properties

Use the CROSEC program to create arbitrary sections and to calculates their section properties (area, moment-of-inertia, center-of-gravity, etc).

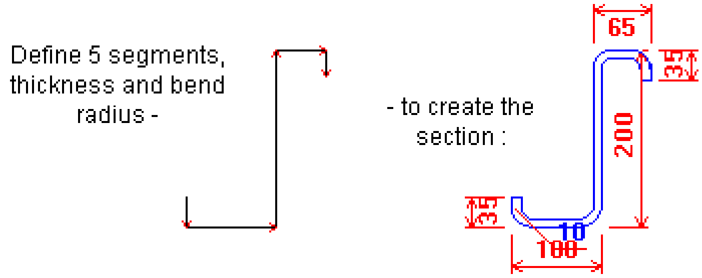
Two general types of sections may be defined:

- **Solid sections:**
  Sections formed by a closed contour. For example:



  - Several separate contours may be defined to form a section; the additional contours may be specified to be "holes".
  - The properties of Solid sections may be copied to *STRAP* geometry.
  - Properties calculated include: Torsional moment-of-inertia (exact).

- **Line sections:**
  Sections composed of a series of connected lines, each with a specified thickness. For example:



- Segments may be defined in any direction (diagonal).
- Several separate sections may be defined to form a section; the additional sections are called "subsections". Each subsection may have a different thickness and bend radius.
- The properties of a Line section may be copied to *STRAP* geometry
- Tables of cold-formed sections may be added to the *STRAP* property tables; the user first defines the dimensionless general shape and then enters the dimensions for each segment of every section in the table. Different segments may be defined as "equal" (only one of the dimensions must be entered).
- Properties calculated include: Shear center, warping constant, torsional moment-of-inertia and torsional-flexural buckling constant

To display a demo video that explains how to create a cold-formed section table and how to transfer it to STRAP:

- *click on* *to start the video*
- *then click on* *to enlarge the display.*

Recreate models list

Use this option to recreate a corrupted model list.

- The model list is stored in the file **ZZMODEL.DIR** (each working folder has a different ZZMODEL.DIR file).
- This option scans the current folder to locate all model files (GEOMnnn.DAT, STATnnn.DAT, etc) and rebuilds the model list.

AutoSTRAP

Run the AutoSTRAP program to convert DXF drawings to *STRAP* models. if this option is selected AutoSTRAP creates the model in the current folder and then automatically displays the *STRAP* geometry screen with the created model.

Convert a STAAD file to STRAP

Refer to - STAAD file import 145.

Create STAAD file

Create a STAAD format geometry file for the current model.

Create a StruCAD / SDNF file

Create a structural steel detailing file in one of the following formats:

- STRUCAD
- SDNF

Note:
- The program writes the model geometry and the beam end results to the files. The steel section written to the file is the current one in *STRAP* ***geometry***; different sections selected in the Steel Postprocessor must be transferred to the geometry using the **Exit and update geometry** option in the postprocessor.
- the height direction must be **X3** for a STRUCAD format file.

Create an AFES file

Create an export file for the AFES foundation design program.

Import an SIX file

Import a geometry file created by the "SI Xchange" program for REVIT Structure and STRAP.

STRAP creates a backup *xxxx.BAK* of the *xxxx.SIX* file because the importing process changes the SIX file.

Note:
- you can add new options 146 to this menu. This is convenient if you want to run a *STRAP* batch operation according to the *STBatch* utility, but any Windows "Target" command can be initiated (i.e. run any other program).
- The STBatch utility can be used for running *STRAP* in batch mode or for generating ASCII files with geometry, load and/or result data in user specified format.

2.6.1 Combine results of 2 projects

Use this utility to combine the result files from several models.

This option is useful in models where the geometry changes for different loading cases (e.g. supports, properties, etc.) and the structure must be run as two or more separate models, but where it is necessary to search all loading cases from all of the models in the various design post-processors.

***** the Stage option can achieve the same result more efficiently *****

Note:
- The number of nodes or elements in the two models that are combined do not have to be identical but ***the node/element at the same location in both models must have the identical numbering***.
- the program combines three files: RESLTnnn.DAT, FORCEnnn.DAT, STATnnn.DAT
  By combining STATnnn.DAT, the program adds the load commands from the load cases of the second model to those of the first. If you later decide to solve the first model again, remember to erase the unnecessary load cases (the load commands are required by the various post-processors to calculate the span moment diagrams).

To run the utility:
- Highlight one of the models that are to be combined; select the **larger** of the two models (or the largest of **all** of the models if you are combining a series of models).
- Select **Utilities** in the toolbar
- Select **Combine results of 2 projects**
- The program displays a list of the models that have been solved;select the model whose results will be added to those of the first model selected.

The process may be repeated to add additional models to the combined ones; the models must be added in order of decreasing size (largest to smallest).

Note:

* ***Only the first model selected is revised***

2.6.2 Convert STAAD File

This utility module converts STAAD input files to ***STRAP*** format. Geometry, loading and load combination commands are translated.

* select **Utilities** in the toolbar.
* select **Convert a STAAD input file to STRAP.**
* type in the name of the STAAD file

Note:

* The program creates the *STRAP* **GEOMnnn.DAT, STATnnn.DAT**, and **COMBnnn.DAT** files.
* STAAD commands with syntax errors are not translated. Error messages are listed on the screen and written in an ASCII file ERRS.LST.
* The utility does not check that entire geometry has been defined (e.g. restraints).

Commands not translated by *STRAP**:***

STAAD Options not available in STRAP:

* elements with variable thickness
* partial end release and shear release at beam end nodes
* CABLE MEMBER
* MASTER/SLAVE
* WIND LOAD
* PRESTRESS LOAD
* partial load or linear loads on individual finite elements

STAAD input options that are in STRAP result/postprocessor modules:

* All DESIGN commands
* All commands related to printout options
* All commands relating to Mode Shape and Natural Frequency Analysis
* UBC LOAD - generated by Seismic Response Analysis
* All DRAW commands
* Commands related to solution options

Commands not translated:

* SEPARATOR command - ";" must be used
* MESH command with 8 corners
* A UNITS command may not be defined within a load case
* SUBST commands are ignored
* PERFORM ROTATION commands are ignored
* The following sections are ignored:
    * sections in user tables
    * sections in steel tables other than American
    *][, COMPOSITE, section with slab, TUBE
* Beam load with offset from the shear centre: the load is applied but the additional moment due to the offset is ignored
* AREA and FLOOR commands are ignored
* STRAIN commands are ignored
* FIXED END commands are ignored
* SET Z UP commands are ignored

2.6.3 Add new options

To add new options to the Utilities menu:

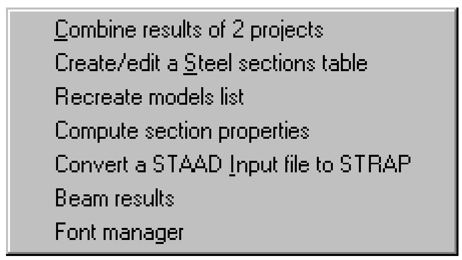
- Edit the file **STRAP.INI** in the program folder
- Add a section **[UTILITIES]**
- Add the line **NumMenus=*n*** , where n = the number of options that are added
- For each option, add the lines:
  Name*n*=*menu_text*
  Command*n*=*target_command*
  Note that the default folder for the commands is the *STRAP* program folder.

Example:
Add two commands to the menu:
- Run the *STBatch* file **Report1** in the program folder; the option in the menu is "**Beam results**"
- Start the program **abcxyz.exe** in folder D:\abc; the option in the menu is "**Font manager**"

Add the following lines to STRAP.INI:

[UTILITIES]
NumMenus=2
Name1=Beam results
Command1=stbatch report1
Name2=Font manager
Command2=d:\abc\abcxyz.exe

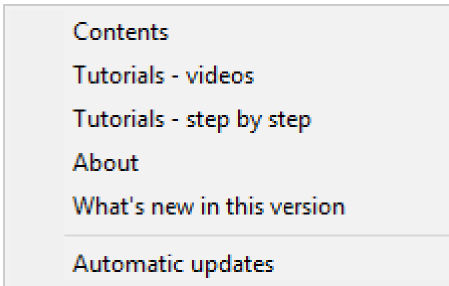
The **Utilities** menu is now displayed as:



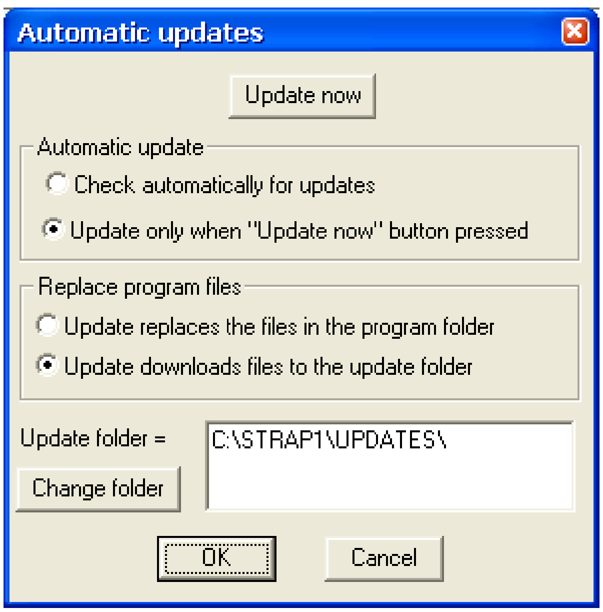
2.7 Automatic updates

The program files can be updated directly from the *STRAP* web site. The update may be either automatic or manual.

- Click "Help" in the Menu bar at the top of the screen
- Select "Automatic updates":



- Select the options:



Check automatically for updates
The program automatically checks the files when the program is started and downloads any that have been updated since the previous download.

Update only when "Update now" button pressed

The program checks the files and downloads the updated ones only when Update now in this menu is pressed.

Note that this option refers to two folders:

- the program folder - normally **\STRAP1**
- the updates folder - normally **\STRAP1\UPDATES** (click Change folder to select a different folder)

Update replaces the files in the program folder
The program copies the files to the program folder ***and*** copies the old version from the program

folder to the updates folder and renames it to ***.OLD (you can copy the old file back to the program folder in the event that there is a problem with the new file).

Update downloads the files to the Update folder
The program copies the updated files to the Updates folder. You must then copy them manually to the program folder

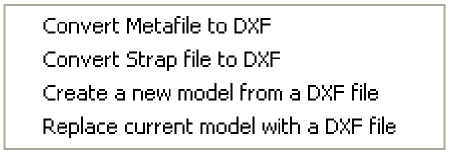
---

Note:

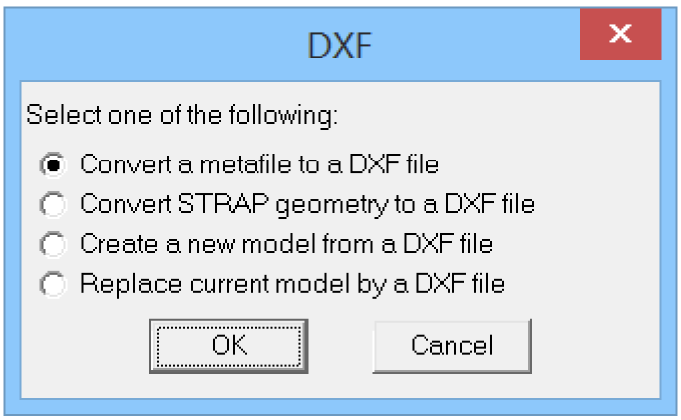
- The program that copies the files is named **UPDSTR.EXE** - ***adjust your firewall settings*** to allow the program to download the files.
- if STRAP is installed in a folder where the user does not have permission to create or modify files (e.g. on a network):
  - select **Update downloads the files to the Update folder** and specify a folder to which you have access rights.
  - ask the network administrator to copy the files to the program folder.
- The first time the program is run after it is installed it checks whether there is an internet connection and whether it can write to the program folder (\STRAP1). If the answer to ***both*** tests is 'yes', the program sets the default option to **Update replaces the files in the program folder**; otherwise **Update downloads the files to the Update folder** is the initial default option.
- When you initially install the program it creates a file named **UPDLST.TXT** (in the Updates folder) that contains a list of the files to be checked. If you later purchase and install a new program module its files will not be in the list and so it will not be updated automatically. Delete **UPDLST.TXT**; the program will create a new one the next time you Update (with the new module files names).
- both the program files (*.EXE) and the Help files (*.CHM) are updated.

2.8 DXF options

Create a ***STRAP*** model from a DXF file, create a DXF file from a model or convert a Metafile to a DXF file.



If  is clicked in the icon bar:



2.8.1 Metafile to DXF

Create a two-dimensional DXF file of any ***STRAP*** graphic display.

- the program displays a list of the Metafiles (extension .WMF, .EMF) in the current folder; select the file to convert.

Note:

- the metafile file is created by selecting **Print drawing** in the **Output** pull-down menu (on the menu bar) and then selecting **Metafile** in the '**Send output to**:' box.
- It is recommended that **Print/edit saved drawing** be used to create DXF files.

2.8.2 Convert STRAP to DXF

This module converts the ***STRAP*** geometry file to three-dimensional DXF files. The generated DXF files are in ASCII format.

This option differs from the ***STRAP*** **Print drawing** option (and subsequent **Convert Metafile to DXF**) in that it creates a full 3D model instead of a 2D image of the current screen display.

- the program works on the current ***STRAP*** model
- plane models generate a 2D DXF drawing
  space models generate a 3D DXF drawing
- Beams generate a DXF "LINE" command.
  Elements generate a DXF "3DFACE" command.
- Each ***STRAP*** property is assigned to a different layer and drawn with a different colour in the DXF file.
- Dummy beams and elements are drawn in a separate layer.
- The following is ***not*** transferred to the DXF file:

Dimension lines, sections, materials, text (numbering, etc).

2.8.3 Convert DXF to STRAP

The AutoSTRAP program is a more powerful option for creating STRAP models from DXF files.

This option converts a DXF format file to a ***STRAP*** geometry file. The DXF files may be in ASCII or binary format. It identifies the LINE and mesh commands and converts them to beams or elements, as requested by the user.

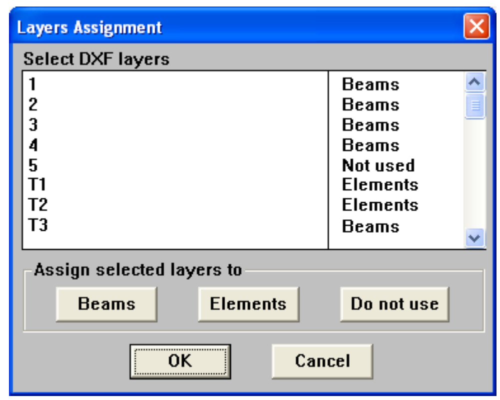
Refer to - Convert DXF - General 153

Select DXF layers to import:

The lines in each layer in the AUTOCAD drawing may be -

- ignored
- converted to beam elements
- converted to finite elements

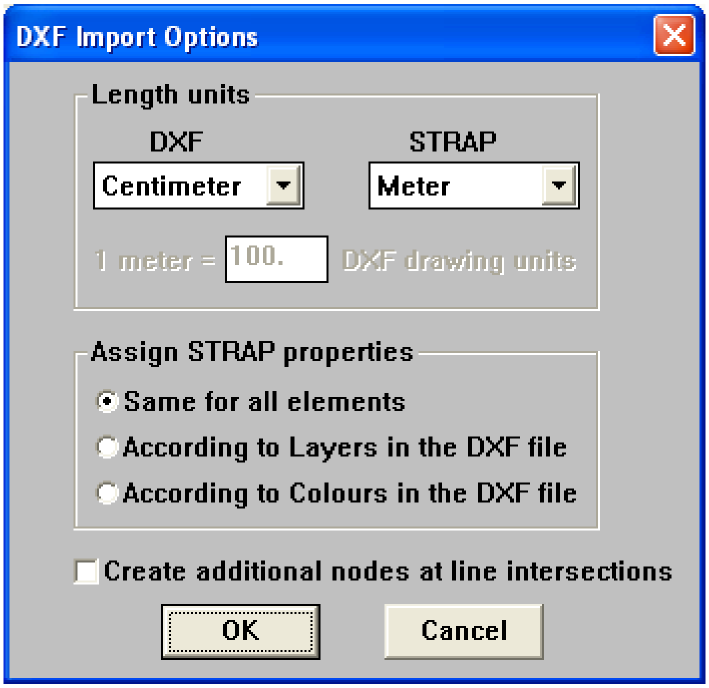
Click and highlight a layer, then click Beams , Elements or Do not use .



Note:

- if 3DFACE, polygon mesh or polygon face elements are converted to beams, the program 'explodes' them and generates beams from all visible borders.
- if a layer was "Freezed" or turned "Off" at the time the DXF file was created, "Do not use" will be the default for that layer.

Specify units and assign colours or layers to STRAP property groups:



Units

Specify the length of unit assumed in the DXF drawing and the default length unit in ***STRAP***. The program then converts the DXF drawing dimensions to ***STRAP*** length units.

Note:

- It is important that the correct DXF length unit be specified. For example, if the DXF drawing is defined in meters, but **Centimeter** is erroneously selected in the menu, a dimension of 5.00 meter in the DXF drawing is assumed to be 5 centimeters. If meters is specified as the ***STRAP*** default length unit this dimension is then converted to 0.05 meters.

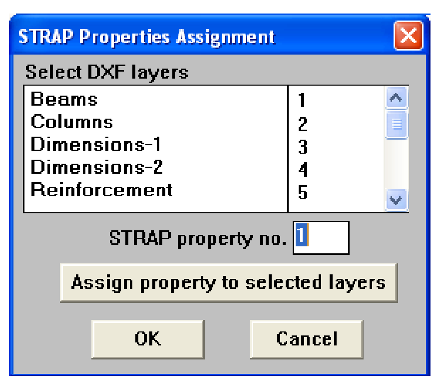
Assign STRAP properties

Same for all elements
All elements (beam and finite elements) are assigned with property group 1.

According to layers in the DXF file
Different ***STRAP*** property group numbers may be assigned to different elements if the different element types are drawn in different layers.

- Select a layer (or layers) in the list box.
- Type in a property number in the **STRAP property no.** text box
- Click the **Assign property to selected layers** button.

The property numbers at the right side of the list box are then updated by the program.



According to colors in the DXF file

Different ***STRAP*** property group numbers may be assigned to different elements if the different element types are drawn with different colours.

- Select a colour (or colours) in the list box.
- Type in a property number in the **STRAP property no.** text box
- Click the **Assign property to selected colours** button

The property numbers at the right side of the list box are then updated by the program.

Note that the search is carried out over ***all*** layers; beams and elements are assigned with the same property number if they are drawn with the same colour.

Create nodes at line intersections

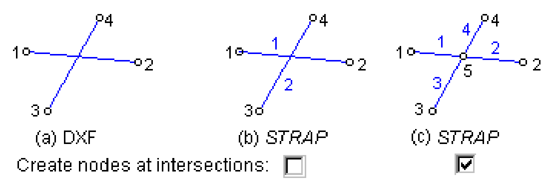
The program searches for intersection points of any two entities in the DXF drawing over If the option is set to:

☑ ***STRAP*** nodes are created at the intersection points and in finite element layers, the 3DFACES are exploded to lines. (In beam layers they are exploded automatically).

☐ ***STRAP*** nodes are not created at the intersection points. ***This is the faster option but must be used carefully***, i.e. only when there are no intersection points that are not the end/corner points of entities.

The following are examples of node, beam and element generation according to the "intersection" option selected.

Beam layer:

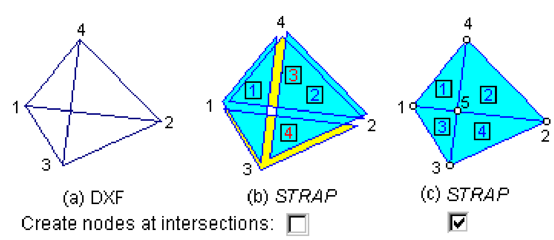


In Figure (a), DXF line 1-2 intersects line 3-4.
If the intersection option is set to:

☐ node 5 is not created and only two beams are generated - Figure (b).

☑ the program creates a new *STRAP* node at the intersection point 5 and creates four beams -1-5, 2-5, 3-5, 4-5 - Figure (c).

Finite element layer:



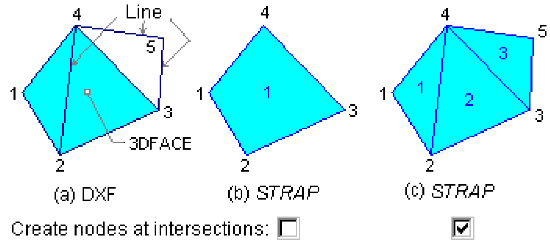
Example 1:

In Figure (a), there are 6 DXF lines; line 1-2 intersects line 3-4.

If the node intersection option is set to -

☑ the program creates node 5 and 4 triangular elements as shown in Figure (c).

☐ the program creates 4 triangular elements as shown in Figure (b).



Example 2:

In Figure (a), there are 3 DXF lines (2-4, 3-5, 4-5) and 1 3DFACE entity (1-2-3-4).

If the node intersection option is set to -

☑ the 3DFACE is exploded to lines and the program creates 3 triangular elements as shown in Figure (c).

☐ the program creates 1 quad element as shown in Figure (b) and ignores the lines as ***the program does not close lines with a 3DFACE to form an element***.

2.8.3.1 Convert DXF - General

This option converts a DXF format file to a ***STRAP*** geometry file. It identifies the LINE and mesh commands and converts them to beams or elements, as requested by the user. The DXF files may be in ASCII or binary format.

Note:

- each Autocad "layer" may be specified as containing beams or elements 150.
- ***STRAP*** property group numbers may be assigned to the generated beams/elements according to Autocad layer.
- ***STRAP*** property group numbers may be assigned to the generated beams/elements according to Autocad line colour

The program handles the different Autocad entities as follows:

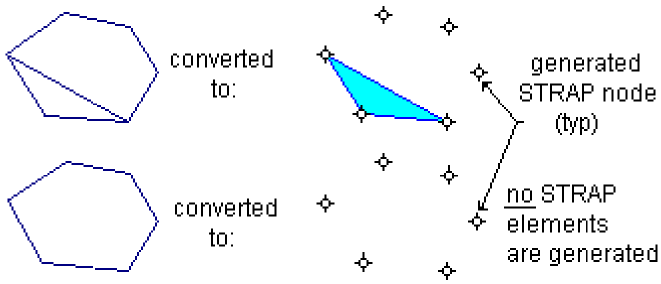
- Beam layer:
  - each line in Autocad is converted to a ***STRAP*** beam.
  - each Autocad mesh element (3DFACE) is exploded to a series of lines which are converted to beams.

- Element layer:
  - areas enclosed by three or four lines and creates triangular or quad finite elements.
  - each 3DFACE or surface of a mesh generates a single ***STRAP*** quad or triangular element.

The program also handles intersections between lines, between elements and between lines and elements.

Note:

- DXF space models create ***STRAP*** space models.
  DXF plane models (all Z coordinates = 0) create ***STRAP*** plane .
- The program defines element local axes according to the ***STRAP*** defaults.
- The program creates quad finite elements wherever possible; non-planar quads are divided into two triangles.
- Each "3DFACE" element, polygon mesh or polyface mesh surface generates one ***STRAP*** finite element.
- Double lines are treated as single lines.
- Concave elements defined by lines, 3DFACE, polygon mesh or polyface mesh are ignored.
- When creating finite elements the program accepts only quads or triangles. Nodes are created at intersection points, even if no elements are generated: For example:



- Program capacity:
  - the program does not generate models that contain more nodes or elements than the *STRAP* program capacity.
  - the program cannot handle DXF files that exceed the following limits:
    maximum number of line end points = 32,000
    maximum number of lines = 32,000
    maximum number of blocks = 600
    maximum number of line end points in all blocks = 16,000
    maximum number of lines in all blocks = 16,000
    maximum number of vertices in any polyface mesh = 600

    Note that if one end of a LINE is at the same location as an end of the ***previous*** LINE, only one end point is considered for the above limits.
  - the program cannot handle DXF files with coordinates greater than 1000 meters (rounding off errors would occur); a warning message is displayed and the program terminates. Move the drawing to the (0,0) coordinates in your drafting program and import again.

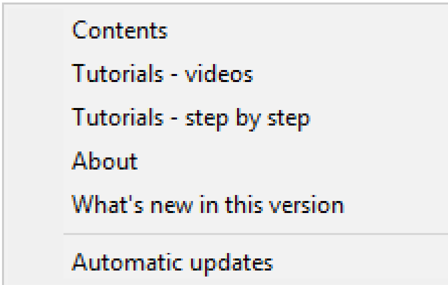
- The program recognizes the following AUTOCAD commands and entity types:
  LINE (or 3DLINE in old AUTOCAD versions)
  TRACE (transforms it to a central line)
  3DFACE
  POLYLINE (2D and 3D)
  polygon mesh
  polyface mesh
  blocks (but only the above elements are converted.)

- The program ignores the following AUTOCAD entities:
  circle, arc, text, dimension lines, solids and hatching.

2.9 Tutorial videos

To display the list of videos:

- Click "Help" in the Menu bar at the top of the screen
- Select "Tutorial videos"



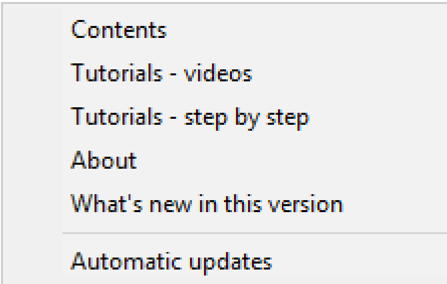
To display a video :

- *click on*  *to start the video*
- *then click on*  *to enlarge the display.*

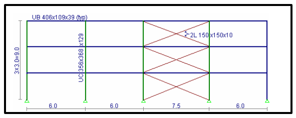
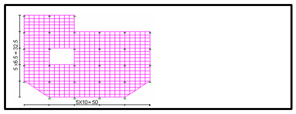
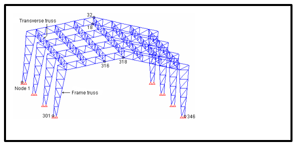
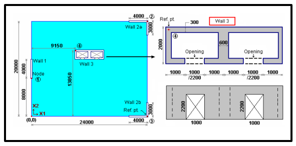
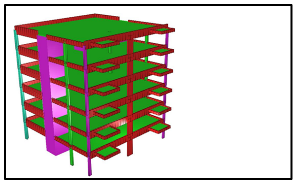
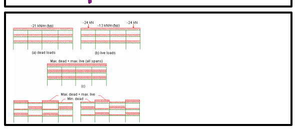
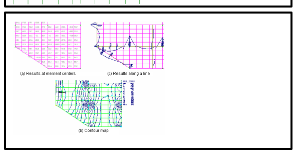
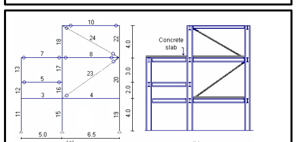
2.10 Tutorials - step by step

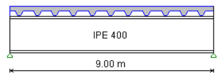
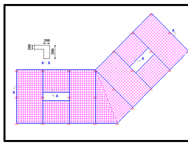
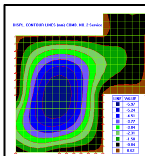
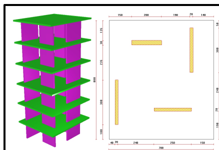
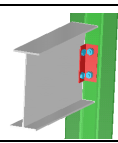
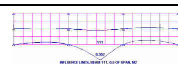
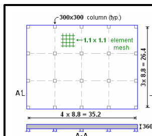
To display the list of step-by-step tutorials:

- Click "Help" in the Menu bar at the top of the screen
- Select "Tutorials- step by step":



- Select on of the following tutorials:

Plane frame - 1 • Wizard • copy geometry		***Plane frame - 2*** • Nodes - line equal • combinations
Plane grid - 1 • Element mesh - floor slab		***Plane grid - 2*** Refine an element mesh
Space frame - 1 • Wizard • submodel • copy geometry		***Space frame - 2*** • cylindrical coordinate system • rotated local axes • end releases
Space frame - 3 • define wall section • add wall to model • define slab with dummy elements		***Space frame - 4*** • Nodes - equations • Copy and rotate geometry
Submodel - 1 • attach a submodel to the Main model		***Submodel - 2*** • create submodel • add several instances • modify connection points • define loads.
Chess loads • chess loads		***Moving loads*** • global loads • moving loads
Results - elements • Results at element centres • Contour map • Results along a line		***Results - combinations*** • load case groups • load case combinations
Steel design - hot rolled sections • limit to section type • identical beams • intermediate supports • combined beams		***Steel design - cold formed sections*** • define new section • limit to section type • identical beams • intermediate supports • combined beams

Steel design - composite beam • Geometry: define composite section • Steel: ▫ select beam for composite section ▫ deflection parameters		***Concrete - beams & columns*** • Define beams and columns • Create column table
Concrete - slab detailing • define 'spaces' • add to drawing • create bar schedule		***Solid section - column*** • import DXF section into *CROSEC* • define corner bar & group locations in section
Concrete - slab deflections • calculate cracked section properties, immediate and long-term deflections.		***Dynamic analysis - walls*** • add weights • calculate mode shapes • seismic analysis • transfer load cases to *STRAP*
Dynamic - seismic analysis • Code seismic analysis • transfer results to *STRAP*		***Dynamic - time-history*** • mode shape calculation • define time-history function • display results and transfer to *STRAP*
Connections - steel • design beam-column connection • design column base plate		***Bridge design - lanes*** • Create Lanes
Bridge design - results • display influence lines • define Lane loads • define Load cases • transfer results to *STRAP*		***PRESTRESS - beams*** • define beam • define Stages • specify cable trajectory
PRESTRESS - slabs • define slab boundaries • define Stages • specify cable trajectory		

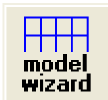
Part III

Geometry

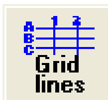
3 Geometry

3.1 Preliminary menu

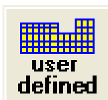
There are three methods for defining a new *STRAP* model:



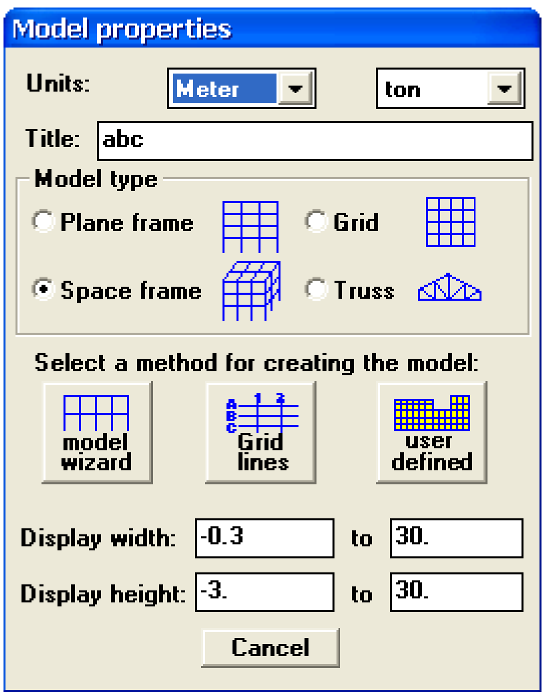
Select a model from the *STRAP* library of standard structure types. Define the geometry, loads and load combinations. The basic model created by the wizard may be revised later using any of the *STRAP* geometry and loading definition options.



Create the orthogonal grid lines as the basis for the new model. Only nodes are defined in this option.



The program displays a blank screen. Define the geometry and loads from scratch using any of the *STRAP* geometry and loading definition options.



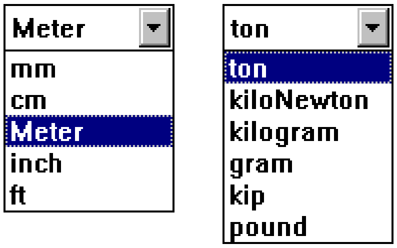
Units

By default, data is input and results are displayed according to the default force and length units selected here. The units may be revised elsewhere without changing the default units:

- Section and material properties may be defined according to different units.
- Results may be displayed according to any user specified units.

Select:

Length units: Force units:



Note:

- If the default units are revised for an existing model, the program only adjusts the geometry values to the new units; loads are not revised. The program will request authorization to revise the coordinates (materials and properties are always revised).
- The program always uses the default units to calculate the scale of a drawing to be sent to the plotter/ printer.
- If **Foot** is selected as the default length unit, the program automatically assumes **Inch** as the default unit for Material and Property definition, and **kip/in²** for stress results output.

Title

Update the model title. The title defined here appears in the model list in the ***STRAP*** main menu.

Model type

Although ***STRAP*** solves general three dimensional models, this option allows you to specify two-dimensional models or trusses as such. The program then restrains unnecessary degrees-of-freedom, allows only loads in relevant directions, etc. and displays only the relevant options in the program menus.

Select one of the following model types:

Plane Frame
- two-dimensional frame; all loads act in the plane of the model.
- plane stress finite elements

Plane Grid
- two dimensional grid; all loads act perpendicular to the model.
- plate bending finite elements

Space Frame
- general three-dimensional space model
- combined plane stress and bending elements.

Truss
- general three-dimensional space truss; all beam elements are assumed pinned at both ends. Finite elements cannot be defined

Display width/height

Enter the approximate dimensions of the model in the current units. This information is required for the initial graphic display. Note that the screen dimensions may be revised at any time.

3.1.1 Model wizard

The program contain a library of standard structure types such as plane frames, grids, and various truss shapes. The basic geometry and loads of these structures can be defined by inputting a limited number of parameters such as number of bays, storey height, etc. The geometry and loads defined in the wizard are then transferred to the regular geometry and load modules where the model may be completed and revised.

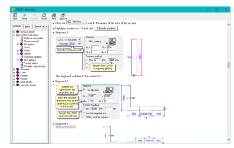
The program automatically assigns property groups to the beams and elements and defines supports.

After the parameters have been defined, the wizard creates and displays the model graphically. Options are available at this stage for revising all parameters and dimensions. ***Note that revising the dimensions at this stage does not change the basic shape of the model***. For example, if the length of a single panel is revised in a "Triangular truss", the top chord remains a straight line.

Continue 161 to an example of the definition of a model using the wizard, or -

Plane Frame 164
Plane Grid 165
Space Frame 165
Truss 166

To display step-by-step tutorials that explain how to use Wizards:
- ***Plane frame - 1***
- ***Space frame -1***

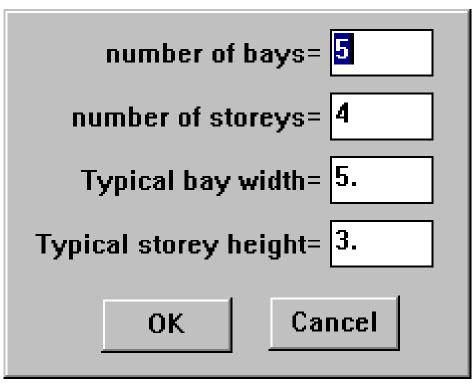


To add more models to the wizard library, refer to Wizard - Add new models 1220

3.1.1.1 Example

Geometry

The model geometry is created by defining a limited number of parameters. For example, the nodes and beams of the **Plane frame** can all be defined by specifying four parameters: the number of bays, the number of storeys, typical bay width and typical storey height.



Enter the parameters and click the 

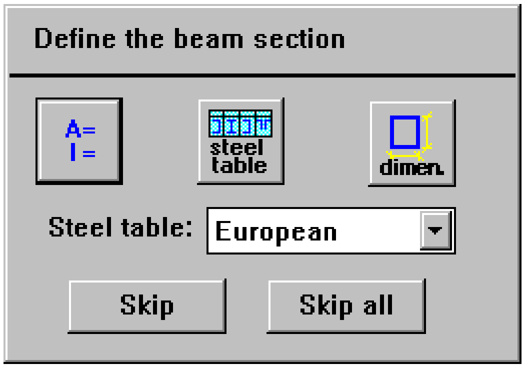
button.

The program creates the model according to the parameters (after property groups and load cases are defined), displays it on the screen and allows limited revisions 163 to the geometry via a Dialog Box at the bottom of the screen.

Note:
- the program creates the models on the X1-X2 global plane. To create the model on the X1-X3 plane, select this plane in the **Setup - Miscellaneous - Additional** option in the *STRAP* model list screen.

Properties

The following menu is displayed only if property commands were included in the wizard file for the model wizard that you selected. A different menu is displayed for each property group; all *STRAP* property definition options are available.



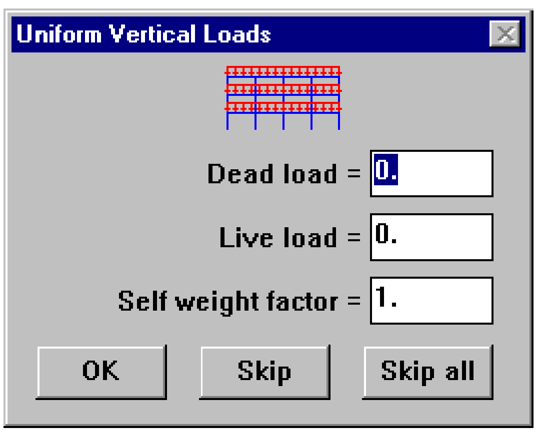
- do not define the properties for the current group

- do not define the properties for all remaining groups; skip to loads.

Loads

The following menus are displayed only if load commands were included in the wizard file for the model wizard that you selected.

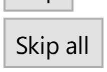
The menus prompt for various load values and load combination factors. The program then automatically generates a series of load cases based on the values (the loads can be viewed only in the Loading module). For example, "Plane frame":



Dead/Live load - uniform load per unit length (in current units)

Self weight factor - enter '0' if you do not want to apply self-weight as a dead load

- do not define the loads for the current case



- do not define the loads for all remaining cases.

Two additional similar dialog boxes are then displayed:

<u>**Dialog box title**</u>	<u>**Data requested**</u>
Uniform wind loads:	Wind on left column Wind on right column
Combination factors:	Factors for: - Dead+Live - Dead+Live+Wind - Dead + Wind

The program generates the following 9 load cases:

Load case	Description
1 - Dead load	Dead load X 1.0
2 - Live load	Live load X 1.0
3 - Wind load	Wind load X 1.0
4 - Alternate dead and live - 1	Staggered loads on alternate spans: - Dead*max. factor + Live *factor - Dead * min. factor
5 - Alternate dead and live - 2	Similar to 4
6 - Dead + live	Dead* max. factor + Live* factor - all spans
7 - Dead + live + wind	Loads x 2nd set of factors (above)
8 - Alternate dead + wind - 1	1.0*Dead + Wind * 3rd set factor
9 - Alternate dead + wind - 2	Dead * max. factor + wind * 3rd set factor

Loads may be defined for the following Wizard models:

Plane frame 1212	Frame truss 1206
Vierendael 1219	Triangular rafter 1216
Truss	Cross diagonal truss 1205
Truss on columns 1218	Flat Warren truss (a) 1208
Triangular truss 1215	Flat Warren truss (b) 1208
Parallelogram truss 1212	Howe truss 1210
Trapezoidal truss 1217	Continuous beam 1204

Refer to Loads for more information on load input.

3.1.1.2 Revise dimensions

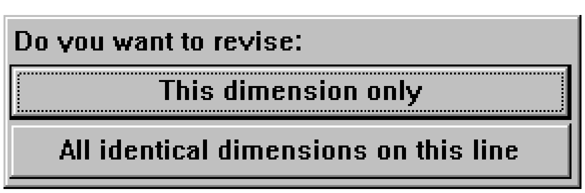
The program creates the model according to the parameters, displays it on the screen and allows limited revisions to the geometry via a Dialog Box at the bottom of the screen. For example, the plane frame wizard:

No. of bays/storeys

Select this option to revise the parameters displayed on the bottom line. The cursor jumps to the bottom of the screen. When OK is selected, the model is redrawn according to the revised parameters.

Dimensions

Use this option to revise dimensions displayed on the dimension lines. Move the near the dimension that you want to revise until it is highlighted by the small rectangular blip ■; click the mouse and type in the new dimension.



- Select the first option to revise the selected dimension only. To revise all identical dimensions on the line, select the second option.
- Select another dimension to be revised or press [Esc] to end the corrections; the program redraws the model with the new dimensions.

Dimensions in grey are for information only and cannot be revised.

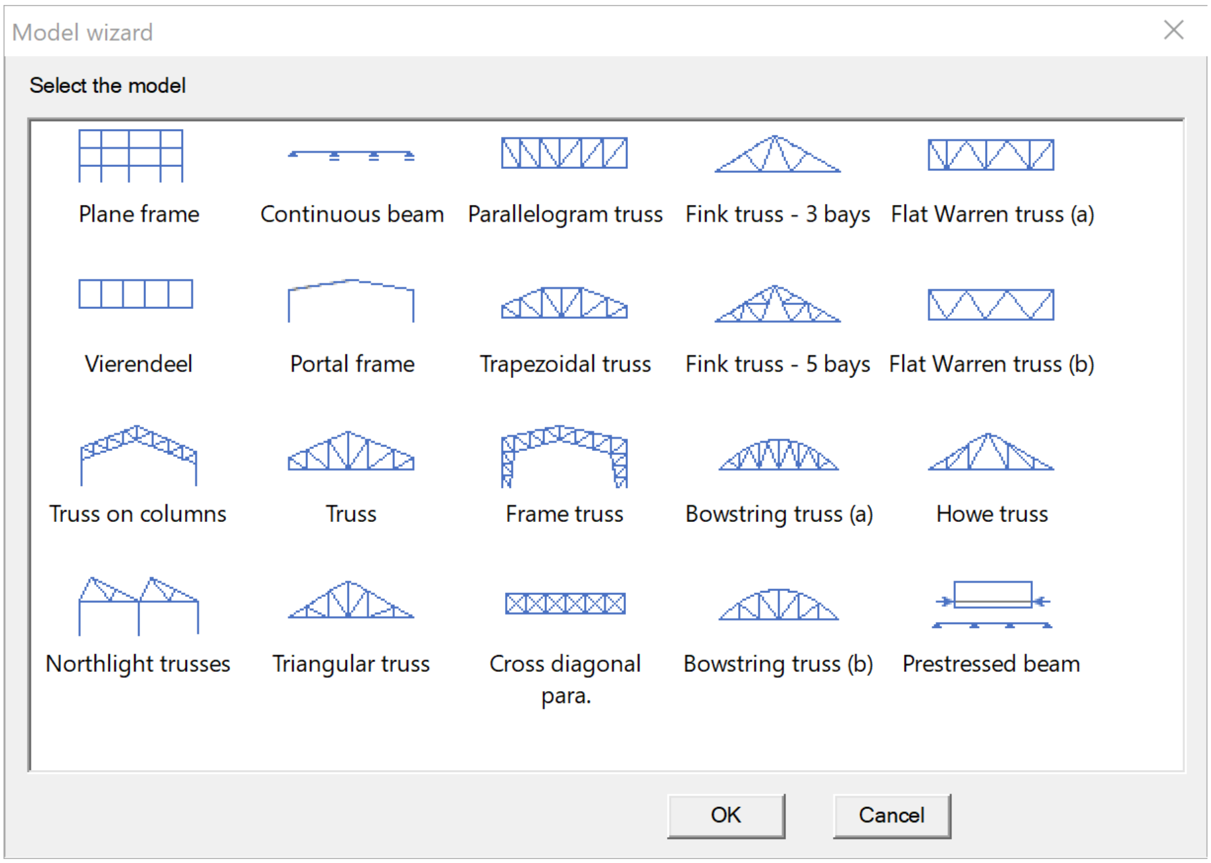
Cancel

Leave the wizard ***without*** saving the model and return to the preliminary geometry menu

OK

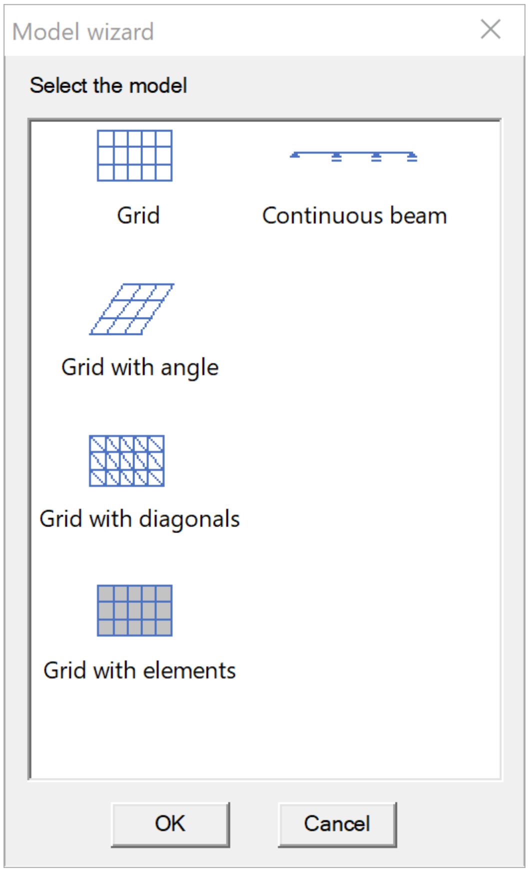
Continue to the regular geometry module and complete the definition (supports, properties, etc.)

3.1.1.3 Plane frames



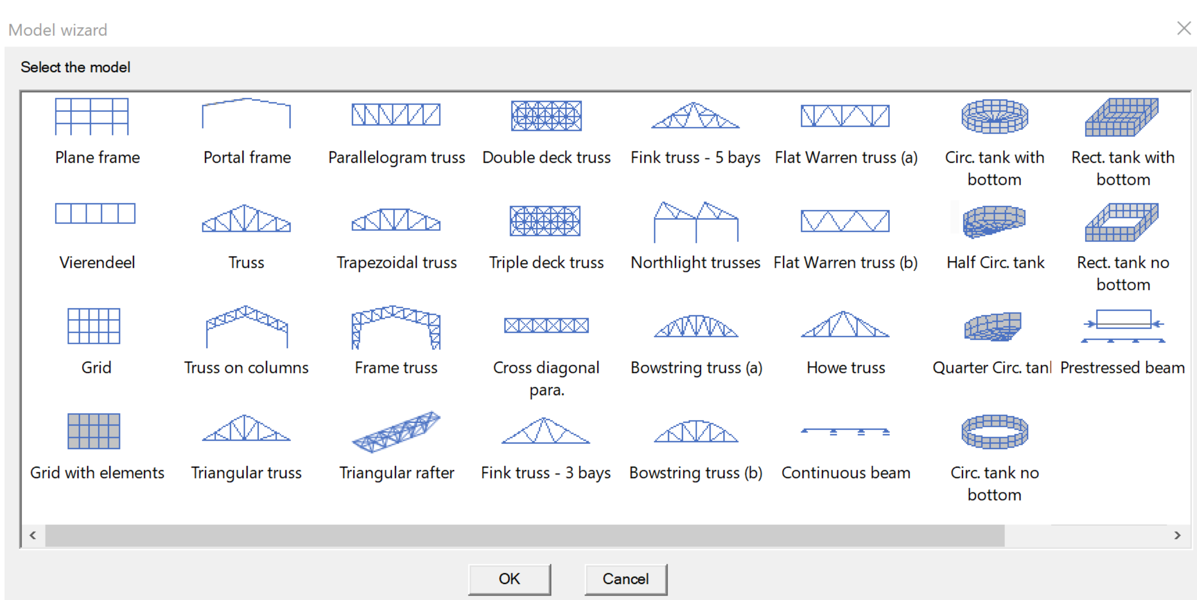
Refer to the Appendix.

3.1.1.4 Plane grids



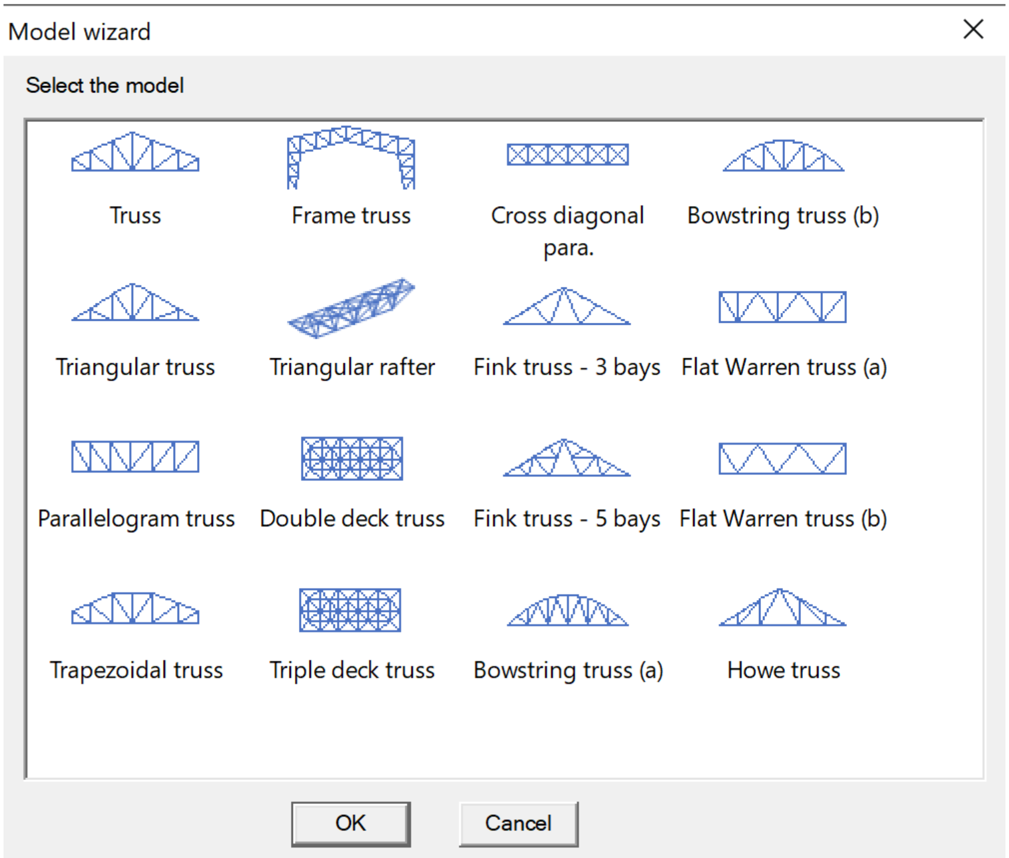
Refer to the Appendix .

3.1.1.5 Space frames



Refer to the Appendix

3.1.1.6 Trusses



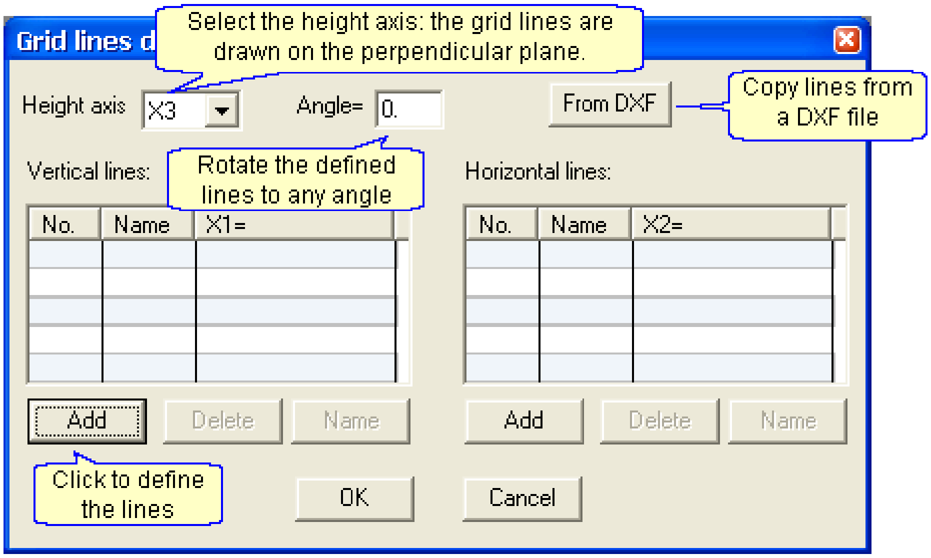
Refer to the Appendix

3.1.2 Grid lines

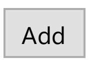
Create the orthogonal grid lines as the basis for the new model.

- the program creates a grid of nodes on the plane perpendicular to the height axis
- beams and elements are not created in this option; Columns may be defined later at the nodes using the Beam - column option and the floor beams/slabs can then be defined using the various geometry options.
- grid lines defined here may be revised in the Geometry grid line option.

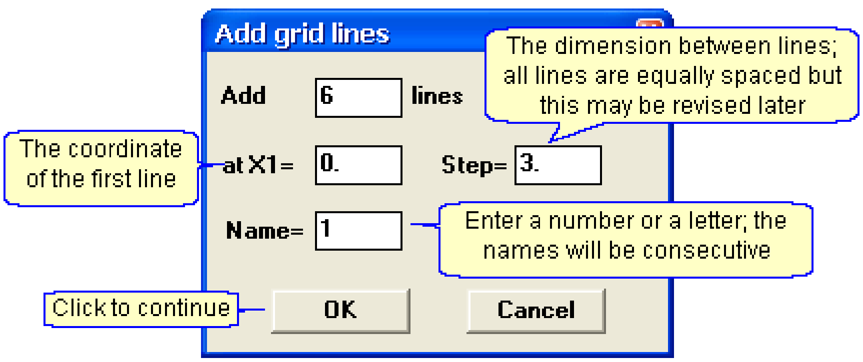
Grid lines may be added manually 167 and/or may be retrieved from a DXF file 168. Define grid lines in both directions:



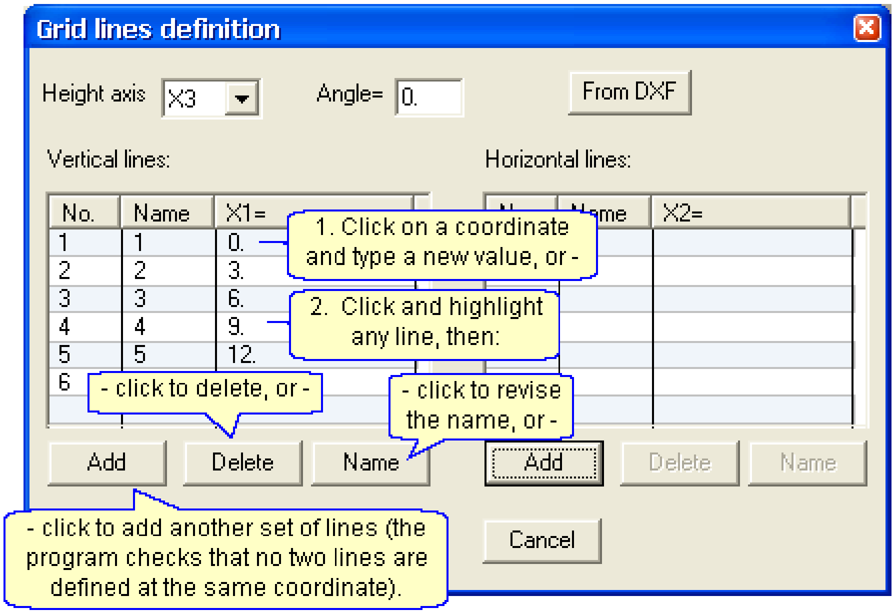
Add

Click 

under the vertical or horizontal lines table; specify the parameters for each direction, for example:



The grid lines are added to the table; the following dialog box show the lines created for the parameters above. To modify the lines select one of the following options:



Click OK when the definition is complete. The program creates the grid lines and nodes at the line intersections and enters the *STRAP* geometry main menu.

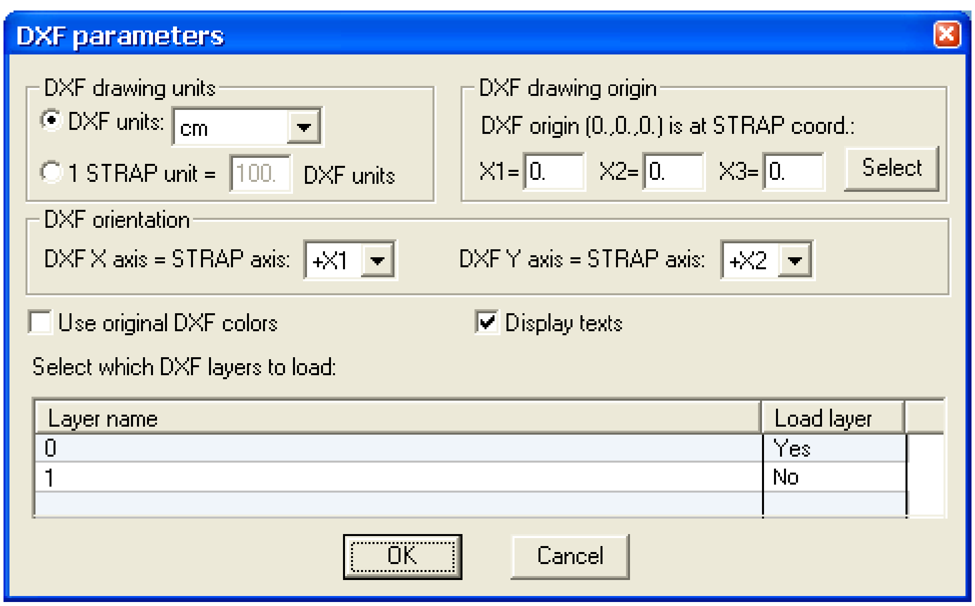
DXF file

The program searches for orthogonal sets of parallel lines and and any text found at the ends of these lines.

Note:

- ***this option erases all existing grid lines***

Click from DXF and select a DXF format file. Specify the various parameters:



DXF drawing units:

The DXF drawing will be scaled according to the *STRAP* geometry units. Two options are available:

DXF units:
Select a unit from the list box

1 STRAP = x DXF units
Select the ratio between the units (the DXF dimensions will be divided by the value entered here). For example, *STRAP* units are meter, the DXF units are feet, but you want to double the size of the drawing in the background: Enter 3.281/2 = 1.6405

Orientation:

Specify:

- the location of the DXF (0,0,0) coordinate on the *STRAP* drawing:
  Specify the global axis coordinates.
  Note: this option is relevant only if **Use the location in the DXF file** is selected in the following menu.
- the orientation of the DXF X and Y axes relative to the model axes:
  - Select *STRAP* global axes (Note that the DXF drawing may be inverted by selecting **-Xn** as the global axis), or -

Use original colors:

☑ display the DXF lines with the same colors as in the DXF drawing
☐ display all DXF lines with the color selected in Setup - Colors - General.

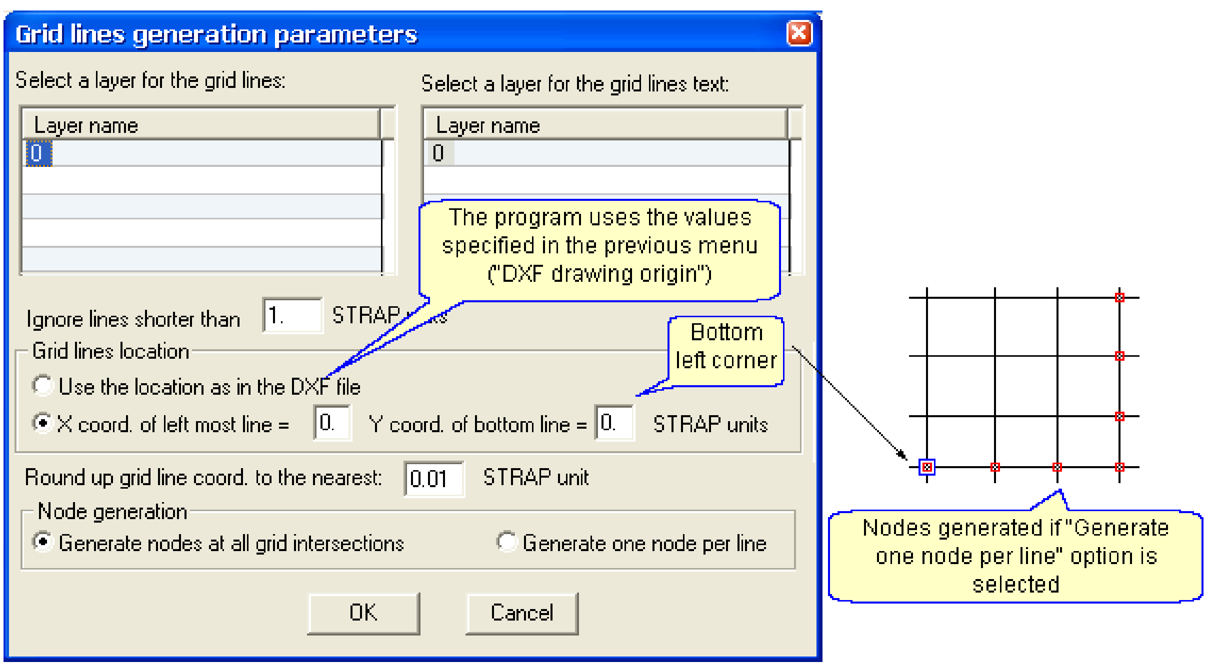
Display text:

Display the text in the DXF drawing on the model. The text will be ignored when defining nodes.

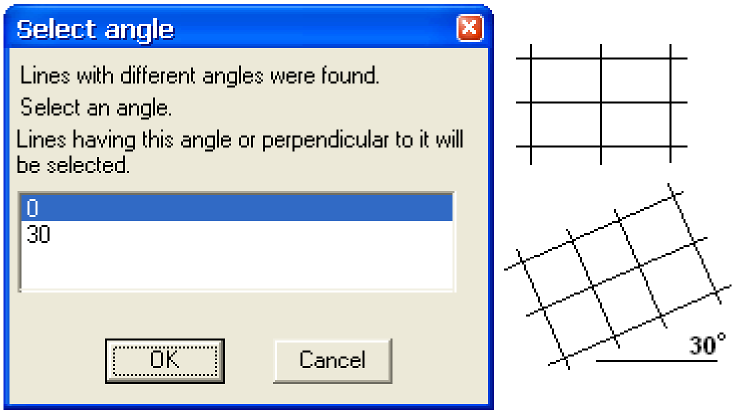
Layers:

All layers selected will be drawn in the background and the grid lines will be retrieved from one of them.

Select the layers that contain the grids and the text:



If more than one set of orthogonal lines at different angles are found, only one set of lines may be selected:



The angle value is inserted into the **Angle =** box in the initial menu

---

The program displays the grid line values in the table of the initial screen. Modify the coordinates and names or add new lines as explained in Add 167.

Click OK when the definition is complete. The program creates the grid lines and nodes at the line intersections and enters the *STRAP* geometry main menu.

3.2 Main menu

The program displays the geometry definition screen (including the current model if an existing model was selected). The geometry definition options are displayed at the bottom-right side of the screen:

Nodes	Define node 172 coordinates
Restraints	Define restrained supports and rigid links 201.
Beams	Define beam 210 elements, including location of the beam (between two nodes), properties, materials, pinned connections, rigid offsets, local axes directions
Elements	Define quadrilateral and triangular plate finite elements 275, including location of the element, properties, materials (including orthotropic).
Slabs	Define a single element to fill a slab area bounded by a contour
Springs	Define elastic supports 323.
Solids	Define general solid elements 344, including location and shape and material type of the elements. Note that the elements may have 4, 5, 6, 7 or 8 corner nodes. Note: ***the Bridge postprocessor cannot solve models that include solid elements***
Walls	Define wall element sections 351 and location
Copy	Duplicate a portion of the model 332 at another location, including nodes, elements and properties. The copied portion of the model may be rotated or a mirror image may be created.
Stages	Define construction 'stages' 394. Different properties and supports may be defined for each stage and beams/elements may be removed
Submodels	Create part of the current model in a separate working area as a "submodel 380". The submodel may then be added one or more times to the main model. For repetitive elements such as typical floors, the use of "submodels" reduces the load input time and the matrix solution time.

The program also displays a list of submodels above these icons (if submodels have been defined):

Main model — Open the list box and click on the submodel (or Main Model) to display on the screen.

From the menu bar:

File Edit Zoom Display Draw Remove Output Help

Note:
- refer to Command Mode for details on defining geometry by typing in commands

3.3 Nodes

Define the location of each node in space by specifying its coordinates in the global coordinate system.

The following options are available:

Node	Define one node 172 only.
Line - eq.	Define a series of nodes 178 all lying on a straight line, with equal spacing between them. When defining the nodes using a cylindrical coordinate system, the nodes are equally spaced along an arc.
Line - uneq.	Define a series of nodes all lying on a straight line, with varying spacing 179 between them. When defining the nodes using a cylindrical coordinate system, the nodes are located along an arc.
Grid	Define a parallelogram grid of nodes 180. Define a 'base' line and a 'height' line of nodes (similar to the previous two options) by specifying the location of three corner nodes. When defining nodes using a cylindrical coordinate system, this command creates a series of parallel arcs or concentric arcs.
Equations	Define a series of nodes using an equation 183, e.g. parabola, sphere, etc. The equation may be one supplied with the program or may be user defined. The option may also generate beams and/or elements.
Move	Move 186 an existing node to a new location.
Delete	Delete 193 an existing node.
Renumber	Assign a new node number 193 to an existing node.
System	Select a coordinate system 196 for node definition: • a Cartesian plane, either one of the global planes or any arbitrary plane in space. • a Cylindrical coordinate system.
Unify	Search for locations where more than one node has been defined and delete all nodes except one, thereby 'unifying' the model 198 at those locations.

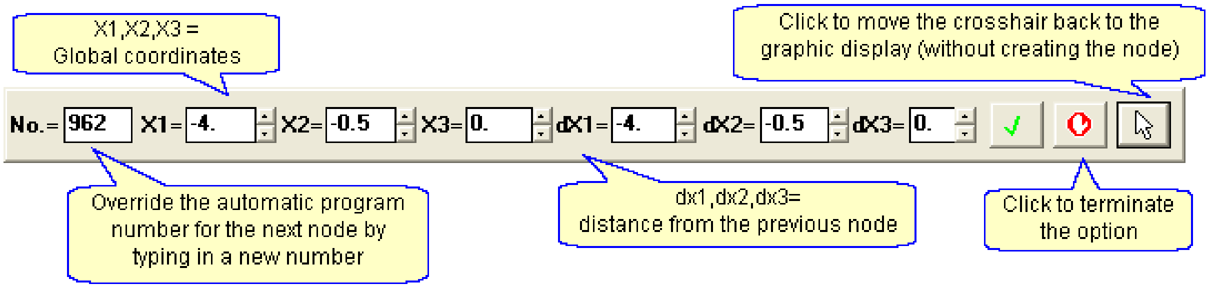
3.3.1 Single Node

Create a single node at any location in the model.

Plane models:

There are two methods:

- Move the mouse so that the is located at the correct coordinates as displayed in the Dialog Box (adjust the Step if necessary); click the mouse.
- move the into the **X1=** text box in the Dialog Box. Either type in the correct value or click the buttons until the correct value is displayed. Repeat for **X2 =** and press to complete the definition.



Space models

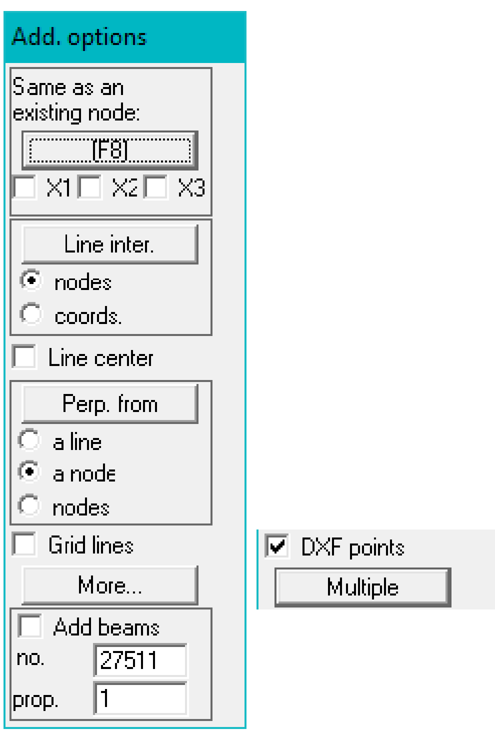
The cursor moves along the current 'working plane' (normally one of the Global planes), even if the the model is rotated. The two coordinates on the plane may be defined as explained for *Plane models*, but the third coordinate must be defined in the Dialog box:

- move the into the X3 text box in the Dialog Box. Either type in the correct value or click the button until the correct value is displayed.

Note that if you click and hold the left mouse button until the is in the Dialog box, then the X1 and X2 coordinates in the box will be those at the location when the mouse was clicked.

Refer also to:
[Additional options](173)
[Cylindrical coordinate system](196)

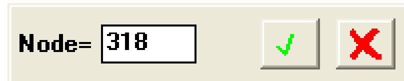
3.3.1.1 Additional options



Same as existing node

- Specify that one or more of the global coordinates is identical to the same coordinates of an existing node:

- select the identical global coordinates by turning on their checkboxes
- click the [F8] button
- select an existing node; Move the to an existing node and click the mouse or type in the node number in the dialog box and press [Enter].

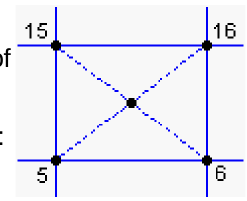


Intersection

Intersection by nodes

Using the , define two lines by pointing to their start and end nodes; the new node is created at the point of intersection of the two lines.

For example, create the node at the centre of the bay:
first line: 5-16 second line: 6-15



Intersection by coordinates

Using the , define two lines by pointing to their start and end coordinates; the new node will be created at the point of intersection of the two lines.

Line center

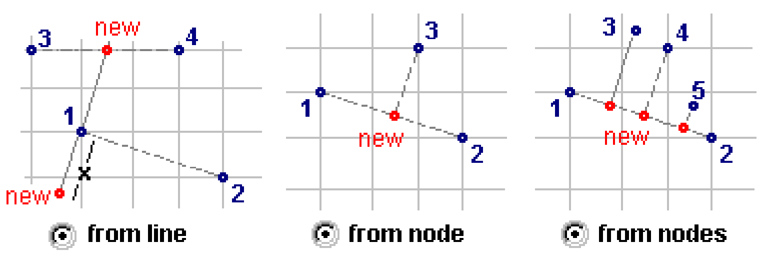
Create a new node at the centre point of the imaginary line connecting two existing nodes.

- select the start and end nodes of the line

Perpendicular from

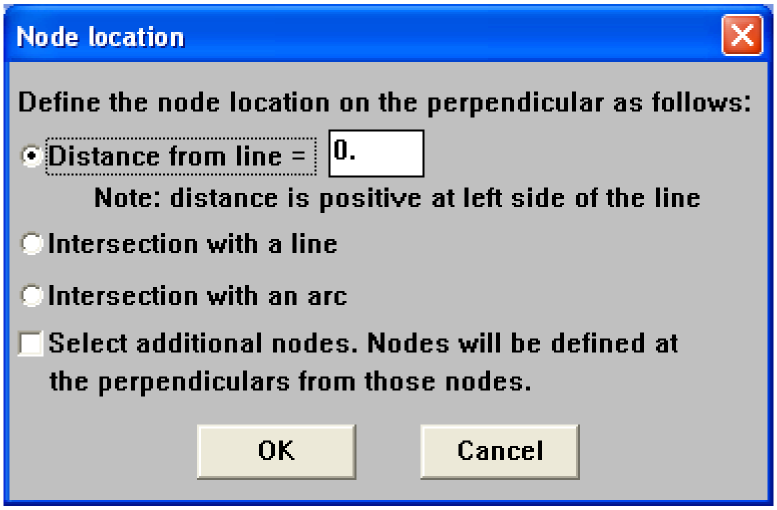
A node (single node, start/end node of a line, etc) may be defined at:

- offset from a node and perpendicular to the line starting at that node; the node is at a specified distance or the intersection of the perpendicular with another line or arc.
- the intersection of a line (defined by two nodes) and the perpendicular to the line from a node(s)



from line

select nodes **1** and **2**; the new node is on the line perpendicular to line **1-2**, drawn from node **1**, according to one of the following options:



- at a specified distance from node **1**. Note that a positive value is to the left when standing at **1** and looking towards **2** (the example above shows a negative value)
- at the intersection with another line (defined by two additional nodes - **3** and **4** in the example above)
- at the intersection with an arc (defined by three additional nodes)

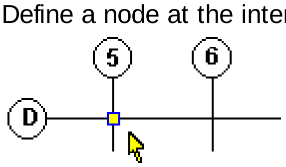
Select additional nodes to create nodes at the perpendiculars from multiple nodes to the same line.

from node
select nodes **1**, **2** and **3**; the new node is created at the perpendicular from node **3** to the line joining nodes **1** and **2**.

from nodes
select nodes **1** and **2,** then select a series of nodes using the standard node selection option; the new nodes are created at the perpendicular from these nodes to the line joining nodes **1** and **2**.

Grid lines

Define a node at the intersection point of any two grid lines. For example:



Note that this option is not available when a background DXF drawing is displayed.

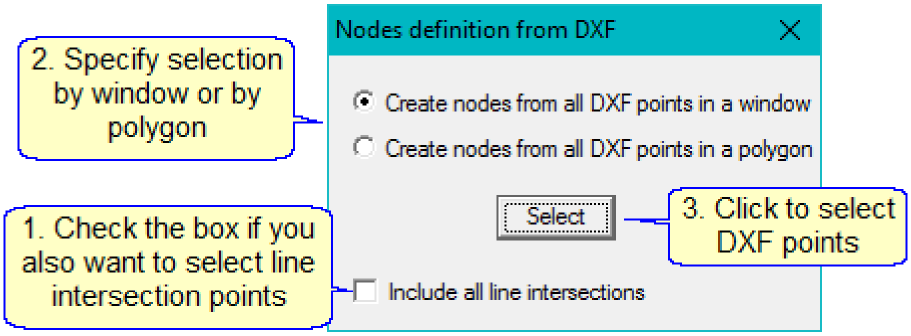
DXF points

This option is displayed only if a DXF drawing has been added to the display background.

only the end points of DXF lines may be selected as node locations; the selection is for individual nodes.

the DXF end points are not highlighted.

To select multiple end points and/or DXF line intersection points - click Multiple :



Note:

- The program sets all Z coordinates = 0 when DXF points are used to define nodes in a Plane Grid model.

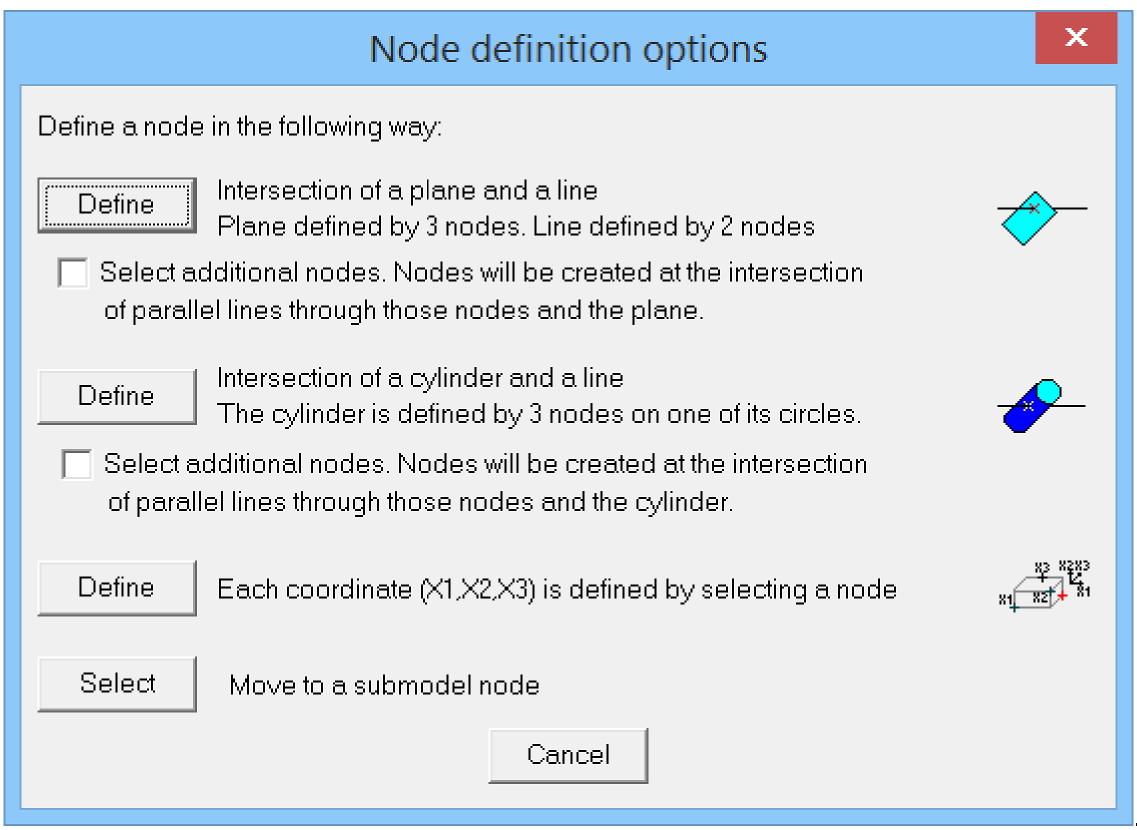
Add beams

The program automatically defines a beam between any two consecutive nodes that are defined. The beam number and the beam property number may be specified here; when defining a line/grid of nodes the program generates a line/grid of beams.

Note:

- for a line/grid - the beam number specified is assigned to the first beam that is generated; the remaining beams are numbered consecutively (if an existing beam is found he program arbitrarily selects a different number for that beam and all following ones).

3.3.1.2 'More' Additional options

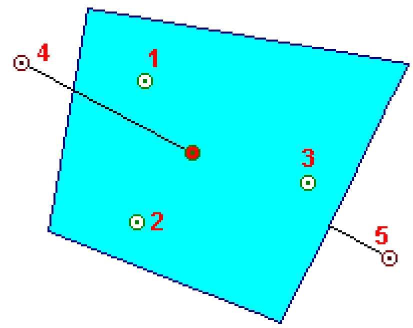


.

Intersection of plane and line

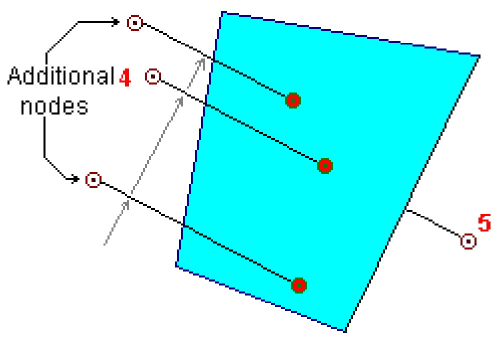
- Select any three existing nodes that define the plane
- Select any two existing nodes that define the line

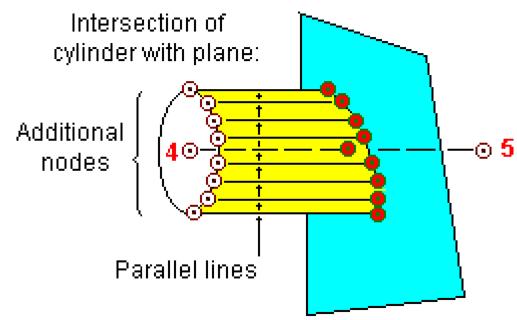
The five nodes cannot lie on the same plane.



Select additional nodes

Select a series of nodes; the program creates nodes at the intersection of the plane and lines that are parallel to the original line but pass through the additional nodes. For example:

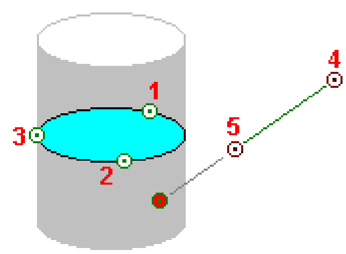




Intersection of cylinder and line

The new node is created at the intersection of a cylinder and a line:

- Select any three existing nodes that define the a circle on the cylinder (the program assumes that the cylinder extends infinitely in both directions)
- Select any two existing nodes that define the line.



Note:

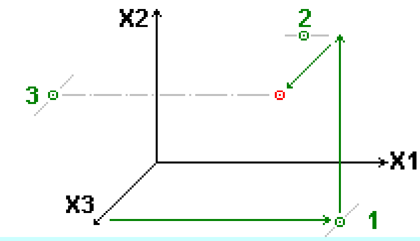
- There are two intersection points; the new node is created at the point closest to the line end node.

Select additional nodes

Select a series of nodes; the program creates nodes at the intersection of the cylinder and lines that are parallel to the original line but pass through the additional nodes.

Each coordinate defined by node

Select three nodes:
- X1 of the first node = X1 of the new node
- X2 of the second node = X2 of the new node
- X3 of the third node = X3 of the new node



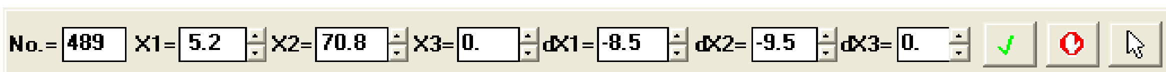
A submodel node

- the program displays a list of submodel instances; select one.
- the program displays the submodel; select a node
- the program restores the previous display and places the crosshair at the selected submodel node location (even if the submodel has been removed from the display). The crosshair can now be moved an additional dX1/dX2/DX3 from the selected node to define multiple nodes.

3.3.2 Line- equal spacing

Use this option to define a line of nodes where the spacing between them is equal.

To define the line:
- define the start node of the line as explained in Single node 172
- define the end node location:
  The dialog box at the bottom of the screen is:



where:

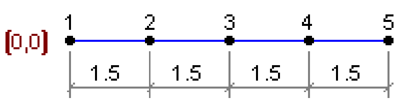
X1,X2,X3 = location on the screen (global coordinates)

dX1, dX2, dX3 = distance **from the start node**.

Move the until the correct coordinates are displayed or type in the correct values in the text boxes.

- Specify the number of segments. For 'n' nodes (including the end nodes), there are 'n-1' segments.

Example:

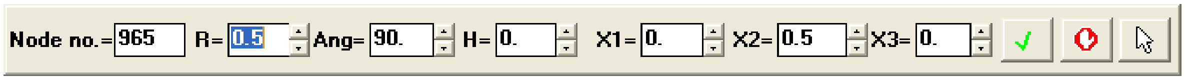


- move to: X1 = 0.0 X2 = 0.0 ; click the mouse
- move to: X1 = 6.0 X2 = 0.0 ; click the mouse
- specify four segments

Refer also to:
Additional options 173
Define an arc 178 (Cylindrical coordinate system)

3.3.2.1 Arc - Equal Spacing

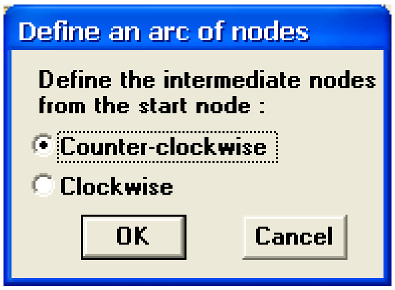
Define nodes equally spaced along an arc.
- Set up the cylindrical system as explained in System - Cylindrical 196.
- Select the start and end nodes as explained in Line - Equal 178, except that the Dialog box is:



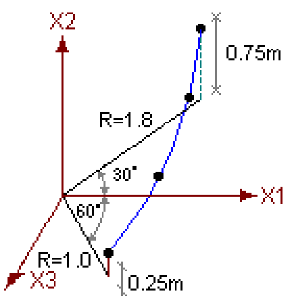
Note:

- movement is relative to the cylindrical system. Remember that moving the mouse vertically revises the radius, while moving the mouse horizontally revises the angle.
- you may type in either the cylindrical coordinates or the global coordinates (X1,X2,X3) in the dialog box.

- Enter the number of segments.
- Define the arc direction about the non-cylindrical axis:



Example:



- define start node at: R = 1.0 Ang = -60.0 H = 0.25
- define end node at: R = 1.8 Ang = 35.0 H = 0.75
- specify three segments.
- select **Counter-clockwise**

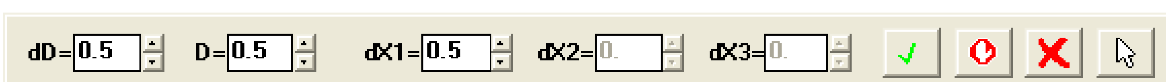
3.3.3 Line - unequal spacing

Use this option to define a line of nodes where the spacing between them is ***not*** equal.

- Define the start and end nodes of the line as explained in Line - Equal 178.
- Point to the intermediate nodes along the line, as follows:

  The moves along the line joining the start and end nodes.

  The dialog box at the bottom of the screen is:



where:

dD = distance ***from the last node defined on the line*** as measured along the line.

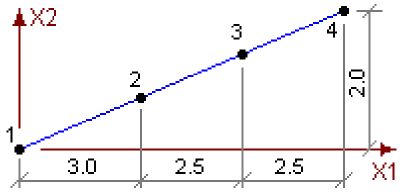
dX1,dX2,dX3 = distance ***from the last node defined on the line*** (global coordinates).

D = distance ***from start node*** as measured along the line.

Note that dX1 or dX2 (the closest to the line) and not dD is updated at the Step rate, a feature that is very convenient when only the projected spacings of a diagonal line on a global axis are known.

In many cases it may be more convenient to type the spacing directly in the text boxes rather than by continuously adjusting the Step for each node.

Example:



- move the to the location of Node 1 and click the mouse.
- move the to the location of node 4 - dX1 = 8.0, dX2 = 2.0 and click the mouse.
- move the along the line until dX1 = 3.0 is displayed; click the mouse; node 2 is created.
- move the along the line until dX1 = 2.5 is displayed; click the mouse; node 3 is created.
- move the outside the line and click the mouse.

Refer also to:
Additional options 173
Define an arc 180 (Cylindrical coordinate system)

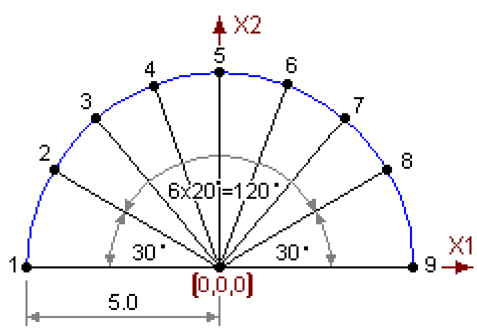
3.3.3.1 Arc - unequal spacing

- Set up the cylindrical system as explained in System - Cylindrical 196.
- Select the start and end nodes as explained in Line - General 179.
- Define the intermediate nodes as explained in Line -General 179, except that the dialog box is:

where:

R, Ang, H = location relative the the cylindrical system origin.
dR, dAng, dH = location ***from the last node defined on the line***.

Example: define the arc - nodes 1 to 9:



- define the start node:
  move the to: **R** = 5.0 **Ang** = 180.0 ; click the mouse
- define the end node:
  move the to: **R** = 5.0 **Ang** = 0.0 ; click the mouse
- select **Clockwise**

The moves only along the arc:

- define the intermediate nodes:
  Node 2 :move to: **R** = 5.0 **Ang** = 150.0 ; click the mouse.
  (or type in the values in the text boxes).
- Similarly define nodes 3 to 8.

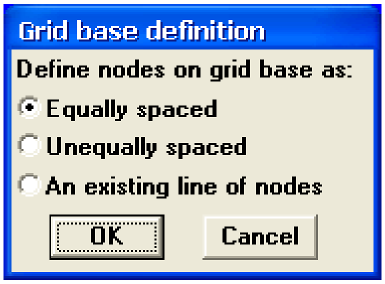
3.3.4 Grid of nodes

A parallelogram grid of nodes is defined by specifying:

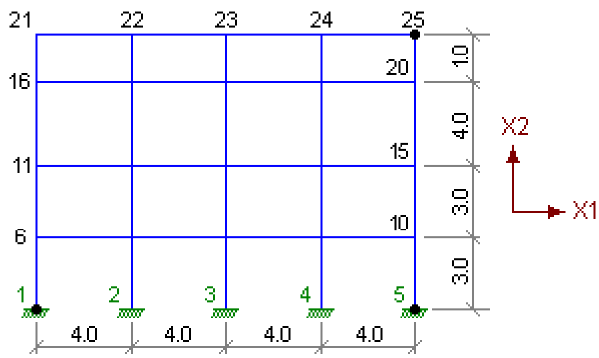
- the nodes on the "base" line of the parallelogram
- the nodes on the "height" line of the parallelogram

Since the end node of the base line is the start node of the height line, three nodes define the general shape of the grid. Once the distribution of the nodes along the two lines is defined the program automatically calculates the location of all of the remaining nodes and creates them.

The distribution of the nodes along the lines is specified with options similar to Line - equal 178 and Line - Unequal 179, or may be specified as a line of existing nodes.



Example:



base line:

- specify **Equally spaced**
- move to: **X1** = 0.0 ;**X2** = 0.0 ;click the mouse
- move to: **X1** = 16.0;**X2** = 0.0 ;click the mouse
- specify four segments

height line:

- specify **Unequally spaced**
- move to: X1 = 16.0 X2 = 12.0 ; click the mouse
- move to: dX2 = 3.0 ; click the mouse
- move to: dX2 = 3.0 ; click the mouse
- move to: dX2 = 4.0 ; click the mouse

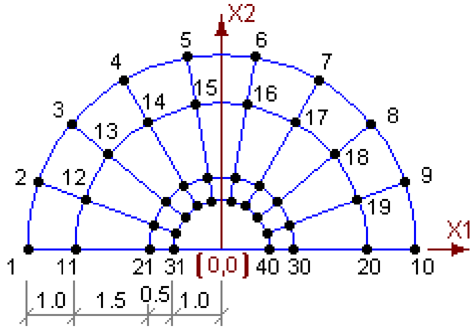
Refer also to:

Additional options 173
Parallel arcs - cylindrical coordinate system 181

3.3.4.1 Parallel arcs (grid)

The **Grid** option generates a series of parallel arcs; the arcs may all lie in the same plane or may be in parallel planes (Space models).

Example (a):



base line:

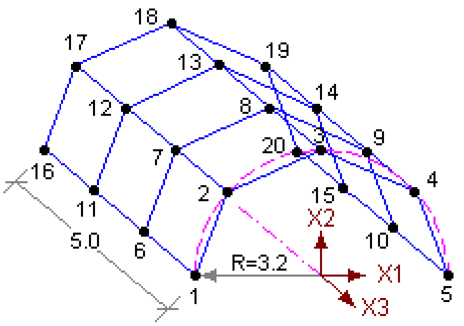
- specify ◉ **Equally spaced**
- move ↖ to: R = 4.0 Ang = 180.0; click the mouse
- move ↖ to: R = 4.0 Ang = 0.0; click the mouse
- specify nine segments
- select **Clockwise**

height line:

- specify ◉ **Unequally spaced**
- move ↖ to: R = 1.0 Ang = 0.0; click the mouse
- move ↖ along the line to:
- R = 3.0 Ang = 0.
- R = 1.5 Ang = 0 .

  etc.

Example (b):



base line:

- specify ◉ **Equally spaced**
- move ↖ to: R = 3.2 Ang = 180.0 H = 5.0 ; click the mouse
- move ↖ to: R = 3.2 Ang = 0.0 H = 5.0 ; click the mouse
- specify four segments
- select **Clockwise**

height line:

- specify ◉ **Equally spaced**
- move ↖ to: R = 3.2 Ang = 0.0 H = 0.0 ; click the mouse
- specify three segments

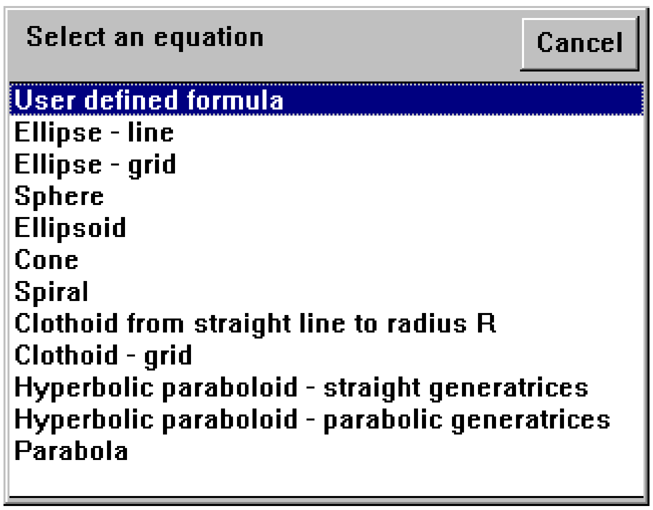
3.3.5 Equations

This option allows nodes to be defined along lines specified by an equation. The equations may represent plane shapes, e.g. an ellipse, or complex three dimensional shapes, such as spheres, cones, etc. The user may also define beams or elements connecting the nodes.

The equations are stored in a general form with constants and variables; after selecting the equation the user must enter values for the constants and variables in order to specify the actual size of the shape.

In addition the user must specify:
- the location of the reference point of the shape
- the number of node increments along the line(s)
- whether beams and elements should also be automatically generated.

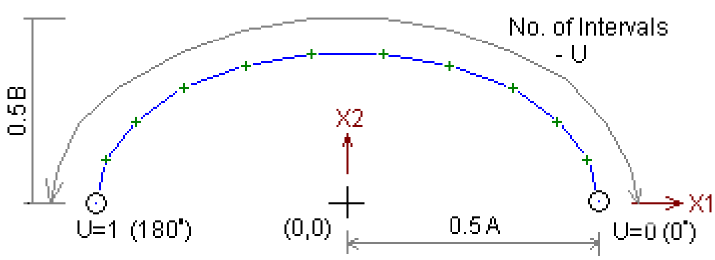
The following menu is displayed:



- select any of the program formula for an explanation of the parameters and variables. Note that only Ellipse - line184 contains an explanation on the program menus.
- select "User defined formula" for an explanation on how to define a new formula.

The equations are stored in a general form with constants and variables; after selecting the equation the user must enter values for the constants and variables in order to specify the actual size of the shape.
- Constants :   A, B, C, D, E, R
  normally represent dimensions of the shape.
- Variables   :   U, V
  represent variable defined for a range of values. The nodes are generated over a range specified for U and V.

For example, a half-ellipse is defined by the equation $x^2/A^2 + y^2/B^2 = 1$;

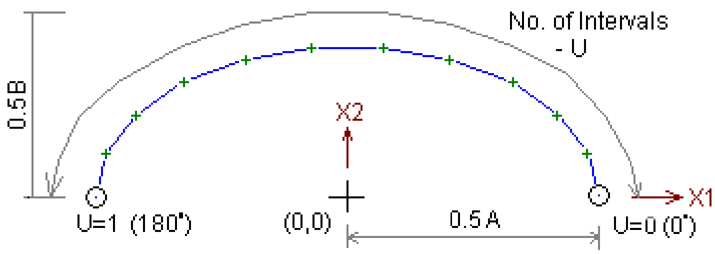


The required values are:

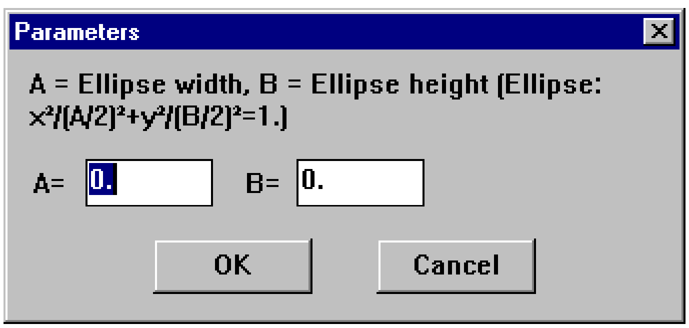
A - 0.5 * ellipse width
B - 0.5 * ellipse height
U - a variable in the range 0.0 to 1.0 specifying the angle of rotation between the horizontal axis and any point on the ellipse, where 0. represents 0° and 1. represents 180°. To define an ellipse from 45° to 135°, specify U from 0.25 (45/180) to 0.75 (135/180). To define a full ellipse, specify U from 0.0 to 2.0.

3.3.5.1 Ellipse - line

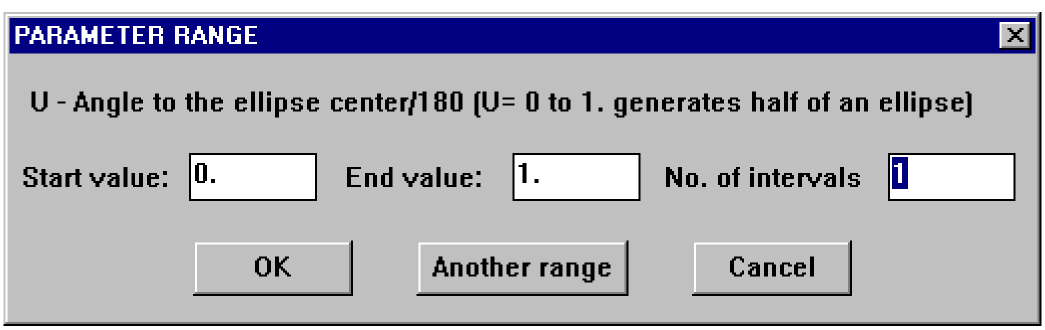
The general shape for an ellipse is:



The program prompts for the dimension constants A and B:

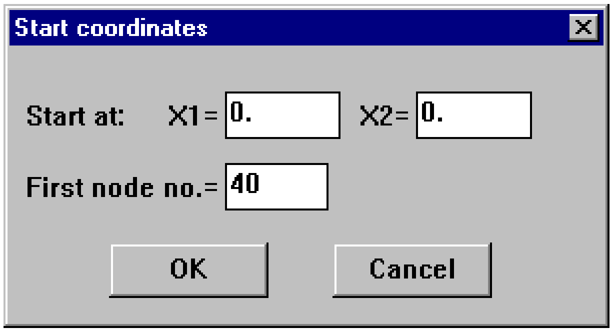


and the range constant U:

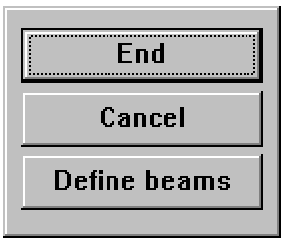


U - referring to the figure above, U from 0. to 1. represents half of a full ellipse (180°). To generate an ellipse from 30°, define U with "**Start value**" = 0.1667 (30/180) and "**End value**" = 1.25 (225/180).
Intervals - to specify the number of node spacings in the range of U. For example, 11 intervals were specified in the example above.
Another - to continue from the "End" point of the previous range with a different interval size. For more complex equations, this option can be used to vary the spacing of an element mesh

Specify the number of the first node and define the coordinate of the reference point (the point with coordinates <0,0,0>). Note that an ellipse is always generated on the X1-X2 axis.



Finally, you may instruct the program to link the nodes with beams (or elements, for equations that generate appropriate models).



Select "**Cancel**" to abort the equation option.

3.3.5.2 User defined equation

Define your own formula and add it to the equation list.

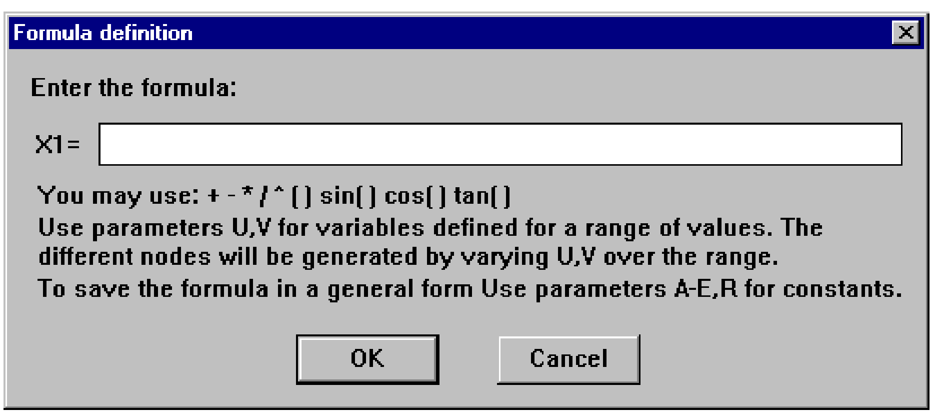
All equations must be defined in terms of constants A,B, .. , R and variables U and V. In all program formulas, the range U = 0 to 1 and V = 0 to 1 represents the complete shape.

For example, define the equations for a full circle lying on the X1, X2 plane. The equations for any point on the perimeter of a circle are:

$X1=R\cos\theta$
$X2=R\sin\theta$

Assume that U represents the angle θ measured counter-clockwise from the X1 axis and the angle varies in the range from 0 to 2π radians. (All angles are measured in radians). Therefore, U = 0. represents 0 radians and U = 1.0 represents 2π radians = 360°.

The program prompts for the equations defining X1, X2 and X3:

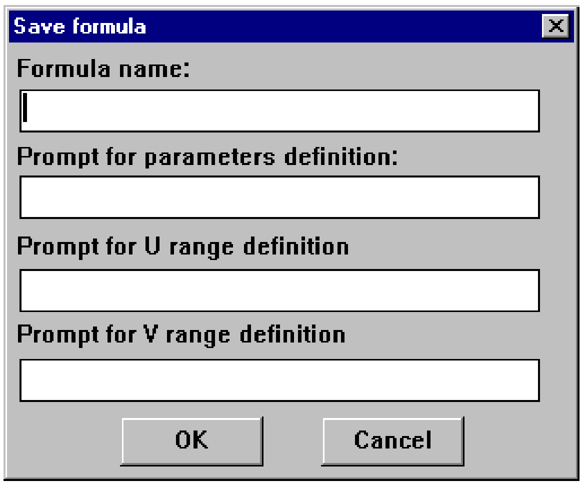


type: **R*cos(6.283185*U)**

Similarly, for X2 type: **R*sin(6.283185*U)** and for X3: Press [Enter].

The formula may be added to the equation list (saved in the ASCII file **FORM.DAT**):

Define a title that will be displayed in the list and the prompts that are displayed when you define the constants for this equation:



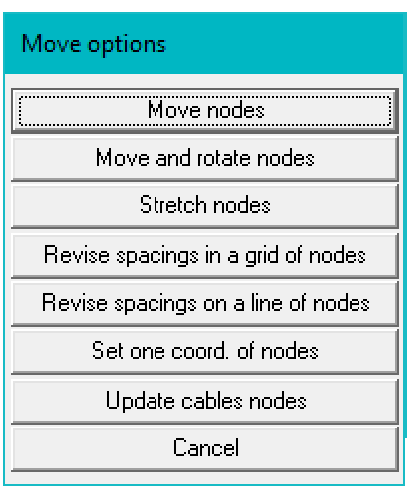
For our example

- define the prompt as **Enter circle radius**. (If you press [Enter] without defining a prompt, "**Enter value**" will be displayed when defining the nodes).
- define prompts for the definition of **U**. For example, "**U = 0. to 1. generates a full circle**".

The equation definition is now complete. The program now begins prompting for values and generate the nodes as explained in Ellipse 184.

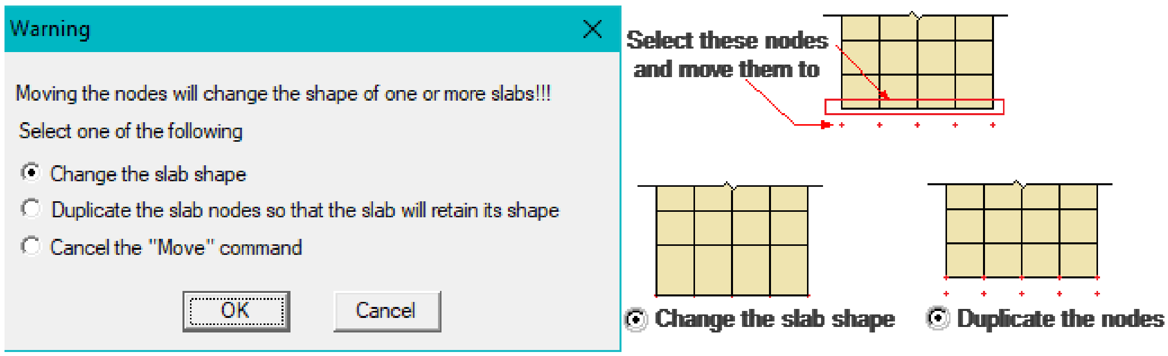
3.3.6 Move Nodes

Use this option to move nodes from their current location to a new location.

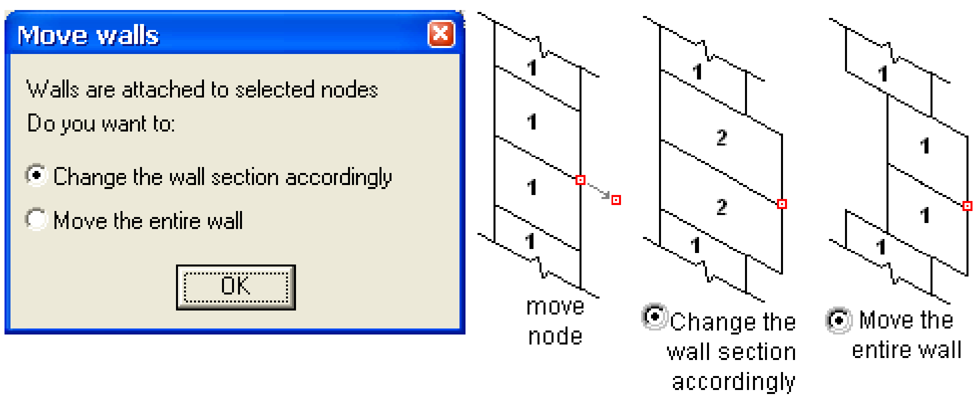


Note:

- Submodels: if ***all*** the main model nodes at the connection points of a submodel instance are moved ***equally***, then the program moves the instance accordingly; otherwise the program does not move the submodel and creates rigid links between the connection point and the new location of the main model node. Refer to Submodel - add to main.
- If a Slab contour node is moved, the Slab shape can be either maintained or revised accordingly:



- If a wall is connected to the selected node, the program asks how to revise the wall section:



The program creates new nodes where necessary.

3.3.6.1 Individual nodes

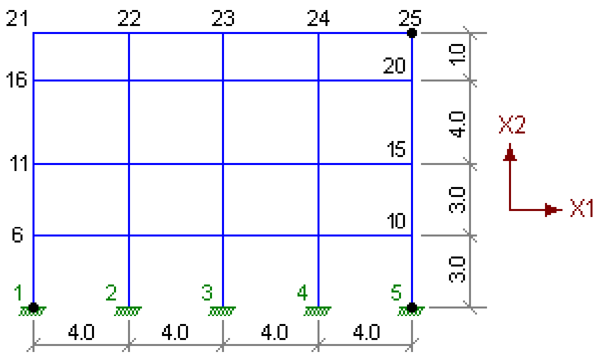
Select one or more nodes using the standard node selection option.

- move one node: move the to the new location and click the mouse.
- move several nodes: select a reference node and its new location; all selected nodes are moved by the ***same*** dimension.

If the current coordinate system is cylindrical, the radius, angle or height values may be revised for the selected nodes (relative to the origin of the cylindrical system).

Example:

For the grid below, increase the width of the first bay (between nodes 1 and 2) to 5.0 m without changing the dimensions of the other bays.

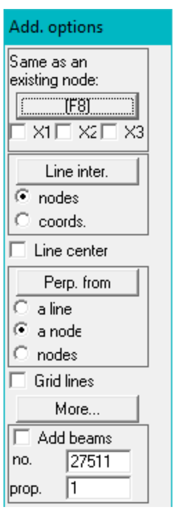


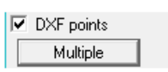
- Select all nodes except those on line 1-21 by defining a rectangular window around them.
- Select node 2 as the reference node.
- Move the so that X1 = 5.0 and dX1 = 1.0.
- Click the mouse; the model is redrawn.

Note that if the window is defined around the nodes on line 2-22 only, the distance from line 1-21 to line 2-22 is revised to 5.0 but the distance from 2-22 to 3-23 is reduced to 3.0.

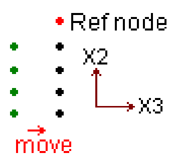
Additional options:

The following options are available when specifying the new location of the ***reference node***:





- Same as existing node: select only specified coordinates (X1, X2 and/or X3) of the reference node. For example:



Select the **X3** coordinate only of the reference node in the side menu that is displayed when the reference node is selected

- define the reference location at the intersection of lines joining existing nodes or coordinates
- select the end points of lines in a DXF background drawing
- etc.

Refer to Nodes - additional options 173

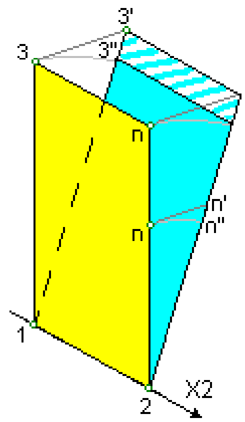
3.3.6.2 Move & rotate

Use this option to both move and rotate nodes selected using the standard node selection option.

The rotation and translation is defined by specifying the new location of reference nodes; the new location of each node can be either at any existing node or at a coordinate. The nodes form a plane and the translation and the rotation of this plane is applied to all of the selected nodes.

There are four options available: 2 options maintain the shape of the selected geometry and the other 2 stretch/squeeze the block proportionally according to the new distances between the reference nodes:

The option is best illustrated by examples:



Example 1:

Move and rotate all of the nodes on plane 1-2-3 to plane 1-2-3'; "*n*" is any point on the plane and line 1-2 is parallel to the global X2 axis.

- Rotate the plane to 1-2-3" without changing its dimensions:
  - Select **Select 2 nodes and a global axis = X2**
  - Select node 1 and its new location at node 1 (same location)
  - Select node 3 and its new location at node 3'

  The plane is rotated about X2 to plane 1-3' but the plane is not stretched or squeezed, i.e. node 3 moves to 3" and not to 3' and any node n on the plane moves to n" and not n'
- Rotate the plane to 1-2-3' , i.e. stretch the plane:
  - Select **Select 3 nodes**
  - Select node 1 and its new location at node 1 (same location)
  - Select node 2 and its new location at node 2 (same location)
  - Select node 3 and its new location at node 3'

  The plane is rotated about X2 to plane 1-3' and the plane is stretched; node 3 moves to 3' and not to 3" and any node n on the plane moves to n' and not n"

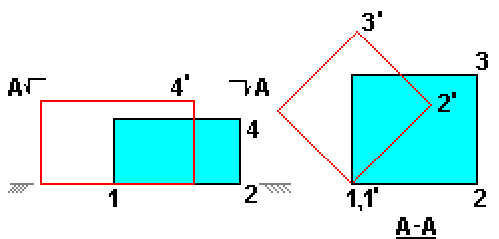
Example 2:

Rotate the base of the space frame 1-2-3 to 1'-2'-3' and increase the height from 4 to 4'

- Select **Select 4 nodes**
- Select node 1 and its new location at node 1' (same location)
- Select node 2 and its new location at node 2'
- Select node 3 and its new location at node 3'

  (the program stretches/squeezes the base if the lengths 1'-2', 2'-3' are not equal to 1-2, 2-3)
- Select node 4 and its new location at node 4'

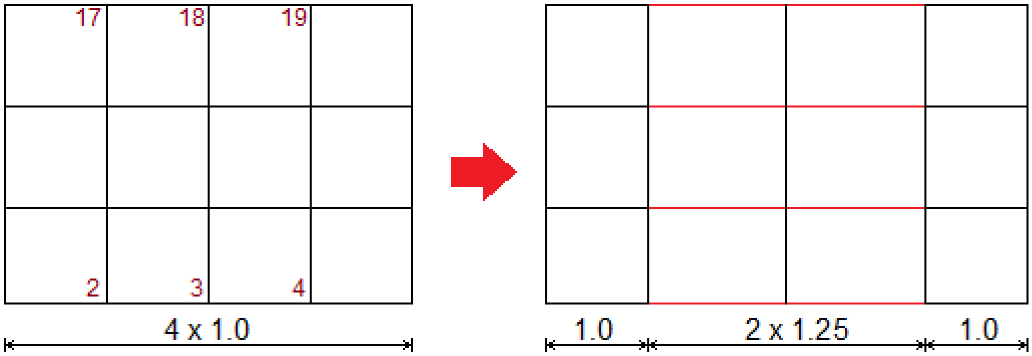
  Note that all intermediate nodes on the height axis of the frame are stretched/squeezed proportionally.



3.3.6.3 Stretch

This option stretches/shrinks the distance between selected nodes while adjusting the connected parts of the structure accordingly. For example, increase the distance between lines 2-17 and 3-18 and lines

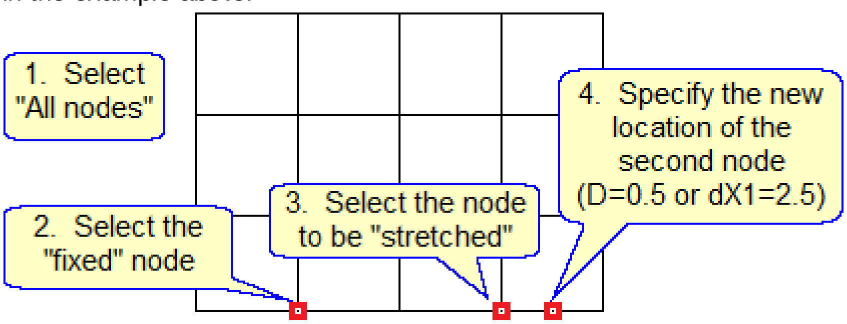
3-18 and 4-19 from 1.0 to 1.25:



The following procedure is followed:

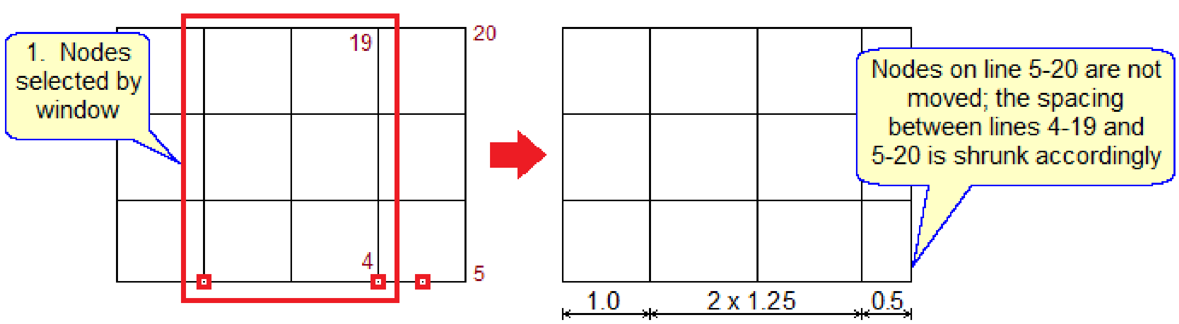
- select the nodes to be stretched
- select a "fixed node" at one side of the stretched segment
- select the node to be stretched at the other side
- specify the distance to move the selected nodes

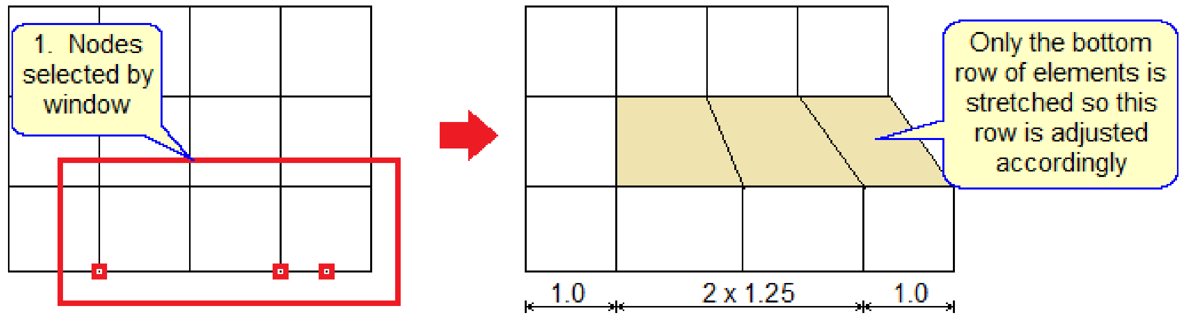
In the example above:



Note:

- the distances are stretched proportionally to their original spacing. For example if the initial distances between 2-3 and 3-4 are 1.5 and 1.0 respectively, the new spacings will be 1.8 and 1.2. (1.5 +1.5/2.5* 0.5 = 1.8)
- if only nodes in part of the model are selected, nodes outside the selected area will ***not*** be stretched. For example:

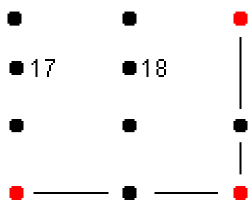




3.3.6.4 Grid of nodes

Define the grid by pointing to its three corner nodes; the program then requests new values for the base line and the height line spacings, as outlined in Line of nodes 191.

Note that the program ignores nodes not lying on the lines drawn through and parallel to the nodes on the base and height lines of the grid. For example:



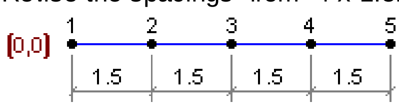
Nodes 17 and 18 are not moved.

3.3.6.5 Line of nodes

This option allows you to revise all of the spacings on a line of nodes in one step.

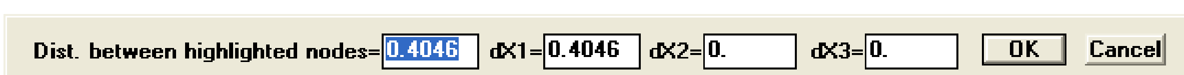
Example:

Revise the spacings from 4 x 1.5m. to 1.6, 1.5, 1.2 and 1.7.

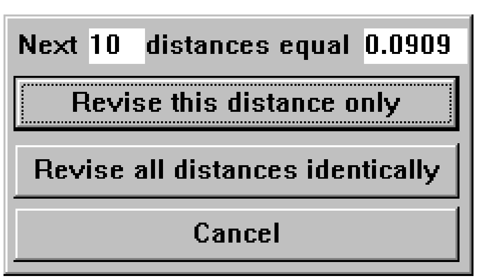


Select nodes 1 and 5 defining the the start and end of the line.

The program highlights the distance between nodes 1 and 2 and displays the following dialog box:



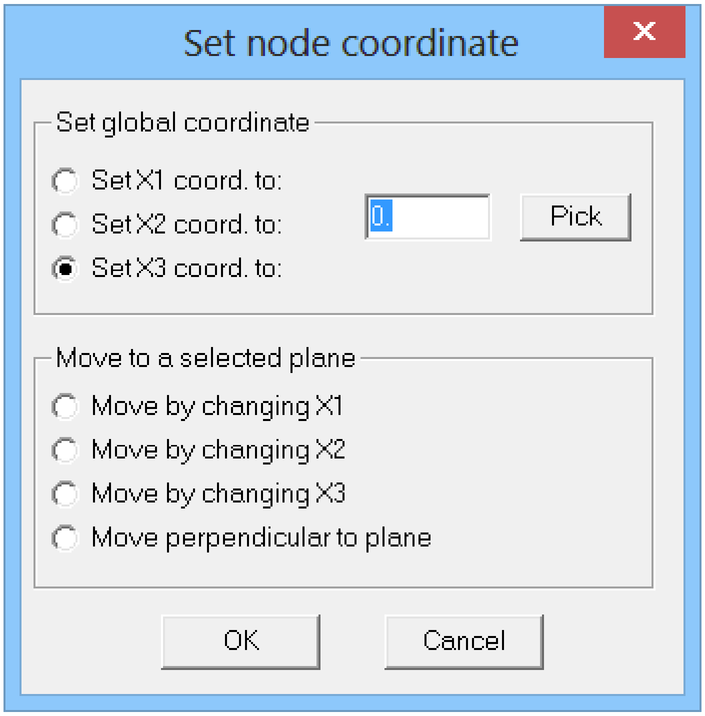
The program identifies lines where spacings are identical and may revise all to a new equal spacing:



For the current example, select the first option and revise each of the following spacings individually.

3.3.6.6 Move one coordinate

Revise one coordinate only of selected nodes or move selected nodes to a plane:



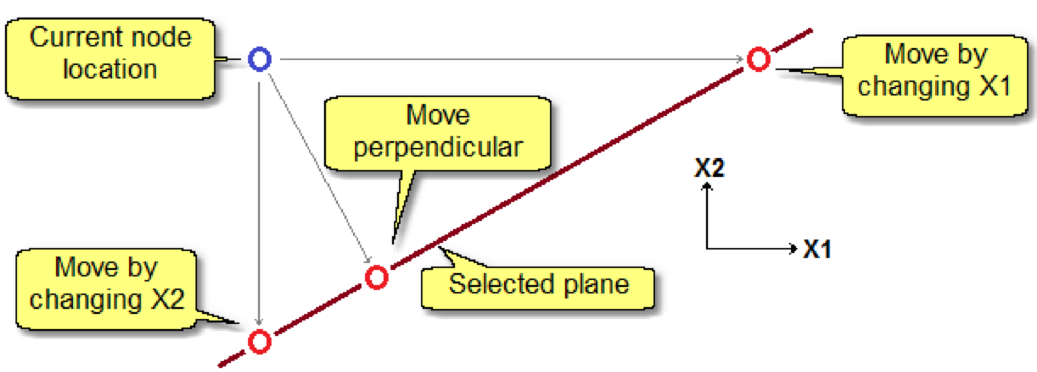
Set global coordinate

- select the nodes to move using the standard node selection option.
- select the option in the menu above.
- select the global coordinate (X1/X2/X3) and specify the value or click  and select an existing node with the same relevant coordinate.

Move to a selected plane

- select the nodes to move using the standard node selection option.
- select the option in the menu above.
- define the plane (select three nodes).

For example:

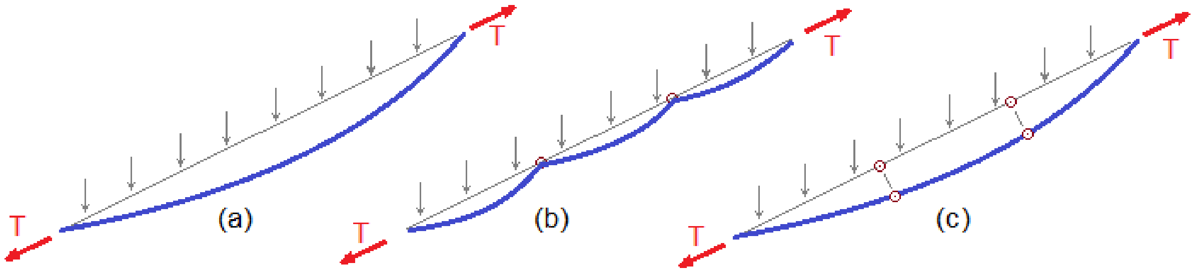


3.3.6.7 Update cable nodes

Update the initial coordinates of **intermediate** cable nodes, i.e. the initial sag of the cables resulting from the self-weight (and additional uniform load, if defined) and the initial tension. The initial coordinates must be updated:
- after any changes to the cable element parameters, including initial tension force
- after the cable property is assigned to any cable with intermediate nodes.

Note:
- the program automatically update the intermediate cable nodes if the the cable element is first assigned to a single beam and the beam is then "split 270" to create the intermediate nodes.

The program automatically calculates the initial geometry for a single cable element:



- If there are no intermediate nodes, the initial geometry is calculated as shown in (a).
- when intermediate nodes are present, the program will calculate the initial geometry incorrectly, as shown in (b).
- this option corrects the initial geometry at the intermediate nodes, as shown in (c).

Select the intermediate nodes using the standard node selection option. Do not select nodes with attached supporting elements.

3.3.7 Delete Nodes

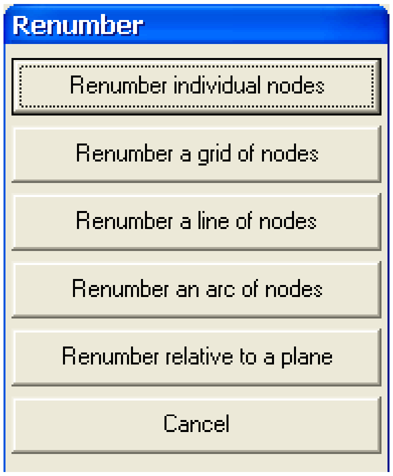
Use this option to delete nodes from the model:
- nodes with beams or elements attached to them cannot be deleted.
- nodes selected to define "Grid lines" cannot be deleted.

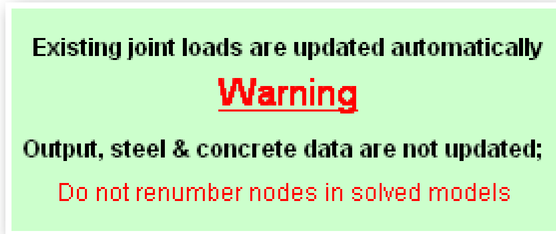
Select one or more nodes using the standard Node Selection option.

Note:
- Nodes that have no beams or elements attached to them are ignored by the program; these unused nodes may be deleted from the model, but it is not strictly necessary.

3.3.8 Renumber Nodes

Use this option to renumber existing nodes:

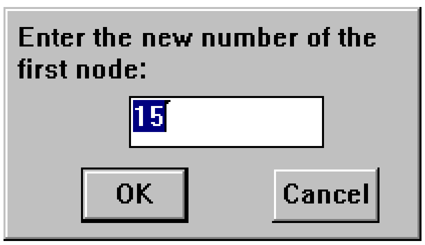




Individual nodes

Select the nodes using the standard Node selection option.

Note that the order of selection is important; ***nodes are renumbered in the order that they are selected***.



Type the new number of the first node selected; all of the nodes selected are renumbered sequentially. If the program discovers that a number has already been assigned to another node, the program assigns the original number of the selected node to the other node.

Example:
- nodes 41, 42 and 43 are selected (in that order).
- 75 is specified as the new number for 41
- the nodes are renumbered 75,76 and 77 respectively
- if, for example, node 76 is an existing node, it will be renumbered 42.

Grid of nodes

Similar to Line of Nodes 194; define the grid by pointing to the three corner nodes defining it and then enter the new number of the node at the start of the base line.

Line of nodes

Use this option to renumber all the nodes in a line:
- Select the two nodes defining the start and end of the line
- Type the new number of the first node in the line; all of the nodes selected are renumbered sequentially. If the program discovers that a number has already been assigned to another node, the program assigns the original number of the selected node to the other node.

Arc of nodes

Use this option to renumber all the nodes lying on an arc:

- Select the two nodes defining the start and end of the arc
- Select any other node lying on the arc
- Type the new number of the first node on the arc.

The program identifies all nodes on the defined arc and renumbers all of them.

All of the nodes selected are renumbered sequentially. If the program discovers that a number has already been assigned to another node the program assigns the original number of the selected node to that node.

Plane of nodes

Renumber all nodes on selected planes. This option is handy for renumbering an entire model or parts of a model consisting of more than one plane. Note that the planes do not have to be parallel.

- select the nodes to be renumbered using the standard node selection option
- define a plane that specifies the renumbering order; the plane is defined by selecting three existing nodes.
- specify the new number of the first node

The renumbering order is determined as follows:

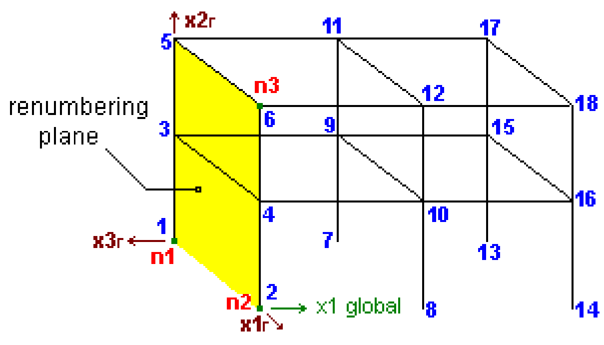
- the first two nodes define the x1r axis of the renumbering plane; the third node defines the x2r axis of the plane; the x3r axis is determined by the right-hand rule
- the program sorts the nodes according to their x3r coordinate, starting with the nodes closest to the renumbering plane. Note that if there are nodes on both sides of the plane, the program first selects all nodes on one side, then all nodes on the other side.
- for nodes with identical x3r coordinates, the program then sorts according to the x2r coordinate, beginning with the smallest value.
- for nodes with identical x3r and x2r coordinates, the program then sorts according to the x1r coordinate, beginning with the smallest value.

All of the nodes selected are renumbered sequentially. If the program discovers that a number has already been assigned to another node, the program assigns the original number of the selected node to that node.

Example:

Renumber the following space frame; the renumbering is to start on the planes perpendicular to X1 global

- select nodes **n1**, **n2** and **n3** to define the renumbering plane
- specify **1** as the new number of the first node



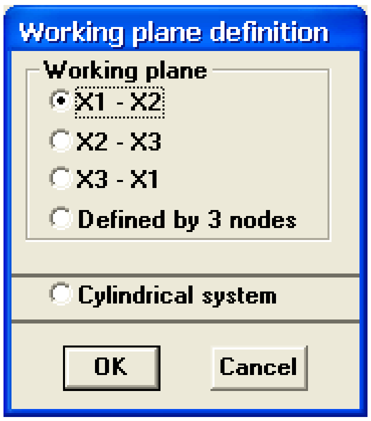
- the nodes on the **x1r-x2r** plane are renumbered first (**1-6**); the bottom nodes have the smallest x2r values and the left node has the smallest x1r value, i.e. it is renumbered first.
- then the nodes on the parallel planes are renumbered (**7-12**) and (**13-18**) are renumbered in the same order.

3.3.9 Node coordinate system

By default the moves on the X1-X2 plane (even if the display is rotated) and node coordinates are defined relative to this Cartesian global system. This option allows you to define coordinates relative to a another system.

A different Cartesian system or a cylindrical system may be defined:

- the will move along the plane of the Cartesian system (referred to as the "Working Plane") or along the arc of the cylindrical system.
- node coordinates are defined relative to the axes of the new system.



There are two options available:

- Working Plane 197
  Define the working plane as a different global plane, or as a plane formed by any three nodes in the model.

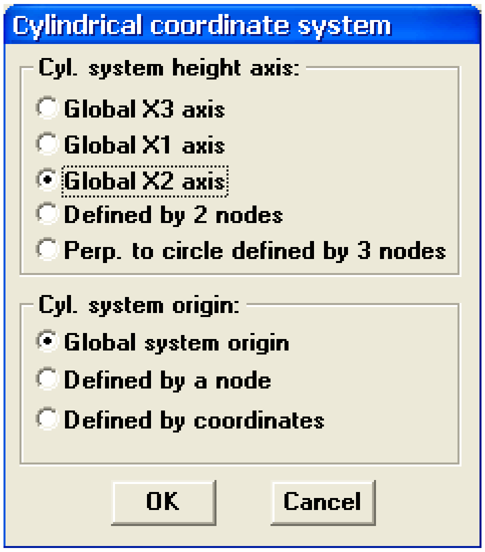
- Cylindrical System 196
  Define a cylindrical coordinate system. The crosshair movement is controlled by defining the angle, radius and height from an arbitrary origin.

3.3.9.1 Cylindrical coordinate System

Coordinates may also be defined relative to a cylindrical coordinate system. If a cylindrical system is active, the coordinates used to define node locations are:

- radius (R) instead of X1
- angle (Ang) instead of X2
- height (H) instead of X3

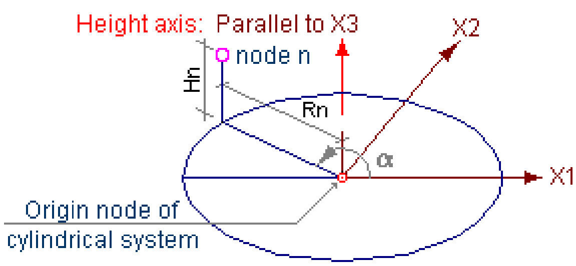
A cylindrical coordinate system has a central non-cylindrical 'height axis' which may point in any direction and which passes through a selected 'origin' node:



The origin location depends on the option used to define the height axis:

Height axis	*Origin:*
• Parallel to global axis:	according to option selected in menu
• Defined by 2 nodes:	at first node selected
• Perp. to circle:	at center of circle

For example, if X3 is chosen as the "height axis" and the origin is set at any arbitrary location, the cylindrical coordinate system is:

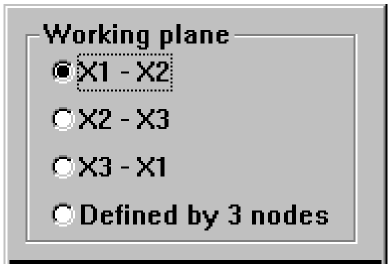


The options **Line equal, Line general** and **Grid** may all be used while a cylindrical coordinate system is in effect. For examples, refer to:
Line equal - define an arc 178
Line general - define an arc 180
Grid - Cylindrical coordinate system 181

Note:
- select a Working Plane 197 to cancel a Cylindrical system

3.3.9.2 Working Plane

The working plane is a plane in space along which the cursor moves. By default, X1-X2 is the working plane. Any of the three global planes or any arbitrary plane may be defined as the current working plane.



If the working plane is an arbitrary plane **not** parallel to one of the global planes (defined by three nodes), the coordinates displayed in the Data Options Area are "U, V and W", where:

The origin is located at the first node selected to define the working plane.

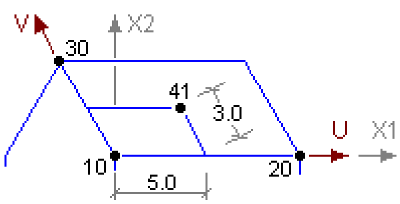
- U is measured from the first node in the direction of the second node.
  The value of U is revised by moving the mouse horizontally.
- V is measured perpendicular to U in the general direction of the third node.
  The value of V is revised by moving the mouse vertically.
- W is the dimension perpendicular to the plane. Its positive direction is determined by the right-hand rule.
- The value of W is revised in the Dialog Box at the bottom of the screen.

Note:

- the Working Plane option is in effect only for node definition.
- the program automatically rotates the model if the working plane is perpendicular (or nearly perpendicular) to the screen when you enter the node definition option.

Example:

Use a working plane to define node 41 on the sloped roof below. The node location relative to the plane of the roof is known (as detailed in the figure) but not relative to the global coordinate system.

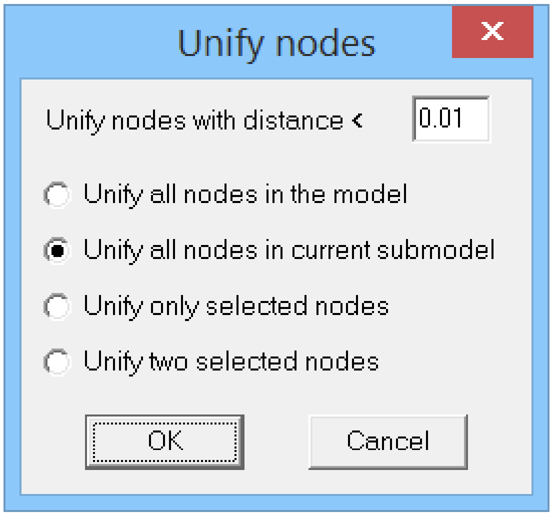


- define a Working plane as **Defined by 3 nodes**.
- select nodes 10, 20 and 30 **in that order** to create the Working Plane U-V-W as shown.
- select **Single node**; move the cursor until U = 5.0 and V = 3.0 are displayed in the Data Options Dialog Box (note that the cursor moves along the working plane only).

3.3.10 Unify nodes

In certain cases it may be convenient to define two different nodes at the same location. This usually occurs in models which have two planes connected along a common line, where each plane was defined by a different **Grid** command. Each command creates nodes at the same locations on the common line; the elements adjacent to the common line are not connected to the same nodes and so the two planes are not physically connected.

The **Unify** option searches for nodes with identical coordinates. If it locates such nodes, it connects all elements attached to them to the node with the lowest number (i.e. the JA, JB, JC and/or JD nodes of the element are revised). The higher numbered nodes are deleted from the model.



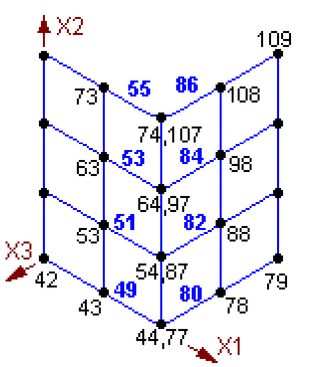
- Specify the 'tolerance'; nodes separated by a distance less than this value will be considered as identical.
- Select:
  - **Unify all nodes**
    the program checks the entire model, including all submodels
  - **Unify all nodes in current submodel**
    the program checks only the current submodel (or main model)
  - **Unify selected nodes**
    select nodes using the standard node selection option; the program checks only these nodes.
  - **Unify two selected nodes**
    Select two nodes; the program eliminates the ***first*** node selected (all beams and elements will be connected to the second node).

Note:
- after unifying the nodes, the program searches for beams, elements and walls with zero dimensions and deletes them.

Example:

The two grids below were defined such that there are two nodes at every location on the common line.

The end nodes for four beams in the model are as follows:



Before the UNIFY option:			After the UNIFY option:		
Beam	JA	JB	Beam	JA	JB
53	63	64	53	63	64
55	73	74	55	73	74
84	97	98	84	64	98
86	107	108	86	74	108

Before:
- Beams 53 - 84 not connected
- Beams 55 - 86 not connected

After:

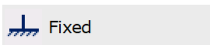
- The beams are connected
- Nodes 77, 87, 97, 107 are deleted

3.4 Restraints

Define the location of restrained nodes, i.e. support locations, by specifying which nodes have restrained degrees-of-freedom. Each restrained degree-of-freedom prevents translation or rotation in a specific direction. Models without any restrained nodes are unstable (i.e. the stiffness matrix is singular).

The restraints are normally defined in the Global Coordinate System directions. In the case of a sloped support, etc., restraints may be defined relative to any arbitrary local axis.

Support locations are specified by moving the crosshair on the graphic display to point to an existing node.



Define supports 201 where all translation ***and*** rotation global degrees-of-freedom are restrained.

Define supports 201 where only translation global degrees-of-freedom are restrained.

Define supports 201 with any other combination of restrained global degrees-of-freedom.

Define a 'local' support coordinate system 202 (a system for supports not parallel to global axes) and to assign supports to these systems.

Connect selected nodes by means of "Rigid links 204" (Master-slave nodes).

Delete supports; Select the nodes with the defined support using the standard Node Selection option.

3.4.1 Global restraints

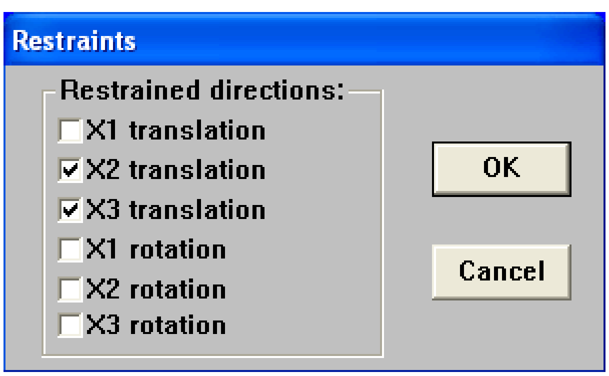
Select the support type:

- **Pinned** - all relevant translation global degrees-of-freedom are restrained
- **Fixed** - all relevant translation ***and*** rotation global degrees-of-freedom are restrained.
- **Other** - any other combination of restrained global degrees-of-freedom

The relevant degrees-of-freedom are:

	Translation	Rotation about
Plane frames	X1, X2	X3
Plane grids	X3	X1, X2
Space frames	X1, X2, X3	X1, X2, X3

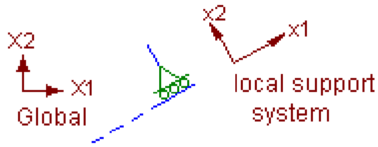
if you select **Other type**, specify the degrees-of-freedom that are ***restrained***. For example:



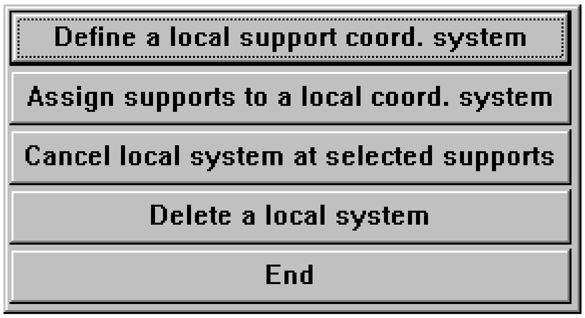
Select the nodes with the defined support using the standard Node Selection option.

3.4.2 Rotated restraints

Restraints may also be defined about any arbitrary non-global coordinate system, referred to as the "local support system". For example, the following support requires a local system to accurately define it:

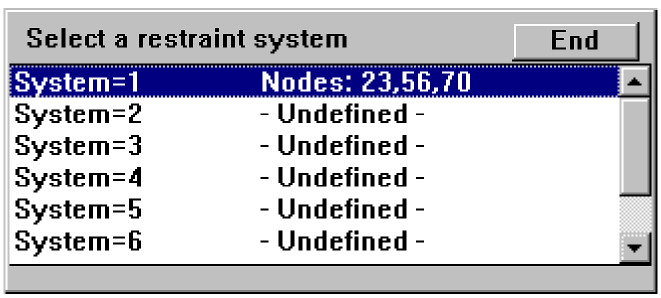


- Support systems may be either 'Cartesian' or 'Radial'; up to 63 different local support systems may be defined.
- A system is defined and assigned to nodes in separate options:

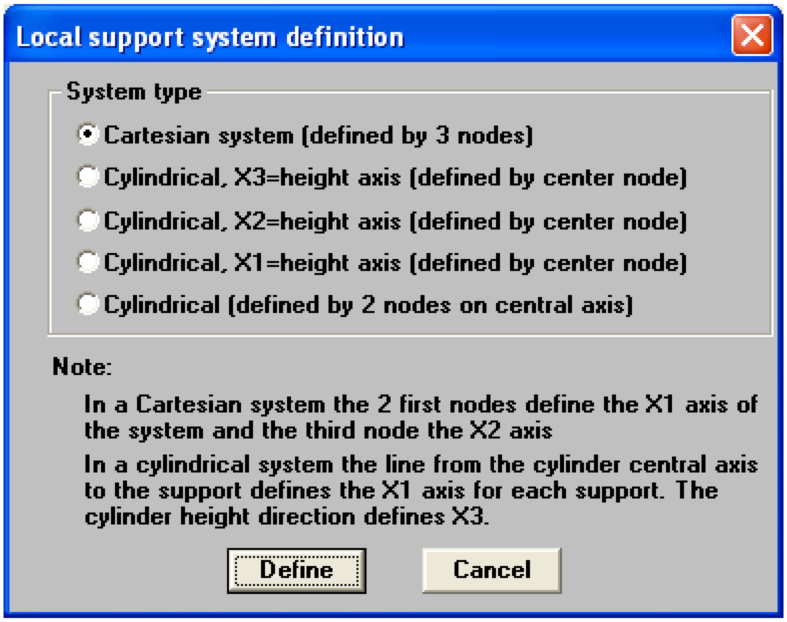


Define

A list of local systems is displayed; select one. Select an **Undefined** system to define a new one.

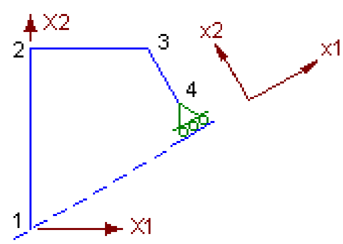


Local coordinate systems may be either Cartesian or Cylindrical:

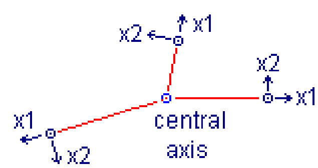


- Cartesian:
  A local support system is defined by three nodes JA, JB and JC, where:
  - the system x1 lies along JA - JB and points towards JB.
  - the system x2 is perpendicular to x1 and points towards JC

  In the example, nodes 1 and 2 define x1, while either 3 or 4 can be selected to define x2.



- Cylindrical:
  The system is defined simply by identifying its central axis; when nodes are assigned to the system, the x1 axis is perpendicular to the line joining the node to the axis. For example:



The central axis may be any of the global axes or may be defined by any 2 existing nodes.

Assign

- The program displays the list of the local systems; select one.
- Select the nodes that the system is to be assigned to using the standard Node Selection option

Cancel

'Canceling' the local system does not delete the support, but merely reassigns the support to the global system.

- Select the nodes with the standard Node Selection option.

Delete a local system

Click on the line in the list box; the system is deleted.

All restraints and springs assigned to this system remain but are now relative to the global coordinate system.

3.4.3 Rigid link

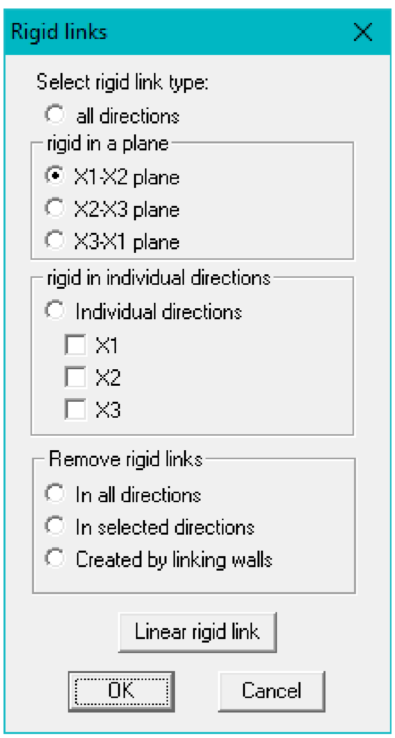
The rigid link option specifies that the specified deflections and/or rotations of selected nodes shall be identical (also referred to as "Master-Slave nodes). Rigid links accurately model the behaviour of many structures; they may significantly reduce the size of the stiffness matrix and hence shorten the solution time.

For example, all nodes in a floor slab may be connected with rigid links so that the entire slab deflects uniformly and rotates uniformly about the vertical axis. Only three degrees-of-freedom are added to the stiffness matrix for the entire floor slab.

Note:

- The lowest number node in the selected group is automatically designated as the Master node.
- All restraints defined at Slave nodes are transferred to the Master node. This may lead to unexpected results. Refer to supports at slave nodes 207 for examples.
- The spring constants defined at Slave nodes are ***summed*** with the spring constants at the master node. This can lead to problems when calculating spring stresses in the Results options because the program uses the tributary area for the master node only.
- Submodels: if rigid links are defined for all nodes attached to the connection points of a submodel instance, the program asks whether to define the same rigid links for ***all*** of the submodel nodes.
- Use the linear rigid link 208 to link a slave node to ***two*** master nodes.

The following types of rigid links may be defined:



All directions

The selected nodes will be rigidly linked in all degrees-of-freedom, i.e. the three-dimensional block formed by the selected nodes will maintain its original shape.

For any pair of nodes A and B:

$\delta_B = \delta_A + \theta_A X_{A-B}$

where: δ_A = the deflection at A in a global axis direction.

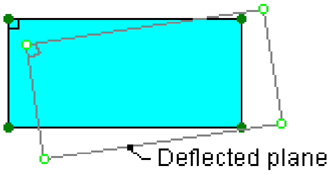
δ_B = the deflection at B in the same direction.

θ_A = the rotation at A about the axis of deflection.

X_{A-B} = the distance from A to B projected on the global axis.

Rigid in a plane

The specified nodes move as a rigid body in the specified plane and maintain its original shape. For example:



For any pair of nodes A and B located in the plane:

$\delta_B = \delta_A + \theta_A X_{A-B}$

where: δ_A = the deflection at A

δ_B = the deflection at B

θ_A = the rotation at A

X_{A-B} = the distance from A to B

Note that the deflection of each node ***perpendicular*** to the plane may be different.

This option is particularly useful for earthquake analysis; a model of a multi-storey building with only a relatively few degrees-of-freedom may be created.

Rigid in individual directions

Select **individual directions** and select one or more of the global directions.

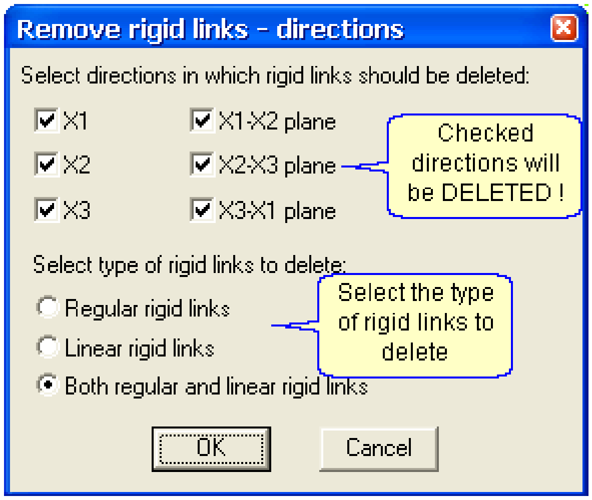
Remove

Delete rigid links defined at the specified nodes or links defined in the Walls option.

The links may be deleted in:

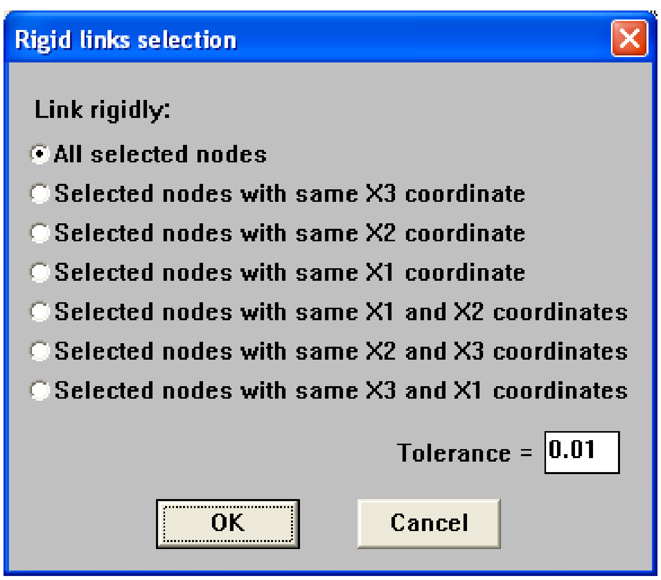
all directions
- all "Rigid in a plane' and all "single directions".
- all regular and Linear rigid links.

selected directions only:



Created by linking walls
Only rigid links created by the Wall - link option.

For all options (including Remove), select the nodes to be connected by the specified rigid link type using the standard node selection option:

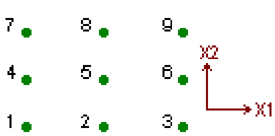


All selected nodes

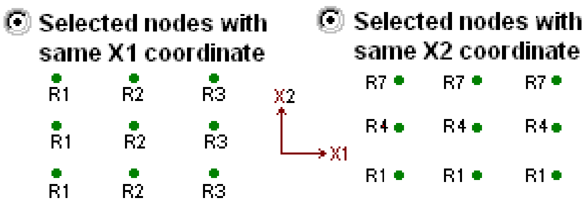
All selected nodes will be included in a single rigid link group.

Selected nodes with same coordinate

Nodes with the same specified coordinate will be included in a rigid link group. For example, the following 9 nodes are selected:



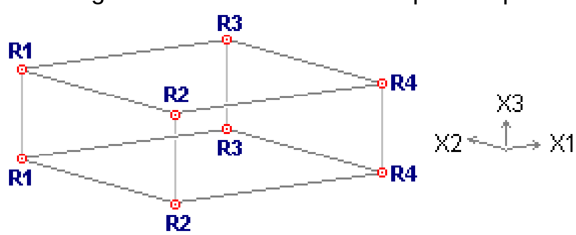
The rigid link groups are created as follows according to the option selected:



Note that the groups are numbered according to the smallest node number of the nodes included in the group.

Selected nodes with same 2 coordinates

Create rigid links between nodes on parallel planes. For example, to create the following rigid links -



- select **Selected nodes with same X1 and X2 coordinates**
- select all eight nodes

The program divides the nodes into the four rigid link groups as shown above; the nodes in each group have identical X1 and X2 coordinates.

Tolerance

Only the selected nodes with the same relevant coordinate as the 'Master node' are included in the rigid link group.

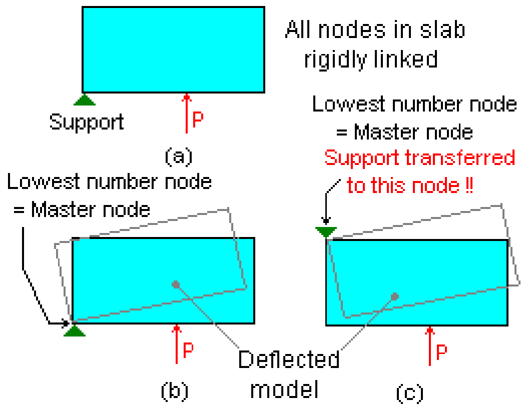
- Specify a tolerance value for the coordinate of the Master node (current geometry length units).

3.4.3.1 Supports at slave nodes

Supports defined at slave nodes are automatically transferred to the master node (the lowest numbered node in the rigid link group). The following two examples show how this may lead to unexpected results.

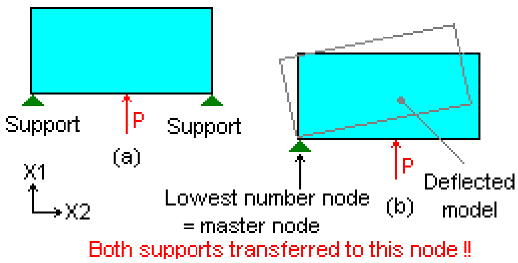
Example 1:

The simplified slab shown in Figure (a) should rotate about the support at the lower-left corner. If the master node is at the support node as shown in Figure (b), then the model will behave as expected. However, if for example, the lowest numbered node is at the ***top-left*** corner, then the model will rotate incorrectly as shown in Figure (c).



Example 2:

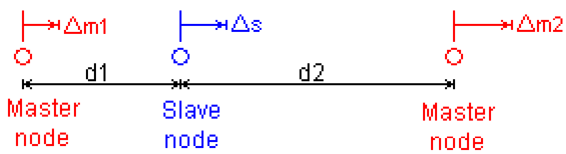
Both bottom corners in Figure (a) are restrained for X1, X2 deflection; the model should not rotate at all. However, the model will rotate about the master node as shown in Figure (b).



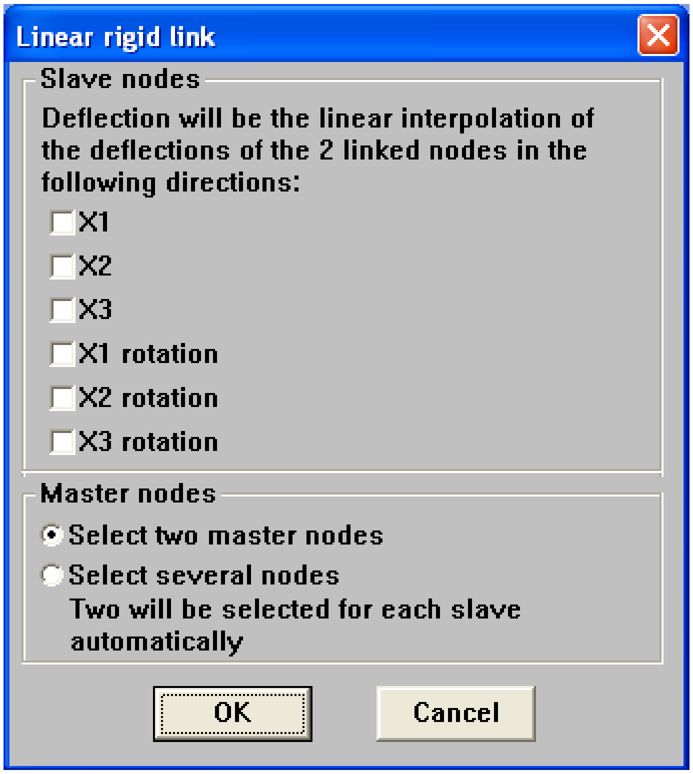
The support at the lower-left corner must be restrained for rotation about X3 to prevent the rotation.

3.4.3.2 Linear rigid link

Connect a slave nodes to ***two*** master nodes; the deflection/rotation of the slave node is a weighted average of the deflections of the two master nodes, proportional to the distances between the nodes:



$$\Delta s = \frac{d2}{d1+d2}\Delta m1 + \frac{d1}{d1+d2}\Delta m2$$

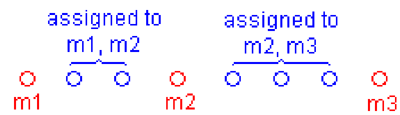


Select the slave nodes using the standard node selection option.

Select two or more several master nodes:

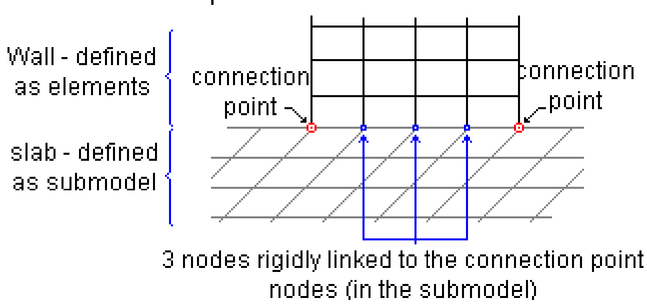
Two master nodes
All the selected slave nodes are assigned to the two selected nodes.

Several master nodes
Each of the selected slave nodes is assigned to the two closest master nodes:



Example:

A slab defined as a submodel is connected to a wall defined as elements; the number of connection points should be kept at a minimum in order to reduce the solution time.



3.5 Beams

Define beam elements by specifying:

- location : designate the end nodes.
- properties : define properties (A,I), dimensions or a steel shape
- material : select a program material or define a new one
- local axes : specify the local x1/x2/x3 axis directions
- orientation : specify major/minor axes orientation (relative to the local axes)
- releases : define pinned connections
- rigid offsets : define rigid segments at the beam ends

Define	• Define 211 one beam element by identifying its end nodes. • Define a series of beams all lying on a straight line or arc. Select the start and the end of the line; the program automatically locates all intermediate nodes and connects them with beams. • Define a continuous string of beams, where the start node of any beam is the same as the end node of the previous beam.
Grid	Define a parallelogram grid of beams 214. The grid is defined by identifying the three corner nodes on the 'base' line and the 'height' line of the grid; the program automatically searches for all intermediate nodes and creates a grid of beams.
Braces	Define a series of bracing 216 beams that all start on a common line and end on a different common line.
Columns	Create columns 218 at selected nodes by defining the number of levels and the story intervals. The program creates the nodes at the levels and connects them with beams.
Delete	Delete 219 an existing beam.
Properties	Define section properties 219 (including material) and assign them to beams.
Releases	Release degrees-of-freedom 255 (translational or rotational) at beams.
Offset	Define rigid offsets 259 at the ends of beam elements and assign them to the beams.
Renumber	Renumber 267 a beam element already defined.
Split	Divide an existing beam 270 into two or more beams if intermediate nodes are located along the length of the beam.
Local axes	Specify or revise the direction of the local axes 272 for beams.

Click **Stages** to select a stage other than **Whole model;** different properties, releases and offsets may be defined for each stage and beams may be removed:

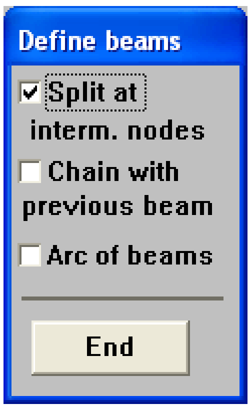
Deactivate	'Remove' a beam from the current stage.
Restore	Restore a beam to the current stage.

3.5.1 Define beams

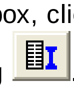
Define:

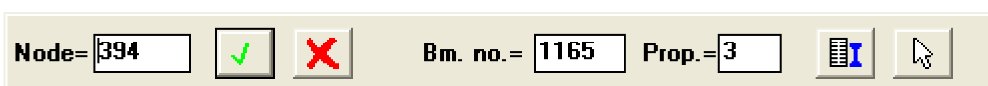
- a single beam 211
- a line of beams 212
- an arc of beams 212
- a chain of beams 213

The required option is selected by setting the following options in the side menu:



All beams are automatically assigned with:

- the program default local axes.
- the **Bm.no**. and **Prop** group number displayed in the Dialog box; to revise these numbers move the to the box, click the mouse and enter a new number (the property number may also be selected by clicking .. Note that selecting **Property 0** will create Dummy beams.

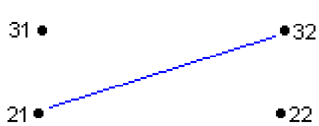


3.5.1.1 Single beam

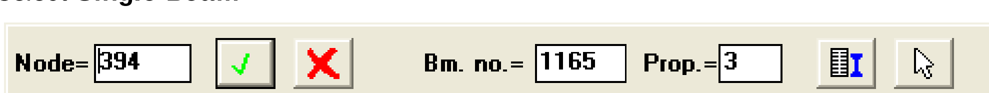
Point to the start node and the end node of the beam; the beam is drawn immediately. The beam is automatically assigned with:

- the **Beam No.** displayed at the bottom of the screen
- the **Prop** group number displayed at the bottom of the screen
- the program default local axes.

Example: Create a beam between nodes 21 and 32



- select **Single Beam**



- move the adjacent to node 21 so that the node is highlighted with the blip ■ and **Node = 21** appears in the Dialog box; click the mouse.
- move the adjacent to node 32 so that the node is highlighted with the blip ■ and **Node = 32** appears in the Dialog box. Note that the program draws a beam from node 21 to the highlighted node. ***Double-click*** the mouse.
- the beam is created and assigned with the **Bm.no**. and **Prop** group number displayed in the Dialog box; to revise these numbers move the to the box, click the mouse and enter a new number (the property number may also be selected by clicking .

Note:

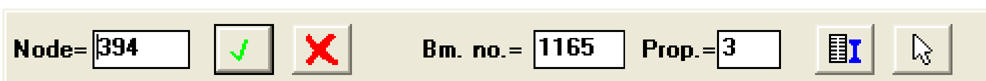
- set **Split at intermediate nodes** to define one beam only when there are intermediate nodes along the line between the two selected nodes.

3.5.1.2 Line of beams

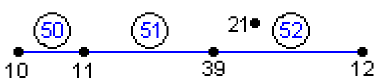
Define a continuous straight line of beams.

All beams are automatically assigned with:

- the **Bm.no**. and **Prop** group number displayed in the Dialog box; to revise these numbers move the to the box, click the mouse and enter a new number (the property number may also be selected by clicking .
- the program default local axes.



Example: define beams 50 to 52 -



- set **Split at intermediate nodes**
- set **Beam No. = 50** in the Dialog box.
- move the adjacent to node 10 so that the node is highlighted with the ■; click the mouse.
- move the adjacent to node 12 so that the node is highlighted with the ■; ***double-click*** the mouse.
- the program identifies nodes 11 and 39 as lying on line 10-12 and creates beams 50, 51 and 52; node 21 is not on the line and is ignored by the program.

Note:

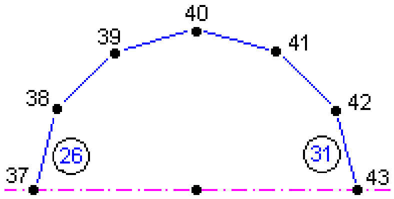
- if **Split at intermediate nodes**, only one beam is created between nodes 10-12.

3.5.1.3 Arc of beams

Use this option to define a line of beams along a circular arc; identify the first and last nodes on the line (as in the regular Line 212 option) and one additional node lying along the arc.

Example:

define beams 26 to 31:



- at the right side of the screen, set the following options:
  ☑ **Split at intermediate node**
  ☑ **Arc of beams**
- set **Beam no. = 26** in the Dialog box
- move the adjacent to node 37 so that the node is highlighted with the ■; click the mouse.
- move the adjacent to node 43 so that the node is highlighted with the ■; click the mouse.
- move the adjacent to any of nodes 38 to 42 to identify the arc location. When the node is highlighted with the ■; ***double-click*** the mouse.

3.5.1.4 Chain of beams

Define a continuous string of Lines where the start node of any line is the same as the end node of the previous line.

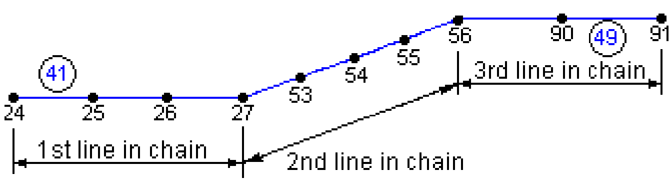
- point to the start node of the first beam of the first line in the chain, to the end node of the first line and then to the end nodes of all of the following lines in the chain.

All beams are automatically assigned with:

- the first beam is assigned the **Beam no.** displayed at the bottom of the screen and the remaining beams are numbered consecutively.
- the **Prop** group number displayed in the Dialog box; to revise the number move the to the box, click the mouse and enter a new number (the property number may also be selected by clicking .
- the program default local axes.

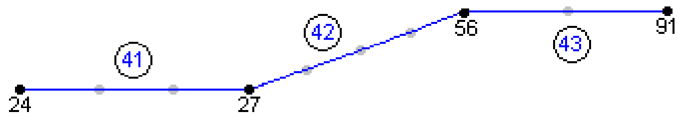
Example:

Define beams 41 to 49.



- set at the right side of the screen.
  ☑ **Split at intermediate node**
  ☑ **Chain with previous beam**
- set **Beam No. = 41** in the Dialog box.
- select the start node of the first line of beams:
  move the adjacent to node 24 so that the node is highlighted with the ■ blip; click the mouse.
- select the end node of the first line of beams:
  move the adjacent to node 27 so that the node is highlighted with the ■ blip; click the mouse.
- select the end node of the second line of beams:
- move the adjacent to node 56 so that the node is highlighted with the ■ blip; click the mouse.
- select the end node of the third line of beams:
- move the adjacent to node 91 so that the node is highlighted with the ■ blip; the mouse..

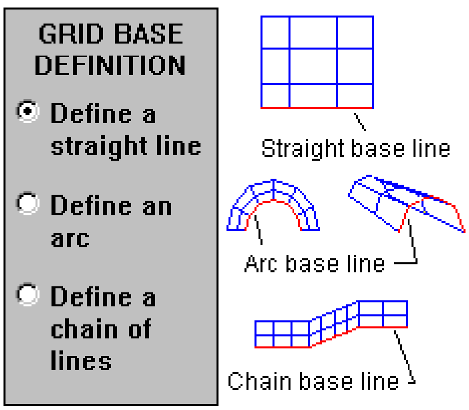
Note:
- If ☐ **Split at intermediate nodes**, only the following three beams will be created:



3.5.2 Grid of beams

Use this option to create a parallelogram grid of beam elements by defining the 'base' line and the 'height' line of the grid, as follows:
- select the start node of the base line
- select the end node of the base line that is also the start node of the height line
- select the end node of the height line

The program automatically searches for all existing intermediate nodes and creates a grid of beam elements.

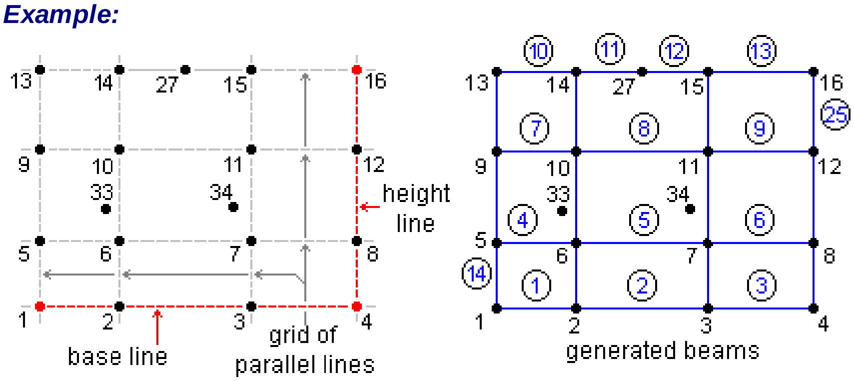


Note:
- All nodes must be defined before this option is selected; this option does not generate nodes.

3.5.2.1 Straight base line

Define a straight base line by selecting two nodes and a height line; the program identifies all nodes lying on the grid and generates the beams.

Example:



- select nodes 1 and 4 to define the base line of the grid
- select node 16 to define the height line of the grid (4-16)

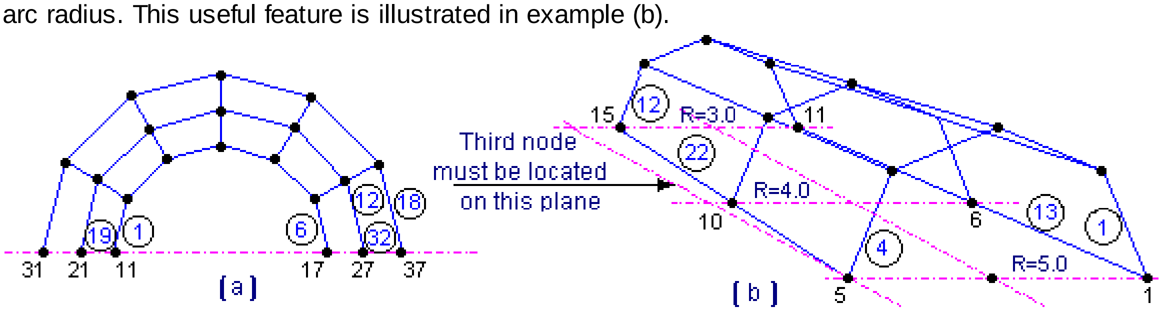
The grid is created as follows:
- the program identifies nodes 2 and 3 on line 1-4 and nodes 8 and 12 on line 4-16.
- all shown nodes lie on the grid of parallel lines, except nodes 33 and 34. Node 27 lies on line 13-16 and this is sufficient to include it in the grid.
- note the generated beams 11 and 12.

3.5.2.2 Arc base line

Use this option to define:
- a series of concentric arcs of beams all lying on the same plane, or,
- a series of parallel arcs concentric about the same **axis**.

First define the "base" arc as explained in Line - Arc of beams 212; then identify a third node defining the height line:
- if the third node lies on the same plane as the base arc, then a series of concentric arcs will be created. (Example 'a' below)
- if the third node does not lie on the same plane, then a series of parallel arcs concentric about the same axis will be created. (Example 'b' below).

The third node must lie on the plane formed by the central axis of the base arc and the radius to the end node of the base arc. The radius from the central axis to the third node ***need not be equal*** to the base arc radius. This useful feature is illustrated in example (b).



Example (a):
- select node 11 as the start node of the base arc
- select node 17 as the end node of the base arc
- select any of the nodes 12 to 16 to complete the definition of the base arc
- select node 37 as the third node of the grid.

Beams 1 to 32 are created.

Example (b):
- select node 1 as the start node of the base arc
- select node 5 as the end node of the base arc
- select any of the nodes 2 to 4 to complete the definition of the base arc
- select node 15 as the third node of the grid.

Beams 1 to 22 are created.

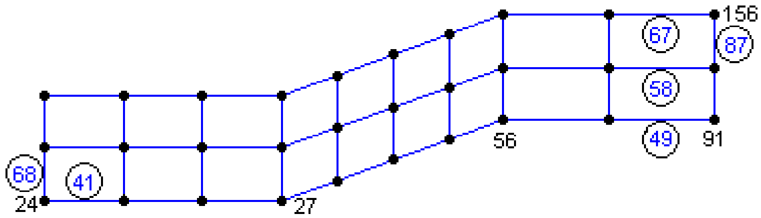
3.5.2.3 Chain base line

Define a grid, where the base line consists of a chain of connected lines (each line containing several nodes) where the lines are not necessarily parallel.

The base line is defined as explained in Line - Chain of Lines 213; the definition is completed by identifying a third node defining the height line.

Example:

Define beams 41 to 87:

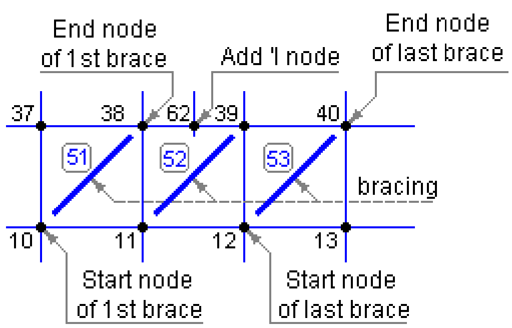


- Select node 24 as the start node of the first line of beams on the base line
- Select node 27 as the end node of the first line.
- Select node 56 as the end node of the second line
- Select node 91 as the end node of the third line of beams on the base line.
- Press [Enter] without moving the crosshair to complete the base line definition.
- Select node 156 to define the height line of the grid.

Beams 41 to 87 are created automatically.

3.5.3 Bracing

Use this option to define a series of beams which whose start nodes all lie on the same line and whose end nodes all lie on another line. This option is useful for defining rows of diagonal bracing, etc. For example:

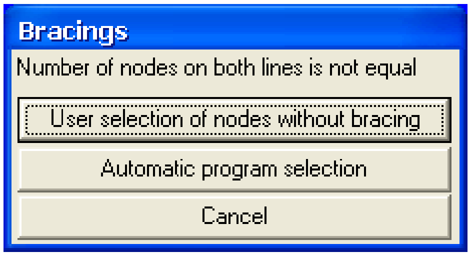


Define all of the bracing beams by defining the first and last beams only; the program automatically searches for intermediate nodes on the lines connecting the two start nodes and the two end nodes.

The option is best explained by an example. Referring to the figure above, define bracing 51, 52 and 53.

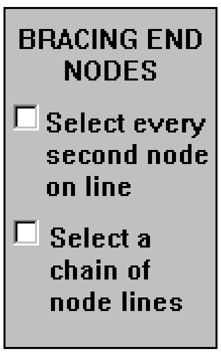
Program prompts:
Select start node of first brace in line: Select node 10
Select start node of last brace in line: Select node 12
Select end node of first brace in line: Select node 38
Select end node of last brace in line: Select node 40

The program searches for intermediate nodes along the lines 10-12 and 38-40, and identifies nodes 11, 62 and 39. If node 62 were not present, the program would draw the three beams immediately. In this case, there are more nodes on one line than the other:

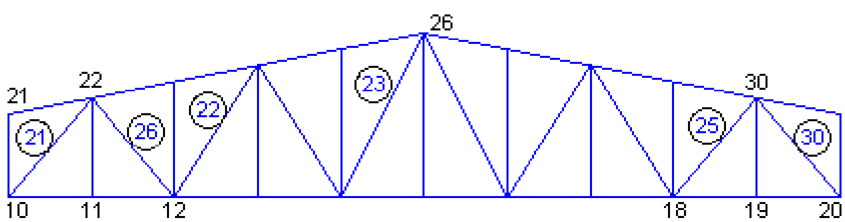


- **User selection ...**
  Select the nodes without bracing using the standard Node Selection option.
- **Program selection**
  The program will select the nodes so that the intermediate bracing is as parallel as possible to the end bracing members. In the above example, the program would select node 39 and not node 62.

Additional Options are available:



The use of these options is best explained by an example. Referring to the following example, the parallel diagonals 21 to 25 are connected to every ***second*** on the top and bottom chords of the truss. In addition, the top chord is not a straight line.



To define the diagonals 21 to 25:

a. bottom chord = start nodes

- set **Select every second node on the line** to
- select node 10 as the start node of the first brace
- select node 18 as the start node of the last brace

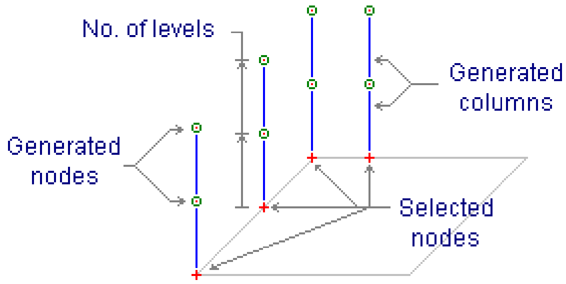
b. top chord = end nodes

- set **Define a chain of lines** to
- select node 22 as the end node of the first brace
- select node 26 as the end node of the last brace **in the first chain**
- select node 30 as the end node of the last brace **in the second chain**
- click the mouse without moving the to end the definition.

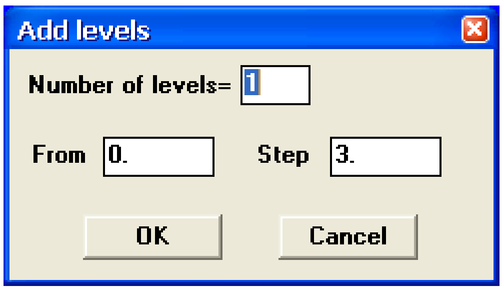
Define bracings 26 to 30 in a similar manner.

3.5.4 Columns

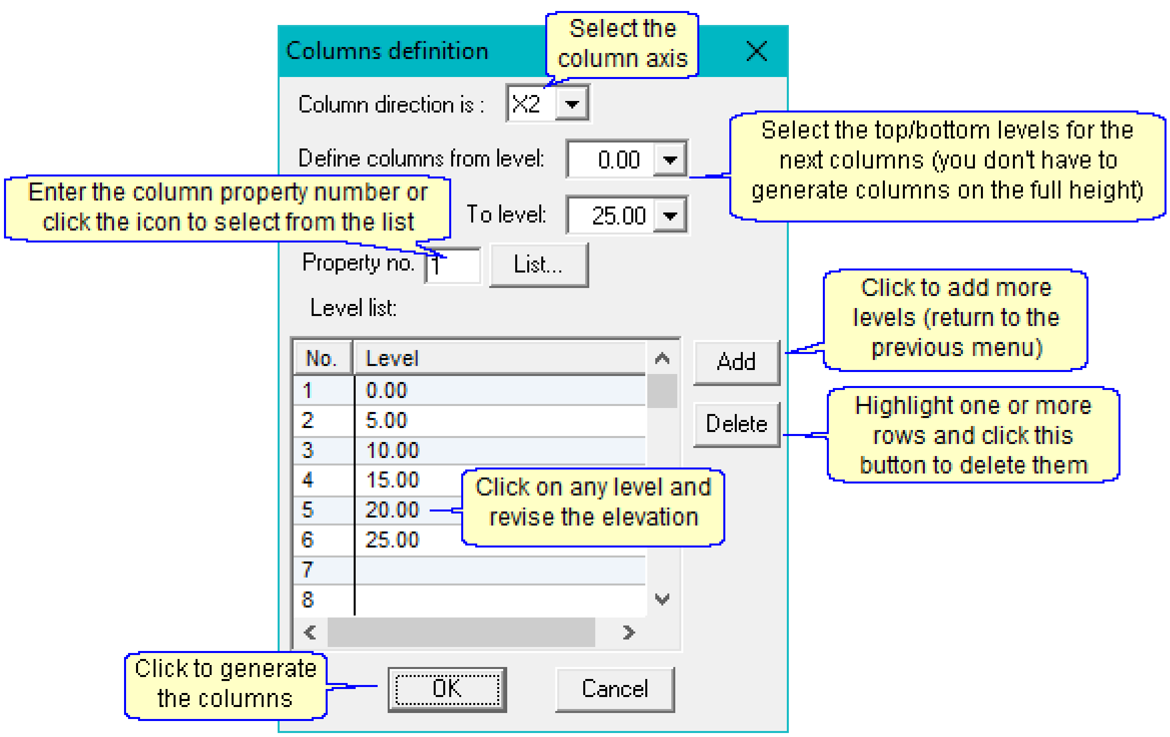
Create columns at selected nodes by defining the number of levels and the story intervals. The program creates the nodes at the levels and connects them with beams.



- Specify the number of levels and the distance between them (the elevation of any level may be revised in the next screen):



- Modify the list of leveLs - revise/delete selected elevations or add more levels to the list:



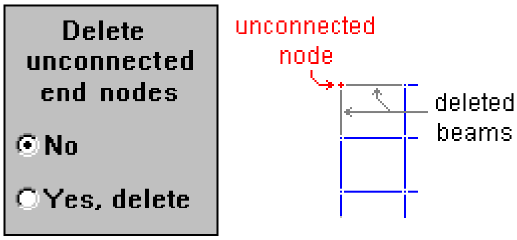
- select the base nodes using the standard Node Selection option. Note that any node on the extension of the column axis may be selected.

- the next time the Columns option is selected the program for the model, the program returns directly to the second menu and displays the same level list.

3.5.5 Delete

Select beams to delete using the standard Beam Selection option.

Note that nodes that are not connected to the model after the beams are deleted may be deleted from the model at the same time; set **Yes, delete** in the side menu.

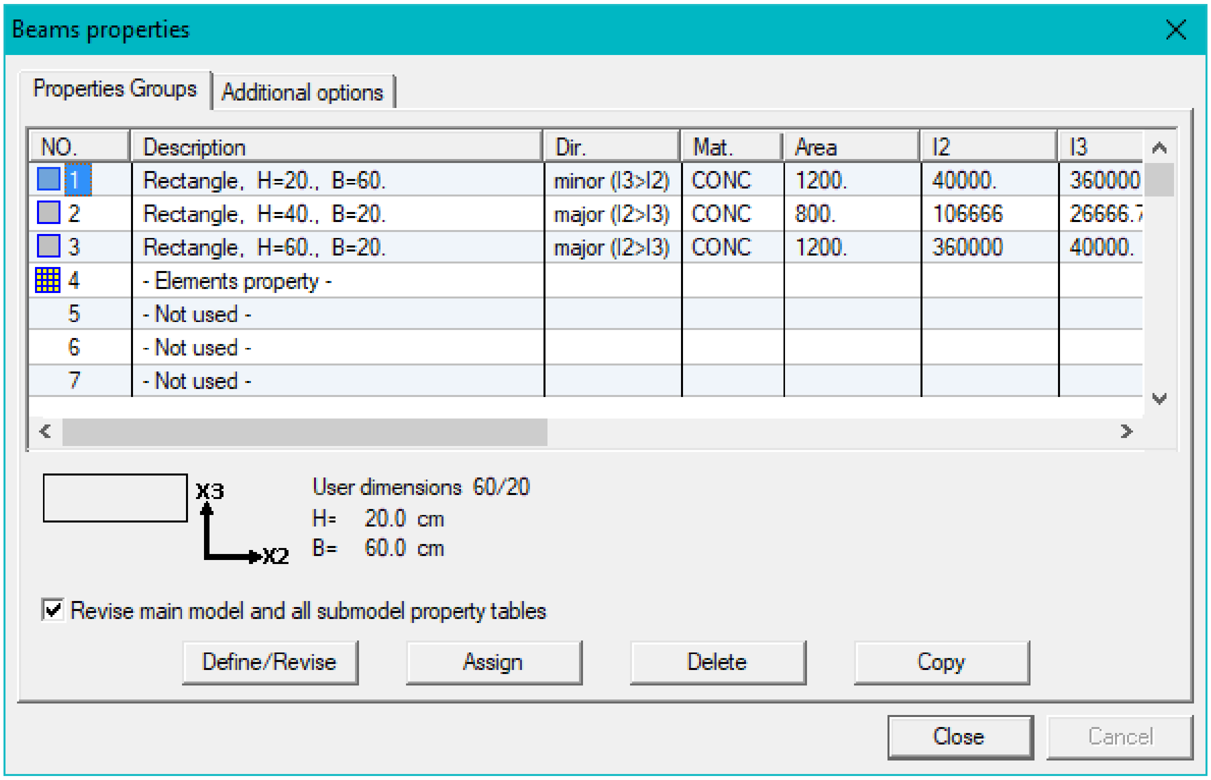


3.5.6 Properties

All beams must be assigned to a property group. When a new beam is defined, it is automatically assigned to the ***Property group*** listed in the Dialog box at the bottom of the screen. The property group number of an existing beam may be revised at any time.

Note:

- a property group may be assigned to a beam although the section properties are still undefined.
- properties may be entered directly, recalled from a steel section table, or alternatively the program may calculate the properties of standard geometric shapes from section dimensions.
- a topping may be added to any section type.
- tapered beam sections may also be defined.
- A beam may be designated as **Dummy**. Dummy beams may be loaded but they do not affect the stiffness of the model and will not appear in the output tables. For example, use a Dummy beam element if you have to define a linear load in a model that consists entirely of finite elements.
- each property group includes a material. ***STRAP*** includes ten permanent materials. The properties of these materials may be revised in the Setup option in the ***STRAP*** main menu. In addition, temporary materials may be defined for the current model (User-defined materials 252).
- each property group definition includes the major/minor axis orientation with respect to the beam local axes.
- new properties cannot be defined when a stage other than **Whole model** is the current one; existing properties may be assigned to any active beam.
- to delete a property:
  - click and highlight the property line in the table and click Delete
  - to delete unused properties, click the Additional options 250 tab and select Delete unused properties



Revise main model and all submodel property tables

If the current model contains submodels they may have their own property tables that are different from the Main model property table (refer to Submodel - new for more details). The property list in the dialog box corresponds to the currently displayed submodel/main model.

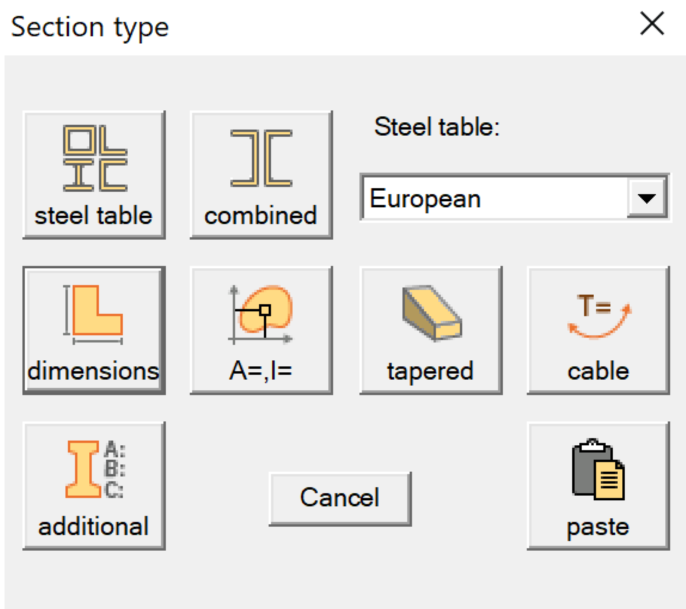
Select one of the following options:

☑ Any revisions to a property in the current list is applied to the main model and all submodel property lists. For example, the list above belongs to a submodel and you revise property 2 dimensions to 50x40 with this option set to ☑; property 2 in the main model list will also be a 50x40 rectangular section.

☐ Any revisions to a property in the current list is applied only to the current list.

3.5.6.1 Define

- first highlight the property to be defined / revised from the property list.
- select the method to define the property:



- for "**Steel table**": select one of the following tables of structural steel sections:

British	-	UB, UC, etc
European	-	IPE, HEA, etc
American	-	W, M, etc
User	-	a customized table of steel sections created by the user. Refer to Utilities - Create/edit a steel section table.

To display a demo video that explains how to create a "User" steel section table:

- *click on to start the video*
- *then click on to enlarge the display.*

Note:
- When a new section is defined, it automatically becomes the default section in the beam definition options.

3.5.6.1.1 General parameters

The following parameters are common to all section types:

Name

Define a name for the property group.

Note:
- for sections from the steel tables - the program automatically uses the section name in the table.
- for sections defined by dimensions - the program automatically builds the section name as the dimensions are typed in.
- the section names can also be modified in the Properties - additional options - property names 251 option

Units

Enter the properties in the units displayed. The units may be revised for each property without changing the model default units. Note that large numbers may be entered in exponential format

Material

Material type is selected from the current list of materials - the permanent program materials 254 and any temporary user-defined materials 252 entered for this model.

SF2 / SF3

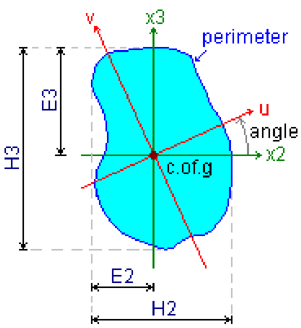
SF2, SF3 are the beam shear shape factors about the beam x2,x3 axes, respectively.

- SF2 is the factor associated with I2, V3 and M2, i.e. SF2 is the factor defined for plane grids.
- SF3 is the factor associated with I3, V2 and M3, i.e. SF3 is the factor defined for plane frames.

Some recommended shear shape factors are:

- Solid rectangular section - 0.85
- Circular tube section - 0.53
- Solid circular section - 0.89
- Rectangular tube section - 0.44

H2 / H3 / E2 / E3 / Angle / Perimeter



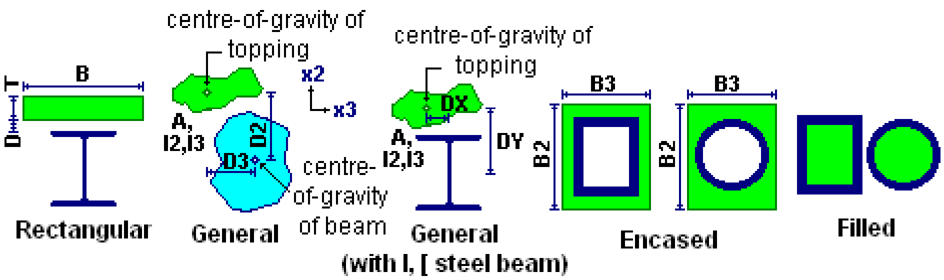
H2	=	maximum dimension parallel to x2 (used for wind loads)
H3	=	maximum dimension parallel to x3 (used for wind loads)
E2	=	maximum dimension from c.of.g to perimeter, parallel to x2 (currently not used)
E3	=	maximum dimension from c.of.g to perimeter, parallel to x3 (currently not used)
Angle	=	angle from x2 to major principal axis (u); positive = counterclockwise

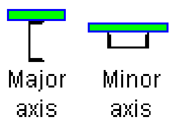
3.5.6.1.2 Composite beams

Define a "topping" for any beam section (steel, dimensions or properties). The topping may be of the following shape:

- ***rectangular*** : for steel **I** and **[** sections only. For **[** sections, the rectangular topping may be about the major or minor axes (see below)
- ***general*** : for all section types, except for hollow pipes and RHS sections
- ***fill/encase*** : for hollow pipes and RHS sections (steel sections or properties) and may be of any material.

The following data is defined by the user:





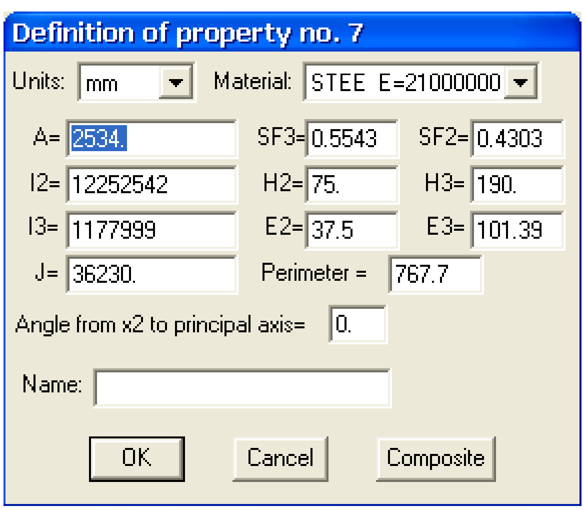
For **[** sections (defined by dimensions or selected from the steel table), rectangular topping may be defined about the major or minor axes.

A gap between the section and the topping cannot be defined for "minor axis".

Note:

- **I2** is associated with **D3**; **I3** is associated with **D2**
- for non-symmetric I-sections defined by dimensions: the topping is added to the "Bf" flange (not "Bf2").
- The area and moment-of-inertia displayed in the **Output** property tables are the ***composite*** properties
- The program automatically modifies the topping area and moment-of-inertia (in both directions) by the modular ratio n = Et/Eb when calculating the properties of the composite section.
- The program calculates **J** as follows:
  - Rectangular topping: **J = Jtop + Jbeam**
  - General topping: **J = Jbeam** (topping is ignored)
  - Box section: **J** = max (**Jtop**, **J** of RHS formed by topping)

3.5.6.1.3 Defined by A, I, etc.



where:

A = section area
I2 / I3 = Moment-of-inertia about the x2, x3 axes, respectively
J = torsional moment-of-inertia

Note:

- a table of section names can be displayed in the Additional options - property 251 names option
- all other parameters are explained in - Properties - general parameters 221.
- composite beams (concrete topping) may be defined; refer to Properties - composite 222.

3.5.6.1.4 Steel section table

The steel section tables are stored in the program.

- select a section type (W, L, 2L, etc.)
- a list of available sections for that type will be displayed; select one.
- define the material type and section orientation.

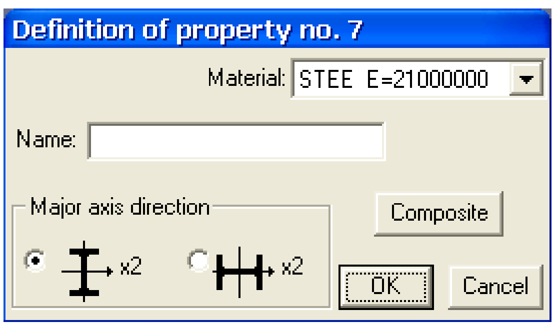
Select one of the following section types for more Help:

- All sections 224, except
- single angles 224
- sections created by Section generator 225 (CROSEC)
- joists 226

Note:

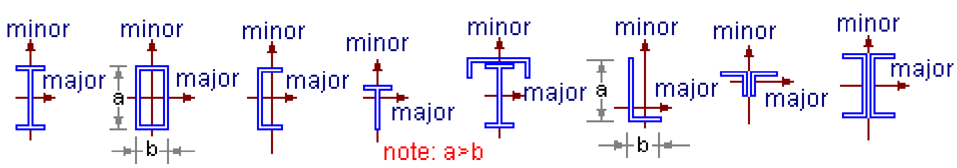
- all parameters not explained are explained in - Properties - general parameters 221.
- composite beams (concrete topping) may be defined; refer to Properties - composite 222.

All sections

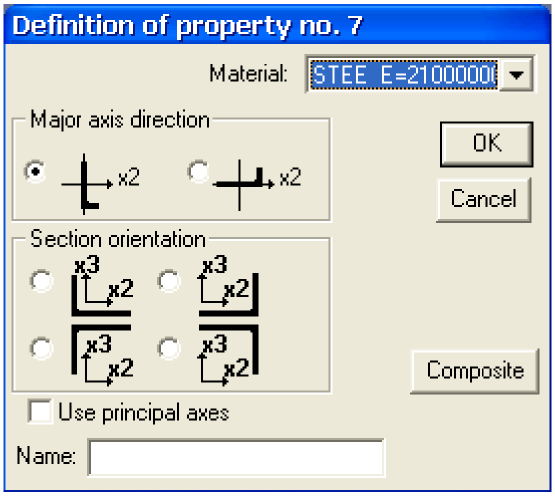


Major axis direction:

The steel section may be aligned so that its major axis resists either the M2 or M3 moments:



Single angles



Name:

- a table of section names can be displayed in the Additional options - property 251 names option

Section orientation:

This option may be selected for all unsymmetric steel sections, sections defined by properties and L-sections defined by dimensions. Although not important for analysis (the moment-of-inertia does not change if the flange location is inverted), the defined orientation will be displayed in the rendered drawing and will be used as the default orientation by the steel postprocessor, etc.

Note:

- The section orientation selected here is the default for this property group. The orientation may be revised for individual beams using the Local axes - flange 272 option
- The option is available for space models only

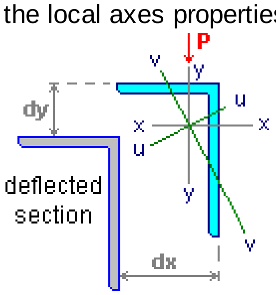
Major axis direction:

Refer to All sections 224.

Use principal axes:

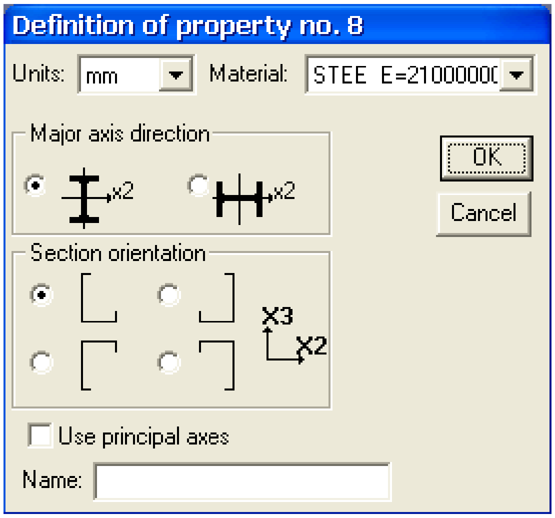
For doubly unsymmetric sections:

When the principal axes of a section are not aligned with the local (geometric) axes, e.g. a single angle, a more refined analysis may be carried out by using the principal axes properties (Iu, Iv, etc) instead of the local axes properties (Ix, Iy, etc).



When principal axis analysis is specified, a load applied in the direction of one local axis causes the section to deflect in both local axes directions. The program internally resolves the load to its principal axis components when solving the model. Note that all results will be displayed relative to the local axes.

Section generator (CROSEC)



Major axis direction:

Refer to All sections 224.

Section orientation:

Refer to Single angles 224 (principal axes).

Joists

The following steel joist types may be selected:

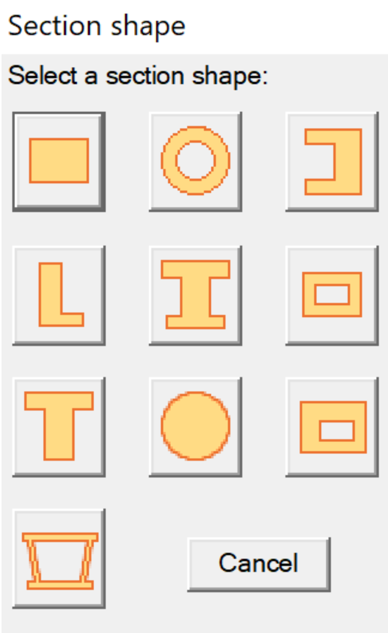
- K-Joist
- KCS
- LH/DLH

as specified by the Steel Joist Institute (1994)

Note:

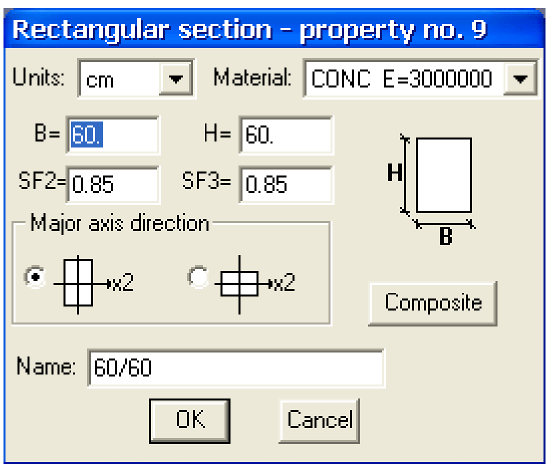
- a topping may not be added to joists
- the program will automatically revise the end releases of a member assigned with a Joist to "Pinned". The release will be revised only upon exiting the Geometry module or if "Save" is selected.
- the area displayed in the properties table for a Joist is used only to calculate self-weight. The area is assumed zero when formulating the stiffness matrix and hence Joists cannot transmit axial force.
- The Steel postprocessor can select and design Joists in a manner identical to that used for regular steel sections. Refer to Steel postprocessor - Joists.

3.5.6.1.5 Define by dimensions

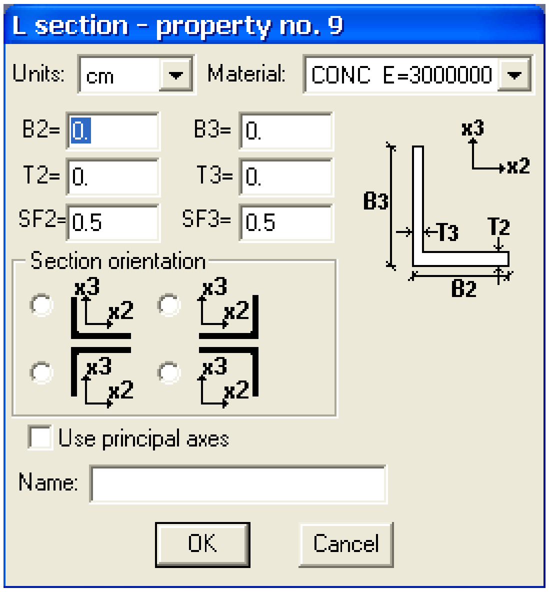


The program then prompts for the dimensions, material and section orientation.

- ***All sections***



- ***L-Sections***

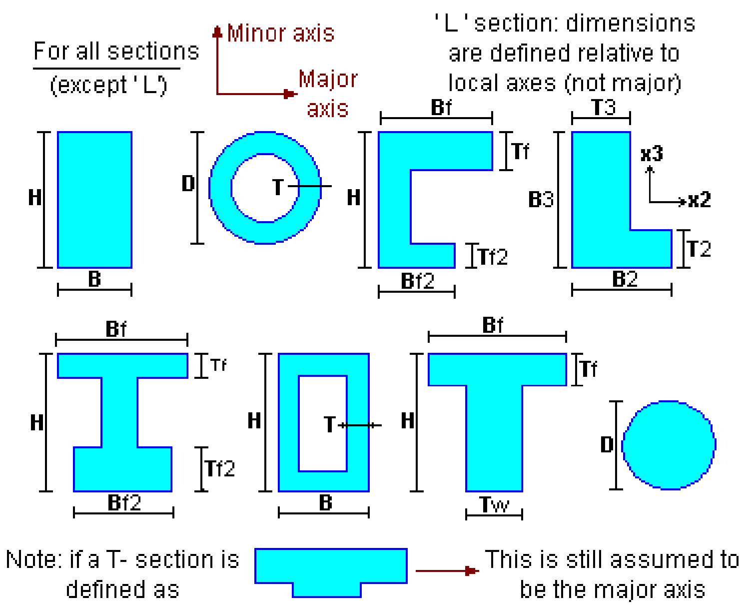


Note:

- The section orientation selected here is the default for this property group. The orientation may be revised for individual beams using the Local axes - flange 272 option.
- the section name is automatically updated as the dimensions are typed in.
- a table of section names can be displayed in the Additional options - property 251 names option

Dimensions

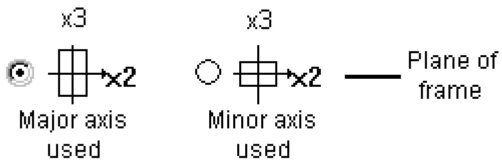
Enter the section dimensions in the current units, as follows: (the units may be revised for each property)



Major axis direction

The section may be aligned so that its major axis resists either the M2 or M3 moments, i.e the major axis is aligned with the beam x2 or x3 axis (remember that the "major" axis is not necessarily the one with the larger moment-of-inertia - refer to dimensions 228).

Define the major axis orientation with respect to the local x2 axis; the user must specify whether:



Section orientation is most easily checked by selecting **Section orientation** in the Display option on the Menu Bar.

L-section orientation

Although not important for analysis (the moment-of-inertia does not change if the flange location is inverted), the defined orientation will be displayed in the rendered drawing and will be used as the default orientation by the steel postprocessor, etc.

The option is available for space models only

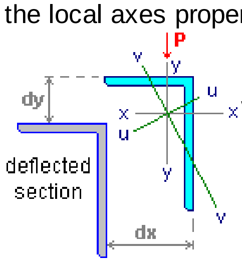
Note:

- The section orientation selected here is the default for this property group. The orientation may be revised for individual beams using the Local axes - flange 272 option.

Use principal axes

For doubly unsymmetric sections:

When the principal axes of a section are not aligned with the local (geometric) axes, e.g. a single angle, a more refined analysis may be carried out by using the principal axes properties (Iu, Iv, etc) instead of the local axes properties (Ix, Iy, etc).

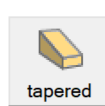


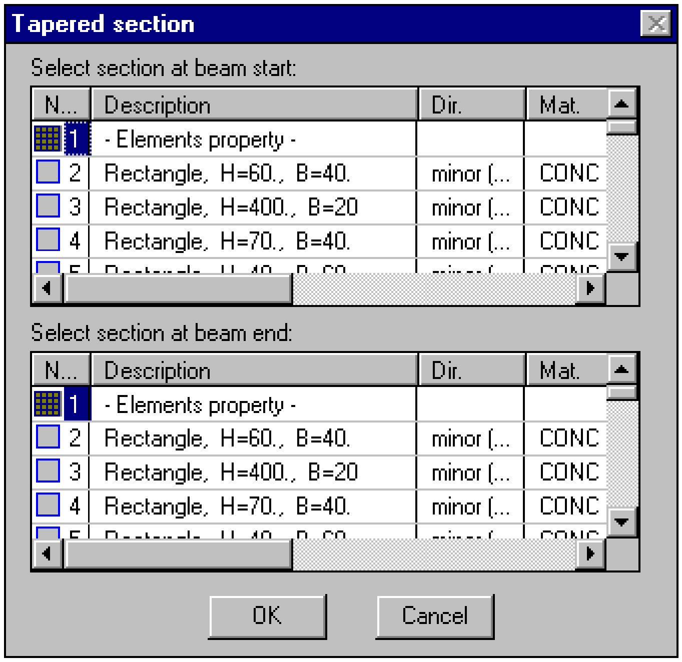
When principal axis analysis is specified, a load applied in the direction of one local axis causes the section to deflect in both local axes directions. The program internally resolves the load to its principal axis components when solving the model. Note that all results will be displayed relative to the local axes.

3.5.6.1.6 *Tapered section*

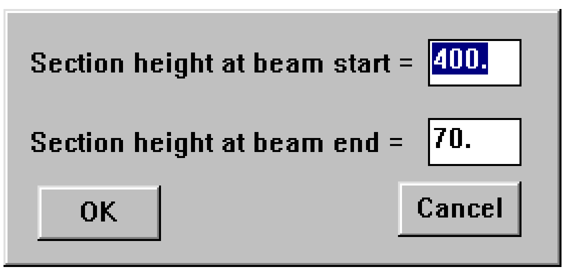
A tapered section is created by defining the section properties at the beam start and beam end using any of the methods: by properties, 223 steel sections 224 or by specifying section dimensions 227.

To define a tapered section:

- define the section properties at the beam start and beam end (if not already defined) using any of the methods: by properties, 223 steel sections 224 or by specifying section dimensions 227.
- select **Define/revise** a property and click the  icon
- Select the properties at the beam start and beam end:

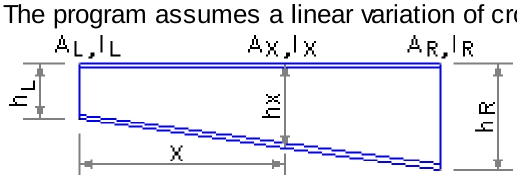


- Define the height at the ends. (if you defined the properties by section dimensions, the height dimensions will appear as the default values).



Note:

- The program assumes a linear variation of cross section along the beam:



the program assumes that the variation of A (area) and I (moment-of-inertia) are:

$$A_x = A_L \left(\frac{h_x}{h_L}\right)^a \qquad I_x = I_L \left(\frac{h_x}{h_L}\right)^b$$

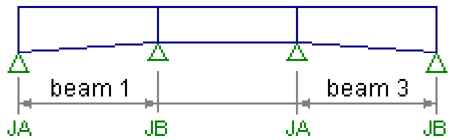
where:

a,b = the exponents which satisfy

$$A_R = A_L \left(\frac{h_R}{h_L}\right)^a \qquad I_R = I_L \left(\frac{h_R}{h_L}\right)^b$$

- the Steel Postprocessor can "check" only tapered sections defined by ***section dimensions***, i.e. enter the dimensions of the flange/web of the section at both ends. Tapered "steel sections" cannot be checked by the postprocessor.

- Tapered properties may be assigned to individual beams or a line of beams. Refer to Assign a property group 248.
- For the beam shown in the figure below, different property groups must be defined for the identical beams 1 and 3 because the "start" and "end" are interchanged.



- "height" values are used only to calculate the exponents "a" and "b". Therefore the end width dimensions may be entered in the "height" dialog boxes for beams with a tapered width.

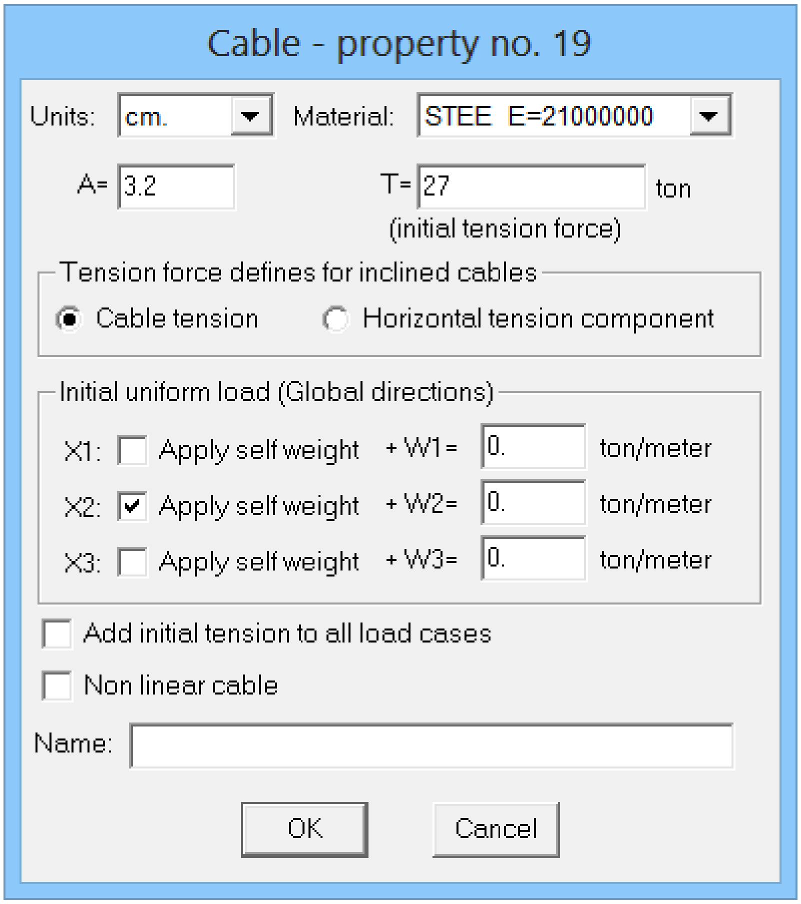
3.5.6.1.7 *Cable element*

A cable can be modeled as either a linear element or a non-linear element.

- linear element 234:
  a simplified element that reduces the axial stiffness to model the initial sag in the cable, i.e. the increased axial deflection represents the taking up of the initial slack in the cable. This option is suitable for models where the loading causes only a minor change in the cable axial force (< 15%).
- non-linear element 234:
  The exact calculation of the tension force in a cable element is essentially a non-linear problem. The program uses an iterative method to achieve equilibrium at the support nodes of the cable elements.

Note:
- only uniform loads and axial temperature loads can be applied to cables; point loads must be applied to intermediate nodes.
- non-linear cables can be defined with intermediate nodes; linear cables with intermediate loads are singular.
- the dynamic and Bridge design modules convert non-linear cables to linear cables. Therefore the cables in models where either these two modules is to be used should not have intermediate nodes.

Define the cable element parameters:



Cable properties

Units = units for cable area only.
A = cable section area
Material = the material selected must have a value for density so that the program may calculate the self weight.

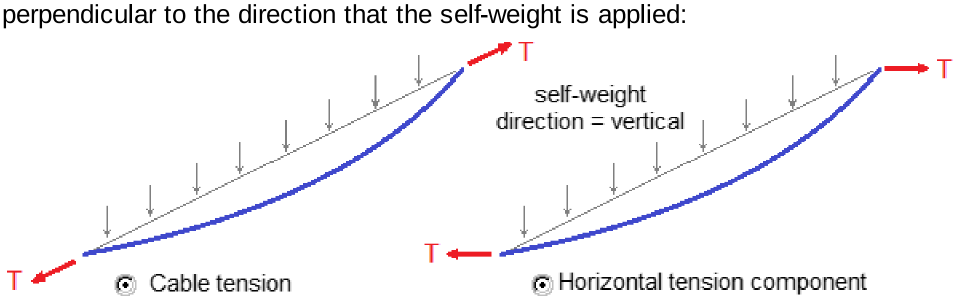
Note:

- for existing cables with intermediate nodes: the initial cable geometry must be updated whenever the cable properties are revised. Refer to Move nodes - update cable nodes 193

Cable Tension

T = the initial cable tension (defined in default force units)

T can be either the cable tension or its horizontal component, where "horizontal" is the direction perpendicular to the direction that the self-weight is applied:



Note:
- for existing cables with intermediate nodes: the initial cable geometry must be updated whenever the cable tension is revised. Refer to Move nodes - update cable nodes 193

Initial uniform load

The initial uniform load defines the initial geometry of the cable. The load must be uniform and consist of two components:
- self-weight - calculated by the program from the section area and the material density
- an additional uniform load. The load can be applied in any global direction.

The initial cable geometry (calculated by the program) can be displayed using the Display - section orientation 55 option

Note:
- a positive load acts in the ***negative direction*** of the global axis
- a cable without initial loads defined here will always remain straight, i.e. it will have a stiffness of **EA/L**.

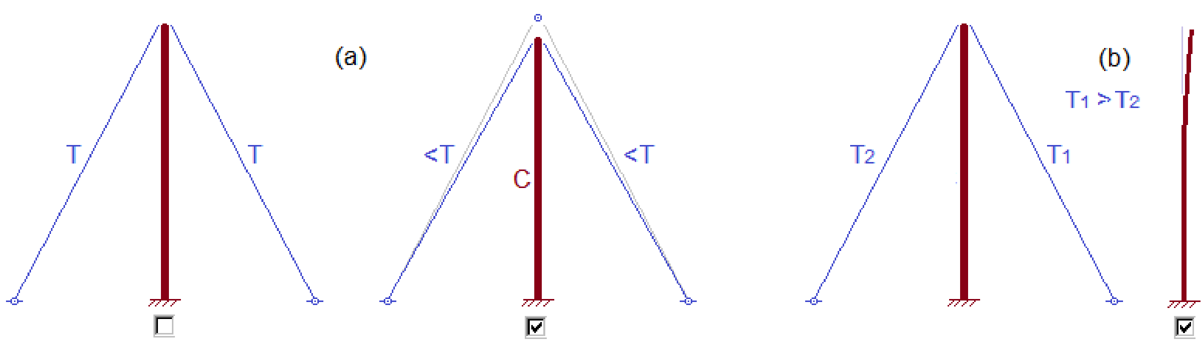
- for existing cables with intermediate nodes: the initial cable geometry must be updated whenever the uniform load is revised. Refer to Move nodes - update cable nodes 193
- for cables with intermediate nodes: if the initial sagged cable geometry is input by the user, there is no need to apply the self-weight (this will speed up the solution).

Add initial tension to all load cases

☐ the initial tension force is not applied to the end nodes
☑ the initial tension is applied

In example (a), **T** is applied to the two cables.
- if ☐ is selected, no compression force is generated in the vertical member and this force will not appear in the member results.
- if ☑ is selected, a force **C** is generated in the vertical member, but the deflection of the top nodes reduces the tension force in the cables.

Refer to non-linear cables for an example that shows how to solve this problem (maintain a tension force **T *and*** generate a compression force **C**) is



In complicated cable models, it may be difficult to create a model that is initially in equilibrium. A simple example is shown in (b), where there is no equilibrium at the top node. Setting this option to ☑ will move the tower to the right, reducing T_1 and increasing T_2 until equilibrium is reached.

Linear/non-linear

Refer to:
- linear 234
- non-linear 234

for detailed instructions and examples.

3.5.6.1.7.1 Cable - linear

This option models a cable as a linear element. The solution is approximate as a non-linear analysis is required for a complete solution. There are several simplifying assumptions:

- The stiffness of the element is provided by the **initial** tension force, i.e. the program does not recalculate the stiffness due to the change in the axial tension resulting from the applied loads.
- the straightening of the cable due to the axial force is linear
- the effect of transverse beam loads is ignored for the straightening and deflection of the cable (moments, shear, etc resulting from these loads are calculated as for regular beams).

Note:

- Linear cable elements should be defined as tension-only 255 to prevent compression forces in the results.
- the axial force results include the initial tension force

The program calculates the cable stiffness as:

$$K=\frac{EA}{L}\left(\frac{1}{1+\frac{w^2L^2EA}{12\,T^3}}\right)$$

where:

w = self weight of the beam (see note below)
E = modulus of elasticity
L = length of beam
A = section area
T = initial tension in the beam

The stiffness is always less than the axial stiffness of a regular beam element = EA/L. This reduced stiffness implies that the applied tension force in the cable (from the frame action) serves two purposes:

- straightening of the sag in the cable
- elastic lengthening of the cable

If the initial tension in the cable is large, the initial sag is small and hence a smaller portion of the applied tension force is used to straighten the cable. It is obvious that as the initial tension F increases, the stiffness approaches that of a regular beam element, i.e. K=EA/L. The stiffness decreases as F decreases, resulting in larger deflections.

3.5.6.1.7.2 Cable - non-linear

The exact calculation of the tension force in a cable element is essentially a non-linear problem. The program uses an iterative method to achieve equilibrium at the support nodes of the cable elements.

Note:

- only uniform loads can be applied to cables; point loads must be applied to intermediate nodes.
- non-linear cables can be defined with intermediate nodes, however they are required only when point loads or varying uniform loads must be applied, or when the cable is intersected by other cables or beams along its length.

For cables with intermediate nodes:

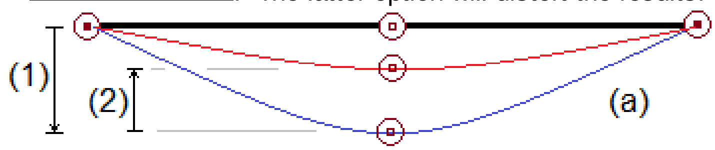
- existing cables with intermediate nodes: the initial cable geometry must be updated whenever the cable properties, initial tension or uniform load is revised. Refer to Move nodes - update cable nodes 193
- if the initial sagged cable geometry is input by the user, there is no need to apply the self-weight (this

will speed up the solution).

- when viewing deflection results, the 

option should be used and not the 

. The latter option will distort the results:



In drawing (a)
- (1) is the initial sagged cable deflection at the intermediate node.
- (2) if the final deflection (additional tension straightens the cable)

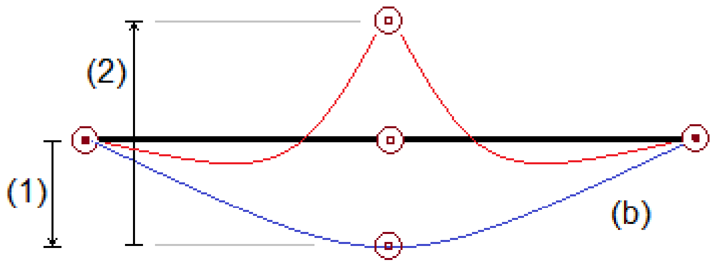
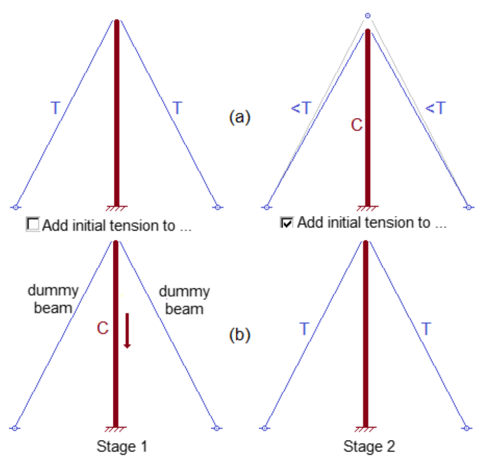


Figure (b) shows the results that will be displayed using 

:
- The initial intermediate node location (1) is displayed at its correct location
- the upwards deflection (2) is displayed according to the user defined "maximum result" dimension; this will distort (2) relative to (1), as shown.

In complicated cable models, geometry "stages" are usually required to model the tensioning sequence and to include the full effect of the cable tensioning in connected beams and elements. The following simple example illustrates the problem and the solution:

In example (a), **T** is applied to the two cables.
- if ☐ **Add initial tension to all load cases** is selected, no compression force is generated in the vertical member and this force will not appear in the member results.
- if ☑ is selected, a force **C** is generated in the vertical member, but the resulting deflection of the top node reduces the tension force in the cables.



Use geometry "stages" to maintain the initial tension **T** while simultaneously generating the compression in the vertical member, as shown in (b):

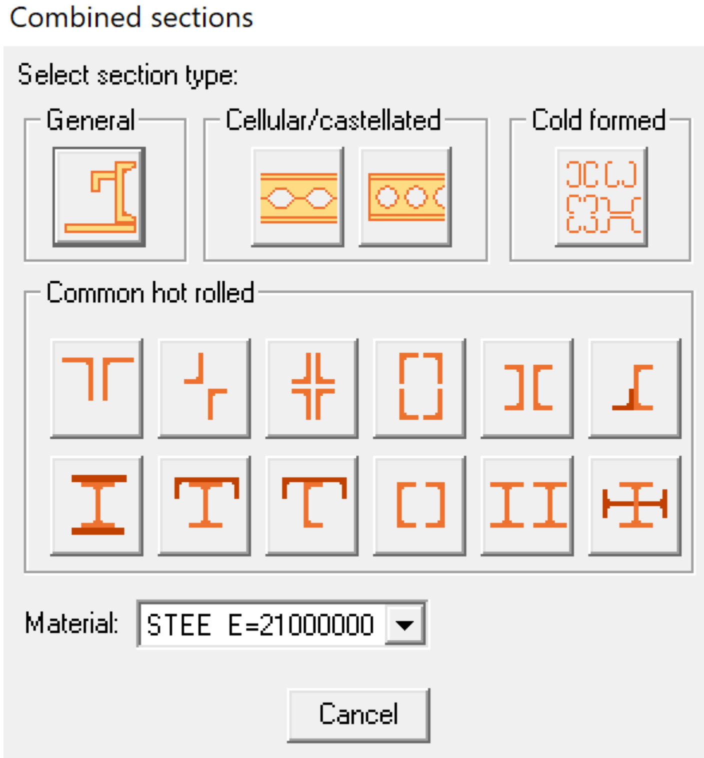
- Stage 1:
  define the cables as "dummy beams"; apply the force **C** as a joint load at the top, or a beam load in the cables, etc.
- Stage 2:
  assign the cable property, including the initial tension force **T**, to both cables.

Three stages should be used if the cables are not tensioned simultaneously.

In general, Stages can be used to model any tensioning sequence in complicated models.

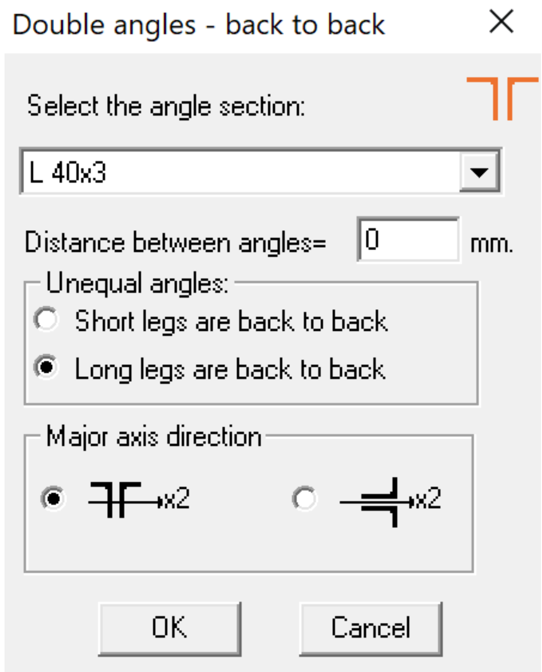
3.5.6.1.8 Combined beam sections

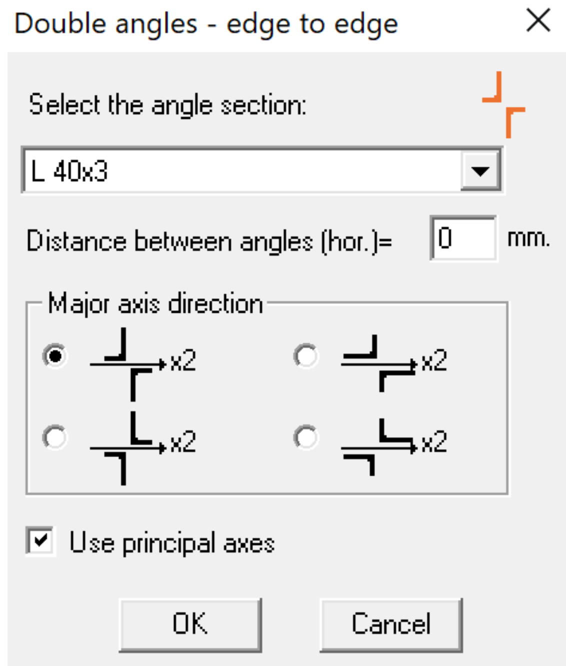
Select steel sections that are made of two or more standard fabricated steel sections:



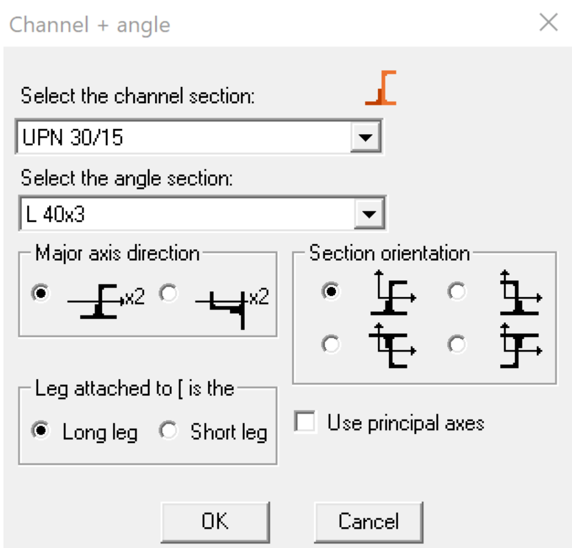
Common hot rolled

- ***Typical section***
- ***Two angles***

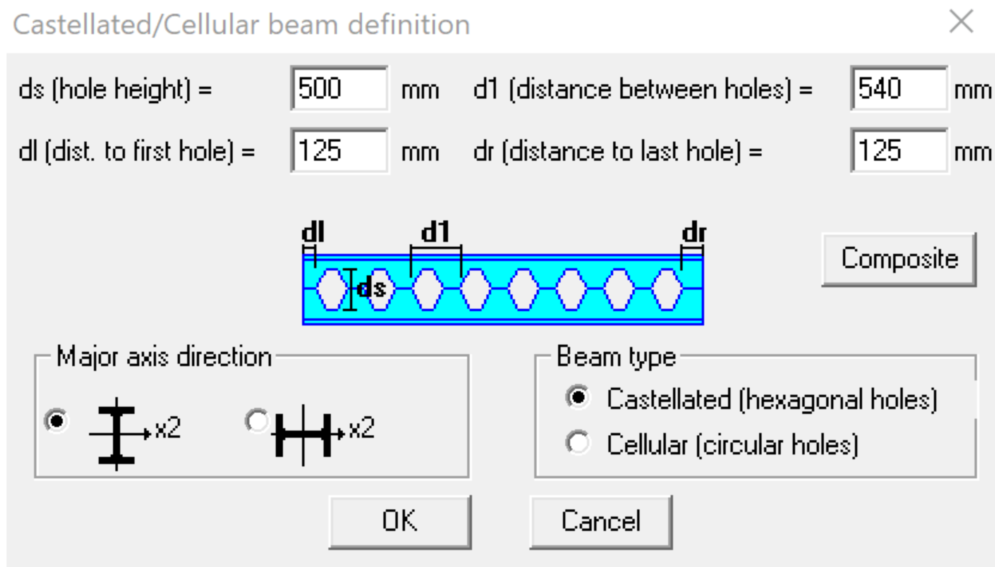




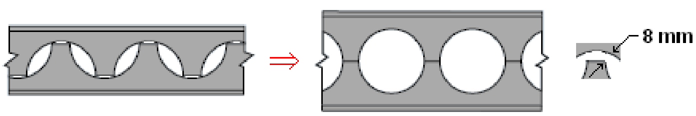
- ***Channel + angle***



Castellated/cellular beams



- the program uses the moment-of-inertia ***at the hole*** to calculate the stiffness of the beam for the analysis
- the program uses an average area for the analysis
- the dimension **dr** is a minimum value; the program stops adding holes before this dimension is reached.
- cellular beams: the program accepts only values of **ds** and **d1** that are feasible:



The program calculates the new section height based on an 8 mm cutting gap; for other dimensions define a fabricated cellular beam.

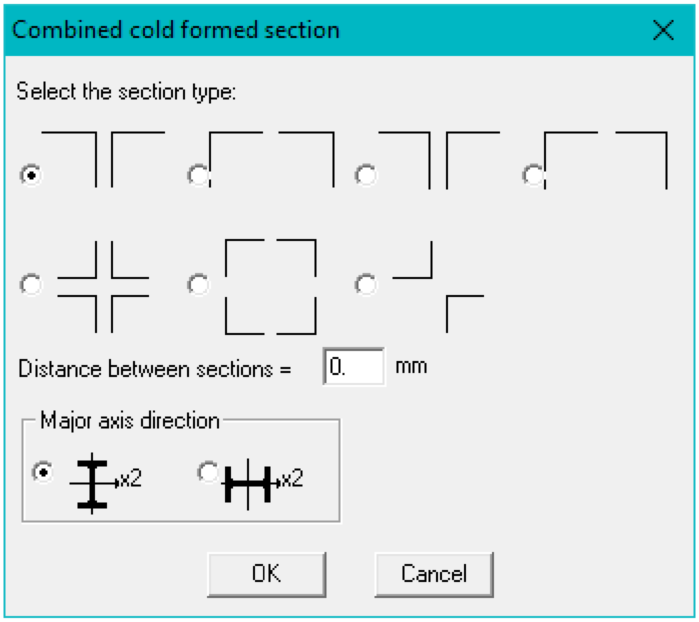
- ***Fabricated cellular beams***

The program designs these beams as 'built-up' (welded) sections.

Combined cold-formed sections

If the two section are defined with zero spacing between them and form a closed section, the program assumes that the section is not closed when calculating the section properties, i.e. the torsional moment-of-inertia, J, is calculated as ΣJ_i of the individual sections. If the exact value of J is required, use the *CROSEC* section generator program to create the section.

For all combined sections:
- select the basic section type from the list, e.g the single angle, channel, etc, that forms the section.
- specify the combined section shape. For example, double angles:



For all sections:

Select section / attach legs, etc.

Specify the section data:
- select a section from the list box
- specify dimensions, if required,e.g. spacing between sections
- specify additional information, if required, e.g. long or short legs back-to-back, etc.

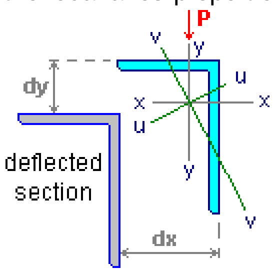
Major axis direction

Specify the orientation of the section major axis relative to the beam local axes.

Principal axes

For doubly unsymmetric sections:

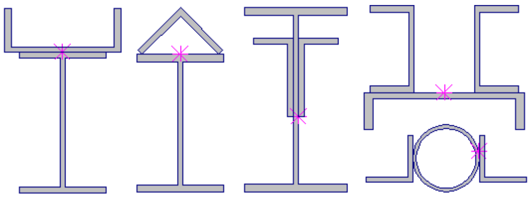
When the principal axes of a section are not aligned with the local (geometric) axes, e.g. a single angle, a more refined analysis may be carried out by using the principal axes properties (Iu, Iv, etc) instead of the local axes properties (Ix, Iy, etc).



When principal axis analysis is specified, a load applied in the direction of one local axis causes the section to deflect in both local axes directions. The program internally resolves the load to its principal axis components when solving the model. Note that all results will be displayed relative to the local axes.

3.5.6.1.8.1 General combined sections

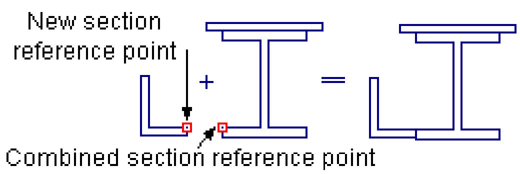
Use this option to create a combined section consisting of any number of steel elements - plates or rolled sections. For example:



To display a demo video that explains how to define general combined sections:

- *click on*  *to start the video*
- *then click on*  *to enlarge the display.*

- "Reference points" are selected on the new subsection and the current combined section; the program combines the sections at the respective reference points. For example:

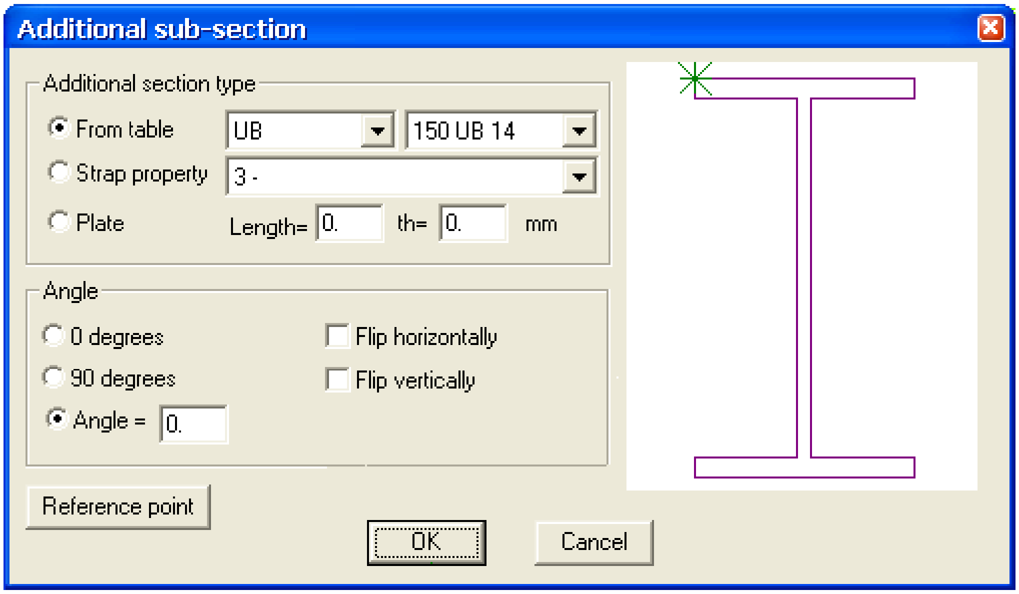


- Subsections may be rolled sections, plates or existing *STRAP* property steel sections
- All sections may be rotated to any angle or "flipped" - horizontally or vertically.

Refer to:
- General combined - menus 240
- General combined - example 243

There are three menus involved in creating a combined section:
- select subsection 240, orientation and connection point
- correct subsection 241 location (displayed starting with the second subsection)
- main menu 242

Select subsection

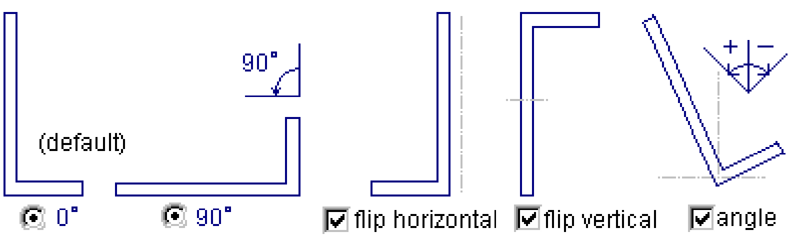


Section type:

Select on of the following:

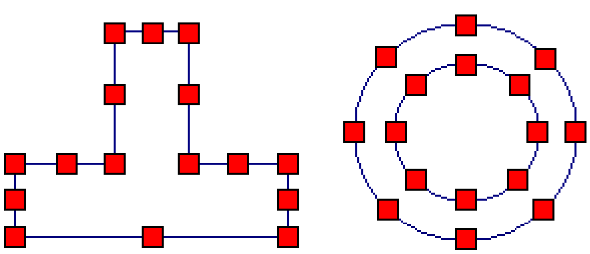
- any steel section from the current table
- any existing property group that is a steel section
- define a plate

Angle:

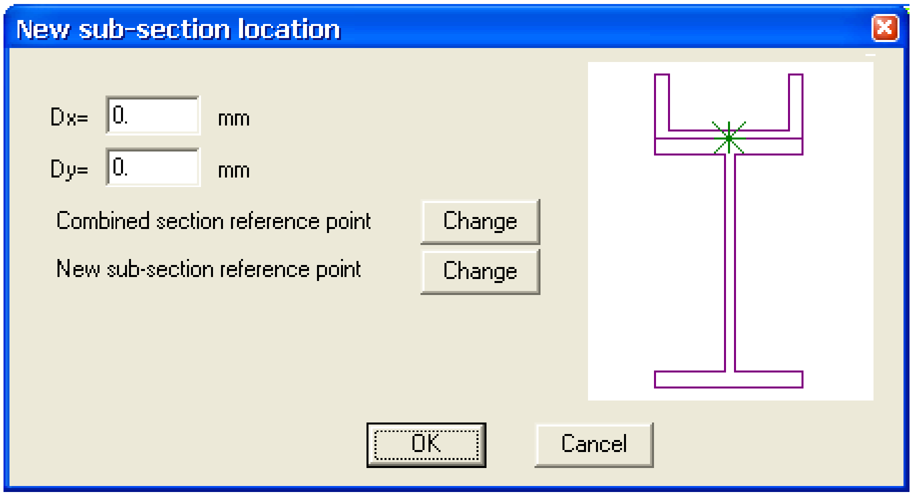


Reference point:

The reference point can be selected at any corner or the mid point of any line (except for round sections). For example:

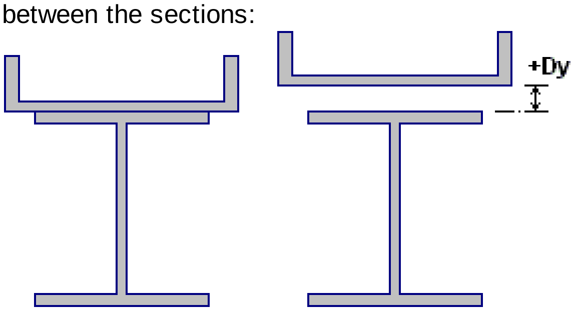


Correct location



Dx, Dy :

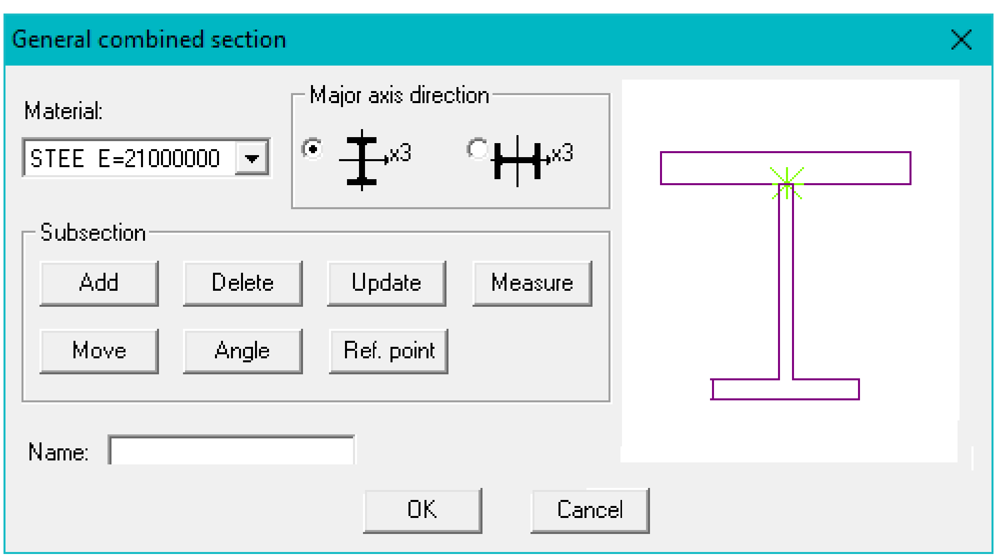
Move the new subsection relative to the reference point. For example this allows a gap to be defined between the sections:



Combined/new section reference point:

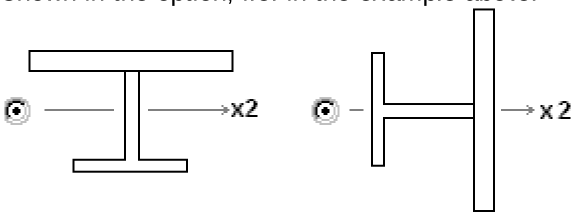
Select new reference points for the current combined section and the subsection just added or the subsection selected.

Main menu



Major axis direction:

The ⦿ I section represents the section ***as displayed*** and the section will align with the x2 axis as shown in the option, i.e. in the example above:

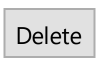


Note that the in unsymmetric sections the larger side is always placed in the positive axis direction, i.e. the section would be oriented in the model in the same way even if the section were defined with the larger flange at the bottom.

Use the Local axes 272 option to modify this default orientation.

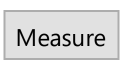
Subsection:

Add a new subsection (first click "Ref. point" to select the point where the next subjection will be joined to it). The Select subsection 240 menu is displayed.

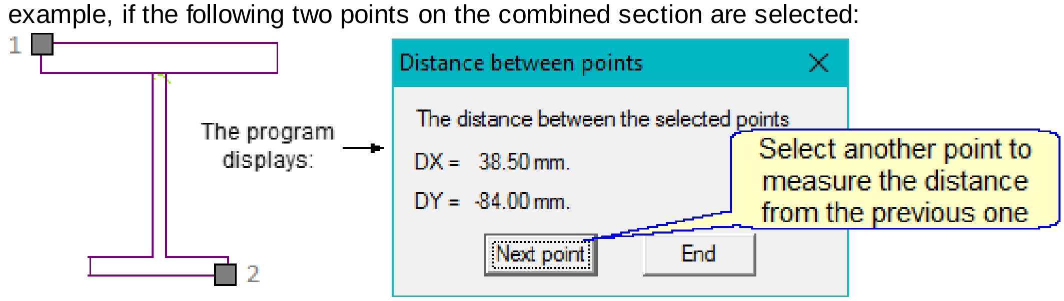


Delete any subsection.

Revise the details of any subsection; the Select subsection 240 menu is displayed with the existing data for the selected subsection.

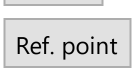


Measure the distance between any two corner points on the combined section. For example, if the following two points on the combined section are selected:



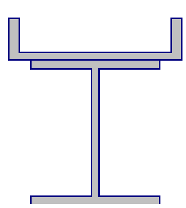
Move any subsection; the Correct location 241 menu is displayed with the existing data for the selected subsection.

Revise the angle/flip for theselected subsection (can also be revised by selecting "Update").

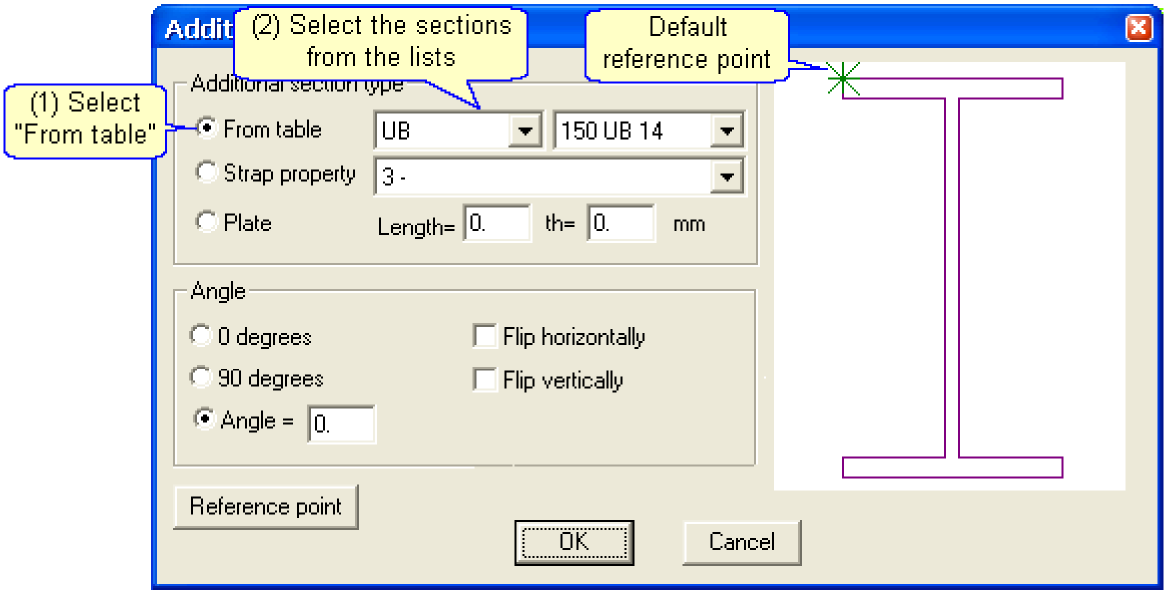


Select the reference point in the combined section where the next subsection will be added.

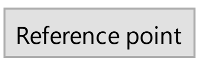
Create the following combined section consisting of an I-section and a channel attached to its top flange:

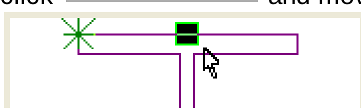


- select the first section:

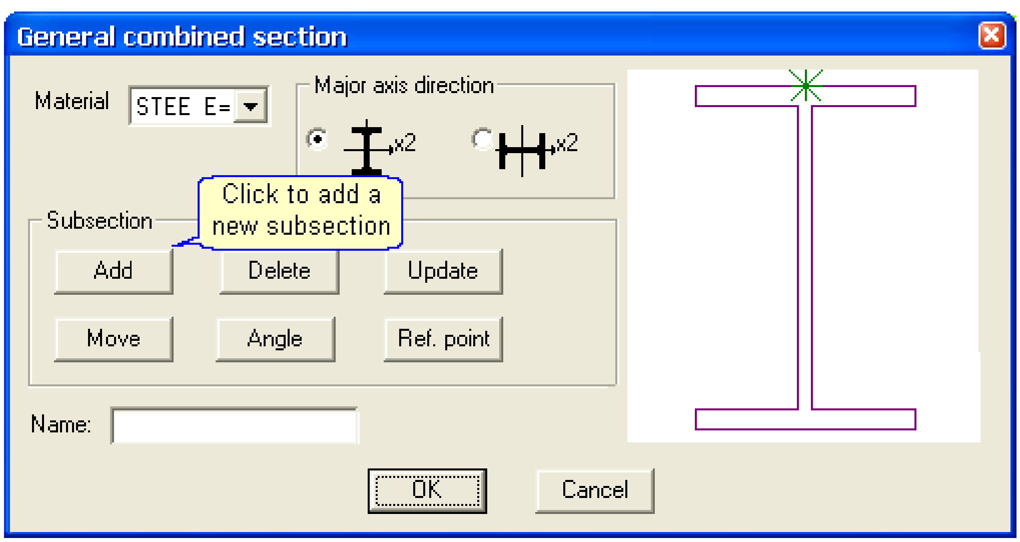


Each section has a "reference point" denoted on the section by a ✳. This point defines where the reference point of the next subsection will be joined. In our example, the reference must be at the centre of the top flange.

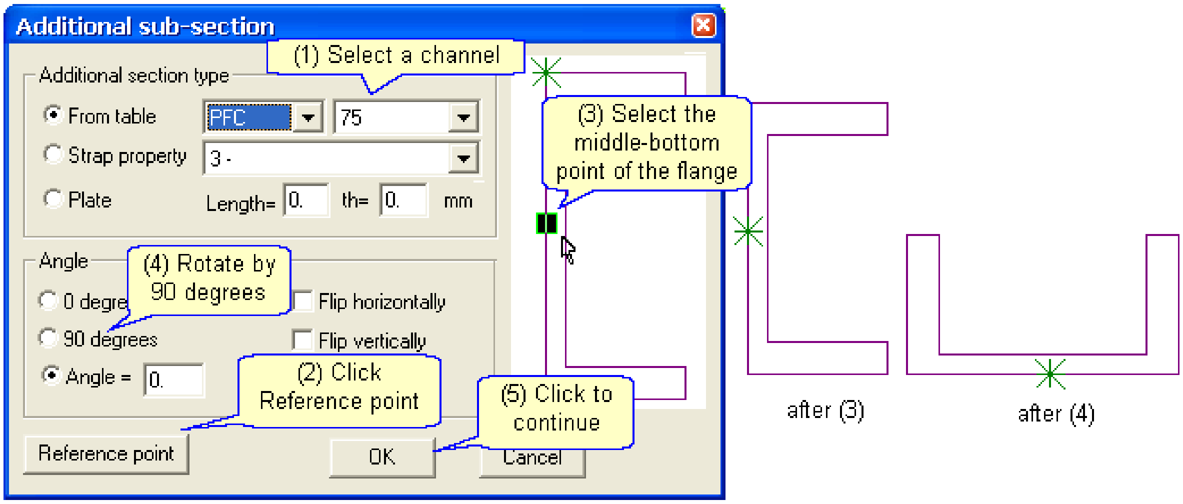
- click  and move the cursor to the center of the flange and click the mouse:



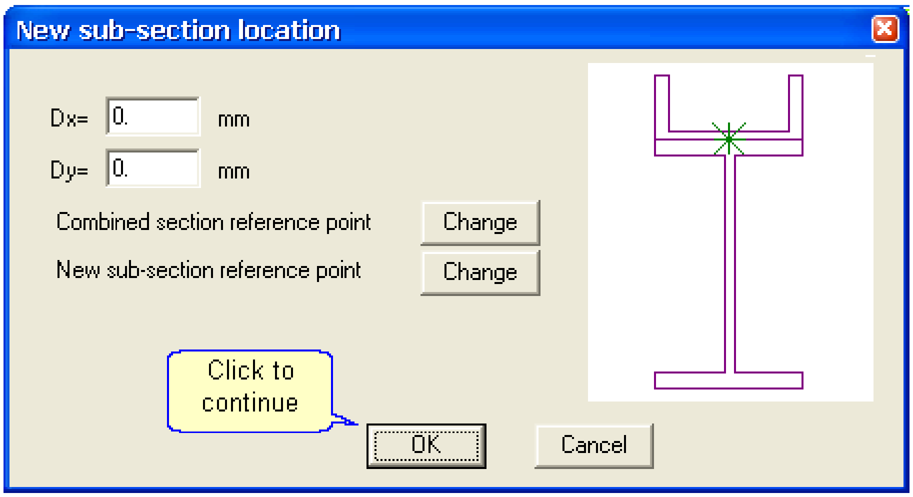
- Add the channel:



- select the channel, select its reference point and rotate it: :



- The program displays the combined section; no corrections are necessary:



3.5.6.1.9 Paste properties

Sections in the 'clipboard' may be pasted into a *STRAP* property group. To copy a section into the clipboard:

- display the model Property list and click Copy

or:

- create and copy a section from *STRAP* Section generator (*CROSEC*):
  - Select **Section generator** in the **File** menu
  - Create the solid section
  - Select **Copy to Clipboard** in the **Output** menu
  - Select **Exit** in the **File** menu to return to the *STRAP* geometry (or press Alt-Tab to toggle back to *STRAP* without closing the section generator).

To paste the sections:

- click and highlight a section in the Property list
- click Define/Revise

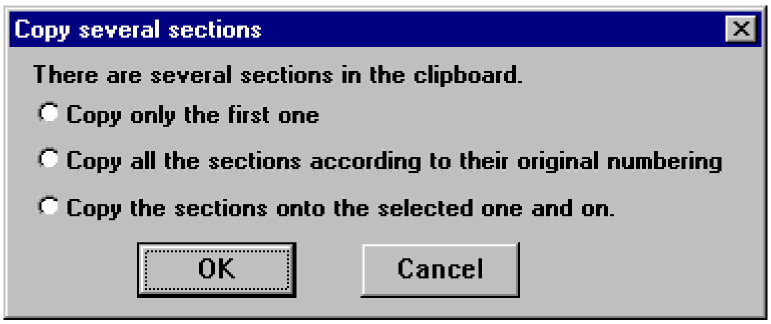
- Click the 

icon

Note:
- All sections copied from the *STRAP* section generator are defined by properties, i.e. (I,A, etc are copied). The exact section shape is copied and is displayed by the rendering option.

- If more than one section was copied to the clipboard (using the 

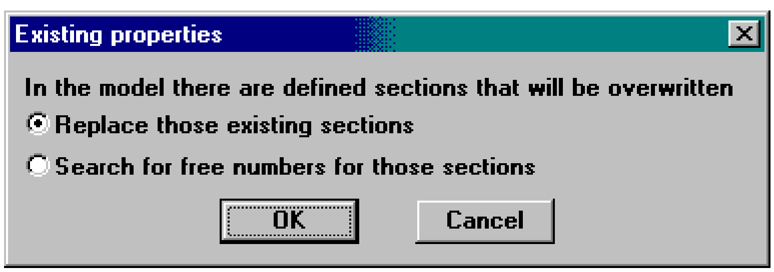
option) -



Copy only the first one
The first section is pasted into the selected property.

Copy all the sections according ...
The property numbers are maintained; for example, if sections 7,9,10 were copied to the clipboard from another model, they are pasted into the same property groups in the current model.

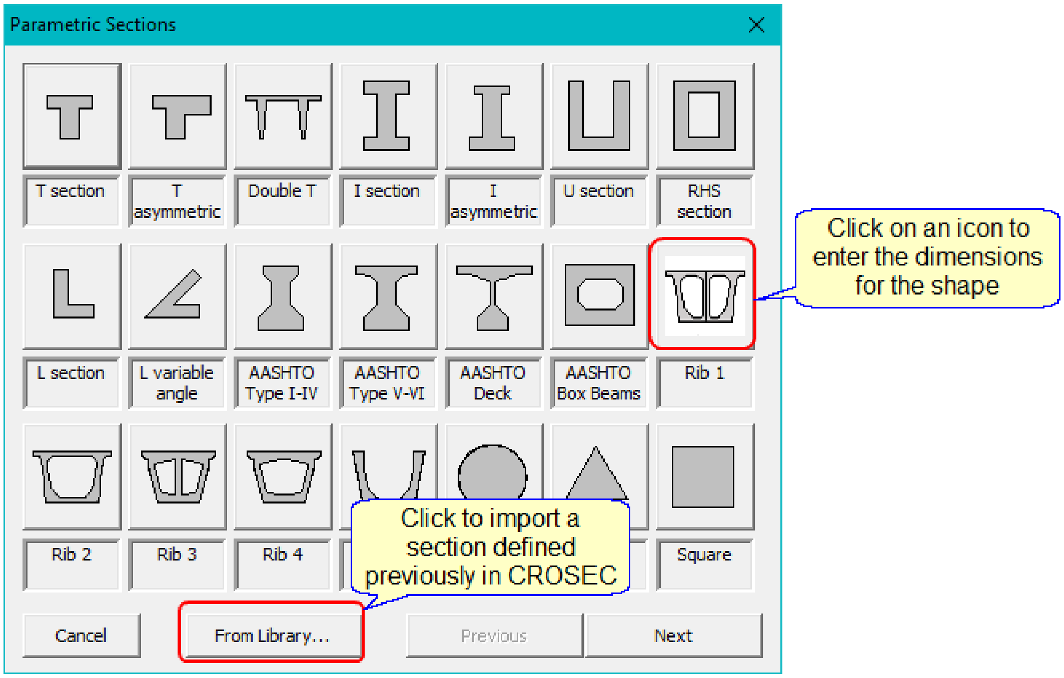
Copy the sections onto the selected one and on
The first section is pasted into the selected property group and the other sections are pasted into the following property groups or empty property groups. For example, sections 7,9,10 were copied to the clipboard from another model and property group 5 is selected in the current model; the following options are available:



- **Replace ..**
  The sections are pasted into property groups 5,6,7
- **Search ..**
  The first section is pasted into property groups 5 and the others into the first two **-Not used-** groups in the table

3.5.6.1.10 More (parametric section)

Predefined solid sections may be defined by entering the dimensions or selecting a section defined and saved in the *CROSEC* program:
- select a section by clicking on an icon:

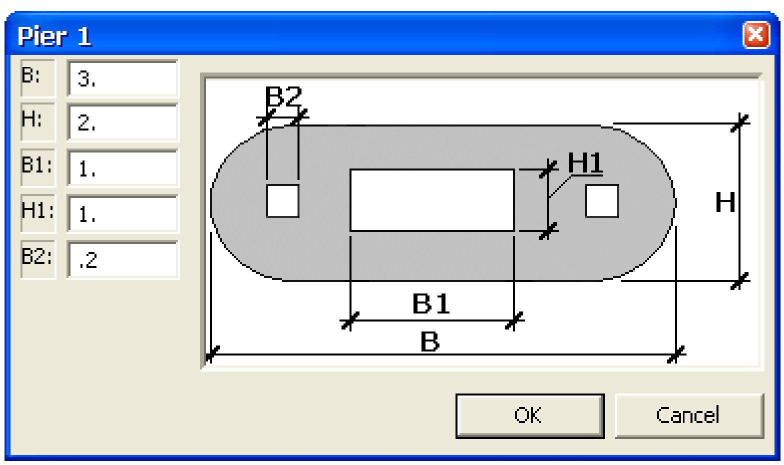


Previous and Next are active if more section screens are available.

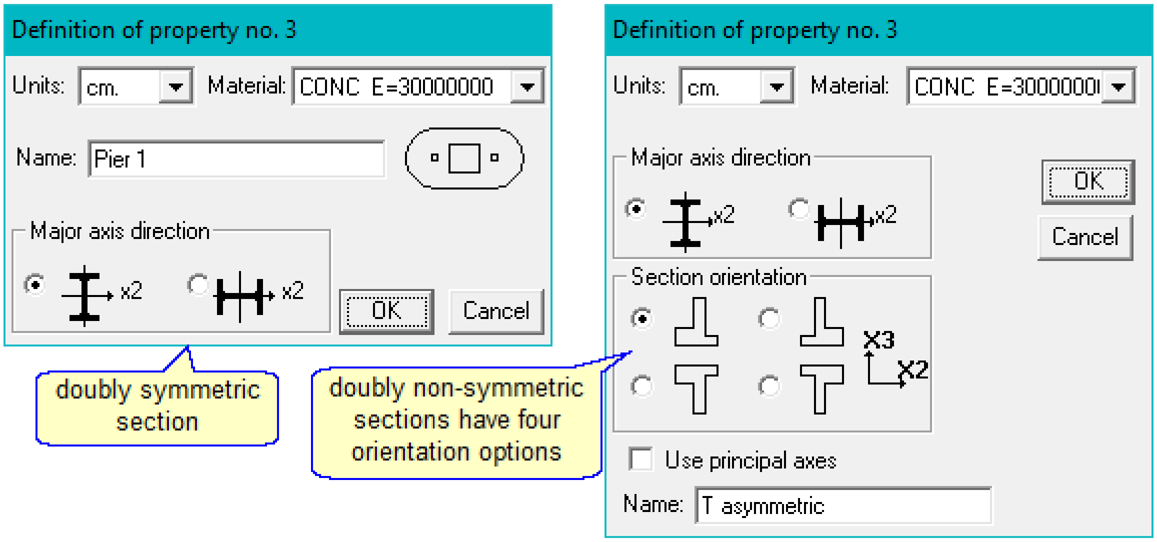
Note:
- this option is available only when the *CROSEC* program is installed in the *STRAP* program folder.
- New section types may be added. Please contact your *STRAP* dealer.

Define by dimensions

- enter the dimensions according to the diagram. For example:



then select the major axis direction and the section orientation (if necessary):

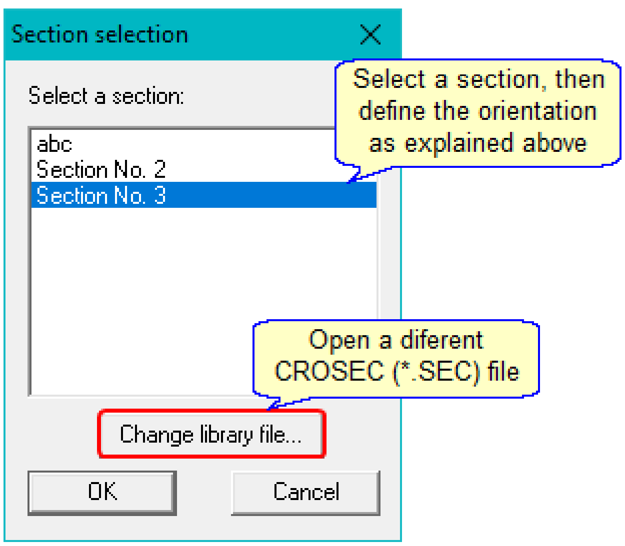


Use principal axes:

Refer to Define by dimensions 229.

From a Library

The program displays a list of sections in the last file saved by *CROSEC*:



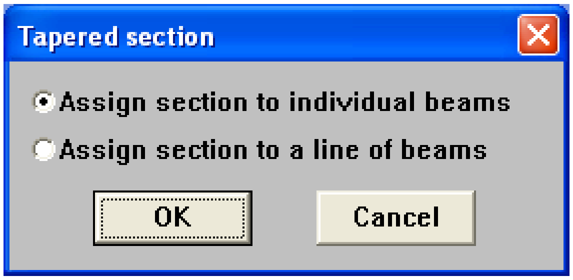
3.5.6.2 Assign

Assign beams to a property group:

- Select a property group from the list that you want to assign to beams
- Select the beams that this property is to be assigned to using the standard beam selection option.

Note that an **-Undefined-** property group may be assigned to beams; the section properties may be defined later.

For "tapered" sections:



Assign sections to individual beams

Select the beams that this property is to be assigned to using the standard beam selection option.

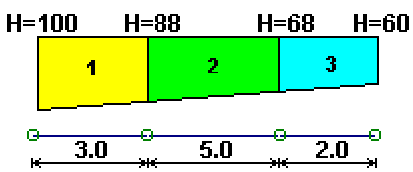
Assign sections to lines of beams

Select the start and end nodes of a line of beams. The program will create a new property group for each beam along the line, the start and end heights calculated proportionally to the distance along the line.

Note:

- If the start/end properties were defined by Dimensions with the same section type, all new properties will have the same section type (all dimensions will be tapered)
- If the start/end property is an I-section defined by Dimensions and the other property is a steel I-section (from a Table), all new properties will be I-section defined by Dimensions (all dimensions will be tapered)
- For all other cases, the new section groups will be defined by Properties (I,A)

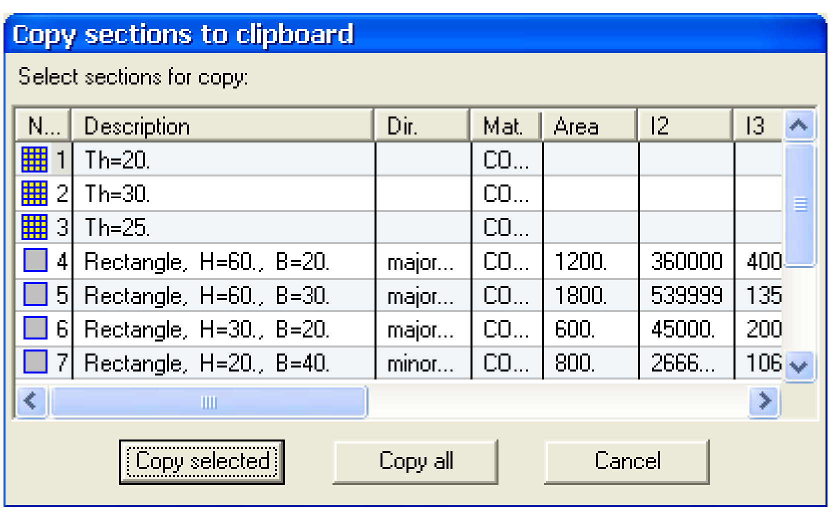
For example, a tapered beam with end heights =100/60 is assigned to a line with three beams:



Three new properties are created.

3.5.6.3 Copy

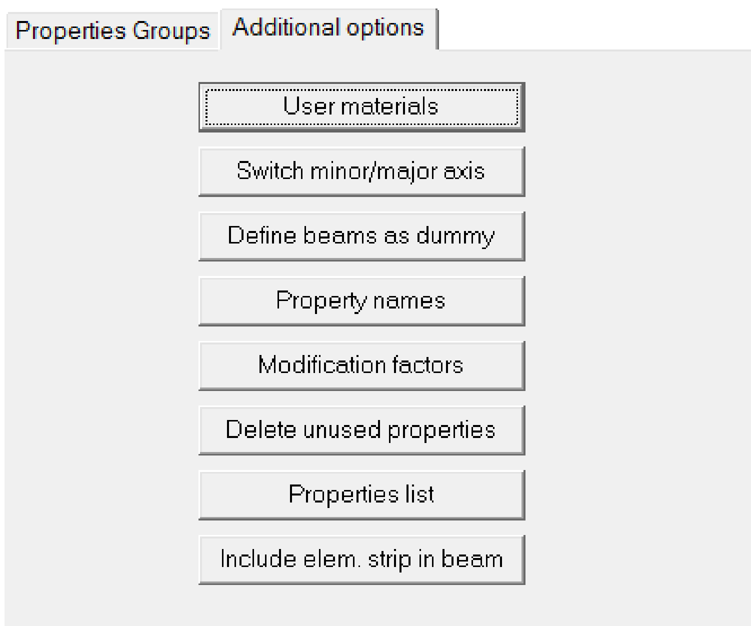
Copy beam properties to the clipboard; the properties can then be retrieved into the current model or any other model.



- click and highlight the properties and click Copy selected
- to copy the entire table to the clipboard, click Copy all

To retrieve the properties into a model, select Beam properties Define/revise, then click on the paste option.

3.5.6.4 Additional options



Define user material

Refer to User-defined material 252.

Switch major/minor

Use this option to interchange the major/minor axes of a section, i.e. to rotate it by 90°.

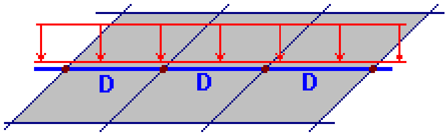
Select the beams with the sections to be rotated using the standard beam selection option.

Note:
- if all beams assigned to the property group are selected, the program revises the property group definition.
- if some of the beams assigned to the group are selected, the program creates a new section group with the rotated properties and automatically assigns the selected beams to the new group.

Define beams as dummy

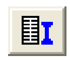
A beam may be designated as "Dummy". Dummy beams may be loaded but they do not affect the stiffness of the model and will not appear in the output tables. Beam loads (including global loads) defined on a dummy beam are applied to the model identically to the loads on a regular beam

For example, use a Dummy beam to define a linear load in a model that consists entirely of finite elements.



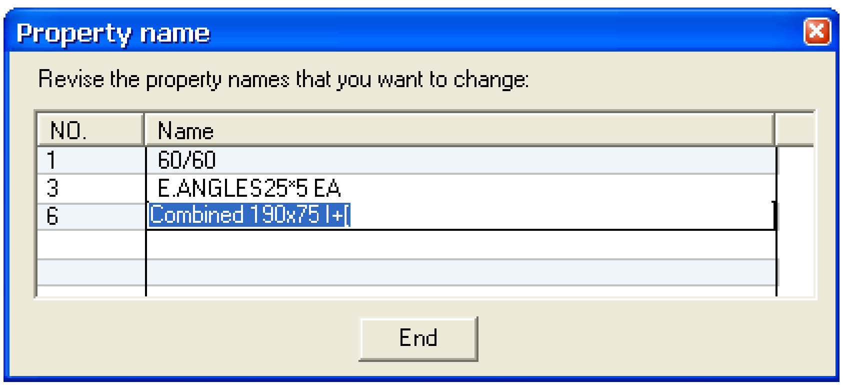
Specify beams as Dummy beams using the standard Beam selection option.

Note:

- dummy beams should be connected to the MODEL at BOTH ends, i.e. dummy beams should not be cantilevered or connected to another dummy beam; the loads on the "unconnected" halves of the beam are lost by the program.
- If you select **Property numbers** in the **Display** option, the letter "**D**" will be displayed alongside the dummy beams.
- Dummy beams can also be defined by assigning  **Property 0** when defining the beam

Property names

Edit the property names defined in any of the beam property dialog boxes:



Property name

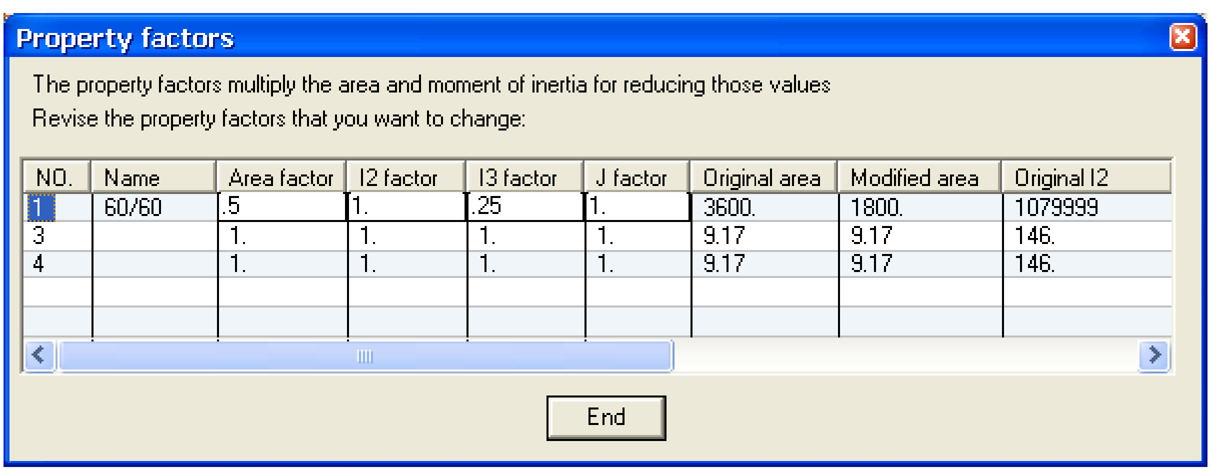
Revise the property names that you want to change:

NO.	Name
1	60/60
3	E.ANGLES25*5 EA
6	Combined 190x75 I+[

End

Modification factors

The area and moments-of-inertia of an property may be reduced by a factor. This option is useful when the cracked section properties of concrete members are required for the analysis. Different factors may be entered for A, I2, I3 and J:



Property factors

The property factors multiply the area and moment of inertia for reducing those values

Revise the property factors that you want to change:

NO.	Name	Area factor	I2 factor	I3 factor	J factor	Original area	Modified area	Original I2
1	60/60	.5	1.	.25	1.	3600.	1800.	1079999
3		1.	1.	1.	1.	9.17	9.17	146.
4		1.	1.	1.	1.	9.17	9.17	146.

End

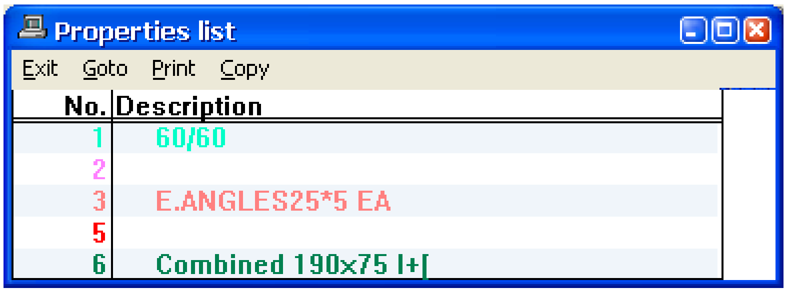
The factors are also displayed in the Output - property table.

Delete unused properties

All properties that are currently not assigned to any beams or elements are deleted from the properties table.

Property list

A property list may be displayed/printed with the property names shown in the same colours as on the screen display:

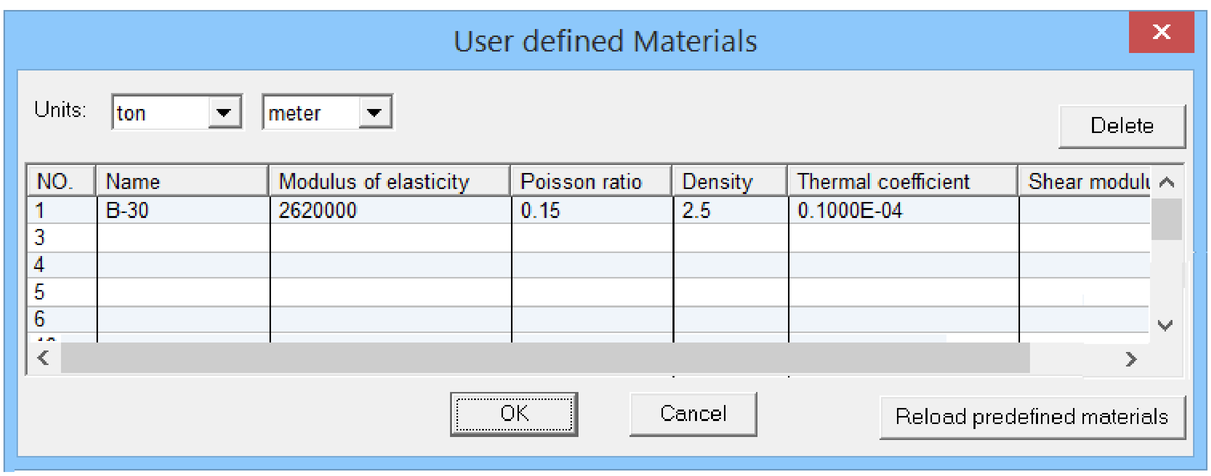


Note:

- element properties are not shown.
- to print the table in colour, set ☑ **Use colour for printing drawings** in Setup - print parameters - general

3.5.6.4.1 User-defined material

Define the properties of a 'user-defined' material. A material defined here is saved for the current model only. To save a user-defined material as a permanent program material 254, select Setup in the ***STRAP*** main menu



Enter:

- the material name; this name will be displayed in the material list whenever a material is selected in the property definition. Type the name of an existing user-defined material to revise its properties.
- the material properties, according to the units selected at the top of the dialog box. Note that large numbers may be entered in exponential format.

Property	Description	Units
Modulus of elasticity	E	force/length^2
Poisson ratio	υ	-
Density	Specific gravity	force/length^3

Thermal coefficient	α	1/°C or 1/°F *
Shear mod. of elasticity	G	force/length^2

* The thermal coefficient may be defined according to either unit. However the temperature difference value entered when a temperature load is applied must be according to the same units.

Note:

- If a zero value is entered for G, the shear modulus of elasticity, the program calculates G from the equation:

$$G = \frac{E}{2(1+\nu)}$$

where ν = Poisson's ratio

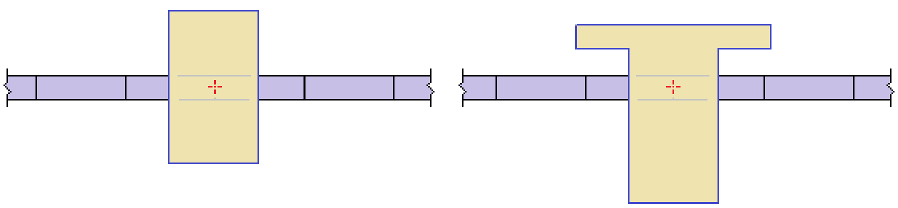
- to delete a material from the table, click and highlight the line in the list; click Delete.

Reload predefined materials

If a permanent program material 254 is revised in Setup (***STRAP*** main menu) after the model geometry is created, select this option to update the material properties accordingly in the current model.

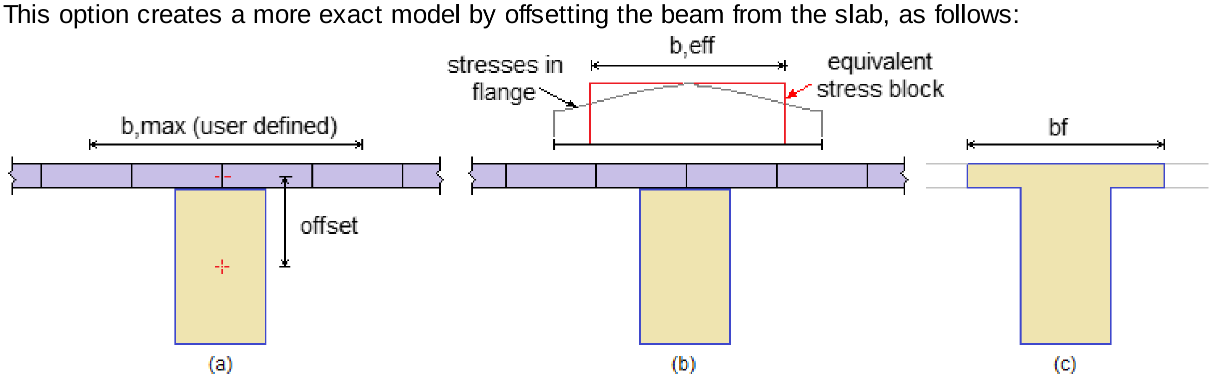
3.5.6.4.2 Add elements to beam

When a beam is defined along a line forming the boundary of slab elements the following models are created:



The center-of-gravity of the beam is aligned with the center-of-gravity of the elements, reducing the overall stiffness of the slab-beam floor system.

This option creates a more exact model by offsetting the beam from the slab, as follows:

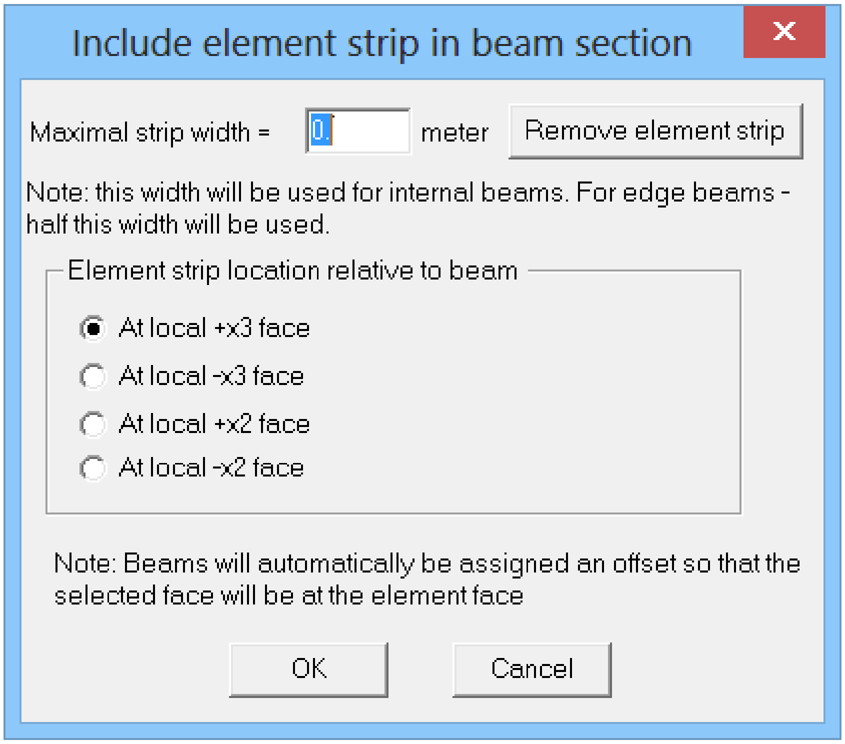


- the user specifies b,max - the maximum permissible effective slab width. This should be approximately the value specified by the code. Refer to Figure (a).
- the program automatically offsets the selected beams and solve the model.
- in "Results", the program calculates the actual stresses in the slab and then calculates an equivalent rectangular stress block with an effective width = b,eff. Refer to Figure (b)
- The program translates the separate beam results and. slab results into a moment (and axial force) acting on the section shown in Figure (c). These moments, shears, etc are the ones displayed in the Results module and the ones used by the Steel and Concrete design modules.
- The beam may have any type with dimensions - Tee, L, I, a steel section, etc, however -

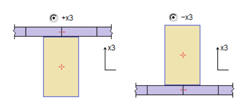
- the Concrete design module recognizes the actual shape but can only design beams and columns with a rectangular section.
- the Steel design module does not automatically recognize these composite sections.

To create the offset:

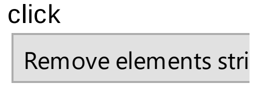
- define the maximum flange width and the flange location relative to the beam:



- defined b,max for the selected elements. Note that the program centers this width on the beam axis and searches for adjacent free edges. It will not use a width greater than the maximum available. For example: If you define b,max=100 and there is an opening on one side of the bea, the program uses b, max=50.
- attach the beam above or below the slab, for example:



- click



to delete the offset from the selected beams.

- select beams using the standard beam selection option.

The program automatically creates the offset groups and assign them to the beams.

3.5.6.5 Program material

Ten permanent materials are stored in the program:

- six predefined materials (steel, concrete, etc.)
- four user-defined materials

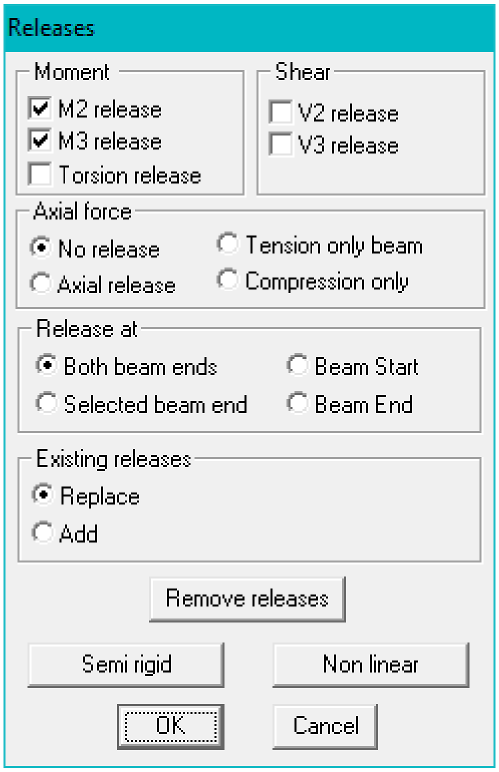
The properties (modulus of elasticity, Poisson ratio, density and thermal coefficient) for all ten materials may be defined or revised, as follows:

- Select **Files** in the ***STRAP*** main menu
- select Setup

- select **Materials**

3.5.7 End releases

Release degrees-of-freedom at beam ends to simulate pinned connections, sliding connections, etc.
- axially loaded members may be defined as tension only or compression only.
- semi-rigid moment connections and plastic hinges may be defined



Moment/shear

M2 release - moment release about local x2 axis (at either or both ends)
M3 release - moment release about local x3 axis (at either or both ends)
Torsion release - MT release (always at both ends)
V2 shear - sliding release parallel to the local x2 axis (at either or both ends)
V3 shear - sliding release parallel to the local x3 axis (at either or both ends)

Note:
- Beams released for shear at both ends in the same direction must not be loaded.
- The graphic deflection of beams with shear releases will be displayed incorrectly.

Axial force

No release
default status

Axial release
axial force is released

Tension only / compression only
Define beams that can take either axial compression forces or tension forces, but not both.

Note:

- these beams are ***non-linear*** elements and require several iterations of the solution.
- the rules of superposition do not apply for non-linear elements. Therefore, load combinations for models with tension/compression only elements ***must be defined*** in loading ("Combine load cases") and not after the solution.
- the stiffness matrix is calculated separately for each load case.
- the moment stiffness is independent of the axial force stiffness.

Release at

Both beam ends
Select beams using the standard beam selection option. The ■ blip is displayed at the beam centre; both ends are released

Selected beam end
Beams must be selected individually; highlight the end of the beams to be released (the ■ blip is displayed at both ends).

Beam start/end
Select beams using the standard beam selection option. The ■ blip is displayed at the beam centre; only the specified end is released

Note:
- Torsion and axial force are always released at both ends.

Existing releases

Replace
The program assigns the specified releases (☑ or ☐) to the selected beams, replace the existing releases.

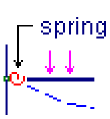
Add
The programs adds the specified ☑ the the existing ones (ignoring the ☐ releases)

Remove release

ALL releases will be removed (the ☑ checkboxes are not relevant for this option).

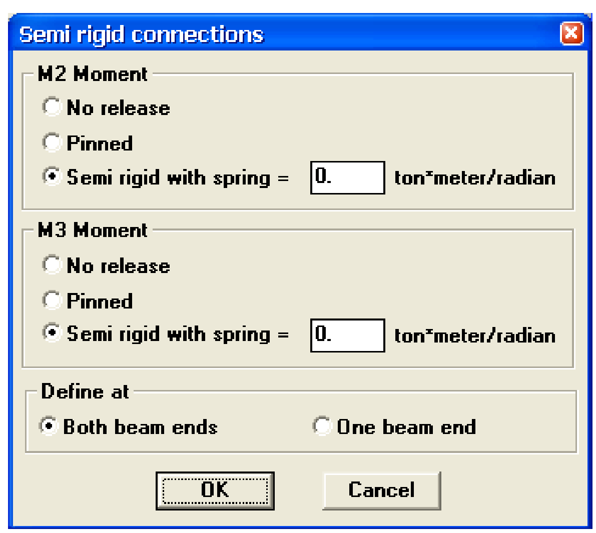
Semi-rigid

Define a semi-rigid beam end support by defining a spring connecting the end of the beam to the support node



The end moment applied to the node is proportional to the spring stiffness and will vary between zero and full fixity.

Similar to regular releases, semi-rigid connections can be defined separately for M2 and M3 moments and may be defined at either or both ends of the beam:



Non-linear

Refer to Releases - non-linear257.

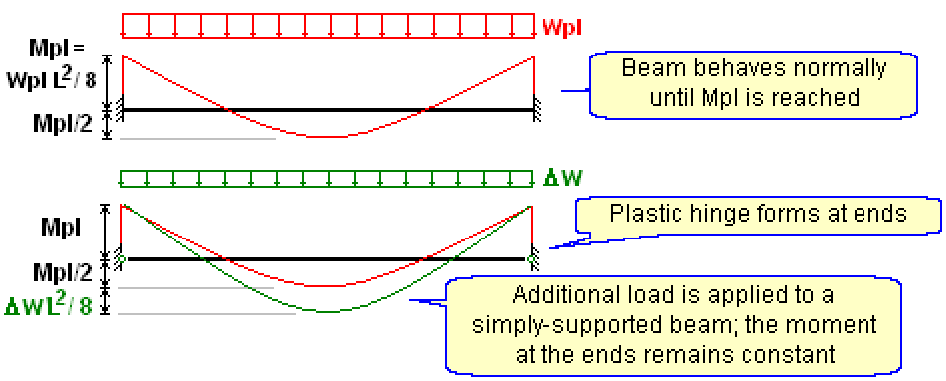
Note:

- For members with section "Joist"property (American steel table only): the program will automatically revise the end releases of a member assigned with a Joist to "Pinned". The release will be revised only upon exiting the Geometry module or if "Save" is selected.
- Every unrestrained node must have at least one unreleased beam connected to it for every degree-of-freedom of the ***node***. A "ZERO STIFFNESS" warning message will be generated during the solution for every node having only released beams connected to it. Note that the single fixed beam will behave as if pinned because the other beams connected to the node have no end moment to transfer to it.

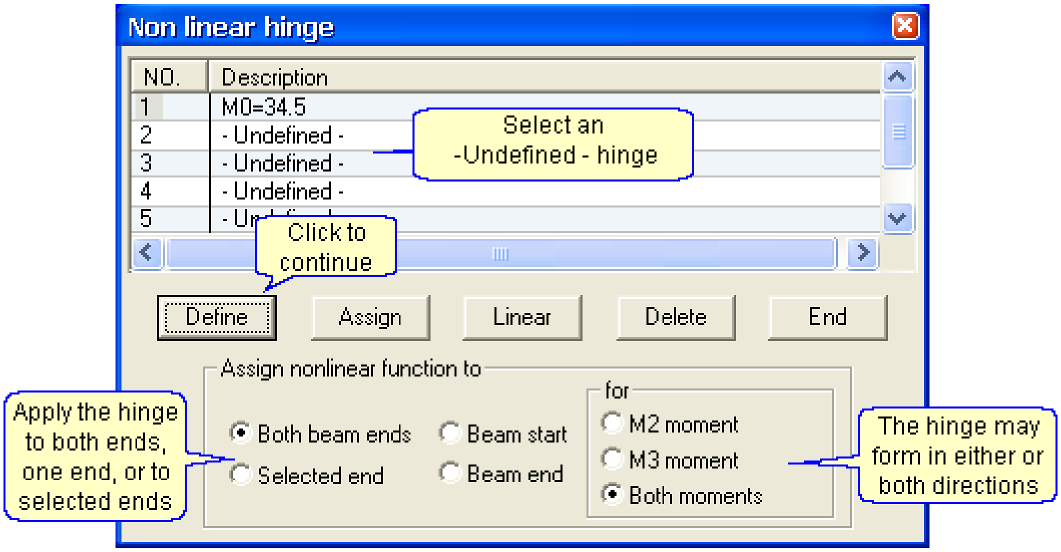
  For example, the restraint for X6 at a node may be supplied by not releasing the M2 (M3) moments of a beam lying on the X1-X2 plane or not releasing the torsional moment of a beam parallel to the X3 axis.

3.5.7.1 Non-linear

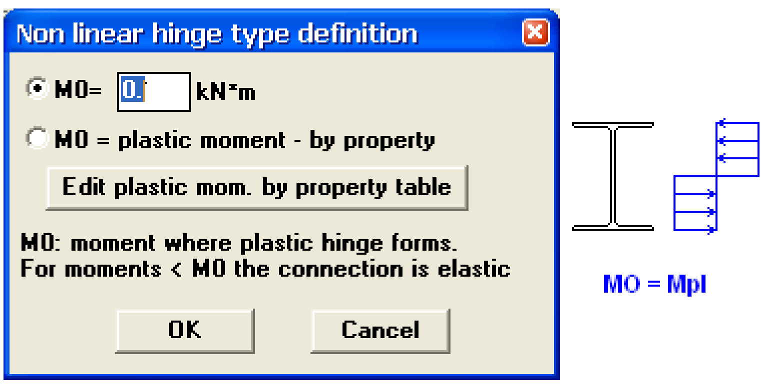
Define plastic hinges at selected beam ends. The program assumes that the beam end behaves normally until the"plastic moment" value is reached. A plastic hinge then forms and the beam behaves as if there is a pinned connection under additional load. For example, a simple beam under uniform load:



To define a non-linear hinge:

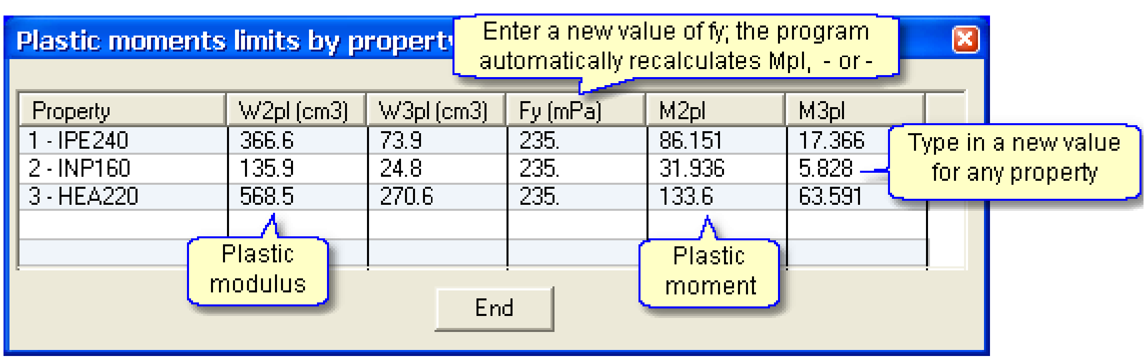


Define the plastic moment:



There are two options:

- **Mo =**
  Enter a value according to the displayed units
- **Plastic moment by property table**
  The program automatically calculates the plastic moment for defined properties. For example:



Note:

- the option is suitable for steel sections only; reinforcement and cracking are ignored for concrete sections
- tapered sections: the capacity is calculated for one end only

- sections from CROSEC: the plastic moment is accurate only for symmetric section, otherwise it is only an estimate.

Select one of the following:

Assign	Highlight a function in the list and assign it to beams. Select beams using the standard beam selection option.
Delete	Delete the highlighted functions from the list. The "assignment" to beams is not deleted.
Linear	Cancel the "assign" for selected beams. Select beams using the standard beam selection option; beams that were "assigned" with this function will revert to regular end connections.

Note:

- one function can be assigned to the M2 moment at a beam end and a different function for M3 may be assigned to the same end.
- non-linear releases are shown on the graphic display as ***n2,n3*** at the beam ends, where *n2, n3* are the function numbers assigned to M2 and M3 respectively.

3.5.8 Offsets

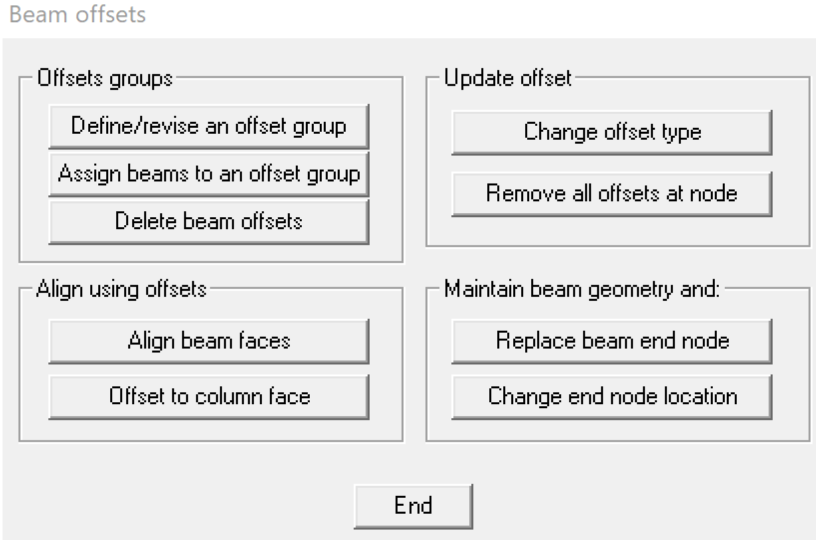
In some cases the assumption that the ends of a beam element are located at the end nodes is inaccurate. A typical example is a shear wall with openings forming horizontal beams; the span of the beams should be measured from the face of the walls rather than from the wall centre where the end nodes are located. The effect on the beam results can be significant when this "offset" length (one-half of the wall width in this example) is large compared to the beam length,

Use this option to define rigid ***Offsets*** at the beam ends; the program assumes that the beam element is ***infinitely*** rigid in the offset length. The actual beam length is measured from the end of the Offsets and program adds moments resulting from the eccentricity of the new beam end to the node.

New JA, JB locations are assumed to be at the end of the offsets. This can lead to a modification in the directions of the local coordinate system axes. ***These modified local axes are used throughout the program*:**

- local axes displayed graphically or in tables will be the modified axes.
- load locations will be measured from the new JA.
- total load applied to a beam will be the distributed load multiplied by the modified length.
- All beam results will be relative to the modified axes.

Offset definition is similar to Property definition; an Offset is defined and assigned to selected beams in separate options:

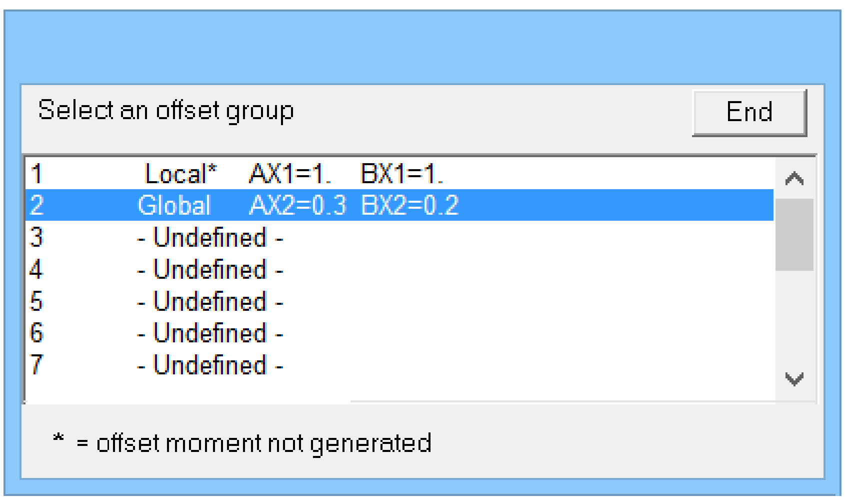


Note:

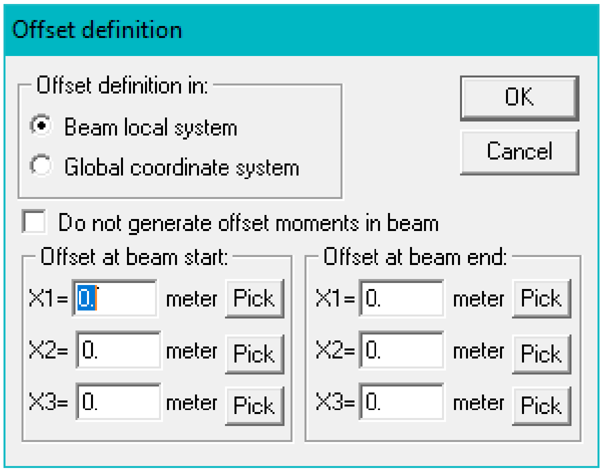
- ***different x1 offsets should not be defined in a 'stage' for loaded beams***; the program uses the total length of the beam from the 'whole model' when calculating the bending moment diagrams for all combinations (the end moments are always correct).

3.5.8.1 Define/revise offset group

Select an Offset group to define or revise; the program displays the Offset group table:



Define the Offset dimensions:



Offset definition in

Beam local system
the dimensions are relative to the beam local coordinate system

Global coordinate system
the dimensions are relative to the global coordinate system

Offset at start/end

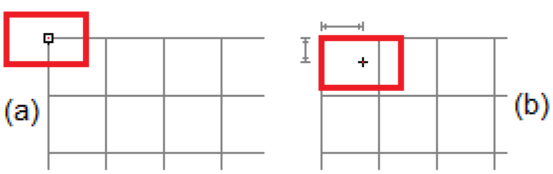
The length of the rigid offsets are measured from the JA and JB nodes of the beam:

Offset at beam start: the length of the offset starting at JA
Offset at beam end: the length of the offset starting at JB

Pick Define the offset dimension by pointing to two nodes. The program calculates the dimension between the two nodes and enters the value in the menu.

Do not generate offset moments in beams

Offsets sometimes generate unwanted moments in attached beams and columns. For example, the column in Figure (a) is defined at the slab corner node but the column faces are in reality aligned with the slab edge as shown in (b):



If the offset as shown in (b) is defined, the column will appear at the correct location in all of the drawings but an additional column moment will be generated equal to product of the beam/slab reactions and the offset distance .

Set the option to ☑ to calculate the moments and forces as if the offset does not exist.

If an offset is defined for a beam element, new JA, JB locations are assumed to be at the end of the offsets. Referring to example (b) below, this can lead to a modification in the directions of the local coordinate system axes. ***These modified local axes will be used throughout the program***:

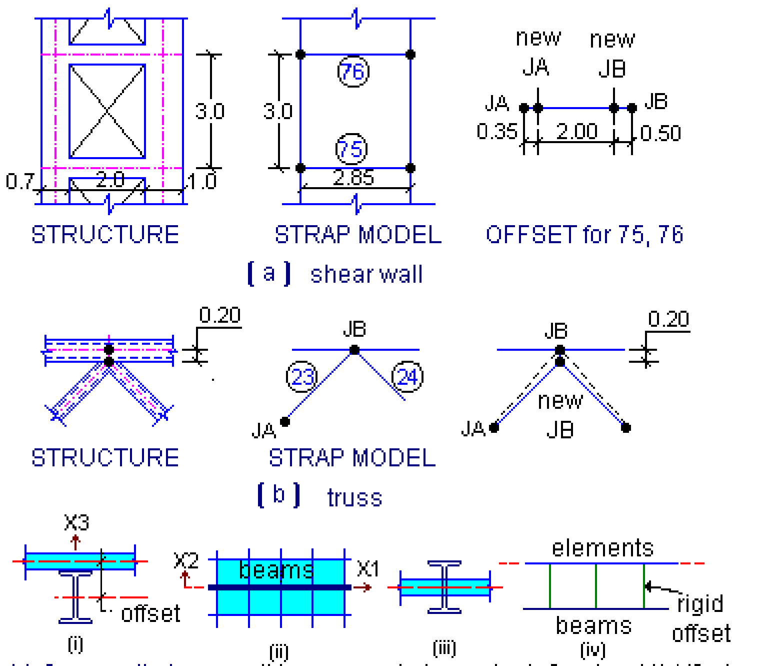
- local axes displayed graphically or in tables will be the modified axes.
- load locations will be measured from the new JA.

- total load applied to a beam will be the distributed load multiplied by the modified length.
- All beam results will be relative to the modified axes.

Note:
- ***different x1 offsets should not be defined in a 'stage' for loaded beams***; the program uses the total length of the beam from the 'whole model' when calculating the bending moment diagrams for all combinations (the end moments are always correct).

Examples:

Example (c) uses offsets to model a composite beam. When all beams and elements are defined in the same plane (c-ii) the centre of the beam is at the centre of the slab (c-iii) and the enhanced properties of the composite section are not utilised. The model is corrected by offsetting the beams by the distance from the centre of the beam to the centre of the slab. This creates a model similar to a Vierendel truss with the rigid offsets forming the vertical links (c-iv). This model has properties similar to that of the composite beam (note that the beam must be defined as a space model in order to specify X3 offsets).



(c) Composite beam: all beams and elements defined on X1-X2 plane

Define:

(a) Beams 75, 76:

set **Global coordinate system**

Beam start: **X1 = 0.35** Beam end: **X1 = -0.50**

(b) Beams 23, 24:

set **Global coordinate system**

Beam end: **X2 = -0.20**

(c) All beams:

set **Global coordinate system**

Beam start: X3 = offset Beam end: **X3 = offset**

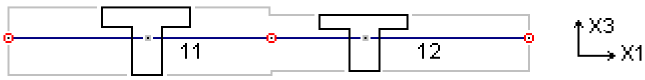
3.5.8.2 Offset groups - assign

Select beams that this Offset is to be assigned to using the standard Beam Selection option.

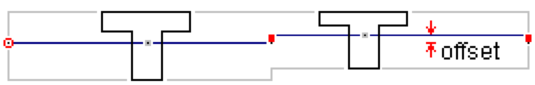
Note that you can assign an **-Undefined-** offset group to beams and define the section properties of the Offset later.

3.5.8.3 Align faces

When a beam is defined the program aligns the line through the centre-of-gravity with the line connecting the end nodes. In the following example the top faces of the adjacent beams are not aligned because of the different dimensions:

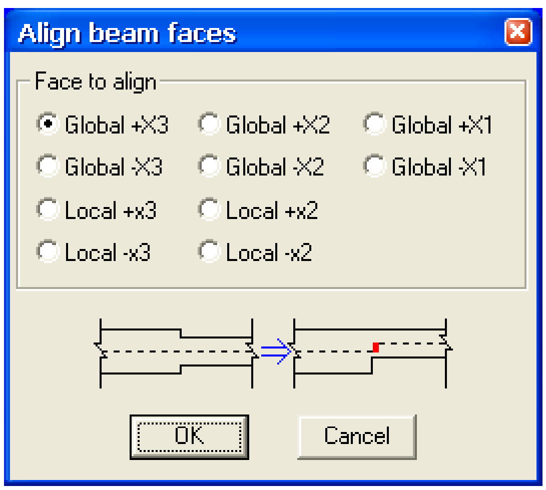


Vertical offsets must be defined to align the top beam faces:



This option aligns the beams automatically:

- select the face to align (Global +X3 in our example):



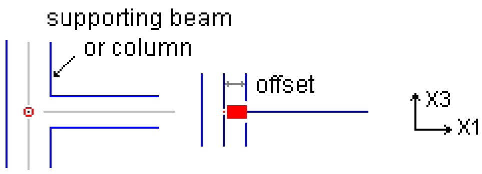
- select a reference beam - all selected beams will be aligned with it (beam 11 in our example).
- select the beams that are aligned with the reference beam using the standard Beam Selection option (beam 12 in our example).

Note:

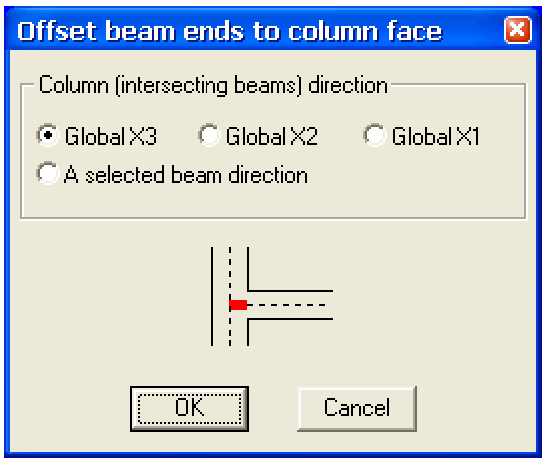
- sections defined by "Properties" must have the relevant H2/H3 dimension value.
- the program searches for a local x2/x3 axis that is parallel to the specified global axis or the closest local axis. The program does not create offsets if the angle between the global axis and the closest local x2/x3 axis is > 45°.
- The program automatically creates offsets with a dimension up to 0.5 meter. The program displays a warning if offsets > 0.5 meter are necessary and allows the user to change the tolerance value. This option is useful when beams on the wrong level are accidentally selected.

3.5.8.4 Offset from columns

Automatically create offsets at beam-to-column or beam-to-beam connections, for example:



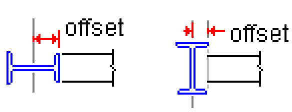
- Select the direction of the supporting beam or column (Global X3 in our example):



Alternatively, set **A selected beam direction** and select any beam parallel to the support beam

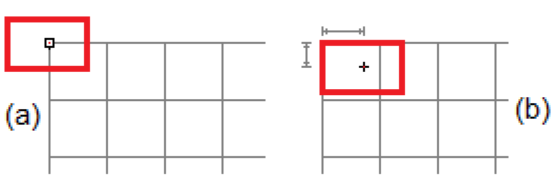
- select the supported beams using the standard Beam Selection option; offset groups are created and assigned to these beams.

Note:
- the offsets is always axial; the option does not align the flange of the beam with the flange of the supporting beam/column.
- supporting sections defined by "Properties" must have the relevant H2/H3 dimension value.
- steel sections: the offset is created according to the relevant exterior dimension:

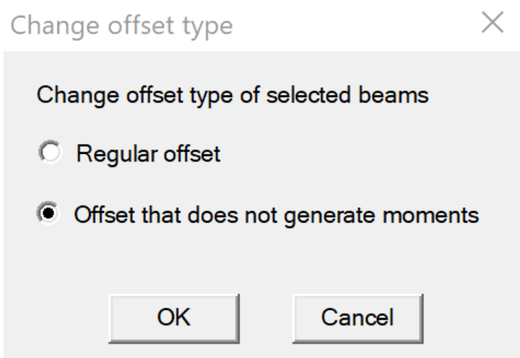


3.5.8.5 Change offset type

Offsets sometimes generate unwanted moments in attached beams and columns. For example, the column in Figure (a) is defined at the slab corner node but the column faces are in reality aligned with the slab edge as shown in (b):



If the offset as shown in (b) is defined, the column will appear at the correct location in all of the drawings but an additional column moment will be generated equal to product of the beam/slab reactions and the offset distance .



Select one of the following:

Regular offset
The program will add the moment due to offset.

Offsetthat doesn't generate moments
Moments due to offsets are not added

And select beams that this Offset type is to be assigned to using the standard Beam Selection option.

Note:

- If the selected beam was assigned to an offset group 260 with a different offset type prior to offset type change, the program will revise the group or if necessary create a new group and assign the selected beams to the new group automatically.

3.5.8.6 Remove all offsets at node

Remove all beams offsets at a node. All beams offsets will be deleted at a selected node.
Select the nodes using the standard Node Selection option.

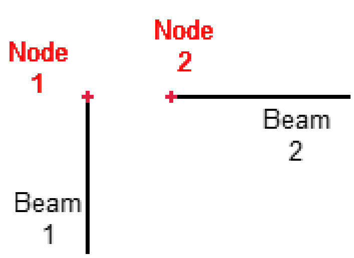
Note:

- If the selected node is connected to a beam that was assigned to an offset group 260, the program will revise the group or if necessary create a new group and assign the selected beams to the new group automatically.

3.5.8.7 Replace beam end node

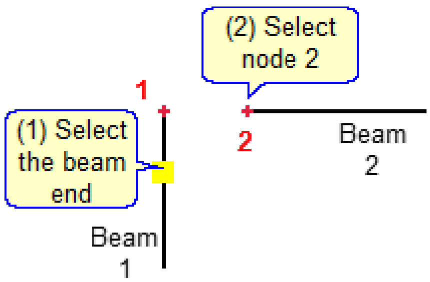
Revise the beam end node. The program will connect the selected beam end to the selected node and assign an offset to the beam end to maintain the beam's original location.

For example:
Change the end node Beam 1 from node 1 to node 2.

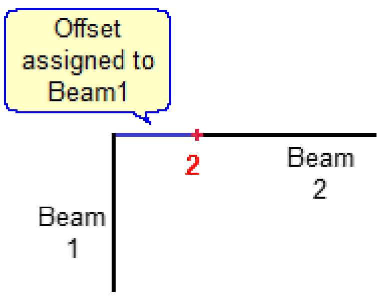


- select the option **Replace beam end node**:
- select the beam end closer to node 1.

- select node number 2.



The program connects Beam 1 to Beam 2 at node number 2 and assigns an offset to beam 1.



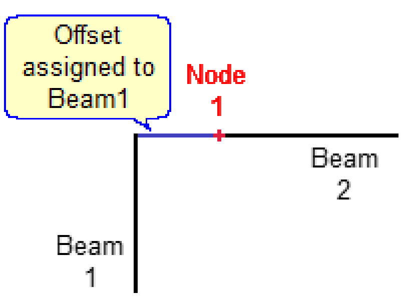
3.5.8.8 Change end node location

Change the end node location by selecting a node or nodes and a beam element. The program will move the node to the selected beam end and assign new offsets to other beams that are connected to the selected node.

Examples**:**

Individual node:

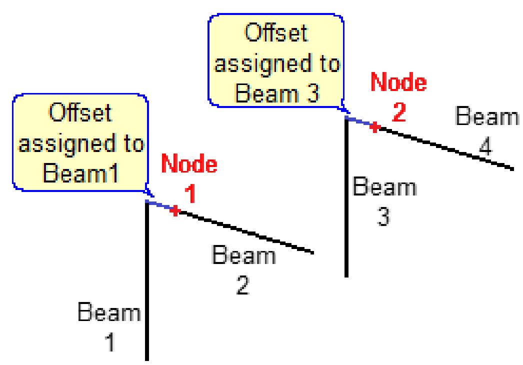
Change the end node location from the end of beam 2 to the end of beam 1.



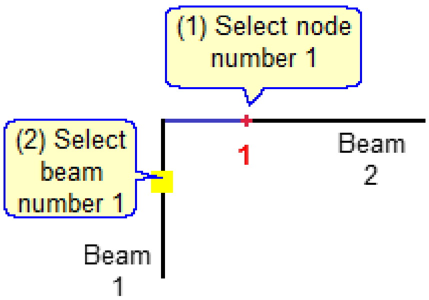
- select the option **Change end node location** :
- select node 1.
- select beam 1.

Multiple nodes:

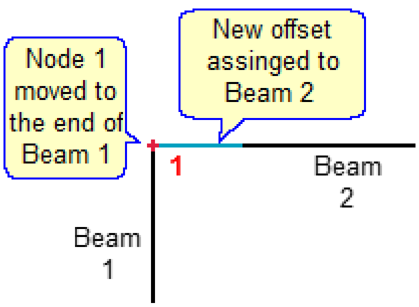
Change the end node location from the end of beam 2 and beam 4 to the end of beam 1 and Beam 3 respectively:

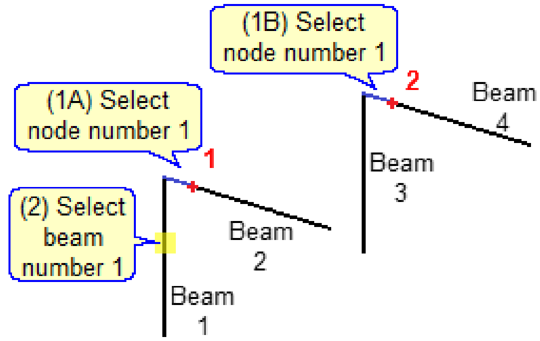


- select the option **Change end node location**:
- select node 1 and node 2.
- select beam 1.

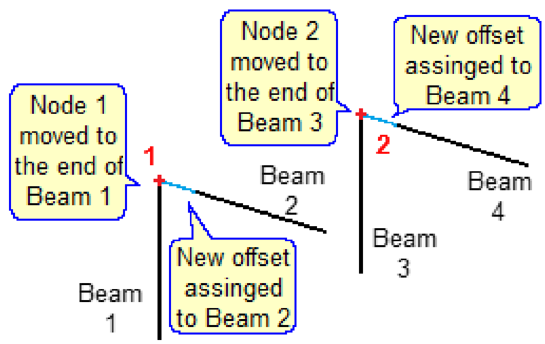


The program moves node 1 to the end of beam number 1, deletes the offset assigned to beam 1 and assigns a new offset to beam 2.



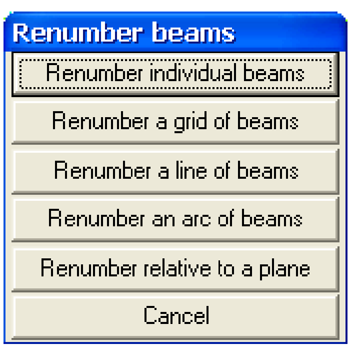


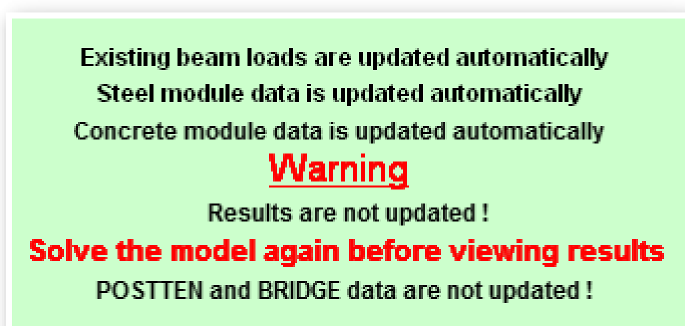
The program moves nodes 1 and 2 to the end of Beam 1 and Beam 3 respectively and assigns new offsets to Beams 1 and 3.



3.5.9 Renumber

Use this option to renumber existing beams:

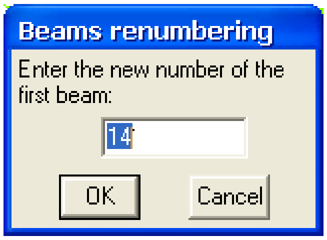




Individual beams

Select one or more beams using the standard Beam selection option.

Note that the order that the beams are selected is important; they will be renumbered in the order that they are selected.



Type the new number of the first beam selected; all of the beams selected are renumbered sequentially. If the program discovers that a number has already been assigned to another beam, the program assigns the original number of the selected beam to that beam.

Example:

- beams 41, 42 and 43 are selected (in that order).
- 75 is specified as the new number for 41
- the beams will be renumbered 75,76 and 77 respectively
- beam 76 is an existing beam; it will be renumbered 42.

Grid of beams

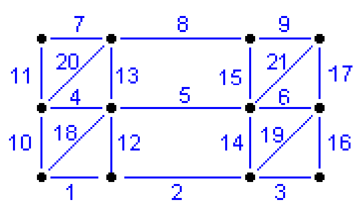
Similar to Line of beams 268.

The program instructs you to define the grid by pointing to the three corner nodes defining it and then requests the new number of the beam at the start of the base line.

All beams with ***both*** end nodes lying in the parallelogram defined by the three nodes will be renumbered, in the following order:

a. all beams parallel to the base line
b. all beams parallel to the height line
c. all other beams

For the following example, the renumbering order is:



Line of beams

Use this option to renumber all the beams in a line:

- Select the two nodes defining the line
- Type the new number of the first beam in the line

All of the beams selected are renumbered sequentially. If the program discovers that a number has already been assigned to another beam, the program assigns the original number of the selected beam to that beam.

Arc of beams

Use this option to renumber all the beams lying on an arc:

- Select the two nodes defining the start and end of the arc
- Select any other node lying on the arc
- Type the new number of the first beam on the arc

The program identifies all nodes on the defined arc and will renumber all beams connecting sequential nodes.

All of the beams selected are renumbered sequentially. If the program discovers that a number has already been assigned to another beam, the program assigns the original number of the selected beam to that beam.

Relative to a plane

Renumber all beams on selected planes. This option is handy for renumbering an entire model or parts of a model consisting of more than one plane. Note that the planes do not have to be parallel.

- select the beams to be renumbered using the standard beam selection option
- define a plane that specifies the renumbering order; the plane is defined by selecting three existing nodes.
- specify the new number of the first beam

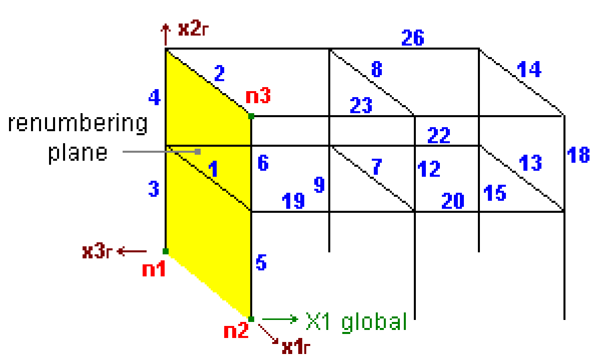
The renumbering order is determined as follows:

- the first two nodes define the ***x1r*** axis of the renumbering plane; the third node defines the ***x2r*** axis of the plane; the ***x3r*** axis is determined by the right-hand rule.
- the program sorts the beams according to the angle between their x1 local axis and the ***x3r*** axis of renumbering plane, starting with the beams closest to the renumbering plane. Note that if there are beams on both sides of the plane, the program will first select all beams on one side, then all beams on the other side.
- for beams on the same plane, the program then sorts according to the angle between the beam x1 local axis and the renumbering plane ***x2r*** axis, beginning with the largest angle. Beams with identical angles are sorted according to their renumbering plane coordinates, beginning with the smallest value.
- for beams with identical x1-***x2r*** angles, the program then sorts according to the ***x1r*** coordinate, beginning with the smallest value.

All of the beams selected are renumbered sequentially. If the program discovers that a number has already been assigned to another beam, the program assigns the original number of the selected beam to that beam.

Example:
Renumber the following space frame; the renumbering is to start on the planes perpendicular to X1 global

- select nodes **n1**, **n2** and **n3** to define the renumbering plane
- specify **1** as the new number of the first beam



- the beams on the **x1r-x2r** plane are selected first (**1-6**);
- the horizontal beams are perpendicular to x2r (i.e. angle = 90°) and are selected first. Beam 1 has the smallest x2r coordinate and is renumbered first. Similarly, beam 3 will be renumbered first among the vertical beams.
- then the beams on the parallel planes are renumbered (**7-12**) and (**13-18**)

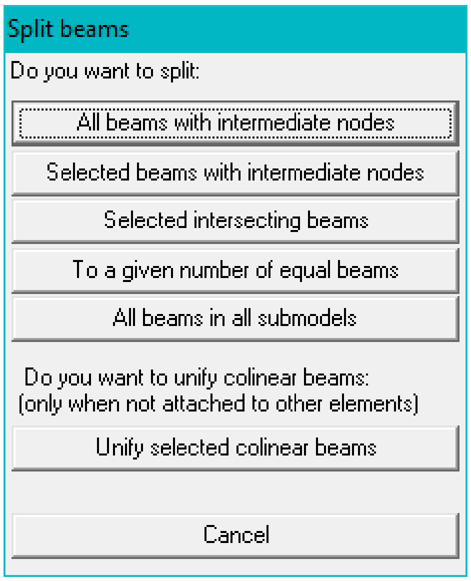
- then the beam perpendicular to the **x1r-x2r** plane are renumbered.(**19-26**).

Diagonal beams are renumbered according to the same algorithm.

3.5.10 Split

Use this option to:
- divide a beam into two or more elements where existing nodes are located along the line of the beam.
- combine colinear beams (that are not attached to other beams/elements/walls)

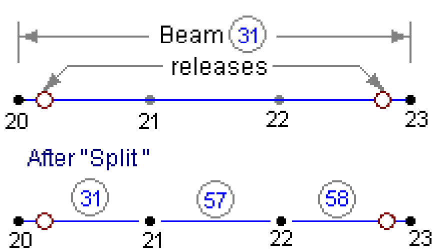
Split selected beams or instruct the program to automatically search the entire model and split all beams where applicable:



- The first new beam is assigned the same beam number as the original beam; the others are numbered according to **Beam no. =**.
- All beams are assigned the same property number as the original beam. If releases were defined for the original beam they will be at the same nodes after the beam is split.

All beams with intermediate nodes
Selected beams with intermediate nodes

Example:

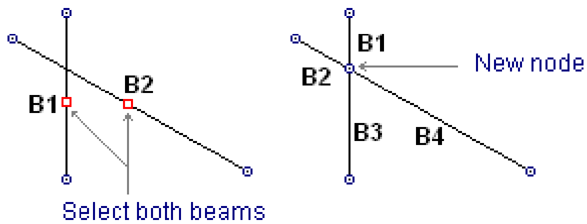


- beam 31 is connected to nodes 20 and 23 and nodes 21, 22 are located along the path of the beam.
- select beam 31; the beam is divided into three beams as shown in the figure.

Note:
- "All beams" refers to the beams in the current submodel only.

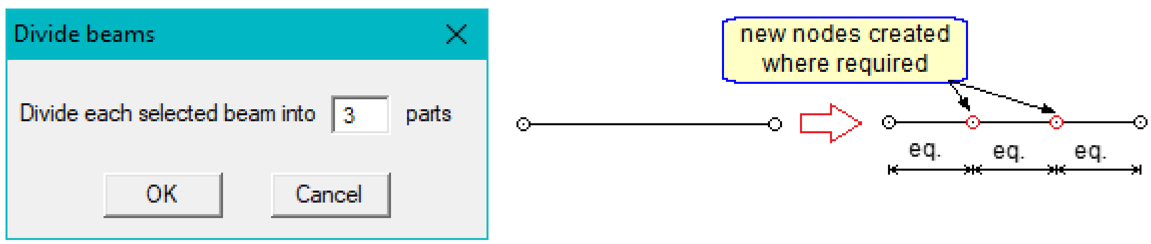
Selected intersecting beams

Select ***both*** of the intersecting beams:



A given number of equal beams

Divide selected beams into series of beams with equal length. The program creates new nodes at the beam ends. For example:



All beams in all submodels

All beams in the entire model will be split.

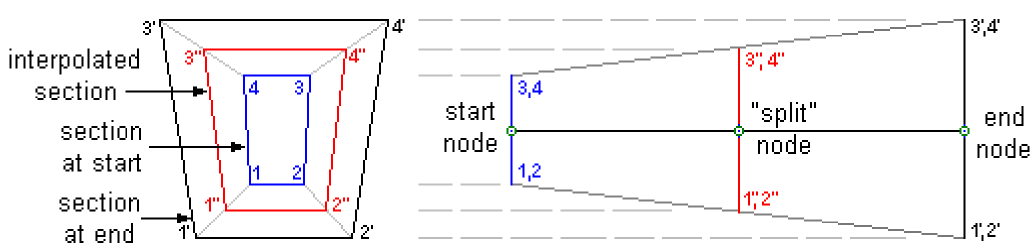
Unify colinear beams

Combine selected colinear beams into one single beam.

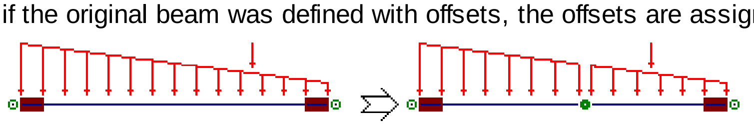
- only beams that are not attached to other beams/elements/walls may be combined.

Note:

- for tapered beams: the program automatically creates new sections at the intermediate nodes:
  - if the two end sections were defined by "Dimensions" and have the same shape type, or the two end sections were defined in *CROSEC* (solid sections) and have the identical number of lines, the program interpolates the dimensions to create the new section with dimensions. For example, two solid sections:



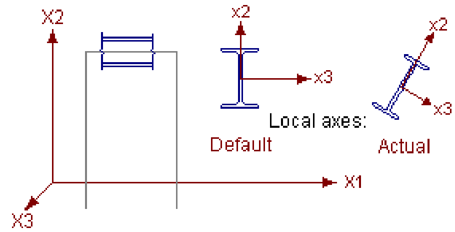
  - for all other combinations of end sections, the program creates a section defined by "Properties" by interpolating the section properties (I,A,h, etc) of the end sections.
- if original beam previously had loads defined on it, the loads are also split to the new beams.
- if the original beam was defined with offsets, the offsets are assigned accordingly to the new beams.



3.5.11 Local axes

The program creates a default local coordinate system x1,x2,x3 for each beam; the major/minor axes of the beam section are always aligned with the local x2/x3.

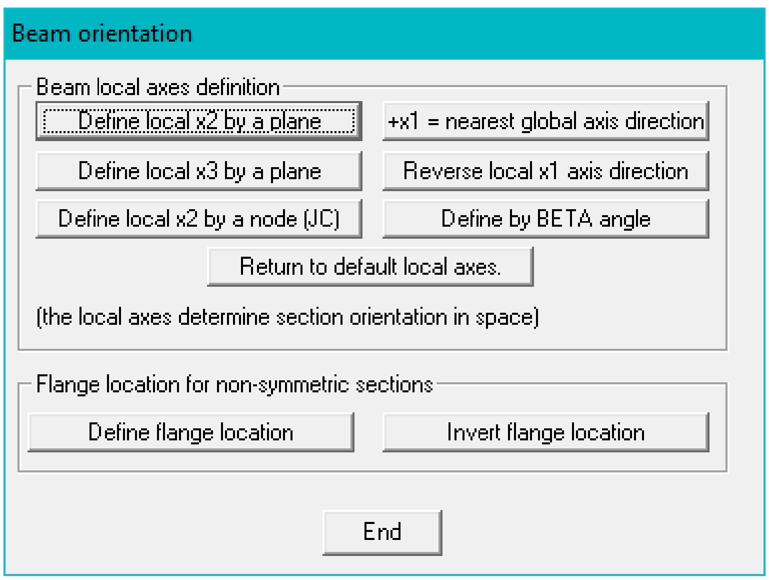
In certain cases the major/minor axes of the beam may not be parallel to the default local axes and the local axes must be rotated. For example:



Use this option to rotate the x2,x3 local axes in any direction other than the default. The x2(x3) axes may be defined as lying parallel to any plane in space, as pointing to a specified node or may be rotated about the local x1 axis by any angle β.

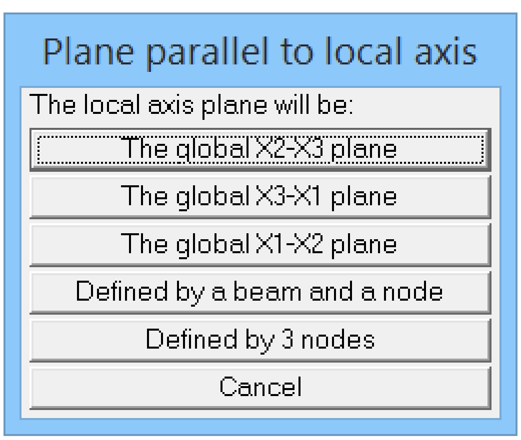
The x1 axis direction may also be reversed.

The current flange location may be displayed using the Rendering option or the  Display - section orientation option



Define local axis by plane

The beam local x2 or x3 axis can be defined as parallel to any plane in the model:



Use one of the above options for defining the plane, then select the beams using the standard Beam Selection option.

Define local axis by node

Select any existing node in the model; the x2 axis of the selected beams will point towards it.

+x1 = nearest global axis direction

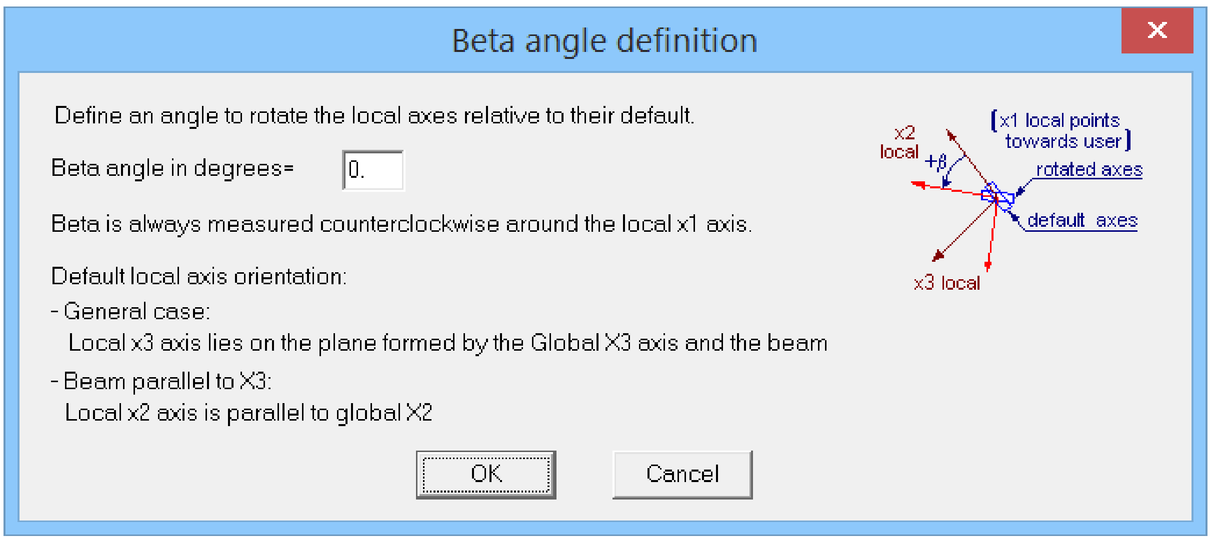
The local x1 axis is always parallel to the axis of the beam. Revise +x1 so that it points in the general direction of the closest positive global axis.

Reverse local x1 axis

The local x1 axis is always parallel to the axis of the beam. Select this option to reverse the positive direction of the axis.

Define by Beta angle

Rotate the beam about its axis by a specified angle. The conventions are explained in the dialog box:



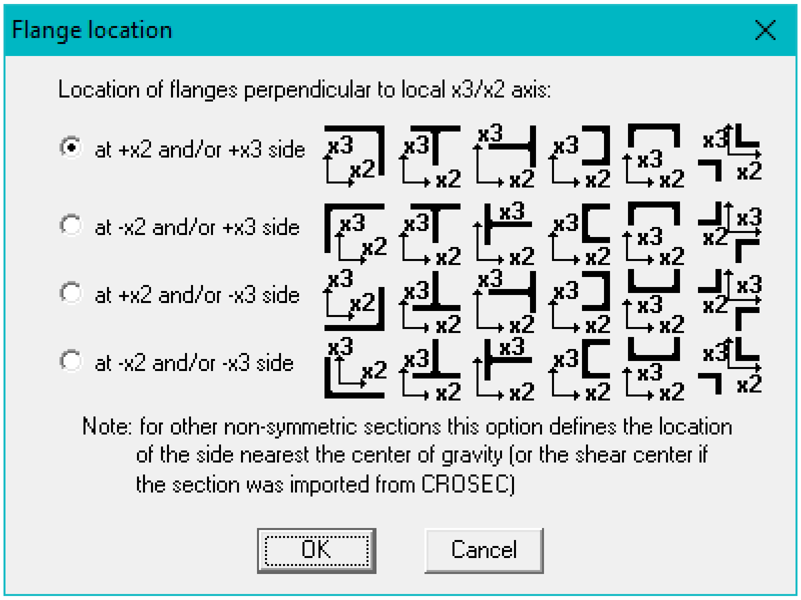
Refer to beam local axes for additional details on the default orientation of the local axes.

Return to default axes

Restore the default local coordinate system for the selected beams.

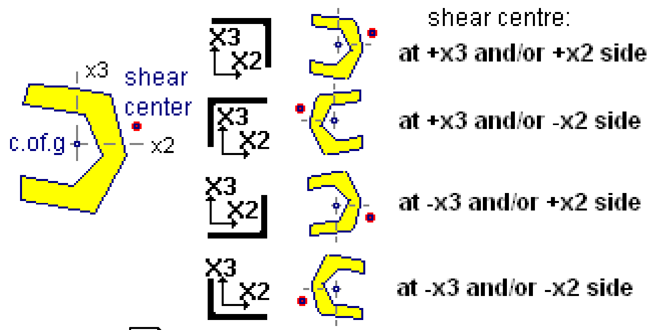
Define flange location

This options allows the exact flange location to be defined for the following non-symmetric sections. Although not important for analysis, unless "principal axes" is selected for L-sections (the moment-of-inertia does not change if the flange location is inverted), the defined orientation will be displayed in the rendered drawing and will be used as the default orientation by the steel postprocessor, etc.



For all other non-symmetric sections:

- sections defined in CROSEC: this option defines the location of the shear center relative to the centre-of-gravity of the section"



- "Combined 236" sections: the "flanges" are assumed to be on the X/Y faces that are closest to the center-of-gravity, where X/Y refer to the horizontal/vertical axes in the section definition dialog box.

The current flange location may be displayed using the Rendering option or the Display - section orientation option.

Invert flange location

Invert the current flange location about ***both*** local axes. The current flange location may be displayed using the Rendering option or the Display - section orientation option.

3.5.12 Stages

Beams may be removed from the current stage or restored.

- Select beams using the standard beam selection options

Note:

- inactive beams are not displayed
- new beams cannot be defined when a stage other than "Whole model" is active.

3.6 Elements

Define quadrilateral or triangular elements by specifying:

- location : select the corner nodes
- properties : define the element thickness
- material : elements may be isotropic or orthotropic
- local axes : specify the local x3 axis direction

The following options are available when "Elements" are selected in the geometry Main Menu:

triangle

- Define one triangular element 276 by identifying its end nodes.
- Define a chain of triangular elements where only the third node of successive elements need be defined; the program uses two nodes of the previous element (common to both elements) to complete the new triangle.

quad.

- Define one quadrilateral element 277 by identifying its end nodes.
- Define a chain of quadrilateral elements where only the third and fourth nodes of successive elements need be defined; the program uses two nodes of the previous element (common to both elements) to complete the new quadrilateral.

Grid

Define a parallelogram grid 278 of elements by identifying the three corner nodes defining the 'base' line and the 'height' line of the grid. The program automatically searches for intermediate nodes and creates a grid of elements.

Surface

Define elements on any surface 281:
Define a 'base' line consisting of a chain of nodes and a 'height' line, also a chain of nodes. The program copies the base line to every level on the height line, generating nodes and elements. For example, if the base line is a semi-circle and the height line is a perpendicular line, the program generates a half cylinder; if the second base line is a semi-circle with a smaller radius, the surface is conical. This option also generates plane grids.

Mesh

Generate a mesh 282 of nodes along with the corresponding mesh of elements. The grid outline is defined by specifying a contour, and the size of the generated elements is determined by user defined parameters.

Delete

Delete 298 elements already defined.

Renumber

Renumber 298 elements already defined.

Properties

Define element properties 300 (including material) and assign them to finite elements.

Local axes

Revise the direction of the local x3 axis 306. This option is used to reverse the direction of the local x3 axis as set by default by the program.

Releases

Define the edges of bending elements as "pinned 307".

Offset

Define offsets 309 perpendicular to the element

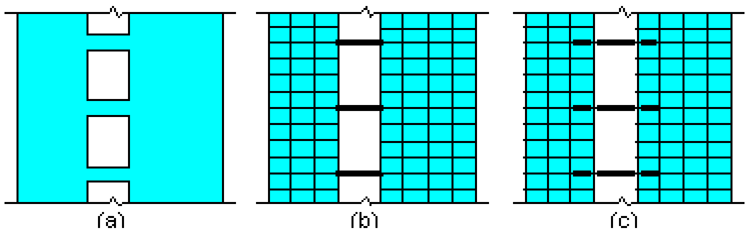
Click **Stages** to select a stage other than **Whole model;** different properties may be defined for each stage and elements may be removed:

Deactivate

'Remove' an element from the current stage.

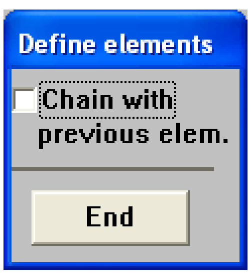
Restore an element to the current stage.

Note:
- Models may contain both beam elements and finite elements.
- the end nodes of finite elements do not transfer bending moments to adjacent elements, i.e. the corners of these elements are all 'released'. Referring to the shear wall in Figure (a), if the lintel beams are modeled by beam elements as shown in Figure (b), no bending moments will be generated in them. The beams should be extended into the wall as shown in Figure (c).



3.6.1 Triangle

Define a single triangle or a chain of triangles, where any two successive elements in the chain have two common nodes.



☐ *Single element*

Point to the three corner nodes of the element; the element is drawn immediately. All elements are automatically assigned with:
- the **El no.** displayed at the bottom of the screen
- the Prop group number displayed at the bottom of the screen
- the program default local axes.

Refer to Single quad 277 for an example.

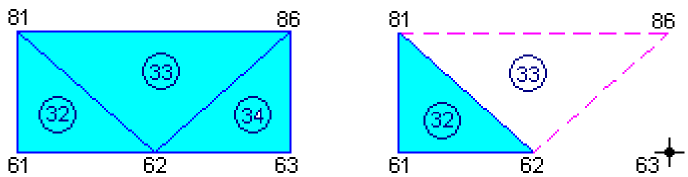
☑ *Chain of elements*

Define a continuous string of triangular elements, where any two successive elements in the chain have two common nodes.
- set the option to ☑
- define the three corner nodes of the first element in the chain, then define the ***third node only*** of the following element; the program will use two nodes of the previous element to complete the triangle.
- continue defining the third node of more elements in the chain.

All elements are automatically assigned with:
- the **Prop** group number displayed at the bottom of the screen
- the program default local axes.
- the first element is assigned the **El no.** displayed at the bottom of the screen and the remaining elements are numbered consecutively.

Example:

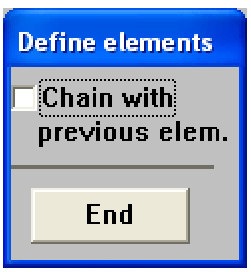
Define elements 32, 33 and 34 using the chain option.



- Define element 32 by pointing to nodes 61, 62 and 81. Refer to Single triangle 276.
- define element 33 by pointing to node 86 only; the program automatically selects nodes 62 and 85 as the other nodes of the triangle (Figure b).
- In a similar manner, select node 63 only to define element 34.

3.6.2 Quadrilateral

Define a single quad or a chain of quads, where any two successive elements in the chain have two common nodes.

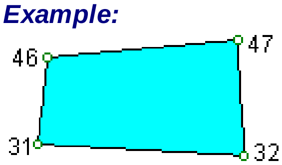


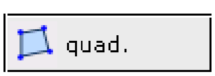
☐ *Single element*

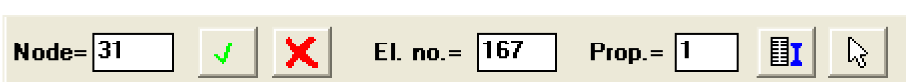
Point to the four corner nodes of the element; the element is drawn immediately. All elements are automatically assigned with:

- the **El no.** displayed at the bottom of the screen
- the Prop group number displayed at the bottom of the screen
- the program default local axes.

Example:

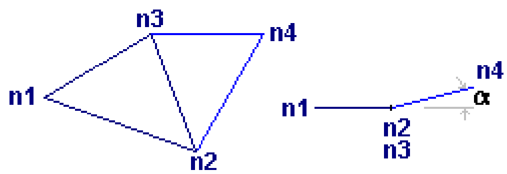


- select 



- move the cursor adjacent to node 31 so that the node is highlighted with the blip ■ and **Node = 31** appears in the Dialog box. Click the mouse.
- Similarly select nodes 32, 47 and 46.
- the element is created and assigned with the **El.no**. and **Prop** group number displayed in the Dialog box; these numbers may be revised by moving the cursor to the edit box, clicking the mouse and entering a new number. An existing property number may be selected by clicking .

Note:
- The four selected nodes must be co-planar. If they are not, the program internally divides the quad into two triangles and checks the angle α between them:



  - if α > 0.5°, the program displays a warning but creates the 'warped' quad element (reduced accuracy).
  - if α > 3.0°, the program displays a warning but does not create the element.

Chain of elements

Define a continuous string of quadrilateral elements, where any two successive elements in the chain have two common nodes.

First define the four corner nodes of the first element in the chain, then define the ***third and fourth nodes only*** of the following element; the program will use two nodes of the previous element to complete the quadrilateral. The program will only accept nodes which form convex / coplanar quadrilaterals.

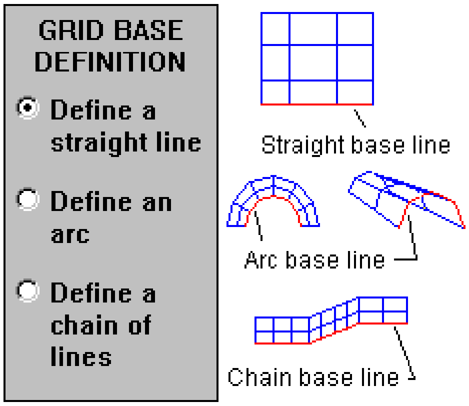
Refer to Chain of triangles 276 for a similar example.

All elements are automatically assigned with:
- the **Prop** group number displayed at the bottom of the screen
- the program default local axes.
- the first element is assigned the **El no.** displayed at the bottom of the screen and the remaining elements are numbered consecutively.

3.6.3 Grid

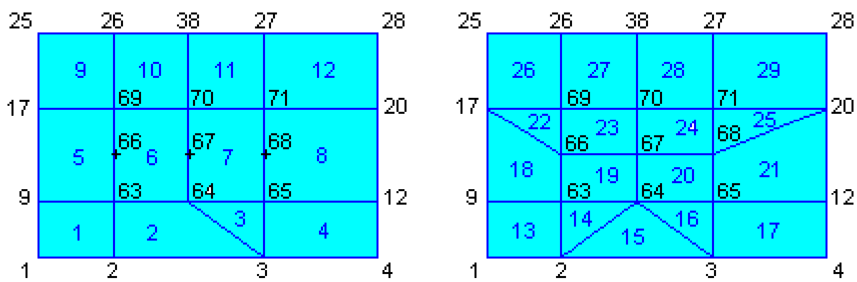
Define a parallelogram grid of elements by identifying the three corner nodes defining the 'base' line and the 'height' line of the grid. The program automatically searches for intermediate nodes and creates a grid of elements, as follows:
- this option does ***not*** generate new nodes.
- the program searches for intermediate nodes on the base and height lines ***only***.
- only the nodes lying on the lines parallel to the base line and running through the nodes on the height line are used to generate the elements; all other nodes are ignored.
- the program creates quadrilateral elements unless intermediate nodes require it to add triangular elements.

Three options are available for defining the base line:



Straight line

Example:



(a) Grid (b) Mesh

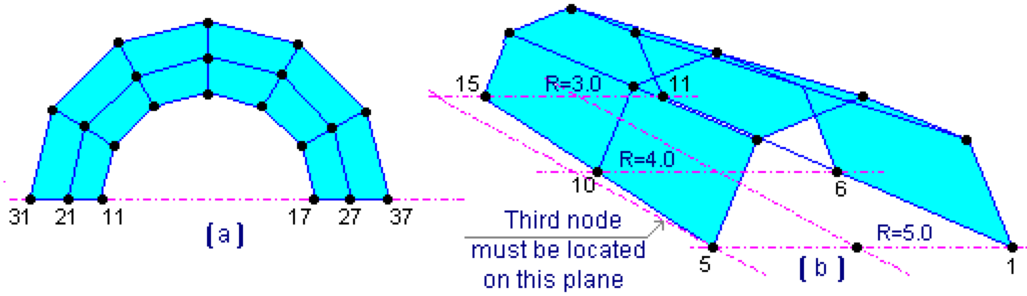
Nodes 1, 4 and 28 were defined as the corner nodes of the grid:
- the program identifies nodes 12 and 20 on height line 4-28.
- all nodes are identified as lying on the grid of parallel lines running through these nodes, except nodes 66, 67 and 68.
- node 38 lies on line 25-28 and this is sufficient to include it in the grid.
- the program then creates the grid using only the identified nodes.

The example is also used to demonstrate the difference between the Grid and Mesh commands; The mesh in Figure (b) is generated on the same pattern of nodes.
- The Mesh option searches for ***all*** nodes lying in the area formed by the grid (or contour) and uses all of them when generating the elements. If several element patterns are possible, the program generates the optimal one, i.e. the pattern where the element shape is as near to rectangular as possible.
- the Mesh option is most useful in cases where the node pattern is not regular, e.g. around openings where more elements are usually required because of local stress concentrations.

Arc

Use this option to define:
- a series of concentric arcs of elements all lying on the same plane.
- a series of parallel arcs concentric about the same forming a shell.

Examples:



First define the "base" arc, then identify a third node defining the height line:

- if the third node lies on the same plane as the base arc, then a series of concentric arcs is created. (Figure a)
- if the third node does not lie on the same plane, then a series of parallel arcs concentric about the same axis is created. (Figure b)

The third node must lie on the plane formed by the central axis of the base arc and the radius to the end node of the base arc. The radius from the central axis to the third node ***need not be equal*** to the base arc radius. This useful feature is illustrated in the example of Figure (b).

Example (a):

- select node 11 as the start node of the base arc
- select node 17 as the end node of the base arc
- select any of the nodes 12 to 16 to complete the definition of the base arc
- select node 37 as the third node of the grid.

Example (b):

- select node 1 as the start node of the base arc
- select node 5 as the end node of the base arc
- select any of the nodes 2 to 4 to complete the definition of the base arc
- select node 15 as the third node of the grid.

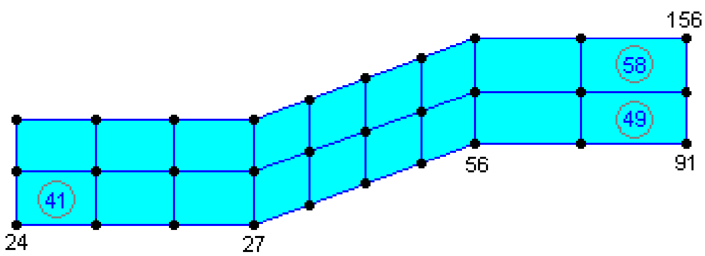
Chain

Define a grid, where the base line consists of a chain of connected lines (each line containing several nodes) that are not necessarily parallel.

- The base line is defined as explained in Chain - Quadrilateral 278; the end node of each line is the start node of the following line.
- The definition is completed by identifying a third node defining the height line.

Example:

Define elements 41 to 58:



- Select node 24 as the start node of the first line of elements on the base line
- Select node 27 as the end node of the first line.
- Select node 56 as the end node of the second line
- Select node 91 as the end node of the third line of elements on the base line.

- Press [Enter] without moving the to complete the base line definition.
- Select node 156 to define the height line of the grid.

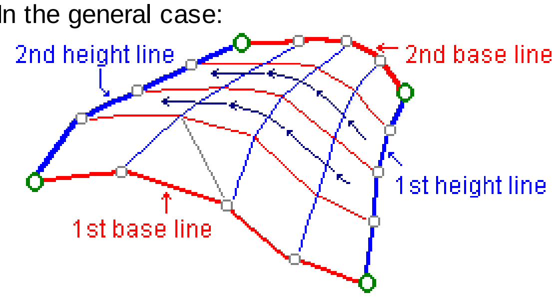
Elements 41 to 58 are created automatically.

3.6.4 Surface

Automatically generate a surface of elements by specifying a 'base line' and a 'height line' only. Unlike the grid option, these base lines do not have to be straight and the pair of opposite lines do not have to be identical.

- define a 'base' line consisting of a chain of nodes and a 'height' line, also a chain of nodes, that starts at one end of the base line.
- the program copies the base line to every level on the height line, ***generating nodes*** and connecting them with elements.
- if the base line is a semi-circle and the height line is a perpendicular line, the program generates a half cylinder; if the second base line is a semi-circle with a smaller radius, the surface is conical.

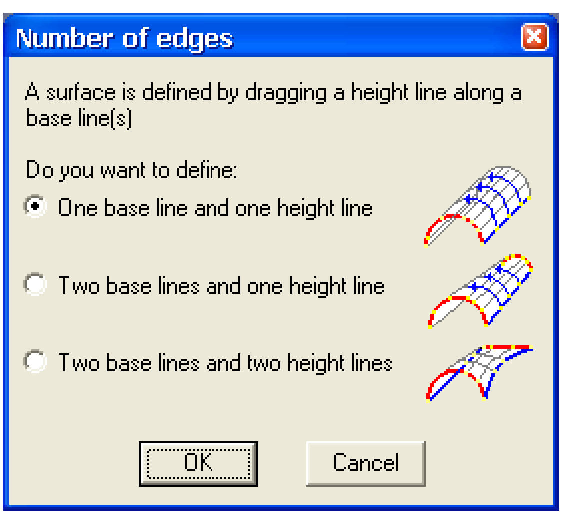
In the general case:



The program 'drags' the 1st height line along the two base lines, interpolating the coordinates of the nodes that it creates

- the number of nodes on the two base lines must be identical; the number of nodes on the two height lines must be identical
- the program creates rectangular elements only if the four nodes are planar; otherwise two triangular elements will be generated

Most models are more regular and the program has simplifying options:



Define:

One base line and one height line

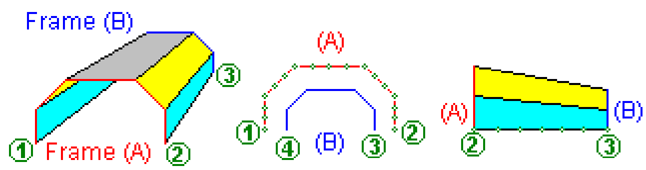
The second base line is identical to the defined one; the two height lines are also identical. For example, if the base line is a semi-circle and the height line is straight, the program will generate a half-cylinder

Two base lines and one height line
The two height lines are identical. For example, if the base lines are semi-circles with different radii and the height line is straight, the program will generate a truncated half-cone

Two base lines and one height line
This is the general case described above

Example:

Create the following space model:

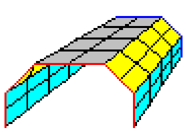


Frame (A) and Frame (B) are the two base lines and line (2)-(3) is the height line.

- define the nodes in **Frame (A)** using any of the Nodes--Line/Chain options
- define the nodes in **Frame (B)**; note that the number of nodes on Frames (A) and (B) must be identical.
- define the nodes along line (2)-(3)
- select the 

option
- select **a 3D surface by selecting nodes along 2, 3 or 4 edges**
- select **Two base lines and one height line**
- "**Select the start node of the base line**": click and highlight node (1)
- "**Select the next node in the line or press [End selection]**": click and highlight all of the nodes in Frame (A) in sequence ; double click on node (2).
- "**Select start node of the height line (one of the base ends)**": click on node (2) again
- "**Select the next node in the height line or press [End selection]**": click and highlight all of the nodes on line (2)-(3) in sequence; double click on node (3).
- "**Select the next node in the base line or press [End selection]**": click and highlight all of the nodes on line Frame (B) in sequence; double click on node (4).

The program creates the following model (fewer elements have been drawn for clarity):



3.6.5 Mesh

The Mesh option is a more powerful and extended version of the Surface - Grid 278 option. The Mesh option generates both the elements ***and*** the nodes within a user-defined area:

The mesh area is specified by defining a 'perimeter' which joins existing nodes. The perimeter consists of straight lines connecting nodes or arcs. It may be of any shape and 'holes' may be defined inside it. Note that the mesh definition may be revised at any time.

The program automatically generates elements of a specified size within a user-defined perimeter and generates new nodes if required.

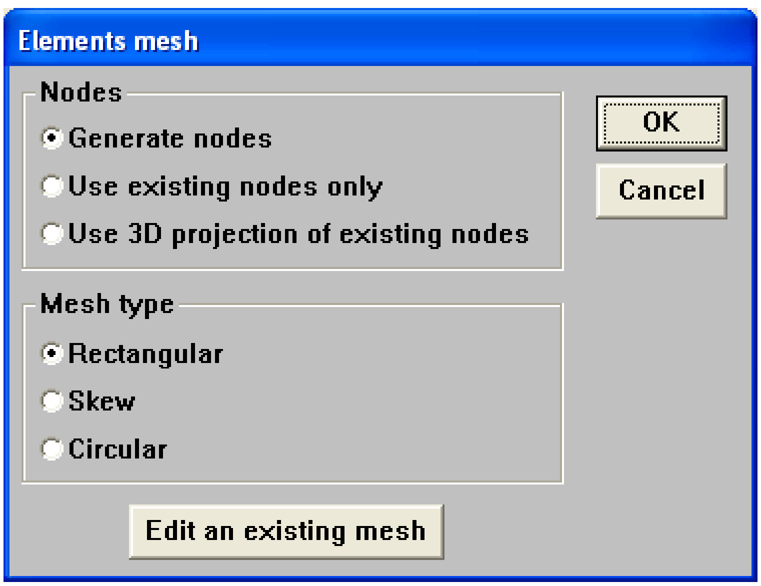
- the perimeter circumference may consist of straight lines or arc joining nodes
- the element shape may be specified as orthogonal or skew, or a circular pattern may be generated.

- holes may be defined in the perimeter area.
- the generated elements are connected properly to existing elements bordering the perimeter area.
- the grid is adjusted to include existing nodes within the perimeter area.
- existing meshes in the perimeter area are automatically erased.
- The program recognizes existing beams within the mesh contour and tries to form element boundaries along the beam lines.
- When a contour line of a new mesh is colinear with a contour line of an existing mesh, the program generates the new mesh without creating new nodes on the common line.
- If a mesh in the main model is supported by a 'wall' made of elements (not a wall element), the mesh element now connects to the existing 'wall' nodes.

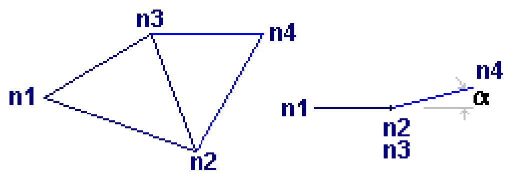
The program initially draws a preliminary mesh over the contour area. These preliminary mesh lines may then be moved by the user. The program then creates quad and triangular elements from this mesh.

Two alternatives are available for defining the element size:
- generation of elements of a ***user-specified*** size within the perimeter. ***The program generates new nodes if required***.
- automatic element generation, where the elements simply connects the existing nodes, similar to the Grid option.

Three alternatives are available for specifying the shape and arrangement of elements within the mesh area:
- Rectangular: the program generates as many rectangular elements as possible.
- Skew: the program generates parallelogram elements whose sides are parallel to user-defined directions.
- Circle: the program generates parallel arcs of elements and tries to maintain a uniform element size.



Note:
- The four generated nodes for each element must be co-planar. If they are not, the program divides the quad into two triangles:



- if α > 0.3°, the program creates two triangular elements.

3.6.5.1 Node options

Generate nodes

The mesh area is specified by defining a 'perimeter' which joins existing nodes. The perimeter consists of straight lines connecting nodes or arcs. It may be of any shape and 'holes' may be defined inside it. Note that the mesh definition may be revised at any time.

The program initially draws a preliminary mesh over the contour area. These preliminary mesh lines may then be moved by the user. The program then creates quad and triangular elements from this mesh.

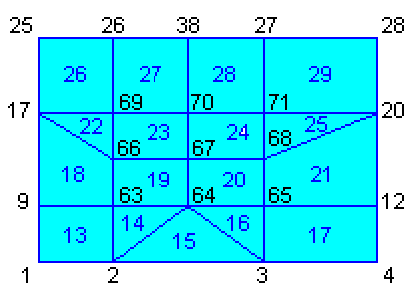
The option is best illustrated by examples:
Example 1 294 : A typical concrete floor slab; an orthogonal mesh is generated.
Example 2 296: A wedge with an arc on the perimeter; a circular mesh is generated.

Use existing nodes

Define a 'perimeter' which joins existing nodes, similar to Generate nodes 284. The program generates elements by connecting the existing nodes only and does not generate any new nodes.

- All nodes (selected and used) must lie on the same plane.
- ***All*** internal nodes are used.

Example:



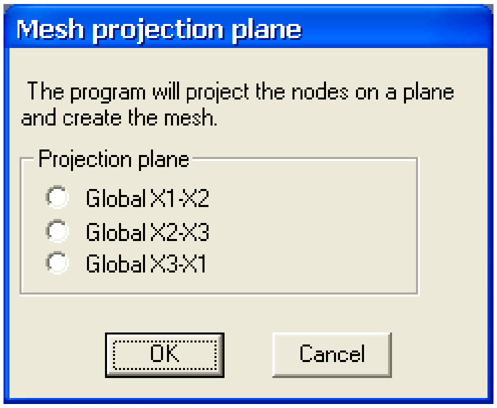
The perimeter was created by selecting nodes 1-4-28-25-1. (all nodes are 'existing').

Use 3D projection

This option is available for space models, and is similar to Use existing nodes only 284.

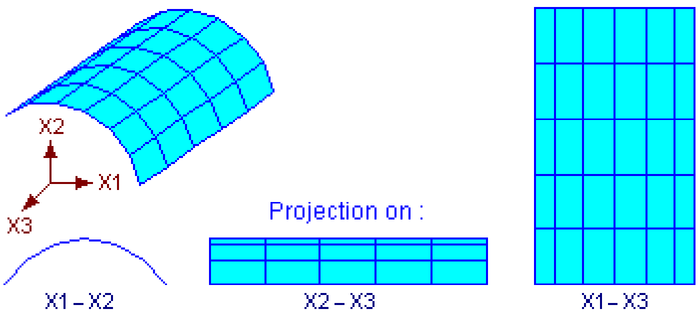
- Define a 'perimeter' which joins existing nodes as explained in Generate nodes 284. The program generates elements by connecting the existing nodes only and does not generate any new nodes. ***All*** existing nodes are used.
- The program projects all of the nodes ***displayed*** onto one of the global planes and connects the adjacent nodes; use the "Remove" option to temporarily delete nodes from the display.

Select the global plane:



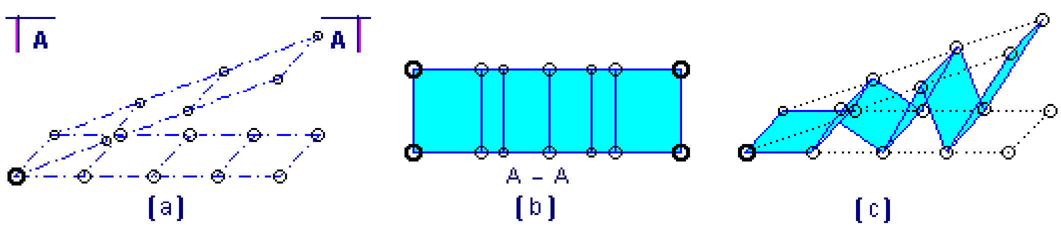
Note that the model does not have to be rotated to the selected plane.

For example, in the shell roof below, the roof must be projected on the X1-X3 plane for the elements to be generated correctly.



Note:

- the element generation may be incorrect if nodes on more than one plane are displayed simultaneously. The following model (a) is projected on a global plane as shown in (b) and the elements are generated between adjacent nodes. The program creates the erroneous mesh displayed in (c). The mesh should be generated separately for each plane.



3.6.5.2 Type

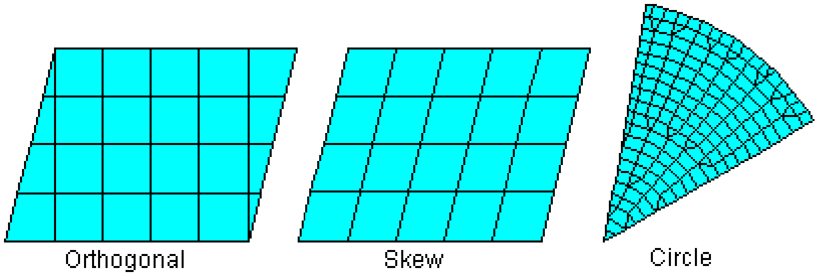
Three alternatives are available for specifying the shape and arrangement of elements within the mesh area:

Rectangular: the program generates as many rectangular elements as possible. The grid may be rotated to any angle; triangular elements may be generated along the mesh perimeter or around existing nodes.

Skew: the program generates parallelogram elements whose sides are parallel to user-defined directions.

Circle: the program generates parallel arcs of elements and tries to maintain a uniform element size. The program generates nodes on arcs. As the radius to the arcs increases, the program increases the number of nodes on the arc, maintaining a spacing between them approximately equal to the spacing between the arcs.

For example:



3.6.5.3 Define

To define a mesh:

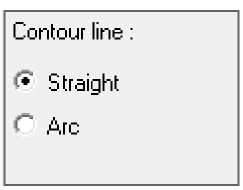
- define the mesh contour 286
- define the mesh parameters 287
- revise the mesh line spacing or move the lines 292 (horizontal/vertical)

For examples refer to:
Example 1 294 : A typical concrete floor slab; an orthogonal mesh is generated.
Example 2 296: A wedge with an arc on the perimeter; a circular mesh is generated.

3.6.5.3.1 Contour

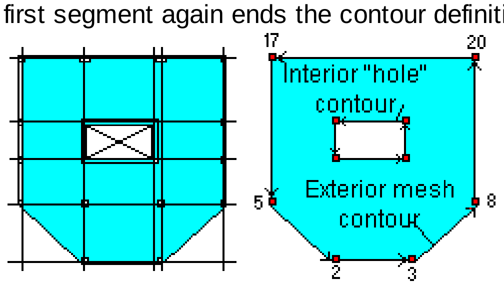
The mesh area is specified by defining a 'perimeter' contour which joins existing nodes. The perimeter consists of combinations of straight lines and/or arcs defined between nodes. It may be of any shape and 'holes' may be defined inside it.

- select the segment type from the following side menu:

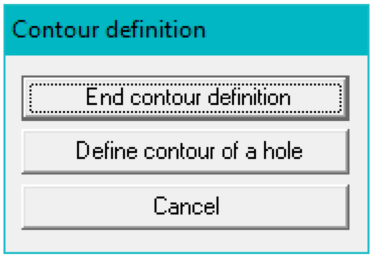


Note that **Arc** may be selected even if the mesh type is **Rectangular** or **Skew**; similarly **Straight** may be specified for **Circular** meshes.

- Select the nodes at the start and end of the current segment. Then define additional segments, where the end node of the last segment is the start node of the first segment. Selecting the start node of the first segment again ends the contour definition.



- Define openings in the contour or end the contour definition:

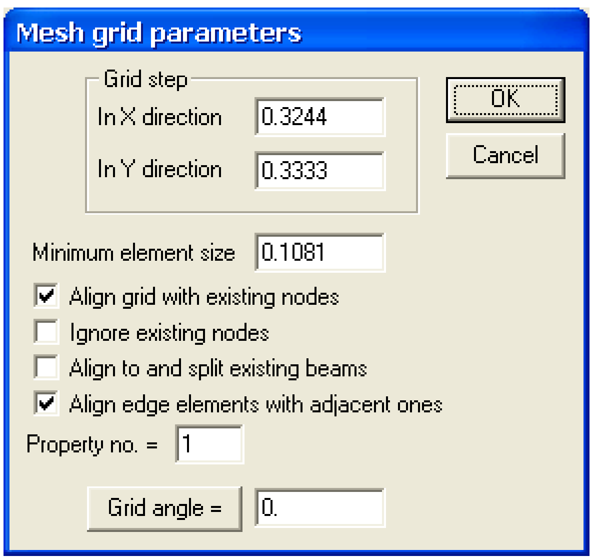


Select:

- **End contour definition**
  Select this option if there is no area within the contour (e.g. stair openings) where elements are **not** to be generated. If openings exist, select **Define contour of a hole.**
- **Define contour of a hole**
  Define the contour of an opening using the same methods as for the main contour. More than one hole may be defined within the main contour. Select **End contour definition** when all holes have been defined.

3.6.5.3.2 Parameters

Specify the mesh parameters; the program creates a preliminary mesh for the contour area according to the mesh parameters specified in this menu.



Specify the parameters and click  to continue; the program then superimposes a ***preliminary*** mesh on the contour area according to the specified spacing and options.

Grid step

Specify the size of the elements in the contour area. For example, if 1.0 x 1.0 elements are adequate for our example, set **in X direction** and **in Y direction** equal to 1.00.

Minimum element size

Increasing this value prevents the generation of small elements; if a generated node on the mesh is less than the distance specified in this option from an existing node (x or y projection), then the program does not generate the node.

Align with nodes

- ☑ The program modifies the grid spacings so that lines pass through existing nodes (the distance between lines will not be less than the minimum element size), i.e. not all elements will have the same dimensions.
- ☐ The program maintains the grid step and the grid lines do not pass through the existing nodes. The existing nodes are used as element corner nodes, but irregular shaped elements are created.

The significance of this option is illustrated in the example below and in Example 1 294 and Example 2 296.

Ignore existing nodes

Existing nodes may lie ***within*** the contour area. These nodes may be connected to elements or may be ignored, i.e only the generated mesh nodes are used. Note that all existing nodes on the contour ***boundary*** are ***always used***, no matter what is specified in this option.

- ☐ All existing nodes in the contour area will be connected to elements.
- ☑ All existing nodes in the contour area will be ignored.

The significance of this option is illustrated in the example below.

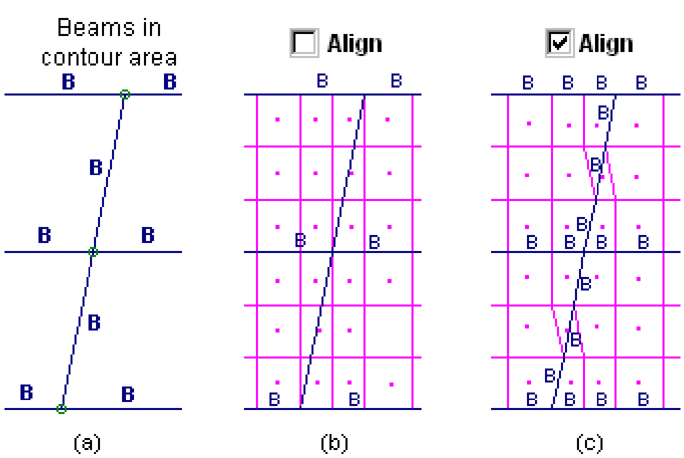
Align with and split existing beams

Select one of the following options:

- ☐ ignore existing beams when creating the mesh
- ☑ align the element boundaries with existing beams and split the existing beams according to the element corner nodes

Example:

The beams in the contour area are shown in Figure (a):



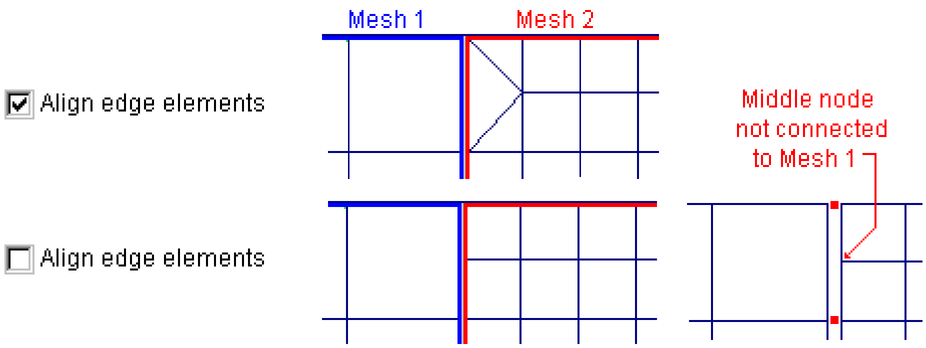
- ☐ referring to Figure (b), no new beams are created by the mesh option. The beams are connected to the elements only at the beam end nodes. Note the elements generated above the diagonal beams - the program ignored the beams when generating them.
- ☑ referring to Figure (c), all of the original beams were split at the element corner nodes. Note how the elements are aligned with the diagonal beams.

Align edge elements with adjacent ones

The program always connects the elements generated by a Mesh command with any existing larger/

smaller elements adjoining the new mesh contour. Setting this option will prevent unconnected nodes along the common boundary of the new and old meshes

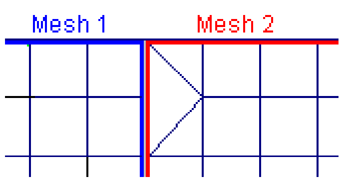
The reason is illustrated by the following example where a more refined "Mesh 2" is generated adjacent to the existing 'Mesh 1'. Setting the option to ☐ generates a node along the common line that is connected only to 'Mesh 2'



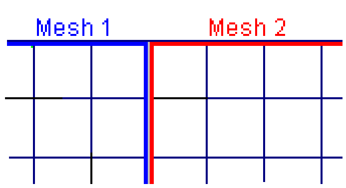
It is recommended that the option be set to ☑ ***when generating a new mesh***.

This option should be set to ☐ only when editing existing meshes. For example, refine 'Mesh 1' in the upper figure to the same density as 2:

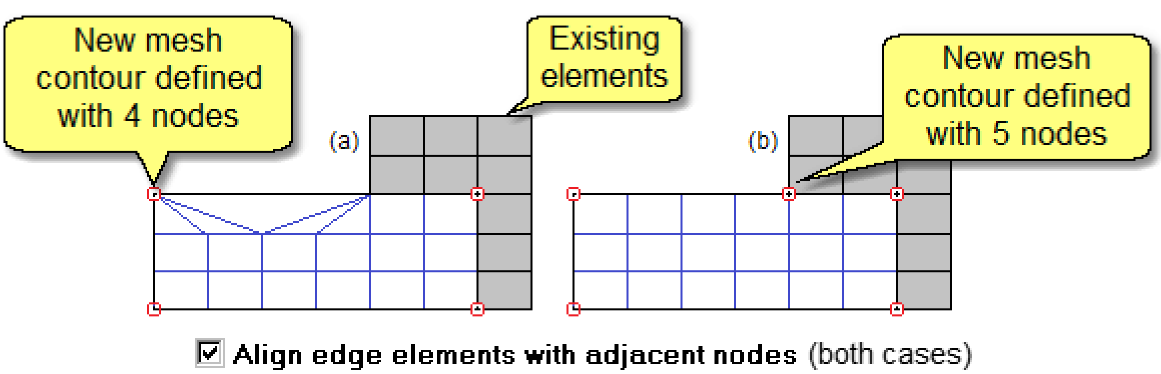
- edit 'Mesh 1', revise the dimension parameters and set the option to ☐ to avoid generating triangular elements opposite the triangular elements in Mesh 2.



- Then edit mesh 2, keep the same parameters and set the option to ☑



Note that the program searches for existing elements separately along each line of the contour that defines the mesh area. Therefore the mesh contour nodes must be selected carefully. For example:



(a) for the top contour line in the new mesh, the program finds the three nodes in the existing mesh and connects all of the adjacent new elements to them, creating the unwanted triangular elements !

(b) The top contour line is defined in two parts as shown. There are no existing nodes adjacent to the left top contour line so the program generates a regular mesh as shown.

Property no.

Specify the property group to assign to all of the generated elements.

Grid angle

The element edges are drawn parallel and perpendicular to the base line of the grid. The base line of the grid may be defined in two ways:

- type in the value of the angle in the text box, where the angle is measured from a base line as follows:

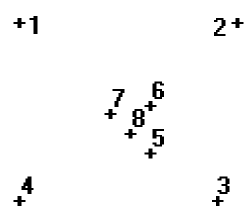
plane models	-	parallel to X1
space models	-	parallel to the projection of X1 on the plane. If the plane is parallel to X1, the base line is parallel to X2.

The grid angle is measured counterclockwise from the base line.

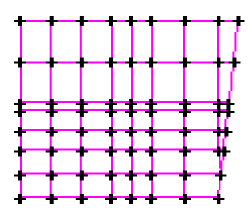
- click the [Grid angle =] button and select two existing nodes defining the base line.

Example:

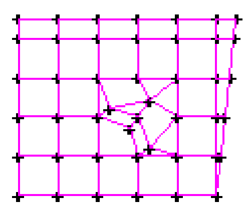
- Nodes 1-2-3-4 define the contour
- Nodes 5, 6, 7, 8 are located within the contour boundary



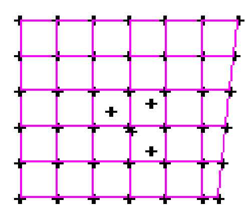
☑ **Align grid with exiting nodes**
☐ **Ignore existing nodes**



☐ **Align grid with exiting nodes**
☐ **Ignore existing nodes**

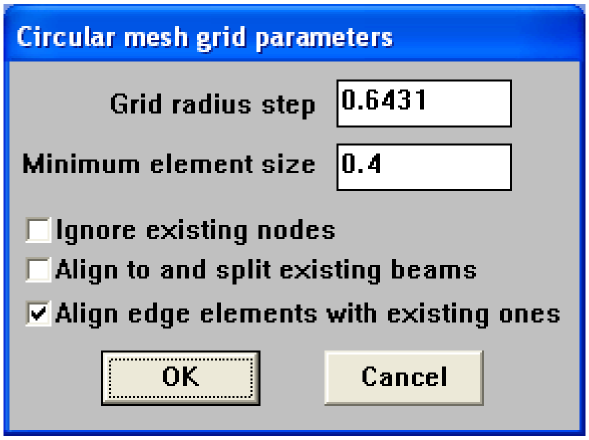


☐ **Align grid with exiting nodes**
☑ **Ignore existing nodes**



3.6.5.3.2.1 Parameters - circle

The program creates a preliminary mesh for the circular contour area according to the mesh parameters specified in this menu.



Specify the parameters and click OK to continue; the program then superimposes a ***preliminary*** mesh on the contour area according to the specified spacing and options.

Grid radius step

Specify the spacing between the concentric arcs. The radial lines are generated so that approximately equal sided quadrilaterals are created. Whenever the length of the longest size is 150% greater than the spacing specified here, the program adjusts the location of the radial lines.

For example, set the Initial node step to 0.5 if approximately 0.5 x 0.5 elements are adequate for our example.

Minimum element size

Increasing this value prevents the generation of small elements; if a generated node on the mesh is less than the distance specified in this option from an existing node (x or y projection), then the program does not generate the node.

Ignore existing nodes

Existing nodes may lie ***within*** the contour area. These nodes may be connected to elements or may be ignored, i.e only the generated mesh nodes are used. Note that all existing nodes on the contour ***boundary*** are ***always used***, no matter what is specified in this option.

☐ all existing nodes in the contour area will be connected to elements.

☑ all existing nodes in the contour area will be ignored.

Align with and split existing beams

Refer to Mesh parameters - align with beams 288.

Align edge elements with existing ones

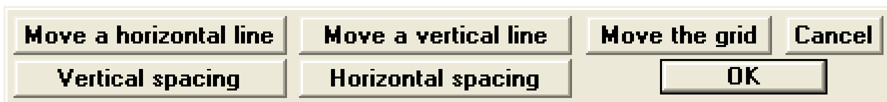
Refer to Mesh parameters - align edge elements 288.

Refer to:
Example 2 296: A wedge with an arc on the perimeter; a circular mesh is generated.

3.6.5.3.3 Move lines

The lines in the preliminary mesh may be modified:
- move a single horizontal or vertical line
- revise the number of spacings between any two selected lines, i.e. add/delete lines in specific areas
- move the entire grid by a specified distance.

For an orthogonal mesh:



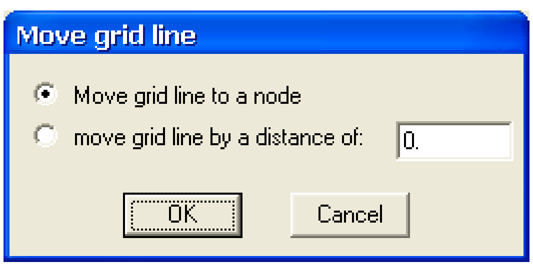
For a circular mesh:

Select 

to create the ***final*** mesh after the following corrections have been completed.

Move lines

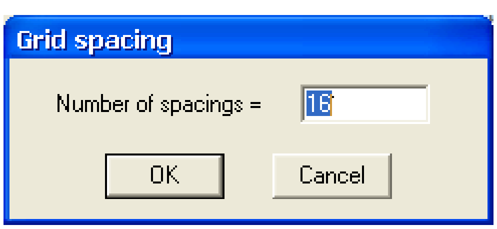
Use this option to move a line to a different location. Select a grid line and then specify the new location; two options are available:



Move grid line to a node: select an existing node
Move grid line by a distance: enter a value

Revise spacing

Use this option to add new lines to the grid. Select two parallel grid lines (not necessarily adjacent); define the new number of spacings between them:



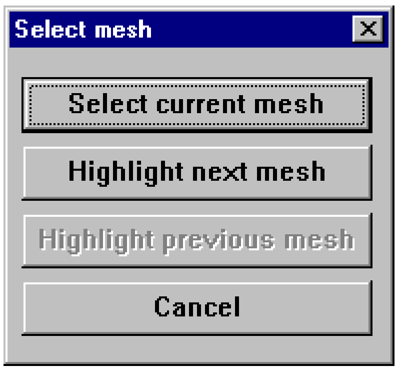
Note that the distance between the two selected grid lines will always be divided into ***equal*** spacings.

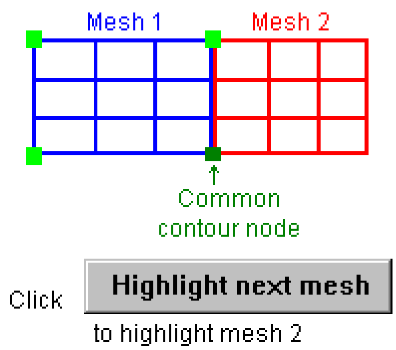
For examples refer to:
Example 1 294 : A typical concrete floor slab; an orthogonal mesh is generated.
Example 2 296 : A wedge with an arc on the perimeter; a circular mesh is generated.

3.6.5.4 Edit

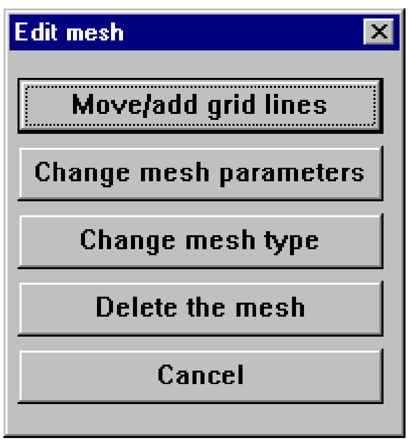
Edit an existing element mesh; change the mesh type, the mesh parameters, or manually move the grid lines, i.e. jump to any stage of the mesh definition process. Note that the mesh contour cannot be revised.

- Select the mesh by selecting any node in the contour. If the node belongs to two or more meshes:





- Revise the mesh:



Move/add grid lines

Refer to Define mesh - move lines 292

Change mesh parameters

Refer to Define mesh - parameters 287

Change mesh type

Refer to Mesh type 285

Delete the mesh

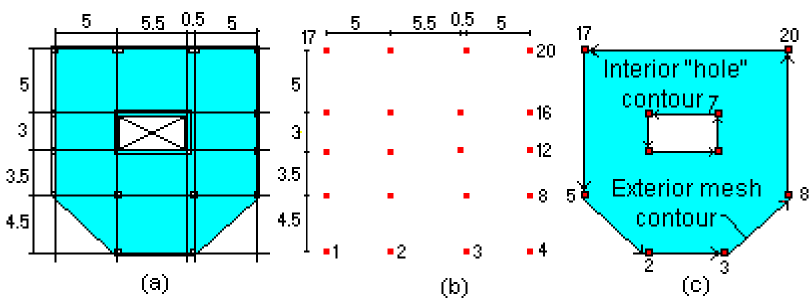
Select one of the following options:

Delete ...	the mesh definition and the elements in the mesh are deleted
Retain ...	the mesh definition is deleted but the elements remain in the model.

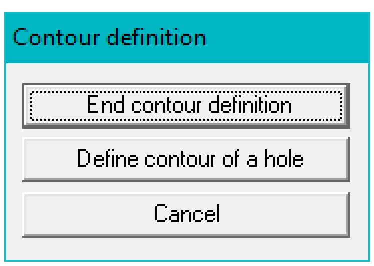
3.6.5.5 Mesh - Example 1

Create a model of the flat slab shown in Figure (a) with 1.0x1.0 elements (approx.).

- First, define the nodes shown in Figure (b).

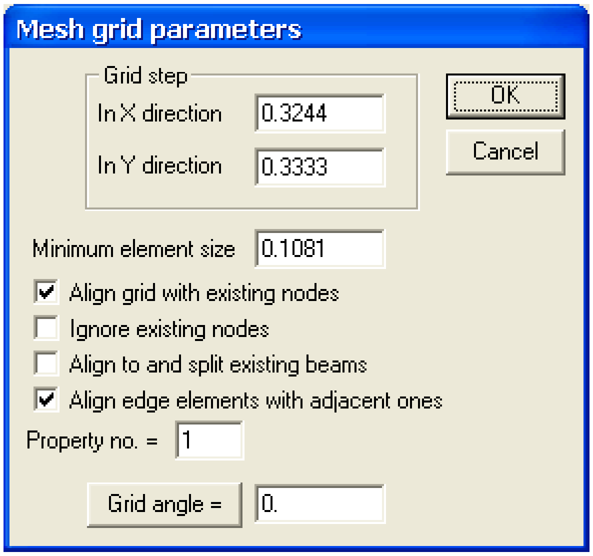


- Select the **Mesh** option and begin defining the contour:
  Move the mouse and highlight the corner nodes as shown in Figure (c).
- Define the opening:

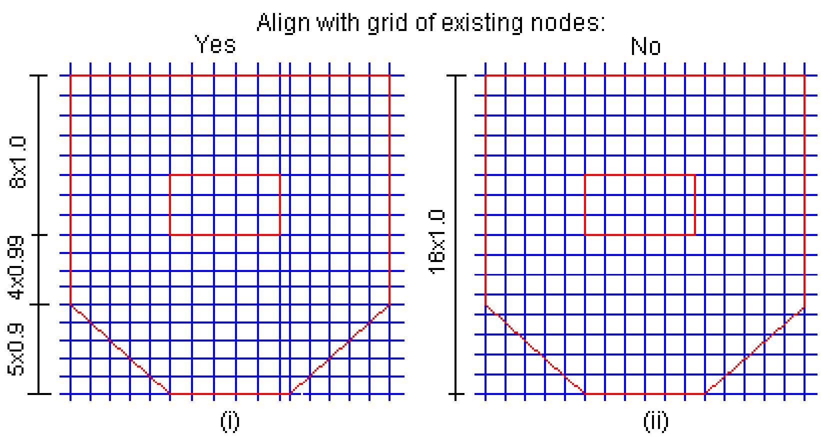


Select:
- **Define contour of a hole** and select the four corner nodes of the opening as shown in Figure (c).
- **End contour definition** when the above menu is displayed again.

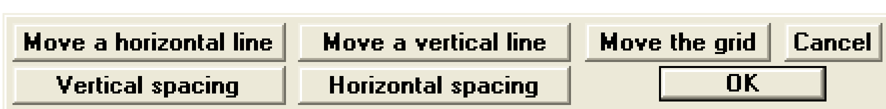
The program now displays the mesh parameter menu; define the parameters as shown:



The program now superimposes a mesh on the contour area according to the specified spacing and options. Two variations are shown below according to the parameter specified for **Align grid with existing nodes**:



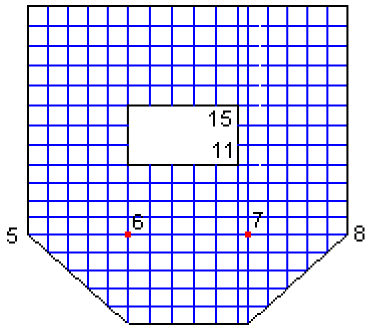
The mesh line locations may be modified:



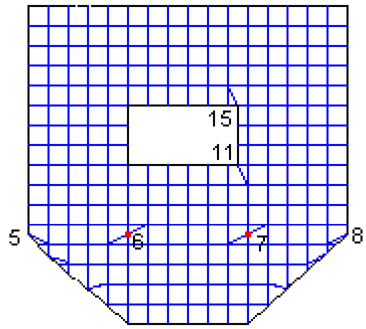
Select 

to create the mesh after all corrections have been completed.

For the two variations in the figure above, the following meshes are created:

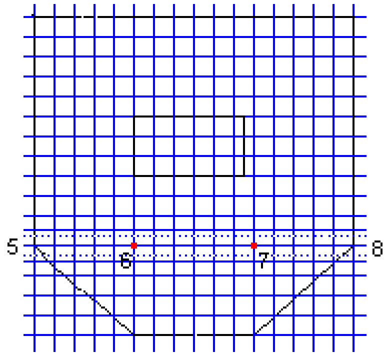


(iii)

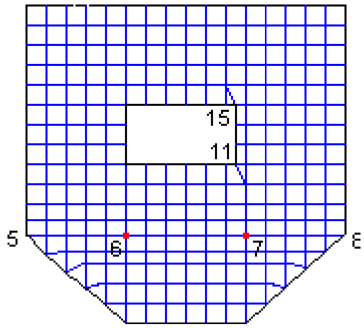


(iv)

For the variation shown in Figure (iii), the dotted lines could be aligned with nodes 5,6,7,8 as shown in Figure (v) below using the "Move line" and "Change spacing" options. The program would then generate the elements as shown in Figure (vi).



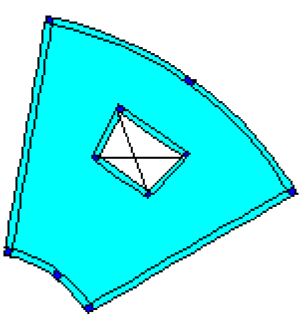
(v)

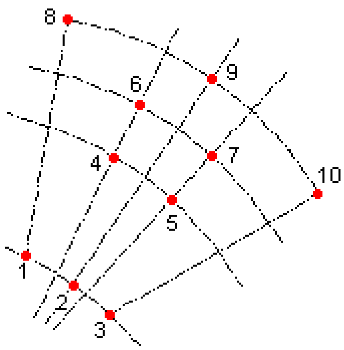


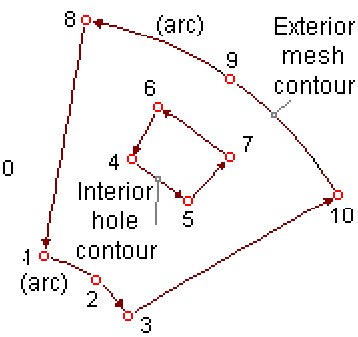
(vi)

3.6.5.6 Mesh - Example 2

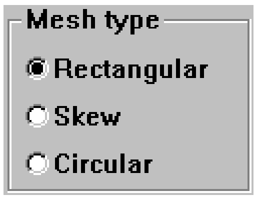
Define a model for the arc shaped floor below containing elements approximately 0.5 x 0.5



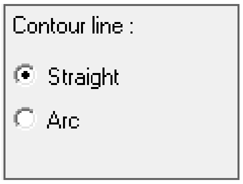




- First define the nodes shown in Figure (b) above.
- Select the **Mesh** option and begin defining the contour:
- Specify the mesh type:

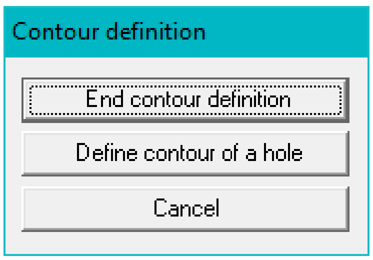


- Define line 1-2-3; specify the line as an **arc** in the menu at the side of the screen:



Select node 1 as the start node of the arc; node 3 as the end node of the arc; node 2 as the third node on the arc. The program draws arc 1-2-3.

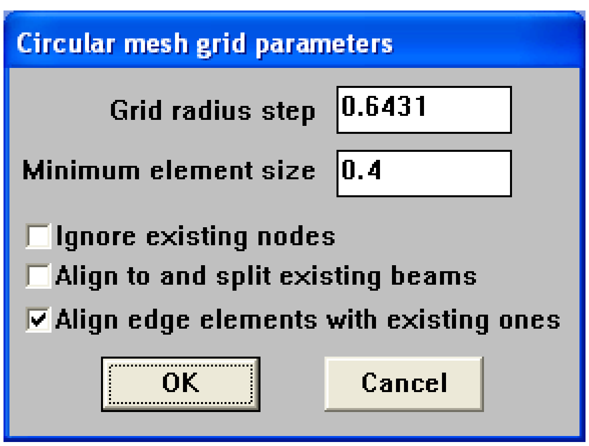
- Note that the side menu reverts to **straight;** select node 10 as the next node on the contour.
- Define arc 8-9-10 (similar to arc 1-2-3) and close the contour by selecting node 1.
- Define the opening:



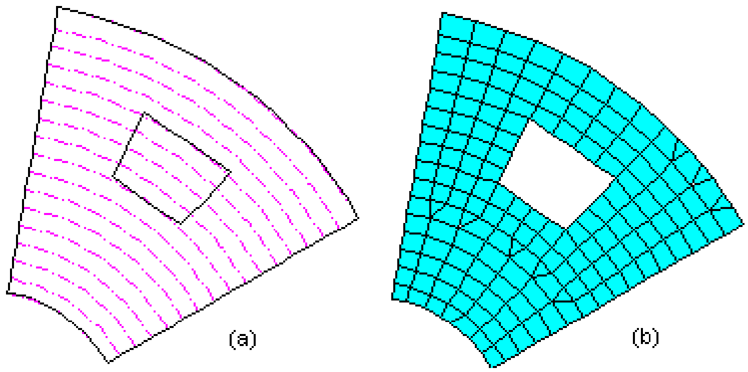
Select:

- **Define contour of a hole** and select the four corner nodes 4-5-6-7 of the opening as shown in Figure (b) above.
- **End contour definition** when the above menu is displayed again.

- Define the centre point of the parallel arcs of the elements to be generated by identifying any of the arcs; select any three nodes on an arc, e.g. 8,9 and 10 (or 1, 2 and 3)
- Specify the mesh parameters for the circular mesh:



- The program now superimposes a mesh on the contour area according to the specified spacing and options (Figure a):



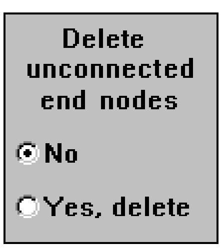
The generated arcs may be relocated or the spacing between any two arcs may be adjusted:

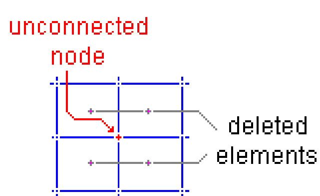
- **Move an arc**
  select an arc and move by a defined distance or to an existing node.
- **Change arc spacing**
  select any two arcs (not necessarily adjacent) and enter the revised number of spacings; the program redivides the distance (equal spacing) between the two selected arcs.
- **OK**
  To generate the grid of elements. The grid created for this example is shown in Figure (b) above.

3.6.6 Delete elements

Select elements using the standard Element Selection option.

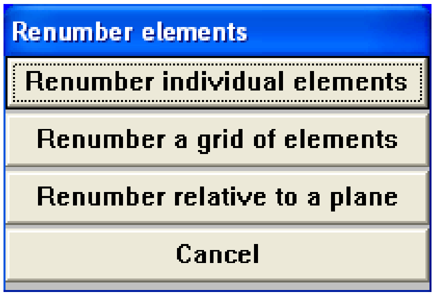
Note that nodes that are not connected to the model after the elements are deleted may be deleted from the model at the same time; set **Yes, delete** in the side menu.

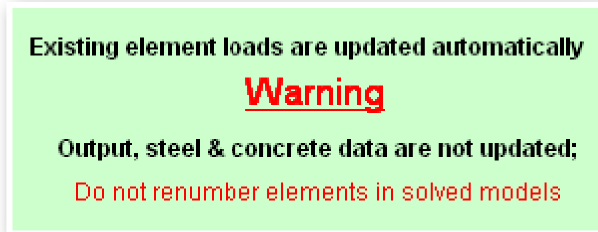




3.6.7 Renumber

Use this option to renumber existing elements:

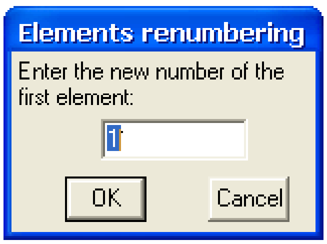




Individual elements

Select one or more elements using the standard Element selection option. Note that the order that the elements are selected is important; they are renumbered in the order that they are selected.

Type the new number of the first element selected:

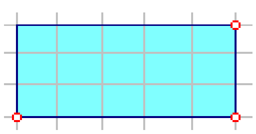
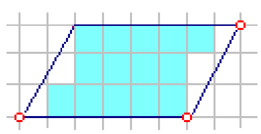


All of the elements selected are renumbered sequentially. If the program discovers that a number in the sequence has already been assigned to another element, the program assigns the original number of the selected element to that element.

For example:
- elements 41, 42 and 43 are selected (in that order).
- 75 is specified as the new number for 41
- the elements are renumbered 75,76 and 77 respectively
- element 76 is an existing element; it is renumbered 42.

Grid of elements

Define the grid by pointing to the three corner nodes defining it; the three nodes define a rectangle or a parallelogram. ***All*** elements with ***all*** end nodes lying in the parallelogram defined by the three nodes are renumbered.

Type the new number of the first element (lower-left corner):

All of the elements selected are renumbered sequentially. If the program discovers that a number in the sequence has already been assigned to another element, the program assigns the original number of the selected element to that element.

Plane

Renumber all elements on selected planes. This option is handy for renumbering an entire model or parts of a model consisting of more than one plane. Note that the planes do not have to be parallel.

- select the elements to be renumbered using the standard element selection option
- define a plane that specifies the renumbering order; the plane is defined by selecting three existing nodes.
- specify the new number of the first element

The renumbering order is determined as follows:

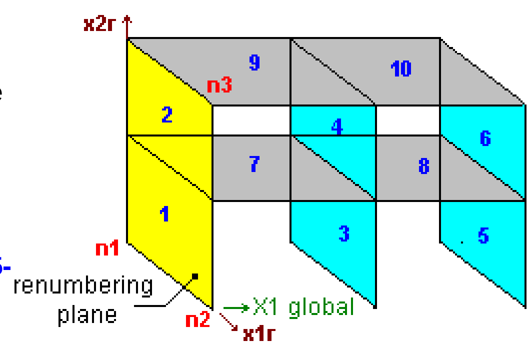
- the first two nodes define the ***x1r*** axis of the renumbering plane; the third node defines the ***x2r*** axis of the plane; the ***x3r*** axis is determined by the right-hand rule.
- the program sorts the elements according to the angle between their plane and the renumbering plane, starting with the elements closest to the renumbering plane. Note that if there are elements on both sides of the plane, the program first selects all elements on one side, then all elements on the other side.
- for elements on the same plane, the program then sorts according to the coordinates of the element center beginning with the smallest values.

All of the elements selected are renumbered sequentially. If the program discovers that a number has already been assigned to another element, the program assigns the original number of the selected element to that element.

Example:

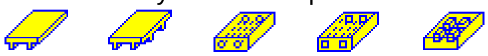
Renumber the following space frame; the renumbering is to start on the planes perpendicular to X1 global.

- select nodes **n1**, **n2** and **n3** to define the renumbering plane
- specify **1** as the new number of the first element
- the elements on the **x1r-x2r** plane are renumbered first (**1-2**)
- then the elements on the parallel planes are renumbered (**3-4**) and (**5-6**)
- then the elements perpendicular to the **x1r-x2r** plane are renumbered.(**7-10**)



3.6.8 Properties

Elements may be either a plate with uniform thickness or one of the following sections:



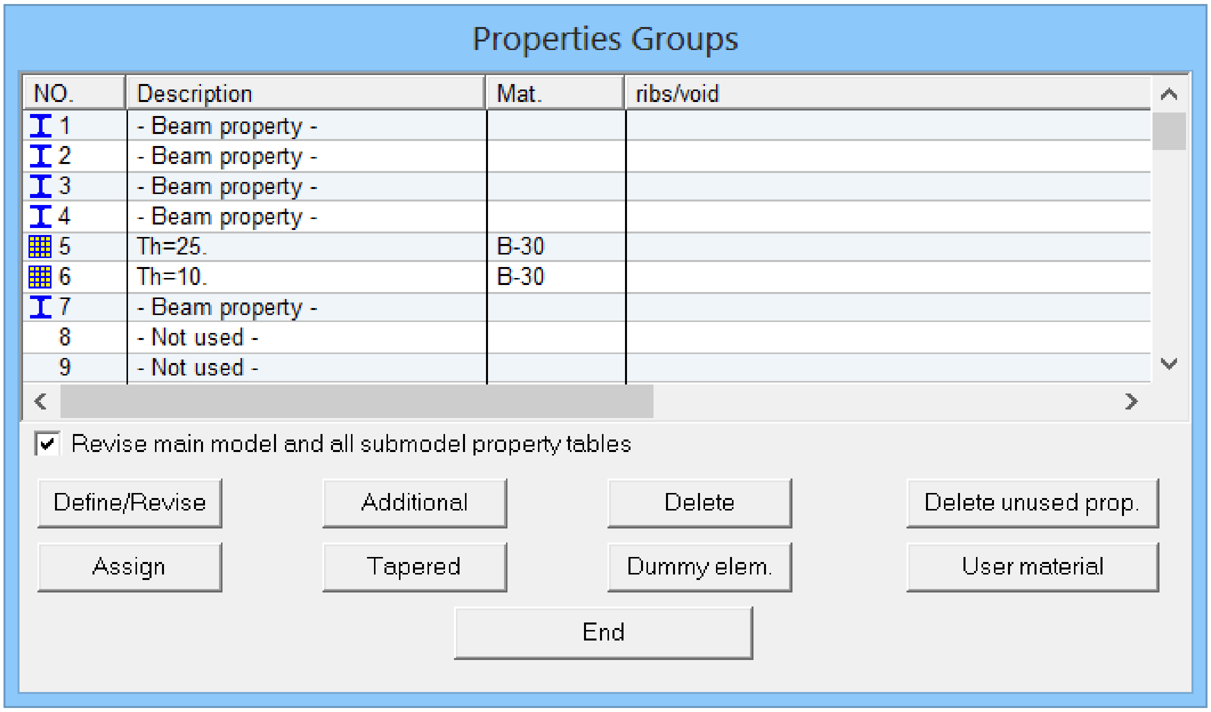
The element properties are:

- dimensions (e.g. plate thickness)
- material (may be isotropic or orthotropic)

An element may also be designated as a dummy element (zero stiffness).

Each element must be assigned to a property group. When an element is created it is automatically assigned the current **Prop** group number displayed at the bottom of the screen. The property group assigned to an existing element may be revised at any time.

Note that a property number may be assigned to an element even if the section properties have not yet been defined.



Define/revise

Define a standard plate element (uniform thickness). Refer to Element properties - define 302

Additional

Define one/two-way ribbed slabs, slabs with rectangular/round/elliptical holes, closed waffle slabs. Refer to Element properties - additional 305.

Assign

- Select and highlight a property group in the list displayed on the screen.
- Select elements that this property is to be assigned to using the standard Element Selection option.

Note:

- an **-Undefined-** property group may be assigned to elements; the section properties may be defined later.

Delete

- click and highlight one of the element properties displayed in the list box
- click the Delete button

Delete unused property

Delete all property groups that are not assigned to any beams or elements/

Dummy elements

An element may be designated as "Dummy". Dummy elements may be loaded but they do not affect the stiffness of the model and do not appear in the output tables.

For example, use a dummy element if you want to define an area load in a model that consists entirely of beam elements.

Select elements using the standard Element Selection option.

Note:

- dummy elements should be connected to the MODEL at ALL corners, i.e. dummy elements should not be cantilevered or connected to other dummy elements; the loads associated with the "unconnected" corners of the element are lost by the program.
- If you select **Property numbers** in the Display option, the letter "**D**" will be displayed in the dummy elements.

User material

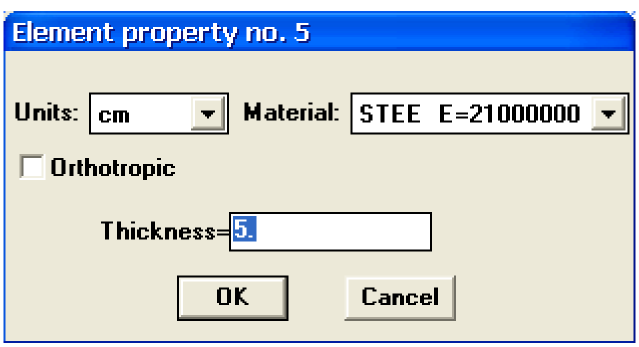
Refer to Beams - user material.

Revise main model and all submodel property tables

Refer to Beam properties.

3.6.8.1 Define/revise

- Select and highlight a property to be defined / revised from the property list.
- Click the Define/revise button
- Define the element properties:



Units, Material

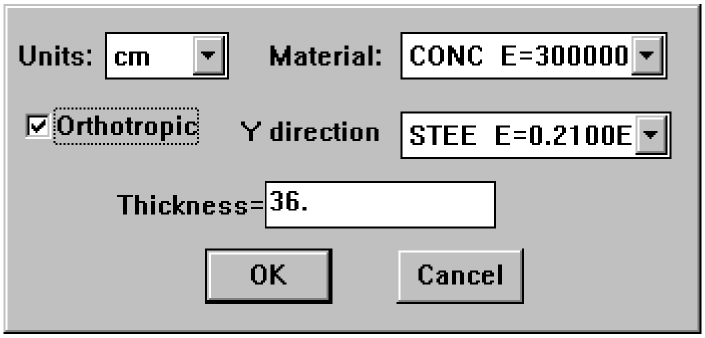
Refer to Beam properties - general

Thickness

Enter the element thickness according to the selected unit. Note that the thickness is symmetric about the centre axis of the element; refer to Offsets 309 for more details.

Orthotropic

To define an orthotropic material, set the check box to ☑. The dialog box is then revised to:



Define the Material in the Y direction, where
- 'X' refers to the direction parallel to the ***local*** element x1 axis
- 'Y' refers to the direction parallel to the ***local*** element x2 axis.

It is important to verify the uniformity of the local x1 and x2 axes of elements in orthotropic models.

The shear modulus-of-elasticity **G** and Poisson's ratio ν are calculated as follows:

- Isotropic: $G = \dfrac{E}{2(1+\nu)}$
- Orthotropic:
  - bending: $G = \dfrac{\sqrt{E_1 E_2} - E_1 \nu_{12}}{2(1 - \nu_{12}\nu_{21})}$
  - plain strain $\dfrac{1}{G} = \dfrac{1+\nu_{12}}{E_1} + \dfrac{1+\nu_{21}}{E_2}$

where:

E_1 = larger modulus-of-elasticity

E_2 = smaller modulus-of-elasticity

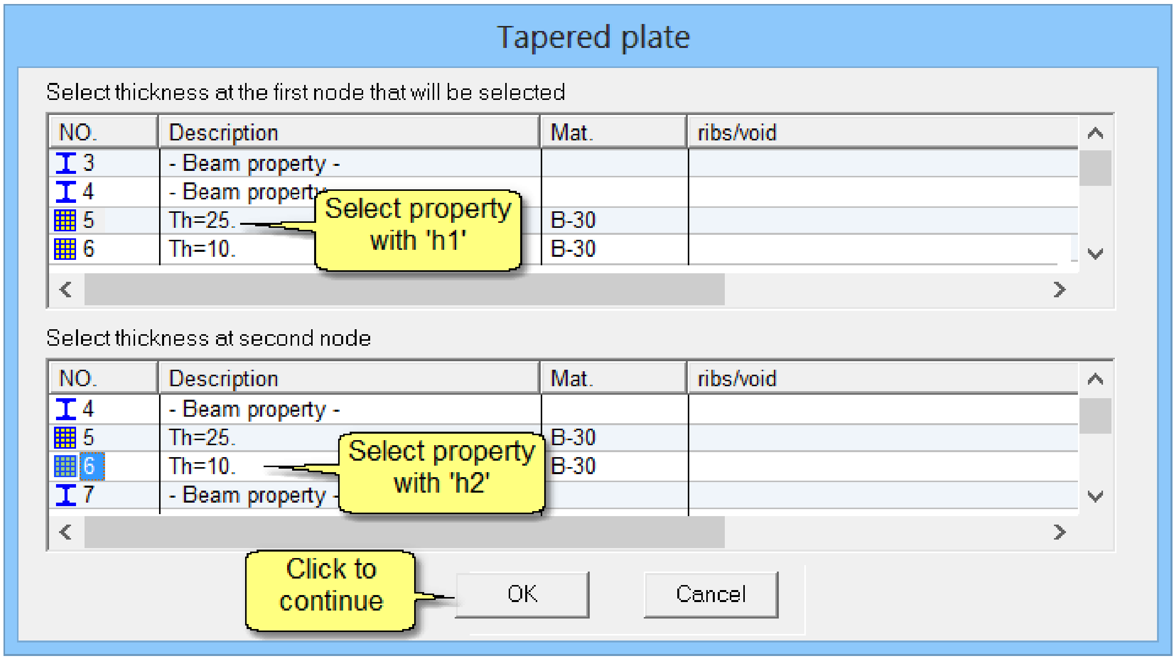
ν_{12} = Poisson's ratio of material with E_1

$$\nu_{21} = \nu_{12}\frac{E_2}{E_1}$$

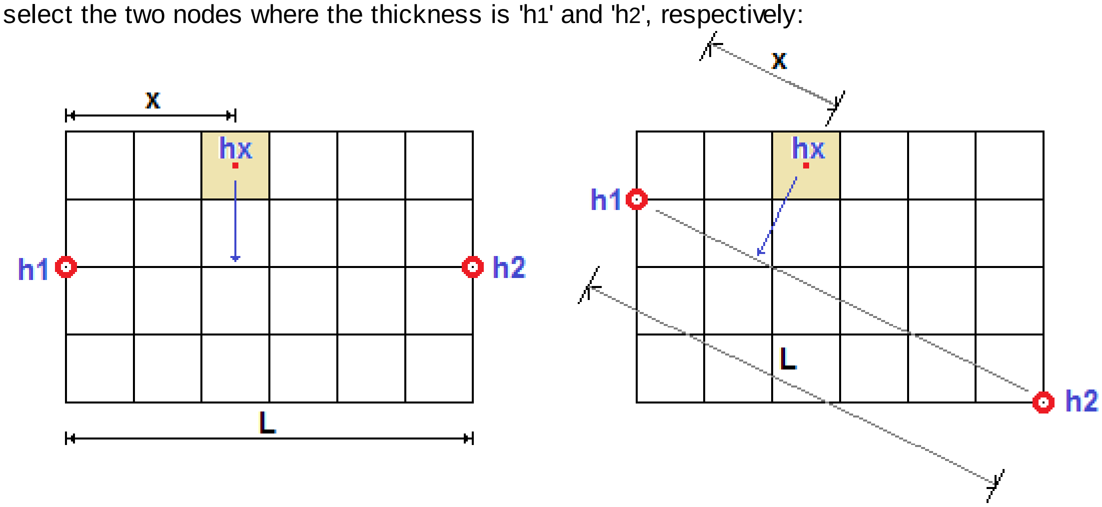
i.e. the program ignores ν of material with E_2.

3.6.8.2 Tapered elements

Create a series of elements with a linearly varying depth to model a tapered slab (the individual elements are not tapered).
- Define the two element properties, each having the thickness at opposite ends of the tapered slab ('h1' and 'h2' in the following examples).
- Click on Tapered in the properties menu.
- Select the all elements in the tapered slab
- Select the properties with 'h1' and 'h2'. For example:

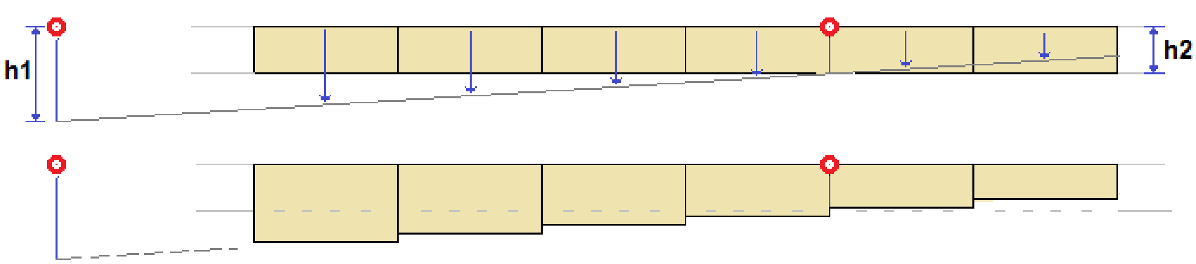


- select the two nodes where the thickness is 'h1' and 'h2', respectively:

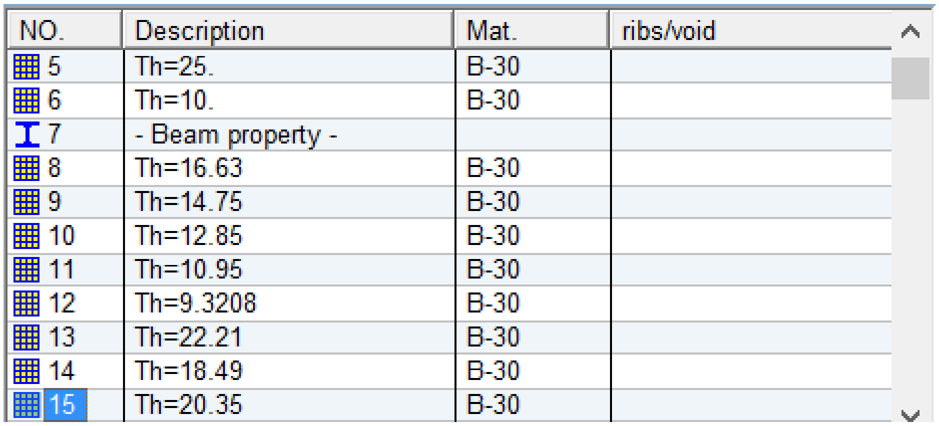


For each selected element, the program draws a perpendicular from the element centre to the line 'h1-h2 (length = L); the element thickness 'hx' is proportional to the ratio (x/L)

In section:



The program adds the new thicknesses to the property table:

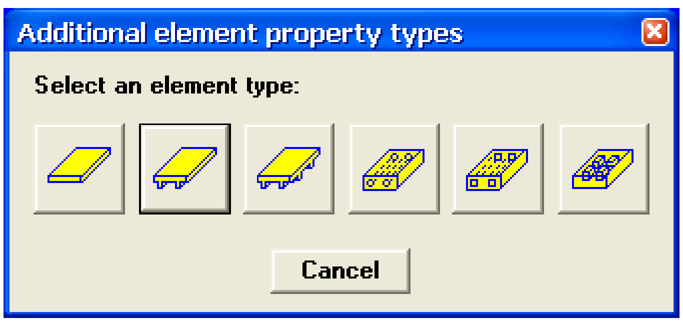


NO.	Description	Mat.	ribs/void
5	Th=25.	B-30	
6	Th=10.	B-30	
7	- Beam property -		
8	Th=16.63	B-30	
9	Th=14.75	B-30	
10	Th=12.85	B-30	
11	Th=10.95	B-30	
12	Th=9.3208	B-30	
13	Th=22.21	B-30	
14	Th=18.49	B-30	
15	Th=20.35	B-30	

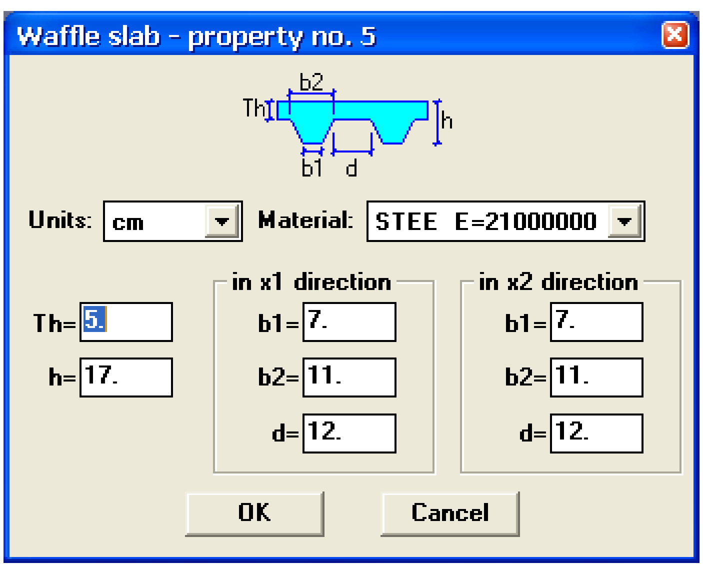
3.6.8.3 Additional

Create an element that represents a one/two-way ribbed slab, a slab with rectangular/round/elliptical holes or closed waffle slab. Note that the program only creates an element with the equivalent stiffness and does not check whether these elements "fit" dimensionally into the model.

Select the section type:



and enter the dimensions. For example, a two-way ribbed slab:



where:

- all dimensions are shown in the sketch

- "direction" refers to the element local axes.

Units, Material

Refer to Beam properties - general

Note:
- the ribs/holes are always parallel to one of the two element local axes x1,x2. Care must be taken to define a uniform local axis direction in any slab.
- for one-way ribbed slabs, the slab stiffness in the direction perpendicular to the ribs is the stiffness of the slab only (**Th**).
- for slabs with holes, the slab stiffness in the direction perpendicular to the holes is the stiffness of the top and bottom plates rigidly connected by a web with no stiffness.
- the program only creates an element with the equivalent stiffness and does not check whether these elements "fit" dimensionally into the model. To define a single rib, the element dimension perpendicular must equal **b_2+d** (refer to the dialog box above for "Waffle slab").

3.6.9 Local axes

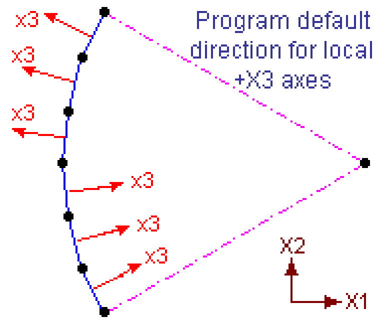
Use this option to revise the direction of:
- the local element x3 axis (the x3 axis is always perpendicular to the element plane). This option is used to reverse the direction of the local x3 axis as set by default by the program.
- the local element x1 axis. This axis can be parallel to any one of the element sides.

As explained in Coordinate Systems, the program tries to ensure uniformity of the direction of the local x3 axis so as to avoid confusion in the results. Thus all x3 axes will point in the same direction in a plane of elements.

The following is an example where the program defaults are not adequate.

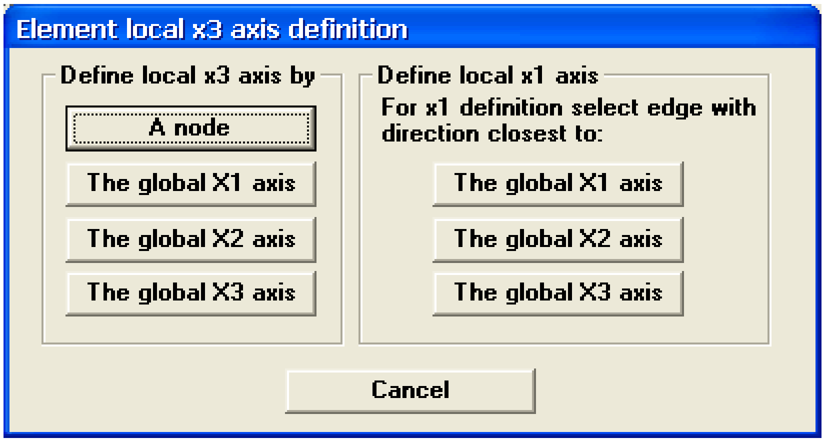
The figure shows a cylinder whose central axis is parallel to X3.

By default, the +x3 axes point in the general direction of +X2. Consequently +x3 of half of the elements point inward, while +x3 of the other half point outward. In such cases it is recommended that ***all*** +x3 point either inward or outward



Select the elements using the standard Element Selection option.

Specify the axis and direction:



x3 axis

- A node

  The positive direction of the x3 axis will point in the general direction of the node. The node cannot lie on the plane of the elements.

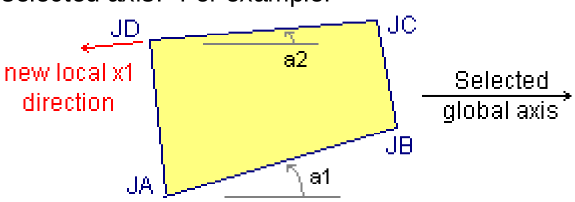
  In the example above select node 1; all the elements will point inward.

- By global axis

  The positive direction of the x3 axis will point in the general direction of the ***positive*** global axis selected. The axis cannot lie on the plane of the element.

x1 axis

Select one of the global axes; the program search for the smallest angle between the sides and the selected axis. For example:



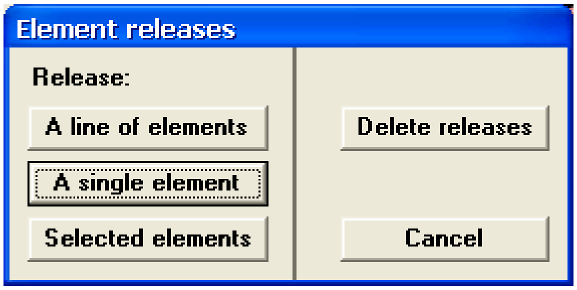
- a2 is the smallest angle so JC-JD becomes the new x1 axis.
- the direction is from JC to JD (the end node of the line).i.e. the direction that maintains the local x3 direction.
- when two lines are parallel the first of the two lines (i.e. JA-JB or JB-JC) has priority.

3.6.10 Releases

Define pinned edges for bending elements. This option is similar to the moment release option for beams.

Example:

- define pinned edges at the connection of two precast slabs; the joint is designed to transfer the vertical shear, but not moments.



Line of elements

- move the so that the first element in the line is highlighted with the ■ and click the mouse.
- select the two nodes defining the edge to be released.
- move the so that the last element in the line is highlighted with the ■ and click the mouse.

Two small circles are drawn at the ends of the edges of all elements in the line.

Single element

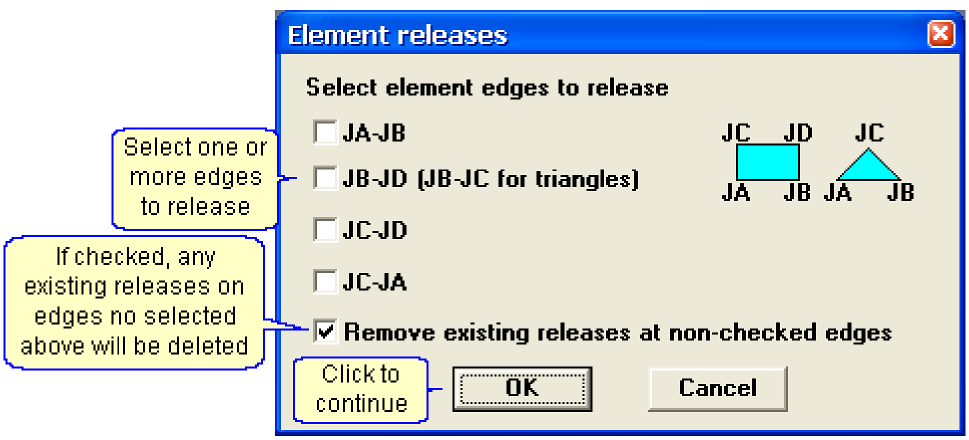
- move the so that the element is highlighted with the ■ and click the mouse.
- select the two nodes defining the edge to be released.

Two small circles are drawn at the ends of the edge.

Selected elements

Select one or more elements and one or more edges for all of them:

- select elements using the standard Element selection option.
- select the edges to release:



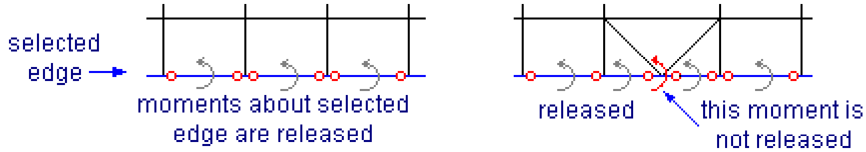
Two small circles are drawn at the ends of the edge.

Delete

Select elements with releases using the standard Element selection option. Releases along all edges of the selected elements will be deleted.

Note:

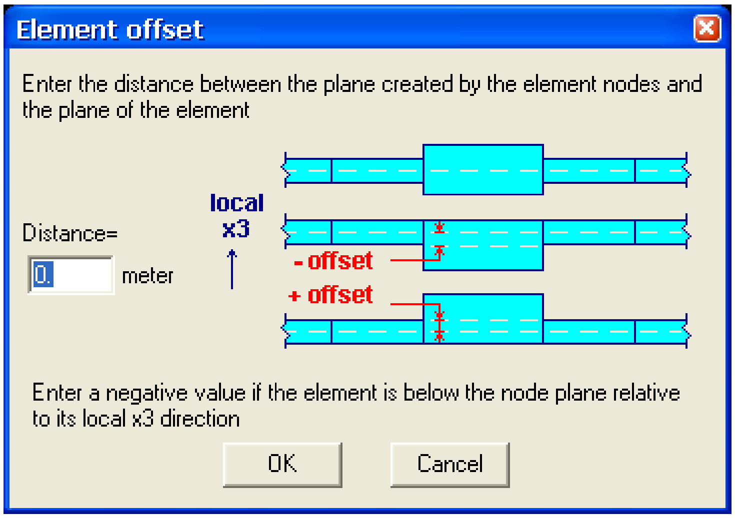
- moments ***about*** a selected ***edge*** are released; releases cannot be defined at the single node of a triangular element connected to the line. For example:



- elements with releases are less accurate than regular bending elements; the element density should be relatively greater along released lines to compensate for the loss of accuracy.

3.6.11 Offsets

Elements may be offset in a direction perpendicular to their plane. This option may be used to defined drop panels or changes in slab thickness where in both cases the top surface remains constant. The sketches in the dialog box show examples:



A positive offset value moves the element in the +x3 local axis direction.

3.6.12 Stages

Elements may be removed from the current stage or restored.

- Select elements using the standard element selection options

Note:

- inactive elements are not displayed
- new elements cannot be defined when a stage other than ***Whole model*** is active

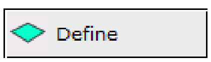
3.7 Slab

A slab element represents an area of a floor slab that has a uniform property. The Slab option is a more powerful and extended version of the Element-Mesh option. The Slab option internally generates both the elements and the nodes within a user-defined area:

The mesh area is specified by defining a 'perimeter' which joins existing nodes. The perimeter consists of straight lines connecting nodes or arcs. It may be of any shape and 'holes' may be defined inside it. Note that the mesh definition may be revised at any time.

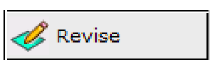
- the slab element is defined by a contour of any shape and may include openings.
- the program automatically connects the slab element to all existing columns, walls, other slab elements and regular elements that attach to it.
- if the slab is in a submodel, the program creates 'connection points' at the wall and column locations.
- when saving or solving the model, the program divides the slab into individual elements according to user-defined parameters (similar to the Mesh option).
- ***if columns, walls, etc, are moved or deleted, no changes to the slab element have to be made because the program automatically redefines the elements and connection points internally every time the model is solved***.
- The Slab elements are viewed only in Geometry and Loads. The element mesh automatically created by the program from the Slab is displayed in Results and all design modules.

The following options are available when "Slabs" is selected in the geometry Main Menu:



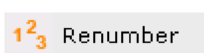
Generate a slab element. The grid outline is defined by specifying a contour, and the size of the generated elements is determined by user defined parameters.

Delete 314 slabs already defined or convert them to regular elements

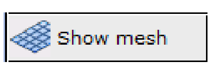


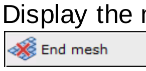
Revise the parameters of an existing slab element..

Define releases or restraints on the edges of slabs.

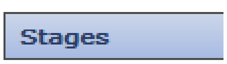


Renumber 315 slabs already defined.



Display the mesh of elements that the program creates for the slabs. Click 

to continue to the slab display.

Click 

to select a stage other than **Whole model;** different properties may be defined for each stage and slabs may be removed or restored:

'Remove' a slab from the current stage.

Restore an slab to the current stage.

Define/revise slab properties 317 and assign them to selected slabs.

3.7.1 Define

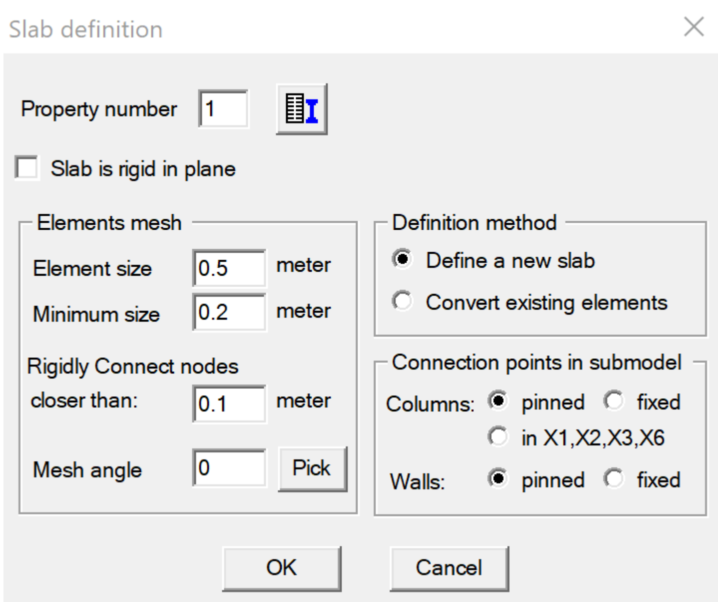
The Slab option is a more powerful and extended version of the Element-Mesh option. The Slab option internally generates both the elements and the nodes within a user-defined area:

The mesh area is specified by defining a 'perimeter' which joins existing nodes. The perimeter consists of straight lines connecting nodes or arcs. It may be of any shape and 'holes' may be defined inside it. Note that the mesh definition may be revised at any time.

- the slab element is defined by a contour of any shape and may include openings.
- the program automatically connects the slab element to all existing columns, walls, other slab elements and regular elements that attach to it. ***Nodes with nothing attached to them are ignored.***
- before solving the model, the program divides the slab into individual elements according to user-defined parameters (similar to the Mesh option).
- ***if columns, walls, etc, are moved or deleted, no changes to the slab element have to be made because the program automatically redefines the elements and connection points internally every time the model is solved***.
- Refer to Notes 314 for hints and suggestions for defining a slab.

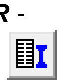
To define a slab:
- specify the slab parameters
- create a contour 313 that defines the slab area, including openings

Parameters:



Property number

Each Slab must be assigned to a property group. The property group assigned to an existing slab may be revised at any time.

- Enter a property number. Note that a property that has not yet been defined may be assigned to an slab **- *OR* -**
- click the  icon to select a property from the list or to define a new property.

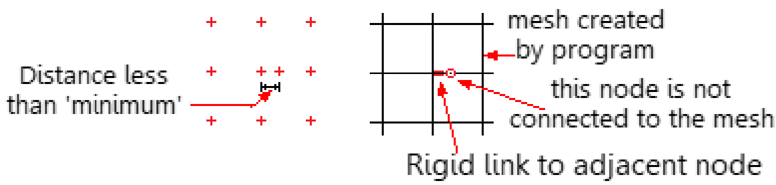
Slab is rigid in plane

☑ For slabs on global planes: all nodes within and on the slab perimeter will be rigidly linked to each other.
  - slabs on non-global (diagonal) planes will not be rigidly linked
  - Example: a slab defined an the X1-X2 plane: the X1,X2,X6 dofs are rigidly linked.

Elements Mesh

The Slab option internally generates both the elements ***and*** the nodes within a user-defined area. The program tries to create square elements according to the following parameters:

- ***Element size:***
  Specify the approximate size of the internal elements in the slab area. For example, enter 1.0 if 1.0 x 1.0 elements are adequate for your slab.
- ***Minimum size:***
  Increasing this value prevents the generation of small elements; if a program generated node is less than this specified distance from an existing node (x or y projection), then the program does not generate the node.
- ***Rigidly connect nodes closer than:***
  If the distance between two existing adjacent nodes within the contour is less than the "minimum element size", the program will use only one of the nodes to create the mesh and will rigidly link the other node to it. For example:



Refer to Notes 314 for hints and suggestions for defining a slab.

- ***Mesh angle:***
  The base line angle of the grid may be defined in two ways:
  - type in the value of the angle in the text box, where the angle is measured from a base line as follows:

    plane models - parallel to X1
    space models - parallel to the projection of X1 on the plane. If the plane is parallel to X1, the base line is parallel to X2.

    The grid angle is measured counterclockwise from the base line.
  - click the [Grid angle =] button and select two existing nodes defining the base line.

Definition method

Define a new slab
Create a slab element in an area with no existing slabs or elements.

Convert existing elements
Create a slab element from one or more existing rectangular and triangular elements. Select the elements using the standard element selection option.
- all selected elements must have the same property
- all selected elements must be connected

Note:
- an edge of a Slab element will be released only if ***ALL*** the nodes along the same edge in the element model were released.
- the elements in a Mesh that were defined as "Skew" or "Circle" will be converted to rectangular elements.
- Springs and Rigid links will be maintained only at the Slab contour nodes; the ones defined at links in the slab interior will be lost.
- Loads defined on the elements are lost.

Connection points in submodel

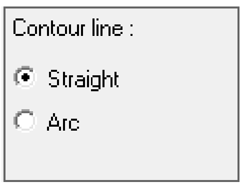
If the defined slab is located in a submodel:

- the program automatically identifies all columns and walls in the main model that are within the slab contour.
- the program automatically creates 'connection points' at the column and wall locations.
- these connection points can be either **pinned**, **fixed** or **pinned, fixed around height** (for columns only). The latter option is required to prevent singularity when a column is attached only to slab elements in submodels at both ends.

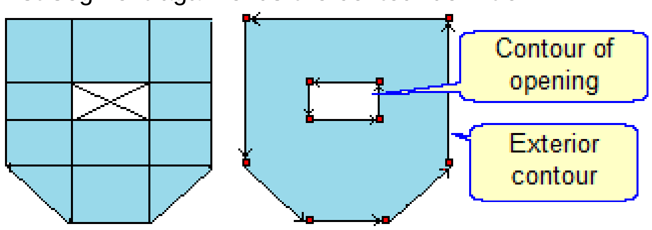
3.7.1.1 Contour

The slab area is specified by defining a 'perimeter' contour which joins existing nodes. The perimeter consists of combinations of straight lines and/or arcs defined between nodes. It may be of any shape and 'holes' may be defined inside it.

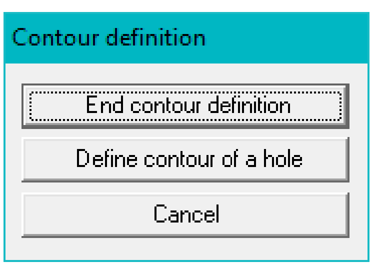
- select the segment type from the following side menu:



- Select the nodes at the start and end of the current segment. Then define additional segments, where the end node of the last segment is the start node of the first segment. Selecting the start node of the first segment again ends the contour definition.



- Define openings in the contour or end the contour definition:



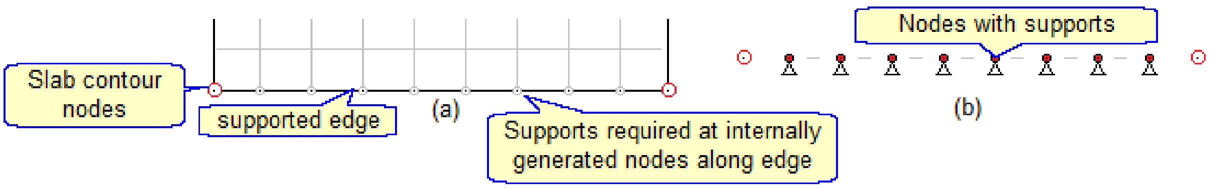
Select:

- **End contour definition**
  Select this option if there is no area within the contour where elements are **not** to be generated (e.g. stair openings, etc). If openings exist, select **Define contour of a hole.**
- **Define contour of a hole**
  Define the contour of an opening using the same methods as for the main contour. Multiple holes may be defined within the main contour. Select **End contour definition** when all holes have been defined.

3.7.1.2 Notes

Supported edges

Supports cannot be defined at nodes created internally by the Slab option. For example: a supported edge [Figure (a)]:

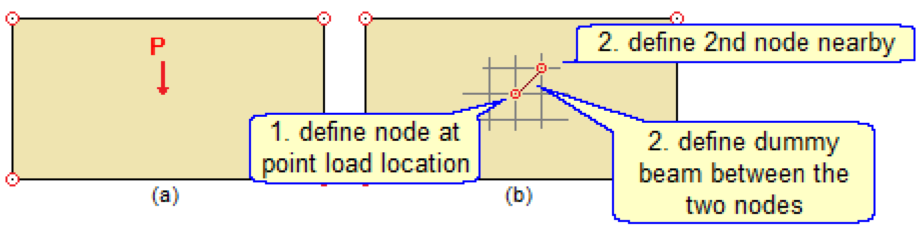


- define a line of nodes with supports along the slab edge - Figure (b). The distance between the nodes should be approximately the desired mesh density.
- the program will then create the internal mesh so that it connects to these existing supported nodes.

Point loads

Point loads cannot be defined directly on a Slab element [Figure(a)]. There are two ways to define the load:

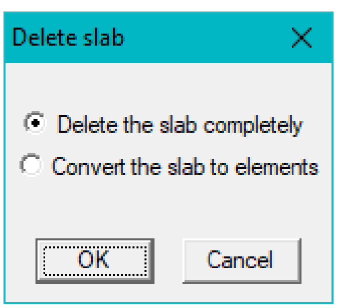
- as a global point load defined by coordinates.
- as a joint load defined on a node at the point load location.
  Not that nodes with nothing attached to them are not connected to the slab. Therefore, define another node near the point load location and connect the two nodes with a Dummy beam - Figure (b):



The internal mesh created by the program will connect to these nodes.

3.7.2 Delete

A Slab can either be deleted entirely or else it can be converted to regular elements:



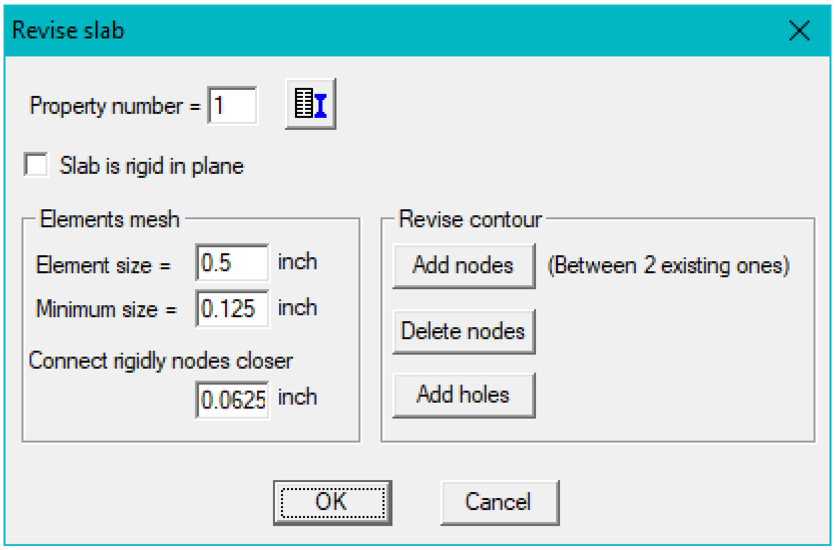
Select slabs using the standard Slab selection option.

Note:

- all Slab loads are lost when a Slab is converted to elements.

3.7.3 Revise

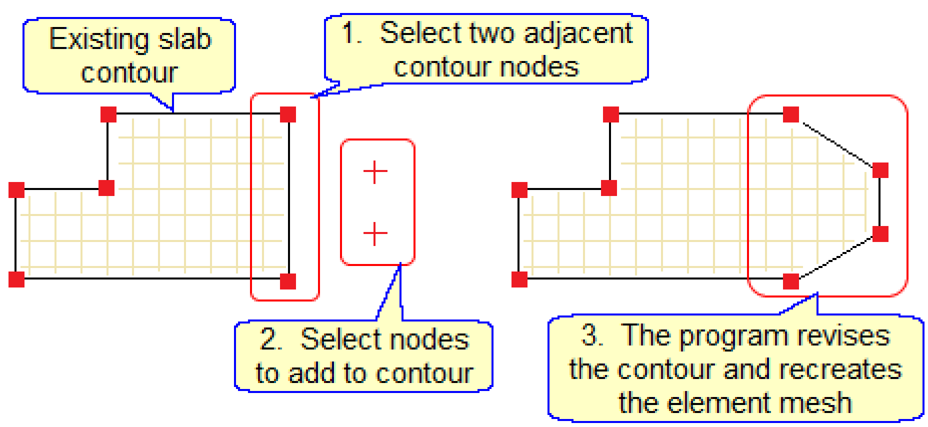
To revise an existing slab:
- Select a slab using the standard Slab selection option.
- revise the parameters or the contour:



Revise contour

Add nodes or delete nodes from the contour or add an opening. The program immediately recalculates the slab elements.

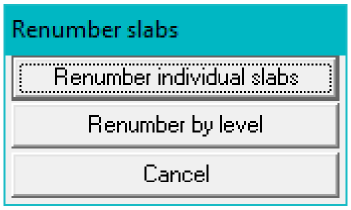
For example, add nodes:

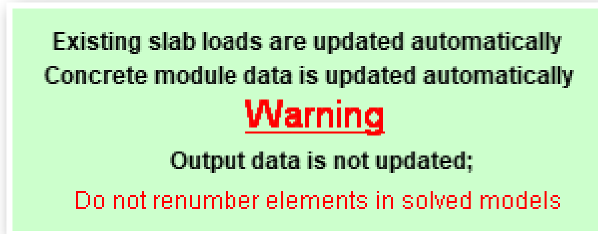


Note:
- if you move a node in a slab contour, you can either maintain the slab shape or revise it accordindly. Refer to Move nodes.

3.7.4 Renumber

Use this option to renumber existing slabs:

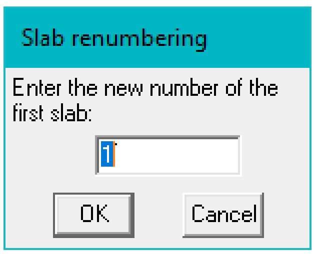




Individual slabs

Select one or more elements using the standard Slab selection option. Note that the order that the slabs are selected is important; they are renumbered in the order that they are selected.

Type the new number of the first slab selected:



All of the slabs selected are renumbered sequentially. If the program discovers that a number in the new sequence has already been assigned to another slab, the program does not renumber that slab.

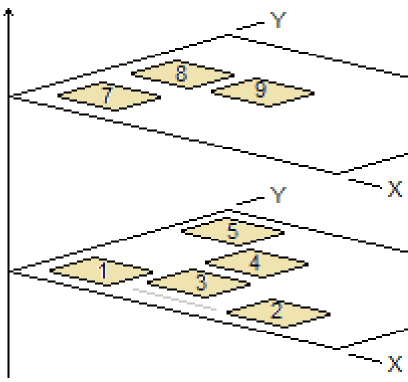
For example:
- slabs 41, 42 and 43 are selected (in that order).
- 75 is specified as the new number for 41
- the slabs are renumbered 75,76 and 77 respectively, but ....
- slab 76 is an existing slab; it remains unchanged at 42.

Renumber by level

Renumber all slabs in the entire model.

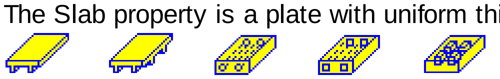
The renumbering order is determined as follows:
- the program first renumbers the slabs parallel to one of the global planes, starting with the plane having the smallest coordinate of the perpendicular axis.
- the program searches for the smallest X and Y coordinates of contour corners in each slab, the slabs with the smallest Y coordinate are then renumbered according to increasing X coordinate, and so one.
- the process is repeated for the levels above and then for levels parallel to other global planes.

For example:



3.7.5 Properties

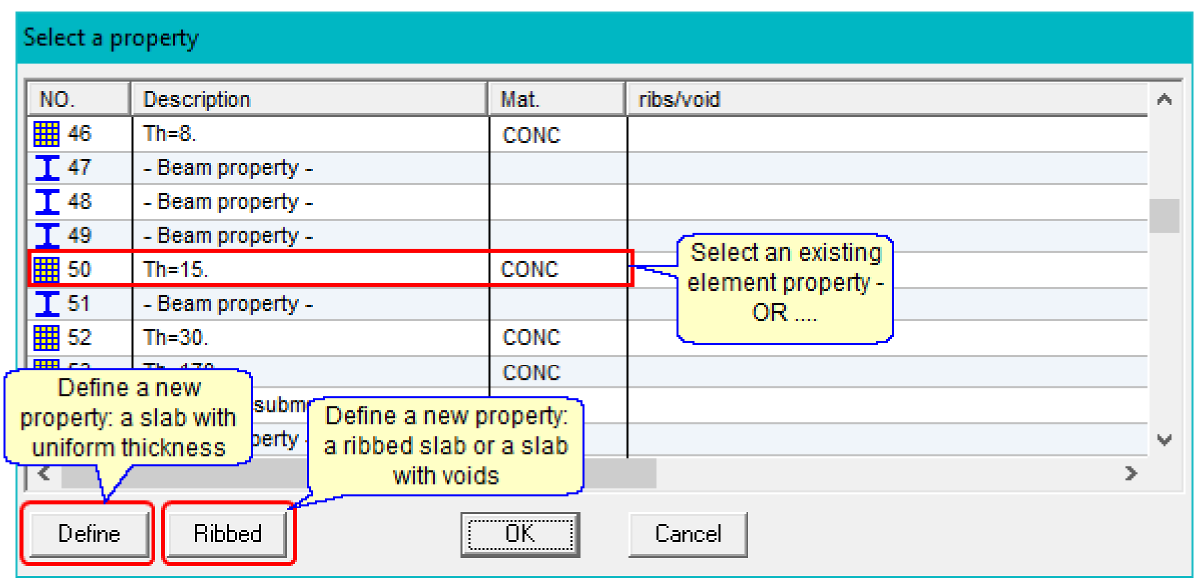
The Slab property is a plate with uniform thickness or one of the following sections:



The element properties are:
- dimensions (e.g. plate thickness)
- material (may be isotropic or orthotropic)

Each Slab must be assigned to a property group. The property group assigned to an existing slab may be revised at any time.

Note that a property number may be assigned to an slab even if the properties have not yet been defined.



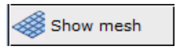
Define

Define a standard plate element (uniform thickness). Refer to Element properties - define

Ribbed

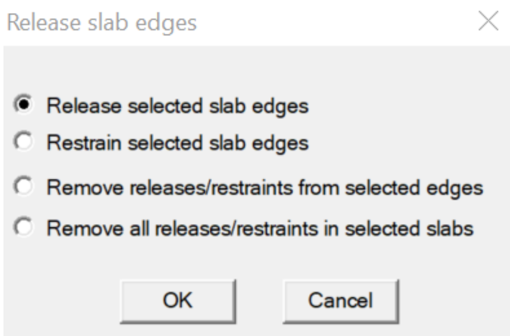
Define one/two-way ribbed slabs, slabs with rectangular/round/elliptical holes, closed waffle slabs. Refer to Element properties - additional.

Note:

- the rib direction is defined as parallel to either the element x1 or x2 axis. To display the local element axes in the generated mesh, click on 

and then select Display - Local axes.

3.7.6 Releases



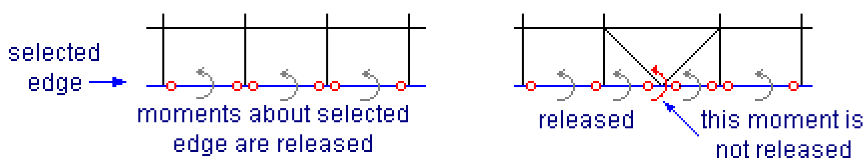
Release/Restraint selected slab edges

Release:

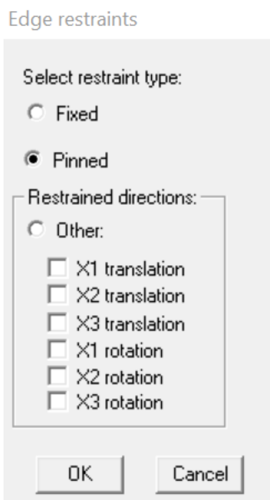
- move the cursor so that a slab contour line is highlighted with the and click the mouse.
- select additional contour edges.
- double click on the last selected line to end.

Small circles are drawn at the ends of the selected contour lines.

Note:
- moments ***about*** a selected ***edge*** are released; a special case occurs when the program creates a triangular or rectangular element attached to a slab edge only at one of its nodes. The moment at the node is not released and there will be a local stress concentration at that node. For example:



Restraint:
- move the cursor so that a slab contour line is highlighted with the and click the mouse.
- select additional contour edges.
- double click on the last selected line to end.



Select the support type:

- **Pinned** - all relevant translation global degrees-of-freedom are restrained
- **Fixed** - all relevant translation ***and*** rotation global degrees-of-freedom are restrained.
- **Other** - any other combination of restrained global degrees-of-freedom

The relevant degrees-of-freedom are:

	Translation	Rotation about
Plane frames	X1, X2	X3
Plane grids	X3	X1, X2
Space frames	X1, X2, X3	X1, X2, X3

Remove releases/restraints from selected edges

Select contour edges as explained in the previous option; existing releases will be deleted.

Remove all releases/restraints in selected slabs

Select slabs with releases using the standard Slab selection option. Releases along ***all*** edges of the selected slabs will be deleted.

3.7.7 Stages

Slabs may be removed from the current stage or restored. Different properties may be defined/assigned to a slab in a Stage.

- Select slabs using the standard slab selection options

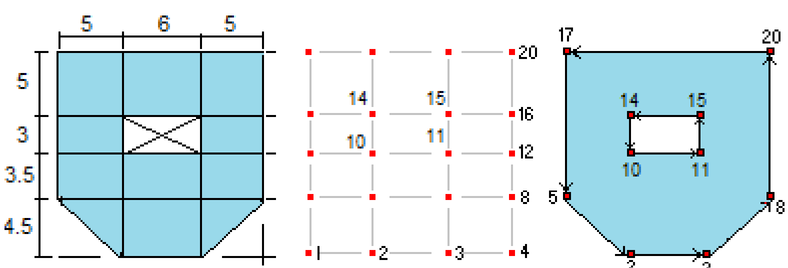
Note:

- inactive slabs are not displayed
- new slabs cannot be defined when a stage other than ***Whole model*** is active

3.7.8 Slab - Example 1

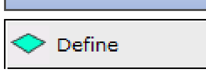
Create a model of the flat slab shown in Figure (a) with 1.0x1.0 elements (approx.).

- First, define the nodes shown in Figure (b).

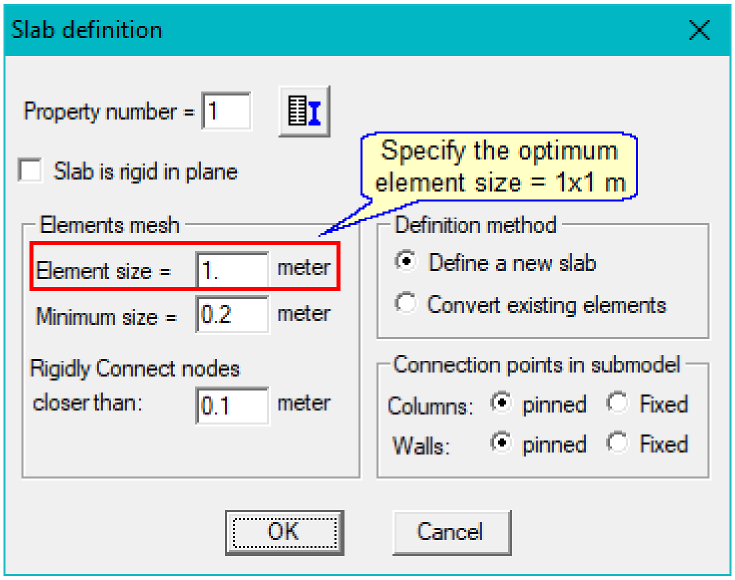


(a) (b) (c)

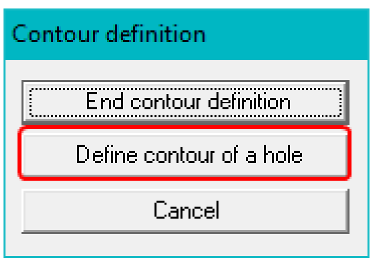
- Select the  option

- Select the  option

- define the slab parameters

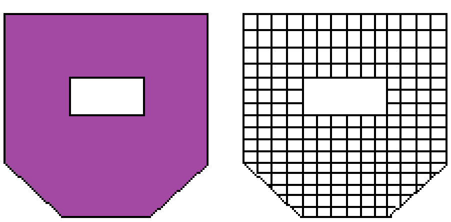


- define the exterior contour:
  Move the mouse and highlight the corner nodes 2-3-8-20-17-5-2 as shown in Figure (c).
- Define the opening: select -

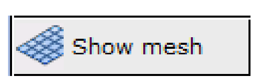


Select:
- select the four corner nodes 10-11-14-15 of the opening as shown in Figure (c).
- **End contour definition** when the above menu is displayed again.

The program now creates the mesh - Figure (a):



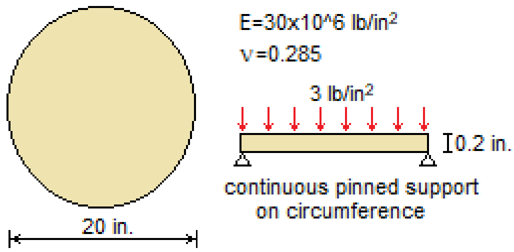
(a) (b)

To show the elements that will be created by the program, click 

- Figure (b)

3.7.9 Slab - Example 2

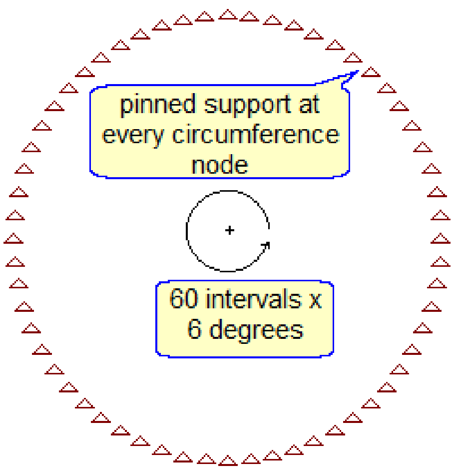
Define a model for the following disc containing elements approximately 0.5 x 0.5

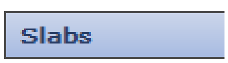
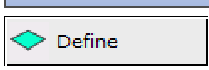


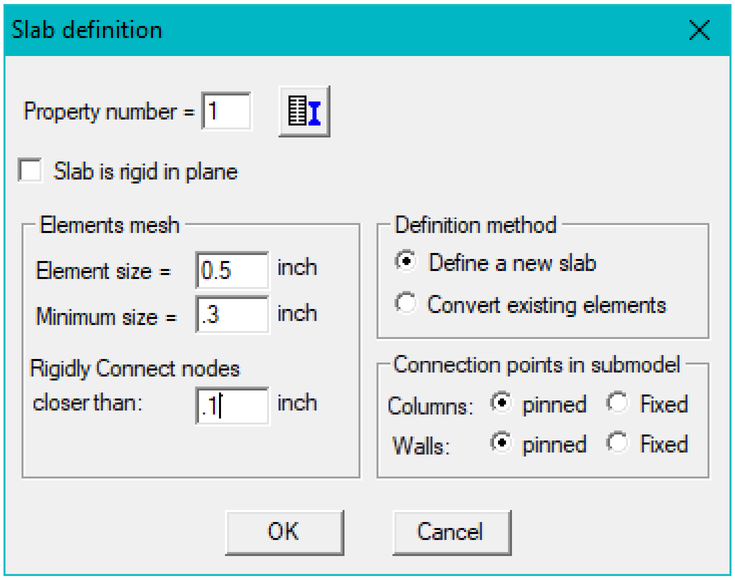
The example is taken from "Roark's Formulas for Stress and Strain" - 6th Edition, p 393

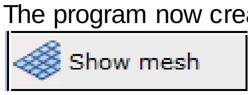
The example also compares the results calculated for the orthogonal array of elements created by the "slab" option to those calculated for the radial array of elements created by the "Elements - Mesh" option.

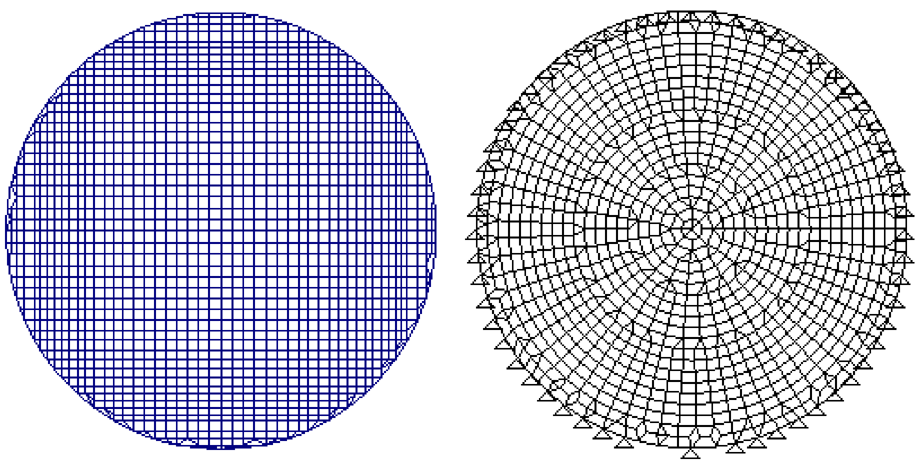
- First, define the nodes on the circumference. Create a Cylindrical coordinate system:



- Select the  option
- Select the  option
- define the slab parameters:



- define the exterior contour:
  The program now creates the mesh. To show the elements that will be created by the program, click  -:



(a) (b)

The example also compares the results calculated for the orthogonal array of elements created by the "slab" option to those calculated for the radial array of elements created by the "Elements - Mesh" option (Figure (b))

The results are:

	STRAP - Slab	*STRAP* - Mesh	Exact
Deflection	-0.0886	-0.0833	-0.0833
Mc,max	61.6	61.6	61.5

3.8 Springs

Define linearly elastic spring supports - translational and rotational - at nodes.

When an elastic support is defined in a specific direction at a node, ***the node must be unrestrained in that direction***.

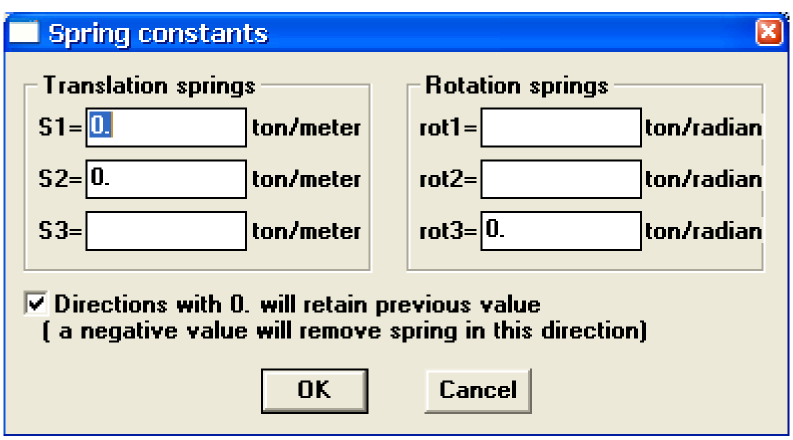
Define	Define or revise [323] springs. Springs may be defined in global (default) or local directions.
Delete	Delete springs; select the nodes with springs using the standard Node Selection option.
Show value	Write the spring constant [325] values on the graphic display (for defined springs).
Unidirectio...	Define translational springs that act in either the positive or negative direction [325] only of the selected axis, but not both.
Area/line	Define spring constant per area/length [326] and then select elements/nodes defining area/length; program will automatically calculate spring constants for relevant nodes.
Non linear	Define non linear springs [328]. These springs have a a spring constant that varies with the deflection.
Gap	Define a gap element [329]. A gap does not transfer forces until it is closed and then it acts as a regular node.

Note:

- To revise a spring, select Define, select the node and enter the revised value.
- To define a translational spring in a direction ***not*** parallel to a global direction, assign a local restraint system to the node, as explained in Springs - local [324]. ***Do not define the global components of the spring constants using S·sin(θ) and S·cos(θ)***.

3.8.1 Define/revise

Define the spring constants.



Directions

S1 = translational spring constant in the global X1 direction units: force / length
S2 = translational spring constant in the global X2 direction
S3 = translational spring constant in the global X3 direction

rot1= rotational spring constant about the global X1 axis units: moment / radian
rot2= rotational spring constant about the global X2 axis
rot3= rotational spring constant about the global X3 axis

Note:
- To define the springs in a non-global (local) direction, refer to Springs - local 324.

Zero values

Select one of the options:

☑ the program ignores all zero values in the menu

☐ the program sets spring constants = 0. in directions with zero values

Example:
- a value for **S1=3250** was previously defined at a node and you now want to add **S2=1000** to the same node:

S1= 0.

S2= 1000.

- Enter the values above in the dialog box; the spring constants for the node will be:

☑ **S1=3250** and **S2=1000**

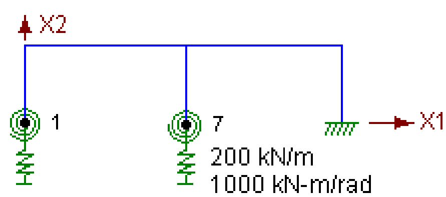
☐ **S1=0** and **S2=1000** (this is incorrect !)

Note:
- to set a spring constant to zero in a specific direction, enter a negative value in that direction.

Example:

Enter: **S2 = 200** and **rot3 = 1000**

and select nodes with these spring constants using the standard Node Selection option.



3.8.1.1 local axes

Springs, by default, are defined relative to the global axes.

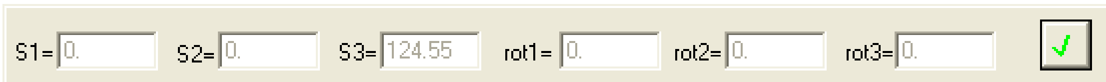
To define a spring relative to any local x1-x2 system:
- define the spring constants using the Define option, but assume that **S1, S2, ..., rot3** refer to the local system
- define a local restraint system using the Restraint - Rotate "**Define a local support coordinate system**" option
- assign the local support system to the node with the spring using the Restraint - Rotate "**Assign supports to a local coordinate system**" option

Note:
- the spring constant may be defined after the local support system has been defined and assigned.
- local support systems may also be defined for unidirectional springs.
- local support systems will be applied to springs define using the "area/line" option.
- if a local support system has been assigned to a node where a spring has been defined, the local system will be noted in the graphic and tabular displays of spring data
- when defining a translational spring in a direction **not** parallel to a global direction, always use the method detailed above; ***do not define the global components of the spring constants using S·sin(θ) and S·cos(θ)***.

3.8.2 Show value

Select a node with springs; the spring constants are displayed at the bottom of the screen and all nodes with the same constants are highlighted.

For example:



Note that the spring constants cannot be edited; to revise, select Define 323, select the node and enter the revised value.

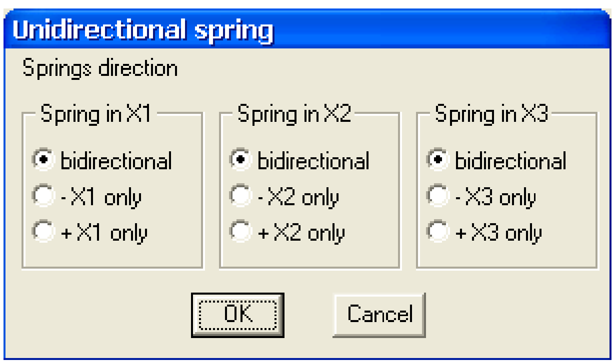
3.8.3 Unidirectional

Define springs that act in either the positive or negative direction only of the selected global axis, but not both. The spring stiffness will be assumed equal to zero in the opposite global direction.

Note:

- unidirectional springs are non-linear elements and require several iterations of the solution, which may increase the solution time significantly.
- the rules of superposition do not apply for non-linear elements. Therefore, load combinations for models with unidirectional springs in loading ("Combine ld") and not after the solution.

Select the active spring directions for each global axis:

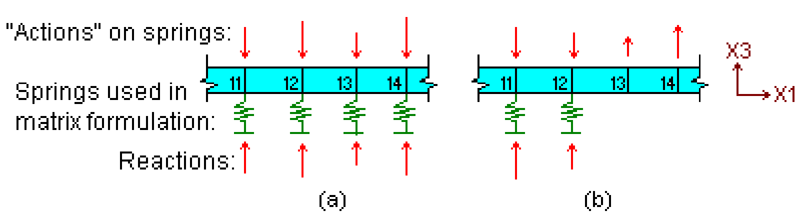


where:

- the direction indicates the direction of the force acting on the spring.
- **bidirectional** is the default value and indicates a regular spring.

Example:

A foundation is subject to uplift forces under specific combinations of loading. In such a case there is no contact between the foundation and the underlying soil. Define unidirectional springs in the -X3 direction.

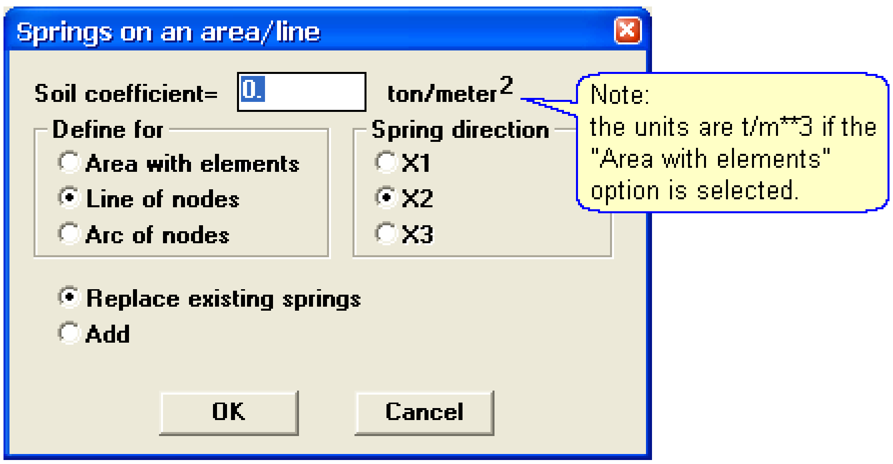


3.8.4 Area/line

This option automatically calculates the spring constant at the nodes along a line or at the nodes enclosed in an area based on a soil coefficient defined by the user. This option is useful for the analysis of beams and mats on elastic foundations.

Beams: The user selects a line of nodes; the program calculates the distance between adjacent nodes, multiplies the distance by the specified soil coefficient and defines springs at the nodes.

Mats: The user selects a group of elements; the program calculates the tributary area for each node attached to the elements, multiplies the area by the soil coefficient and defines springs at the nodes. Use dummy elements in models with beams only



Soil coefficient

Enter a soil coefficient as follows (in the current geometry units):

Area with elements

Enter a value with units Force/Length^3; the program will multiply the coefficient by the tributary area and calculate a spring constant (units = Force/Length).

Line/Arc of nodes

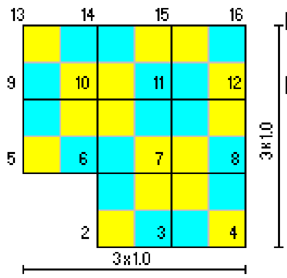
Enter a value with units Force/Length^2; the program will multiply the coefficient by the the distance between adjacent nodes and calculate a spring constant (units = Force/Length).

Area

The spring coefficients are calculated as follows:

- the user selects elements using the standard Element selection option.
- the program identifies all nodes connected to the elements.
- the program calculates the tributary area associated with each node, multiplies by the soil coefficient to calculate the spring constant and defines the spring.

For example, soil coefficient = 100 (F/L^3);.8 selected elements:



Node 2, 4, 5, 13, 16: Area = 0.5 x 0.5 = 0.25
Spring constant = 0.25 x 100 = 25 F/L

Node Area = 1.0 x 0.5 = 0.50
Spring constant = 0.50 x 100 = 50
Area = 3 x 0.25 = 0.75
Spring constant = 0.75 x 100 = 75
Area = 1.0 x 1.0 = 1.00
Spring constant = 1.00 x 100 = 100

Node 6:

Node 7, 10, 11:

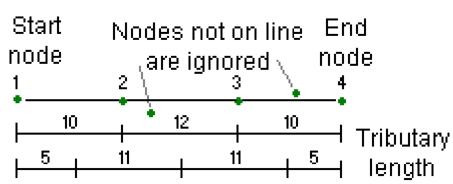
Line

The spring coefficients are calculated as follows:

- the user selects the start node and end node of the line using the standard Node selection option.
- the program identifies all intermediate nodes on the line
- the program calculates the tributary length associated with each node, multiplies by the soil coefficient to calculate the spring constant and defines the spring

For example, soil coefficient = 100

Spring at nodes 1, 4 = 5 x 100 = 500
Spring at nodes 2, 3 = 11 x 100 = 1100



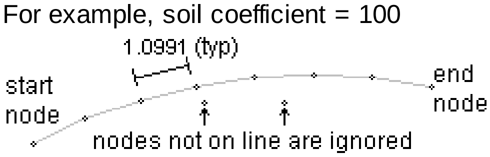
Arc

Calculate the spring coefficients along an arc of nodes. Note that to define radial springs, the relevant nodes must be assigned to a local radial restraint system.

The spring coefficients are calculated as follows:

- the user selects the start node and end node of the arc and an intermediate node along the arc.
- the program identifies all intermediate nodes on the arc
- the program calculates the tributary length associated with each node, multiplies by the soil coefficient to calculate the spring constant and defines the spring

For example, soil coefficient = 100



Spring at intermediate nodes = 1.0991 x 100 = 109.91
Spring at end nodes = 1.0991/2 x 100 = 54.96

Direction

Select a ***global*** direction.

Replace/add

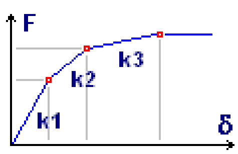
Select one of the following:

- **Replace existing springs**
  The program erases the current spring constant ***in the selected global direction*** at the selected nodes and replaces it with the calculated one.
- **Add**
  The program adds the calculated spring constant to the current value at the specified nodes.

Select the nodes or elements using the standard Node selection or Element selection options; the program then calculates the spring coefficients and defines the springs at the relevant nodes.

3.8.5 Non-linear springs

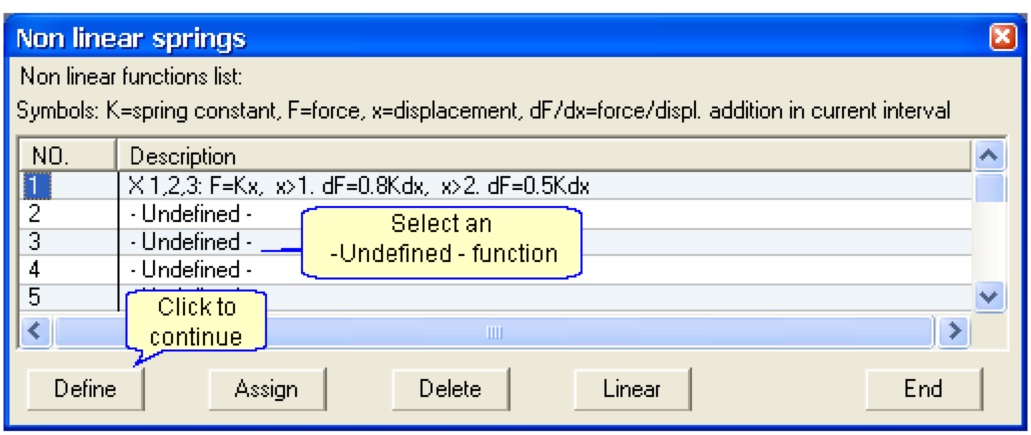
Define non-linear springs. These springs have different spring constants for different ranges of applied force, e.g.



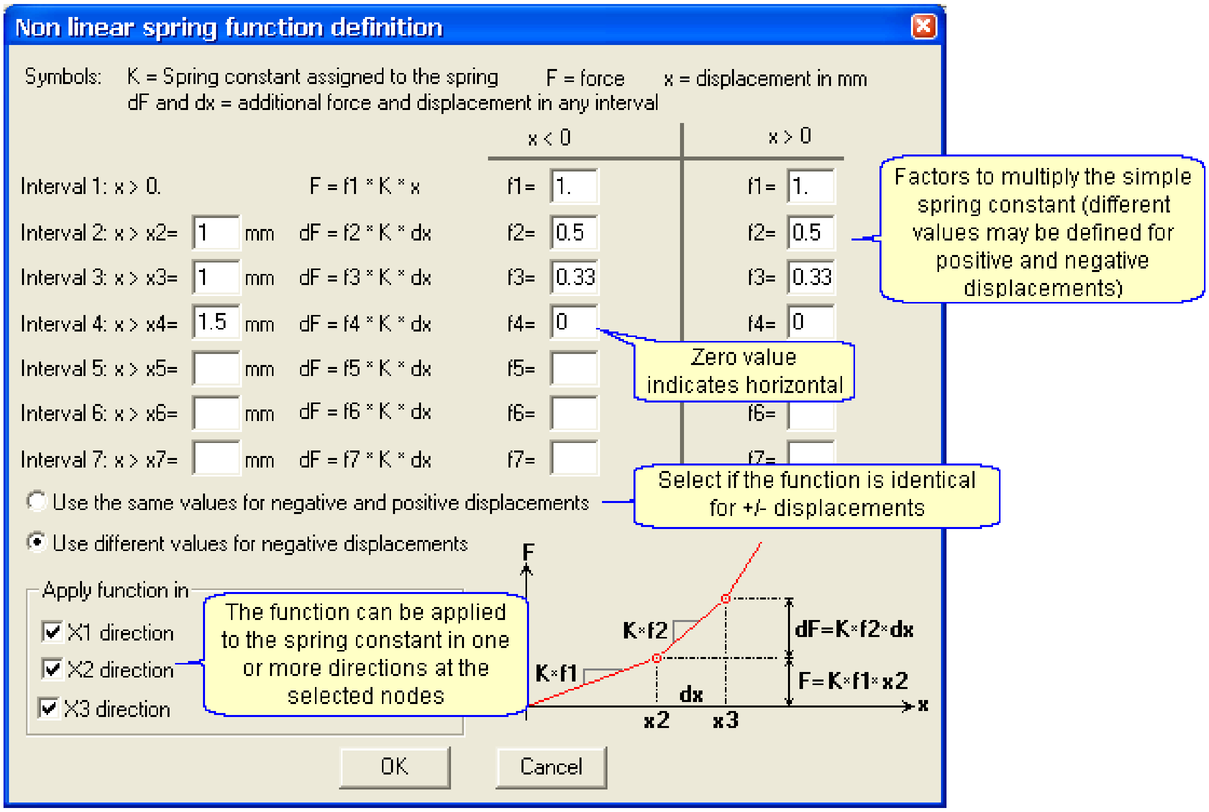
The procedure is as follows:

- define regular springs at the selected nodes
- define a non-linear spring function that modifies simple springs.
- apply the function to selected simple springs. In this way the same function can be applied to springs with different spring constants.

To define a non-linear spring function:

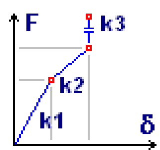


The function consists of the factors to multiply the simple spring constant to which the function is applied. For example, referring to the small graph above, assume that k1=6000, k2=3000, k3=2000 for intervals of 1 mm, 1mm, and 1.5 mm, and a simple spring has been defined with k=6000. The factors that must be defined in the function are 1.0, 0.5 and 0.33 :



Note:

- if the motion stops at a certain displacement, define a large constant after that displacement, e.g. k_n=1000.



- to define a unidirectional spring 325 (zero stiffness in one direction), define f_1, ... f_n = 0 in the relevant column

---

Select one of the following:

Assign Highlight a function in the list and assign it to existing simple springs. Select nodes using the standard Node Selection option. Note that springs with different constants may be selected; the same function will be applied to all of them.

Delete Delete the highlighted functions from the list. Note that the "assignment" to the springs is not deleted.

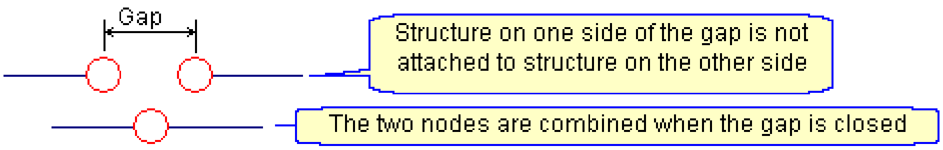
Linear Cancel the "assign" for selected nodes. Select nodes using the standard Node Selection option; springs that were "assigned" with this function will revert to simple springs.

3.8.6 Gap element

The Gap element is a non-linear element that represents a space between two nodes. The gap can be either opened or closed:

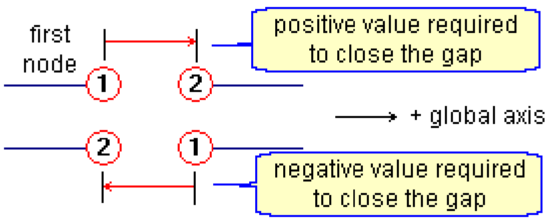
- when the gap is opened no forces are transferred between the members on either side of the space.
- when the gap is closed forces are transferred through the node - in the selected direction - as if no gap was defined.

Gaps are created by defining two nodes at the ***same location*** and an initial space between them:



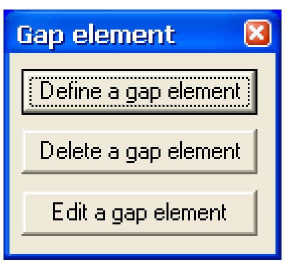
Note:

- the motion is always in a global axis direction.
- a gap element has its own "coordinate system":
  - the first node selected is the"start node"
  - a positive gap value indicates that ***movement of the "start node" in the positive global axis direction is required to close the gap*** :

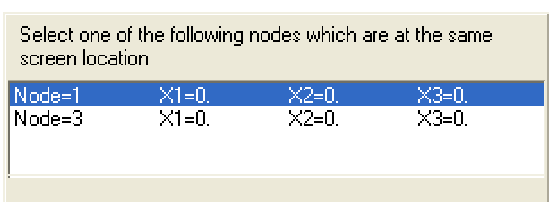


To define a gap element:

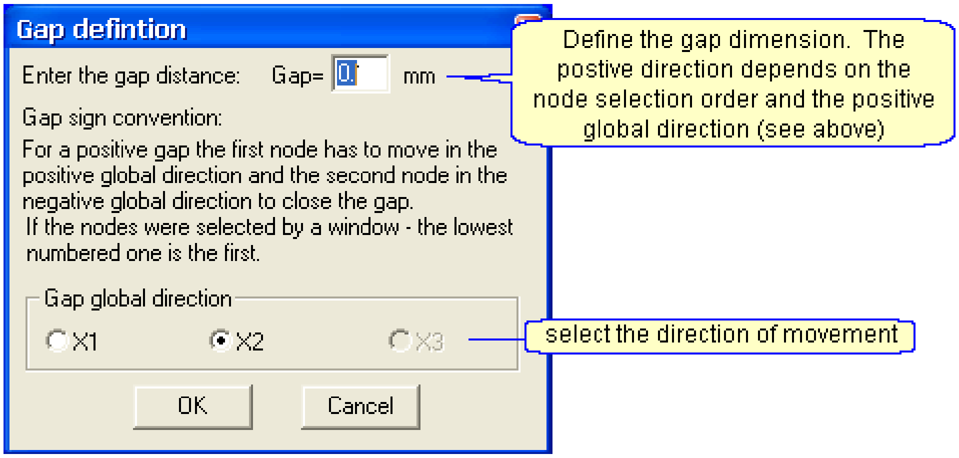
- define two nodes at the same location
- select:



- select nodes using the standard Node Selection option.
  - if you select the nodes using the Window option, the lower numbered node is the"start node"
  - if you select the nodes individually, the program prompts you to select the start node:



- enter the gap distance:



Note:

- when the gap is closed, ***forces are transferred only in the one selected global direction***; define rigid links connecting the two nodes to transfer forces in the other global directions..
- nodes with gaps are shown on the graphic display as ***n1-n2***

3.9 Copy geometry

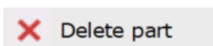
Copy is a powerful option which enables the user to copy a block from the model - including nodes, elements, properties and releases - to a different location.

All elements in the copied block are drawn parallel 333 to the original ones. The location of the copied block is at a specified distance from the original.

The copied block may also be rotated 334 with respect to the original block. The program may stretch or shrink the dimensions of the block, but will always maintain the same node/element layout

Create a mirror image 338 of the original block about an axis of symmetry.

Copy geometry from one submodel to another.



Delete part of a model. All elements types may be selected.

Note:
- more than one copy of the block may be created with one command
- the program does not generate a new node at the location of an existing node but uses the existing node when creating the elements in the new block.
- the program does not generate a new element at the identical location of an existing element, including walls.
- the copied element is assigned with the property group number of the original element.
- Releases are automatically copied.
- Geometry in submodels can be copied to other submodels, but geometry cannot be copied from the main model to a submodel or vice versa.
- Beam local axes (Copy+rotate/Mirror):
  The program tries to rotate the local axes along with the rotation of the block. For example: a beam with x2 pointing to the centre of a circle is copied radially around the circle; the x2 axis of all the copies of the beam will also point towards the centre of the circle.
- Element local axes:
  The local coordinate systems of the copied elements are selected so that the axes of the copied elements point in the directions that are as near as possible to the directions of the axes of the original elements.

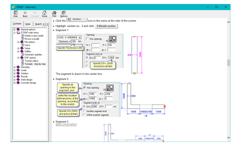
The block to be copied is defined by selecting a group of nodes using the standard Node Selection option; only elements with ***all of their end nodes*** selected are included in the block.
- the location of the new block may be defined by rotating and translating the original block.
- the two blocks may have a common intersection line; the program automatically Unifies the two blocks so that they are connected.
- the program may be instructed to connect the new nodes and the corresponding original nodes with beams.
- the command may stretch or shrink the dimensions of the block, but will always maintain the same node/element layout.

For all options, select the nodes that define the block.

3.9.1 Translate only

To display a step-by-step tutorial that explains how to "Copy":

- ***Space frame - 1***

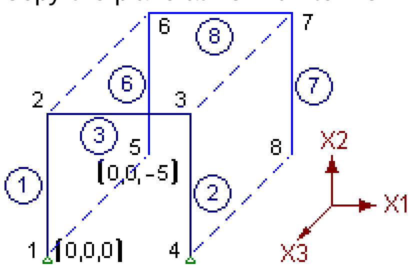


All elements in the copied block are drawn parallel to the original elements.

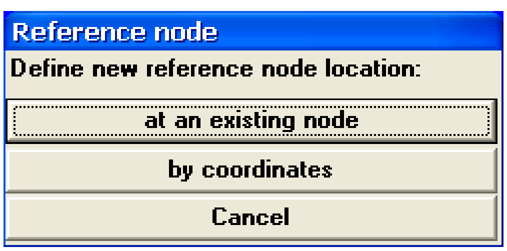
- The block to be copied is defined by selecting a group of nodes using the standard Node Selection option; only elements with ***all of their end nodes*** selected are included in the block. Only one reference node is required since all generated nodes are offset an identical distance from their original nodes.
- More than one copy can be created with the same command; the offset from the second copy and the first will be identical to the offset from the first copy to the original block.

Example:

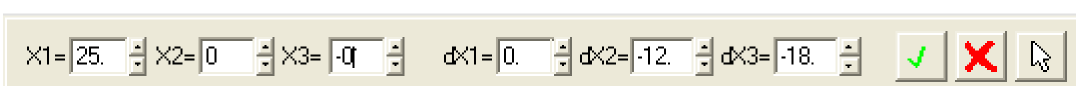
Copy the plane at X3 = 0. to X3 = -5.0



- Define the block to be copied by selecting the frame 1-2-3-4 using any of the standard Node selection options.
- Select node 1 as the reference node.
- Select the new location of the reference node:

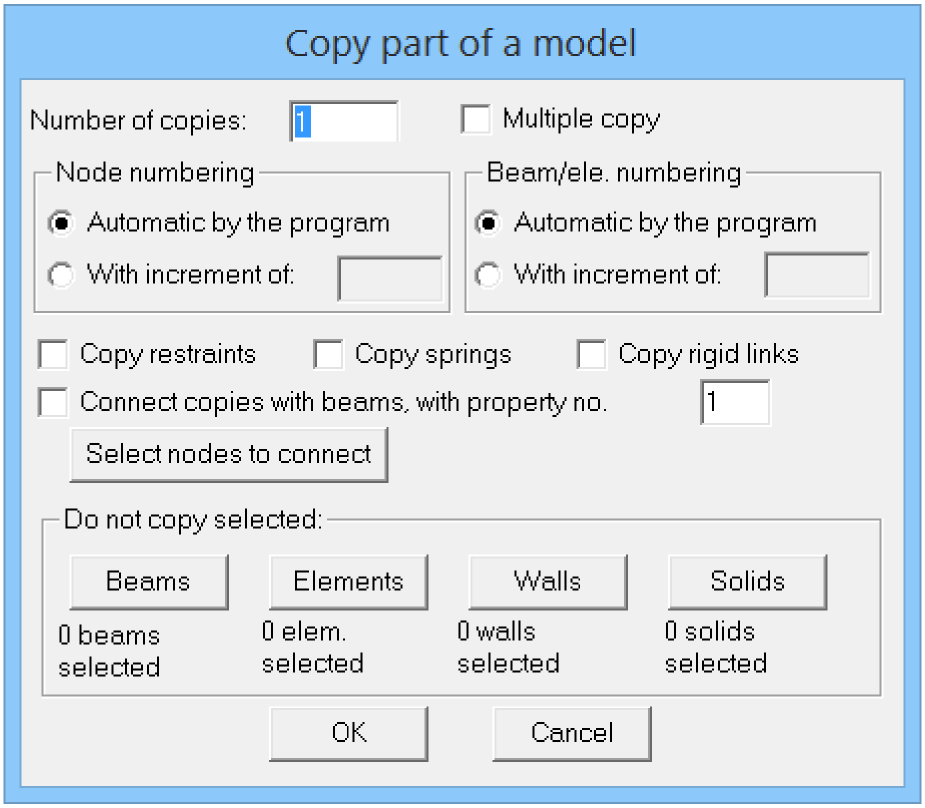


- select by coordinates and move the so that



and click the mouse.

- set the copies/numbering/increment options to the following values.



Note:

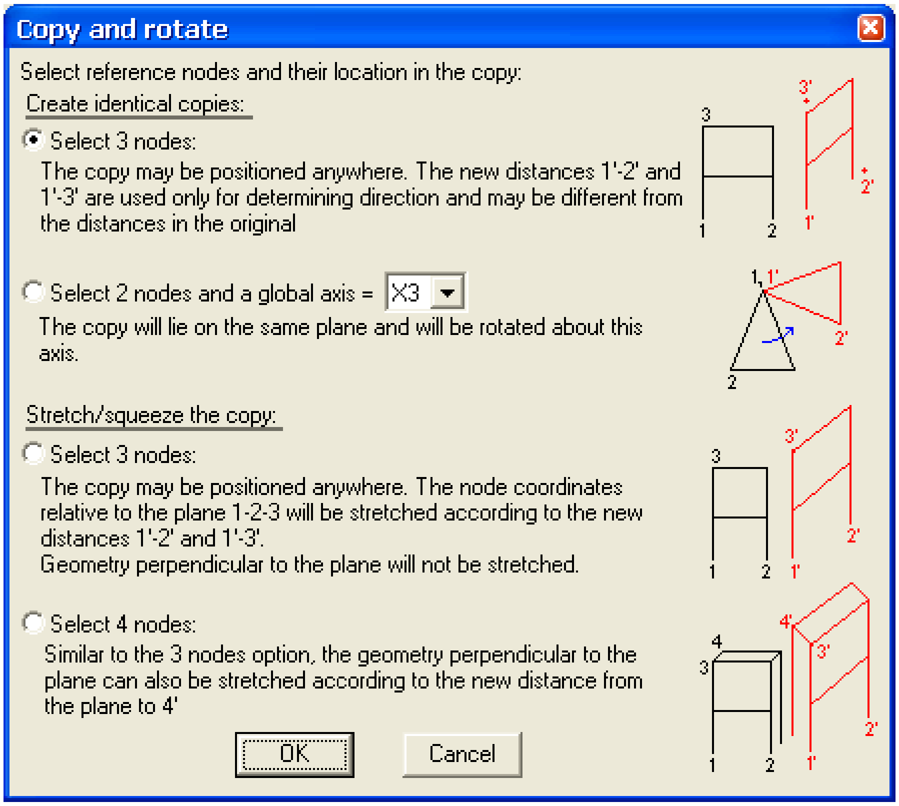
- beams 1-5, 2-6, 3-7, 4-8 are generated if the **Connect copies with beams** option is set to ☑.
- restraints at nodes 1,4 are copied to nodes 5,8 if the **Copy restraints** option is set to ☑.

3.9.2 Translate and Rotate

The block to be copied and rotated is defined by selecting a group of nodes using the standard Node Selection option; only beams and elements with ***all of their end nodes*** selected will be included in the block.

The rotation and translation is defined by specifying the new location of reference nodes; the new location of each node can be either at any existing node or at a coordinate. The nodes form a plane and the translation and the rotation of this plane is applied to all of the selected nodes.

There are four options available: 2 options maintain the shape of the selected geometry and the other 2 stretch/squeeze the block proportionally according to the new distances between the reference nodes:

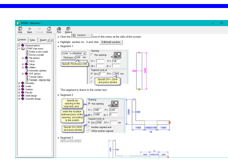


- if the stretch/squeeze options are selected, the program will 'stretch' or 'squeeze' all of the elements in the block according to the change in distance between the reference nodes; ***the program will not distort the block***, i.e. all parallel beams will be stretched or squeezed ***by the same factor***.
- the block may be copied more than once by the same command; the offset and rotation between successive copies is the same amount as that between the original block and the first copy.

Refer to the examples 335 for a more detailed explanation, or:

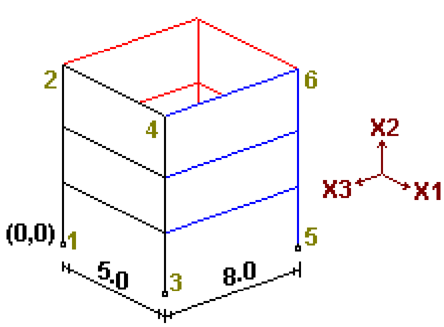
To display a step-by-step tutorial that explains how to "Copy and rotate":

- ***Space frame - 2***



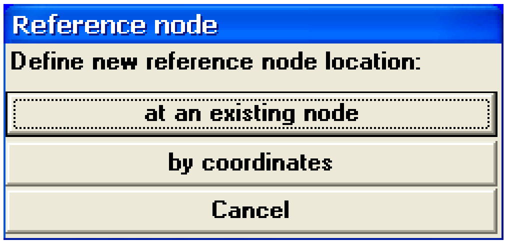
3.9.2.1 Examples

Create the following model:



- Create frame 1-2-3-4 using the Model wizard

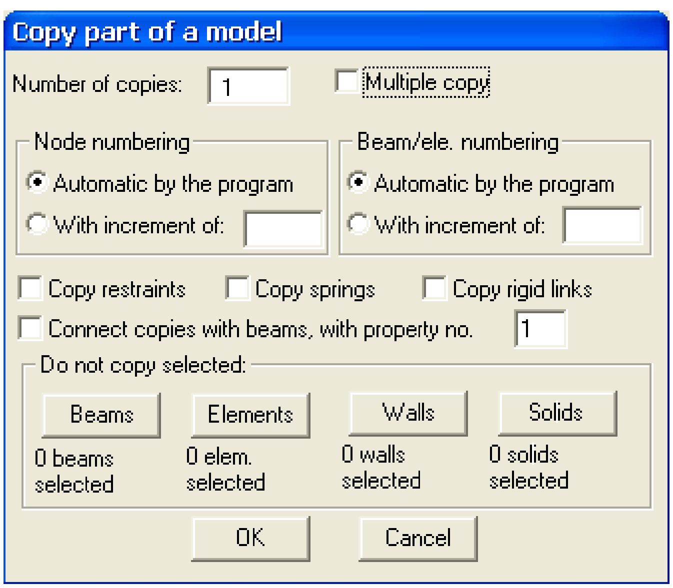
- Copy frame 1-2-3-4 to frame 3-4-5-6:
  - select all the nodes in the frame using the standard Node Selection option.
  - select **Select 3 nodes** (stretch/squeeze the copy)
  - define the three reference nodes and their new locations:



Ref. node:	No:	Option:	Select/define:
1st	1	at an existing node	node 3
2nd	2	at an existing node	node 4
3rd	3	by coordinates	at coordinates (5, 0, -8.0)

Note that the distance from the first to the third reference node is changed from 5.0 in the original block to 8.0 in the copied block; all of the dimensions in this direction are revised proportionally. The perpendicular distance remains unchanged in the copied block and all vertical dimensions will remain constant.

  - Define the number of copies and node/beam numbering increment:



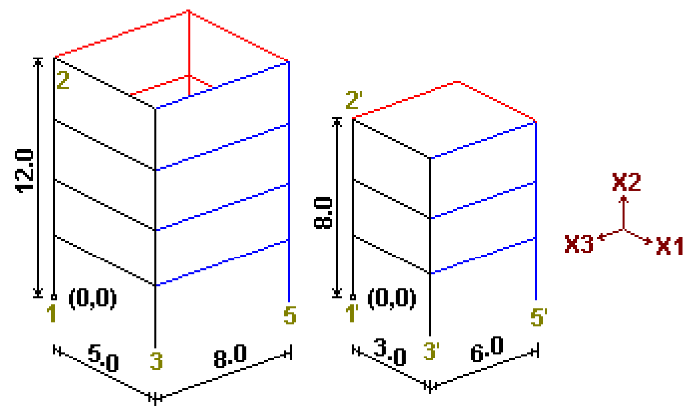
The program should create a node at the new location of node 1. However it checks whether there is an existing node at the location of each new node and if yes, it does not create the new node. Similarly, new beams are not created on the line 3-4.

- Create the remaining two planes:
  - select all the nodes in both frames using the standard Node Selection option.
  - select **Select 3 nodes** (Create identical copies)
  - define the three reference nodes and their new locations:

Ref. node:	No	Option:	Select/define:
1st	3	at an existing node	node 5
2nd	4	at an existing node	node 6
3rd	5	at an existing node	node 1

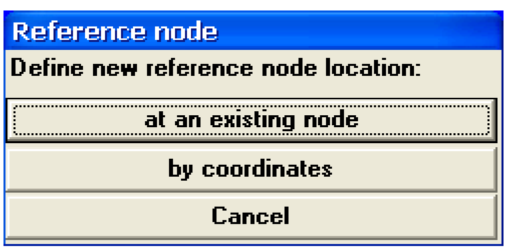
- set **Number of copies =1**

Create the smaller frame at the right from the frame of Example 1:



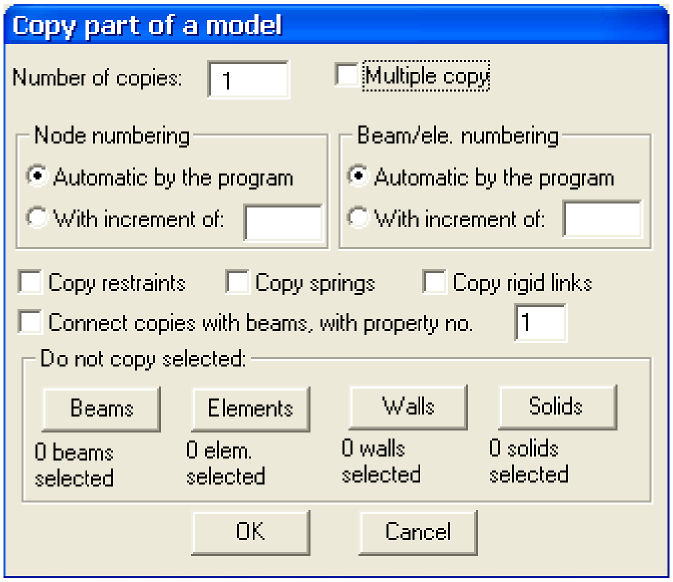
Define 1-3-5 as the reference nodes forming the plane and node 2 as the node in the perpendicular direction:

- select all the nodes in the frame using the standard Node Selection option.
- select **Select 4 nodes** (stretch/squeeze the copy)
- define the four reference nodes and their new locations:



Ref. node:	No:	Option:	Select/define:
1st	1	at an existing node	node 1'
2nd	3	at an existing node	node 3'
3rd	5	at an existing node	node 5'
4th	2	at an existing node	node 2'

- Define the number of copies and node/beam numbering increment:

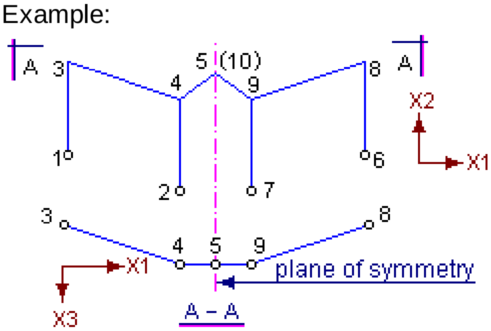


3.9.3 Mirror image

Create a mirror image of the original block about an axis of symmetry.

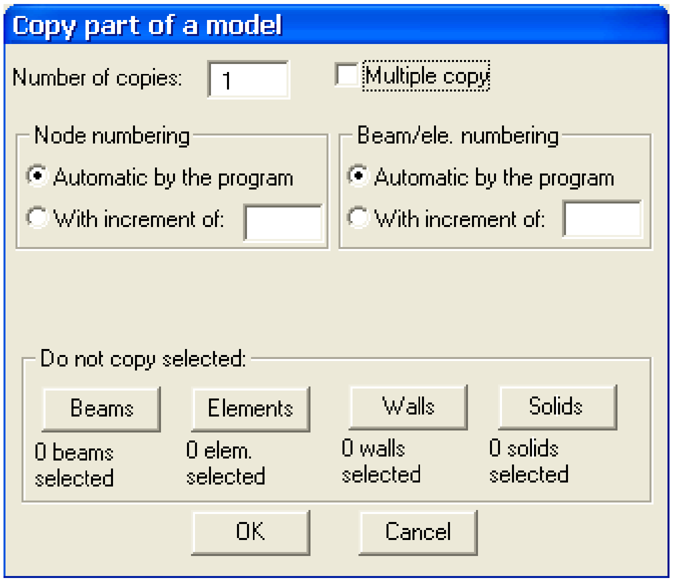
- Only one reference node is required; the program joins the old and new locations of the reference node with an imaginary line and passes a plane perpendicular to it through its mid-point.
- All of the selected nodes are recreated on the other side of this plane.
- The reference node must not lie on the plane of symmetry.

Example:



1-2-3-4-5 is an existing frame; create a mirror image about a plane through node 5.

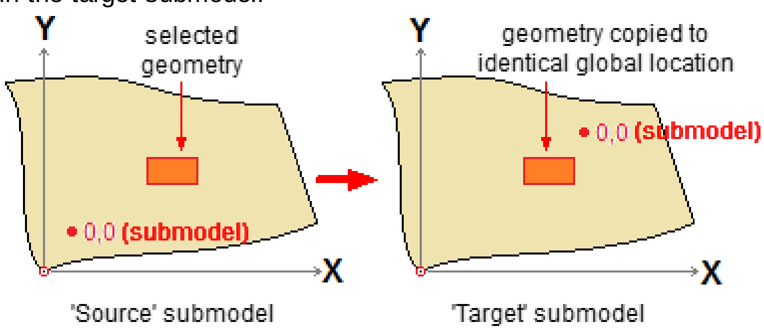
- define the block using the standard node selection option.
- select node 2 as the reference node.
- select the location of node 7 as its new location.
- Define the number of copies and node/beam numbering increment;



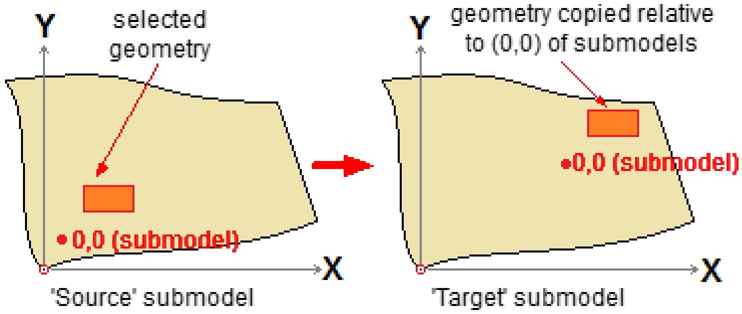
3.9.4 To submodel

Geometry from one submodel can be copied to another submodel

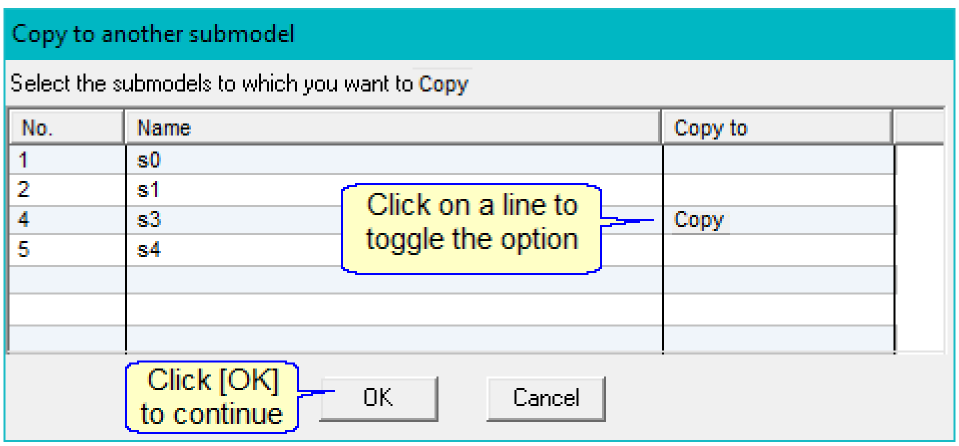
- the option is similar to Translate only 333.
- there is no reference point; the option should be used when the source submodel and the target submodel are both attached to the model ***and*** are stacked one above the other, i.e. they are at the same location in the plane perpendicular to the height axis.
- the selected geometry is copied as follows:
  - if both the 'source' submodel and the 'target' submodel are attached to the main model, the program copies the selected geometry to the same global coordinates, even if the (0,0) coordinates in the submodels are not at the same location. For example, if both submodels lie on the X-Y global plane, the selected geometry in the source submodel will be copied to the identical X,Y global coordinates in the target submodel:



  - if either of the submodels is not attached, the location will be relative to the (0,0) coordinates of the submodels:



When the source submodel is displayed:
- select a group of nodes using the standard Node Selection option.
- select one or more target submodels that you want to copy the geometry to:



Note:
- the program does not create duplicate elements only if they are completely identical in the target submodel. If, for example, the element density was changed in the source submodel and the area is copied to the target submodel, two sets of elements will be found at the same location in the target and the original ones must be erased manually.
- refer to the notes in Copy geometry 332 for other limitations.

3.9.5 General options

No. of copies

More than one COPY can be created with the same command; the offset to the next copy from the previous will be ***identical*** to the offset from the first copy to the original block.

Multiple copies

Use this option to create identical copies with ***different*** offsets.
- all copies use the same reference node in the original block; each copy prompts for a different new location of the node.
- this option can be used together with "No. of copies".

Node numbering

The node numbering increment is blank by default:
- if values are not specified the program will use the first available number for the first node in the Copy and then number ***consecutively***.

- if values are specified, the program will add the increment to the existing node numbers.

Beam numbering

The beam/element numbering increment is blank by default:
- if values are not specified the program will use the first available number for the first beam/element in the Copy and then number ***consecutively***.
- if values are specified, the program will add the increment to the existing beam/element numbers.

Copy restraints

☑ The program copies all restraints at selected nodes to the new locations.

Copy springs

☑ The program copies all springs at the selected nodes to the new locations.

Copy rigid links

☑ The program copies rigid links the new locations only if both the master and slave nodes are selected

Connect copies with beams

The program can connect the nodes in the original block to the corresponding nodes in the new block with beams. There are three options:

☑ a new beam will be generated for ***all*** nodes; specify the property number of the connecting beams
☐ no beams will be generated
Select nodes to connect — beams are generated only at the selected nodes; specify the property number.

Do not copy selected beams/elements/walls/solids

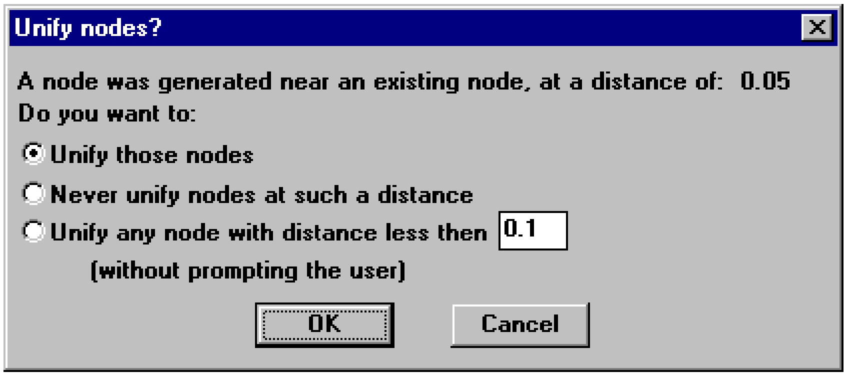
Select beams, elements, walls and/or solids that are ***not*** to be copied, using the standard selection options

Reference node

Define the new location of the reference node at:
- another existing node
- any coordinate (there does not have to be a node at the coordinate)

Unify nodes

After the copy is generated the program searches for existing nodes very near the location of new nodes:



Select one of the following options:

Unify
The program does not generate new nodes but connects all new beams/elements to the existing nodes

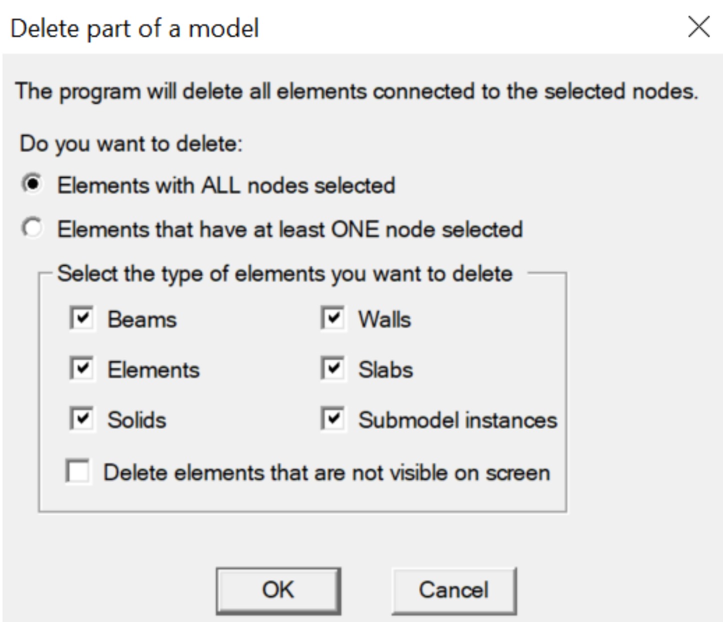
Never unify
The program generates new nodes and connects all new beams/elements to the new nodes

Unify with distance
Specify a new unify tolerance (initially 0.005 units). The program will unify all nodes where the spacing between the existing and new is less than the value specified

Note that the option selected here applied only to the current Copy command.

3.9.6 Delete part

Delete a part of a model with one command by selecting nodes and type filtering.



Elements with ALL nodes selected

Only elements with **ALL** end nodes in the window/polygon are deleted.

Elements that have at least ONE node selected

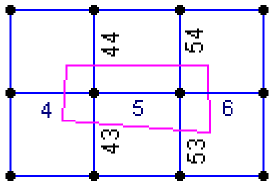
All elements with at least **ONE** end node in the window/polygon are selected

Select the type of elements you want to delete

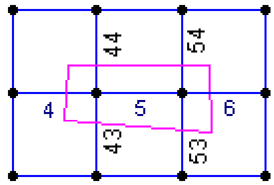
Filter the elements to be deleted by type.

Examples

- Beam element:

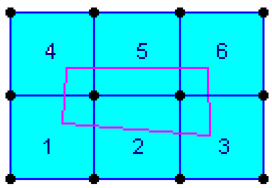


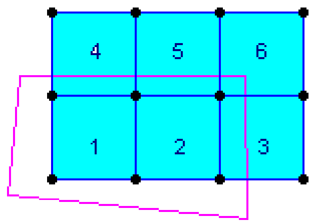
All nodes:
Beam 5 only is selected



One node:
all numbered beams are selected

- Elements:

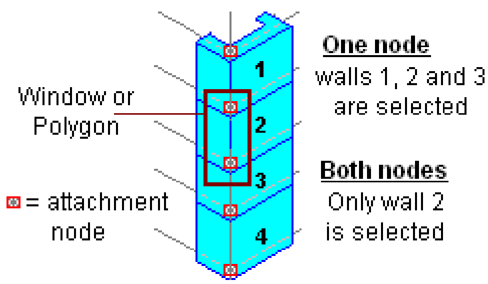




Elements selected:

All nodes: no elements	**All nodes**: elements 1 and 2
One node: all elements	**One node**: all elements

- Walls:

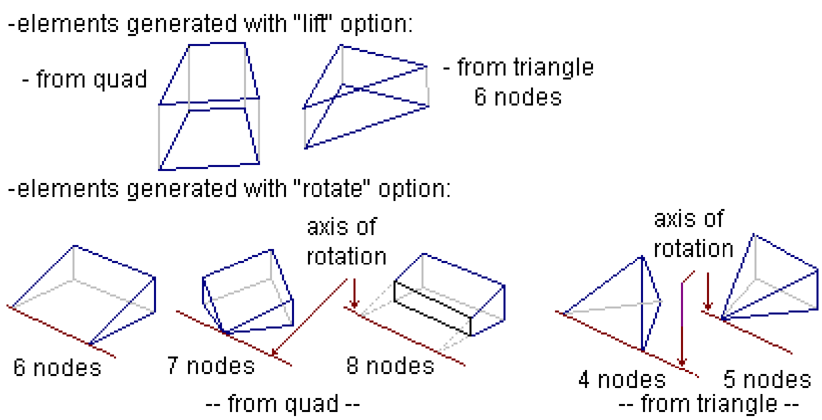


3.10 Solids

Solid elements are stress elements with actual thickness defined by the distance between end nodes. The element result types are stresses and principal stresses at the corner nodes.

The elements are defined by:
- specifying the corner nodes of a single element
- lifting or rotating existing quadrilateral or triangular plate elements to the nodes on the opposite face. The plate elements used to generate the solid elements may then be erased.

The generated elements may have 4,5,6,7 or 8 nodes:

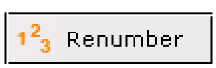


Note:
- solid element numbering is independent of beam/plate element numbering.
- **the Bridge postprocessor cannot solve models that include solid elements**

Generate solid elements by "lifting 345" existing plate elements. Refer to the Figure above for examples.

Generate solid elements by "rotating 346" existing plate elements. Refer to the Figure above for examples.

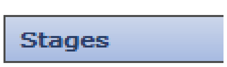
Delete existing solid elements; select the solid elements to delete using the standard element selection option.



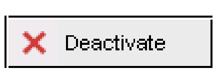
Renumber 347 existing solid elements

Define the material properties 349 for solid elements

Define a single element 349 by specifying the corner nodes

Click 

to select a stage other than **Whole model;** different properties may be defined for each stage and elements may be removed:



Solids may be removed from the current stage or restored.

Restore an element to the current stage

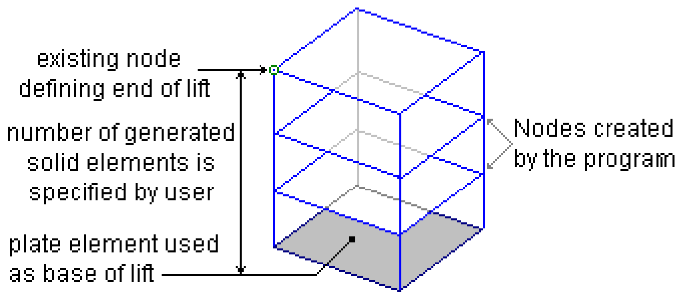
Note:
- Select solids using the standard element selection option.

- inactive solids are not displayed.
- new solids cannot be defined when a stage other than **Whole model** is active.

3.10.1 Lift

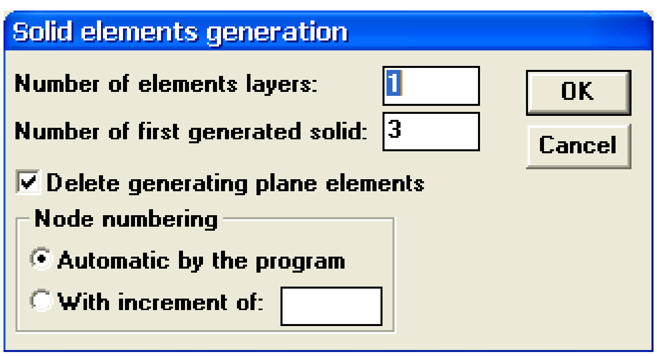
Create solid elements by "lifting" existing plate elements (quad or triangle) to a plane defined by an existing node. The distance between the plate element and the node defines the 3rd dimension and this dimension may be divided into one or more solid elements.

Note that this option also generates all required nodes. For example:



To define the solid elements:

- define the plate base elements if they do not already exist
- select the plate base elements to be lifted using the standard element selection option.
- select the existing node that specifies the end location of the lift
- specify the lift parameters:



Number of element layers

Define the number of parallel solid element layers to be generated between the base plate element and the reference node defined the end of the lift.

Number of first generated solid

Specify the number of the first element to be generated. Note that solid elements may have the same number as an existing plate element or beam.

Delete elements

☑ delete the plate elements used as the base for generating the solid elements.

☐ do not delete the plate elements

Node numbering

The node numbering increment is blank by default:

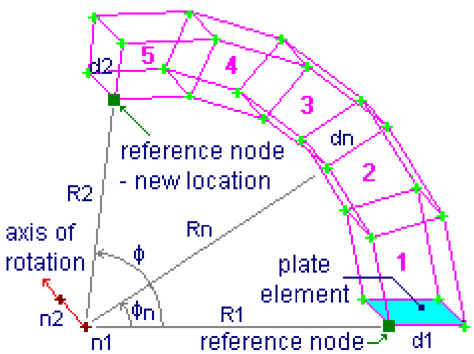
- if values are not specified the program uses the first available number for the first node in the Copy and then number ***consecutively***.
- if values are specified, the program adds the increment to the existing node numbers.

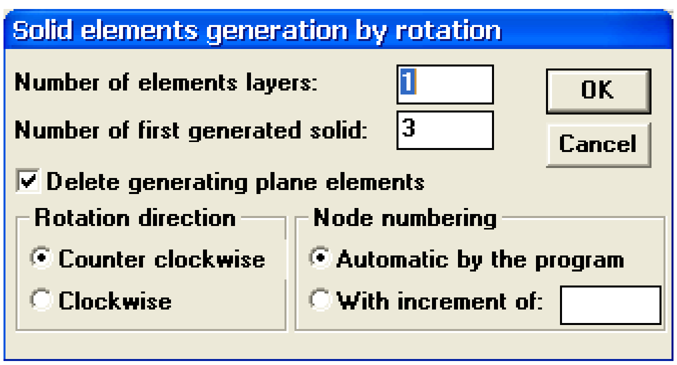
3.10.2 Rotate

Generate solid elements by rotating a plate element. The rotation parameters are defined by specifying:

- the axis of rotation - by two existing nodes (n1 and n2 in the following figure)
- a reference node and its new location
- the generation direction (clockwise or counterclockwise)
- the number of elements to generate

Example:





Number of element layers

Define the number of parallel solid element layers to be generated between the base plate element and the reference node defined the end of the rotation.

Number of first generated solid

Specify the number of the first element to be generated. Note that solid elements may have the same number as an existing plate element or beam.

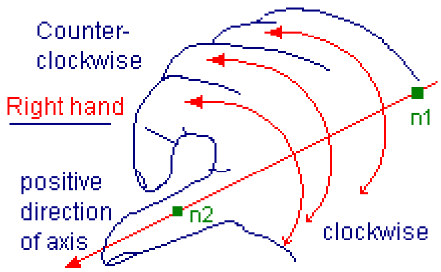
Delete elements

☑ delete the plate elements used as the base for generating the solid elements.

☐ do not delete the plate elements

Rotation direction

The program can generate the elements in either the clockwise or counter-clockwise directions. The direction is determined from the axis of rotation (defined by the two nodes n1 and n2) according to the following rule:



Node numbering

The node numbering increment is blank by default:

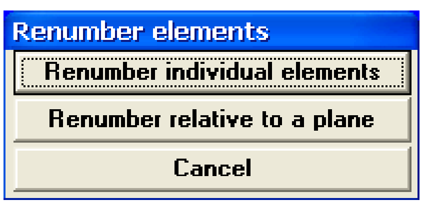
- if values are not specified the program uses the first available number for the first node in the Copy and then number ***consecutively***.
- if values are specified, the program adds the increment to the existing node numbers.

Note:

- the reference node n1 does not have to be a plate element corner node
- all elements will be identical if the distances R1 and R2 are identical, i.e d1=d2. If not, d1/d2=R1/R2 and the dimensions R_n and d_n at any node are proportional to ϕ_n/ϕ.

3.10.3 Renumber

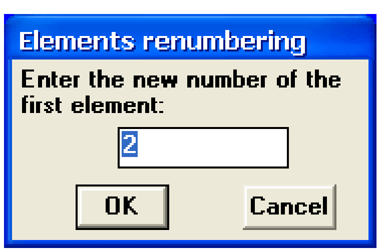
Renumber existing solid elements.



Renumber individual elements

To renumber selected elements:

- select solid elements using the standard element selection option. Note that the order that the beams are selected is important; they are renumbered in the order that they are selected.
- enter the new number of the first element:



Type the new number of the first element selected; all of the elements selected are renumbered sequentially. If the program discovers that a number has already been assigned to another element the program assigns the original number of the selected element to that element.

Example:

- elements 41, 42 and 43 are selected (in that order).
- 75 is specified as the new number for 41
- the elements are renumbered 75,76 and 77 respectively
- element 76 is an existing element; it is renumbered 42.

Renumber relative to a plane

Renumber all elements relative to a selected plane. This option is handy for renumbering an entire model or parts of a model consisting of more than one plane. Note that the planes do not have to be parallel.

- select the elements to be renumbered using the standard element selection option
- define a plane that specifies the renumbering order; the plane is defined by selecting three existing nodes.
- specify the new number of the first element.

The renumbering order is determined as follows:

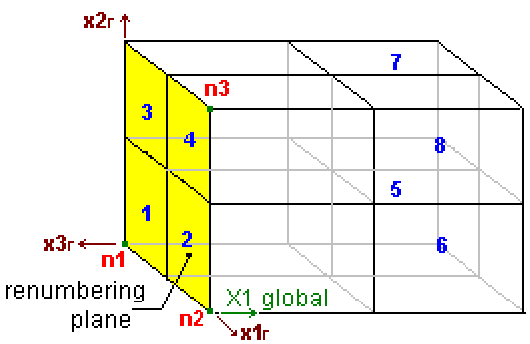
- the first two nodes define the x1r axis of the renumbering plane; the third node defines the x2r axis of the plane; the x3r axis is determined by the right-hand rule
- the program calculates the coordinate of the element centre and sorts the elements according to the center coordinate
- the program sorts the centers according to their x3r coordinate, starting with the centers closest to the renumbering plane. Note that if there are centers on both sides of the plane, the program first selects all centers on one side, then all centers on the other side.
- for centers with identical x3r coordinates, the program then sorts according to the x2r coordinate, beginning with the smallest value.
- for centers with identical x3r and x2r coordinates, the program then sorts according to the x1r coordinate, beginning with the smallest value.

All of the elements selected are renumbered sequentially. If the program discovers that a number has already been assigned to another element, the program assigns the original number of the selected element to that element.

Example:

Renumber the following space frame; the renumbering is to start on the planes perpendicular to X1 global

- select nodes **n1**, **n2** and **n3** to define the renumbering plane
- specify **1** as the new number of the first node



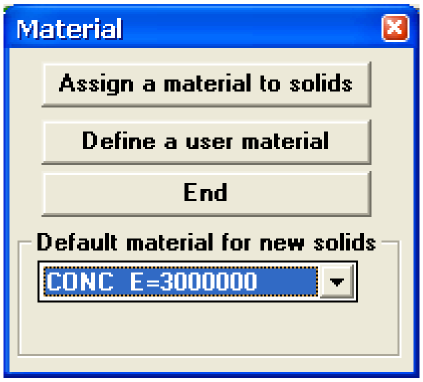
- the elements adjacent to the **x1r-x2r** plane are renumbered first (**1-4**); the bottom elements have the smallest x2r values and the left element has the smallest x1r value, i.e. it is renumbered first.
- then the elements on the parallel planes (**5-8**) are renumbered in the same order.

Note:

- solid element numbering is independent of beam/plate element numbering.

3.10.4 Material

Specify the default material for new solid elements, assign a different material to existing elements or define a new material (user material).



Assign a material to solids

To assign an existing material to existing solid elements:
- select a material from the list displayed
- select the solid elements using the standard element selection option

Note that the material number (from the Output - materials table) is displayed at the center of the element if you select **Display - property numbers** in the menu bar or click the  icon in the toolbar.

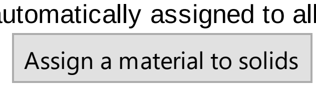
Define a user material

Refer to Beams - user material.

Default material

Select the default material from the existing material types in the list box.

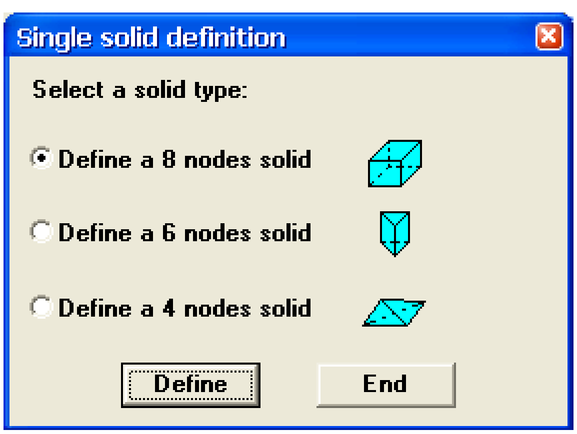
Note that the material selected and displayed here is automatically assigned to all new solid elements.

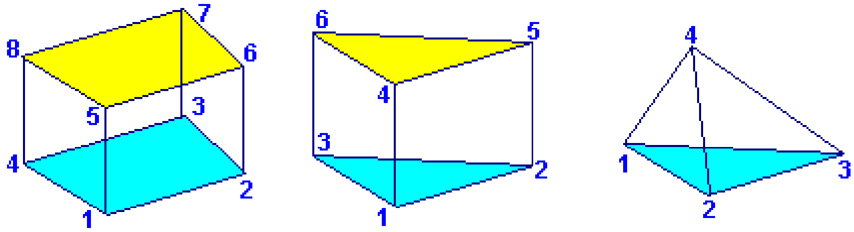
To assign the material to existing solid elements, click 

.

3.10.5 Single

Define a single solid element by selecting the existing corner nodes:

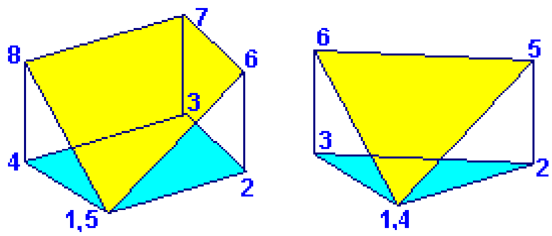




- 8 nodes
  Nodes 1-4 define the bottom plane; nodes 5-8 define the top plane.
  Node 5 must correspond to node 1, etc.
- 6 nodes
  Nodes 1-3 define the bottom plane; nodes 4-6 define the top plane.
  Node 4 must correspond to node 1, etc.
- 4 nodes
  Node 1-3 define the base of a pyramid.
  Node 4 defines the apex.

Note:

- the order of node selection (on any plane) is not important.
- the program checks whether the nodes that form a 4-node 'plane' are all on the same plane
- to define a 7-node element, define an 8-node solid, where one of the nodes on the top plane is the same as one of the nodes on the bottom plane.
- to define a 5-node element, define a 6 node solid, where one of the nodes on the top plane is the same as one of the nodes on the bottom plane.



3.11 Walls

Define wall elements consisting of one or more segments and attach them to nodes in the model. For more information, refer to Walls - general 351.

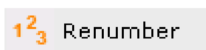
Add a single wall element 354 between two nodes

Add a series of identical wall elements 354 attached to a line of nodes

Define the section 355 of a wall element consisting of a series of connected wall segments with or without openings.

Delete wall elements from the model; select walls to delete using the standard wall selection option.

Rotate wall elements 370 about their "reference point" (elements will remain attached to the same nodes)



Renumber 371 wall elements

Create rigid links 372 connecting the wall elements to nodes that are located within the width of the wall segments.

The following options are available when a stage other than **Whole model** is the current stage:

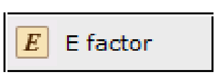
Stages

Click 

to select a stage other than **Whole model;** selected walls may be removed from the model in a stage.

Walls may be removed from the current stage or restored.

Restore a 'removed' wall to the current stage.

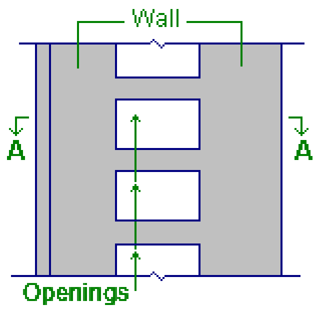


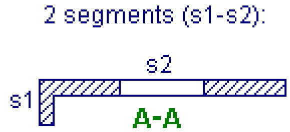
Define the wall stiffness reduction factor 355 applicable to this stage.

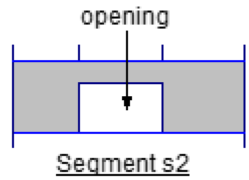
Note:
- Select walls using the standard wall selection options
- inactive walls are not displayed
- new walls cannot be defined when a stage other than **Whole model** is active

3.11.1 General

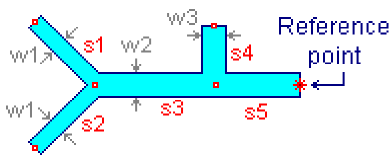
The Wall option enables the quick definition of complex walls that extend across multiple levels in the model.
- the wall cross-section is defined first and then is attached to the model by selecting a vertical line of nodes; the program automatically creates any additional nodes that are required
- each wall section may consist of multiple segments together with openings, for example:

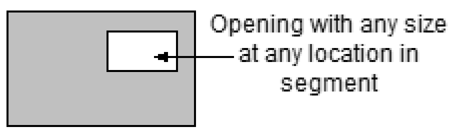




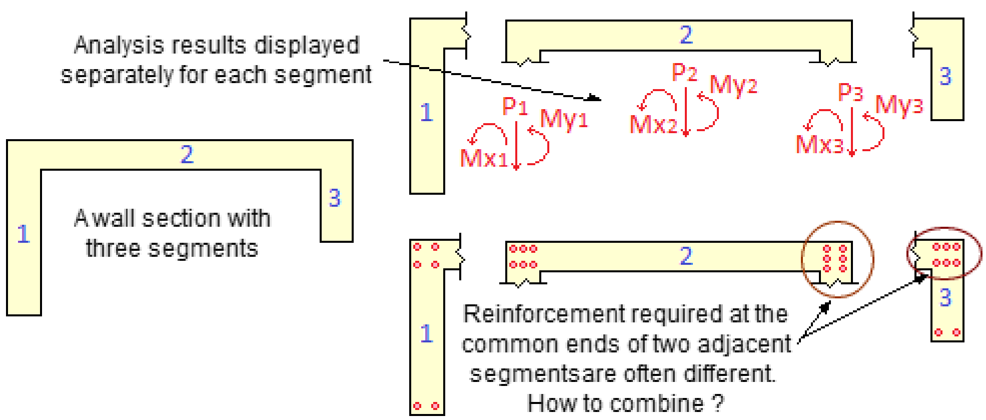


- the number of segments is unlimited, the arrangement and orientation of the segments is not restricted. Each segment may have a different width. For example:

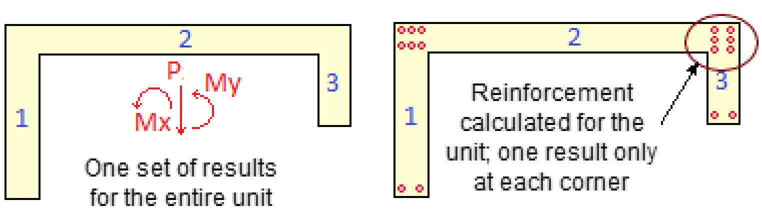




- each opening may have different dimensions and may be located anywhere with the wall segment (vertically and horizontally)
- each wall has a "reference point" at the end of any segment. This point is used to attach the wall to nodes in the model (referred to as the "Attachment nodes").
- segments may be combined to form "design units". Creating these "units" consolidates the results and gives simpler and more economical reinforcement detailing. For example:



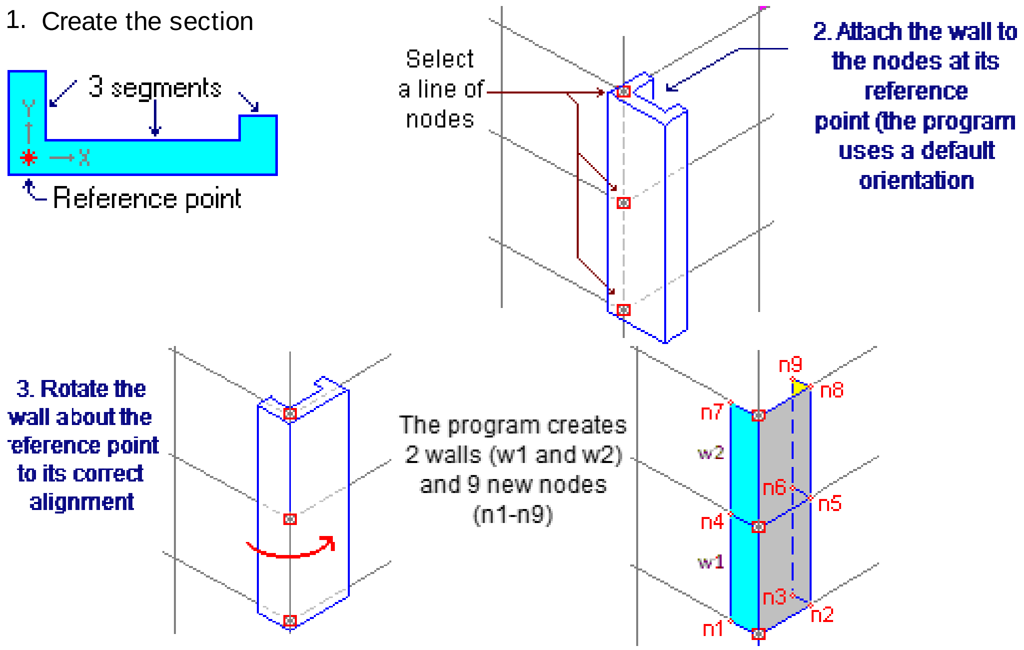
If the segments are combined into one "design unit":



When the wall section is attached to the model, the program generates a series of walls and creates the necessary nodes at the corners of each segment if they do not already exist.

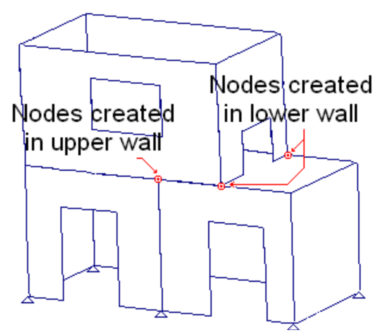
Example: the following wall is defined and attached to the model:

1. Create the section

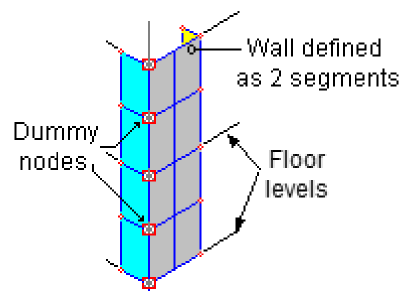


Note:

- additional nodes are automatically added at the top/ bottom faces of the wall when it connects to a wall with a different cross-section above or below:

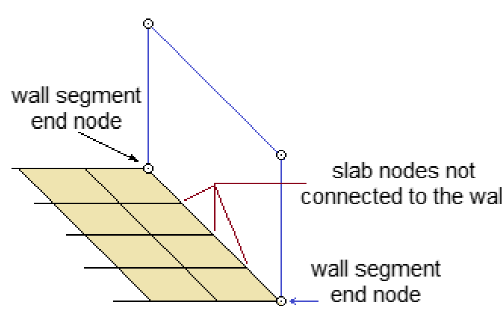


- The program creates a single element between nodes along the height axis for each segment. Normally this implies a single element per storey height. Testing has shown that the results are sufficiently accurate for *typical long multi-storey walls*, i.e. more than 4-5 storeys (refer to the *STRAP* Verification manual). For short walls, improved accuracy will be obtained if dummy nodes are defined between floor levels and long segments are defined as more than one segment.



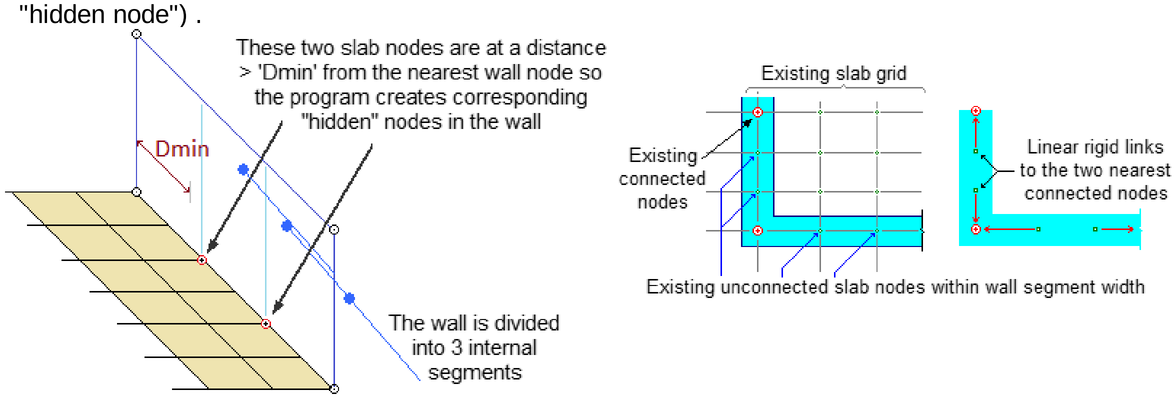
- The wall segments are connected to the adjacent parts of the model only at their end nodes. However there may be existing nodes in the model that lie along the wall segment boundary but are not connected to the wall. For example:

  The wall may not move together with the slab along the boundary when either is loaded.There are two ways to deal with this problem:



  - create rigid links between the unconnected nodes and the wall end nodes. However, results become inaccurate in long wall segments with many rigid links

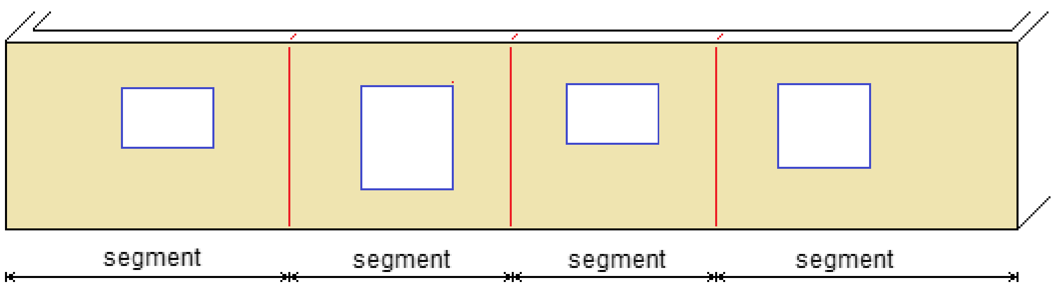
- create 'hidden nodes' in the wall segment at the unattached model node locations, and then connect the wall to the model at those nodes. The program can automatically create these 'hidden nodes' so that no unattached slab node is further than "**Dmin**" from a wall node (a corner node or another "hidden node") .



- The results for wall elements are presented in the form of beam results - one axial force, moment and shear value for each generated element.
- Walls from older versions of *STRAP* (up to Version 2011) with "coupling beams" are automatically converted to walls with openings.

Note:

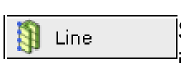
- only one opening can be defined in a segment. A continuous wall segment with no perpendicular segments and several openings must be defined as several continuous segments:



3.11.2 Line

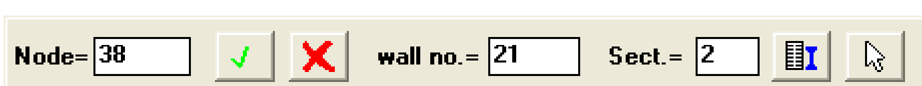
Select a series of "attachment" nodes where the wall section reference point will be attached to the model. There are two options:

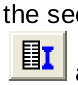
Select the start node and the end node; the program will create only 1 wall element between the two nodes



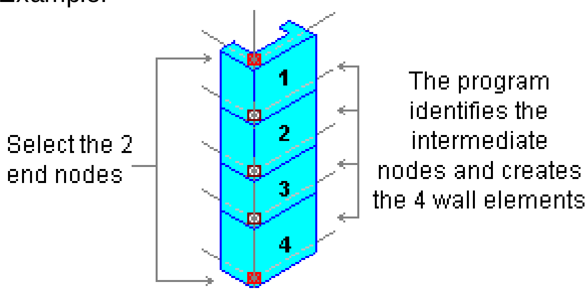
Select a start and end node; the program will identify all of the nodes along the line joining them and create a series of wall elements

For both options, the Wall section to be attached is specified in the dialog box at the bottom of the screen:



- type the section number in the **Sect.=** box, or -
- click  and select a section from the table.

Example:

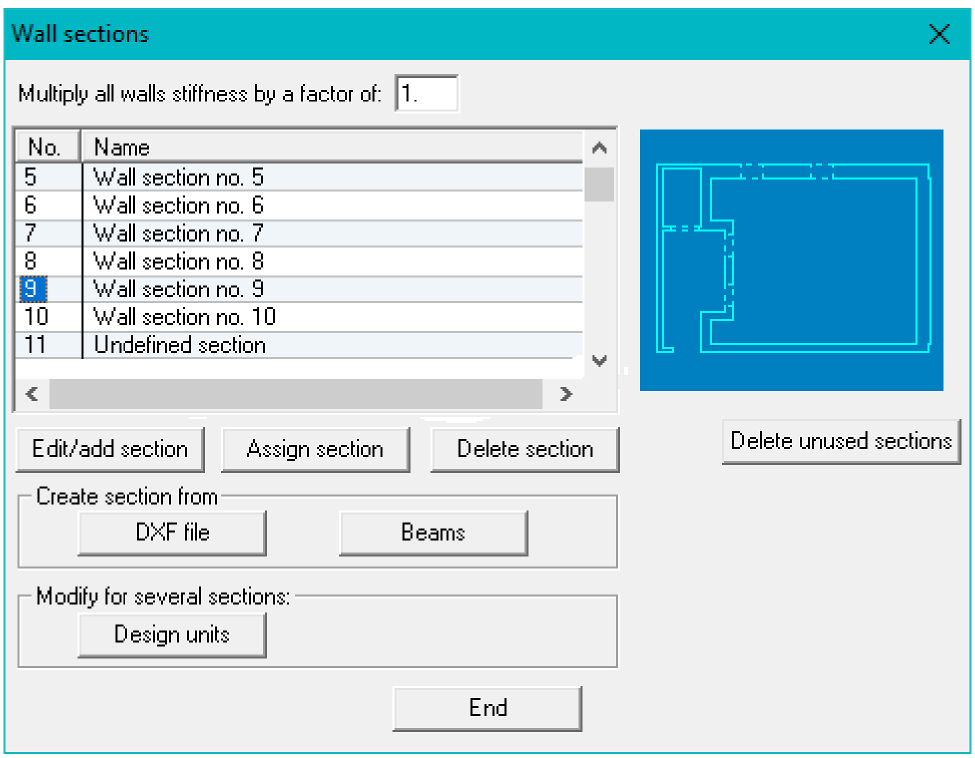


3.11.3 Section

Define/edit a wall section or assign an existing section to an existing wall.

- wall sections may be retrieved from a DXF file 365.
- wall sections may be defined along 368 the center lines of existing beams.

Click and highlight a section and select one of the options:



Delete unused sections

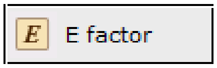
"Unused" sections are sections that are not found in the model. This option deletes them from the section list.

3.11.3.1 Factor

Reduce the wall stiffness by a factor, as required by many seismic and wind design codes.

The stiffness is modified for **all** walls and **all** load combinations. To reduce the stiffness only for the horizontal loads:

- create a Stage containing the entire model and assign all gravitational loads to it; click on



and define a wall stiffness factor = 1.0 for the Stage.

- The 'Whole model' should contain only the horizontal load cases. Specify a wall stiffness factor < 1.0 for the 'Whole model'.
- Carry out the modal analysis on the 'Whole model' (Modal analysis cannot be carried out on a Stage).
- Combine the results from the 'Whole model' and the Stage.

Note:

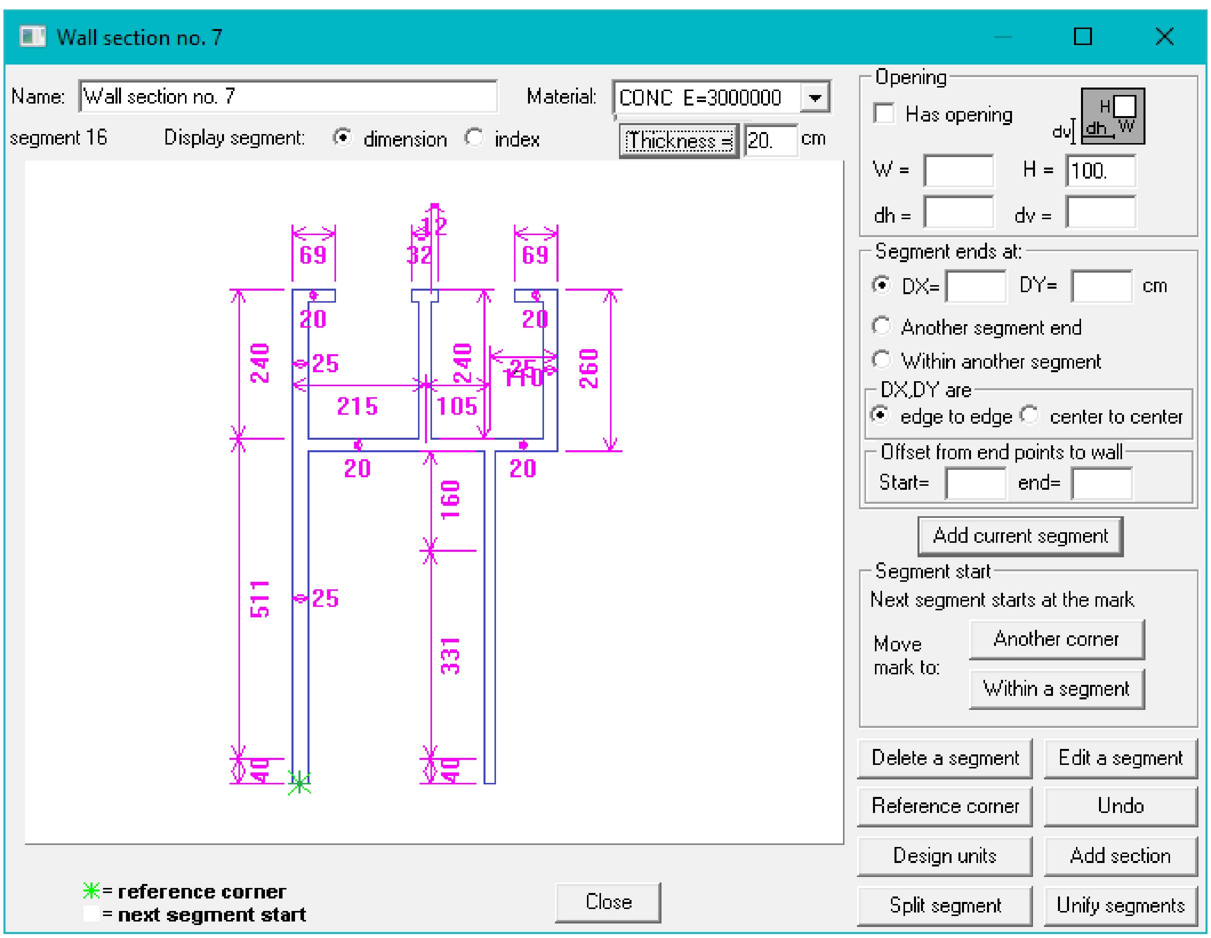
- the program modifies the modulus of elasticity (E), therefore the shear modulus of elasticity (G) is also modified. The wall element is a shell element, therefore both the bending stiffness and the in-plane stiffness (axial forces) are reduced.

3.11.3.2 Edit

Define a wall section consisting of several connected segments; each segment may have an opening.

- each segment may have a different width
- each additional segment must be connected at least one end to an end of a different segment
- segment end points may be defined at a coordinate, at the end of another segment, or anywhere along another segment
- segments can be combined to form "design units"; different design units can be defined for moments and shear
- a wall may be defined with up to 80 design units; each unit can contain up to 200 segments.

Refer to Examples 375 for detailed explanations, or:



Name

Revise the default title for the current wall section.

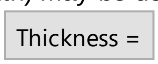
Material

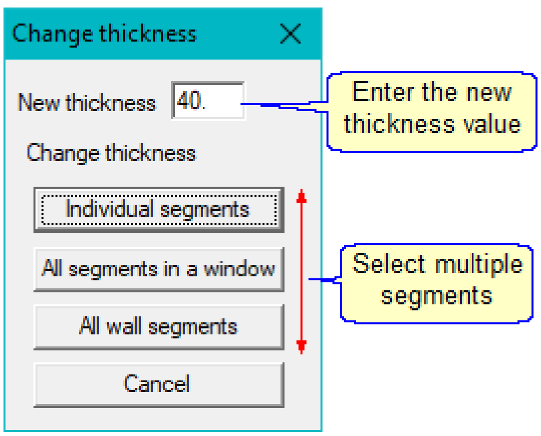
Select a material from the list box. New materials can be created in the Beam/Element option.

Display segment dimension/index

Each segment in the wall has an index number. The number of the current segment is displayed at the left above the section ("16" in the example above). The index numbers for all segments can be display by selecting **index** instead of **dimensions**

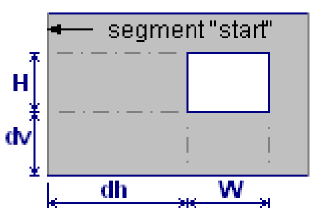
Thickness

A different thickness (width) may be defined for each wall segment. Enter the thickness value for the current segment or click  to revise the thickness for several segments:



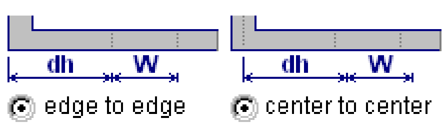
Opening

One opening can be defined in each segment; the opening can be of any size and can be located anywhere in the segment

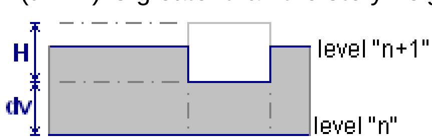


Note:

- **dh** is measured from the 'start' of the segment, either from the outer face if or from the centre:

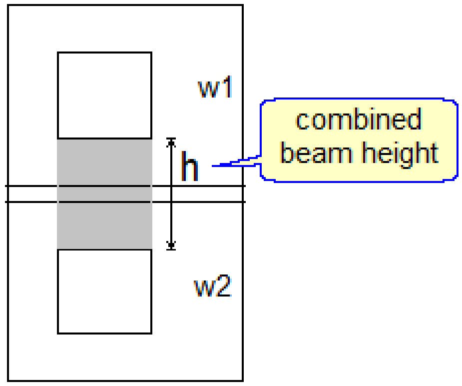


- if (**dv+H**) is greater than the story height, the program builds the model as follows:



Note:

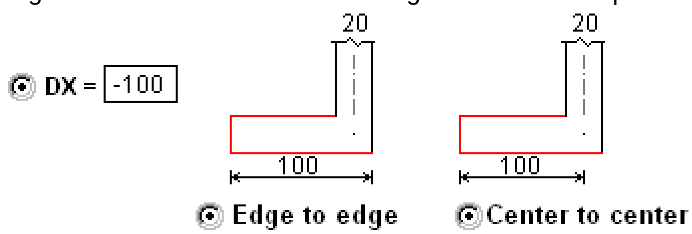
- The concrete design module designs the parts of the wall above and below the opening as beams. If there are openings in the middle of both segments above and below a level, the program combines the two resulting beams to form a single beam for design:



Segment ends at

Each new segment may be defined to end at:

- an offset from the segment start (DX, DY). The distance specified may be measured either from the edge or the center of the current segment. For example:



- at the the start/end point of an existing segment
- anywhere along an existing segment, at a specified distance or perpendicular to either the current segment or the existing segment.

Refer to examples 375

Segment start

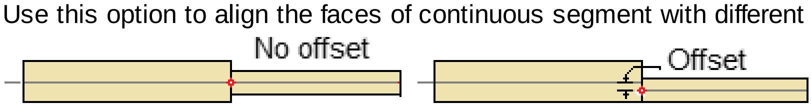
Each new segment may be defined to start at:

- the end of the previously defined segment (default)
- at the the start/end point of an existing segment
- anywhere along an existing segment

Refer to examples 375

Segment offset

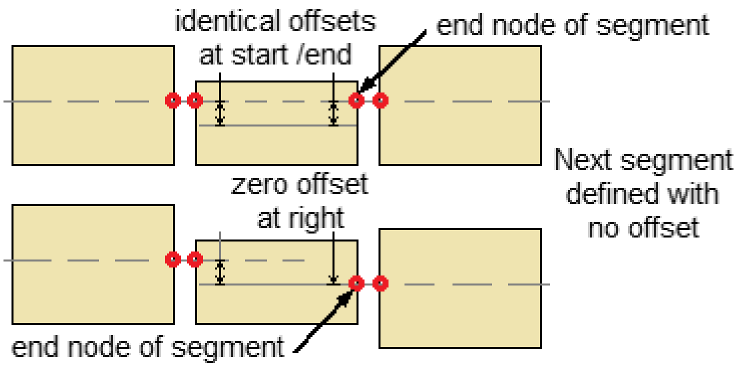
Use this option to align the faces of continuous segment with different thicknesses. For example:



Note:

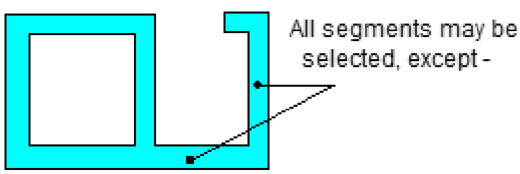
- that the 'start' node of the segment is at the same location as the end node of the previous segment
- the 'end' offset defines the segment end nodes location. A zero offset at the right places the end node on the segment center-line.

For example:



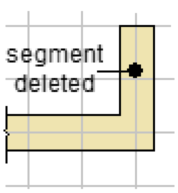
Delete

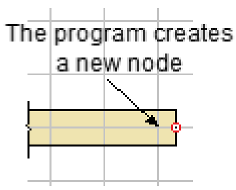
Delete one of the segments in the wall. Note that only segments that do not break the continuity of the wall may be selected. For example:



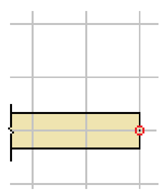
Move the so that the segment is highlighted with the ■ and click the mouse.

If you delete the end segment of a wall, the following problem may arise:





To end the previous segment at an existing node:

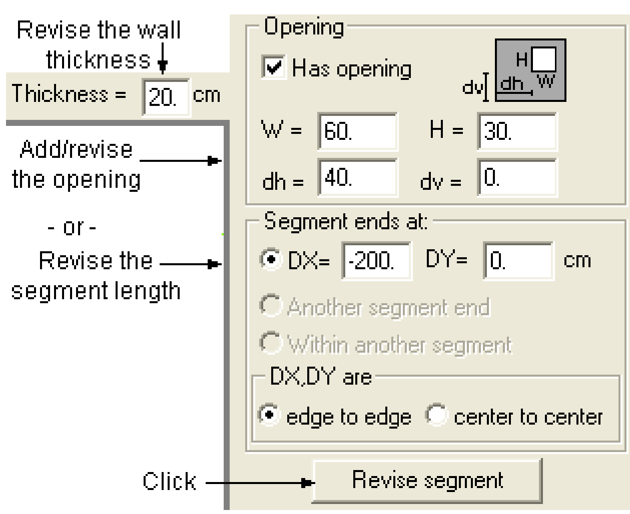


Select:
center-to-center
before deleting

Edit a segment

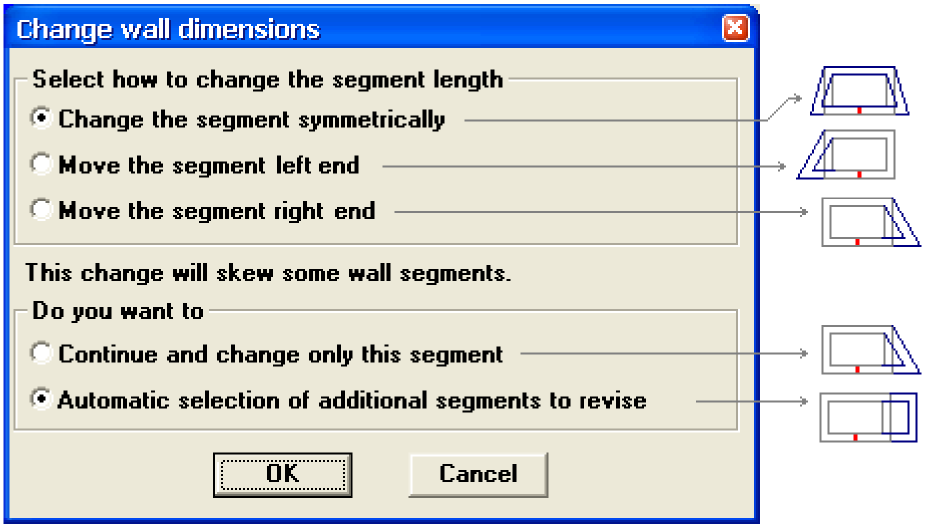
Edit the segment properties or dimensions:

- move the so that the segment is highlighted with the ■ and click the mouse.
- the program displays the data for the selected segment:



- if the segment is part of a closed figure:

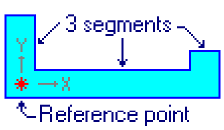
- either the start or the end can be moved or the length change can be divided equally between both points
- the program can automatically identify and modify other segments to maintain the same general shape.



Reference point

Each wall section is defined with a 'reference point' located at the end of one of the segments. The wall is attached to the model at this reference point.

The reference point can be moved to any segment start/end point in the section; move the so that the point is highlighted with the ■ and click the mouse.



Refer to examples 375.

Undo

Click this button to undo the previous action, e.g. delete the last segment defined, restore a deleted segment, etc.

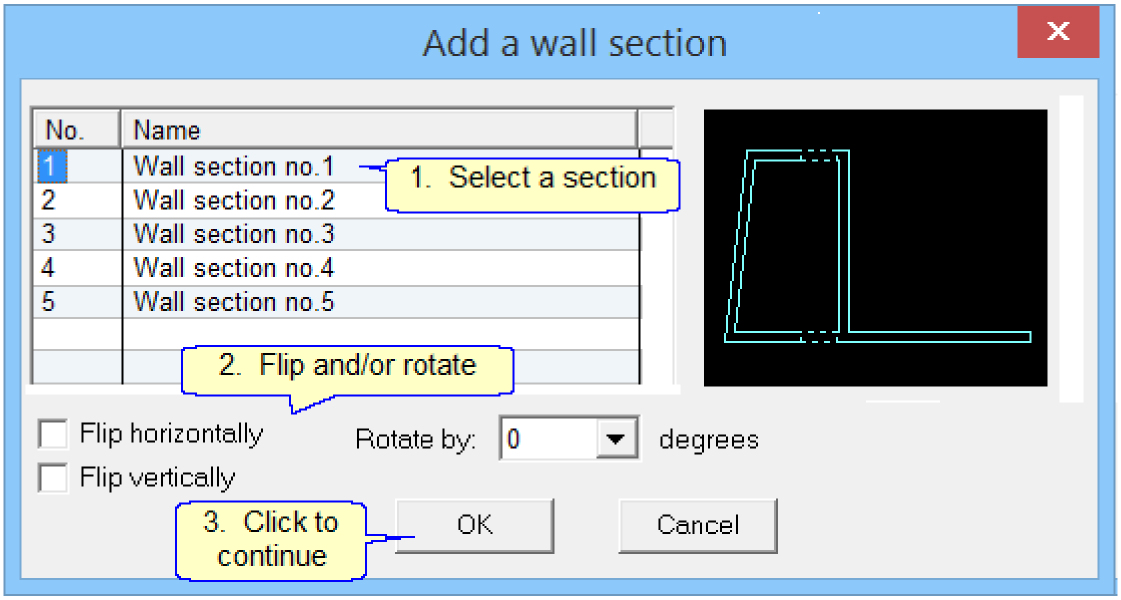
Design units

Refer to Design units 361

Add section

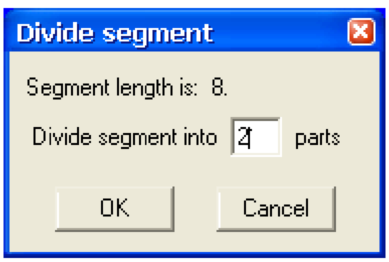
Paste an existing wall section into the current section:

- The reference corner of the added section will be attached at the current mark location; use the **Move mark to** options to select a different mark location.
- The selected section can be rotated or flipped when attached at the mark location.
- select another existing wall section in the model:



Split a segment

Divide a segment into two or more equal segments:



Note:

- one set of results may be displayed for the split segments. Refer to the Colinear360 option.

Unify segments

Combine segments of equal width that lie along the same line, e.g. segments that were formed using the "Split" option.

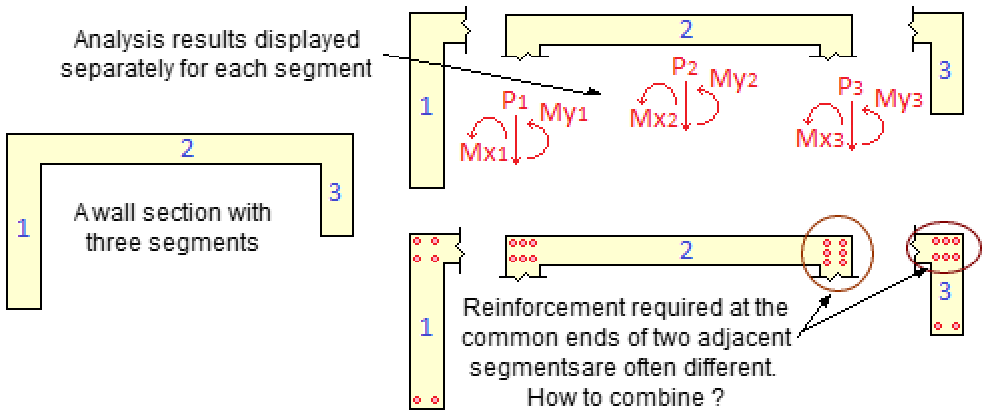
- select the first and last segment in the chain that are to be unified.

3.11.3.2.1 Design units

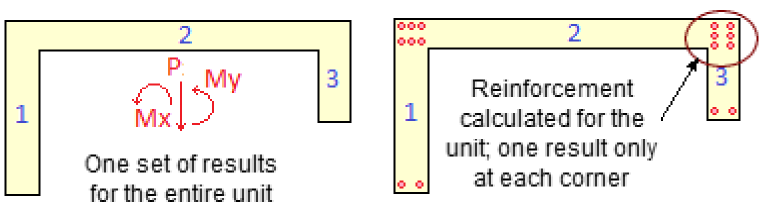
Segments may be combined to form "design units". Different design units can be defined for moments and shear.

- **Moments**:

  Creating these "units" consolidates the results and gives simpler and more economical reinforcement detailing.



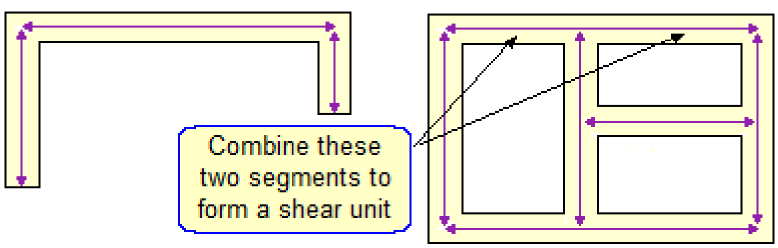
If the segments are combined into one "design unit":



A small wall, such as the one shown above, should be combined into one moment design unit.

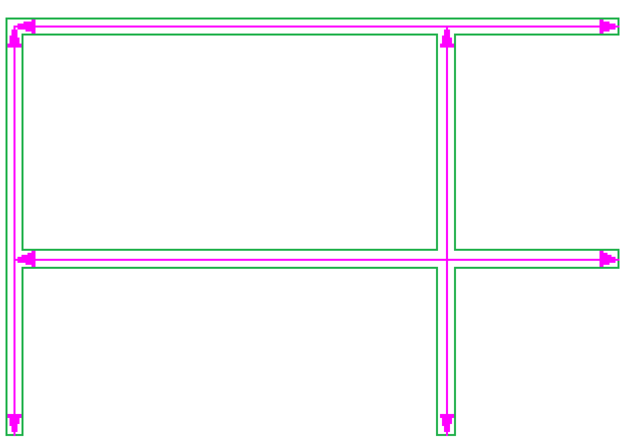
- **Shear**:

  Continuous linear lines of segments should be combined to form shear resisting units. For example:



Default design units:

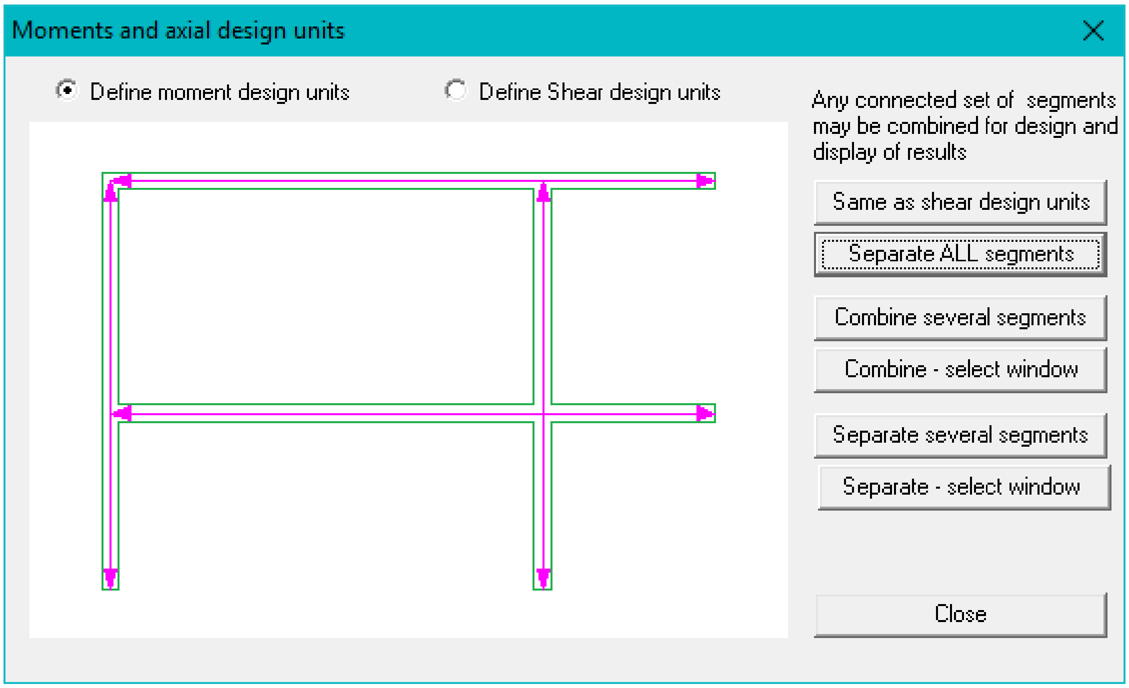
Each straight line of segments is initially assumed to create a single unit for both moments and shear. The units are shown on the drawing as . For example:



This is generally a conservative assumption; the wall should be converted to a single design unit if the designer believes that the wall acts as a single unit. The Codes do not give guidelines as to when a wall can be designed as a single unit.

To modify the units:

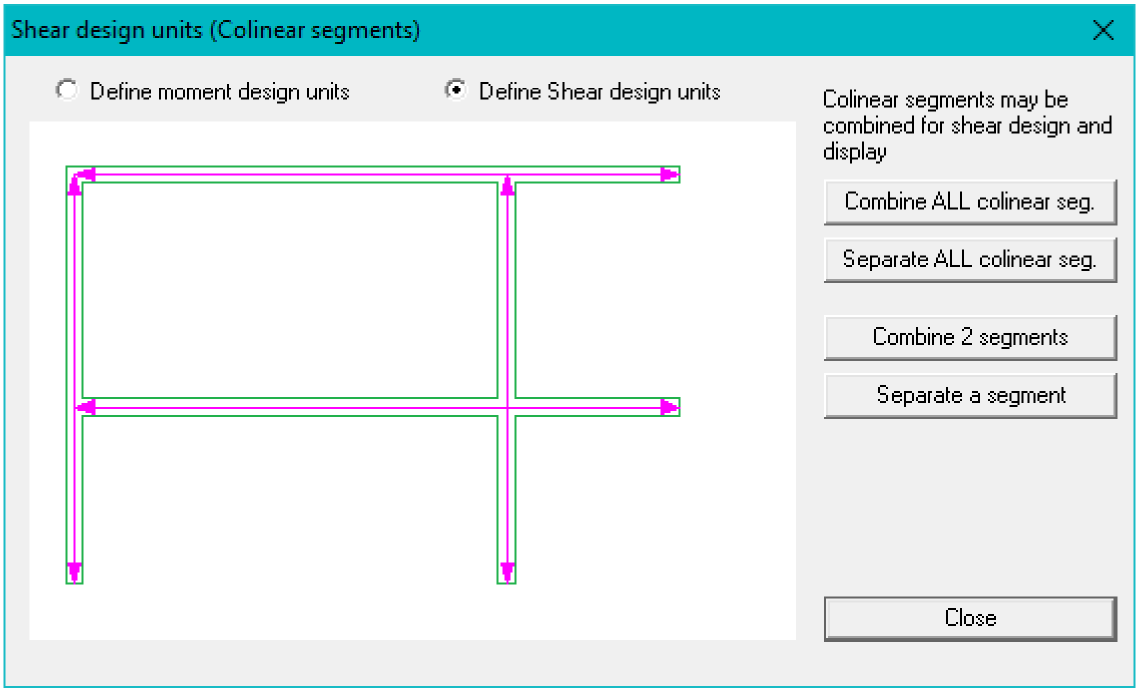
- Moment units:



Select:

Same a shear design units	-	Make the moment design units identical to the shear design units
Separate all segments	-	Each segment will form a design unit (this is not the default configuration)..
Combine several segments	-	Highlight and click on the segments to combine.
Combine - select window	-	Create a window around the segments to combine. Both the start and end of a segment must be enclosed by the window.
Separate several segments	-	Highlight and click on the segments to separate.
Separate - select window	-	Create a window around the segments to separate. Both the start and end of a segment must be enclosed by the window.

- Shear units



Select:

Combine all colinear seg. - The program searches for all colinear segments and combines them automatically

Separate all colinear seg. - The program searches for all combined segments and separates them

Combine 2 segments - Select colinear start and end segments to combined

Separate a segment - Select a combined segment to separate.

3.11.3.3 Assign

Assign a wall section (may be "***Undefined***") to a defined wall:

- click and highlight a section
- click Assign
- select walls using the standard Wall selection option

3.11.3.4 Delete

Delete a wall section:

- click and highlight a section in the list
- click Delete

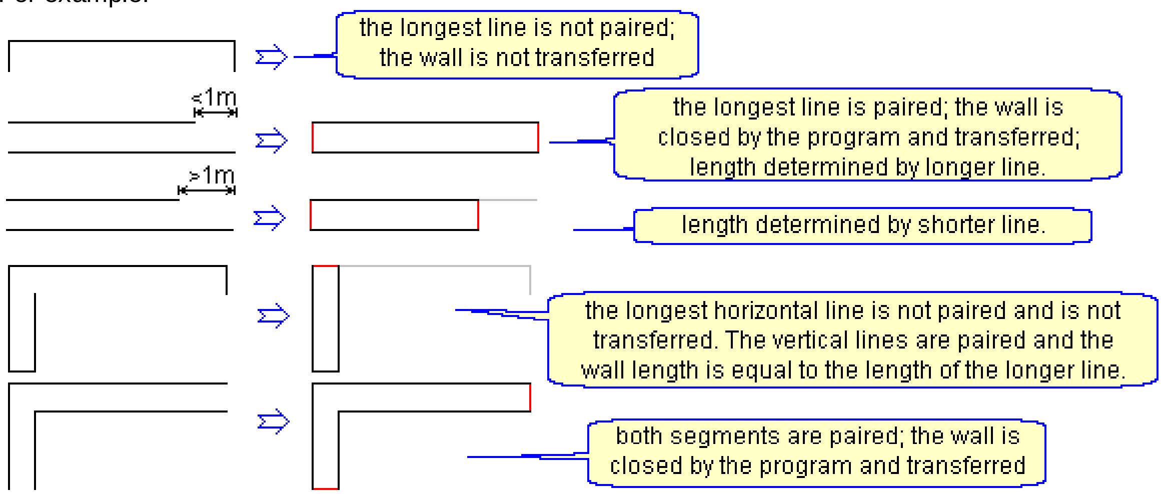
If the section was added to the model, the program requests confirmation prior to deleting; the wall will be deleted from both the model and the list.

3.11.3.5 Import from DXF

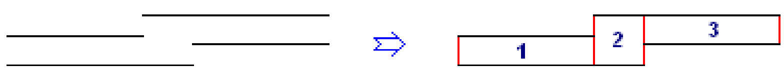
Import wall sections from a DXF drawing:

The program identifies parallel lines in the selected layers and assumes that they form walls.
- the walls do not have to be closed
- the longest line must be paired in a chain of lines or else it is ignored
- the length is determined by the longest line in the pair if the length difference is less than 1 meter; otherwise it is determined by the shortest line.

For example:

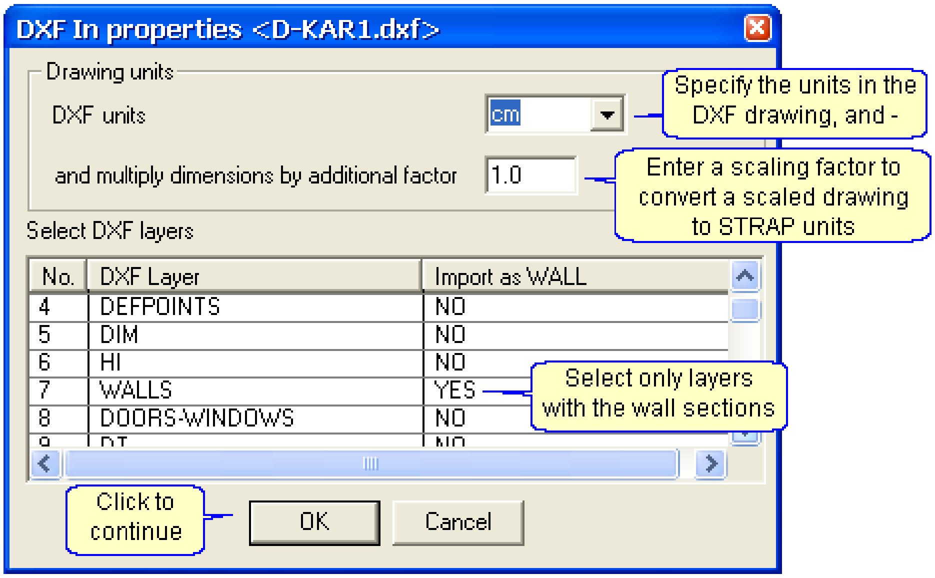


The program also checks whether there are parallel pairs above or below any pair. For example, the followed two pairs creates three wall segments:

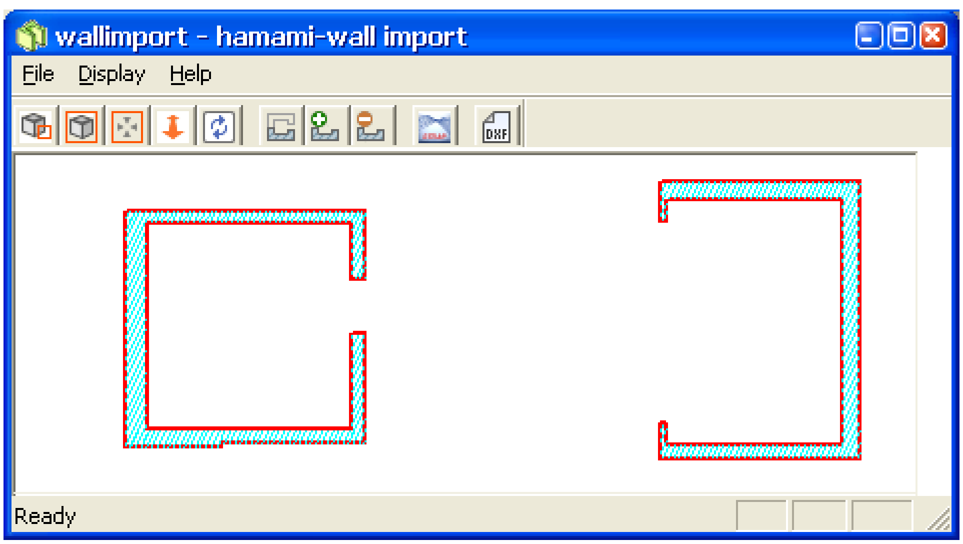


Note:
- all wall sections or selected sections in the DXF drawing may be imported simultaneously.

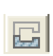
To import the sections:
- select the file
- select the layers to import and specify the DXF units and scaling factor



The program displays ***only the wall sections that it identifies*** in the selected layers, for example:



Select one of the following options:

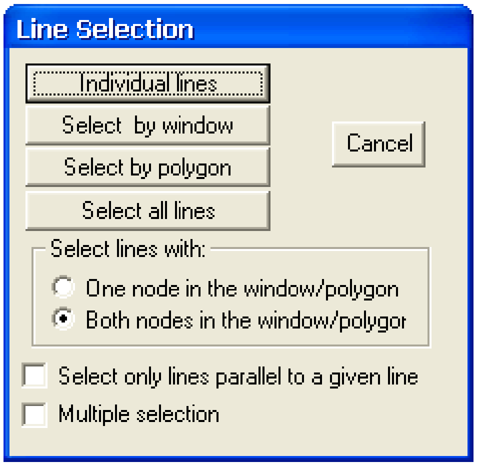
 Display ***all*** of the other lines in ***all*** of the layers as background lines.

 Convert background lines to wall sections or restore deleted wall sections. They program automatically adds the background lines to the display.

 Delete selected lines (convert to background lines). If individual lines in a chain are deleted, the program recreates the sections from the remaining lines according to the rules above.

 Transfer selected sections to the wall section menu. Each individual section selected is transferred as a separate section in the wall section menu.

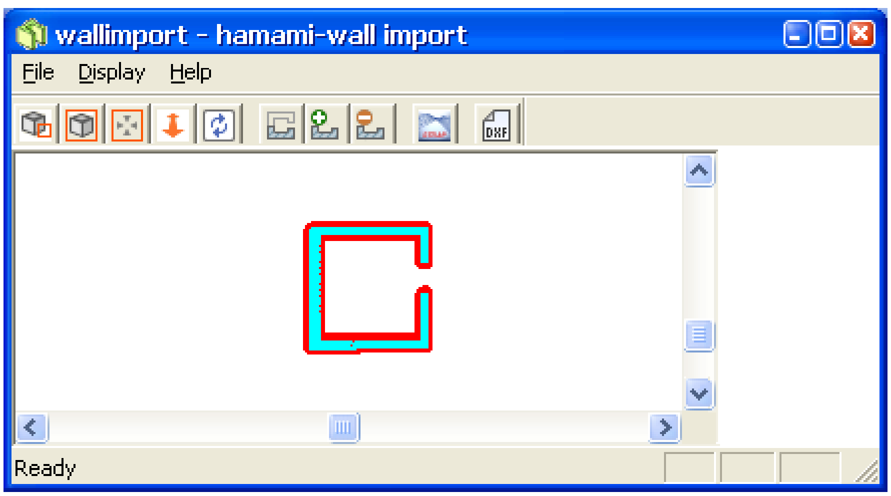
For all of the above options, select one or several lines (similar to the standard beam selection option):



For example, delete the section at the right:

- click 
- create a window around the section

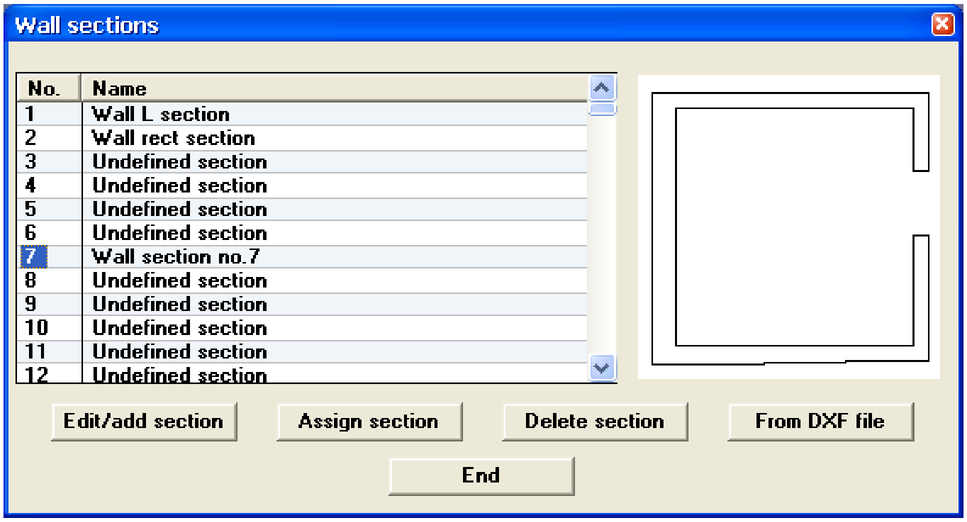
The program displays the remaining section:



To import the section:

- click 
- if there is more than one section, select one or more or click Select all lines to select all of them (the selection menu is not displayed if there is only a single section).

The program displays all selected sections (in this case - one) in the wall sections menu:

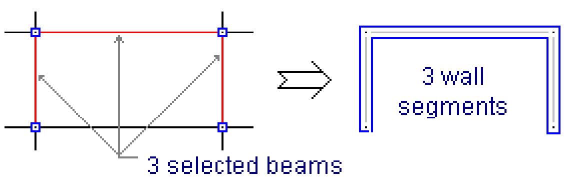


Note:

- if more than one section is selected they are inserted consecutively in the table (in the example above - no. 7,8,9 ...).
- if there is an existing section at any location in the list the program asks whether overwrite it. If No is selected, the section is ***not*** added to the list.

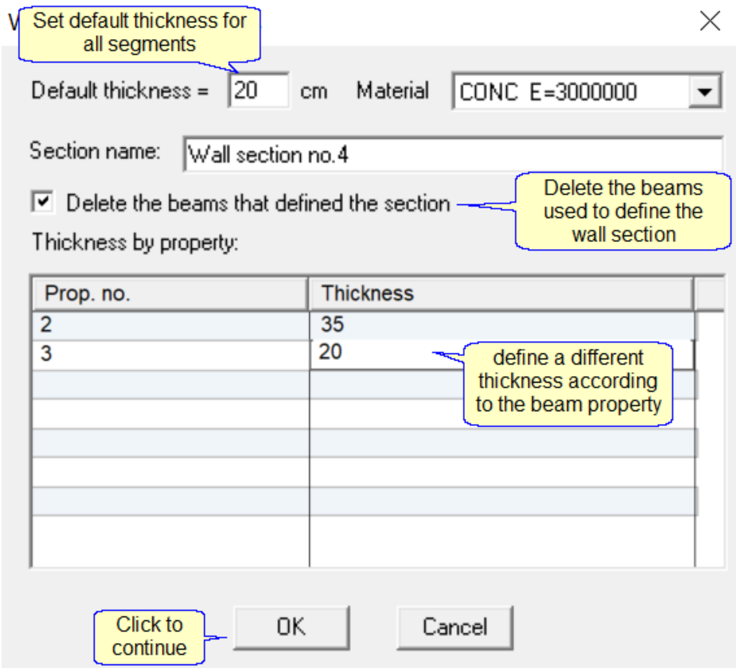
3.11.3.6 Defined by beams

A wall section may be defined according to existing beams: each selected beam forms a segment of the wall section. For example:

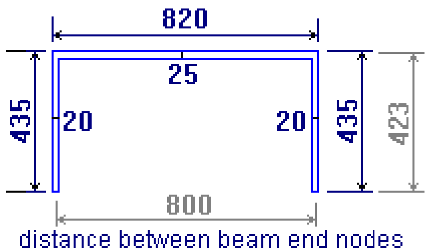


To define a wall using this option:

- selected an **-undefined section-** and click Beams
- select the beams using the standard beam selection option (all selected beams must be connected).
- the program identifies the beam properties; the wall segments associated with each property may be defined with a different thickness at this stage.



- The program creates the wall section:



- the wall may be modified further; refer to Section - edit 356.

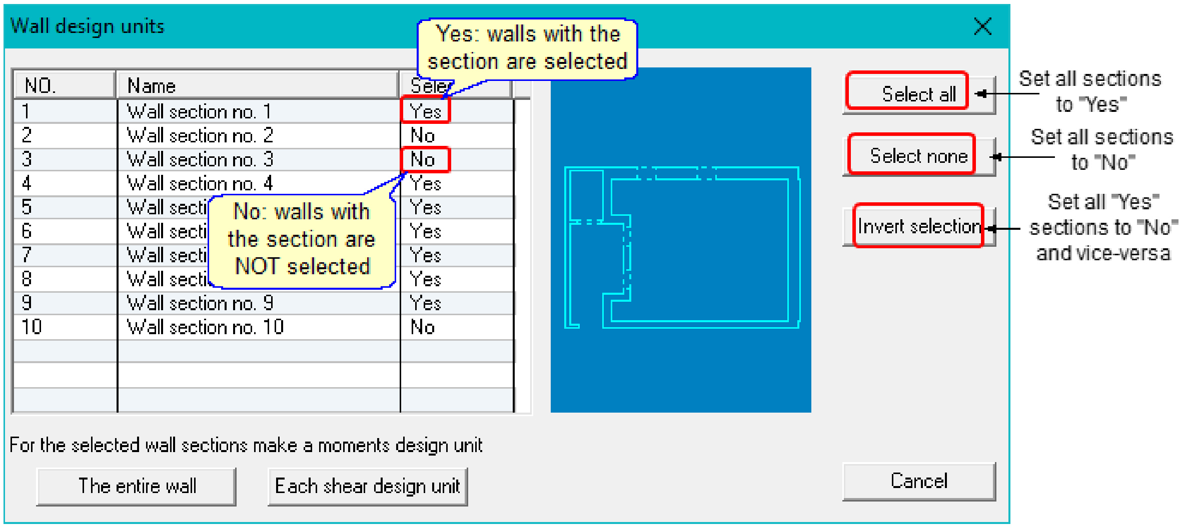
Note:

- the beams define the wall center-lines; the end segments are terminated at the beam end nodes. Refer to the dimensions in the example above

3.11.3.7 Modify design units

Specify for all walls or selected walls that the entire wall is a single design unit 361 or that the moment design units are identical to the shear design units.

- select the wall sections:



* Select:

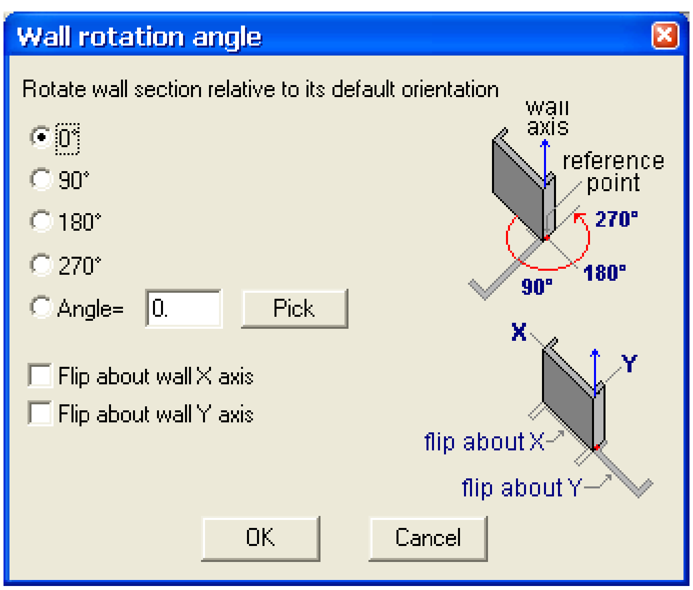
  For walls selected "Yes": The entire wall becomes a single moment design unit.

  For walls selected "Yes": The the moment design units become identical to the shear design units.

Note:
* shear design units are not modified by this option.

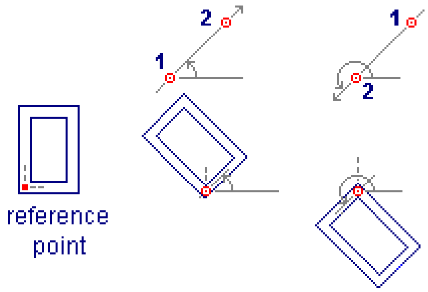
3.11.4 Rotate

The program attaches the wall section at its reference point to the line of nodes selected by the user according to the default orientation[371]. In many cases this orientation will be incorrect; use this option to rotate or flip the wall section to the correct orientation:

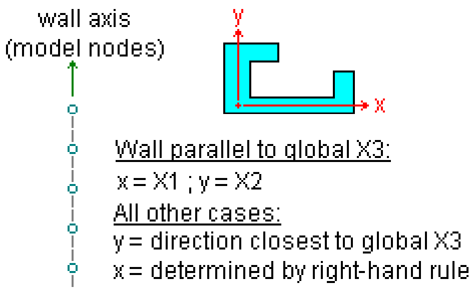


Note:
* the orientation is always relative to the default, not the current orientation
* a positive angle is counter-clockwise

- the wall may be rotated and flipped at the same time
- Select  to define the rotation angle by two nodes. The order of the node selection is important ! For example:



Default axes

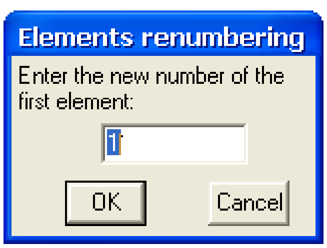


Note
- the positive direction of the wall axis always points in the positive direction of the nearest global axis.

3.11.5 Renumber

Select one or more walls using the standard Wall selection option.

Note that the order that the walls are selected is important; they will be renumbered in the order that they are selected.

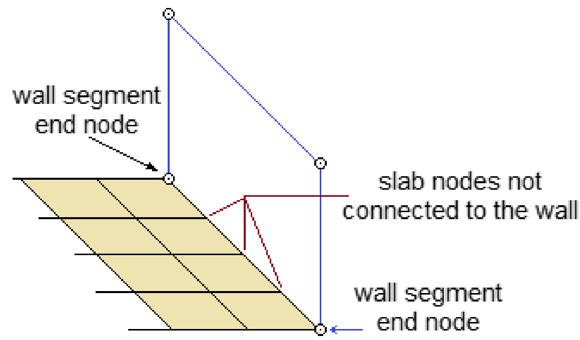


Type the new number of the first wall selected; all of the walls selected are renumbered sequentially. If the program discovers that a number has already been assigned to another wall the program assigns the original number of the selected wall to that wall.

Example:
- walls 41, 42 and 43 are selected (in that order).
- 75 is specified as the new number for 41
- the walls will be renumbered 75,76 and 77 respectively
- wall 76 is an existing wall; it will be renumbered 42.

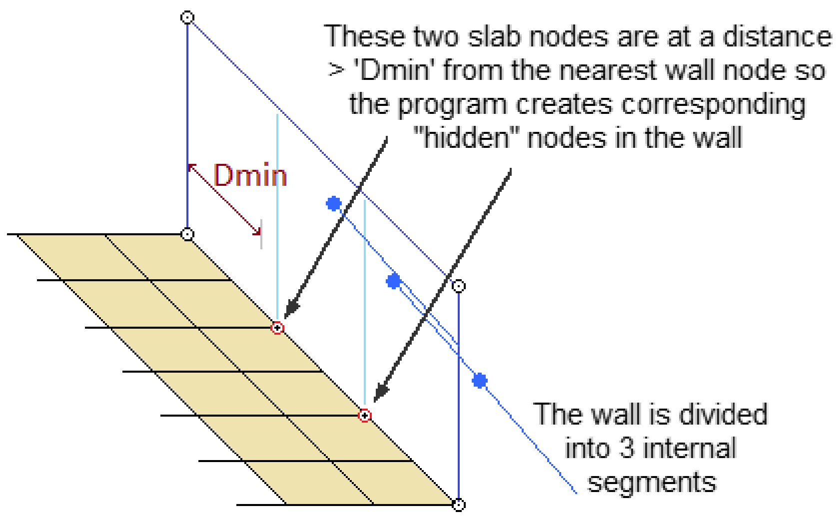
3.11.6 Link

The wall segments are connected to the adjacent parts of the model only at their end nodes. However there may be existing nodes in the model that lie along the wall segment boundary but are not connected to the wall. For example:



The wall may not move together with the slab along the boundary when either is loaded.There are two ways to deal with this problem:

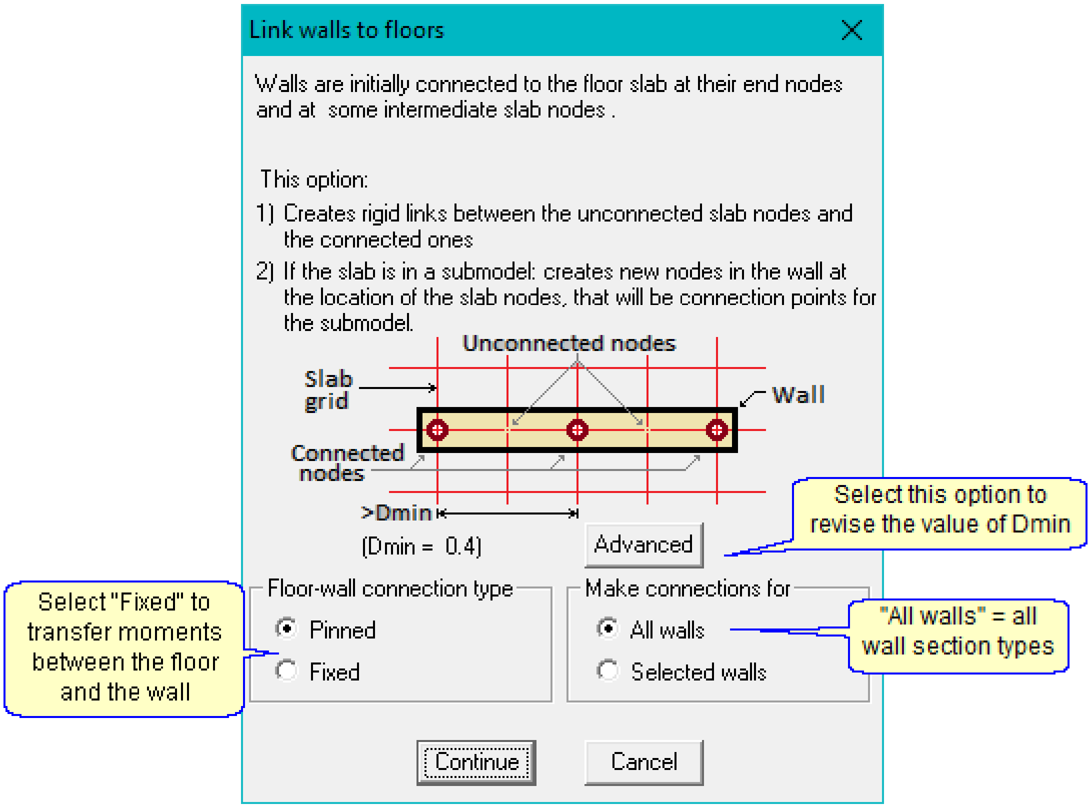
- create rigid links between the unconnected nodes and the wall end nodes. However, results become inaccurate in long wall segments with many rigid links.
- create 'hidden nodes' in the wall segment at the unattached model node locations, and then connect the wall to the model at those nodes. The program can automatically create these 'hidden nodes' so that no unattached slab node is further than "**Dmin**" from a wall node (a corner node or another "hidden node") .



- If the wall and the slab are both in the main model, by default the program ***automatically*** creates the "hidden nodes" but does not create rigid links to the remaining unattached nodes. The **Link** option must be selected in order to create the rigid links.
- If the wall is in the main model but the slab is in a submodel, the program does not automatically create "hidden nodes". The **Link** option must be selected to create ***both*** the "hidden nodes" and the rigid links. Note that you will be able to see these "hidden nodes" in the Main model.
- The default value of **Dmin** = 0.4 m; refer to the Advanced option below to change the value.

Note that the program cannot create these "hidden nodes" if the slab nodes are offset from the wall centre-line. In these cases, only rigid links can be created to all these unconnected nodes.

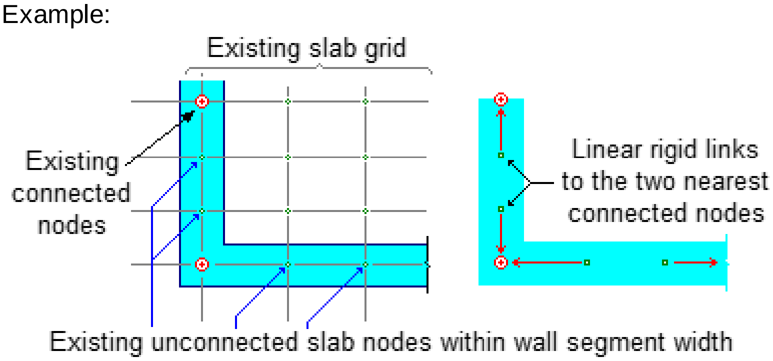
Select one of the following options:



Connection type (rigid links):

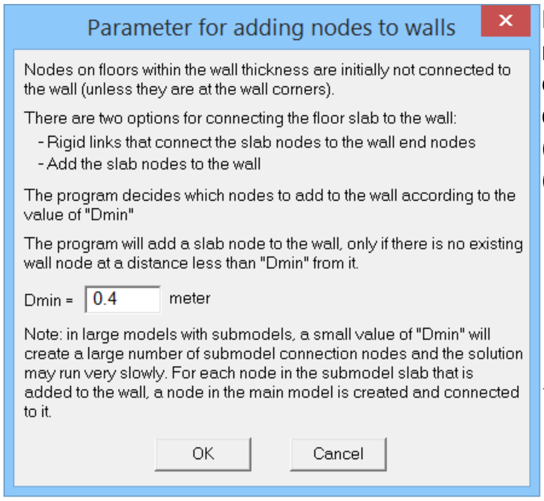
This option searches for unattached nodes and automatically creates linear rigid links to the two nearest attached wall nodes (corner or "hidden" nodes). The link may be either fixed (similar to "Rigid in a Plane " - the plane perpendicular to the wall) or pinned (does not transfer moments from the slab to the wall).

Example:

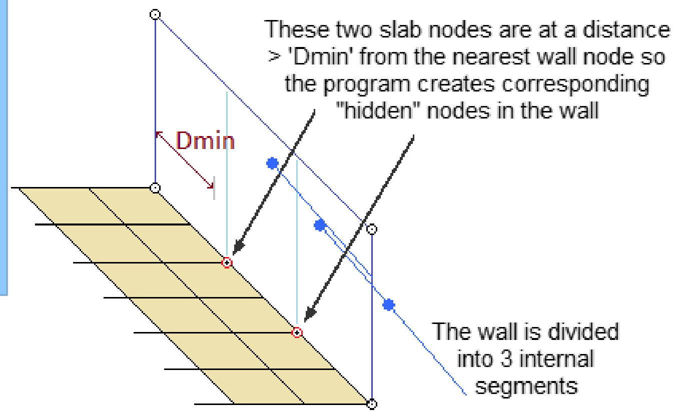


Note that the connections are "fixed" by default at the nodes that are connected to the wall. If the connection between the wall and slab is pinned, releases should be defined for all slab element edges connected to the wall edges.

Advanced options



Results become inaccurate in long wall segments with many rigid links connecting slab nodes to wall segment end nodes. To overcome this problem, the program can divide segments at existing nodes on the wall edges (usually from the attached slab) to create new segments (these segments are not seen by the user):

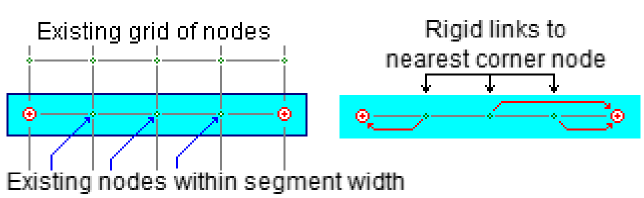


Note:

- If the value of **Dmin** is later revised so that the "hidden" wall nodes are at different locations, the program also automatically updates the rigid links and deletes unnecessary previous nodes from the Main model.
- This option will correctly create the nodes when the wall is in the Main model and the slab is in a submodel.
- The default value of **Dmin** was selected after testing to optimize accuracy vs solution time and changing the value is not recommended. Reducing the value of **Dmin** in large models with many walls will greatly increase the solution time.

Note:

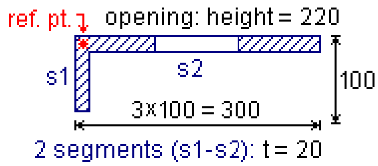
- To check for "hidden nodes": Select the "Data" option  and select one of the unconnected slab nodes. The data table lists the elements connected to the node. If "**Wn**" is displayed, the node is a "hidden node" connected to wall "**n**".
- the program automatically deletes any existing links ***created by this option*** when automatic links are requested again for the same wall.
- Results may be unsymmetric in symmetric models where there is an odd number of nodes in the slab because the rigid links will not be symmetric. For example, a geometrically symmetric wall and slab:



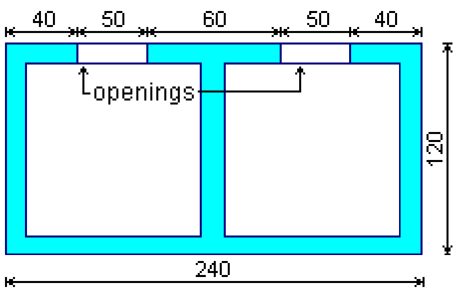
  The middle node is rigidly linked to either of the two end nodes, i.e. 2 nodes are linked to one end node and 1 node is linked to the other; the mode is now unsymmetric.
- In models generated by the "AutoSTRAP" program. *STRAP* knows which linear rigid links connecting the walls were created by AutoSTRAP. These rigid links are automatically revised when the wall is updated.

3.11.7 Examples

- Example 1 375

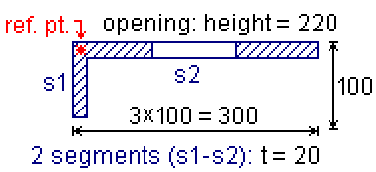


- Example 2 376



3.11.7.1 Example 1

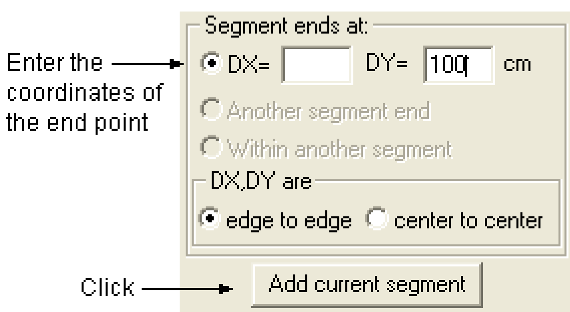
Define the following wall section:

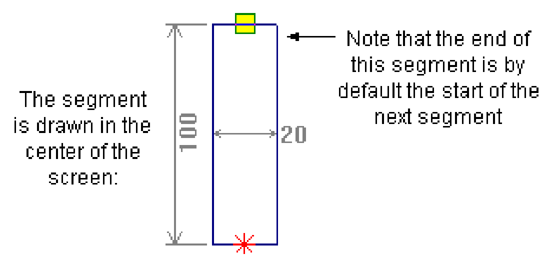


Note that all dimensions are ***exterior***.

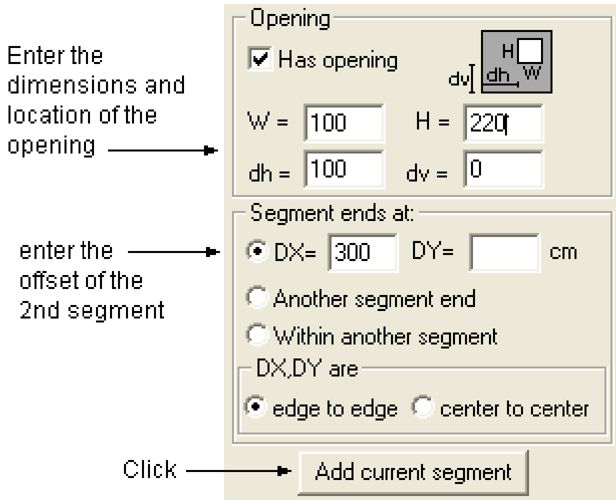
The program assumes that the start point of the first segment is at an arbitrary (0,0) coordinate and that it is also the reference point.

- set: **Thickness = 20 cm**
- define segment 's1':

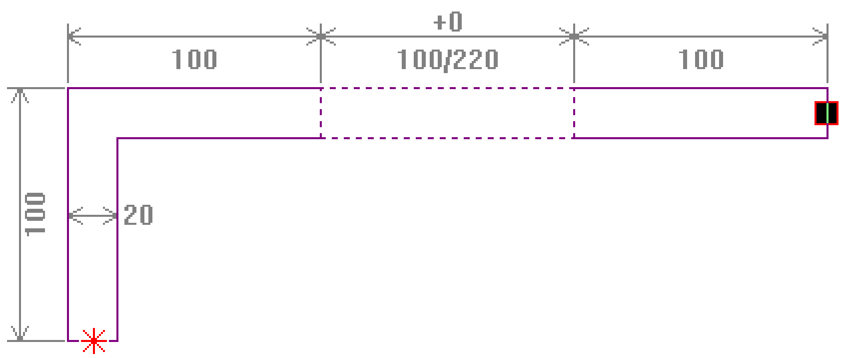




- Define segment 's2':



- The section is displayed as:



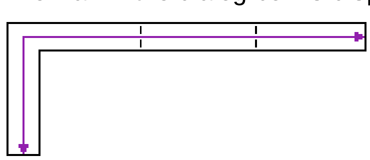
If any of the segments is incorrect, click Undo and redefine them

- Move the reference point to the corner of the 2 walls:

Click Reference corner , move the so that the corner is highlighted with the ■ and click the mouse.

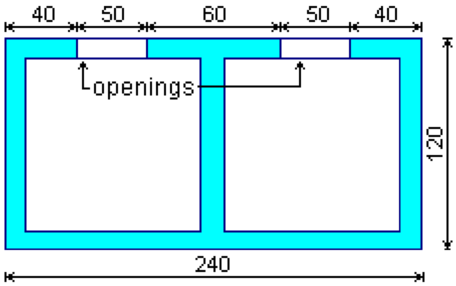
- combine both segments into one design unit for moments:
  - click Design units
  - click Combine several segments
  - highlight and click on both segments

The wall in the dialog box is displayed:



3.11.7.2 Example 2

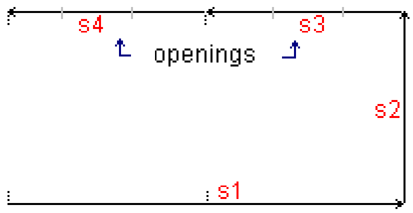
Define the following wall section:



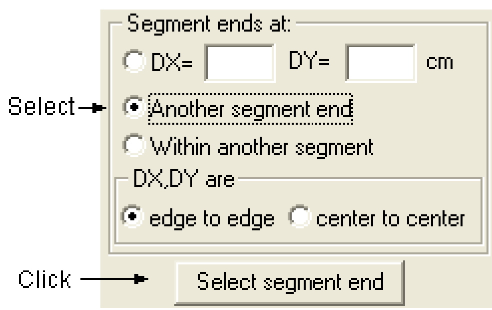
All segments are combined to form one moment design unit.

The top/bottom horizontal segment are combined to form two horizontal shear units.

- Define segments s1 to s4 as explained in Example 1375.

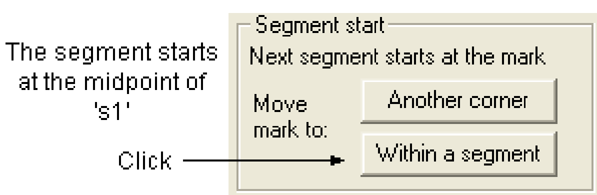


- Close the section:



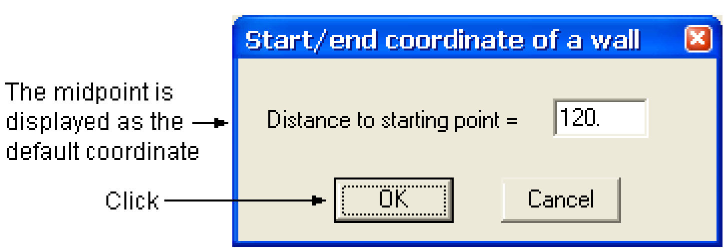
move the so that the initial corner is highlighted with the ■ and click the mouse.

- Add the dividing wall:

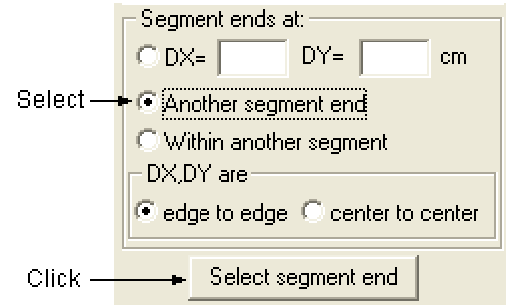


move the so that segment 's1' is highlighted with the ■ and click the mouse.

- specify the location of the dividing wall on 's1':

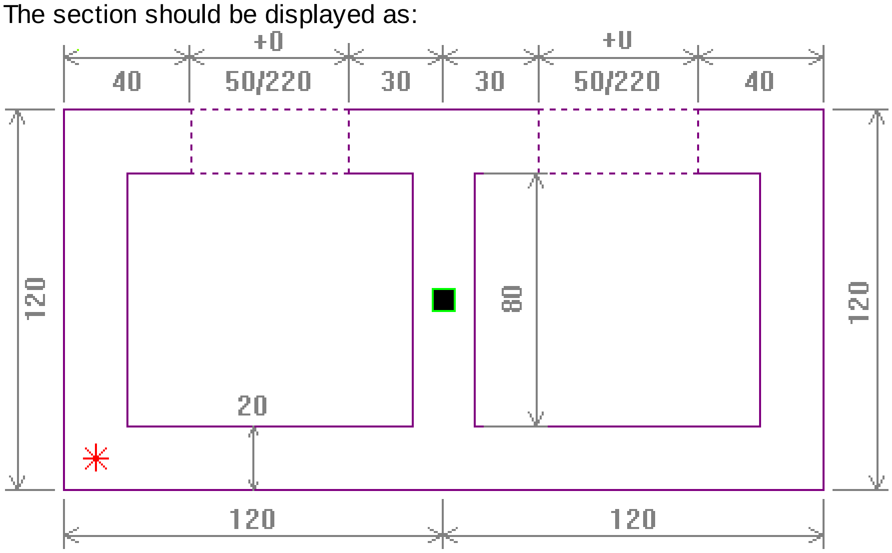


- Close the dividing wall:



move the cursor so that segment 's3' is highlighted with the ■ and click the mouse.

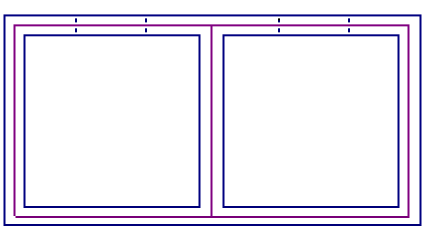
The section should be displayed as:



If any of the segments are incorrect, click Undo and redefine them.

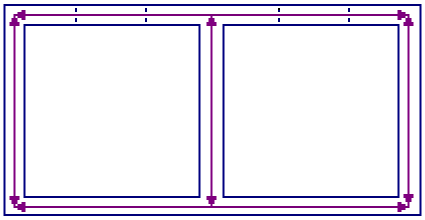
- create the moment and shear design units:
  - click Design units
  - select **Define moment design units**
  - click Combine - select window
  - draw a window around the entire wall and click the mouse

  The wall in the dialog box is displayed:



  - click Design units
  - select **Define shear design units**
  - click Combine ALLcolinear seg.

The wall in the dialog box is displayed:

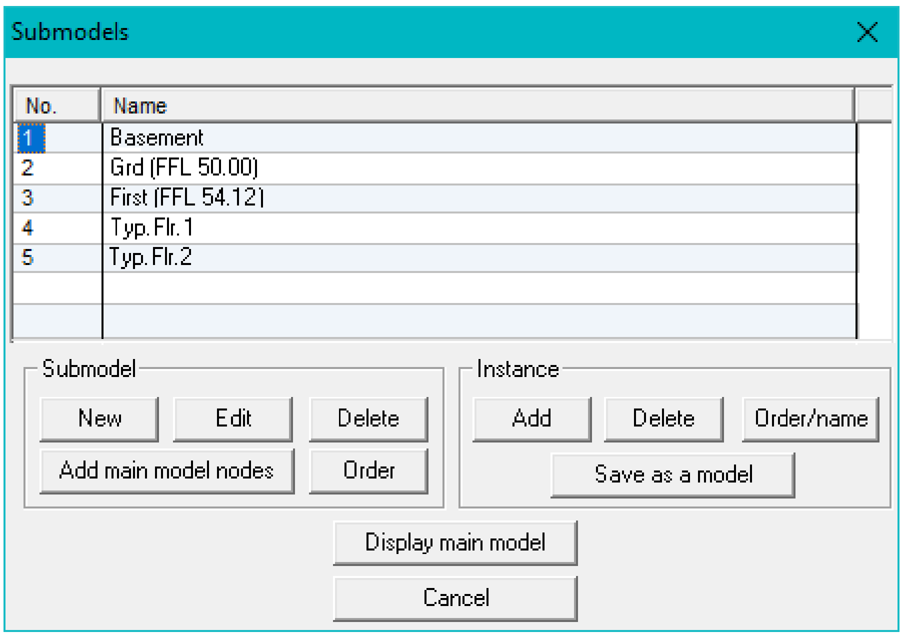


3.12 Submodel

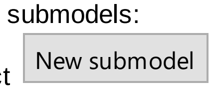
A submodel is a part of the final model defined in a separate working area.

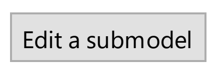
- all the geometry options are available when defining a submodel.
- a submodel may be added to the current model any number of times (referred to as '***instances***') at any location and at any angle. Two options are available:
  - the submodel is added as a ***submodel,*** i.e. the complete model at all stages - geometry, loads, results, etc. - consists of the 'Main model' and one or more 'submodels'. Note that ***each*** submodel may be defined with the maximum number of nodes and/or elements (32,000), i.e. the total size of the model (main model + submodels) is unlimited. (see note 385).
  - as ***individual elements***, i.e. the submodel is merged into the main model when it is added.

Refer to submodels - general 383.

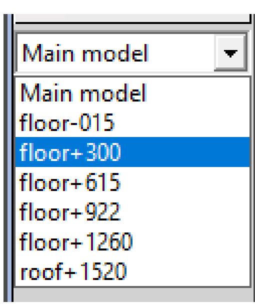


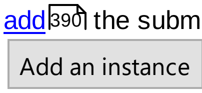
There are three steps when using submodels:

- create 385 the submodel - select 

- edit a submodel - click and highlight a submodel in the list and click 

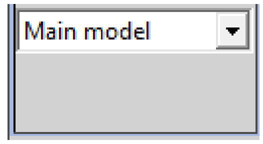
.
Alternatively, select the submodel in the small list box in the side menu:



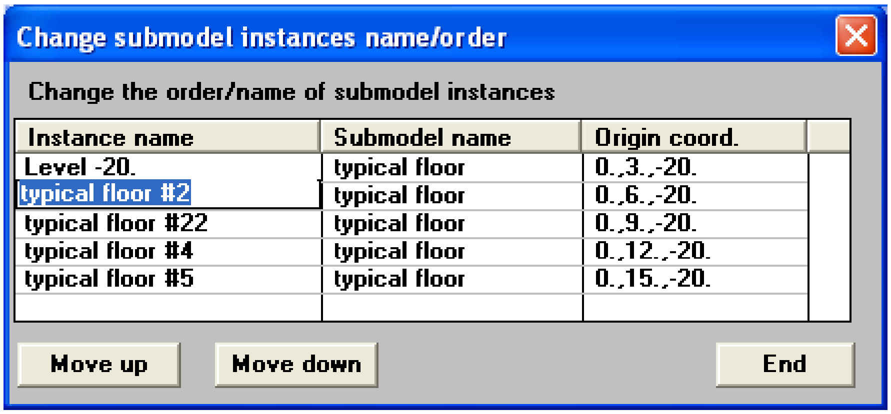
- add 390 the submodel to the main model - click and highlight a submodel in the list and select 

Display main model

Return the main model to the display. Alternatively, select 'main model' in the small listbox in the side menu:



Instances order/name



To change an instance name:

- click on the appropriate name in the left-hand column (as shown above)
- type in a new name

To rearrange the list:

- click and highlight the appropriate line
- click Move up or Move down

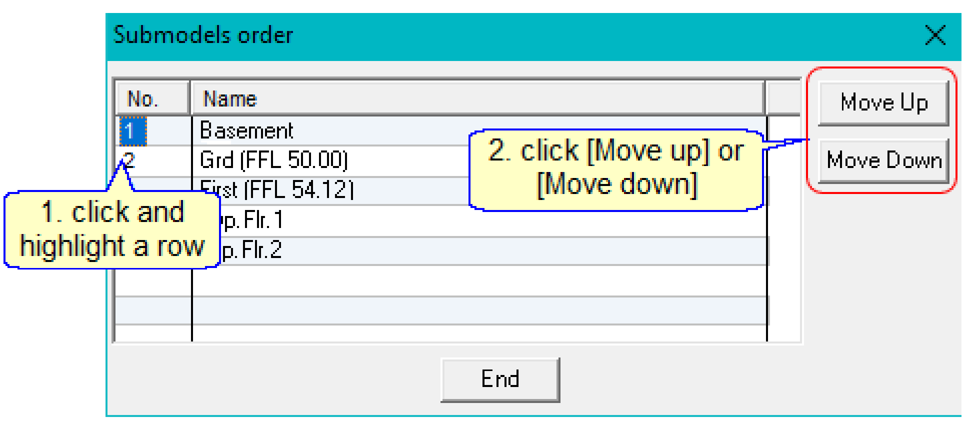
Delete an instance

Delete selected instances of a submodel from the main model; the submodel itself is not deleted.

- click and highlight one or more lines in the listbox
- click Delete

Order

Rearrange the order of the submodels in the list:



Delete a submodel

Delete a submodel and all of its instances. Several submodels may be deleted at the same time.

Save as a model

Create a new model from one of the submodels in the current model:

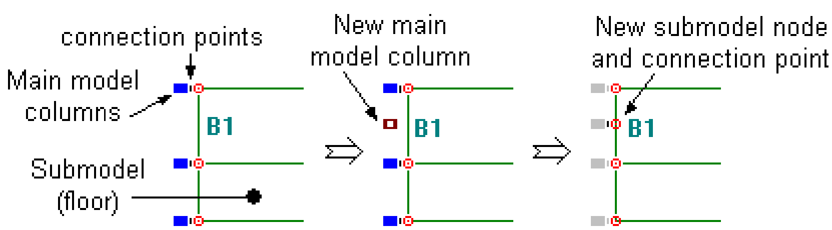
- click Save as a model
- select a submodel (and an instance, if there are more than one)
- enter a name for the new model. Note that the loads are copied only if the ☑ **Copy loads** option is selected. Only loads applied to the selected instance are copied.

Add main model nodes

Nodes added to the Main model after a submodel was defined can be added automatically to the submodel:

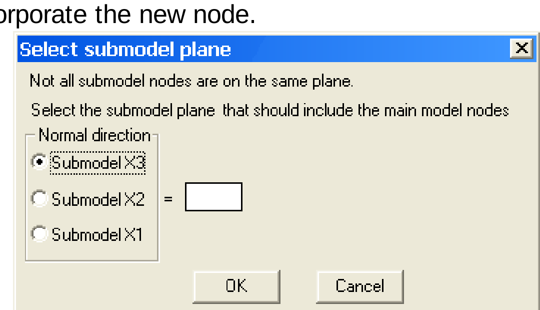
- the program adds the node only and does not add or split beams, etc, in the submodel.
- the new nodes are automatically designated as Connection points.

For example:



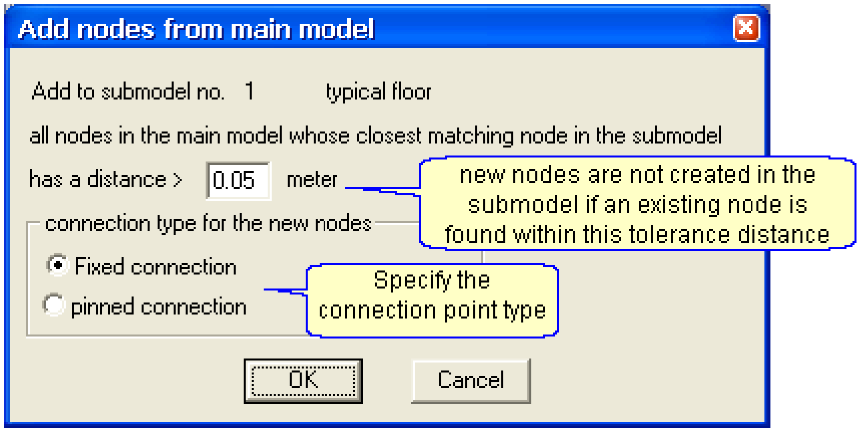
Note:

- the program does not split B1, i.e. the new node is not connected to the submodel (although it is a "Connection point"); edit the submodel geometry to incorporate the new node.
- the program searches for Main model nodes that are at the same level as the submodel. If the submodel contains more than one level the program asks which axis and coordinate to use:

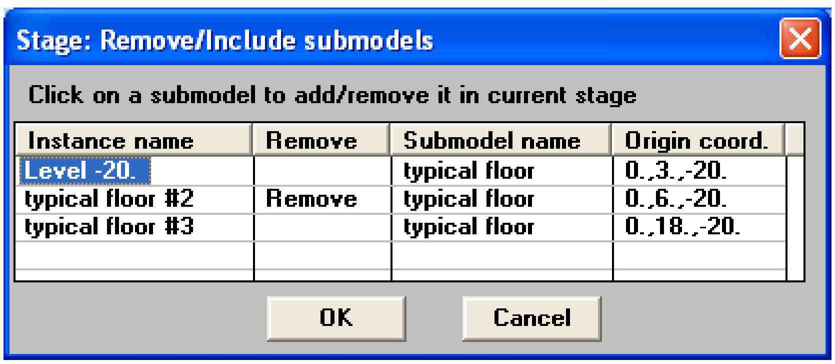


To add the nodes:

- click and highlight one of the submodels in the list
- click Add main model nodes
- specify the parameters:



If the option is selected when a Stage other than "Whole model" is current, an ***entire*** submodel instance may be deleted from the Stage; individual beams,elements, etc. may not be removed from the stage.



Click on a line to toggle the status in the **Remove** column.

3.12.1 General

What are submodels:

A submodel is a part of the final model defined in a separate working area.

A submodel may be added to the current model any number of times (referred to as '***instances***') at any location and at any angle. Two options are available:

- the submodel is added as a ***submodel,*** i.e. the complete model at all stages - geometry, loads, results, etc. - consists of the 'Main model' and one or more 'submodels'. The submodel is retained by the program and may be revised at a later time.
- as ***individual elements***, i.e. the submodel is merged into the main model when it is added. The submodel is not retained by the program.

When to use submodels:

Submodels are recommended when a repetitive element is present in the structure, e.g. a typical floor in a high-rise building.

- each typical floor type is defined as a submodel
- every occurrence of a typical floor type is added to the model as an instance

Advantages:

If the submodels are added as ***submodels***:

- the main model and ***each submodel*** may be defined with the maximum number of nodes and/or elements (32,000); multiple instances of each submodel may be added to the main model, i.e. the total size of the model (main model + submodel instances) is ***unlimited***. (see note 385)

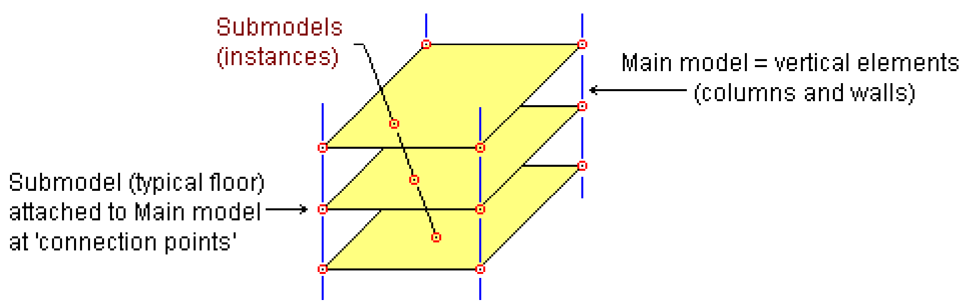
- the ***solution time*** for a model defined with submodels is ***significantly reduced***, i.e. slabs may be defined with more refined meshes without a significant increase in the solution time.
- revising the geometry of a submodel (e.g. a typical floor) automatically revises the geometry in all of the instances.
- Loads may be defined either on the submodel (applied to all instances) or on a selected instance of the submodel. If applied to the submodel, any change to the loads is applied to all of the instances.
- the node and element numbering is independent in each submodel, i.e. all numbering starts from ***1*** in each submodel; large node/element numbers are avoided.
- Each submodel has its own property group list (which may be identical to the Main model property list). Refer to Submodel - new 385.
- Each submodel (or instance) can be displayed separately without defining a View.

Note that the submodel is connected to the main model at connection points 387 only. These points are defined in the submodel similar to restraints, i.e. any degree-of-freedom may be restrained or free.

How to use submodels:

The recommended procedure for a high-rise building is as follows:

- Define all columns, walls and restraints in the main model.
- Define the typical floor at one level in the main model (this ensures correct dimensions in the submodel)
- Create the submodel: select New submodel and the **Use part of a model** option and select all nodes in the typical floor. The program automatically identifies the common submodel/main model nodes and automatically creates the connection points in the submodel.
- Modify the connection point type, if necessary (they are completely fixed by default)
- Add the typical floor submodel to the main model at each level; each level is represented by an ***Instance*** of the same submodel.



- repeat for other typical floor types.

To display a demo video that explains how to use submodels:

- *click on to start the video*
- *then click on to enlarge the display.*

Restrictions:

- all the geometry options are available when defining the submodel, except the following:
  - supports: all supports must be defined in the main model
  - tension/compression-only beams
  - individual submodel beams/elements cannot be removed in a Stage; however entire instances may

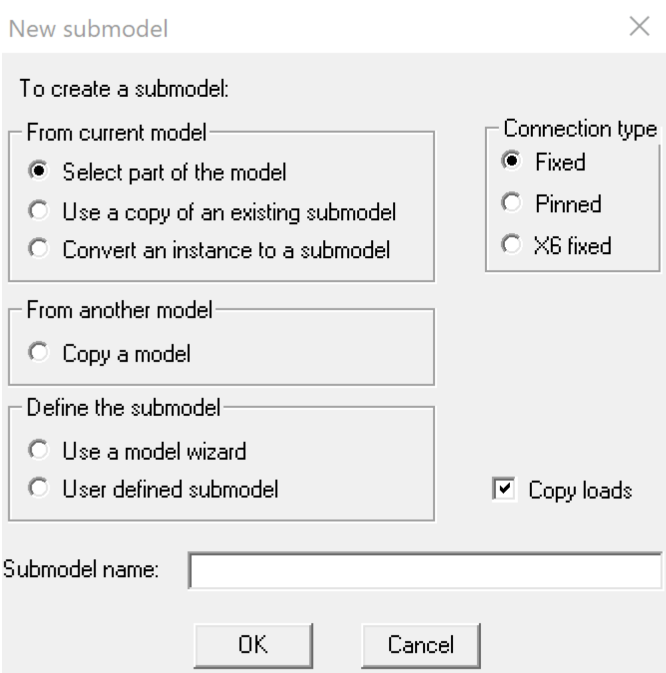
be removed from or added to a stage.
- the concrete design module cannot identify columns and walls in submodels (they must be defined in the Main model)
- The following modules cannot handle models with submodels in the current version: *BRIDGE*, *CONNECT*, *POSTTEN*, Dynamic time-history analysis.

Display:
- either the Main model or any of the submodels may be displayed on the screen:
  - revisions to submodel geometry or loads applied to submodels may be made only when the submodel is displayed
  - submodel results may also be viewed when the main model is displayed
- geometry tables list only submodel data if a submodel is currently displayed.
- 'right-click' ***Model properties*** option:
  - Main model is displayed: total weight and center-of gravity are shown for the entire model (including all submodels)
  - Submodel is displayed: total weight and center-of gravity are shown only for the current submodel.
- Node/beam/element number display: refer to Display node/beam/element numbers.

Capacity:
- the 'unlimited' capacity is available only in the 32,000 element/node version of *STRAP*.
- in smaller capacity versions (e.g 1000 nodes), submodels may be defined but the total number of nodes or elements in the ***entire*** model (main model + submodel instances) may not exceed the purchased program capacity.

3.12.2 New submodel

There are several methods for creating the submodel:



For all options:
- Enter the **Submodel name**

- select the option to create the submodel and click OK
- the program displays the submodel on the screen; edit using the standard geometry options.
- Attach the submodel to the Main model with the Add an instance option.

Note:
- each submodel has its own property list; note the way it is created according to the option selected.
- the **Copy loads** option works differently for each option. Note that certain load types will not be copied.

From the current model

Select part of the model

Select part of the model using the standard Node Selection option;
- only elements with ***all of their end nodes*** selected are included in the submodel.
- nodes that have an ***unselected*** beam/element/wall attached to them are automatically designated as connection points.
- the program automatically ***turns the selected part of the model into a submodel*** instance (even though it was defined as part of the main model).
- the entire Main model property list is copied to the property list for the new submodel.
- Under **Connection type** select the connection point type to be assigned for the new submodel.

Fixed -	All degrees of freedom (translation and rotation) are restraint.
Pinned -	Only translation degrees of freedom (X1,X2,X3) are restraint.
X6 fixed -	Translation degrees of freedom are restraint plus X6 rotation (rotation around X3).

- the **Copy loads** option is not available. Note that certain load types will not be copied.

Use a copy of an existing submodel

Select a submodel from the list.
- the property list from the existing submodel is copied to the property list for the new submodel.
- Set **Copy loads** to copy the loads defined in the other submodel. Note that certain load types will not be copied.

Convert an instance to a submodel

Select an instance of any submodel from the list. The program converts it to the first instance of a new submodel. No changes to the geometry are made and the instance remains in the same location it was previously attached.

From another model

Browse to the correct folder; click and highlight the model name in the list and click OK.
- if the model also contains submodels, you must select either the main model or one of the submodels.
- the property list from the selected main model/submodel is copied to the property list for the new submodel
- Set **Copy loads** to copy the loads defined in the other model. Note that certain load types will not be copied.

Define the submodel

Use a model wizard

Create the submodel from one of the models in the Wizard library.

- the property list defined in the wizard is copied to the property list for the new submodel.
- Set **Define loads** to define loads in the wizard.
- select the **Working plane** for the wizard model - **X1-X2**, **X1-X3** or **X2-X3**. The Add option adds submodels to the Main model ***on the same plane*** by default (The Add + rotate option.does not rotate loads).

User-defined model

The program displays a empty working area; define the submodel using all of the standard geometry options.

- the entire Main model property list is copied to the property list for the new submodel.

Note:

- submodels should be defined in the same planes that they will appear in the Main model (to avoid rotating them when adding them to the model).
- the submodels are attached to the model at connection points 387. These points are the nodes in the submodel that are attached to corresponding nodes in the Main model. The connection may be defined as active or released in any degree-of-freedom (similar to Restraints). Click Connections in the side menu when editing the submodel.

3.12.3 Connection points

The submodel is connected to the main model at common nodes called ***Connection points***. Use this option to define the location and types of connection. The connection points may be either selected by the user or automatically selected by the program.

Fixed Pinned Other	The option is similar to the Restraints option: • specify the connection type (all directions / pinned / other) • select the nodes using the standard Node selection option. Refer to User-defined 388 connection points.
Automatic	The program automatically creates the connection points (***only if the submodel has already been placed in the main model).*** Refer to Automatic 388 connection points.
Delete	Select the nodes in the submodel using the standard Node Selection option.
Rigid links	Rigid links may also be defined in this option: • the rigid links are defined between nodes in the submodel • if two connection points are connected by a rigid link, the program automatically creates the identical rigid link between the corresponding nodes in the Main model when submodel is attached.

Note:

- If the submodel is defined using the **Select part of the model** option, nodes that have an ***unselected*** beam/element/wall attached to them (e.g. columns perpendicular to a selected floor plane) are automatically designated as connection points.
- using submodels with a ***pinned*** connection point is a simple way to connect elements to columns without transferring moments (see note below).
- each connection point in the submodel must be aligned with an existing node in the main model. The program checks that this is true for each instance. In cases where a connection point is not aligned with a node the program creates a rigid link between the connection point and the nearest node in the

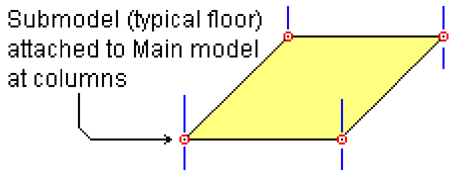
main model.

3.12.3.1 User defined

The option is similar to the Restraints option:

- specify the connection type (all directions / pinned / other)
- select the nodes using the standard Node selection option.

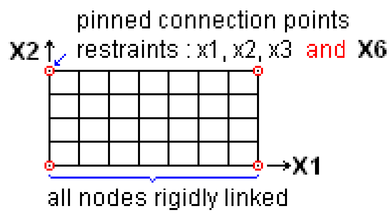
Example:



- select Pinned, Fixed or Other (and select the restrained degrees-of-freedom)
- select the nodes at the locations corresponding to the columns in the Main model using the standard Node selection option.

Note:

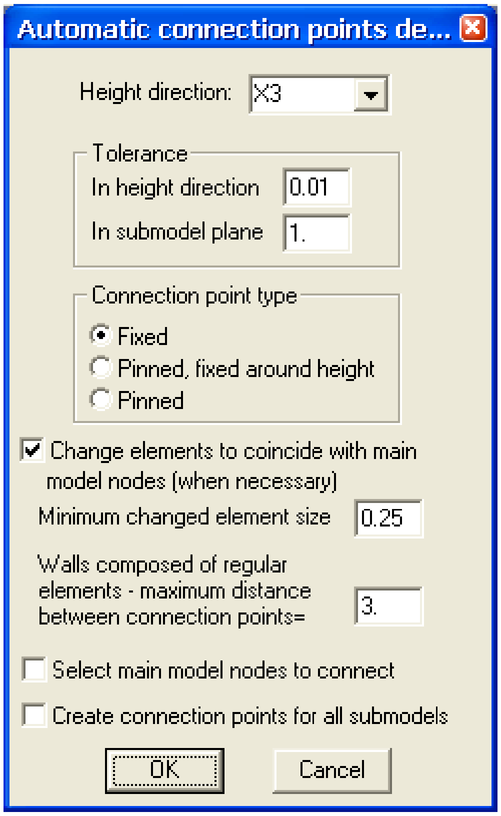
- if the typical floor submodel is rigidly linked ***and*** the connection points are pinned, the program automatically adds restraints at the connection points for rotation about the axis perpendicular to the rigidly linked plane (when the geometry is saved):



3.12.3.2 Automatic

The program identifies main model nodes at the same level as the submodel, then searches for the submodel node closest to the main model node and defines it as a connection point.

The connection points are created ***only if the submodel has already been placed in the main model***.



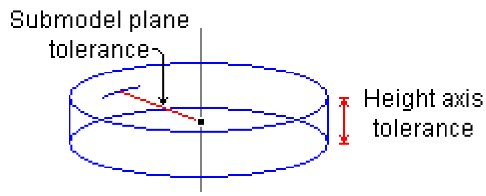
Specify the following options:

Height direction

The program defines the connection points only for submodels perpendicular to this axis.

Tolerance

Specify the tolerance in the height direction and the perpendicular plane; the program searches all main model nodes and looks for submodel nodes that are within the tolerances limits:



The submodel node closest to the main model node and within the tolerance limits is then defined as a connection point.

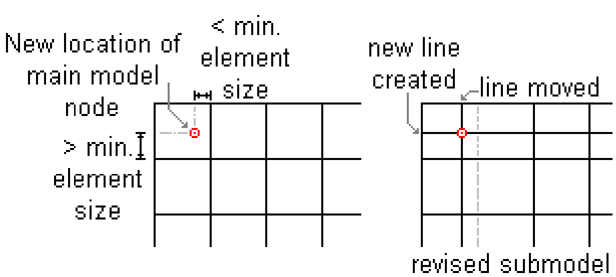
Connection point type

Specify the connection point type (for all selected nodes). Select:

- fixed - all degrees-of-freedom
- pinned & rotational degree-of-freedom about the height axis.
- pinned only.

Change elements to coincide
Minimum changed element size

The program can modify the submodel geometry, if requested, when a main model node attached to a connection point lies ***within the submodel***. The program will either create new elements or move the node lines, depending on the value of the "minimum changed element size". For example:



This option can also be used to recreate the connection points after main model nodes have been moved.

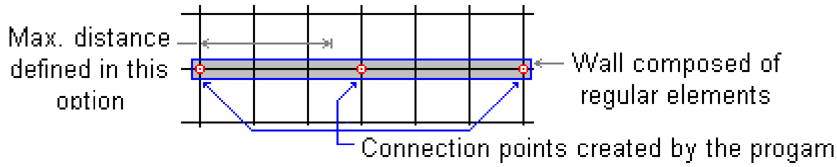
For nodes that are moved outside the submodel:

- if the distance is relatively small, the program connects the node to the submodel connection point with rigid links
- otherwise the connection point is deleted.

Wall composes of ... maximum distance

For walls in the main model composed of ***regular elements*** (not walls created using the "Wall" option):

Connection points do not have to be created at every node along the wall. Specify the maximum distance between connection points. For example:



The program connects all other nodes along the wall to these connection points with Linear rigid links.

Select main model nodes to connect

☑ Select ***main model*** nodes using the standard Node selection option:

- the program will create (or recreate) the connection points in the submodel (as described above) only for the selected main model nodes.
- existing connection points to other main model nodes are not deleted.

Create connection points for all submodels

☑ Connection points are defined automatically for ***all*** submodels in the model
☐ Connection points are created only for the current submodel

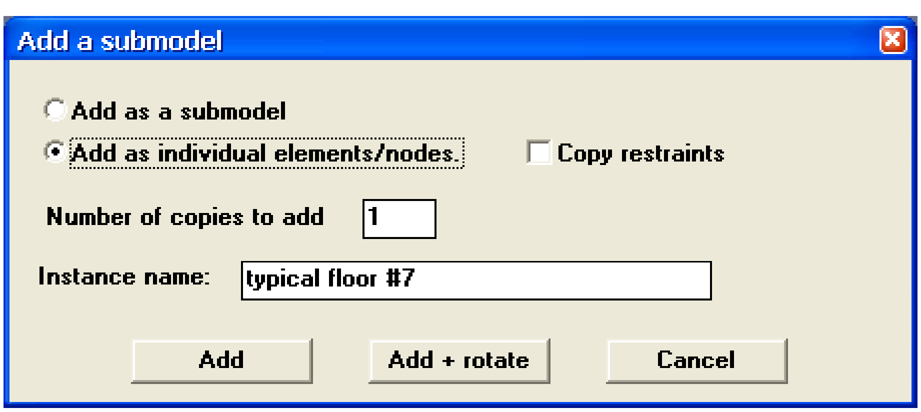
3.12.4 Add to main model

The submodel is added to the main model using a method similar to the geometry Copy option:

- one or three reference nodes are defined in the submodel and the corresponding nodes are selected in the main model. The program then attaches the submodel to the main model at the first reference node; the model may be rotated if three reference nodes are defined.
- the submodel may either be added as a ***submodel*** (i.e. the complete model at all stages - geometry, loads, results, etc. - consists of the 'Main model' and one or more 'submodels') or as ***individual elements*** (i.e. the submodel is merged into the main model when it is added). Refer to Submodels - general 383 for more information.

Note:

- this option copies the ***entire*** submodel, including supports (if added as submodel), offsets, rigid links, springs, etc.
- load cases may be copied if the submodel was created from an "existing model" and is added as "individual elements"



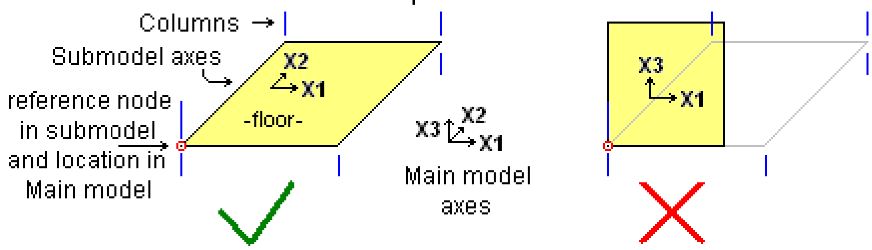
Any number of instances may be added at the same time, each at a different location:

- specify **Number of copies to add**
- define the **Instance name** if you select ◉ **Add as a submodel**

Add as a submodel

- Add

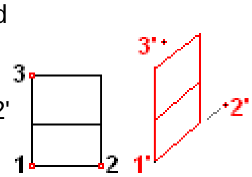
  Select a single reference node in the submodel and the reference node location in the main model; the submodel is added to the main model at the location of the reference node.
  ✍the submodel is neither stretched nor squeezed
  ✍the submodel is added in the same plane in the main model as it was defined in the submodel, e.g.



  ✍the program prompts for the reference node location in the main model for ***each*** of the **Number of copies to add**

- Add + rotate

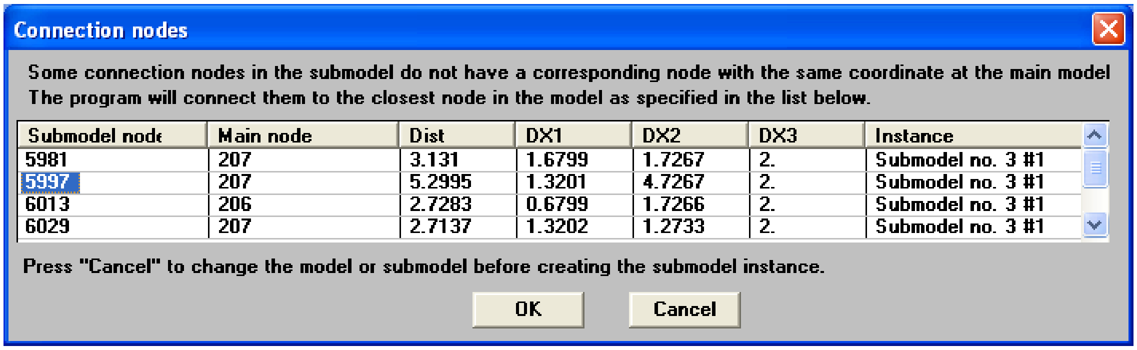
  Select three reference nodes in the submodel and the corresponding locations of the reference nodes location in the main model; the submodel is added to the main model at the location of the reference node.
  ✍the submodel is neither stretched nor squeezed
  ✍Node 1 in the submodel is placed at node 1' in the main model. Nodes 2' and 3' are used only for determining direction; 1-2 is aligned with 1'-2' and the entire submodel is placed in the plane formed by 1'-2'-3'.



  ✍the program prompts for the reference node location in the main model for ***each*** of the **Number of copies to add**
  ✍For "Number of copies" > 1 : the rotation angle is calculated for the first copy and the same rotation angle is used for all other copies.

Note:

- each connection point in the submodel must be aligned with an existing node in the main model. The program checks that this is true for each instance. In cases where a connection point is not aligned with a node the program creates a rigid link between the connection point and the nearest node in the main model and displays a list. For example:



Add as individual elements/nodes

- Add

Similar to "Add as a submodel" above.

- Add + rotate

The copy is generated by defining ***three*** reference nodes:

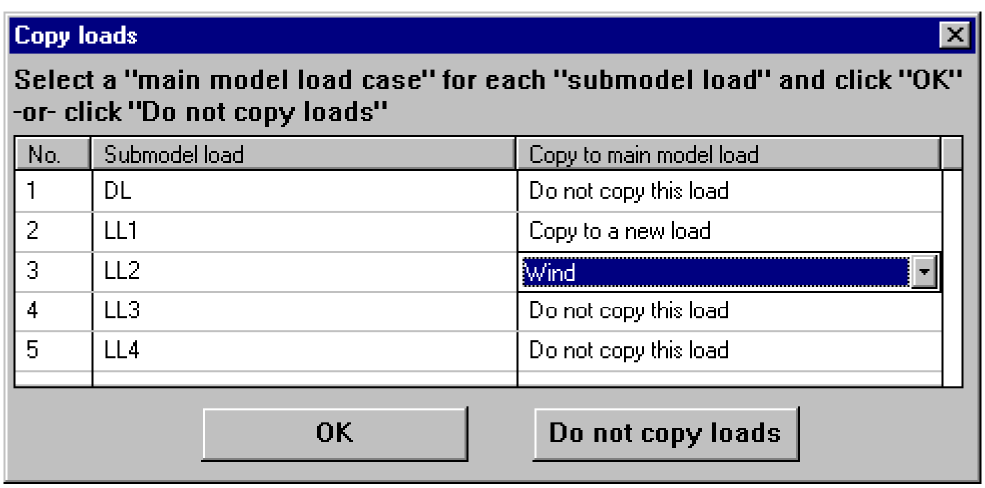
✍ as the three nodes can be located anywhere in space, this option allows the copied block to be rotated with respect to the original.

✍ if the distances from the first reference node to the second or from the second reference node to the third are not the same in the copied block as in the original block, the program will 'stretch' or 'squeeze' all of the elements in the block accordingly; ***the program will not distort the block***, i.e. all parallel beams will be stretched or squeezed ***by the same factor.***

Note:

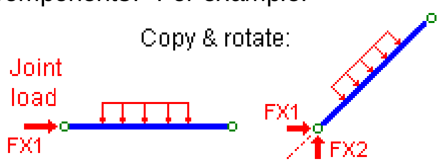
- restraints are copied to the main model only if ☑ **Copy restraints** is selected.
- the program does not create a new node at the location of an existing node; the restraints at the existing node are the sum of the previous restraints in the existing node and the restraints at the submodel connection node (if copied).
- the program will not create a new beam/element at the identical location of an existing beam/element.
- restraints or springs defined parallel to global axes in the submodel will be oriented in the closest possible global direction in the main model.

For a submodel created from an "existing model":



For each load case in the submodel, click on the right column, open up the list box, and select:
- the name of an existing load case in the main model - the loads will be added to the load case.
- "Copy to a new load" - to create a new load case in the main model.

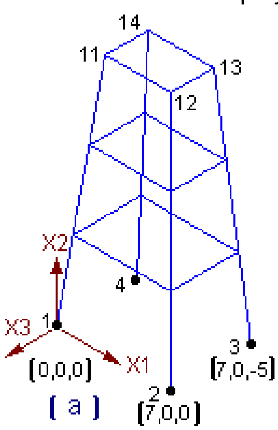
Note:
- Joint loads, beam loads, element loads and global loads only may be copied.
- Support displacements are not copied.
- Loads that were added to the load case using the "**Combine**" command cannot be copied.
- Element loads generated using the "Linear" element load option are not copied.
- Joint loads rotated to a direction not parallel to a global axis are separated to the equivalent global components. For example:

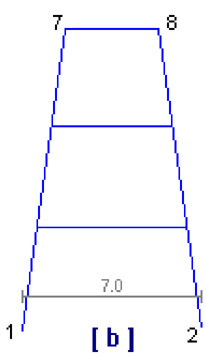


- Global loads defined in a global direction are applied in the closest global direction after rotation. Global loads applied perpendicular to the plane will remain perpendicular to the plane.

The option is best illustrated by an example:

Create the model displayed in Figure (a) from the submodel displayed in Figure (b).





Create plane 1-2-11-12 from the submodel:
- define the three reference nodes in the submodel:
  first reference node: 1
  second reference node: 2
  third reference node: 8
- define their new locations in the main model:
  first reference node: 1
  second reference node: 2
  third reference node: 12

Create planes 2-3-13-12 from the submodel:
- define the three reference nodes in the submodel as above:
- define their new locations in the main model:
  first reference node: 2
  second reference node: 3
  third reference node: 13

Note:
- the distance from the first to the second reference node is changed from 7.0 in the original block to 5.0 in the copied block; all of the dimensions in this direction are revised proportionally.
- As the perpendicular distance from the third reference node to the line joining the first two nodes remains unchanged in the copied block, all vertical dimensions will remain constant.
- The program should create a node at the new location of node 2. However it also checks if two nodes will be at the same location. Upon discovering that the new node is at the same location as node 2, it connects all elements to the existing node. Similarly new beams are not created on the line 2-12.

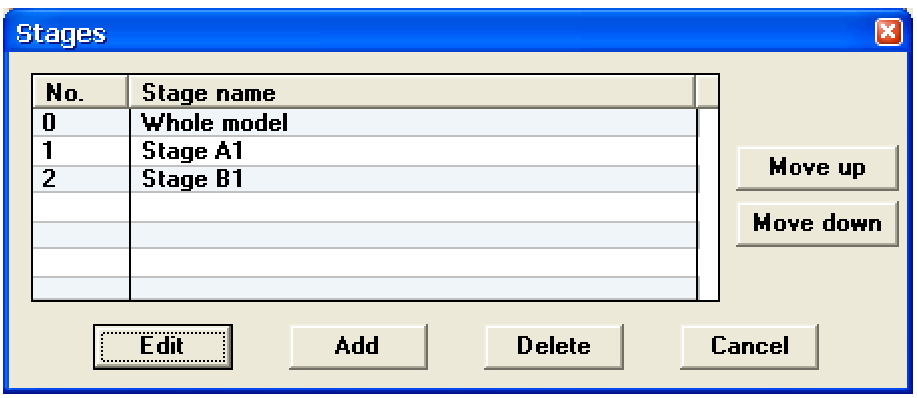
3.13 Stages

Define construction 'stages' for the current model:

- the default stage is the **Whole model**; it represents the final structure and contains ***all*** elements and the properties and supports/springs present at the end of construction.
- all other stages represent intermediate steps during construction; beams and elements may be removed (but ***cannot be created in a stage***), different properties and supports may be assigned, and specified load cases may be applied. Submodels may be removed only in their entirety (individual beams, elements, etc. cannot be removed from a submodel in a Stage).

The program treats each stage as a separate model and calculates the results based on the stage geometry and the applied load cases.

The stages are defined and activated in this option; elements, properties, restraints, etc are applied/ removed from the stages in the various Geometry and Load options.



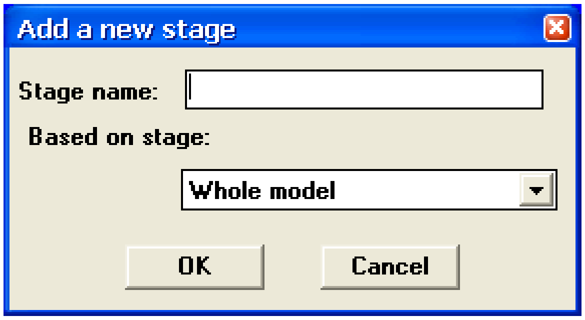
Stages - edit

Select one of the stages as the 'current stage'. Click and highlight the relevant line and click Edit; the program displays the current stage at the top of the screen:

and only the relevant options will be active, e.g. in Beams, only **Remove**, **Restore** and **Property** options will be available.

Stages - add

Add a new stage:



- Enter the stage name
- A new stage may be 'based' on any other existing stage, i.e. it is created with the same geometry as the selected stage.

Note:
- if the 'base stage' is not the **Whole model** and is later revised, the other stages based on it are ***not*** revised.
- if the **Whole model** is revised (e.g. elements are added), the revisions are made to all of the stages.

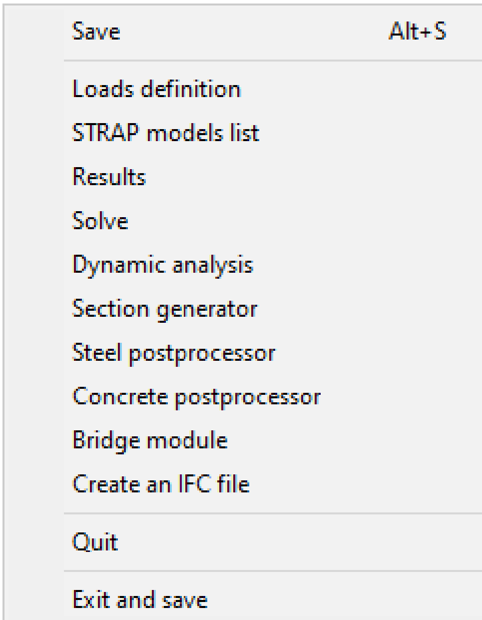
Stages - delete

Click and highlight the relevant line and click Delete.

Stages - move

Click Move up or Move down to rearrange the list.

3.14 File options



Save

Save current geometry and continue. Note that the geometry may be saved automatically at timed intervals. Refer to Setup - miscellaneous.

Loads

Save geometry and continue to load definition.

STRAP models list

Save geometry and return to *STRAP* main menu (list of models).

Results

Save the geometry and go to the Results display.

Solve

Solve the current model and proceed to results.

Dynamic analysis

Save geometry and continue to dynamic analysis

Section generator

Create a beam section using the STRAP section generator (*CROSEC*). The section can be copied to a beam property group as a section defined by properties (A,I,J, etc).

Steel postprocessor

Save the geometry and go to the Structural steel design module.

Concrete postprocessor

Save the geometry and go to the Concrete Design module.

Bridge module

Save the geometry and go to the Bridge Analysis module.

Create an IFC File

Output the geometry to an "IFC" ("Industry Foundation Class") format file:

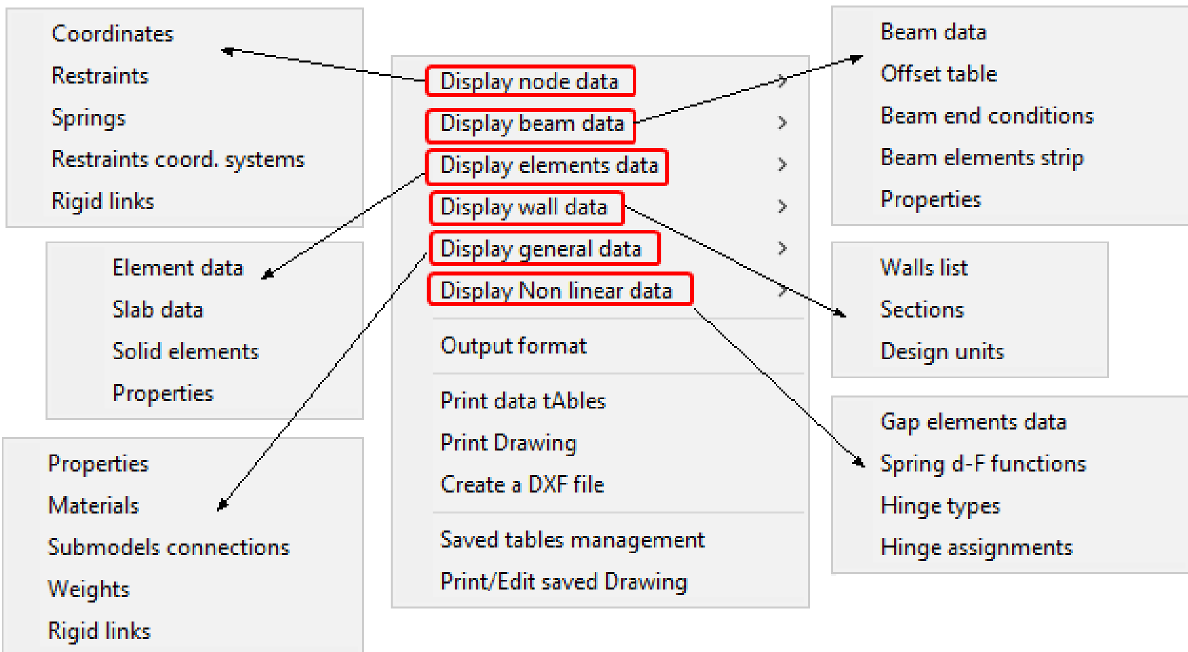
Quit

Exit the program ***without*** saving changes to the geometry. Note that only changes made after the last "Save" or the start of the session are discarded.

Exit and save

Save geometry and leave *STRAP.*

3.15 Output options



3.15.1 Nodes

3.15.1.1 Coordinates

NODAL COORDINATE TABLE (units - meter)

Exit Goto Print Copy

NODE	X1	X2	X3
1	-17.602	23.562	3.490
2	34.137	-24.101	0.000
3	-17.594	24.330	3.490
4	-8.019	24.285	3.490
109	31.864	-24.101	0.000
110	31.407	-24.101	0.000

where:
X1, X2, X3 = coordinates in the global system

3.15.1.2 Restraints

NODAL RESTRAINED DOF TABLE

Exit Goto Print Copy

NODE	X1	X2	X3	X4	X5	X6
3739	0	0	0	0	0	1
3747	0	0	0	0	0	1
4510	0	0	0	0	0	1
9782	0	1	0	0	0	0
11324	1	0	0	0	0	0

where:
0 = free
1 = restrained.

Relevant restraints:
- plane frame : **X1,X2,X6**
- plane grid : **X3,X4,X5**
- space frames : **X1 to X6**

Note:
- when a submodel is the current display, the table shows the restraints for the submodel connection points.

3.15.1.3 Springs

SPRINGS TABLE (units - ton meter)

Exit Goto Print Copy

NODE	S1	S2	S3	S4	S5	S6
165	.0	.0	675.0U-	.0	.0	.0
302	.0	.0	325.0	7500.0	.0	.0
309	.0	.0	325.0	7500.0	.0	.0

where:
S1-S6 = Spring constant in Global X1-X6 directions.
U+/U- = unidirectional spring

3.15.1.4 Restraint coord. systems

RESTRAINTS COORDINATE SYSTEMS

Exit Print Copy

System no.	JA	JB	JC
1	52	22	27

where:
- each local restraint coordinate system is defined by JA, JB, JC.

3.15.1.5 Rigid links

NODAL RIGID LINKS TABLE

Exit Goto Print Copy

	rigid link in plane			rigid link in axes		
Node	X1-X2	X3-X1	X2-X3	X1	X2	X3
3	3			3	3	
8	3			3	3	
9	3			3	3	

3.15.2 Beams

3.15.2.1 Beams

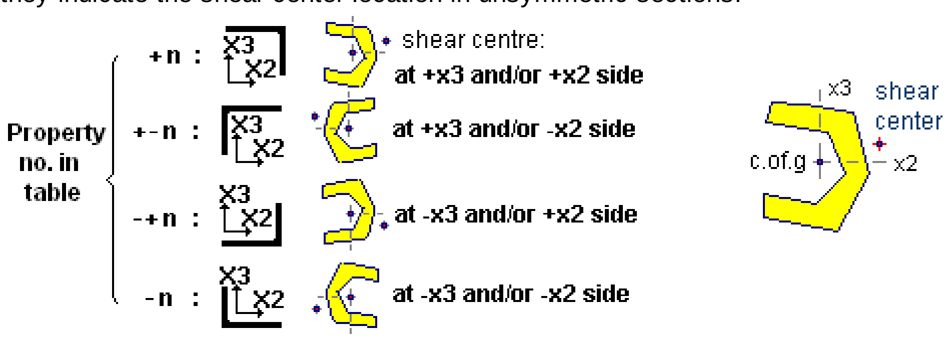
BEAM CONNECTIVITY TABLE

Exit Goto Print Copy

Beam No.	JA	JB	JC/ Beta	Release AJ mvmv	Length	prop no.	mat no.	Beam x2 direction cosines			offs. no.
40	40	41	32	y	4.690	3	1	.000	-1.000	.000	
48	43	35	32	c	1.820	4	1	.000	-1.000	.000	
50	45	37	32	t	1.820	4	1	.000	-1.000	.000	
52	52	53	57	nyy	4.690	DUMMY		.000	1.000	.000	
53	53	54	52	2 3	3.500	2	1	.000	.000	-1.000	
54	54	55	52		3.400	2	1	.000	.000	-1.000	

where:

- **JA** = start node
- **JB** = end node
- **JC** = JC node/or
  BETA = BETA angle (refer to Command Mode manual)
- **Releases** = Releases at JA,JB (refer also to Output - end conditions 401 table)
  The 6 columns are:
  A = Axial force:
  blank or **n** = no release
  y = release
  t = tension only
  c = compression only
  J = torsional release (both ends):
  blank or **n** = no release
  y = release
  MVMV = Moment and shear release at JA/JB
  blank or **n** = no release
  y = release in both x2 and x3 directions
  2 = release in x2 direction only
  3 = release in x3 direction only
  S = semi-rigid connection
- **LEN** = beam length
- **PROP** = property group number (or **DUMMY** beam) If the number is preceded by +/- signs, they indicate the shear center location in unsymmetric sections:



- **Mat** = material type (refer to material table)
- **COSINE** = cos. of angle between global X1,X2,X3 axes and local x2.

- **OFF NO** = OFFSET group number

Note:
- when a submodel is the current display, the table shows the beam list only for the current submodel.

3.15.2.2 Offsets

BEAM OFFSET DEFINITION TABLE (units - meter)

Exit Print Copy

		OFFSET AT JA			OFFSET AT JB		
NO.	System	X1	X2	X3	X1	X2	X3
1	Local no moment	1.000	0.000	0.000	1.000	0.000	0.000
2	Global	0.000	0.300	0.000	0.000	0.200	0.000

where:
No. = offset group number
System = GLOBAL or LOCAL
"**No moment**" indicates that the ☑ **Do not generate offset moments in beams option** was selected.
X1, .. , X3 = distance from JA/JB to end of offset in all directions

3.15.2.3 Beam end conditions

Display beam release data:

BEAM END CONDITIONS (units - kN meter)

Exit Goto Print Copy

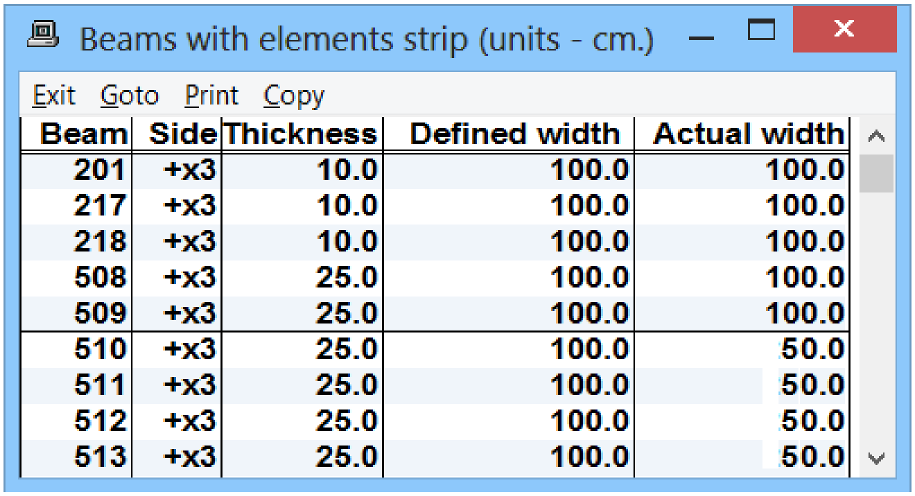
Beam			JA				JB			
no.	Axial	Tors.	M2	M3	V2	V3	M2	M3	V2	V3
281		Free					Free	Free		
408	Free									
424			S=1450.0	S=750.0						
432			Free	Free			Free	Free		
441	Tens.				Free	Free			Free	Free

Data is displayed at both beam ends (JA, JB)

"**Free**" : indicates a release
Axial: : **Tens** = tension only
Compr = compression only
M2,M3 : **S = ..**" indicates a semi-rigid release

3.15.2.4 Element strip

Display beams automatically offset from an element strip:

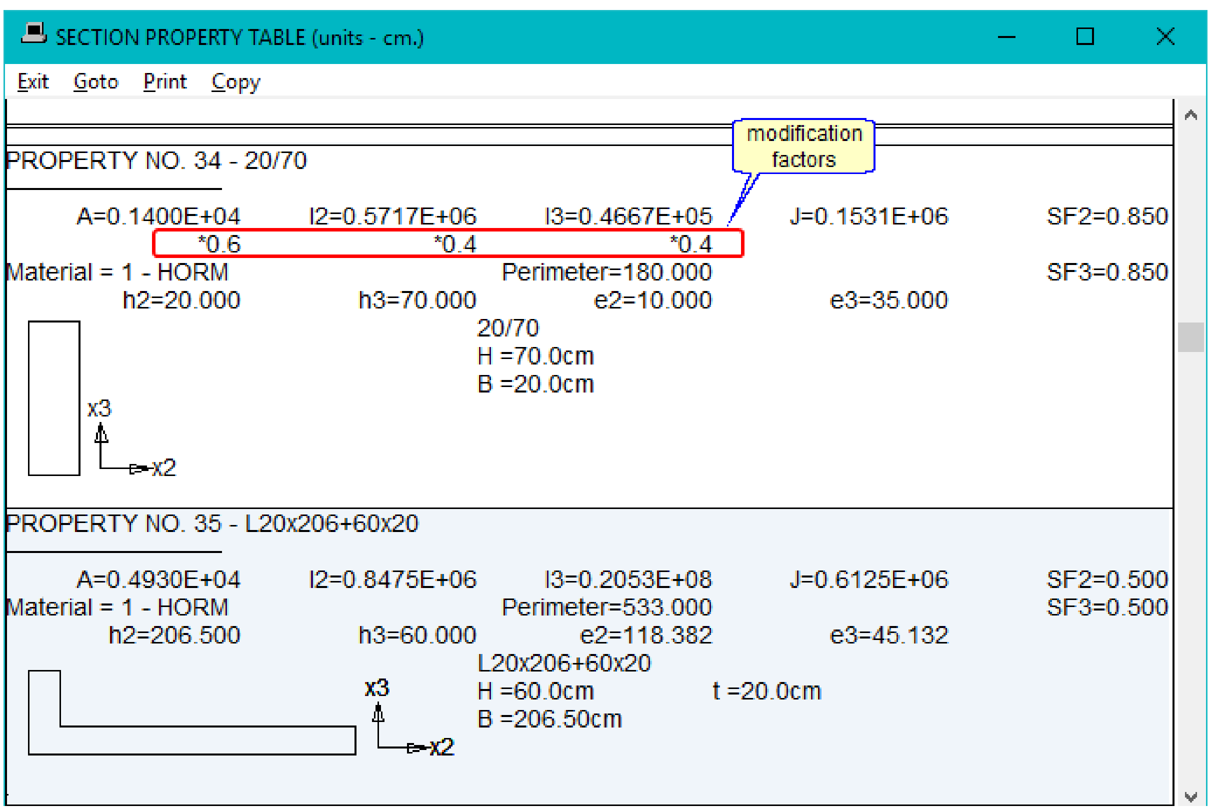


Beam	Side	Thickness	Defined width	Actual width
201	+x3	10.0	100.0	100.0
217	+x3	10.0	100.0	100.0
218	+x3	10.0	100.0	100.0
508	+x3	25.0	100.0	100.0
509	+x3	25.0	100.0	100.0
510	+x3	25.0	100.0	50.0
511	+x3	25.0	100.0	50.0
512	+x3	25.0	100.0	50.0
513	+x3	25.0	100.0	50.0

where:

Side	=	face of the beam, relative to the local axis, attached to the slab
Thickness	=	element thickness
Defined	=	Width input by the user.
Actual	=	Maximum width available (the program searches for slab edges).

3.15.2.5 Properties



Properties are about local axes, where

A = area

I2,I3 = moment-of-inertia about x2,x3.

J = torsional moment-of-inertia

SF2,SF3 = shear shape factors about x2,x3

Material = material number

h2,h3 = maximum dimensions parallel to x2/x3

e2,e3 = maximum distance from c.of.g to perimeter, parallel to x2/x3

Note:

- the gross values of A, I and J are shown; the "modification factors", if defined. are displayed below them.

For "Combined - General" sections:

```
PROPERTY NO. 6 - Combined 91x100 I+[]

     A=0.1702E+02   I2=0.2537E+03   I3=0.1136E+03   J=0.3730E+01        SF2=0.255
Material = 2 - STEE               Perimeter=59.860                      SF3=0.650
          h2=10.000       h3=9.100         e2=5.334        e3=4.846
   Zx=0.6220E+02   Zy=0.3383E+02           Esx=0.7         Esy=0.6  Cw=0.2088E+04
Principal axis angle=  97.4        Iu=0.2561E+03   Iv=0.1112E+03
Combined 91x100 I+[]
```

where:

Zx = plastic modulus about X

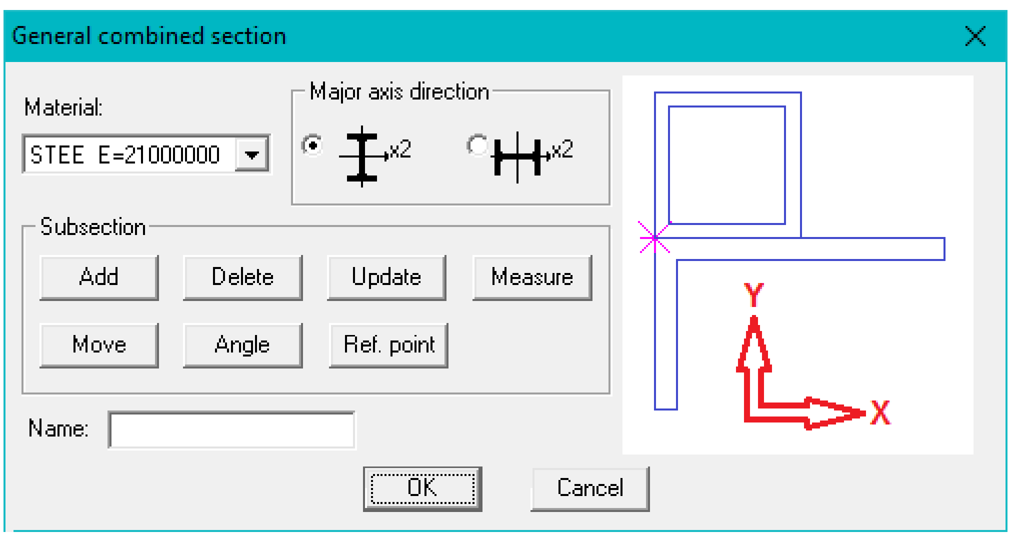
Zy = plastic modulus about Y

Esx = distance from center-of-gravity to shear center - X

Esy = distance from center-of-gravity to shear center - Y

Cw = warping constant

and "X" and "Y" refer to the horizontal and vertical axes when the section was defined:



Note:

- when a submodel is the current display, the table shows the property list for the current submodel.
- Units of all properties = PROPERTY No. 1 units.

3.15.3 Elements

3.15.3.1 Elements

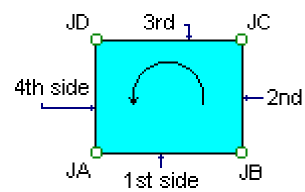
ELEMENTS TABLE

Exit Goto Print Copy

NO.	JA	JB	JC	JD	Area	thick.	mat	release	offset
70	130	131	150	151	1.0000	0.200	1		
71	141	142	161	162	1.0000	0.200	1		
72	142	143	162	163	1.0000	0.200	1		0.100
73	143	144	163	164	1.0000	0.200	1		
74	144	145	164	165	1.0000	0.200	1		
75	145	146	165	166	1.0000	0.200	1	nynn	
76	146	147	166	167	1.0000	0.200	1		
77	147	148	167	168	1.0000	0.200	1		

where:

JA,JB,JC,JD = 4 corner nodes in order of definition.
AREA = element surface area.
Thick = thickness
Mat = material type
Release = moment release along the element sides, where '**y**' indicates that a release has been defined. The first letter represents the side connecting nodes JA and JB. The next three letters represent the following three sides proceeding counterclockwise around the element, starting from JB.



Offset = offset distance perpendicular to the element plane. A positive value is in the +x3 direction.

Note:
- when a submodel is the current display, the table shows the element list only for the current submodel.

3.15.3.2 Slabs

SLAB DATA TABLE (units - meter)

Exit Goto Print Copy

NO.	Area	Prop.	TH	Level	Elem size	Rigid
1	18.0000	1	0.200	8.000	1.00/0.20	No
2	18.0000	1	0.200	0.000	1.00/0.20	No

where:

Area = area enclosed by the slab contour
Prop = property number
TH = slab thickness
Level = coordinate of axis perpendicular to the slab plane
Elem size= Element size and minimum size as specified by the user

Rigid = "Slab is rigid in plane" option

3.15.3.3 Solid elements

SOLID ELEMENTS TABLE

Exit Goto Print Copy

NO.	JA	JB	JC	JD	JE	JF	JG	JH	volume	mat
1	155	156	163	162	169	170	172	171	41.6667	CONC
2	169	170	172	171	173	174	176	175	41.6667	CONC
3	173	174	176	175	29	30	37	36	41.6667	CONC
4	132	133	140	139		177	178		23.9177	CONC
5	132	177	178	139		179	180		23.9177	CONC
6	132	179	180	139		181	182		23.9177	CONC
7	132	181	182	139		6	13		23.9177	CONC

TOTAL SOLID ELEMENTS VOLUME = 220.671

where:

JA, ... ,JH = 8 corner nodes. Note that 4 to 8 noded elements are possible.
Volume = element volume.
Mat = material type

Note:

- when a submodel is the current display, the table shows the element list only for the current submodel.

3.15.4 Walls

3.15.4.1 Walls

Display the wall segment data:

WALLS DEFINITION TABLE

Exit Goto Print Copy

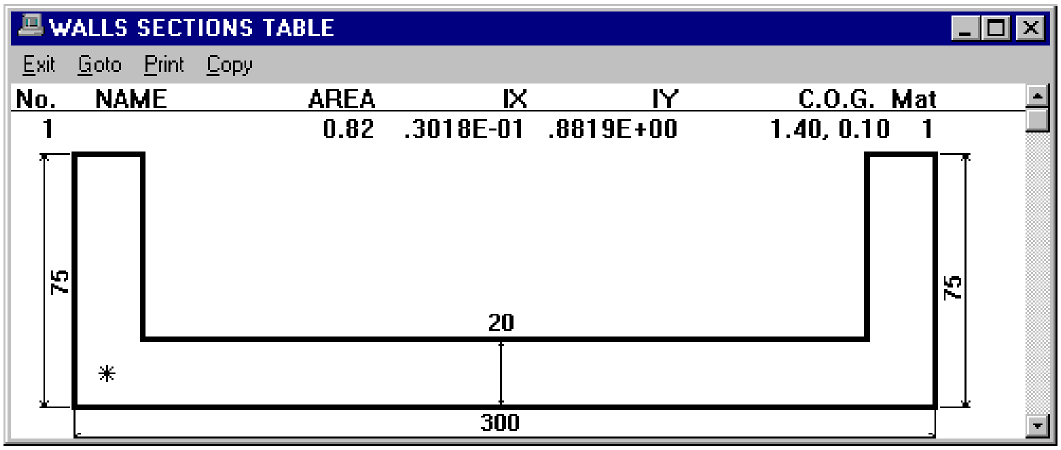
No.	JA	JB	Section	ANG	Length	Volume
5	276	282	1	0.0	3.00	2.46
6	282	288	2 20 cm wall	180.0	3.00	0.67
7	288	294	1	0.0	3.00	2.46
8	294	300	1	0.0	3.00	2.46
12	318	324	1	0.0	3.00	2.46
13	221	227	3	F90.0	3.00	0.00

TOTAL WALLS VOLUME = 17.886 TOTAL HEIGHT = 27.

No = segment number as displayed on the screen
JA,JB = start, end nodes of line segment (when defined)
Section = section number and title (if not default)
Ang = rotation angle, relative to the default orientation. **F** indicates that the wall was "flipped".
Length = Length of wall segment (JA-JB)
Volume = Wall area * Length (does not include coupling beams)

3.15.4.2 Wall sections

Display the dimensions and properties of all defined wall sections (in the model geometry units):



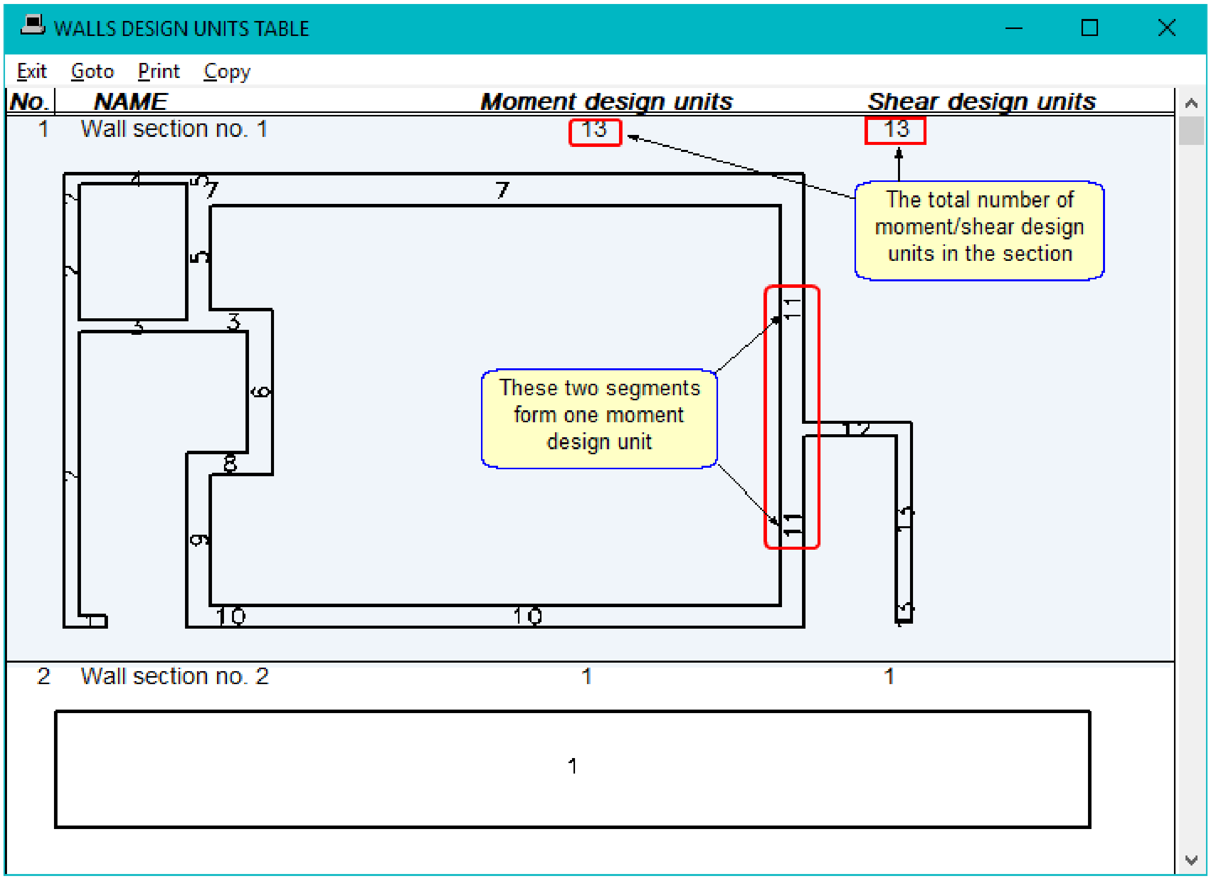
No = wall section number
Name = wall section title (if different from default)
Area = section area
Ix,Iy = moments-of-inertia (about centre-of-gravity)
C.O.G = centre-of-gravity - from "Insertion point" *
Mat = material

Note:

- The area and moment-of-inertia values are for the full section (without openings).
- the format for the section drawing (size, line thickness, text parameters) can be specified in the Output format 412 option.

3.15.4.3 Wall design units

Display the wall sections showing the location of the ***moment*** design units. For example:



3.15.5 General

3.15.5.1 Materials

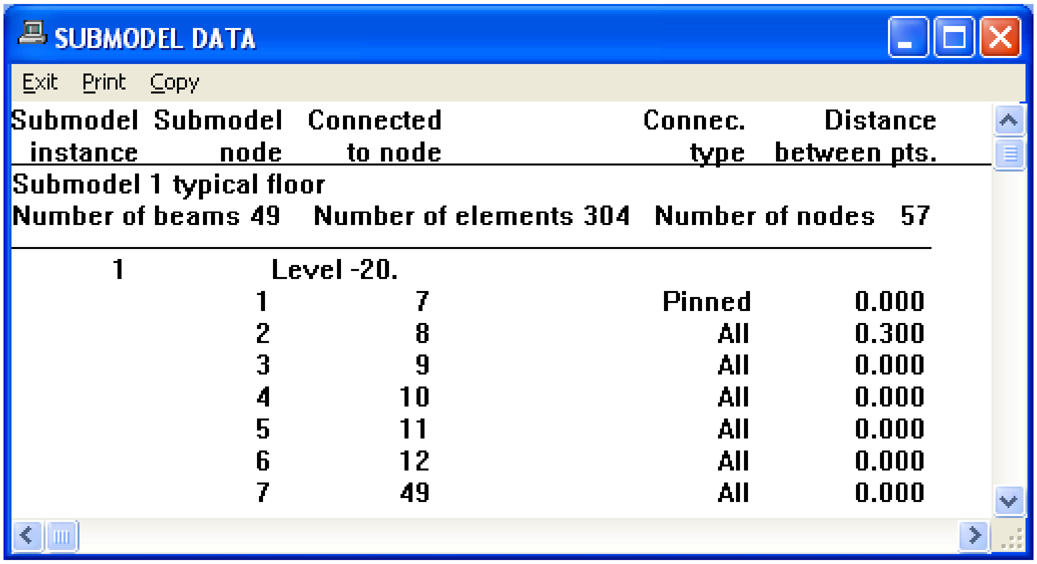
MATERIAL TABLE (units - ton meter)

Exit Print Copy

NO.	Name	Modulus of Elasticity	Poisson ratio	Density	Thermal coefficient	Shear modulus
1	CONC	0.3000E+07	0.150	0.2400E+01	0.00001000	0.1304E+07
2	STEE	0.2100E+08	0.300	0.7850E+01	0.00001000	0.8077E+07
3	ALUM	0.8000E+07	0.300	0.2700E+01	0.00002300	0.3077E+07

The units are displayed in the title bar of the table.

3.15.5.2 Submodel connections



For each submodel instance a list of the connection points is displayed with the following information:

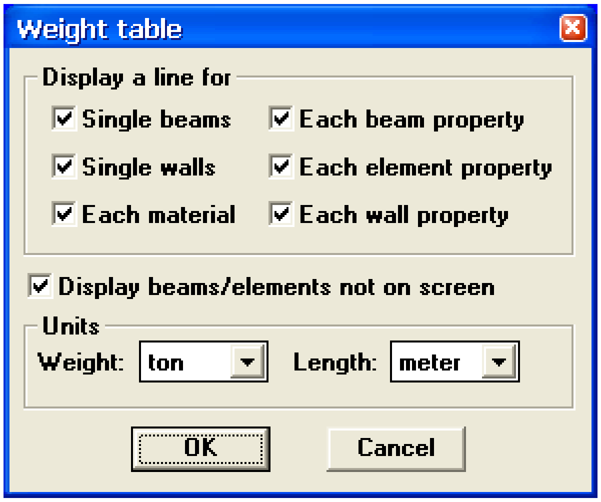
- the number of the connection point node in the submodel.
- the number of the node in the main model that it is connected to.
- the connection type: **Fixed** (X1 to X6 restrained), **Pinned** (X1 to X3 restrained) or a list of restrained degrees -of-freedom.
- the distance between the submodel node and the main model node. If the value is not zero, the nodes are always connected by a 'rigid link'.

3.15.5.3 Weights

Display the following tables showing beam, element and wall weights.

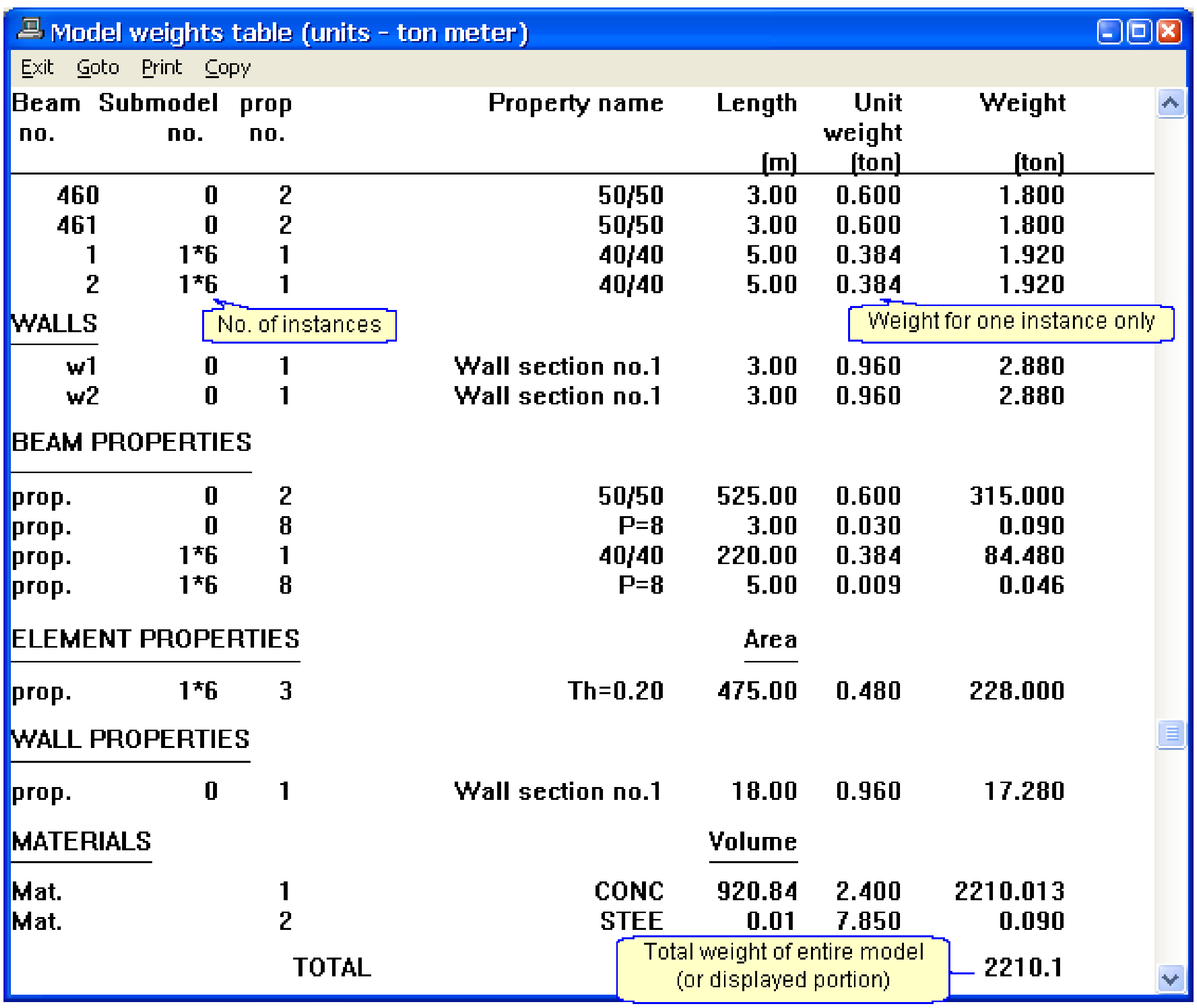
- section area, weight and length of each beam
- weight and length summary of each beam property
- weight and area summary of each element property
- weight and length of each wall section
- weight and volume of each individual wall.
- weight of all elements for each material type

Select the tables to display:



Note: solid elements are not displayed

For example:

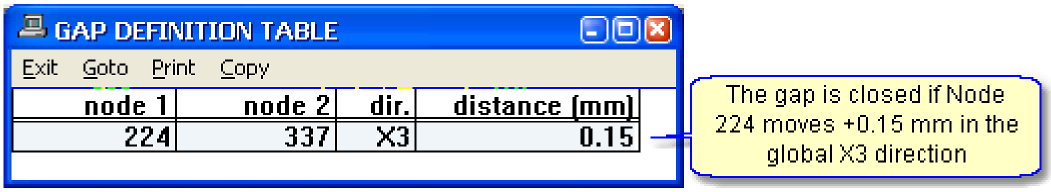


Beam no.	Submodel no.	prop no.	Property name	Length [m]	Unit weight [ton]	Weight [ton]
460	0	2	50/50	3.00	0.600	1.800
461	0	2	50/50	3.00	0.600	1.800
1	1*6	1	40/40	5.00	0.384	1.920
2	1*6	1	40/40	5.00	0.384	1.920
WALLS						
w1	0	1	Wall section no.1	3.00	0.960	2.880
w2	0	1	Wall section no.1	3.00	0.960	2.880
BEAM PROPERTIES						
prop.	0	2	50/50	525.00	0.600	315.000
prop.	0	8	P=8	3.00	0.030	0.090
prop.	1*6	1	40/40	220.00	0.384	84.480
prop.	1*6	8	P=8	5.00	0.009	0.046
ELEMENT PROPERTIES				Area		
prop.	1*6	3	Th=0.20	475.00	0.480	228.000
WALL PROPERTIES						
prop.	0	1	Wall section no.1	18.00	0.960	17.280
MATERIALS				Volume		
Mat.		1	CONC	920.84	2.400	2210.013
Mat.		2	STEE	0.01	7.850	0.090
		TOTAL				2210.1

3.15.6 Nonlinear

3.15.6.1 Gap

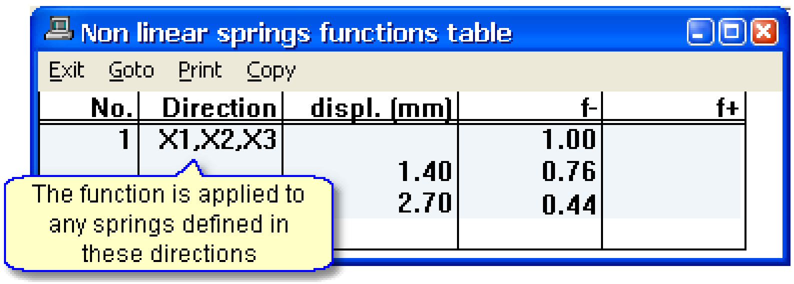
Gap data



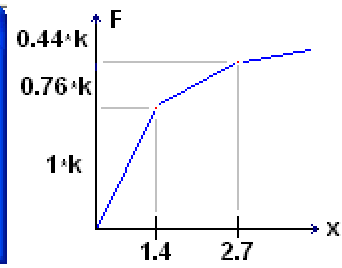
node 1	node 2	dir.	distance (mm)
224	337	X3	0.15

3.15.6.2 Spring d-F functions

Spring d-F functions

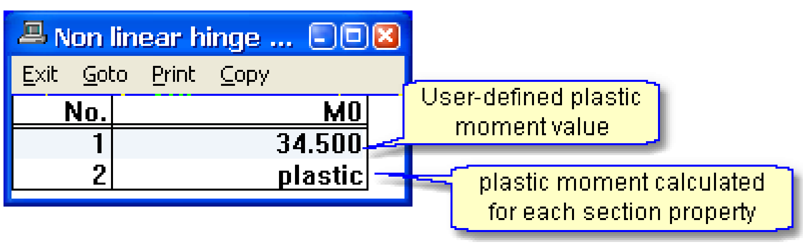


No.	Direction	displ. (mm)	f-	f+
1	X1,X2,X3		1.00	
		1.40	0.76	
		2.70	0.44	



3.15.6.3 Hinge

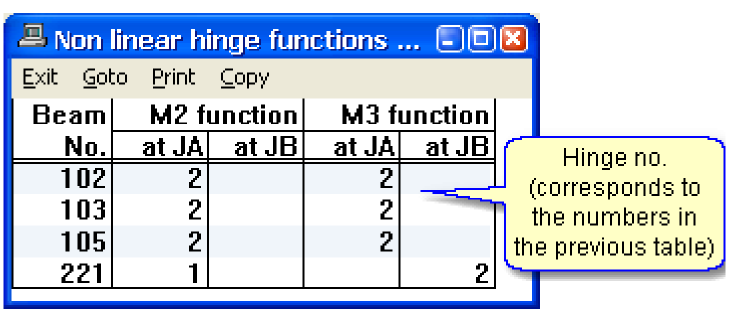
Hinge types



No.	M0
1	34.500
2	plastic

3.15.6.4 Hinge assignment

Hinge assignments



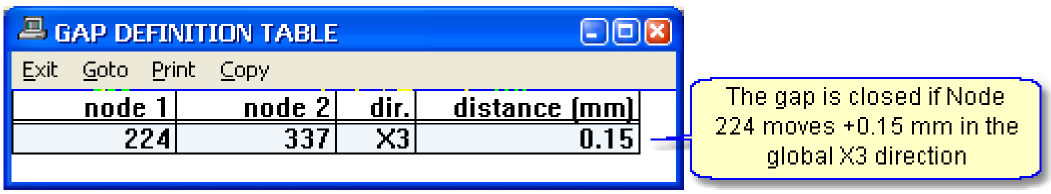
Beam No.	M2 function at JA	M2 function at JB	M3 function at JA	M3 function at JB
102	2		2	
103	2		2	
105	2		2	
221	1			2

3.15.7 Non-linear

Select one of the following options

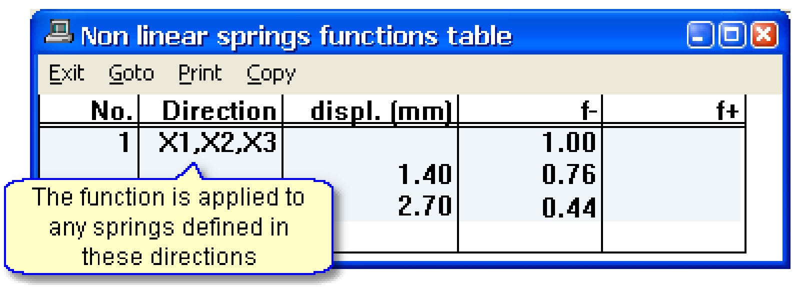
- Display gap data
- Display spring d-F functions
- Display hinge types
- Display hinge assignments

Gap data

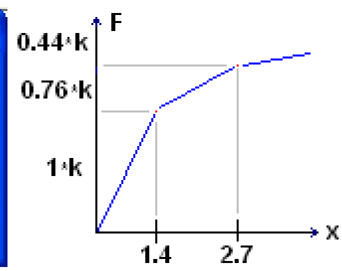


node 1	node 2	dir.	distance (mm)
224	337	X3	0.15

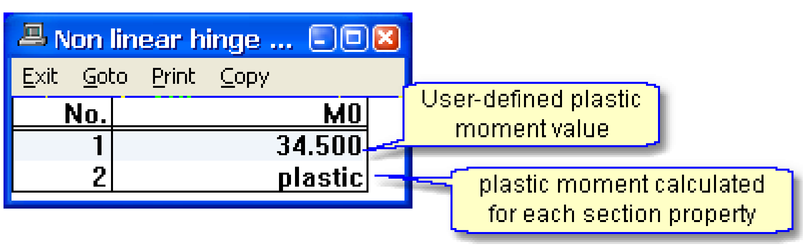
Spring d-F functions



No.	Direction	displ. (mm)	f-	f+
1	X1,X2,X3		1.00	
		1.40	0.76	
		2.70	0.44	

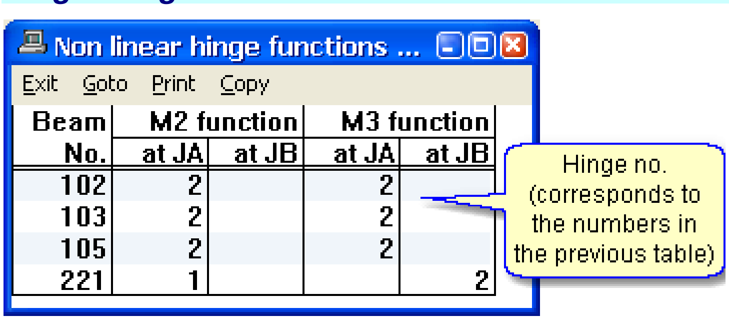


Hinge types



No.	M0
1	34.500
2	plastic

Hinge assignments

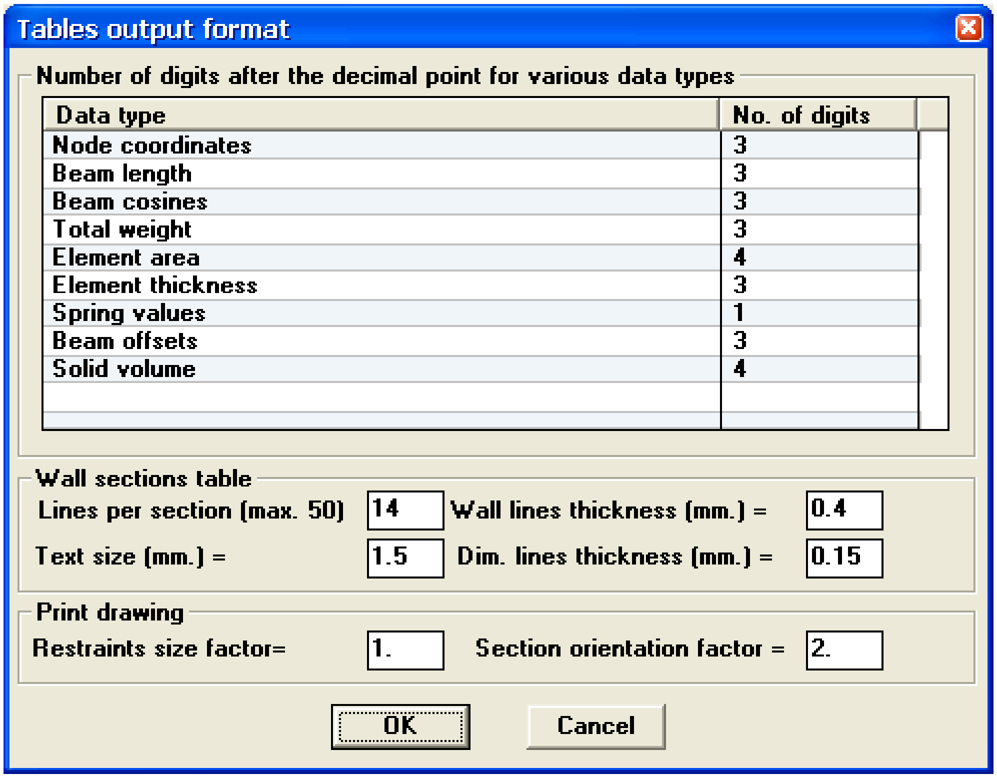


Beam	M2 function		M3 function	
No.	at JA	at JB	at JA	at JB
102	2		2	
103	2		2	
105	2		2	
221	1			2

3.15.8 Output format

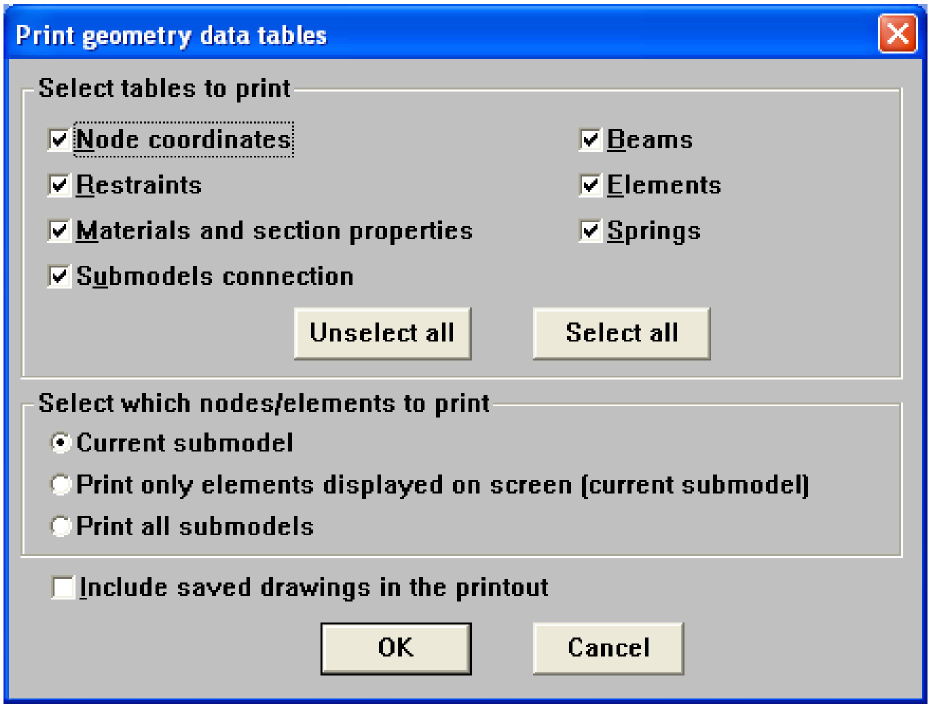
Specify:

- the number of digits after the decimal point for each value in the output tables
- the format for the Display wall sections 406 option
- the size of the beam sections and restraint symbols superimposed in the drawing (restraint size applies to the print option only).



3.15.9 Print

Print selected tables for all or part of the model.



Select tables

Geometry tables set to ☑ are printed.

Select nodes/elements

Current submodel
data for the entire current submodel (or Main model) are printed, even elements if not displayed on the screen.

Elements displayed on screen
only data for beams/nodes/elements currently on the screen is printed.

All submodels
data for the entire model is printed

Include drawings

☑ **Include saved drawings**
add drawings created with the **Save for "print/edit drawing"** option.

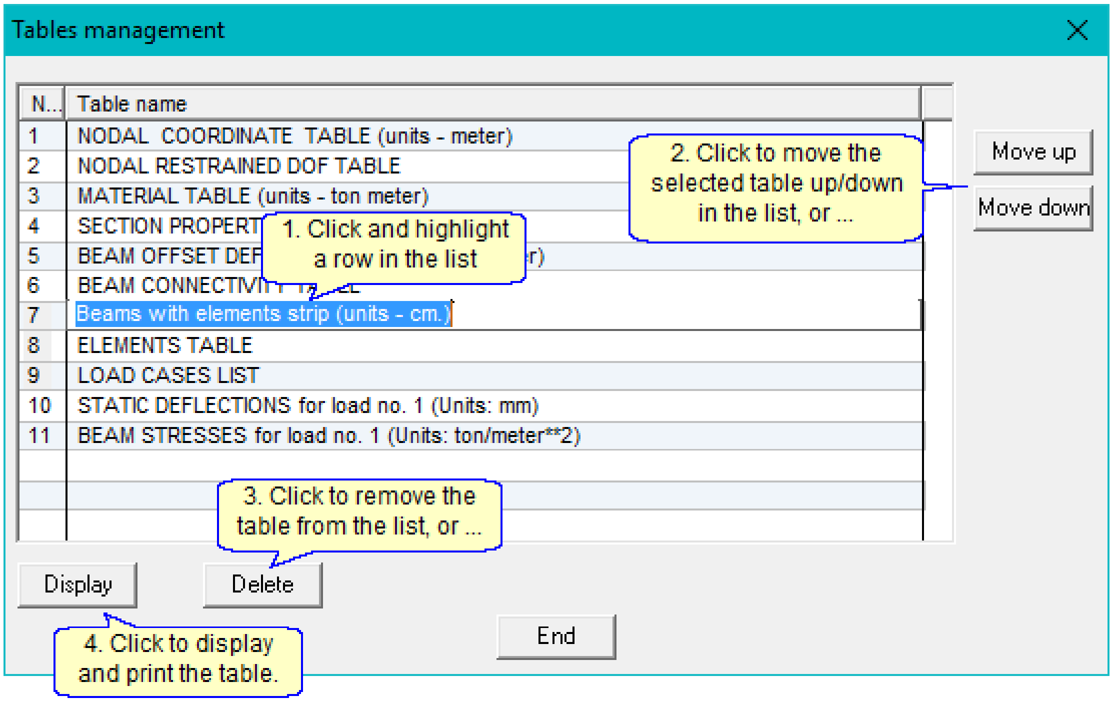
The program allows you to place drawing between any of the tables; refer to Print order.

Note:

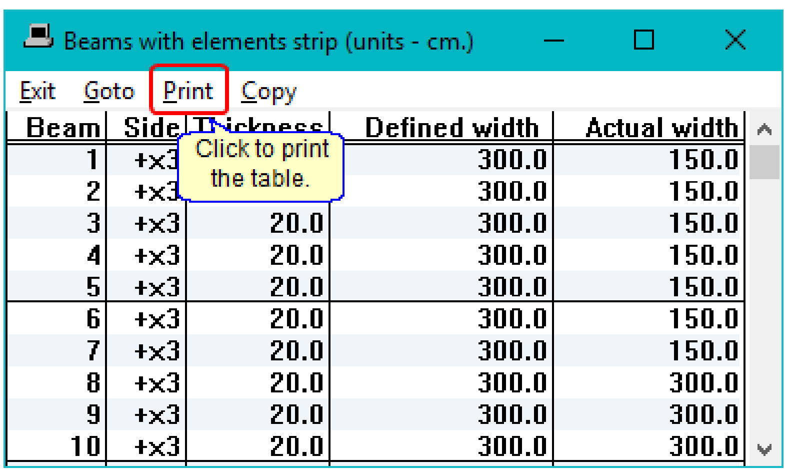
- Tables may be written to ASCII files in user-specified format using the STBatch utility.

3.15.10 Saved table management

Delete tables, rearrange the order of tables or display/print tables that were saved using the **Save output for report generation** option. The program displays a list of saved tables:



If Display is selected, the standard print option is available in the menu bar:



Part IV

Loads

4 Loads

- The program can solve the model for multiple load cases. Each load case may consist of joint loads, beam loads, element pressures and support displacements.
- Load cases may be combined to form a new load case.
- Beam and element loads may also be defined relative to the global coordinate system; standard load patterns may be stored in a file and recalled.
- Each load case may be assigned to a different geometry 'stage'.
- All defined loads for the current load case are displayed graphically superimposed on the geometry.

New load	Begin the definition of a new 417 loading case.
Existing load	Revise 419 the loads in an ***existing*** load case.
Delete	Delete 420 an existing load case.
Deactivate	Deactivated 483 load cases are not solved but are not erased.
Moving load	Automatically generate a series of "moving 484" load cases from a single basic case. The global loads in the basic load case will be offset by a constant increment in each successive generated case.
Chess load	Automatically generate a series of "chess 486" load cases with alternating patterns of live load from basic load cases containing the dead and live loads on all spans. The patterns are arranged according to Code requirements for calculating maximum and minimum moments in beams.
P-delta	Calculate second-order (P-D 489) forces and moments.
Wind load	Define wind 492 load cases according to Code requirements.
Copy load	Copy 531 an entire load case.
Sway	Define sway 533 unit load cases at specified nodes. These load cases are required for the sway/drift control option in the Steel postprocessor.
Combinati...	Define load combinations 534. These combinations can either be solved or they can be saved for the Results module.
1+2= Solve	Solve 558 the model
Submodel	Display an existing submodel instance on the screen and define loads on it. Note that loads applied to one instance of a submodel may be applied at the same time to all other instances of the same submodel.

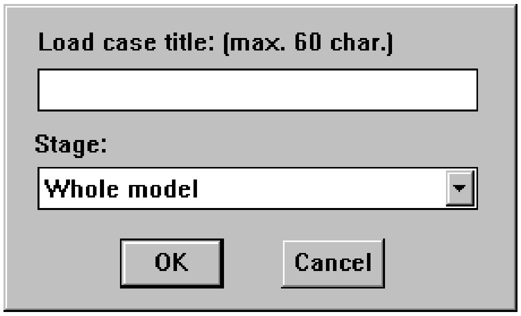
From the menu bar:

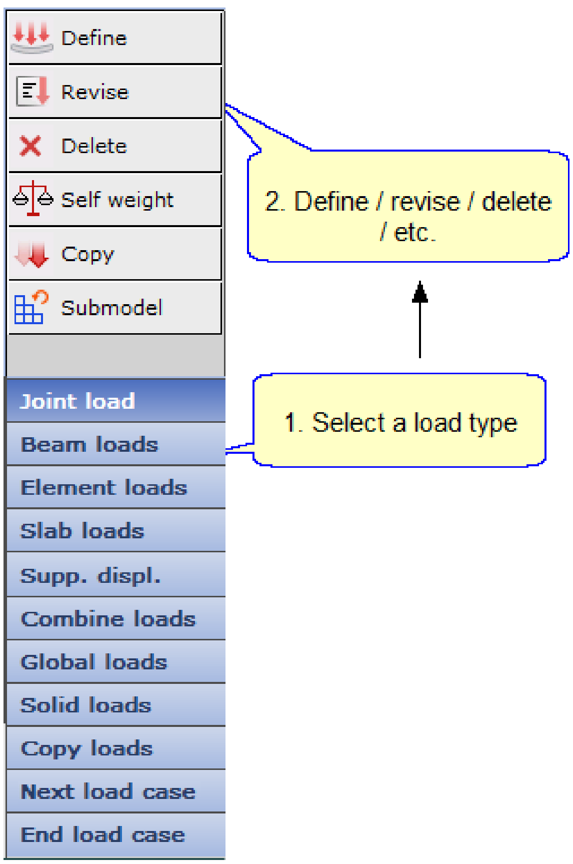
File Edit Zoom Rotate Display Draw ReMove Undo Output Help

Refer also to:
- Command Mode for details on defining loads by typing in commands.
- Display loads for details on the conventions for the graphic display of the loads on the model.

4.1 Define load case

Enter the title of a new load case. Each case may be assigned to a different stage:





Joint loads 423
- Concentrated forces and moments applied at the model's nodes, defined relative to the global coordinate system or any arbitrary local system.
- Self-weight may be defined as a joint load; the program applies the reaction of the uniform self-weight load of each beam/element as a concentrated load at the end nodes.

Beam loads 428
- Uniform, linear, concentrated loads.self-weight and temperature (expansion/contraction or gradient)
- Beam loads can be defined either parallel to the local beam axes or the model global axes.

Element loads 449
- Element pressures are applied to the entire surface area of quadrilateral or triangular elements.
- The pressure is not necessarily applied normal to the element surface; the load can be applied in any

of the local or global axis directions. In all cases, the total load applied is the pressure multiplied by the surface area of the element.

Support displacement 463

- Support displacements may be entered in the direction of any degree-of-freedom, including rotation.
- These displacements should only be defined at nodes which have been restrained in the same degrees-of-freedom (Restraints).

Combine loads 464

- combine existing load cases to create a new load case; the existing load cases may be increased by a factor.
- Note that the ***STRAP*** result module also has an option for combining load cases. In general, it is more convenient to define combinations ***after*** the solution rather than at this stage, because:
  - the solution time is decreased
  - combinations may be revised without solving the model again
- Since the P-Delta effect is ***non-linear***, the rules of superposition do not apply. Therefore, load combinations for models with P-Delta must be defined here in LOADS, rather than in the results module.
- This option may also be used to insert an existing loading case into a new loading case.

Global loads 466

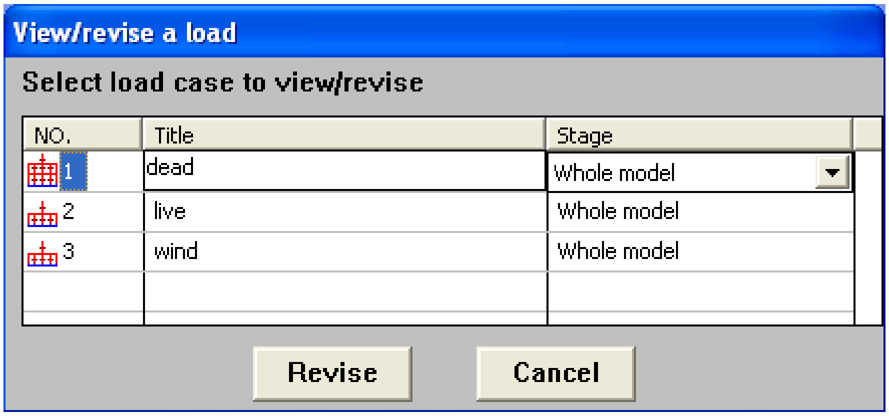
- Load locations may be defined relative to the global coordinate system. The program locates the nodes and elements surrounding the "global loads" and converts the loads to equivalent joint loads or element loads, as requested by the user.
- This option is useful in models characterized by load patterns which do not coincide exactly with the nodes or elements, such as bridges.
- Global loads can be entered directly or may be stored and recalled from a file.

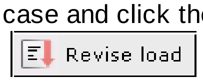
Solids loads 478

Define temperature loads or self-weight for solid elements. All other loads (pressure, linear loads, concentrated loads, etc) must be defined by applying them to beams or plate elements (dummy or regular) defined parallel to the solid elements surfaces.

4.2 Revise load case

Revise the loads in an existing load case or revise the stage assigned to the load case. The program displays a list of the existing cases:



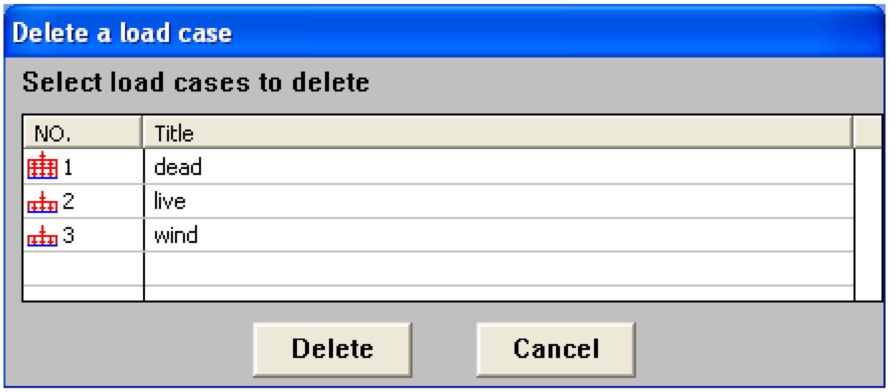
- to revise a title - click on the title in the relevant row and type in the corrections.
- to select a different stage - click on the relevant row, click the to open up the listbox and select a stage from the list.
- to revise loads select a load case; move the into any cell on the line in the table corresponding to the load case and click the mouse.
- click the 

 icon
- continue as explained in Define loads 417

4.3 Delete load case

Delete an entire load case:

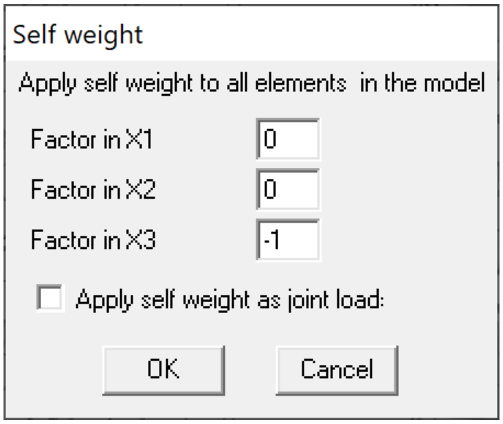
The program displays a list of existing load cases;



- Highlight a load case using the and click the mouse.
- Click the Delete button
- Click OK in the following menu to confirm the deletion.

4.4 Self weight - All elements

Apply self-weight to ***all*** elements (beams, elements, slabs and walls) in the model.



Note:
- Self-weight can also be applied to selected elements by selecting **Beam loads/ Elements loads/ Wall loads/ Slab loads - Define - Self-weight**.
- If self-weight is defined more than once on an element, only the last definition is used by the program, i.e. self-weight commands are ***NOT*** superimposed by the program.
- The program automatically computes the load by multiplying the elements area (depended on the element type: beams, elements, walls, etc.) by the material density:
  - If the material is User defined, verify that the density was defined.
  - incorrect loading result if fictitious beams are defined with an arbitrarily large area and their self-weight is applied; define the material density in these beams equal to zero or exclude them from the list of beams.
- Do not define self-weight on tension/compression only beams that are sloped.(i.e. an axial component is applied) The program always applies these loads to the model, even if these beams are not active for the relevant load case.
- For composite beams, the self weight includes the weight of the steel section and the weight of the concrete topping.

Factor

The self-weight can be applied in any direction multiplied by any factor:

Examples:
Assuming that X2 is the height axes of the model -
- apply self-weight as a vertical service load:

  Factor for X2 -1.00
- apply 10% of the self-weight as a horizontal load:

  Factor for X1 0.10

Note:
- The self-weight is automatically calculated and applied for ***all*** elements in the model.
- To delete self-weight loads, define again and set all factors equal to zero.

As joint load

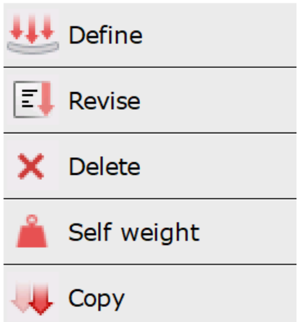
The self-weight can be applied in any direction multiplied by any factor:
Apply the self-weight as a load at the nodes. The program computes the reaction of the uniform self-weight load of each element and apply it as a concentrated load at the end nodes of the element.

Note:

- Self-weight applied to nodes is ***NOT*** displayed graphically on the model; a footer line "**Self-weight on nodes ...**" is displayed (and printed) below the model.

4.5 Joint loads

Joint loads are defined relative to the global coordinate system or to any arbitrary local system defined by three nodes.

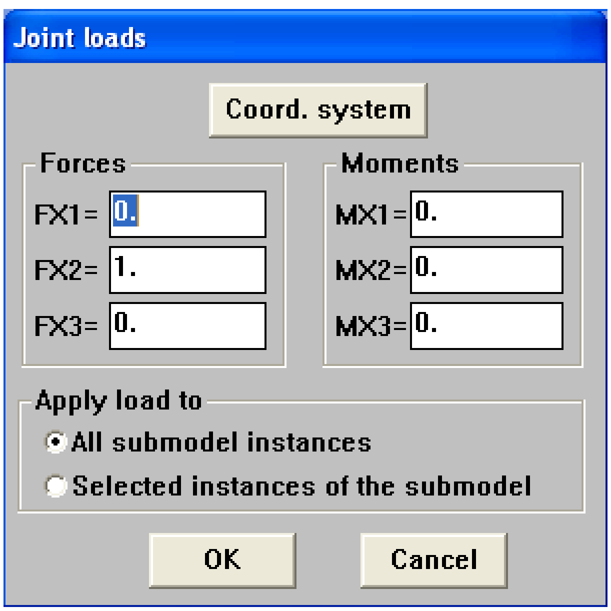


Note:

- The self weight option is available **only** in the main model. Before clicking self weight, select Main model from the submodels drop-down menu.

4.5.1 Define

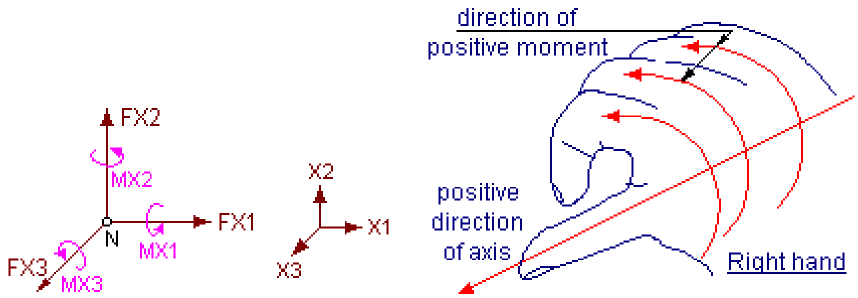
Define the joint loads - forces and moments.



- Enter the load values; note that loads in more than one global direction can be defined at the same time.
- The loads are defined relative to the global axes by default; click Coord. system to define loads relative a local system.
- Select the nodes where the loads are to be applied using the standard Node Selection option.

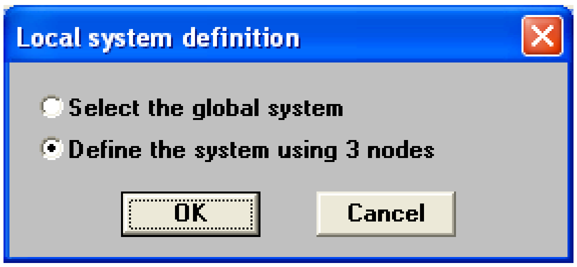
Forces / moments

The positive force and moment conventions are:

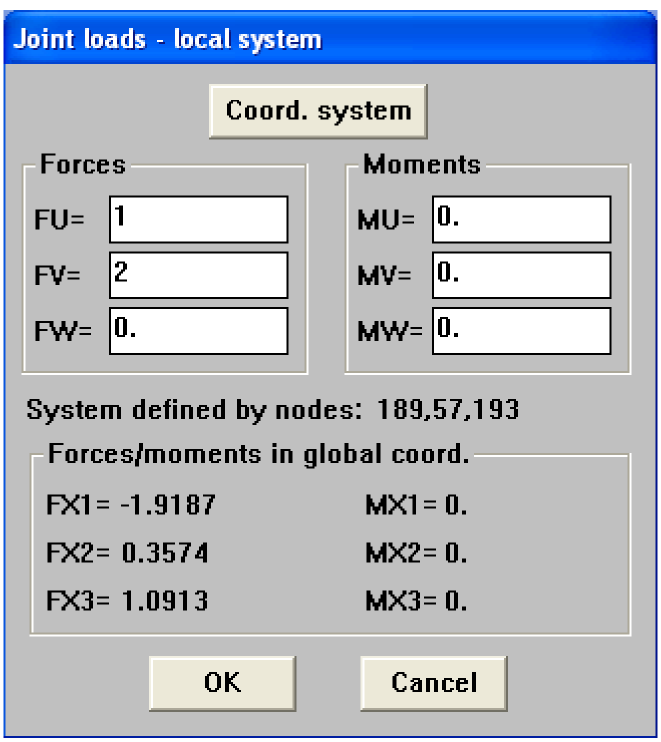


Coordinate systems

Joint loads may be defined relative to any local system U,V,W defined by three nodes:



- U lies along the line joining nodes 1 and 2 and points in the direction of 2.
- V lies on the plane formed by 1-2-3, is perpendicular to 1-2 and points in the general direction of node 3
- W is defined by the right-hand rule

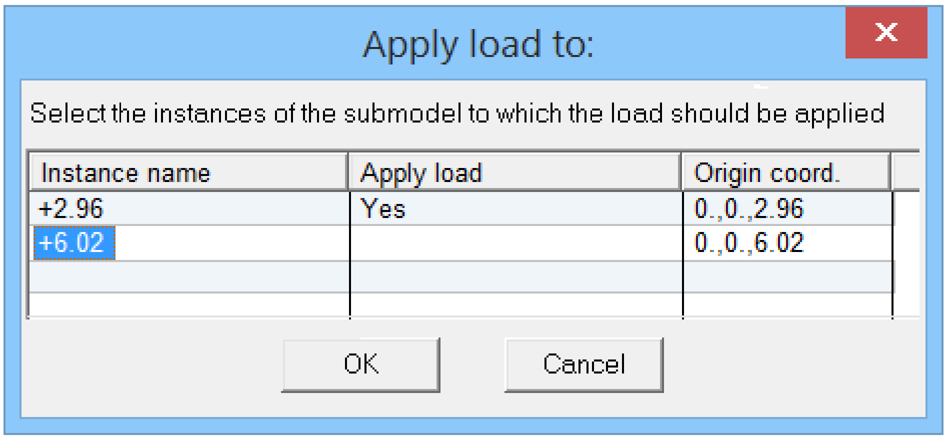


- Enter the load values; note that loads can be defined in more than one local direction at the same time. The program displays the global components of the loads in the bottom half of the dialog box.
- Select the nodes that the loads are applied to using the standard Node selection option

Note:
- Joint loads are always displayed with their global components.

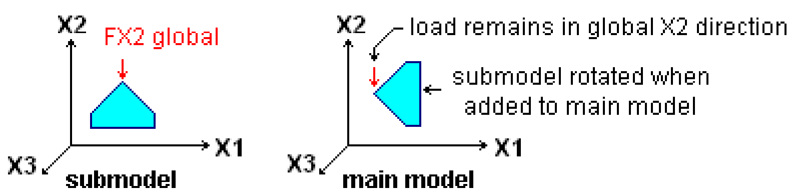
Submodels

This option is displayed only when a submodel is currently displayed and there is more than one instance attached.



Click on a line to toggle the **Yes/No** option.

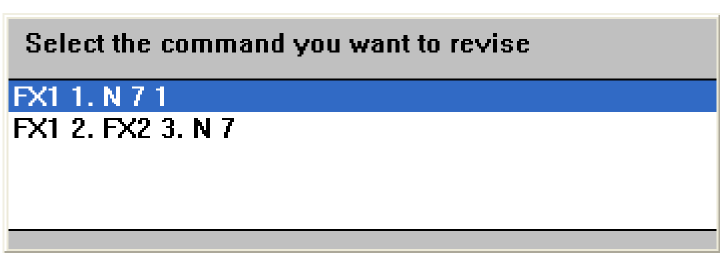
Note:
- if the submodel joint loads are defined relative to the global coordinate system the program uses the ***Main model*** system, not the submodel system, when applying them. For example:



Loads defined relative to a local system are not rotated; define a local system parallel to the submodel global system (**Define using 3 nodes**) to avoid rotating the loads in the situation described above.

4.5.2 Revise

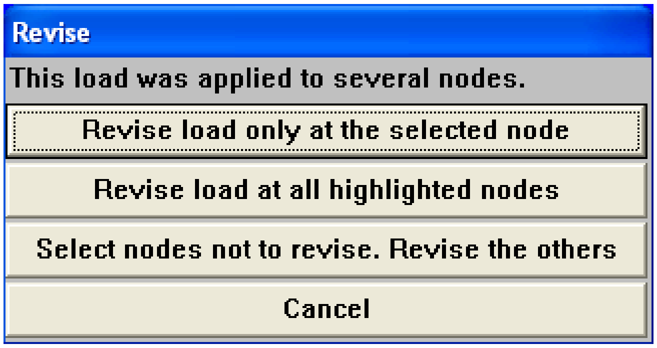
Select ***one*** node with a joint load to be revised/deleted using the standard single node selection option.
- If more than one load was defined for the selected node you must decide which one of them to revise. For example, a node with a two joint loads:



Select one of the loads.

- The corresponding current load values are displayed on the screen; revise them.
- If the selected load was applied to more than one node in the same command, the program highlights

the nodes with a ■; you may correct the load on all of them or some of them at the same time:



- **Revise load only for the selected node**
  Load on the selected node only are revised.
- **Revise load for all highlighted nodes**
  Load are revised on all nodes highlighted with the ■.
- **Select nodes not to revise**
  Use the standard node selection option to select some of the nodes with the ■; the loads on the selected nodes are ***not*** revised.

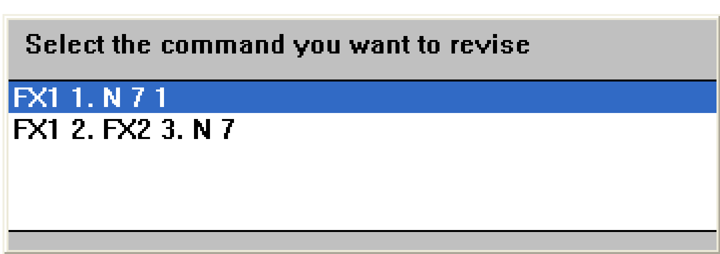
Note:

- Loads that were added to the current load case by the Combine loads 464 option cannot be corrected; the original load case must be revised.

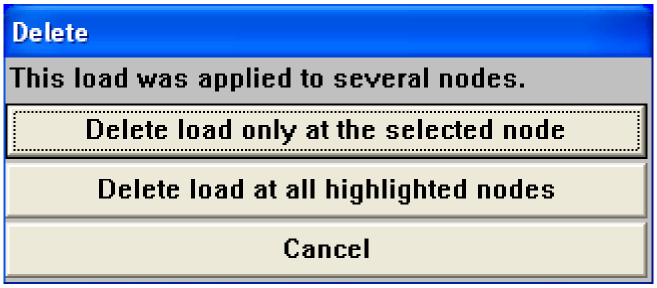
4.5.3 Delete

Delete all loads on nodes or delete selected loads:

- Select the node with the load to be deleted using the standard single node selection option.
- If you specified Delete a selected load**,** the program displays a list of loads applied to the node; select one. For example, a node with a two loads:



- If the selected load was applied to more than one node in the same command, you may correct the load on all of them at the same time (the nodes will be highlighted):



- **Delete load only for the selected node**
  Load are deleted on the selected node only
- **Delete load for all highlighted nodes**
  Load are deleted on all nodes highlighted with the ■

Note:

- Loads that were added to the current load case by the Combine loads 464 option cannot be deleted; the original load case must be revised.

4.6 Beam loads

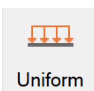
Beam loads may be defined either in a direction parallel to one of the global coordinate axes or parallel to one of the local coordinate axes of the beam.

The options displayed for **Beam loads** are:

Note:

- Do not define beam loads with an axial component (e.g. axial temperature, global linear load on a sloped beam, etc) on tension/compression only beams. The program always applies these loads to the model, even if these beams are not active for the relevant load case.

4.6.1 Define

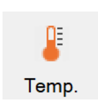


A uniformly distributed load 429 along the ***entire*** span length.

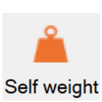
A linear distributed load segment 432 anywhere along the span or a uniformly distributed load acting on part of the span.

A point load 434 acting anywhere along the span

A linear load acting anywhere across a line of beams 436.



Axial expansion/contraction or a gradient 437 on beam height



Apply the self-weight 440 of beams as a uniform load

A parabolic or straight prestress force 441 (axial force and eccentricity)

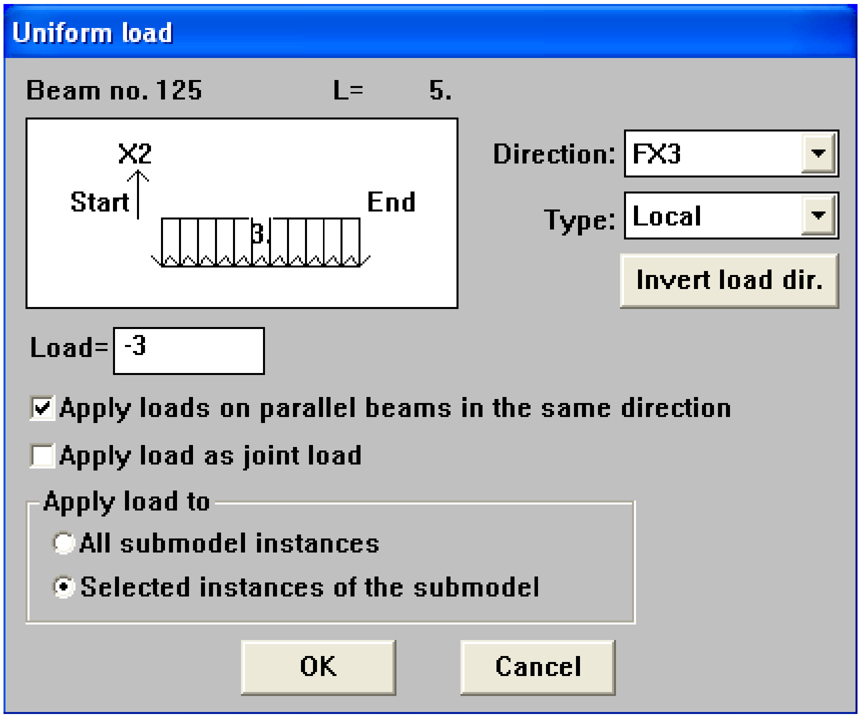
apply the loads generated by a beam that is initially too short or too long 445

Apply a load to the perimeter surface area 445 of the beam

4.6.1.1 Uniform

Define a uniformly distributed load - force or moment - along the entire length of a beam.

- select the beam to which to apply the loads using the standard Beam selection option.
- enter the load value, type and direction:



Direction

Forces: select the local/global axis the load acts parallel to.

FX1 - parallel to X1 or x1 - relevant for plane and space frames
FX2 - parallel to X2 or x2 - relevant for plane and space frames
FX3 - parallel to X3 or x3 - relevant for grids and space frames

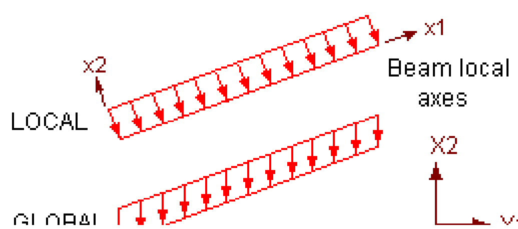
Moments: select the axis that the load acts ***about***.

MX1 - about local x1 - relevant for grids and space frames

Type

Local : the load is parallel to the ***beam local*** axis specified.

Global: the load is parallel to the ***global*** axis specified and is applied to the total beam length.



Global projected: the load is parallel to the axis specified, but is applied to the projected beam length as projected onto the specified global axis (uniform and linear loads only).

Load =

Enter the load according to the current units.

Remember that a positive load is in the positive direction of the specified local / global axis. In the examples above, all loads should be defined with ***negative*** values.

Invert load direction

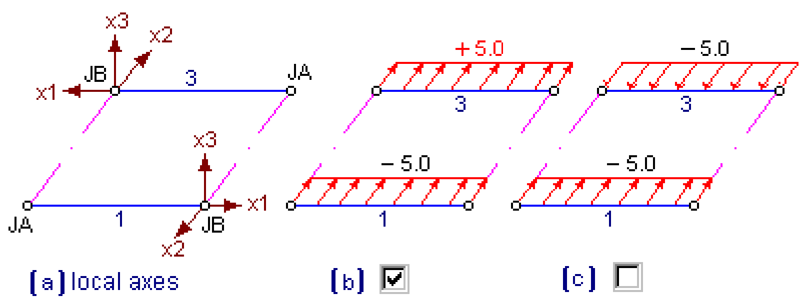
The load direction is displayed graphically in the small box; click this button to reverse the direction.

Apply loads on parallel beams

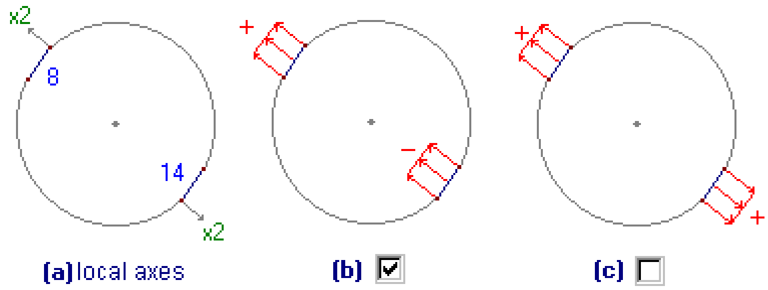
For beam loads applied in a local axis direction, the program checks whether the relevant local axis points in the same direction for all of the parallel beams in the group. If not, the user may instruct the program to apply all of the loads in the same direction.

The importance of this option is explained by two examples:

- **Example 1:**
  Parallel Beams 1 and 3 in Figure (a) below are selected together and a load of -5.0 is defined parallel to the local x2 axis. However, note that the local x2 axes of beams 1 and 3 point in opposite directions. The direction box is displayed for Beam 1
  ☑ **Apply loads on parallel beams in the same direction**
  The loads are applied in the same direction as shown in Figure (b)
  ☐ **Apply loads on parallel beams in the same direction**
  The loads are applied in opposite directions as shown in Figure (c).



- **Example 2:**
  Beams 8 and 14 are on opposite sides of the surface of a cylinder. The local x2 axis of both beams point outwards, i.e. in opposite directions. The direction box is displayed only for beam 8 because the two beams are parallel.

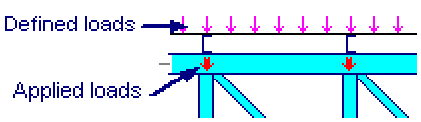


☑ **Apply loads on parallel beams in the same direction**
The loads are applied in the same direction as shown in Figure (b)

☐ **Apply loads on parallel beams in the same direction**
The loads are applied in opposite directions as shown in Figure (c). This is most likely the correct option.

Apply as joint load

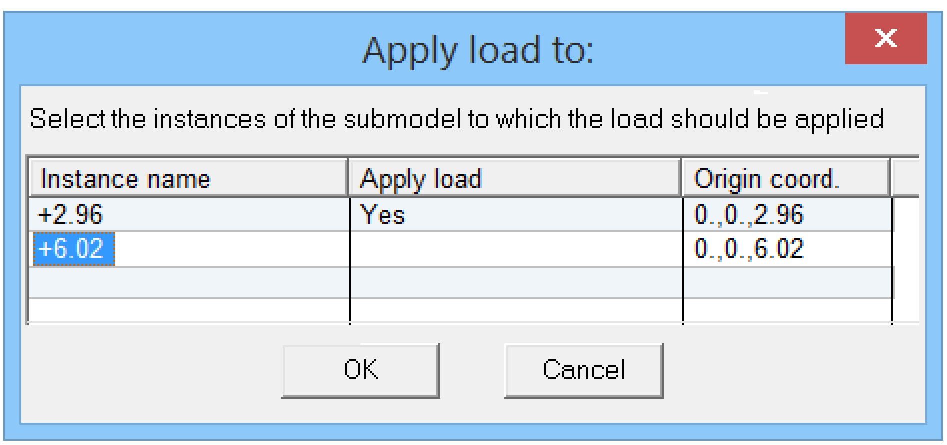
In many cases it is convenient to define a load as a linear load even when the load is applied to the model only at the supporting nodes. For example:



☑ the program calculates the end reactions from the beam loads and applies them to the nodes as joint loads ; no bending moments and shear are generated in the beam.

Submodels

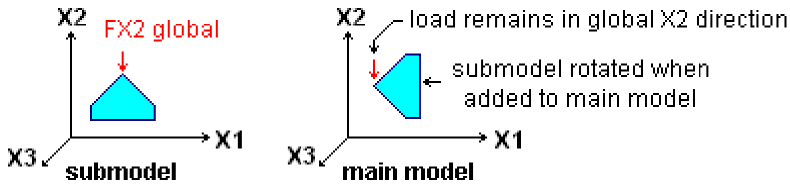
This option is displayed only when a submodel is currently displayed and there is more than one instance attached.



Click on a line to toggle the **Yes/No** option.

Note:

- if the submodel beam loads are defined relative to the global coordinate system the program uses the ***Main model*** system, not the submodel system, when applying them. For example:



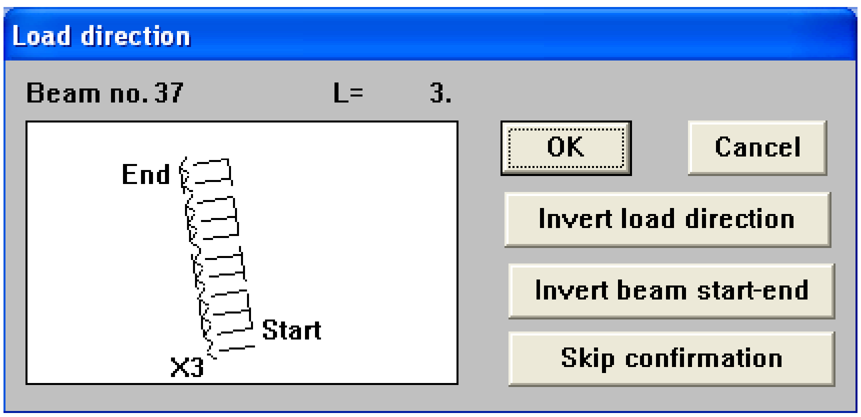
Loads defined relative to the beam local coordinate system are rotated.

Beams not parallel:

The beam is displayed in the box with the same orientation as on the screen.

If more than one beam is selected, a separate box is displayed for each group of beams with parallel local x2/x3 axes, i.e. the box is displayed only once if relevant axis of all the beams selected are parallel. Note that the positive direction of the axis is not important; refer also to Apply loads on parallel beams in same direction 430.

If the box is displayed for a second group of beams, the following options are available:



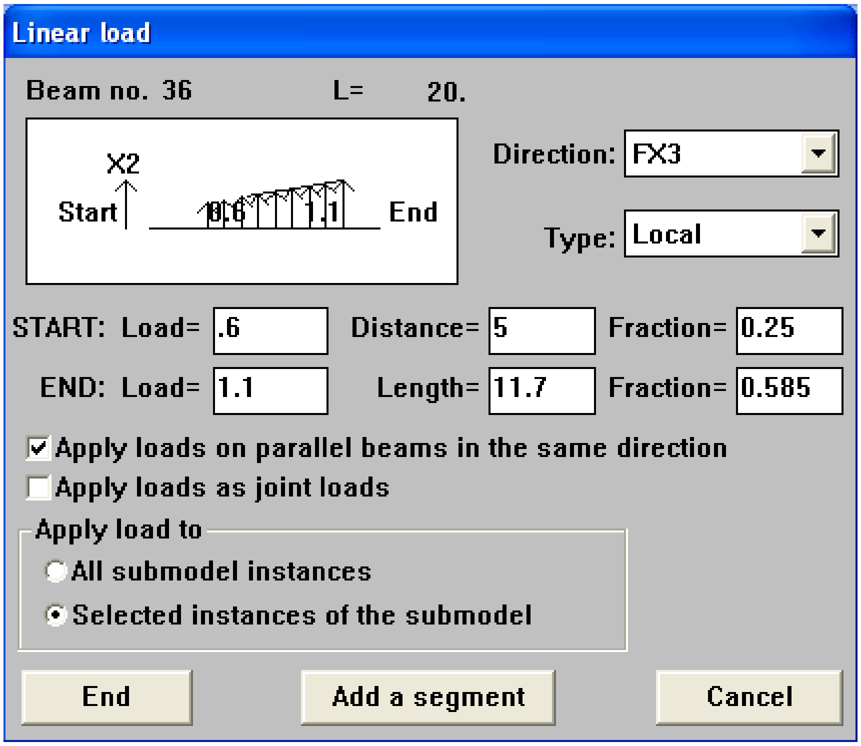
Select one of the following options:

OK	-	apply load as displayed
Cancel	-	do not apply load to the selected beams (cancel command)
Invert load direction	-	to change the sign of the load definition
Invert beam start-end	-	to create a mirror image of the loads defined
Skip confirmation	-	to cancel the display of the centre box for the remaining beams selected.

4.6.1.2 Linear

Define a general linear load made up of load 'segments'.
- more than one segment (per beam) may be defined.
- each segment is defined by specifying the location of both ends and the magnitude of the load applied.

First, select the beam to which to apply the loads using the standard Beam selection option. Then enter the load value, type and direction;



Direction / Type / Load =

Refer to Uniform loads 429.

START:

For the first segment in the command, **Distance** is measured from the start of the beam; for all following segments, **Distance** is always measured from the end of the previous ***segment.***

END:

The program assumes that the load extends to the end of the beam, i.e. **Length** is displayed with the distance to the end of the beam.

Apply loads on parallel beams

Refer to Uniform loads 430.

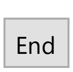
Apply as joint loads

Refer to Uniform loads 431.

Submodels

Refer to Uniform loads - apply to submodels 431.

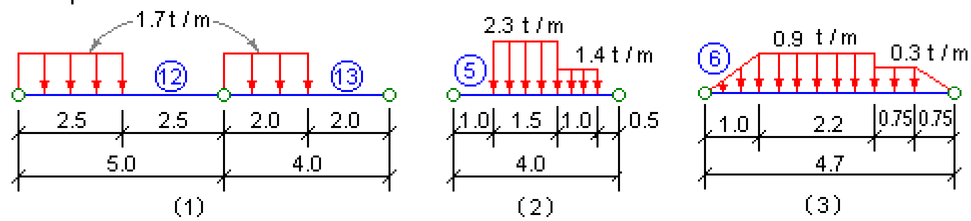
After each load has been defined, select:



to terminate the command.

Add a segment to add another load segment to the command; the distance (or fraction) is measured from the ***previous segment***.

Examples:



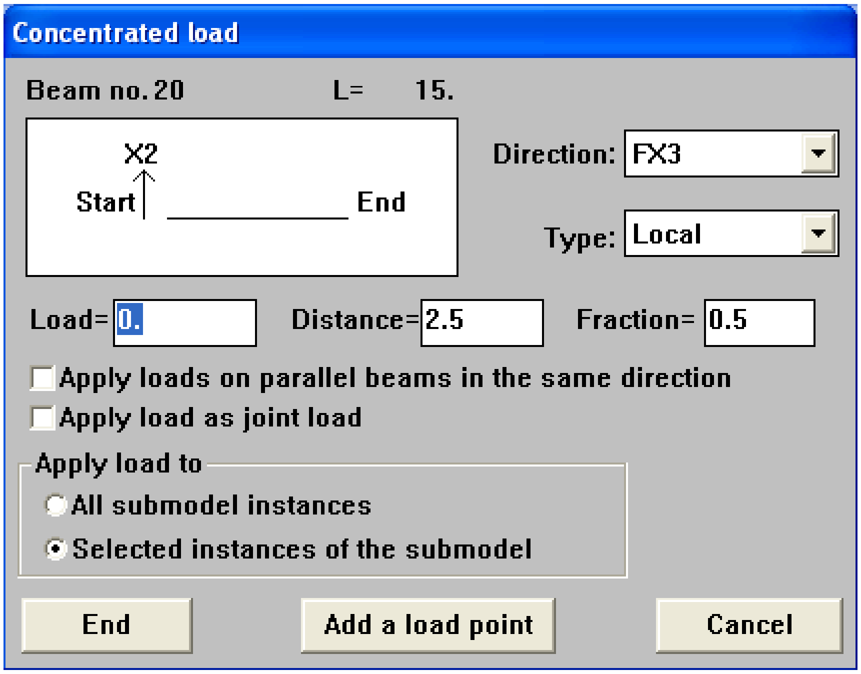
The input is: (the default values are underlined)

		Example (1)		Example (2)		Example (3)	
Segment	Loc.	Dist.	Load	Dist.	Load	Dist.	Load
1	Start	<u>0.0</u>	-1.7	1.0	-2.3	<u>0.0</u>	<u>0.0</u>
	End	2.5	<u>-1.7</u>	1.5	<u>-2.3</u>	1.0	-0.9
2	Start			<u>0.0</u>	-1.4	<u>0.0</u>	<u>-0.9</u>
	End			1.0	<u>-1.4</u>	2.2	<u>-0.9</u>
3	Start					<u>0.0</u>	-0.3
	End					0.75	<u>-0.3</u>
4	Start					<u>0.0</u>	<u>-0.3</u>
	End					<u>0.75</u>	0.0

4.6.1.3 Concentrated

Enter point loads or moments at any location along the length of the beam.

First, select the beam to which to apply the loads using the standard Beam Selection option. Then enter the load value, type, direction and location;



Define the load value and the distance from the start of the beam to the load as a length or a fraction of the beam.

Direction

Forces: select the local/global axis the load acts parallel to.

FX1	- parallel to X1 or x1 -	relevant for plane and space frames
FX2	- parallel to X2 or x2 -	relevant for plane and space frames
FX3	- parallel to X3 or x3 -	relevant for grids and space frames

Moments: select the axis that the load acts ***about***.

MX1	- about local x1 -	relevant for grids and space frames
MX2	- about local x2 -	relevant for grids and space frames
MX3	- about local x3 -	relevant for plane and space frames

Distance / fraction

The load location along the beam may be defined as an absolute distance (in the current units) or as a fraction of the beam length.

- for the first load in the command, the distance is measured from the beam start (JA).
- for all following loads in the same command, the distance is measured from the ***previous load***.

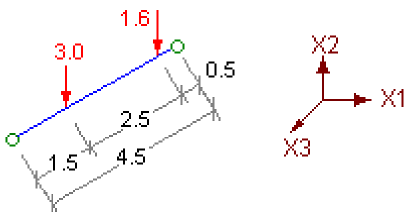
All other options

Refer to Uniform loads - Define 429.

After each load has been defined, select:

End	To terminate the command.
Add a load point	to add another concentrated load to the command; the distance (or fraction) is measured from the ***previous load point***. Five loads may be added using this option; if there are more than 6 point loads on the beam, define the remaining loads in a separate command.

Examples:



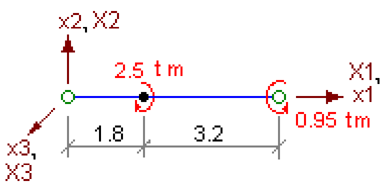
Set options to:

- Direction : **FX2**
- Type: **Global**
- Load = **-3.0**   Distance = **1.5**   Fraction = **0.33**

Select Add a load point

Set options to:

- Load = **-1.6**   Distance = **2.5**   Fraction = **0.566**



Set options to:

- Direction : **MX3**
- Type: **Local**
- Load = **-2.5**   Distance = **1.8**   Fraction = **0.36**

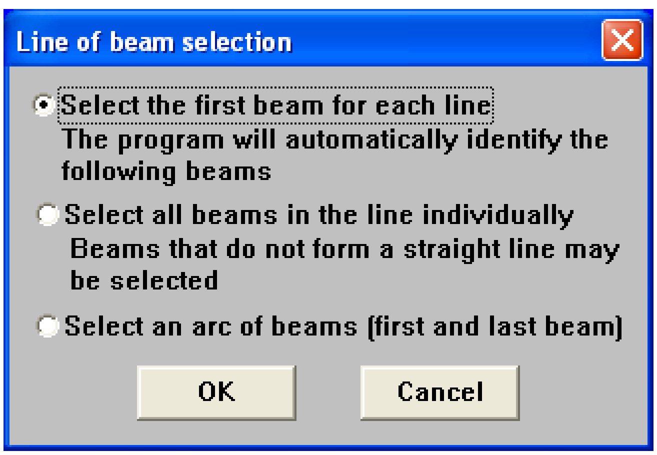
Select Add a load point

Set options to:

- Load = **0.95**   Distance = **3.2**   Fraction = **0.64**

4.6.1.4 Linear - line

The option is similar to Linear 432, but allows linear loads to be defined over a chain of continuous beams.



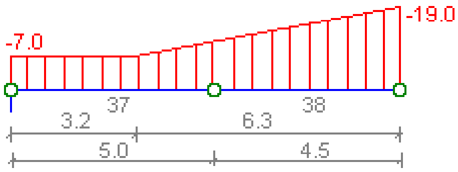
Select the first beam ...
Select the *start* beams of the lines using the standard beam selection option; the program automatically identifies the continuation beams for each start beam (beams connected to JB).

Select all beams ...
Select a chain of beams not necessarily forming a straight line but having common nodes

Select an arc ...
Select the *start* and *end* beams of an arc using the standard beam selection option; the program automatically identifies the continuation beams lying on the arc.

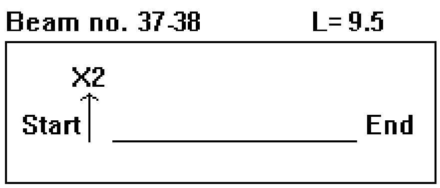
The chain is treated as a single beam and the linear load segments are defined as in Linear 432.

Example:



To define the load:

- select beam 37 - the start beam in the chain (selecting 38 along with 37 gives the same results).
- the program automatically identifies all following beams (connected to JB). Note that if you select beam 38, the program does not identify beam 37.
- the load definition box is displayed with the ***combined*** length:



- define the first load segment from 0.0 to 3.2 with Load = -7.0
- select Add a segment
- define that second load segment from 0.0 to 6.3 with Load = -7.0 at start and Load = -19.0 at the end.
- select End

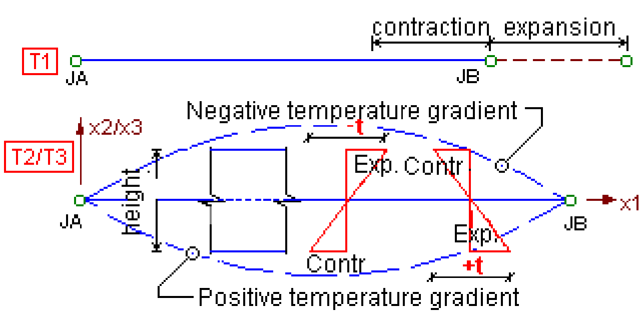
Note:

- the program divides the linear-line load into individual linear loads on each beam, i.e. to revise the load in the example, the loads on beams 37 and 38 must be revised separately. ("Undo" erases the entire command).
- a beam is considered as part of the chain if the angle between its x1 axis and the x1 axis of the previous beam is less than 5°.
- the length of the chain is limited to 80 beams (use two or more commands for 80+ beams)

4.6.1.5 Temperature

Temperature loads are always defined relative to the local coordinate axes.

- a temperature change in the x1 direction expands/contracts the beam and applies a force to the model at the beam ends.
- a temperature change in the x2 or x3 directions creates a temperature gradient across the height/ width of the beam, resulting in a curvature of the beam, similar to bending.

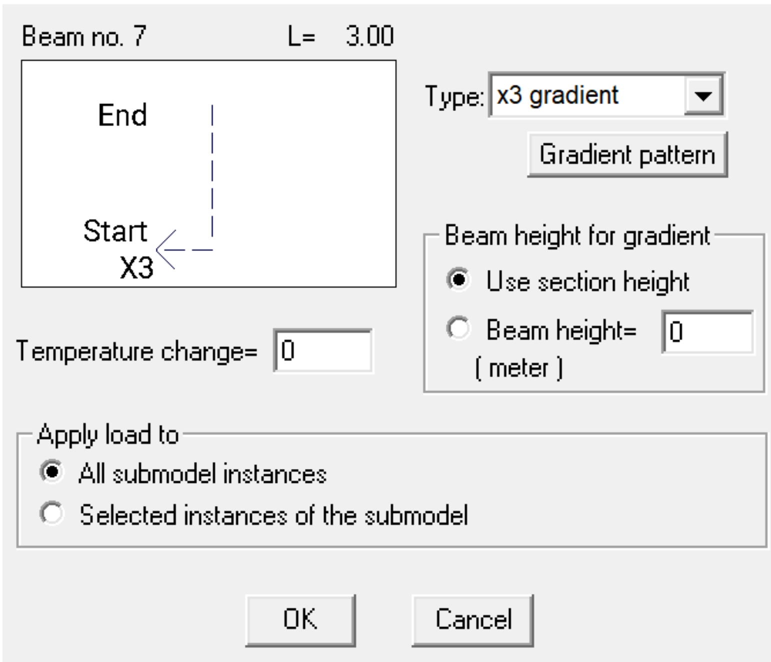


Note:

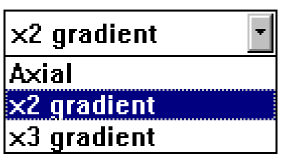
- x1 temperature loads should not be applied to 'tension/compression only' members. Temperature forces will be applied to the end nodes even if these members are 'removed' in an iteration and as a result the solution may not converge.

To define a temperature load:
- select the beams to which to apply the loads using the standard Beam Selection option.
- enter the load value, type and direction.

Temperature load



Type



Select:

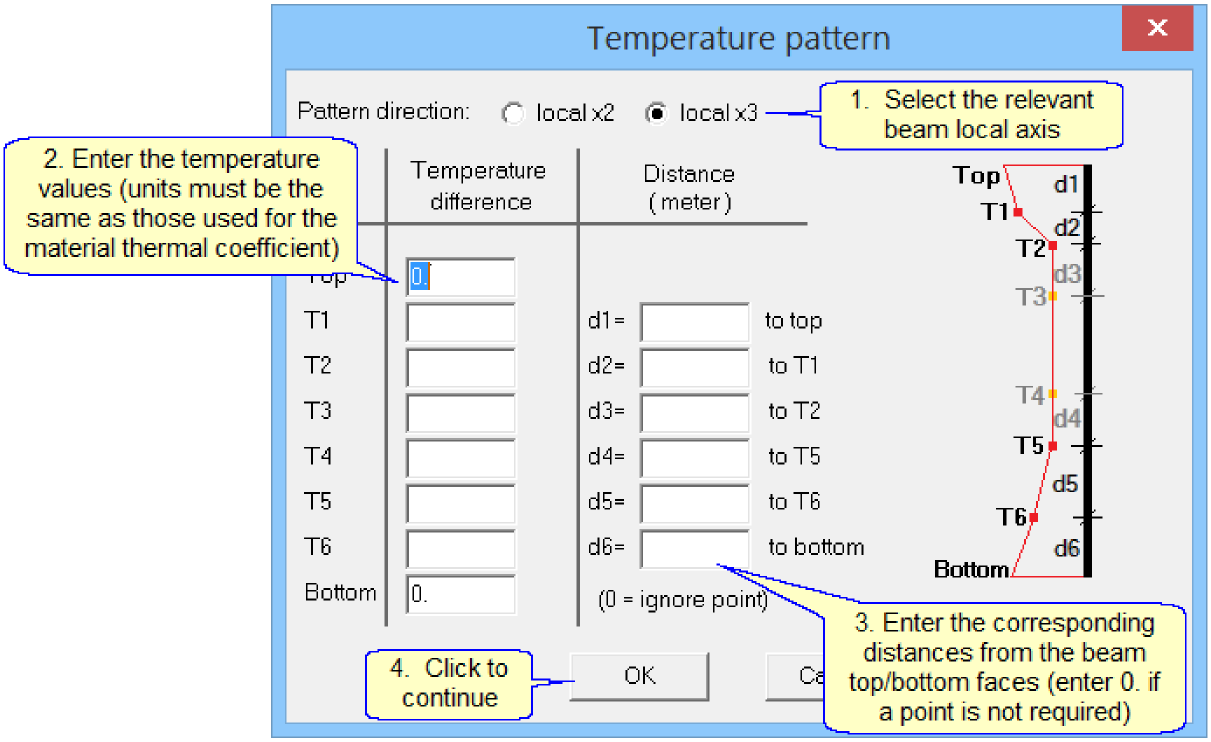
Axial - a uniform temperature change that expands/contracts the beam along its axis.

Gradient - a uniform temperature gradient across the height of the beam that produces a curvature in the beam. For non-uniform gradients, select Gradient pattern 438

Note:
- do not define an axial temperature load on tension/compression only beams. The program always applies these loads to the model, even if these beams are not active for the relevant load case.

Gradient pattern

Define a non-uniform gradient pattern over the height of the beam:



Note that **d1** is measured from the top face of the beam and that **d6** is measured from the bottom face; all other distances are measured from the adjacent points.

Temperature change

Enter the temperature difference in degrees Celsius (°C). Note the sign conventions.

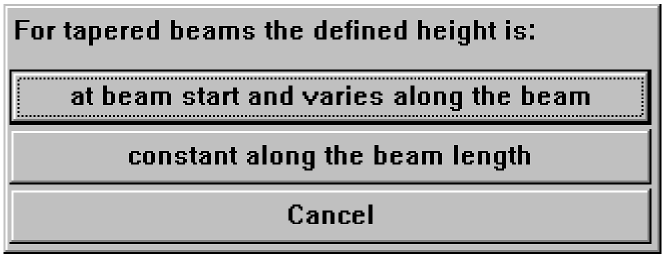
The program multiplies the temperature difference by the thermal coefficient of the material(s); therefore the temperature and the coefficient must have the same units. Check the value of the thermal coefficients and enter the difference in degrees Celsius (°C) or degrees Fahrenheit (°F) accordingly.

Beam height

The height is required in the case of a **gradient** to calculate the curvature and hence the applied loads. Two options are available:

- **Use section height -**
  The program assigns the beam height defined in the geometry. The height is determined according to the selection of the temperature type.
  For example: The selected type is X2 gradient. The program assigns the section height measurement in the local X2 direction to the beam height of the temperature gradient.
- **Beam height =**
  Insert the beam height manually

Notes:
For ***tapered*** beams:
- axial: the program takes into account the tapered section when calculating the applied nodal forces.
- gradient:



Varies: The program assumes that the height defined here is at the beam ***start*** and then varies linearly to the beam end according to the (hr/hl) ratio calculated from the tapered section dimensions defined in Beam Properties.

Constant: The program assumes that the height defined is constant throughout the beam length.

Submodels

Refer to Joint loads - apply to submodels 425.

Examples:

- Axial expansion due to temperature rise of 30°C:
  Select: **Axial**
  Enter: **Temperature change** = 30
- Temperature gradient of 25°C on height = 0.50 m. (5°C on +x3 face, 30°C on -x3 face).
  Select: **X3 gradient**
  Enter: **Temperature change** = 25 **Beam height** = 0.5

4.6.1.6 Self-weight

Self-weight defined as a Beam Load is applied as a uniformly distributed load on the beam.

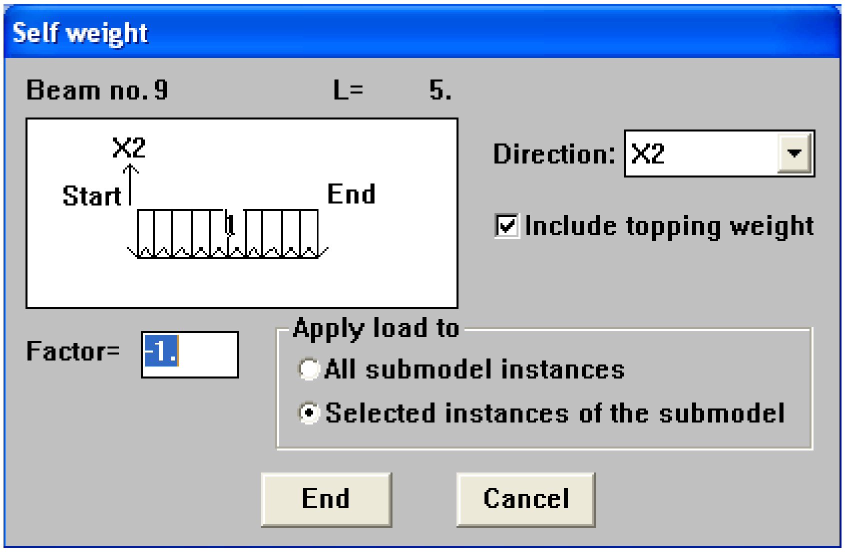
- The load is always applied in the direction of one of the global axes.
- Self-weight may be defined as acting on all beams in the model or only on selected beams.

Note:
The program automatically computes the load by multiplying the beam area by the material density:

- If the material is User defined, verify that the density was defined.
- incorrect loading result if fictitious beams are defined with an arbitrarily large area and their self-weight is applied; define the material density in these beams equal to zero or exclude them from the list of beams.
- do not define self-weight on tension/compression only beams that are sloped.(i.e. an axial component is applied) The program always applies these loads to the model, even if these beams are not active for the relevant load case.
- Self-weight of elements is applied separately as an Element load.

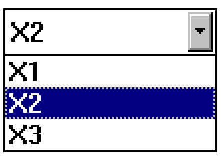
To define the self-weight:

- select the beam to which to apply the loads using the standard Beam selection option.
- enter the load factor and direction:



Direction

Select one of the **GLOBAL** directions:



Topping weight

For composite beams only:

☑ calculate the self-weight as the sum of the weights of the beam and the topping

☐ calculate and apply the self-weight of the beam only

Factor

The calculated self-weight is multiplied by the factor; the sign of the factor defines the direction of the applied load relative to the selected axis.

Submodels

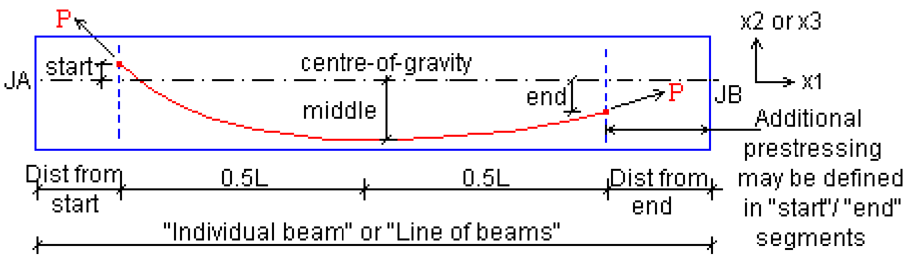
Refer to Joint loads - apply to submodels 425.

Examples:

- apply self-weight as a vertical service load:
  Direction = X2 **Factor** = -1.00
- apply 10% of the self-weight as a horizontal load:
  Direction = X1 **Factor** = 0.10

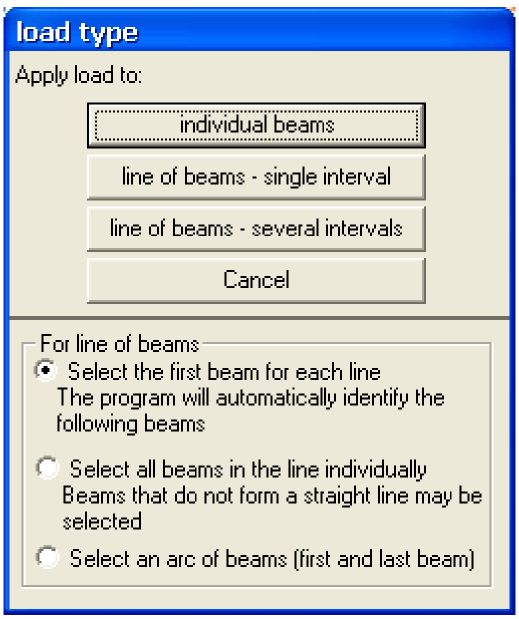
4.6.1.7 Prestress

Define a prestress force for a beam by specifying a prestress force and eccentricity. Different eccentricities may be defined at start and end points (not necessarily the beam ends) and at the mid-point between the start/end to simulate a parabolic cable. The prestress load applies a force and moments (due to the eccentricity of the load) to the end nodes of the beam.



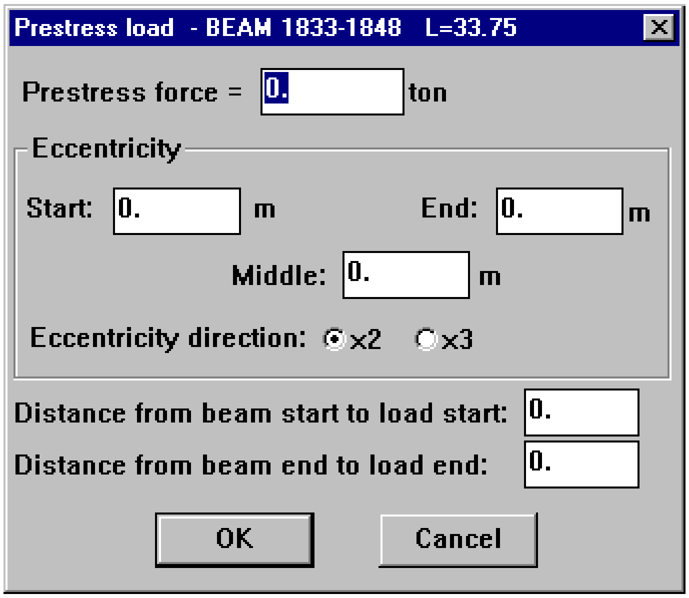
Note:

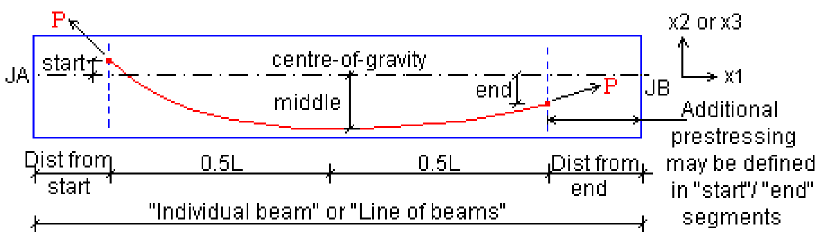
- several different prestressing segments may be applied to the same beam.
- prestressing may be defined in a line of beams, with either uniform prestressing (one interval) or several intervals.
- Submodels: prestress loads may be applied to one or more instances of the submodel. Refer to Joint loads - apply to submodels 425.



4.6.1.7.1 Individual beams

- Select beams using the standard beam selection option
- Define the prestress data:





Force

Enter the prestress force in the default force unit. The value must be positive.

Eccentricity

Enter eccentricity values in the default length unit:

- First enter the **Start** and **End** values; the program assumes that the cable is straight and writes the corresponding mid-point value in the **Middle** box. If the cable is parabolic, enter the actual mid-point eccentricity in the **Middle** box.
- Eccentricity is measured from the centre-of-gravity of the section.
- a positive eccentricity is measured in the positive direction of the local axis; in the example above, **Start** = positive, **Middle** = negative, **End** = negative.

Select the eccentricity axis (local) - x2 or x3.

Distance

The length of the individual beam or the total length of the line of beams is displayed in the title line of the dialog box.

Enter the distance from the start of the beam to the start of the prestressing and the distance from the end of the beam (or line of beams) to the end of the prestressing.

4.6.1.7.2 Line - single interval

- Select the beams:

  Select the first beam ...
  Select the *start* beams of the lines using the standard beam selection option; the program

automatically identifies the continuation beams for each start beam (beams connected to JB).

Select all beams ...
Select a chain of beams not necessarily forming a straight line but having common nodes

Select an arc ...
Select the *start* and *end* beams of an arc using the standard beam selection option; the program automatically identifies the continuation beams lying on the arc.

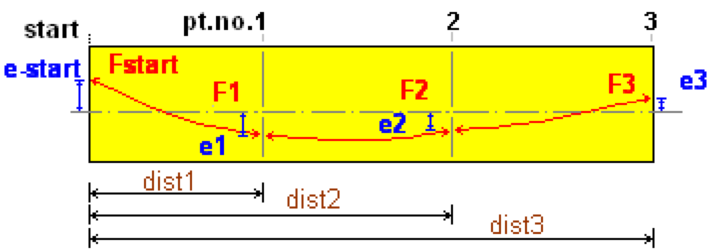
Define the prestress data; refer to Prestress - individual beams 442.

Note:
- the length of the chain is limited to 80 beams (use two or more commands for 80+ beams)

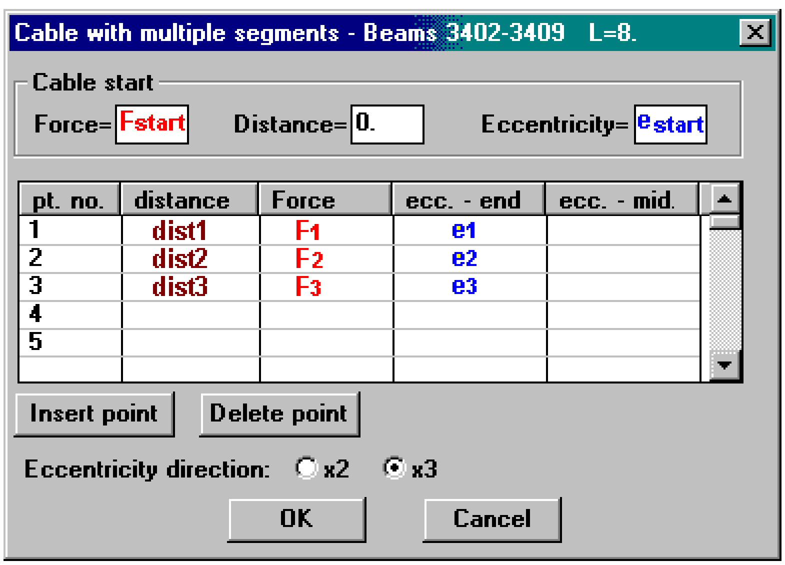
4.6.1.7.3 Line - several intervals

Define several intervals of prestressing in the selected line of beams:
- the prestressing force may be different in each interval
- the trajectory may be defined with a different parabola in each interval

Example:



Enter the following values according to the sketch above:



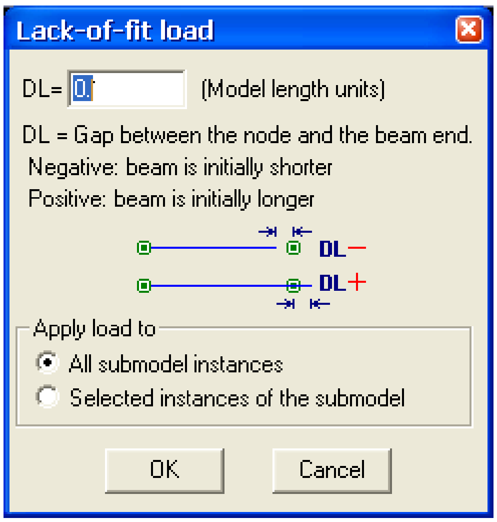
Note:
- the start and last points do not have to be at the beam ends.
- the program calculates a default value for **ecc-mid** that creates a parabola with the same slope at its start as at the end of the previous interval; the **ecc-mid** value may be revised by the user, but this introduces a 'kink' in the cable trajectory.

- to insert a new point, highlight the following line and click Insert point .
- to delete a point, highlight the line and click Delete point.

4.6.1.8 Lack-of-fit

Apply the load generated by a beam that is initially too short or too long and is stretched/squeezed to fit the distance between the end nodes:

- select the beams according to the standard beam selection option
- enter the length of the gap between the end of the beam and the end node (note the sign convention displayed in the dialog box).
- Submodels: lack-of-fit loads may be applied to one or more instances of the submodel. Refer to Joint loads - apply to submodels 425.

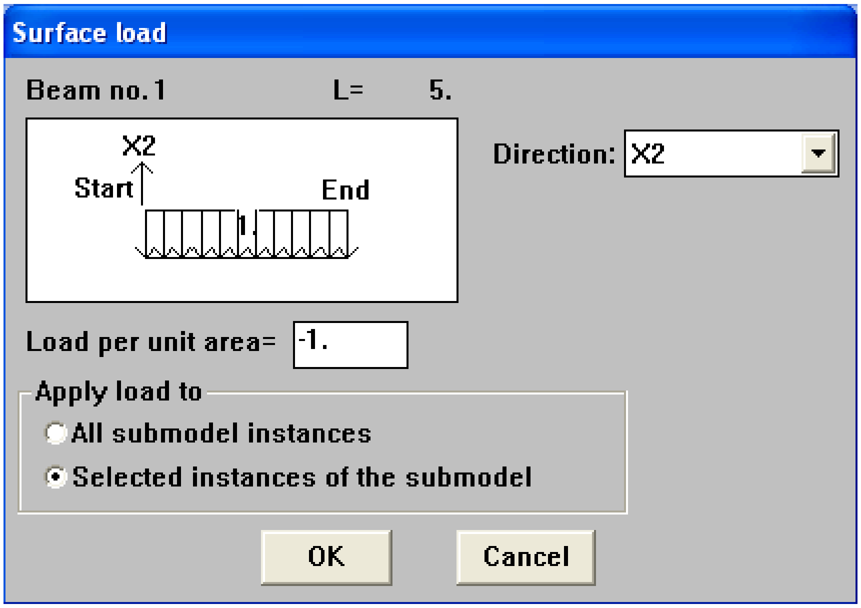


The load applied is similar to a temperature load.

4.6.1.9 Surface

Enter an area load applied to the perimeter surface area of the beam. The program applies the resulting uniform load to the entire beam length.

- select the beam to which to apply the loads using the standard Beam Selection option.
- specify the surface load value and direction;



Submodels

Refer to Joint loads - apply to submodels 425.

Note:

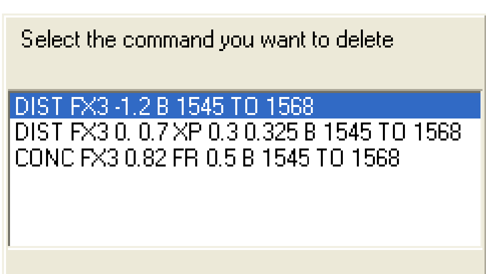
- The "Perimeter" values in the beam property tables are used for this calculation and they must be defined. **Note that Perimeter=0 and hence all surface loads will be zero in all models created prior to Version 9.00**.

4.6.2 Delete

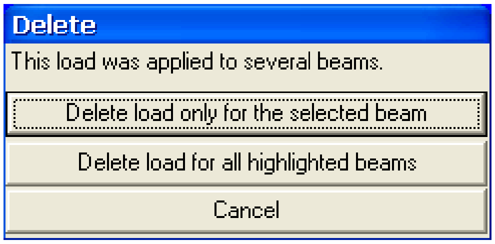
Delete all loads on beam/elements or delete selected loads:

Select a beam/element with the load to be deleted using the standard Element selection option.

- If you select [Delete a selected load] the program displays a list of loads applied to the beam; select one. For example, a beam with a uniform load, a linear load and a point load:



- If the selected load was applied to more than one beam/element in the same command, you may correct the load on all of them at the same time (the beams will be highlighted):



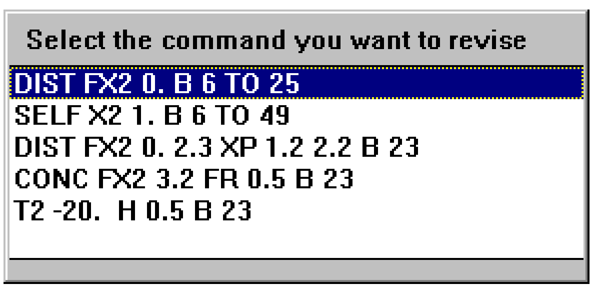
- **Delete load only for the selected beam**
  The load is deleted on the selected beam/element only
- **Delete load for all highlighted beams**
  The load is deleted on all beams/elements highlighted with the ■.

Note:

- Loads that were added to the current load case by the **Combine loads** option cannot be deleted; the original load case must be revised.

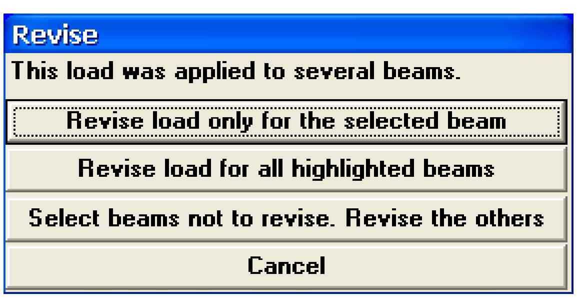
4.6.3 Revise

- Select the beam/element with the load to be revised using the standard single element selection option.
- If more than one load was defined for the selected beam/element, you must decide which one of them to revise. For example, a beam with a two distributed loads:



Select the load to be revised.

- The corresponding current load values are displayed on the screen; revise them.
- If the selected load was applied to more than one beam/element in the same command, the program highlights the beams/elements with a ■; you may correct the load on all of them or some of them at the same time:



- **Revise load only for the selected beam**
  the load is revised on the selected beam/element only
- **Revise load for all highlighted beams**
  the load is revised on all beams/elements highlighted with the ■

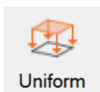
- **Select beams not to revise**
  Use the standard element selection option to select beams for which ***THE ■ WILL BE TURNED OFF***.

Note:

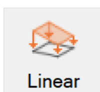
- Loads that were added to the current load case by the **Combine loads** option cannot be corrected; the original load case must be revised.

4.7 Element loads

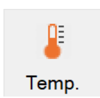
4.7.1 Define



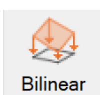
Uniform pressure 449 on the face of the element.



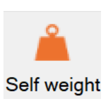
Linear pressure 450 (hydrostatic, soil pressure, etc.) applied to a group of elements that varies in one direction only.



A uniform temperature change 452 that expands/contracts the element in the plane of the element or produces a temperature gradient across the thickness of the element that produces a curvature in the element.



Bilinear pressure 453 (hydrostatic, soil pressure, etc.) applied to a group of elements that varies in two directions.

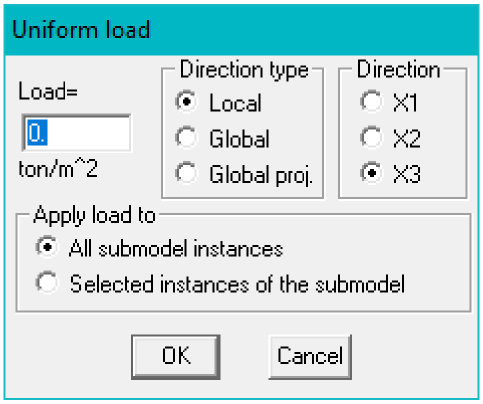


Apply the self-weight 456 of elements as a uniformly distributed element pressure.

4.7.1.1 Uniform

Use this option to define a uniform pressure on the face of the element. The pressure may be defined either in a direction parallel to one of the global coordinate axes or parallel to one of the local coordinate axes of the element.

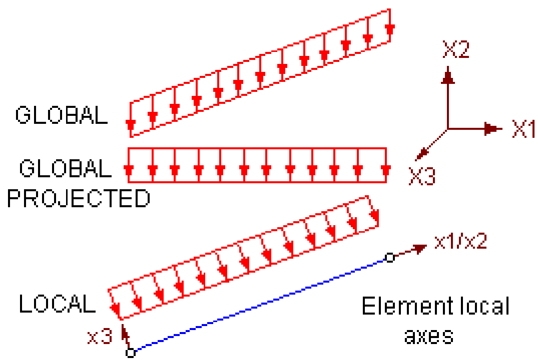
- select the element to which to apply the loads using the standard Element Selection option.
- enter the load value, type and direction:



Direction

Local : the load is parallel to the ***element local*** axis specified.
Global : the load is parallel to the ***global*** axis specified.
Global projected : the load is parallel to the axis specified, but is applied to the projected element area as projected onto the specified global axis

The following figure illustrates the **Global**, **Global projected** and **Local** element loads commands.



Load

- the sign is positive in the positive direction of the axis.
- The total load applied is the load times the element area (for all directions), or the projected element area for "Global projected" loads.

Submodels

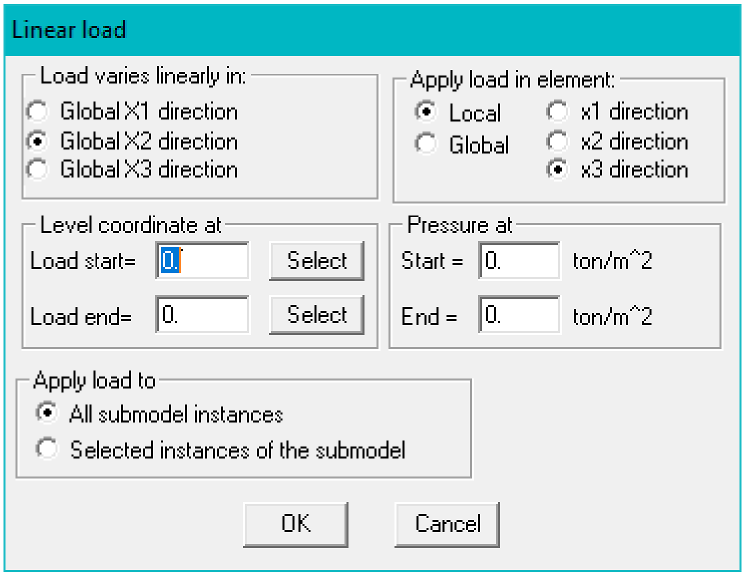
Refer to Joint loads - apply to submodels 425.

4.7.1.2 Linear

Use this option to define a linear pressure (hydrostatic, soil pressure, etc.) applied to a group of elements. The pressure varies in one direction only (refer also to the bilinear pressure 453 option)

The program calculates the total pressure on each element and applies it as a ***uniform*** pressure on the entire face of the element. The program displays the pressures as standard uniform pressures.

Define the parameters:



Load varies linearly in

The pressure can vary in any one of the global axes directions:

Note that this option defines the ***direction in which the load varies*** and not the direction in which it is applied (these are not necessarily the same - see example (b) below).

Apply load in element

The load may be applied in any local or global direction, e.g. perpendicular or parallel to the surface, vertically or horizontally, etc.

Level coordinate

Define the coordinate of the start and end of the pressure diagram. Either enter the coordinate value or click Select and select a node at the coordinate location.

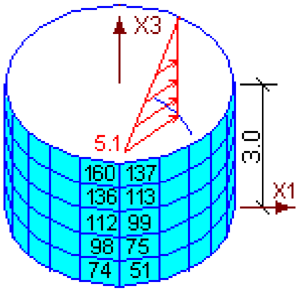
Pressure at

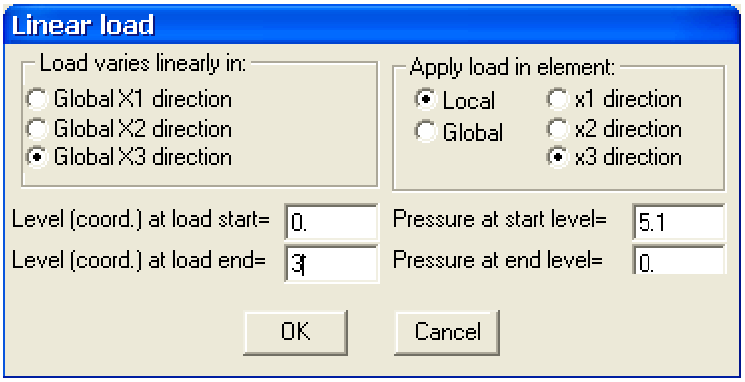
Define the value of the pressure at the start and end of the pressure diagram.

Submodels

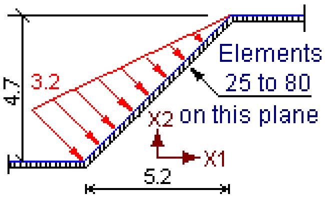
Refer to Uniform loads - apply to submodels 431.

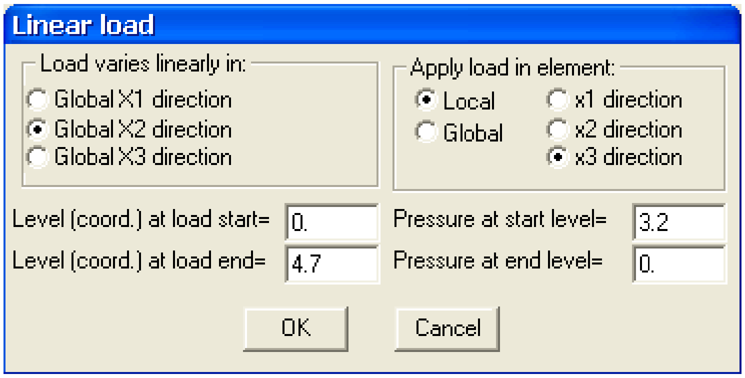
Examples:





Select all of the elements in the tank walls.





Select elements 25 to 80.

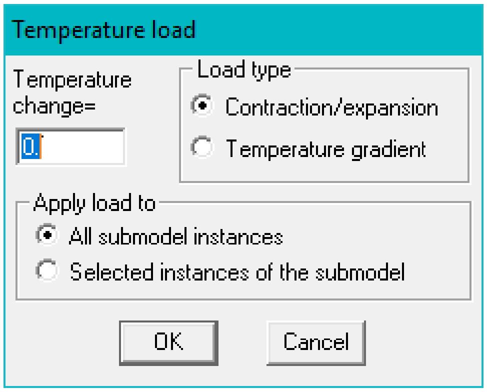
Note that the load could also have been defined as varying in the global **X1** direction from coordinate **0.0 to 5.2**

4.7.1.3 Temperature

Two types of temperature loads may be applied:

- a uniform temperature change that expands/contracts the element in the plane of the element.
- a temperature gradient across the thickness of the element that produces a curvature in the element. Enter the temperature at the +x3 face of the element less the temperature at the -x3 face.

Define the parameters:



Load type

Select:

Contraction / expansion
a uniform temperature change that expands/contracts the element in the plane of the element.

Temperature gradient
a gradient across the thickness of the element that produces a curvature in the element. Enter the temperature at the +x3 face of the element less the temperature at the -x3 face.

Temperature change

Enter the temperature ***difference***.

The program multiplies the temperature difference by the thermal coefficient of the material(s); therefore the temperature and the coefficient must have the same units. Check the value of the thermal coefficients and enter the difference in degrees Celsius (°C) or degrees Fahrenheit (°F) accordingly.

Submodels

Refer to Uniform loads - apply to submodels 431.

Examples:

- Expansion / contraction:
  A plate model is heated uniformly by 27 °C.
  Enter: **Temperature change = 27**
- Gradient:
  In a dome shell structure, the interior (-x3) temperature is 18°C and the exterior (+x3) temperature is -15°C;
  Enter: **Temperature change = -33**

Note:

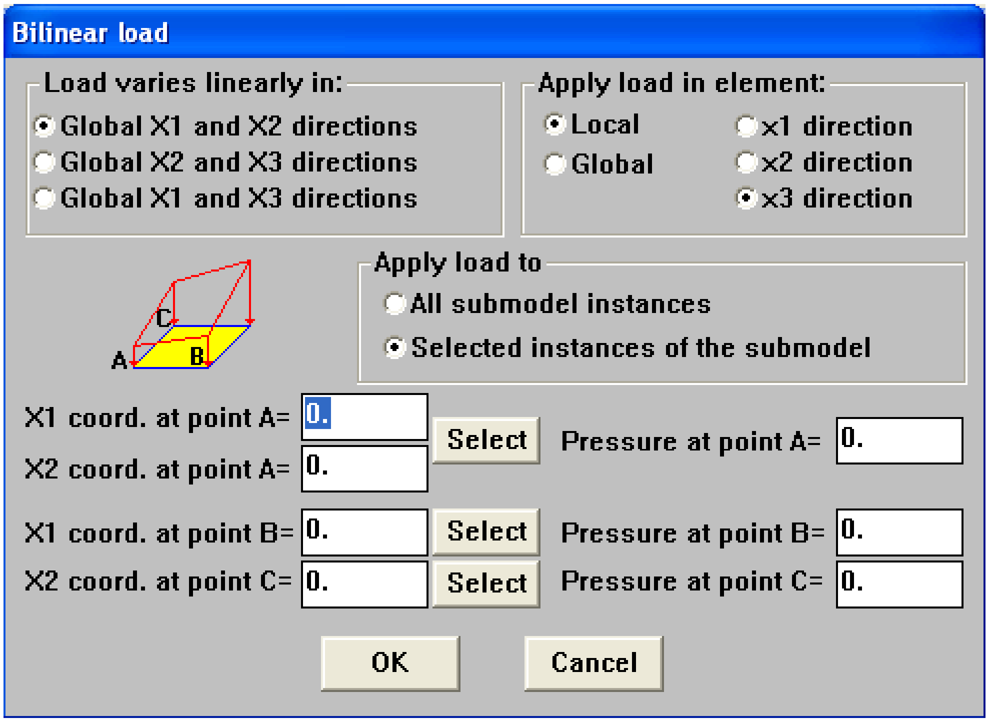
- Temperature loads applied to orthotropic elements are calculated using Ex, αx in the local x1 direction and Ey, αy in the perpendicular direction.

4.7.1.4 Bilinear

Define a linear pressure (hydrostatic, soil pressure, etc.) applied to a group of elements. The pressure varies in two directions (refer also to the regular pressure 450 option to define pressure that varies in one direction only)

The program calculates the total pressure on each element and applies it as a ***uniform*** pressure on the entire face of the element. The program displays the pressures as standard uniform pressures.

Define the parameters:



Load varies in

The pressure can vary in the general direction of any one of the global planes.

Note that this option defines the ***direction in which the load varies*** and not the direction in which it is applied (these are not necessarily the same.

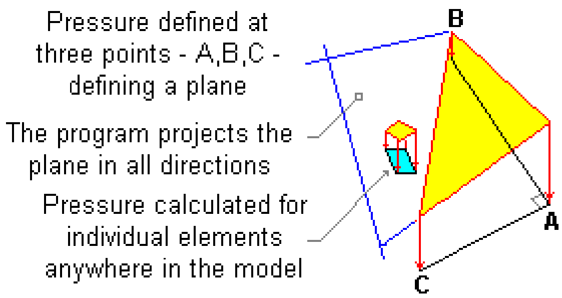
Apply load in

The load may be applied in any local or global direction, e.g. perpendicular or parallel to the surface, vertically or horizontally, etc.

Pressure

Define the pressure values and the coordinates.

The pressure diagram must be planar, therefore it is sufficient to define three corners and their location; the program interpolates the plane over the surface of all elements selected.



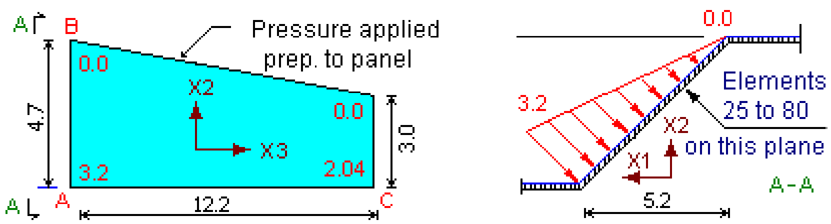
Note:

- **lines AB and AC must be perpendicular**.
- point **A** must be the **middle point** of the chain C-A-B

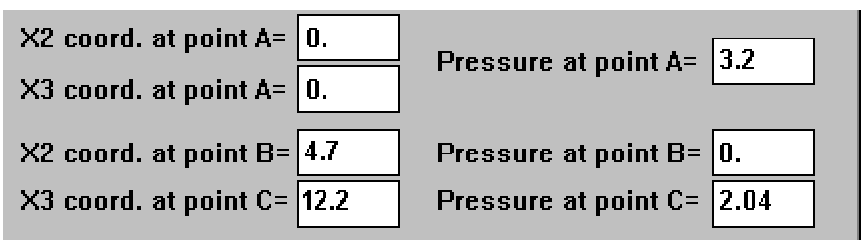
Submodels

Refer to Uniform loads - apply to submodels 431.

Example:



- Select **Global X2 and X3 direction**
- Define the 3 points and the corresponding pressure values:

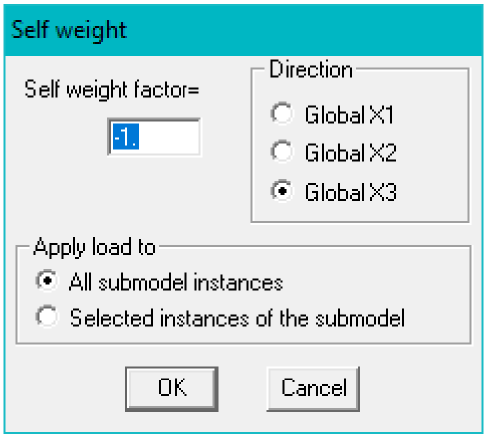


X2 coord. at point A=	0.	Pressure at point A=	3.2
X3 coord. at point A=	0.		
X2 coord. at point B=	4.7	Pressure at point B=	0.
X3 coord. at point C=	12.2	Pressure at point C=	2.04

- Select elements 25 to 80

4.7.1.5 Self-weight

Apply the self-weight as a uniformly distributed element pressure.



Direction

The self-weight can be applied in any ***global*** direction.

Factor

The self-weight is calculated as the element thickness multiplied by the material density. This value may be multiplied by a factor. If a negative factor is defined, the load is applied in the negative direction of the global axis selected.

If the material is ***User-defined***, verify that the density was defined (a warning is displayed).

Submodels

Refer to Uniform loads - apply to submodels 431.

Note:
- self-weight can also be applied to all elements in the model. Refer to Self weight 421
- Select the elements using the standard Element Selection option.

Examples:
- apply self-weight as a vertical load:
  Global X2
  Self weight factor = 1.00
- apply 10% of the self-weight as a horizontal load:
  Global X1
  Self weight factor = 0.1

4.7.1.6 Wall

Apply the self weight of a wall. The self-weight is calculated as the wall area x thickness x material density:
- The load is a applied to the segment corner nodes.
- If the material is ***User-defined***, verify that the density was defined (a warning is displayed).
- this load cannot be revised; delete the load and define again.

Direction

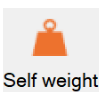
The self-weight can be applied in any ***global*** direction.

Factor

The self-weight value may be multiplied by a factor. If a negative factor is defined, the load is applied in the negative direction of the global axis selected.

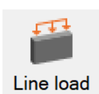
4.8 Wall loads

4.8.1 Define



Apply the self-weight 458 of a wall.

A uniform area load 459 on a face of a wall.

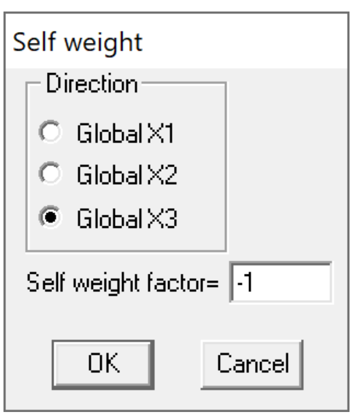


A uniform line load 459 on a the top level of the wall.

4.8.1.1 Self-Weight

Apply the self weight of a wall. The self-weight is calculated as the wall area x thickness x material density:

- The load is a applied to the segment corner nodes.
- If the material is ***User-defined***, verify that the density was defined (a warning is displayed).
- this load cannot be revised; delete the load and define again.



Direction

The self-weight can be applied in any ***global*** direction.

Factor

The self-weight value may be multiplied by a factor. If a negative factor is defined, the load is applied in the negative direction of the global axis selected.

Examples:

Assuming that X2 is the height axes -

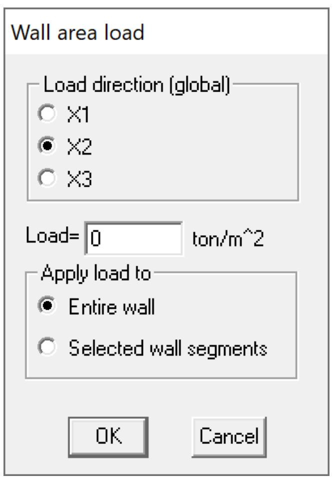
- apply self-weight as a vertical load:
  Global X2
  Self weight factor = 1.00
- apply 10% of the self-weight as a horizontal load:
  Global X1
  Self weight factor = 0.1

Note:

- self-weight can also be applied to all elements in the model. Refer to Self weight 421

4.8.1.2 Area load

Use this option to define a uniform pressure on a face of a wall segment.
The pressure may be defined in a direction parallel to one of the global coordinate axes.



Load direction

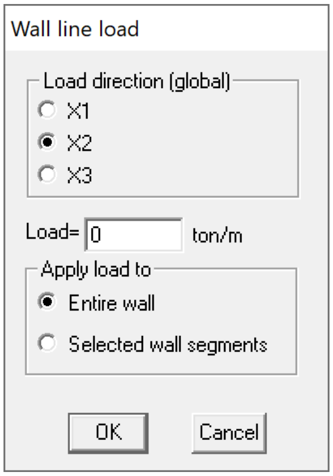
Select one of the global coordinate axes. The load will be defined parallel to the selected direction.

Apply load

Entire wall	The load will be applied to all the wall segments.
Selected wall segments	The load will be applied to selected wall segments.

4.8.1.3 Line load

Use this option to define a uniform line load on a the top level of the wall segments.
The line load may be defined in a direction parallel to one of the global coordinate axes.



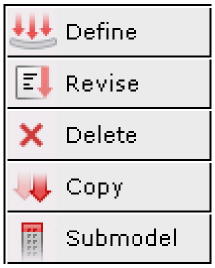
Load direction

Select one of the global coordinate axes. The load will be defined parallel to the selected direction.

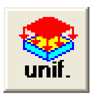
Apply load

Entire wall	The load will be applied to all the wall segments.
Selected wall segments	The load will be applied to selected wall segments.

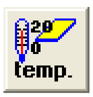
4.9 Slab loads



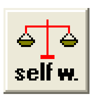
4.9.1 Define



Uniform pressure 461 on the face of the slab.



Apply a uniform temperature change that
expands/contracts a slab in its plane, or -
creates a temperature gradient along the height of the slab, creating a curvature.
Refer to Elements - temperature load 452

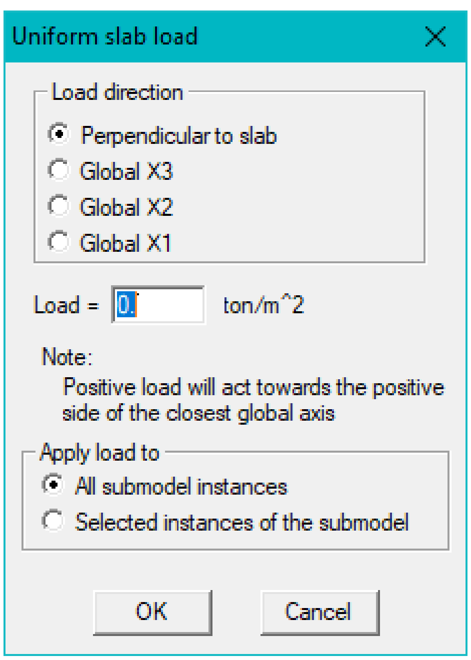


Apply the self-weight of the slab as a uniformly distributed pressure.
Refer to Elements - self-weight load 456

4.9.1.1 Uniform

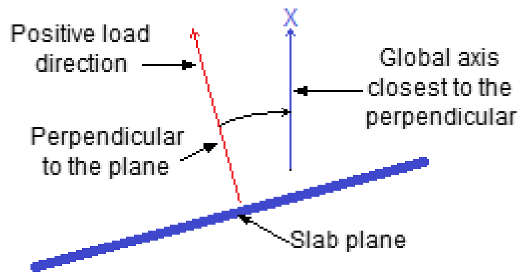
Use this option to define a uniform pressure on the face of the slab. The pressure may be defined either in a direction perpendicular to the slab face or parallel to one of the global coordinate axes.

- select the slab to which to apply the loads using the standard Slab Selection option.
- enter the load value and direction:



Direction

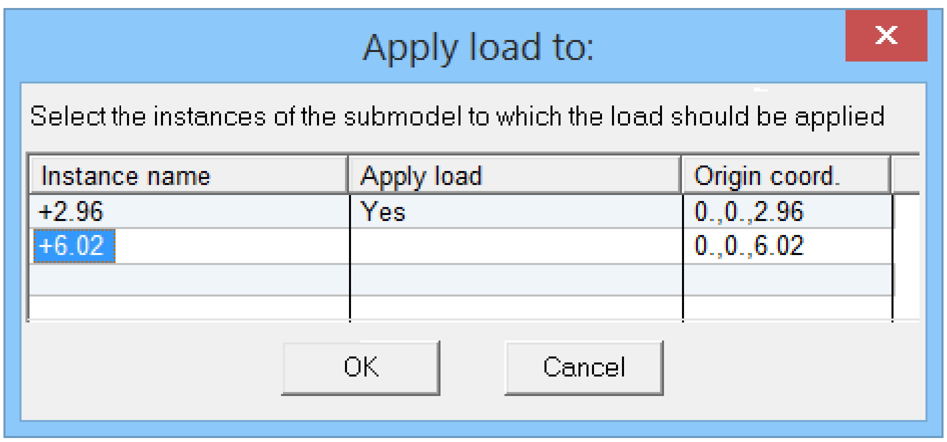
Perpendicular to slab : the load is perpendicular to the slab plane. A positive load is in the positive direction of the global axes closest to the perpendicular to the slab plane:



Global X1/X2/X3 : the load is parallel to the ***global*** axis specified (positive load in the positive direction os the axis).

Submodels

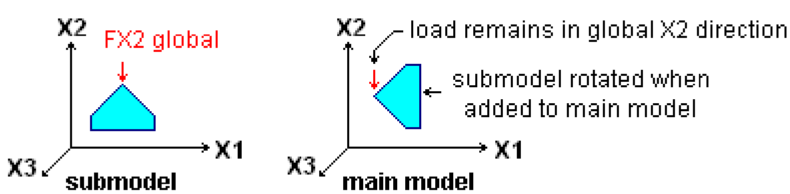
This option is displayed only when a submodel is currently displayed and there is more than one instance attached.



Click on a line to toggle the **Yes/No** option.

Note:

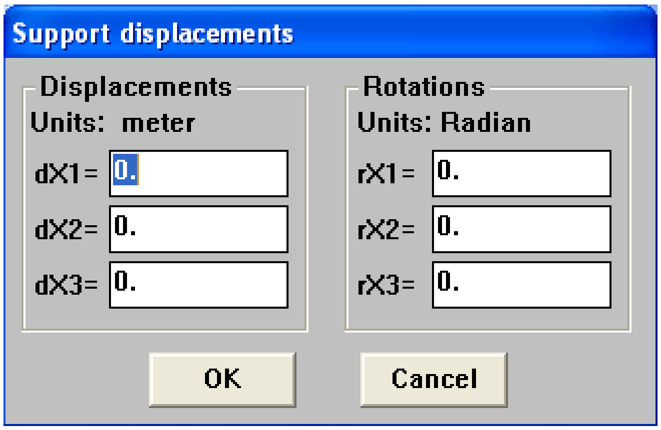
- if the submodel joint loads are defined relative to the global coordinate system the program uses the ***Main model*** system, not the submodel system, when applying them. For example:



Loads defined relative to a local system are not rotated; define a local system parallel to the submodel global system (**Define using 3 nodes**) to avoid rotating the loads in the situation described above.

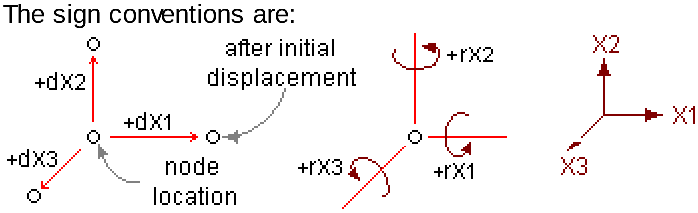
4.10 Support displacements

Support displacements may be entered in the direction of any degree-of-freedom, including rotation. Note that the nodes must be restrained in the same degrees-of-freedom as the defined displacements.



- **dx1, dx2, dx3** = initial translation in the direction of the global X1, X2 and X3 axes respectively.
- **rx1, rx2, rx3** = initial rotation about the global X1, X2 and X3 axes, respectively (radians)

The sign conventions are:



Example:

- Initial settlement in the -X2 axis direction = 2.00 mm.
- Initial rotation about the X3 axis (counter-clockwise) =0.04 radian

Specify: **dx2 = - 0.002** **rx3 = 0.040**

Select the support locations using the standard Node Selection option. Only nodes with restraints may be selected.

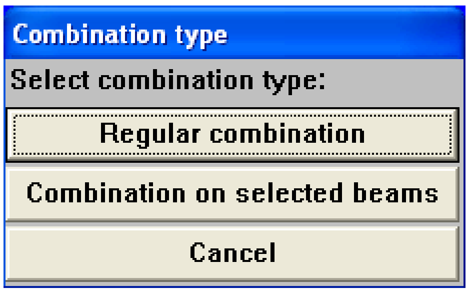
4.11 Combine loads

Use this option to:

- define a load case as a combination of existing load cases
- add an existing load case into the current load case, in addition to other loads.

Note:

- ***unless the "Combination on selected beams" options is required, it is more convenient to create the combinations in the load Combinations*** 534 ***option***.
- all load cases added to a combination may be multiplied by a factor.
- combinations may also be defined in the Results module and then copied back into the Loads module as new load cases. Refer to Copy a load case 531. This very convenient when there are a large number of combined load cases.



Regular combination

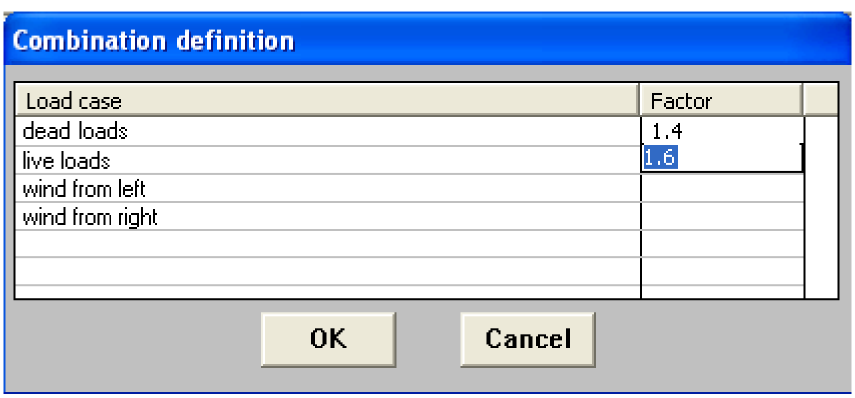
Select load cases and define factors. ***All*** loads in the selected cases are added to the combination.

Combination on selected beams

Select load cases, define factors and select beams. Only ***beam*** loads on ***selected*** beams are added to the combination, i.e. joint loads, element loads, etc. and beam loads on beams not selected are not added to the combination.

For all options:

Select a load case and define factors:



- move the into the correct cell in the "**Factor**" column and click the mouse
- type in the load factor
- repeat for other load cases and click OK to complete.

In the above example, 1.4 * dead load +1.6 *live load + 0.0 * wind loads will be added to the current load case.

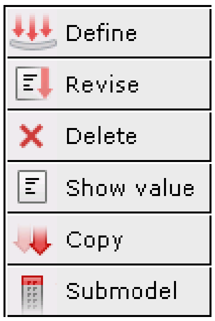
Note:

- It is strongly recommended that combinations be calculated ***after*** the solution rather than when solving the model as results can be obtained for new or revised combinations without solving the model again, however -
- The rules of superposition do not apply for models with non-linear elements. Therefore, load combinations for models with P-Delta, tension/compression only members, unidirectional springs, etc, ***must be defined here or in the combinations*** [534] ***option***, rather than after the solution.
- combinations defined in the loads combination [534] option can be calculated either during the solution or in the Results module.
- combinations may also be defined in the Results module and then copied back into the Loads module as new load cases. Refer to Copy a load case [531]. This very convenient when there are a large number of combined load cases.
- if a load case in a combination is updated, then the combination is updated automatically.
- loads that were defined by a Combination cannot be corrected using the "Revise" option of Joint Loads, Beam Loads, etc. The original load case must be revised.
- a combination cannot include a load case which includes a combination.

4.12 Global loads

Load locations may be defined relative to the global coordinate system. The program locates the nodes and elements surrounding the "global loads" and converts the loads to equivalent joint, beam or element loads as specified by the user. Loads outside the model limits are ignored in certain cases (refer to Method of Application).

This option is useful in models characterized by load patterns which do not coincide exactly with the nodes or elements, such as bridges.

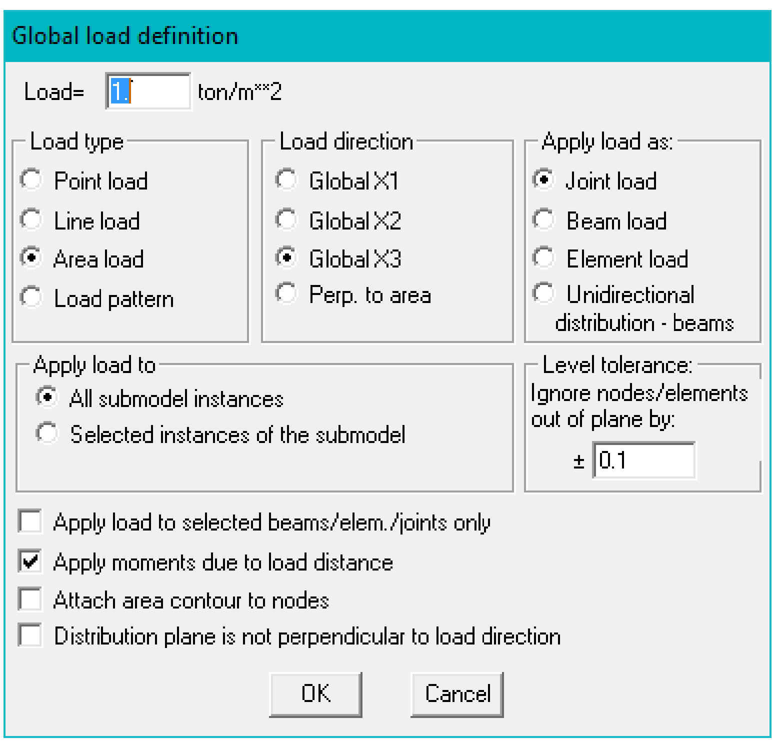


Note:

- Global Loads can be applied to any plane of beams or elements, including planes not parallel to a global plane.

4.12.1 Define

Define the global load values:



for **Load pattern** only:

Load

The value of the area / point load to be applied; a positive value is applied in the positive direction of the AXIS.

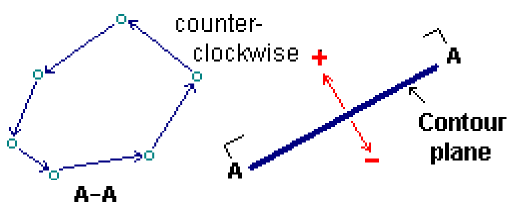
Load type

For an explanation on how to define the load location, refer to :
Point load 470
Line load 470
Area load 472
Load pattern 473

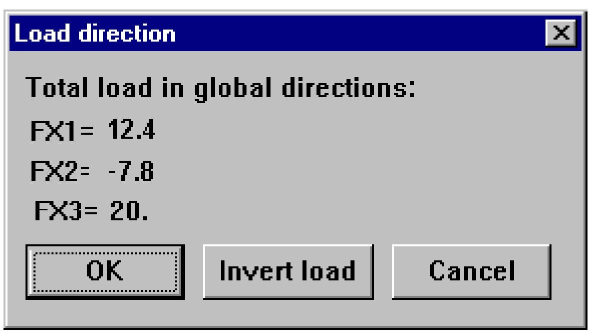
Load direction

Global loads are applied either in global axis directions or perpendicular to the surface of the plane.

For loads applied perpendicular to the plane, the sign convention is determined by the direction that the contour is defined:



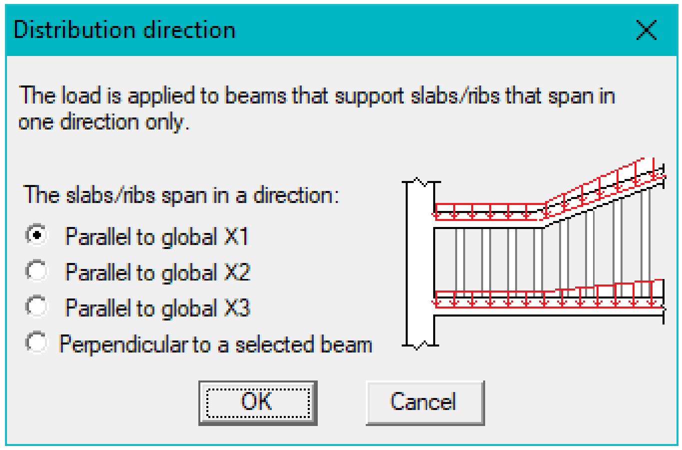
The program then calculates the global components of the total area load and asks the user to confirm or reverse the direction:

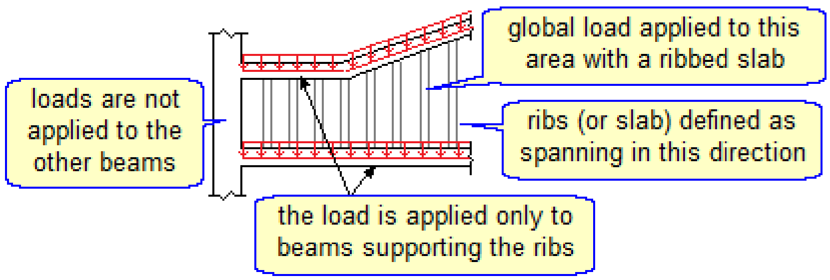


Apply load as

Global loads are converted to joint loads, beam loads or element loads:

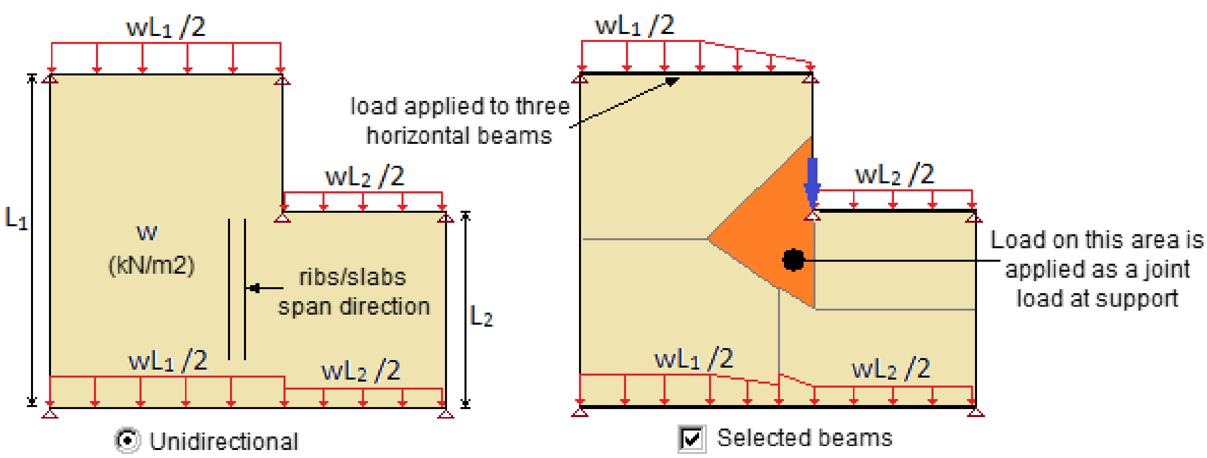
Joint	specifies that the load are converted to joint loads.
Beam	specifies that the load is applied as beam loads to the surrounding beams; part of the loads outside the model boundary is applied as joint loads by the program in certain cases. Refer to Method of application
Element	specifies that the load is applied as an element pressure load on the adjacent elements; loads outside the model boundary are ignored. Refer to Method of application.
Beams - Unidirectional	loads applied to beams can be distributed only to beams that span in one selected direction:





Note:

- Global loads are applied by default to ***all*** adjacent beams/elements. To apply the global loads to selected beams/elements only, use **Apply loads to selected beams** or **Unidirectional.** Note the difference between the two options. For example, a slab area bounded on all sides by beams:



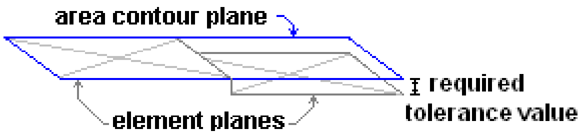
- Global loads applied the dummy beams/elements that are not connected to the model (i.e. that are connected only to other dummy beams/elements) are not applied to the model.

Apply at level

Global loads are applied only nodes/beams/elements lying on the contour plane +/- a tolerance value. The tolerance value is by default set to 0.01 to allow for minor inaccuracies in the contour coordinate definition.

Example:

The global area load is to be applied to the two adjacent parallel planes:



Apply loads to selected

Global loads - point, area or pattern - may be applied to selected beams, elements or nodes only.

- select beams or elements using the standard element selection option, or nodes using the standard Node selection option.
- beam loads:see the note above for an explanation on the difference between **Apply loads to selected beams only** or **Unidirectional**
- for the method of application to selected beams/elements, refer to Method of Application.

Factor

Factor by which all of the loads in the group are multiplied when applied.

Angle

Angle of rotation of entire load group. When the load is applied, it may be rotated about the (0,0) point of the load group, where a positive angle is counterclockwise.

Apply moments due to distance

Applicable only if the global loads are applied as joint loads:

☑ Apply forces and moments at the nodes, as explained in Method of Application

☐ Apply the forces only at the nodes.

Attach area contour to nodes

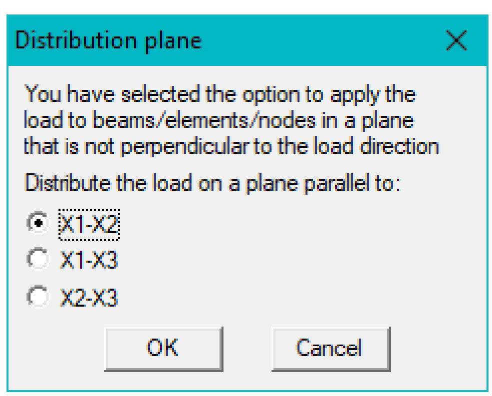
For area loads only - contour defined by nodes:

☑ The program remembers the nodes and recalculates the load area if the nodes are moved.

☐ The program translates the load area to coordinates and the area is not updated if the contour nodes are moved.

Distribution plane is not perpendicular to the load direction

The 'distribution plane' is the plane on which the program searches for nodes/beams/elements to which the global load is applied.



By default, the program assumes that the distribution surface is perpendicular to the load direction.Check ☑ for this option if this is not the case and select the direction of the distribution plane:

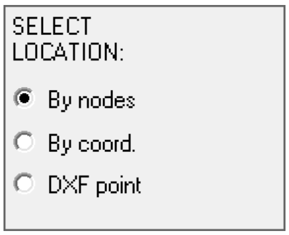
For example: vehicle braking
loads applied parallel to the plane.
of the bridge

Submodels

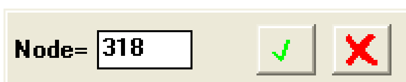
Refer to Joint loads - apply to submodels 425.

4.12.1.1 Point loads

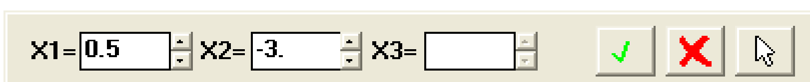
Global point loads may be defined at a node or at a coordinate; select the method in the side menu:



By nodes
highlight and select any node in the model.



By coord.
Specify the global coordinate of the concentrated load on the global plane perpendicular to the load direction:

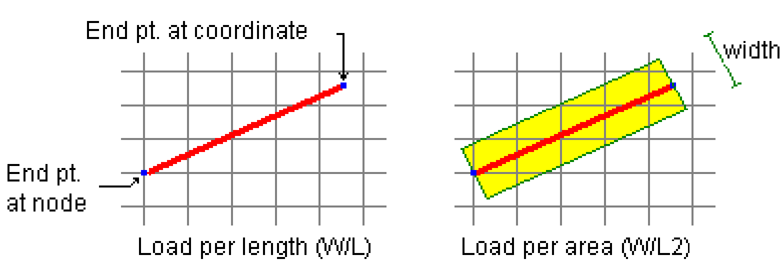


DXF point
Select points at the ends of lines of a DXF file that is displayed in the background. For more info refer to DXF Drawing 549.

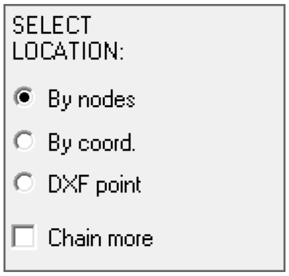
For an example on the application of global point loads, refer to Global Area Loads 472.

4.12.1.2 Line load

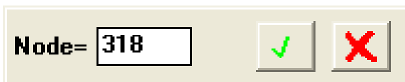
Global line loads are defined by selecting two points on the plane. The load may be defined as a line load (load per length) or as an area load with a load width (load per area):



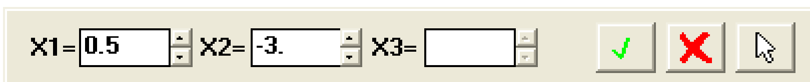
The two points may be defined at a node or at a coordinate; select the method in the side menu:



By nodes
highlight and select any node in the model.

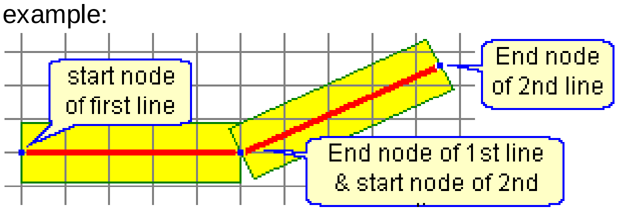


By coord.
Specify the global coordinate:

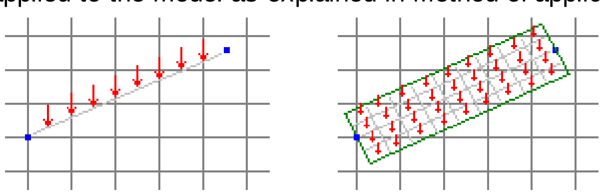


DXF point
Select points at the ends of lines of a DXF file that is displayed in the background. For more info refer to DXF Drawing549.

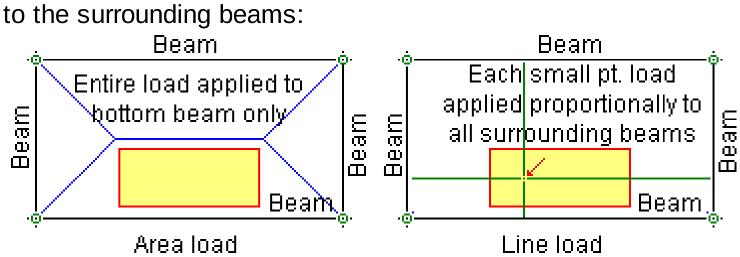
Chain more
Create a chain of line loads; the end node of the previous line is the start node of the next line. For example:



Line loads are converted by the program to a series of global point loads and these point loads are applied to the model as explained in Method of application.

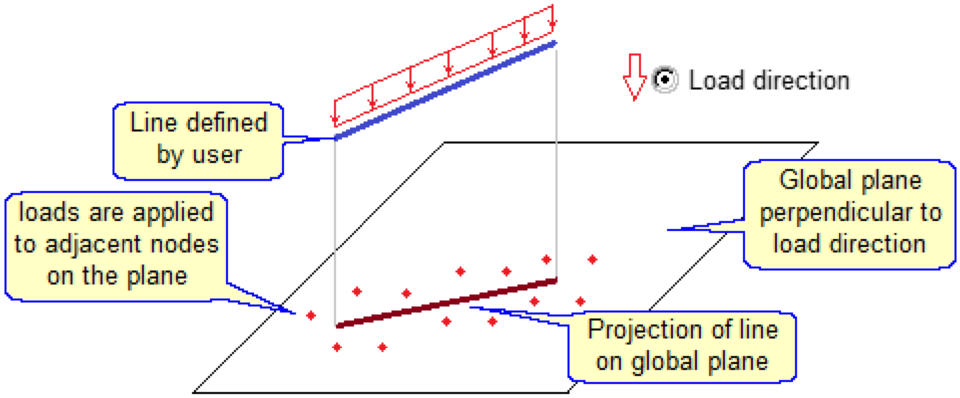


Therefore, a line load 'per area' and the identical area load applied as beam loads are applied differently to the surrounding beams:



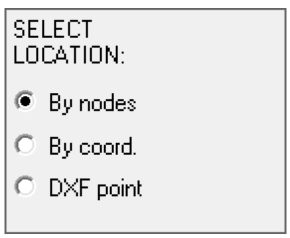
Note:
- the nodes/coordinates that form the line must lie on the same plane, not necessarily perpendicular to the selected load direction. However the load will still be applied to nodes/beams on the nearest plane

perpendicular to the selected global direction. For example, a line load applied as joint loads:



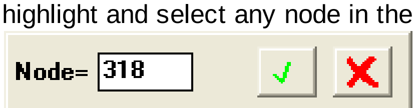
4.12.1.3 Area loads

Area loads are applied to a polygon area defined by a planar contour connecting coordinates and/or nodes (not necessarily parallel to a global plane). The contour corners may be defined at a node or at a coordinate; select the method in the side menu:

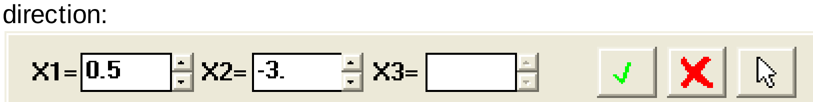


A different option may be used for each corner:

By nodes
highlight and select any node in the model.



By coord.
Specify the global coordinate of the area load corner on the global plane perpendicular to the load direction:



This option is not available if ☑ **Attach area contour to nodes** was specified in the previous menu.

DXF Point
Select points at the ends of lines of a DXF file that is displayed in the background. For more info refer to DXF Drawing 549.

Double-click the last corner or click the End button to end the definition.

Note:

- the contour nodes must all lie on the same plane, not necessarily parallel to a global plane
- the uniform load may be applied in any global direction or perpendicular to the surface of the plane. Refer to Global load - direction 467 for the sign conventions when the load is applied perpendicular to the surface
- For area loads applied as beam loads:

There are inaccuracies in the algorithm. The program calculates the total load defined and the total load applied. A warning is displayed if there is a discrepancy greater than 3% of the defined load.

This situation may arise in the following cases:

- two beams do not intersect at a node. If a "level" is not defined in a space model, it is sufficient that the projections of two beams intersect (Figure a).
- two beams, or their projections, have a common segment (Figure b).

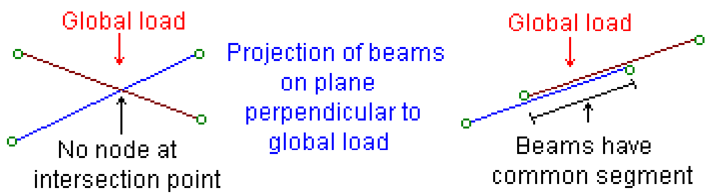
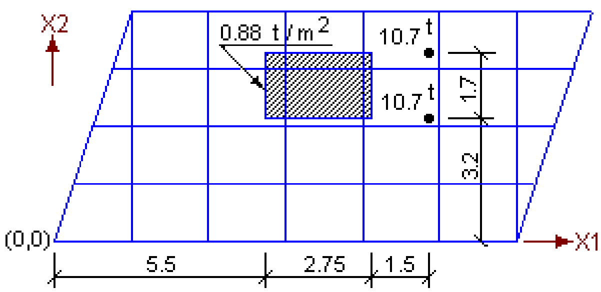


Figure a   Figure b

In rare cases the algorithm may fail when global area loads are applied to very complicated models.

Example:

Apply the global POINT and AREA loads as displayed in the following figure:

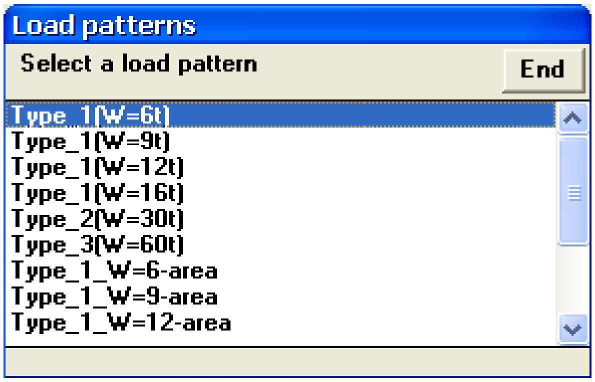


- specify:
  Point load
  Global X3
  Load = -10.7
- move to **X=9.75 , Y=3.2** and click the mouse; repeat for coordinates **X=9.75 , Y=4.9**
- specify:
  Area load
  Global X3
  Load = -0.88
- move to **X=5.5 , Y=3.2** and click the mouse to select the bottom-left corner.
- repeat for **X=8.25/Y=3.2**, **X=8.25/Y=4.9, X=5.5/Y=4.9**
- click the first point to close the contour

4.12.1.4 Pattern loads

Global loads may be stored in an ASCII format file and recalled directly into a load case. This option is useful for applying vehicle loads to a bridge model. The load patterns are stored in the file **PATTERN.DAT** 476.

The program displays a list of the load patterns stored in the file. For example:



Select one, then move the to the location of the pattern origin on the global plane.

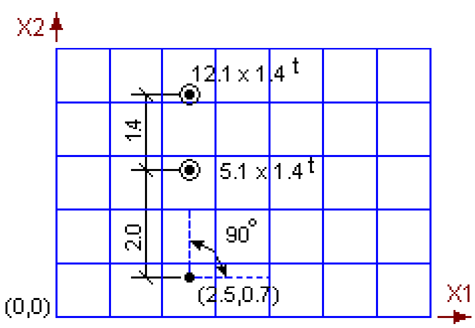
Note:

- Each load pattern is defined on an X-Y plane and the loads are referenced to an arbitrary zero coordinate. When the pattern is recalled into a load case, the location of the zero coordinate on the model is selected at a node or a coordinate.

  In space models, when the load is applied to a plane parallel to the:
  - X1-X2 plane: X corresponds to X1, Y corresponds to X2
  - X2-X3 plane: X corresponds to X2, Y corresponds to X3
  - X1-X3 plane: X corresponds to X1, Y corresponds to X3
- Each group may be recalled several times to a single loading case.

Example:

Apply the "H2" in the PATTERN.DAT file example 476, as shown in the following figure. Note that the load is defined horizontally so it must be rotated by 90° :

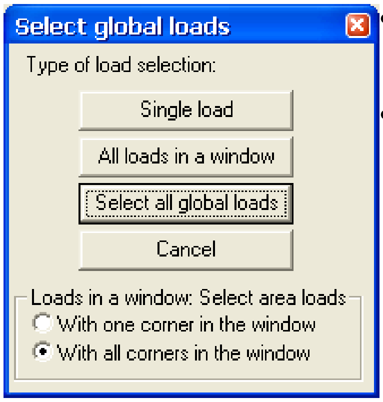


Specify:
Load direction = X3
Angle = 90
Factor = 1.4

Set coordinates to: **X1 =2.5 , X2=0.71**

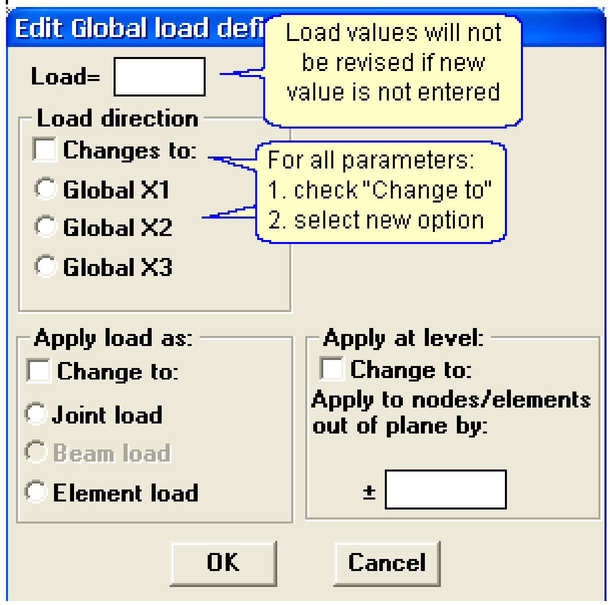
4.12.2 Revise

Revise a single or several global loads, including the load value, the type, level, application and direction:



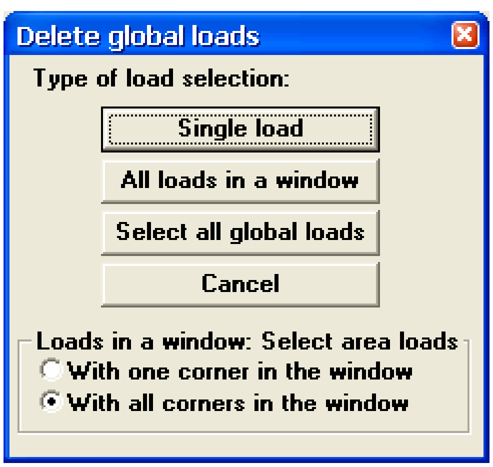
- Single load:
  Select the global load and proceed according to Define 466

- All/selected loads:
  Select several loads and change one or more of the following parameters for ***all*** of them:

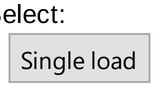


4.12.3 Delete

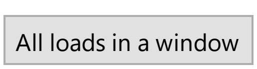
Delete one, several or all global loads:



Select:

- 

  move the adjacent to the load so that it is highlight with the ■ and click the mouse.

- 

For area loads the program displays the ■ at each of the area corners; select one of the options at the bottom of the box:

..**one corner** .. - the load is erased if any one of the corner ■ is in the window

..**all corners** .. - all of the corner ■ must be in the window to erase the load

- Select all global loads

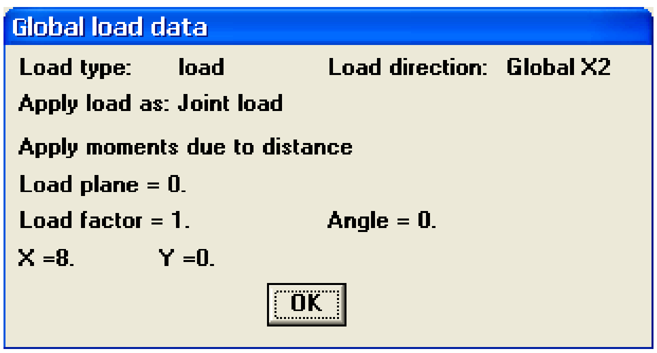
All global loads in the load case are deleted

4.12.4 Show

Display the data for any existing global load:

- move the adjacent to the load so that it is highlighted with the ■ and click the mouse.

Example:



4.12.5 PATTERN.DAT

Global loads may be stored in an ASCII format file and recalled directly into a load case. This option is useful for applying vehicle loads to a bridge model. The load patterns are stored in the file **PATTERN. DAT.**

- A file with standard vehicle loads is installed with *STRAP* in the program directory. The user may edit, add or delete the loads in the file using any text editor program.
- The user may create a **PATTERN.DAT** file in any working directory. When the user selects the **Load pattern** option, the program first looks for the file in the current working directory; if the file is not found, the program then looks in the program directory.

The commands must be in the standard format - refer to the Command Mode Manual.

Various load patterns can be stored in the file and can be recalled separately.

- Each group begins with a "***load name***" (no blanks allowed) and is terminated with "***END***".
- The load commands are identical to those entered in the Command Mode, except that "BEAM list" or "ELEM list" at the end of the command is not allowed; the loads are applied by default to all beams/ elements (refer to Method of Application) but may be applied to selected elements.

The file format is:

```
NAMES
loadname1
loadname2
loadname3
.
.
END
loadname1
.
global load commands for group 1
.
.
END
loadname2
.
global load commands for group 2
.
.
END
loadname3
.
```

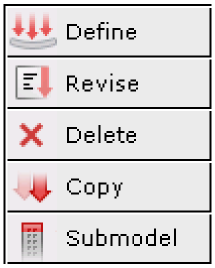
Example:
A file with two load groups,
named "H1" and "H2"

```
NAMES
H1
H2
END
H1
U -1.2 C 0 0 R 3.15 2.87
P -12.7 C 3.5 6.7
P -12.7 C 3.5 8.7
END
H2
P 5.1 C 2 0
P 12.1 C 3.4 0
END
```

4.13 Solids load

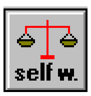
The loads that may be defined for solid elements are temperature loads and self weight. To define any type of surface load (pressure, concentrated, etc) define dummy beam or elements connected to the end nodes of the solid elements and apply loads to them.

Select one of the following options:

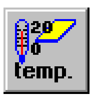


4.13.1 Define

Select the load type:



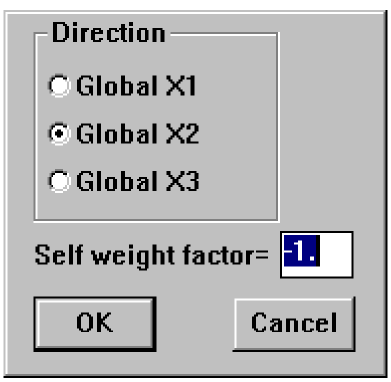
Self-weight 478



Temperature 479

Self-weight

Apply the self-weight as joint loads; the weight of the element (volume x density of the element material property) is divided equally to the element end nodes, even if the element is not symmetric.



- ***Direction:***
  The self-weight can be applied in any ***global*** direction.
- ***Factor:***
  The self-weight is calculated as the element thickness multiplied by the material density. This value may be multiplied by a factor. If a negative factor is defined, the load is applied in the negative direction of the global axis selected.

  If the material is USER-DEFINED, verify that the density was defined (a warning is displayed).

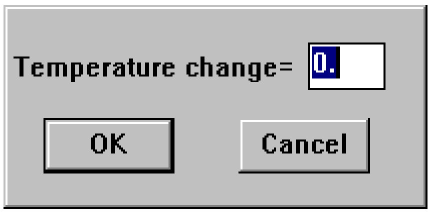
Examples:
- apply self-weight as a vertical load:

Global X2
Self weight factor = 1.00

- apply 10% of the self-weight as a horizontal load:
  Global X1
  Self weight factor = 0.1

Temperature

A temperature load applies a uniform stress in **all** global directions equal to $\mathbf{E\alpha(\Delta T)/(1-2\mu)}$



Enter the temperature ***difference***.

The program multiplies the temperature difference by the thermal coefficient of the material(s); therefore the temperature and the coefficient must have the same units. Check the value of the thermal coefficients and enter the difference in degrees Celsius (°C) or degrees Fahrenheit (°F) accordingly.

Note:

- Select the elements using the standard Element Selection option.

4.14 Copy loads

The **Copy loads** option enables the user to copy existing loads from one part of the model to another. Similar to the "**Copy**" geometry option, the loads may be translated, translated and rotated, or a mirror image may be created.

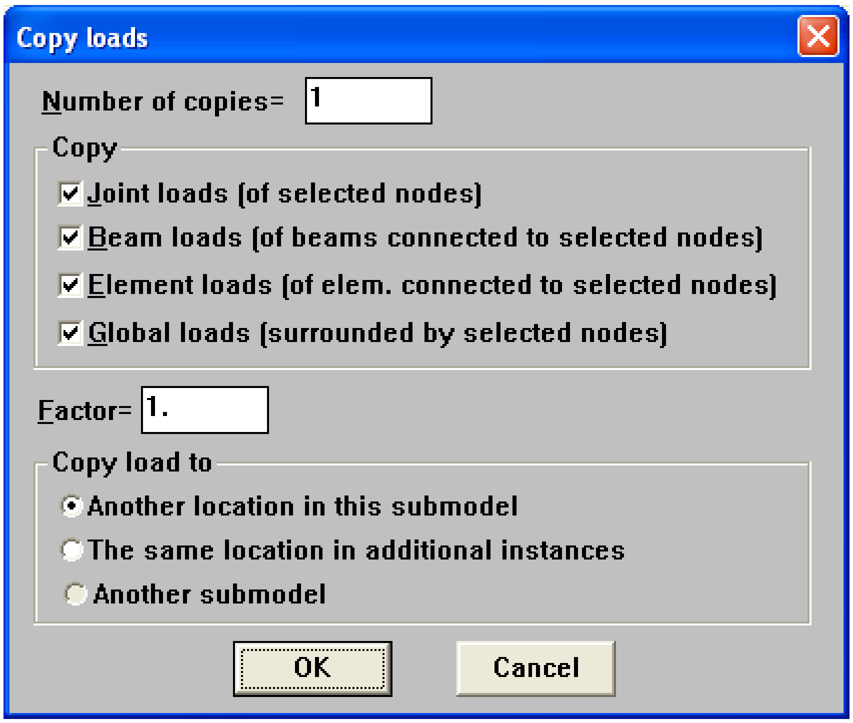
To copy loads:

- select: 

, 

, or 

- Specify the **Copy load** options:



Number of copies

The loads may be copied more than once; each copy is offset an identical distance from the previous one.

Copy

Set the checkboxes for the various load types:

☑ the loads is copied

☐ the loads is not copied

Note:

- beam/element loads: only the loads on beams/elements with ALL end nodes selected are copied.
- global loads must be 'enclosed' by the selected nodes and on the same plane in order to be copied:

area loads:	the contour must enclose at least one of the selected nodes or at least the first three nodes of the contour must be 'surrounded' 482 by selected nodes
point loads:	must be 'surrounded' 482 by selected nodes.

Factor

All copied loads may be multiplied by a factor. A negative factor reverses the load directions.

Submodels

For loads defined on submodels - the selected loads may be copied either to the current instance, a different instance of the same submodel, or to instances of a different submodel.

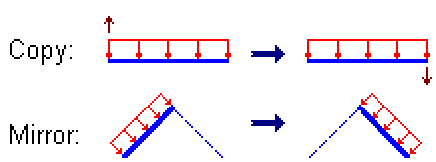
- Use the standard Node selection option to select the nodes specifying the joints, beams and elements whose loads are to be copied. For beams and elements - ***all*** corner nodes must be selected.
- Select the reference nodes and their new locations:

select one reference node and its new location in the first copy. The distance between all subsequent copies will be identical

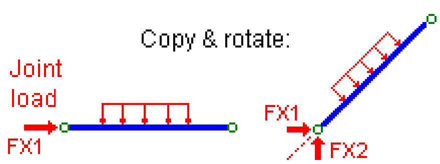
the new location is specified by three reference nodes.

the new location is specified by a single reference node (not on the plane of symmetry). The program joins the old and new locations of the reference node with an imaginary line and bisects this line with a perpendicular plane. All selected loads are copied on the other side of the plane.

Note:
- Joint loads, beam loads, element loads and global loads only may be copied.
- Support displacements are not copied.
- Loads that were added to the current load case using the Combine 464 command cannot be copied.
- Element loads generated using the Linear 450 element load option are not copied.
- Loads must be copied to identical geometry in order to be duplicated:
  - joint loads that do not fall exactly on another joint are not copied
  - beam/element loads that do not fall on another beam/element with identical dimensions are not copied.
  - global loads must be 'enclosed' by the selected nodes and on the same plane in order to be copied:
    area loads: the contour must enclose at least one of the selected nodes or at least the first three nodes of the contour must be 'surrounded 482' by selected nodes
    point loads must be 'surrounded 482' by selected nodes.
- The program maintains the direction of beam/element loads, even if the local axis directions are not consistent. For example:

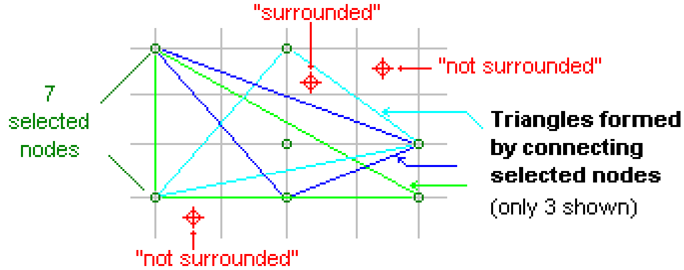


- Joint loads rotated to a direction not parallel to a global axis are separated to the equivalent global components. For example:



- Global loads defined in a global direction are applied in the closest global direction after rotation. Global loads applied perpendicular to the plane remain perpendicular to the plane.

'Surrounded' global point loads or 'Surrounded' global area load contour corners



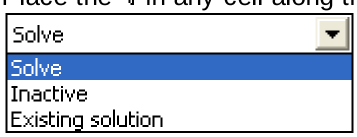
4.15 Deactivate loads

Deactivated load cases are not solved and not erased. They may be reactivated at any time. There are two options:

- **Inactive**
  This option allows you, for example, to define basic loading cases and use them in the **combine ld.** option without solving them, thereby saving solution time. These load cases are not displayed in the results or design module menus.
- **Existing solution**
  The program does not overwrite the existing solution for the selected load case. This is the default option for load cases generated by the Dynamic, Bridge and *POSTTEN* modules.

The program displays a list of existing load cases:

- Place the in any cell along the load case line and click the mouse; select the option from the list:



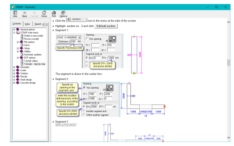
Note:

- the combinations in the result module are automatically updated when **Solve** or **Inactive** is selected for a load case.

4.16 Moving loads

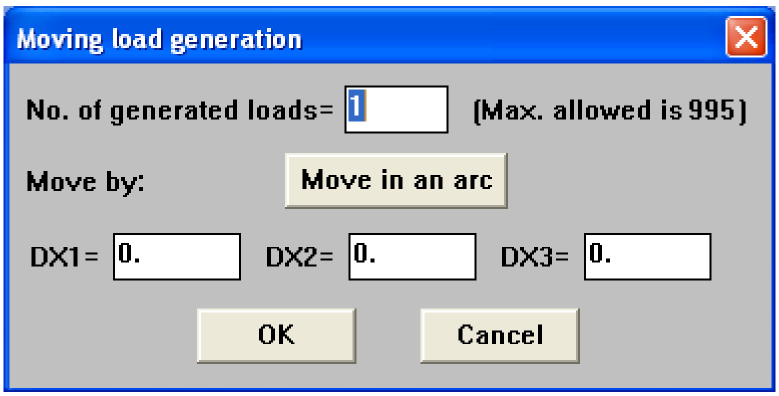
To display a step-by-step tutorial that explains how to define "Moving loads":

- ***Moving loads***



Generate a series of new load cases from an existing load case containing global loads. In each new load case, the global loads are moved by a ***constant*** increment. All other load types (beam loads, etc.) are applied in the generated cases, but remain at the same location.

Select an existing load case; the program displays the following menu:

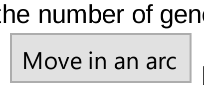


No. of generated loads

Specify the number of new load cases (in addition to the original case). The "maximum allowed" is the maximum number of load cases allowed less the number of cases already defined.

Move in an arc

To generate copies of global loads along an arc:

- Specify the number of generated loads
- click the 

 button
- specify the nodes at the start and end of the arc as well as any intermediate node.

If the existing global load is at the start node, then the last generated copy will be at the end nodes and the intermediate copies will be at equally spaced intervals along the arc.

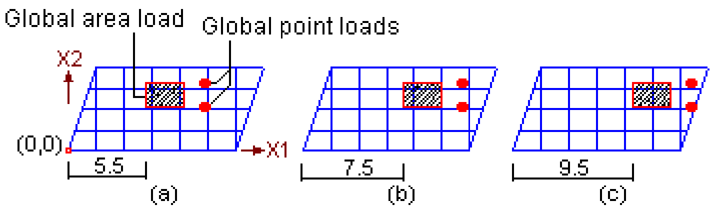
If the reference point of an existing global load in the same load case does not lie on the defined arc, the program will draw an identical arc parallel to the original one starting at load reference point; global load will be generated along the parallel arc as described above.

Move by

Enter the coordinate increment of the global load location.

Example:

The title of an existing load case is "Wheel at +12.3". This load case is shown in the figure (a). Generate the two load cases shown in (b) and (c).

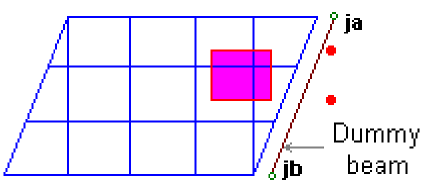


Specify:
No. of loads generated: 2
DX1 = 2.0   DX2 = 0.0

The program generates two new loading cases, "Wheel at +12.3 #2" and "Wheel at +12.3 #3"

Note:
- the program remains in the "Moving load" options after generating the new cases and adds the title of the ***last*** generated load case to the list. If this case is selected for generating new moving loads, the second set of generated loads will be labeled "Wheel at +12.3 **#4**", "Wheel at +12.3 **#5**", etc. This feature allows variable coordinate increments to be easily defined.
- If the original load case is revised, the program can revise all the related moving loads (a prompt is displayed after **End load case** is selected); if a generated load case is revised, the other cases cannot be automatically regenerated.

Referring to the Method of Application for global loads, loads located outside the model that are applied as beam loads will still generate loads on the structure. This may create a problem in the Moving Loads option as the last generated load case may be partially or completely beyond the model boundary. For example:



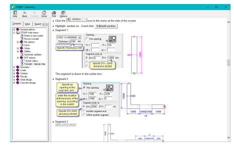
To ensure that such loads are not applied to the model:
- define nodes offset a small distance (1 cm) from the model ('ja', 'jb' in the above figure)
- connect these nodes with dummy beams lying parallel to the model boundary
- do not connect these new nodes to the structure with other beams or elements.

The global loads outside the model boundary (the two concentrated loads in our example) will be applied to the dummy beams. They will have no influence on the structure as the dummy beams are not connected to the model. The solution and results modules of the program ignore dummy beams so no error messages or results are displayed.

4.17 Chess (Staggered) Loads

To display a step-by-step tutorial that explains how to define "Chess loads":

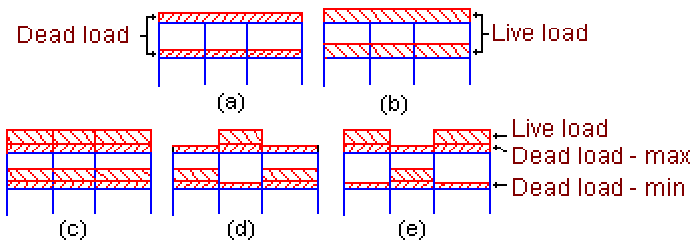
- ***Chess loads***



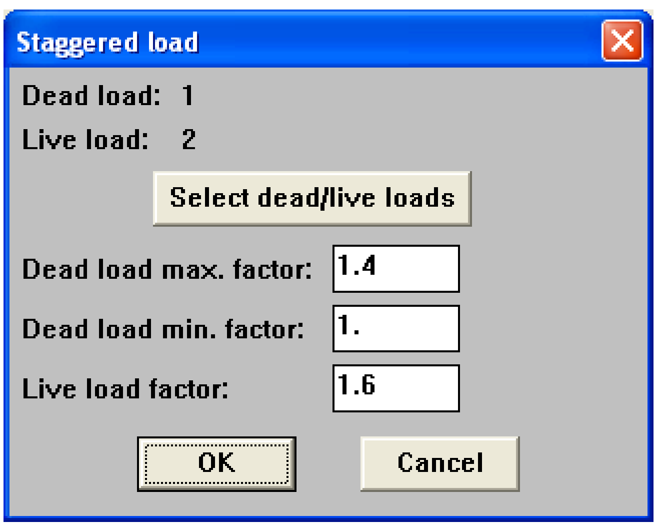
Most design codes require several patterns of dead and live loads for the calculation of maximum support and span moments in beams.

This option automatically generates the required patterns from basic load cases containing the dead loads on all spans and the live loads on all spans; if one of the basic dead or live load cases is later revised, all of the generated cases may be automatically revised.

Example: automatically generate the load cases (c), (d) and (e) from the basic load cases (a) and (b):



The patterns to be generated are listed in the file CHESS.DAT 487 which may be revised by the user.



Select dead/live loads

Identify the load cases that contain dead loads (5 maximum) and live (imposed) loads (5 maximum).

Dead load max. factor

Specify the load factor for dead loads when maximum load is applied. For most Codes, the value is 1.40.

Dead load min. factor

Specify the load factor for dead loads when minimum load applied. For most Codes, the value is 1.00.

Live load factor

Specify the load factor for live loads when applied. For most Codes, the factor is in the range from 1.5 to 1.7.

The program now displays schematically each of the generated load cases in sequence. Click:

To generate the load case.

Return to the previous generated load case.

If the arrangement of the live load is not satisfactory, three options are available for modifying it:

All beams assigned with maximum load are assigned with minimum load and vice versa.

Select loaded beams using the standard Beam Selection option; all selected beams with minimum load are revised to maximum load.

Select loaded beams using the standard Beam Selection option; all selected beams with maximum load are revised to minimum load.

The design postprocessors (steel and concrete) assume that the factored load combinations are defined after the solution and that the load case deflections are from service loads.

If the load combinations are defined using this option, then the loads in the cases are factored. As the postprocessors cannot reduce the deflections, the allowable deflection limit must be correspondingly increased in the postprocessor..

For example:

- allowable service load deflection = L/350
- Average load factor on span = 1.55
- Specify adjusted allowable deflection = 350 / 1.55 = L/226

4.17.1 CHESS.DAT

The list of patterns to generate is stored in the file CHESS.DAT. The file may be revised using any editor program.

Each line is in the format:

LEN nsp PAT n1 n2 ... nnsp

where:

nsp = number of spans in the pattern

n1, n2, .. = 0 or 1 for each of the **nsp** spans; 1 indicates that the span is loaded and 0 indicates that the span is unloaded

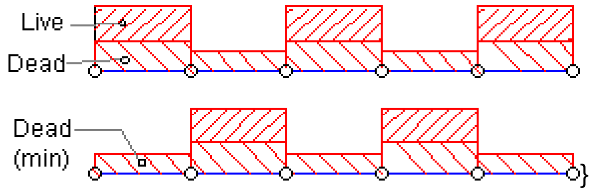
The default patterns are:

LEN 2 PAT 1 0
LEN 4 PAT 1 1 0 0

where:

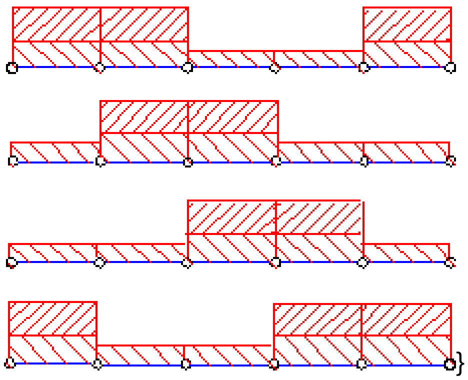
- Line 1: 2 spans in the pattern; 1st span loaded and 2nd span unloaded.

The program produces the load patterns:



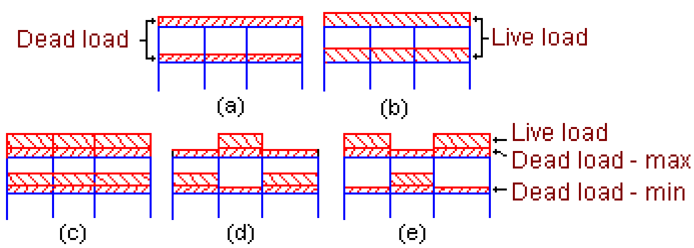
- Line 2: 4 spans in the pattern; 1st and 2nd spans loaded and 3rd and 4th spans unloaded.

  The program produces the load patterns:



For the following example, the file CHESS.DAT should read:

LEN 1 PAT 1
LEN 2 PAT 1 0

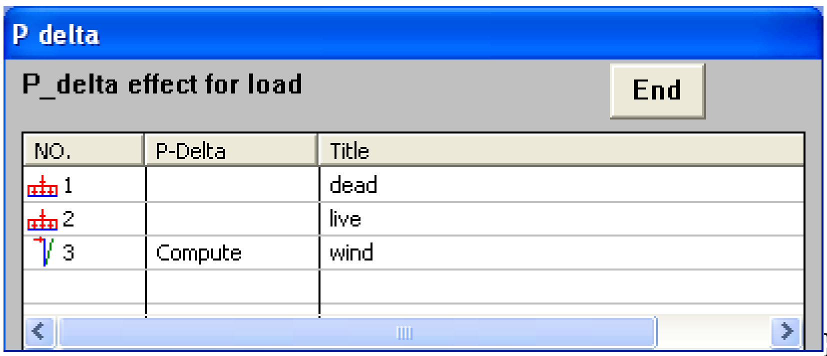


4.18 P-Delta

Secondary moments and forces resulting from the action of the loads on the deflected model (the P-DELTA effect) may be calculated.

Note:
- For details on the algorithm used by the program refer to P-Delta - Method of Calculation 489.
- P-Delta calculation applies only to beam elements. The calculation must be specified for each load case where the effect is required, as explained below.
- The P-Delta effect is ***non-linear***, i.e. the rules of superposition do not apply. Therefore, load combinations for models with P-Delta must be defined here in LOADING, rather than after the solution.

The program displays a list of existing load cases:


P delta

P_delta effect for load   End

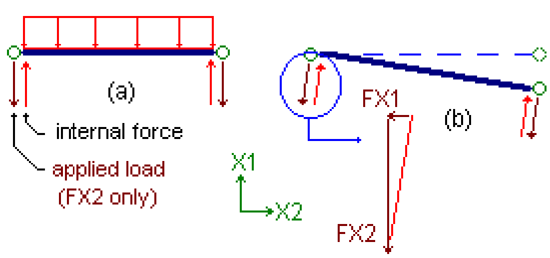
NO.	P-Delta	Title
1		dead
2		live
3	Compute	wind

Move the cursor into any cell on the line in the table corresponding to the load case and click the mouse; the P-Delta status of the loading case is then revised.

4.18.1 P-Delta - method

The program uses the following iterative method for calculating the secondary moments and forces in the model due to the action of the loads on the deflected structure - the P-Delta effect. The P-Delta calculations are applicable to beam elements only.

The program carries out the P-Delta calculation as follows:
- at any node, the applied loads equal the sum of the internal forces at the ends of the beams connected to the nodes, with the opposite sign (Figure a).



- for the second and subsequent iteration, the program first calculates the new orientation of each beam, based on the deflected location of its end nodes from the previous iteration, i.e., a revised local coordinate system is assigned to the beam (Figure b).
- the program calculates the sum of the internal forces at each node based on the new local systems of the connected beams. This new sum is then applied to the node (with the opposite sign) as the new applied load. In the example above, the applied load from the original vertical load now has a horizontal component (Figure b).

- the program solves the model again and continues with the next iteration, if required by the convergence criteria.
  This iteration process is repeated until the following condition is satisfied at ***all*** nodes in the model:

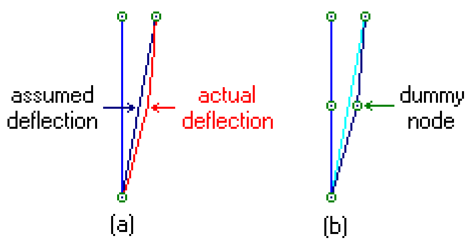
  $$\frac{(\text{defl})_i - (\text{defl})_{i-1}}{\text{max. deflection}} < 0.005$$

  where:
  i = the current iteration.
  i-1 = the previous iteration
  max. deflection: refers to the original solution.

  Relatively flexible structures may not converge. If the solution has not converged after five iterations, the program pauses and asks the user whether to continue. The question is repeated every 5 iterations. Note that almost all models converge within 5 iterations.

Note:

- all iterations are calculated with the ***initial*** stiffness matrix (calculated from the initial, undeflected geometry)
- the support reactions in the result tables include the additional forces due to the P-Delta effect.
- the calculation is generally conservative for relatively slender members with deflections along the length of the member. Referring to Figure (a) in the following drawing, the deflection assumed by the method is not equal to the actual deflection of the member.



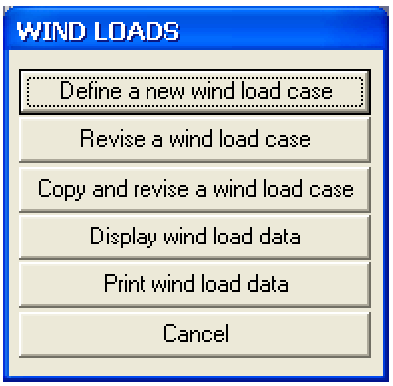
  The accuracy may be improved by inserting a dummy node along the member as shown in Figure (b).

4.19 Wind loads

This option generates wind loads according to Code requirements and applies them as loads either on panel or lattice members. Refer to Wind loads - general 492 for more details.

To display a demo video that explains how to define wind loads:

- *click on to start the video*
- *then click on to enlarge the display.*

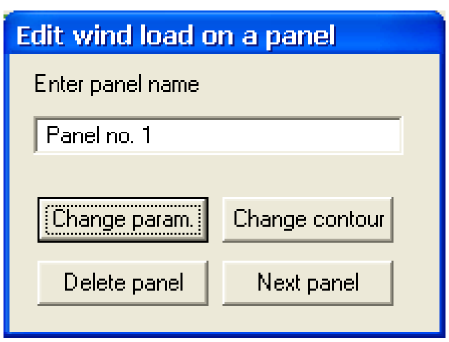


Define a new wind load case

Refer to Wind loads - define 493.

Revise a wind load case

Select a load case; the program displays the following menu for each panel defined in the case:



Select:

Change param.	-	revise the Code based parameters that define the wind load on the panel
Change contour	-	redefine the entire contour for the current panel (you cannot revise individual nodes); the program automatically displays the parameter menu after the contour has been revised.
Delete panel	-	delete the current panel entirely
Next panel	-	do not revise the current panel

Copy and revise a wind load case

Select any existing load case (wind or regular) and add wind loads to the copy. Refer to Wind loads - define 493.

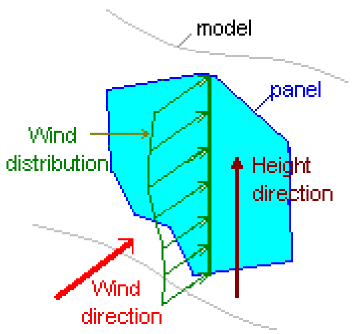
4.19.1 General

This option generates wind loads according to Code requirements.

The user defines:

- the outline of areas referred to as "panels", or selects beams and defines the perpendicular panel width.
- the direction in which the wind acts.
- the "height" direction of the panel.
- various Code parameters.

The program calculates the wind pressure distribution on the panel according to the Code and distributes the load to the nodes/beams/elements in the panel area.

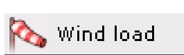


In general, each wall or roof plane is defined as a separate panel.

Note:

- Wind pressures are applied to the nodes/beams/elements using the same methods that are used to apply global area loads.
- Dynamic effects are not considered.

Procedure:

- click the  icon
- select "**Define a new wind load case**"
- specify how to apply the loads - to nodes, beams or elements
- specify the load definition method:
  - user-defined panel
  - automatic creation of panels by the program on the entire structure (walls & roof)
  - lattice/tower loads on individual members
  - selected beams and perpendicular width (for plane models).
- define the Code wind load parameters
- repeat for additional load cases

Note:

- if panels defined in the same wind load case contain common members, the program applies a load only once to each of these members.

To check the loads generated by the program :

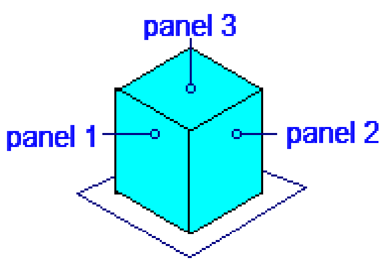
- click the Existing load icon
- select the wind load case
- the global loads generated by the program on the panels will be displayed. ***** Do not revise the applied loads in this option *****

To revise wind loads:
- click the Wind load icon
- select Revise a wind load case and select the case
- for panels defined by contour: revise the load application method (beam/element/joint loads).
- for all wind load types: revise the contour/beam list, the parameters of an existing panel and/or add new panels

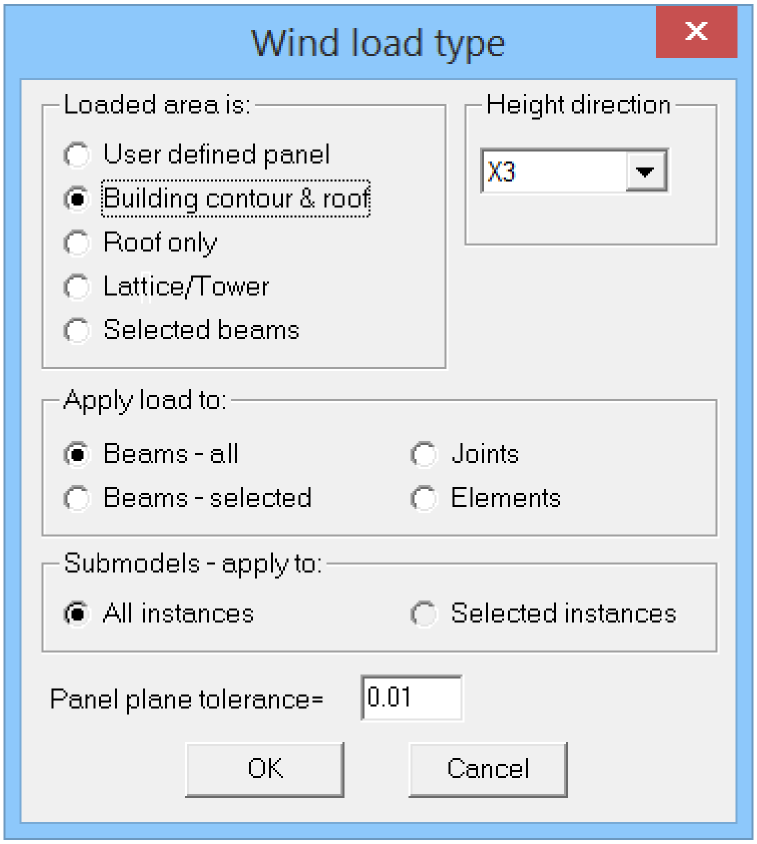
Note:
- a wind load case defined by contours cannot be revised to a case defined by beams & width, or vice-versa.

4.19.2 Define

Wind loads in a load case may be generated as:
- uniform beam loads on lattice/tower structures.
- wind pressure on series of contour areas called "**panels**" which are applied to the structure as element, beam or joint loads. For example:



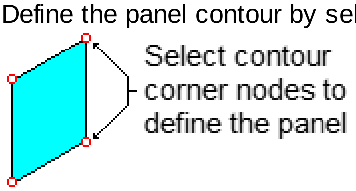
Specify the following options:



Loaded area is

User defined panel

Define the panel contour by selecting its corner nodes. For example:



Building contour and roof

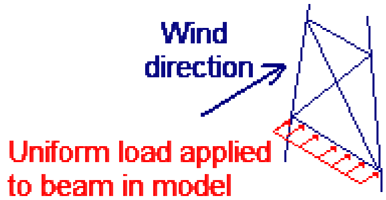
Define the entire building envelope, including the external walls and the roof. The program automatically creates the individual panels on all surfaces and calculates the wind loads for all panels. Refer to Wind loads - contour/roof 496.

Roof only

Define the roof contour. The program automatically creates the panels and calculates the wind loads for the entire roof surface. Refer to Wind loads - contour/roof 496.

Lattice/tower

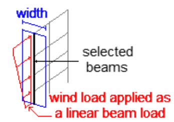
Apply as uniform beam loads on a "lattice" structure. the program assumes that the model is an open structure, i.e. the wind blows through the model. This option is primarily intended for transmission towers, open trusses in industrial buildings, etc.



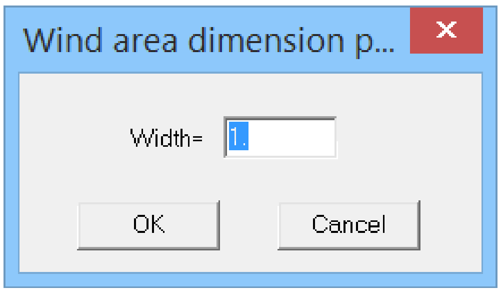
The program calculates the projected surface area of each member perpendicular to the wind direction, calculates the wind load for individual members in a lattice structure according to the Code and applies it either as a distributed load on the selected beams or as joint loads at the beam ends.

- **Selected beams**

  A series of beams and the perpendicular width: the program applies linear beam loads (this is the only option available for plane frames)

  After specifying the code parameters, specify the width of the "panel" perpendicular to the line of beams:

Height direction

Select the global height (vertical) axis of the building. The program assumes that all wind loads are perpendicular to this direction.

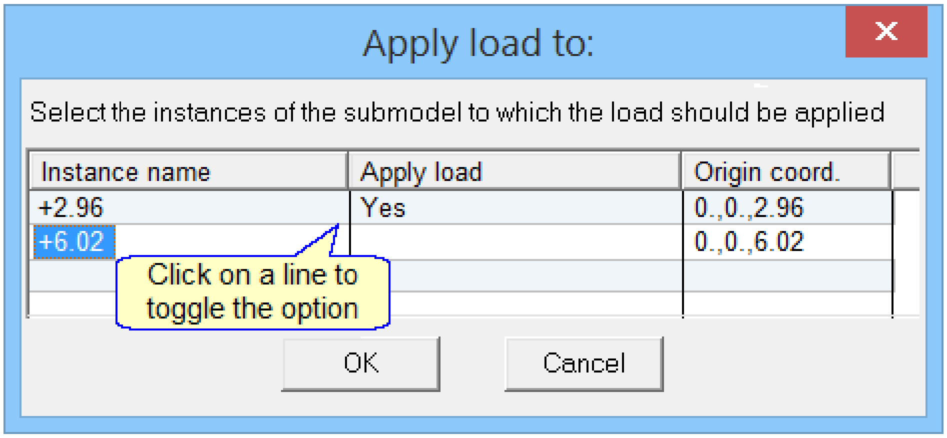
Apply load to

The global area loads calculated by the program are applied to either nodes, beams or elements:

- **Beams - all :** The program identifies the beams in the area bounded by the panel and applies the loads to ***all*** of them.
- **Beams - selected :** The program identifies the beams in the area bounded by the panel and applies the loads only to beams ***selected*** by the user.
- **Joints :** The program identifies the nodes in the area bounded by the panel and applied the loads to ***all*** of them. This option is not available for "Lattice/tower" option.
- **Elements :** The program identifies the elements in the area bounded by the panel and applied the loads to ***all*** of them. This option is not available for "Lattice/tower" and "Selected beams" options.

Submodels

This option is displayed only when a submodel is currently displayed and there is more than one instance attached.

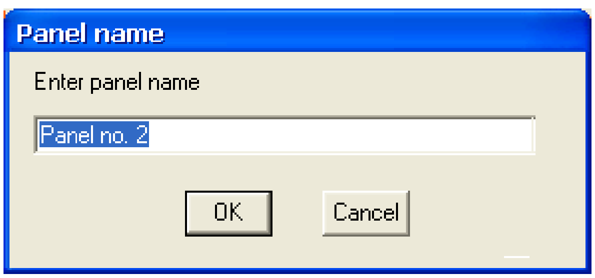


Tolerance

Wind loads are applied only to beams/elements lying on the panel +/- a specified tolerance value. The tolerance value is by default set to 0.01 to allow for minor inaccuracies in the panel corner coordinates.

For all options:

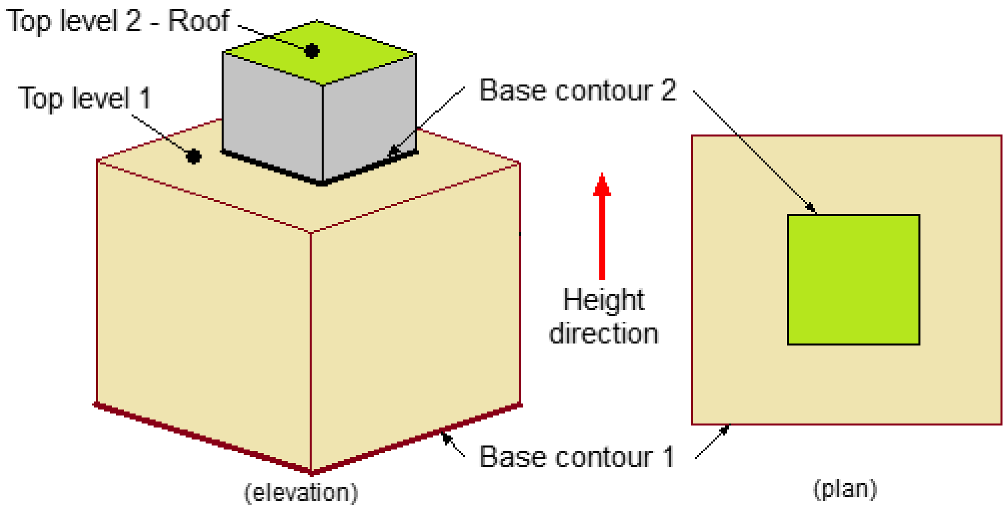
Enter a title for the new panel.



4.19.2.1 Contour/roof

Define the entire building envelope, including the external walls and the roof. The program automatically creates the individual panels on all surfaces and calculates the wind loads for all panels.

The envelope is defined by defining a series of contours perpendicular to the height direction and identifying the top level corresponding to each contour. For example, two contours are required in this structure:

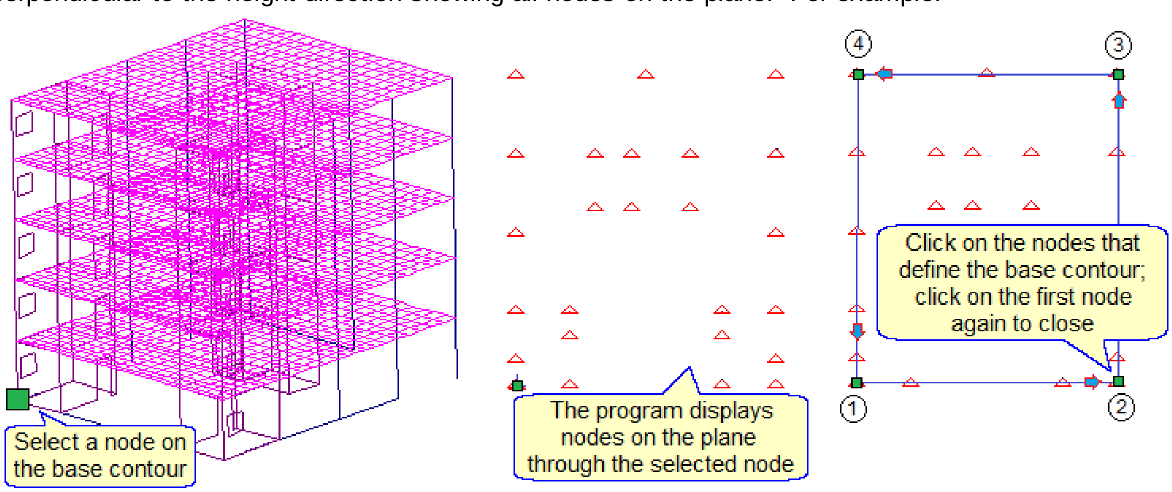


The final top level is the "roof", which can be flat, monopitch, duopitch or multispan (all intermediate top levels are flat).

The program then creates "panels" on all the exterior surfaces and generates the wind loads on all panels according to the specified parameters.

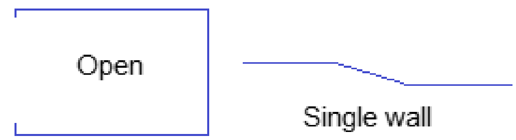
To define the wind loads using this option:

- select **Building contour and roof**, select the height direction and parameters; click OK.
- enter the load case title
- select a node on the first "base contour". The program then displays the plane through this node perpendicular to the height direction showing all nodes on the plane. For example:

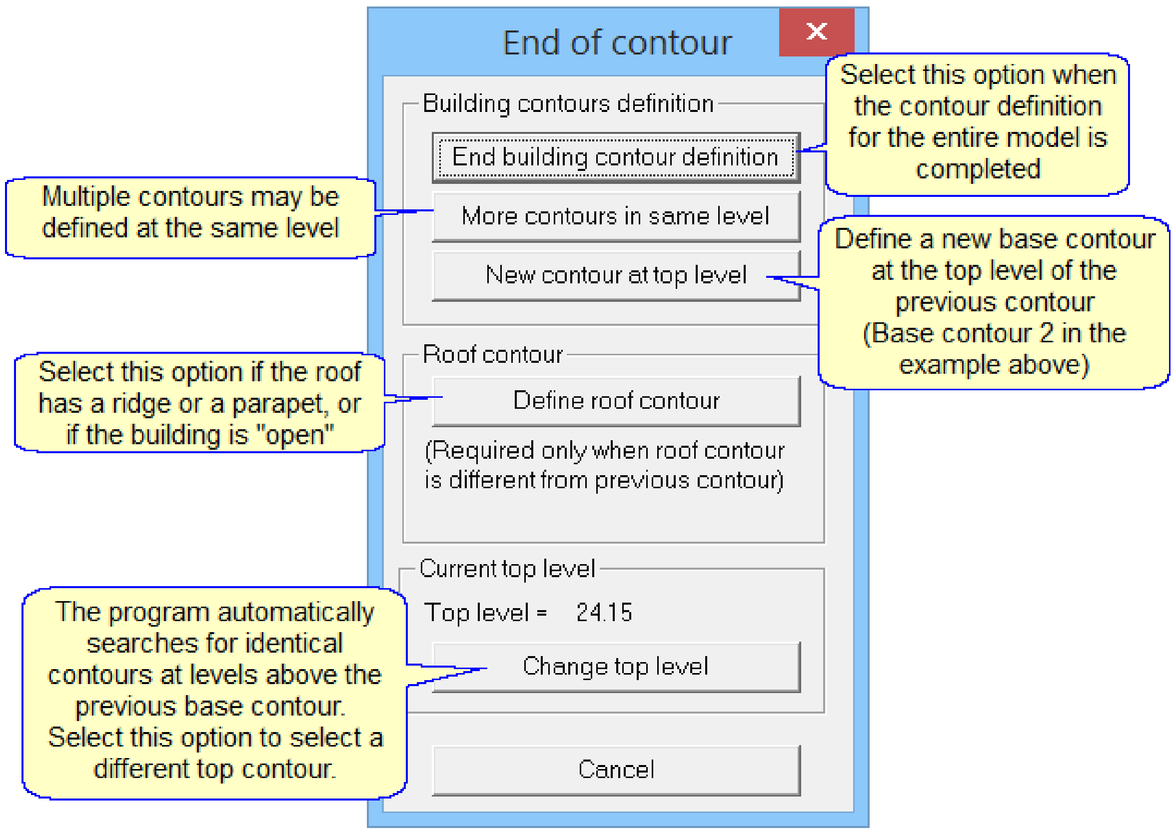


Note that the contour does not have to be closed.
The building can be either -

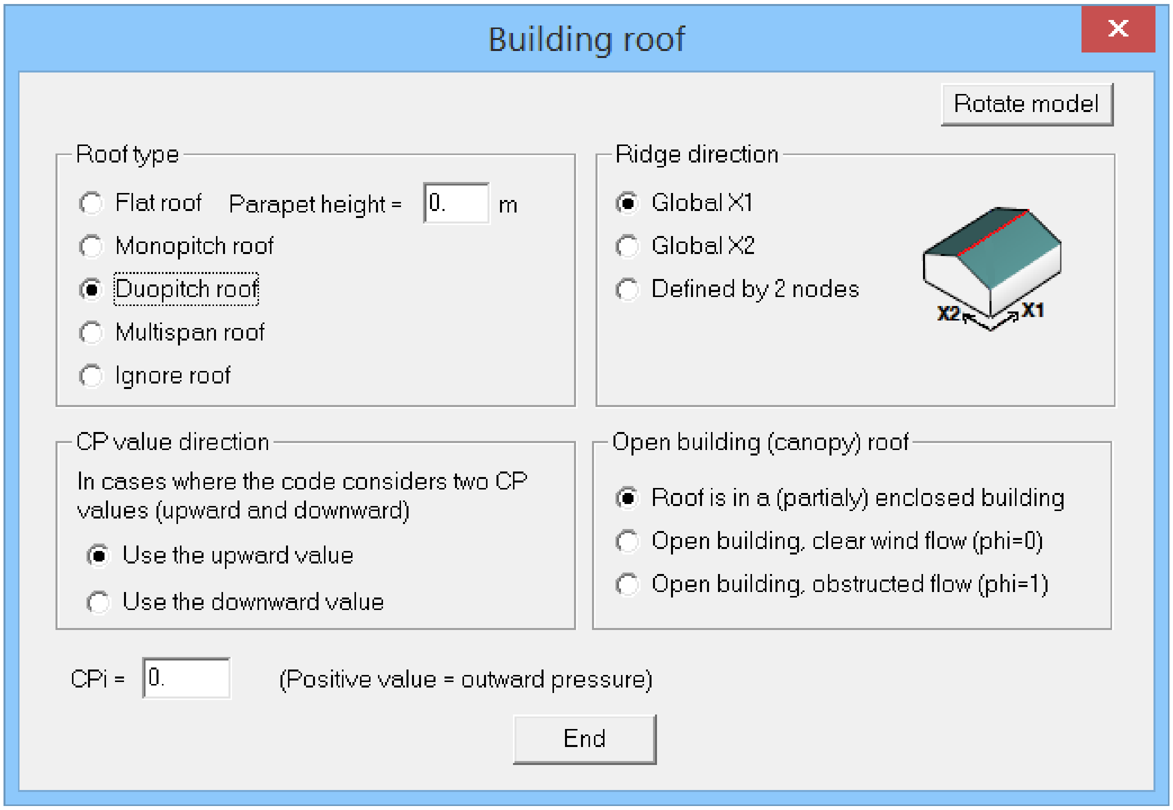
- open
- single wall



- End the definition of base contours or start the next one:



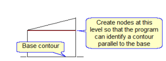
- Define the roof:



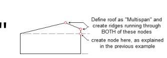
Roof type:

Select the roof type and the parapet height (flat roof only):

Note:



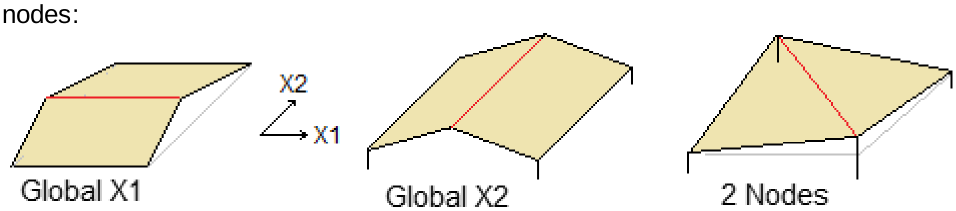
- For a "monopitch" roof, a contour parallel to the base contour must be defined at the bottom level of the roof. This may necessitate defining additional nodes at that level:



- The following roof configuration (columns with unequal heights) must be defined as a " **Multispan**: roof:

Ridge direction:

The roof ridge direction can be parallel to either of the global axes or can be defined by any two nodes:



Cp value direction:

Codes specify two C_p values for roofs: one for upward pressure and one for downward pressure. Two load cases should be defined, each with a different C_p value for the roof.

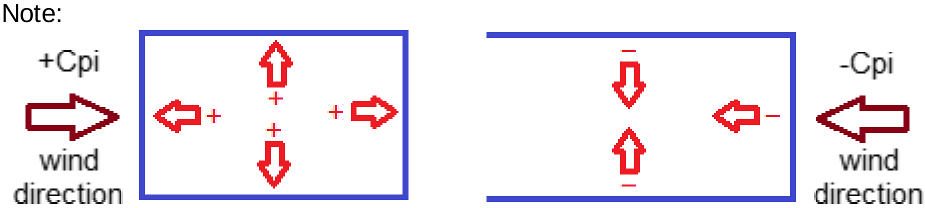
Open building:

The Codes classify buildings as "open" or "closed", depending on the total amount of openings. Open buildings can be classified as "obstructed" if internal blockage impedes the wind flow through the structure.

Cpi value:

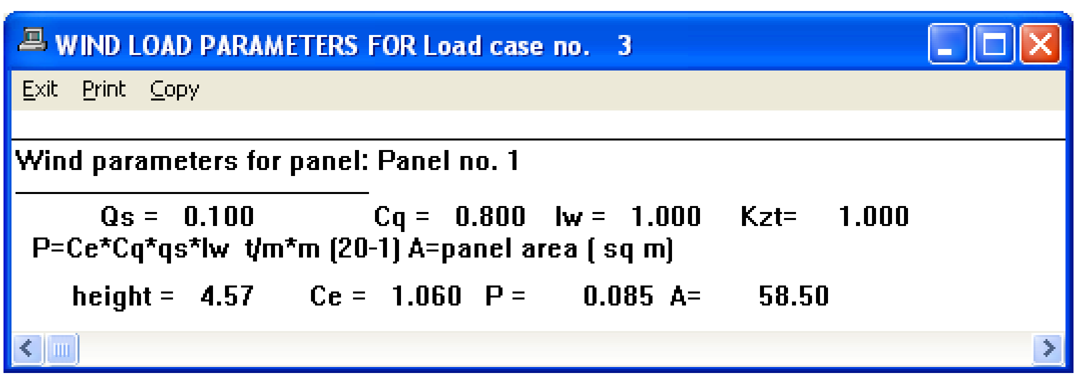
Define C_{pi}, the internal pressure coefficient. Wind codes usually specify different values for "open", "partially enclosed" and "enclosed" building. The internal pressures are added (or subtracted) from the external pressures. This value is also used to calculate the internal pressure on walls.

Note:



4.19.3 Display wind parameters

Display/print a table detailing the calculation of the load pressure for each panel according to the Code parameters. For example:



Refer to:

Wind loads - general 492

Codes - calculation methods

4.19.4 WINDUSER.DAT

A single wind pressure vs. height table may be defined and stored by the user:

- The file name must be **WINDUSER.DAT**
- The file must be saved in the program directory
- The program assumes that the data in the file is in the model units and does not modify them, e.g. if the model units are kips and feet, then the heights must be defined in feet and the pressure in ksf.
- The numbers must include a decimal point
- The file may contain any number of rows and must end with a row that starts with **-99999.0**
- The rows must be arranged in ascending order of height
- Each row includes one height value and one pressure value; the height value must be in columns 1-8 and the pressure value must be columns 9-16

Example:

3.0	0.7
5.0	1.4
7.0	2.5
|	|
|	|
100.0	12.9
120.0	16.2
-99999.0	
(height)	(pressure)

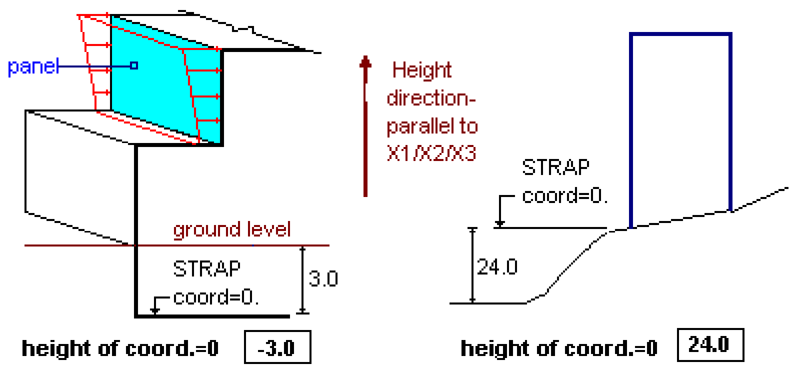
4.19.5 Wind codes

4.19.5.1 General parameters

Height axis / coordinate

Specify:

- the global axis that is the "Height axis" of the panel; the wind will vary along this direction according to the height above the ground surface
- the height of the ground surface at the *STRAP* zero height axis coordinate. For example:



Wind direction

Specify one of the global axes as the "**Wind direction**".

The program compares the wind direction with the orientation of the panel face.

4.19.5.2 IS875

The calculation is based on IS 875 - "Code of Practice for Design Loads" - Part 3 - 3rd Revision (2015).

The wind pressure acting normal to a surface is calculated as:

$\mathbf{F = (C_{pe}-C_{pi})\,p_d}$ (Section 7.3.1)

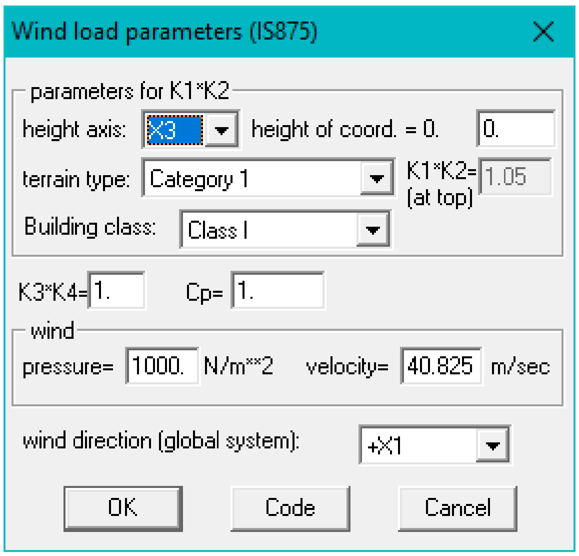
where:

C_{pe} = external pressure coefficient, calculated according to 7.3.3.
C_{pi} = internal pressure coefficient, calculated according to 7.3.2.

p_d = $K_d\ K_a\ K_c\ p_z$ (Section 7.2)
K_d ,K_a ,K_c are assumed = 1, i.e. p_d = p_z For other values modify the value of C_p
pz = design wind pressure = **0.6 Vz²** N/m² (Section 7.2)

Vz = design wind velocity at height 'z' = **Vb·k1·k2·k3·k4** m/sec (Section 6.3)
Vb = basic wind speed
k1 = Risk coefficient (section 6.3.1)

k2 = terrain,height factor (Section 6.3.2)
k3 = topography factor (Section 6.3.3)
k4 = importance factor (Section 6.3.4)



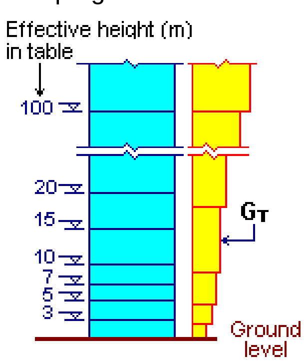
Height axis / Coordinate / Wind direction

Refer to Wind parameters - general 501

Terrain type / K1*K2 / Building class

There are three methods for specifying the value for **k2**: select one of the following options in the "**Terrain type**" list box:

- Terrain categories **Category 1, Category 2, Category 3, Category 4:**
  The program takes the values of **k2** from Table 2 according to the height of the panel segment.



- User table
  The program will use the **k2** vs. effective height/building class table stored in file WINDUSER.DAT instead of Code Table 2 (all other factors are ignored).

- Define k_1*k_2
  Enter a value for k_1*k_2 in the adjoining edit box. The value is uniform over the entire face of the panel.

K3*K4

Enter the value of k_3***k_4**

- **k_3** : as defined in Section 5.3.3 and Annex C.
- **k_4** : as defined in Section 6.3.4

Cp

Enter the value of **$C_p = (C_{pe}-C_{pi})$**, the difference between external and internal pressure coefficients, as defined in Sections 7.3.2. and 7.3.3 and the relevant tables.

Pressure / velocity

Referring to Code Section 7.2:

$$p_z = 0.6 \cdot V_b^2$$

Enter ***either*** the wind speed, **V_b**, ***or*** the wind pressure, **p_z**; the other value is automatically updated according to the equation.

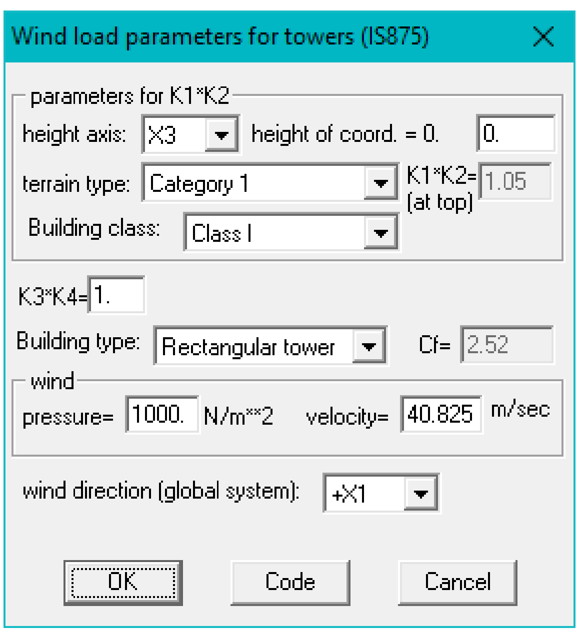
4.19.5.3 IS875 - lattice

Wind on members in lattice structures is calculated according to Section 6.3.3:

$$F_n = C_{fn} \cdot p_d \, K \, (l \, b)$$

where (**l b)** is the projected surface area of the member.

The program determines the projected area perpendicular to the wind direction for each selected beam. The pressure on any member is constant; the program calculates the pressure using the maximum height. The H2, H3 dimensions in the beam property tables are used for this calculation and they must be defined. (*Note that H2=H3=0 in all models created prior to Version 9.00 and hence all lattice wind loads will be zero*)
.



Height axis / Coordinate / Wind direction

Refer to Wind parameters - general 501

Terrain type / k1*k2 / Building class

Refer to IS875 - Terrain type 502

k3*k4

Refer to IS875 - k3*k4 503

Building type

The program uses the building type specified for calculating the value of **Cf**:

- **Rectangular tower** - according to Table 33
- **Triangular tower** - according to Table 33
- **Plane lattice** - according to Table 31
- **User defined Cf** - for all other types, enter a value for Cf in the adjacent box

Note:

- the program automatically calculates the **Solidity ratio** from the ratio of the section surface area and the panel area.

Cf

Enter a value for the force coefficient **Cf** according to Table 29 if **User defined Cf** was selected in the **Building type** option.

Pressure / velocity

Refer to IS875 - Pressure/velocity 503.

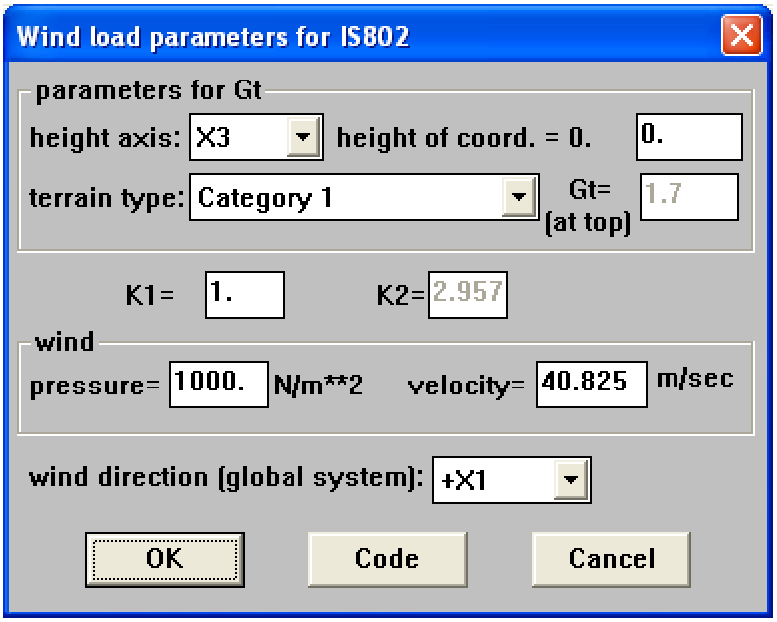
4.19.5.4 IS802 - lattice

The wind force on members in lattice structures is calculated according to Section 9.1.1:

$$F_{wt} = P_d\ C_{dt} \cdot A_e\ G_T$$

where the design wind pressure P_d is calculated according to Section 8.4.

The program determines the projected area perpendicular to the wind direction for each selected beam. The pressure on any member is constant; the program calculates the pressure using the maximum height. The H2, H3 dimensions in the beam property tables are used for this calculation and they must be defined. *Note that H2=H3=0 and hence all lattice wind loads will be zero in all models created prior to Version 9.00*.

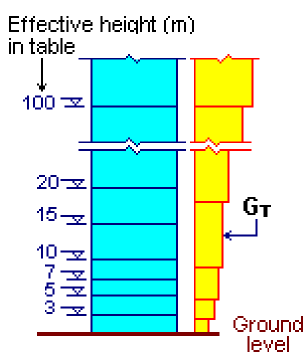


Height axis / Coordinate / Wind direction

Refer to Wind parameters - general 501

Terrain type / Gt

Select one of the Terrain categories from the listbox: **Category 1, Category 2, Category 3**
The program takes the values of **G_T** from Table 6 according to the height of the panel segment:



K1 / Cdt

- **K_1**:
  Enter a value for **K_1**, the risk coefficient, according to the values in Table 2. Note that the value of **K_2** is calculated automatically by the program according to the specified Terrain Category.
- **C_{dt}**:
  The solidity ratio and hence **C_{dt}**, the Drag coefficient, are calculated automatically by the program.

Pressure / velocity

Referring to Code Section 8.4:

$$P_d = 0.6 \cdot V_d^2$$

Enter *either* the wind speed, **V_d**, *or* the Wind pressure, **P_d**; the other value is automatically updated according to the equation.

4.19.5.5 China

Please contact your *STRAP* dealer

4.19.5.6 China - lattice

Please contact your *STRAP* dealer

4.19.5.7 Taiwan

Please contact your *STRAP* dealer

4.19.5.8 Taiwan - lattice

Please contact your *STRAP* dealer

4.19.5.9 UBC 1997

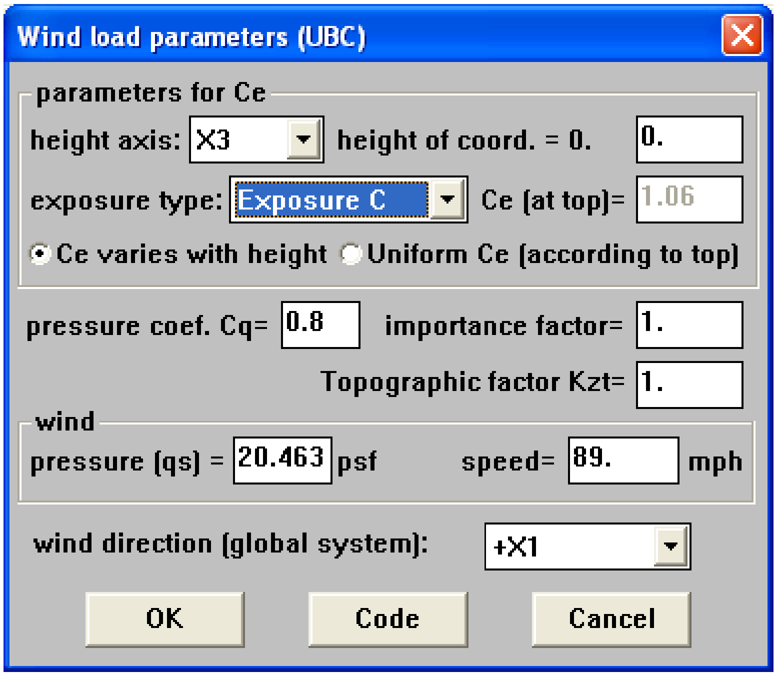
Wind design is based on UBC 1997: Division 3 - Wind Design.

Referring to Section 1620: " Design wind pressures for buildings and structures and elements therein shall be determined for any height in accordance with the following formula:

$$P = C_e C_q q_s I_w k_{zt} \qquad (20\text{-}1)$$

where:

C_e = combined height, exposure and gust factor coefficient (Table 16-G)
C_q = pressure coefficient (Table 16-H)
q_s = wind stagnation pressure at 33 feet (Table 16-F)
I_w = importance factor (Table 16-K)
k_{zt} = topographic factor



Height axis / Coordinate / Wind direction

Refer to Wind parameters - general 501

Exposure type

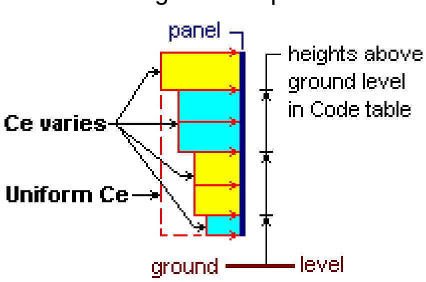
Select one of the following options:

- **Exposure B,C or D**
  The exposure type (refer to Section 16.16) is used to determine the coefficient **Ce** according to Table 16-G.
- **User defined table**
  The program uses the table stored in file WINDUSER.DAT
- **Define Ce**
  Enter a value for **Ce** in the "**Ce(at top)**" edit box.

Ce

Ce varies with height
The program calculates the pressure according to the height gradations in Table 16-G and applies a stepped pressure pattern to the panel.

Uniform Ce (according to top)
The program calculates the pressure at the top of the panel and applies the same pressure over the entire height of the panel



Cq

Enter the Pressure Coefficient, **Cq**, according to Code Table 16-H.

Importance factor

Specify the Wind Importance Factor **Iw** as set forth in Table 16-K.

Topographic factor

The topographic factor, **kzt**, is included in the equation for **P** by other Codes that are similar to the UBC. The default value is 1.00.

Pressure / speed

Referring to Table 16-F, enter **either** the basic wind speed **or** the Wind Stagnation Pressure, **qs**; the other value will automatically be updated according to the Table.

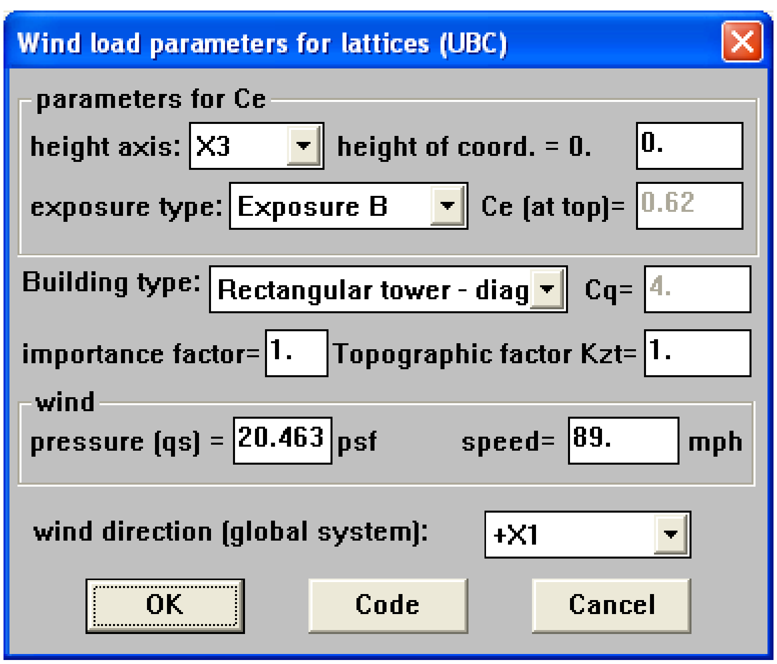
Units:

- basic wind speed - miles per hour (mph)
- wind stagnation pressure, **qs** - psf

4.19.5.10 UBC- lattice

Wind on members in lattice structures is calculated according to UBC, Section 1623:

The program determines the projected area perpendicular to the wind direction for each selected beam. The pressure on any member is constant; the program calculates the pressure using the maximum height. The H2, H3 dimensions in the beam property tables are used for this calculation and they must be defined. *Note that H2=H3=0 and hence all lattice wind loads will be zero in all models created prior to Version 9.00*.



Height axis / Coordinate / Wind direction

Refer to Wind parameters - general 501

Exposure type

Refer to UBC - Exposure type.

Building type

Specify the type of lattice according to Table 16-H -5 (Open frame towers) or enter a value for $\mathbf{C_q}$.

Factors for cylindrical elements are two-thirds of those for flat or angular elements (Table 16-H, Note 8)

Cq

Enter the Pressure Coefficient, $\mathbf{C_q}$, according to Code Table 16-H.

Importance factor

Specify the Wind Importance Factor $\mathbf{I_w}$ as set forth in Table 16-K.

Topographic factor

The topographic factor, $\mathbf{k_{zt}}$, is included in the equation for **P** by other Codes that are similar to the UBC. The default value is 1.00.

Pressure / speed

Refer to UBC - pressure / speed.

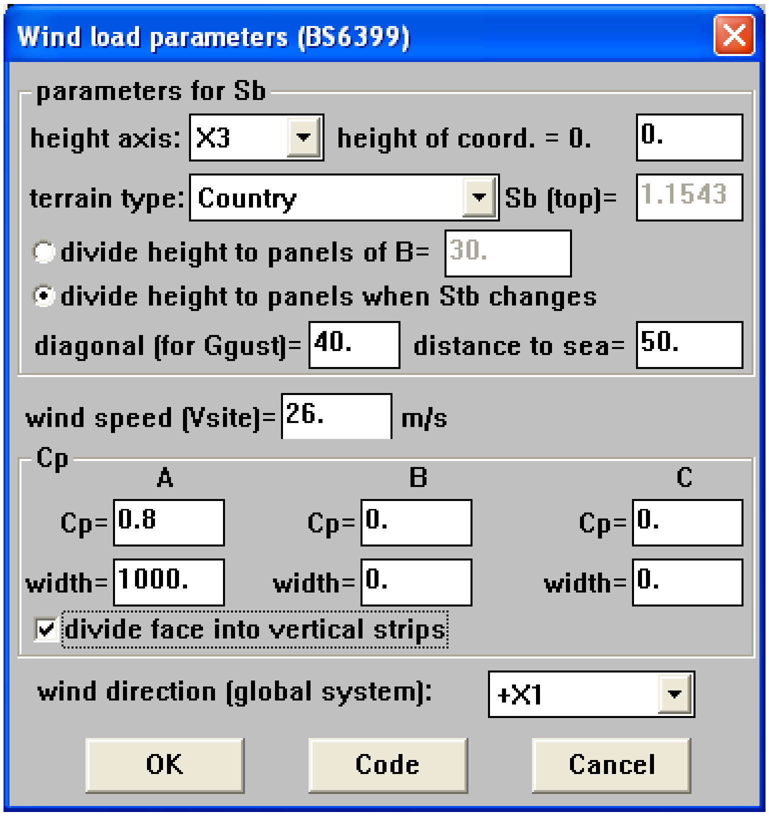
4.19.5.11 BS6399, Part 2

Referring to Clause 3.1.3, the pressure acting on the external surface of a building is calculated as:

$$p_e = q_e \cdot C_p \qquad \text{(Eq. 17)}$$

where:

- $\mathbf{q_e}$ is the reference pressure obtained from Clause 3.1.2.1
  $q_e = 0.613 \cdot V^2_e$ (Eq. 16)
  $V_e = \underline{V}_{site} \cdot S_b$ (Eq. 27)
  is the terrain and building factor obtained from Clauses 3.2.3.2.2 and 3.2.3.2.3
- $\mathbf{C_p}$ is the pressure coefficient obtained from Clause 3.3.



Height axis / Coordinate / Wind direction

Refer to Wind parameters - general 501

Terrain type / Stb

The **Sb** factor is calculated according to the Detailed method in Clause 3.2.3.2:

- For sites in COUNTRY terrain:

$$S_b = S_c \cdot [1 + (g_{GUST} \cdot S_t) + S_h] \qquad \text{(Eq. 28)}$$

- For sites in TOWN terrain:

$$S_b = S_c \cdot T_c \cdot [1 + (g_{GUST} \cdot S_t \cdot T_t) + S_h] \qquad \text{(Eq. 29)}$$

Specify the **Terrain** type; select one of the following:

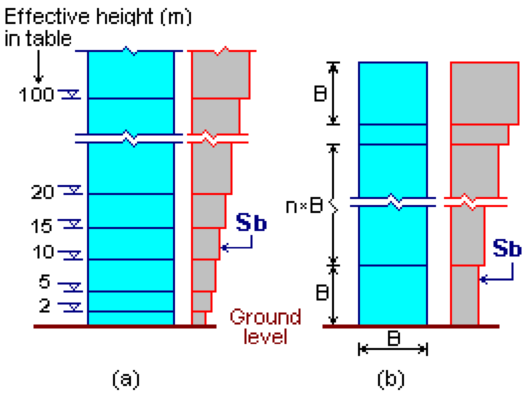
- **Country**, or **Town** and upwind distance from edge of town to site
  Sc, St, Tc, Tt· are calculated by the program according to Code Tables 22 and 23
  Sh is assumed = 0
- **User table**
  The program uses the **Sb** vs. effective height table stored in file WINDUSER.DAT (All other factors are ignored)
- Define Sb
  Enter a value for **Sb** in the adjoining edit box.

Panels

The value of **Sb** varies along the height of the panel. The program selects the heights at which to change the value according to one of the following methods:

divide height to panels of B=
Referring to Code clause 3.2.3.1, the user defines B = maximum horizontal plan dimension and the program applies a uniform pressures on successive panels according to 2.2.3.2 and Figure 11. The reference height for determining the value of **Sb** is the height at the of the panel. Refer to Figure (b) below.

divide height to panels when Sb changes
the program calculates new values at the effective heights listed in Code Tables 22,23. Refer to Figure (a) below.



Ggust

Referring to Code Appendix F, specify the "**Diagonal b**" distance required to calculate the gust peak factor, g_{gust}.

$$g_{gust} = 0.42 \ln (3600/t)$$

$$t = 4.5\, a/V_o \qquad \text{(F.1`)}$$

where V_o is the relevant mean wind speed at height H_r, given by:

- country terrain - $V_o = V_{site}\, S_c$
- town terrain - $V_o = V_{site}\, S_c\, T_c$

Distance to sea

Referring to Code Tables 22 and 23, specify the "**Upwind distance from sea to site (km)**"

Wind speed (Vsite)

Specify the value of **V_{SITE}** as defined in Code clause 2.2.2.

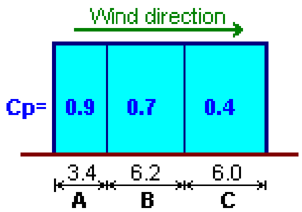
Cp

Define the Pressure Coefficient, **Cp**, for the panel, as outlined in

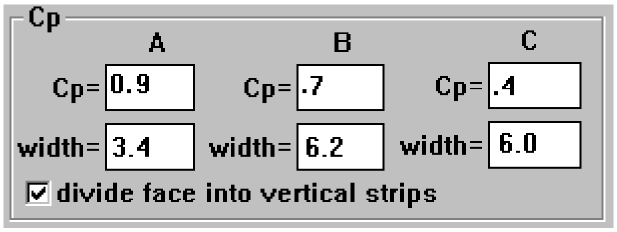
- BS6399: clause 3.3.
- Eurocode 1: section 10

A uniform coefficient may be defined for the entire panel face, or the panel may be divided into vertical strips (refer to BS6399 -Figure 31 or Eurocode 1 - Figure 10.2.3).

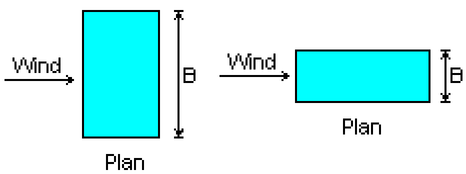
For example:



specify:



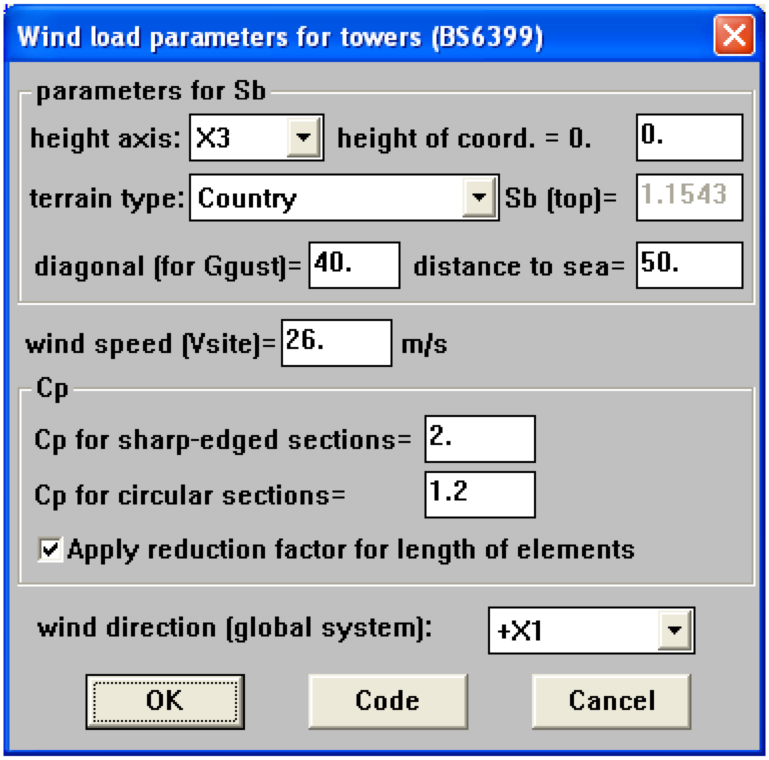
The loaded areas on the side face should be divided into vertical strips from the upwind edge of the face using the scaling length **b** given by: **b = min(B, 2H)**
where **B** = dimension perpendicular to the wind direction



4.19.5.12 BS6399, Part 2 - Lattice

Wind on members in lattice structures is calculated according to BS 6399,Part 2,Section 2.7:

The program determines the projected area perpendicular to the wind direction for each selected beam. The pressure on any member is constant; the program calculates the pressure using the maximum height. The H2, H3 dimensions in the beam property tables are used for this calculation and they must be defined. *Note that H2=H3=0 and hence all lattice wind loads will be zero in all models created prior to Version 9.00*.



Height axis / Coordinate / Wind direction

Refer to Wind parameters - general 501

Terrain type / Stb

Refer to BS6399 - Terrain type 510

Ggust

Refer to BS6399 - Ggust 511

Distance to sea

Referring to Code Tables 22 and 23, specify the "**Upwind distance from sea to site (km)**"

Wind speed (Vsite)

Specify the value of **VSITE** as defined in Code clause 2.2.2.

Cp

Specify the pressure coefficient, **Cp**, according to section 2.7.2 and Table 20.

Apply reduction

☑ the program automatically calculates the reduction factor, κ, according to section 2.7.3 and Figure 25.

Note:

- the program uses L = *STRAP* member length when calculating L/B. The length will be incorrect in the case of combined beams.

- the program ignores the provisions of item 2.7.3 referring to sections that cantilever from the ground, from another plane surface or that span between two planes.

4.19.5.13 SI 414 - 2008

The wind pressures are calculated according to SI 414: Characteristic loads in structures: Wind loads (2008).

The wind pressure, **we**, acting on the external surface of a structure shall be obtained from:

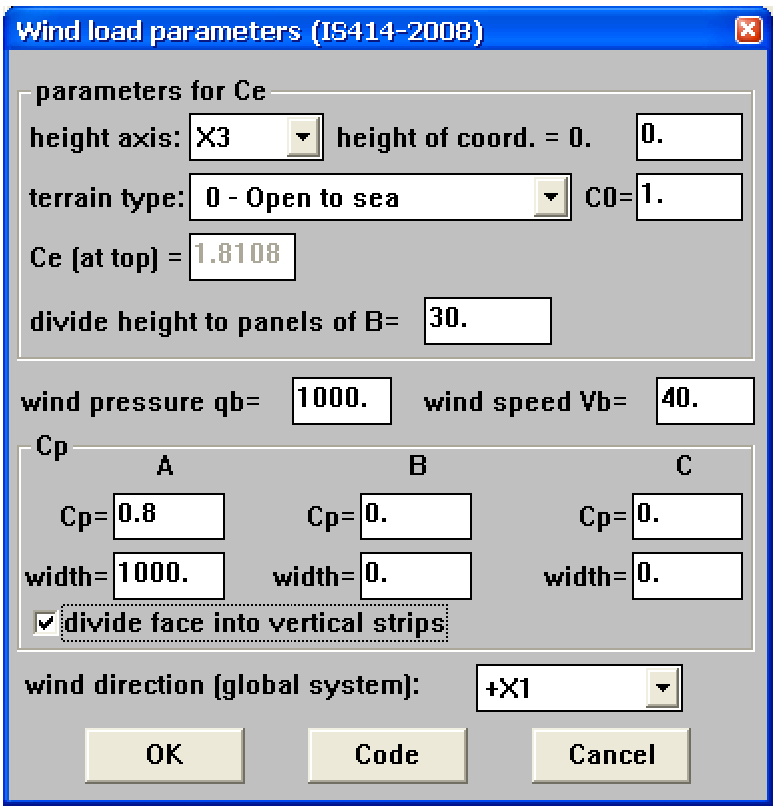
we = qb·ce·cp (4.4)

where:

qb = reference mean wind velocity pressure derived from the reference wind velocity as defined in Code section 3.5.

ce = exposure coefficient accounting for the terrain and height above ground (**z**), given in Code section 5.5

cp = external section pressure coefficient derived from Code section 7.



Height axis / Coordinate / Wind direction

Refer to Wind parameters - general 501

Terrain type / Ct / Ce

The exposure coefficient **Ce** is determined from Code equation (5.9)

$$C_e = C_r^2 \cdot C_o^2 \left[1 + \frac{7k_t}{C_o \cdot \ln(Z/Z_o)}\right]$$

where:

kt = terrain factor as defined in Code section 5.3

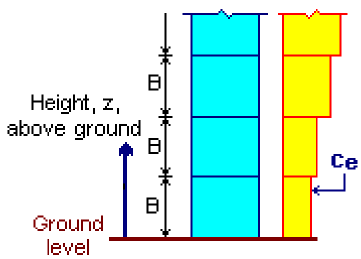
cr = roughness coefficient as defined in Code section 5.2
co = orography coefficient as defined in Code section 5.4
z = height above ground
zo = roughness length

There are three methods for specifying the value for **Ce**: select one of the following options in the "**Terrain type**" list box:

- Terrain categories 0-IV (**Sea**/**Rough/Farmland/Suburban/Urban**) according to Table 5.1:
  The program takes the values of **kt**, **zo**, **zmin** from table 5.1 and calculates **cr** according to Code equation 5.2. **co** is defined by the user in the edit box.
- **User table**
  The program uses the **Ce** vs. effective height table stored in file WINDUSER.DAT 500 (All other factors are ignored)
- Define Ce
  Enter a value for **Ce** in the adjoining edit box

Panels

The value of **Ce** varies along the height of the panel. The program selects the heights at which to change the value according to dimension **B** defined by the user:



q_b / v_b

Referring to Code Section 3.5:

$$q_b = 0.625 \cdot v^2_b \qquad (3.5)$$

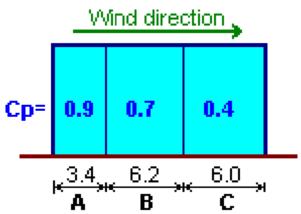
Enter ***either*** the wind speed v_b **or** the Wind pressure, q_b; the other value will automatically be updated according to the equation

Cp

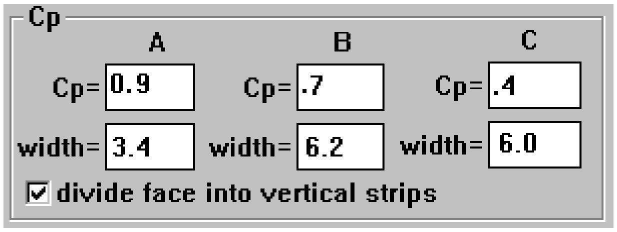
Define the Pressure Coefficient, **Cp**, for the panel, as outlined in Chapter 7.

A uniform coefficient may be defined for the entire panel face, or the panel may be divided into vertical strips (refer to Figure 7.2).

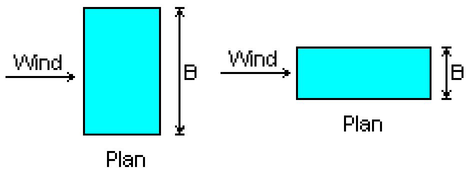
For example:



specify:



The loaded areas on the side face should be divided into vertical strips from the upwind edge of the face using the scaling length **b** given by: **b = min(B, 2H)**
where **B** = dimension perpendicular to the wind direction



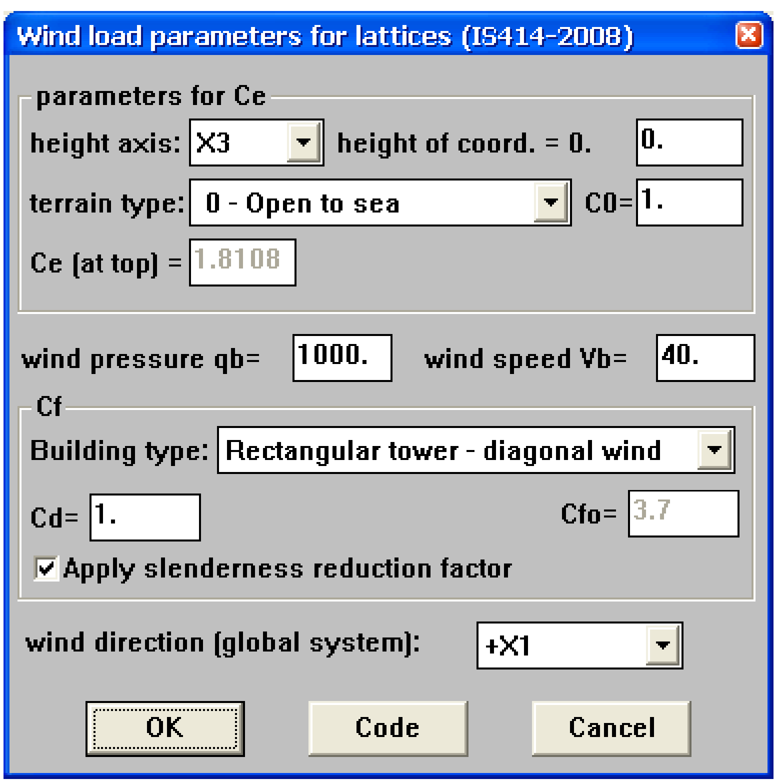
4.19.5.14 SI 414 - 2008 - Lattice

The wind pressures on lattice structures are calculated according to SI 414: Characteristic loads in structures: Wind loads (2008), equation 4.1:

$$F_w = q_b \cdot c_e(z_e) \cdot c_d \cdot c_f \cdot A_{ref}$$

where **cf** is calculated by the program according to sections 7.10 and 7.11.

The program determines the projected area perpendicular to the wind direction for each selected beam. The pressure on any member is constant; the program calculates the pressure using the maximum height. The H2, H3 dimensions in the beam property tables are used for this calculation and they must be defined. *Note that H2=H3=0 and hence all lattice wind loads will be zero in all models created prior to Version 9.00.*



Height axis / Coordinate / Wind direction

Refer to Wind parameters - general 501

Terrain type / Co / Ce

Refer to SI 414 (2008) - Terrain type 514.

q_b / v_b

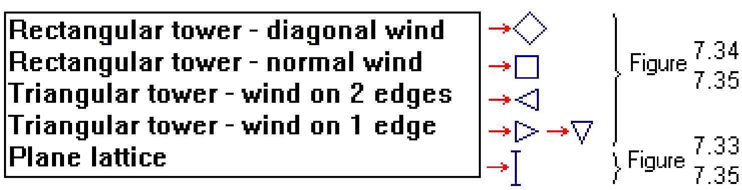
Refer to SI 414 (2008) - q 515 b 515 /v 515 b 515.

Cf

The program calculates the load on individual members of a lattice structure according to Equation (7.11):

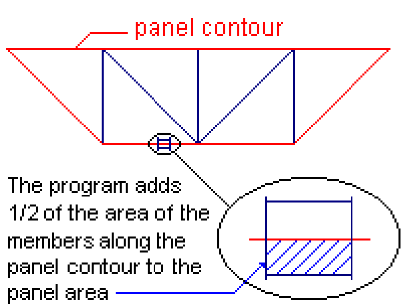
$$C_f = C_{f,o} \cdot \psi_\lambda$$

- ψ_λ, the slenderness reduction factor, is calculated by the program according to Figure 7.19, based on the max/min horizontal and vertical dimensions of the panel contour.
- $C_{f,o}$ is calculated by the program according to Figures 7.33, 7.34 and 7.35 based on:
  - the building type



    Note: for values of φ not on the graphs of Figure 7.35, the program uses the closest line.

  - the solidity ratio, φ, calculated by the program according to Section 7.10.2 as the ratio of the surface area of the members enclosed by the panel to the area of the panel:



  - the Reynolds number, **Re**, required for Figure 7.35 is calculated by the program according to equation 7.4.

Cd

Enter a value for **cd**, the dynamic coefficient, according to Chapter 6.

4.19.5.15 IS414 - 1982

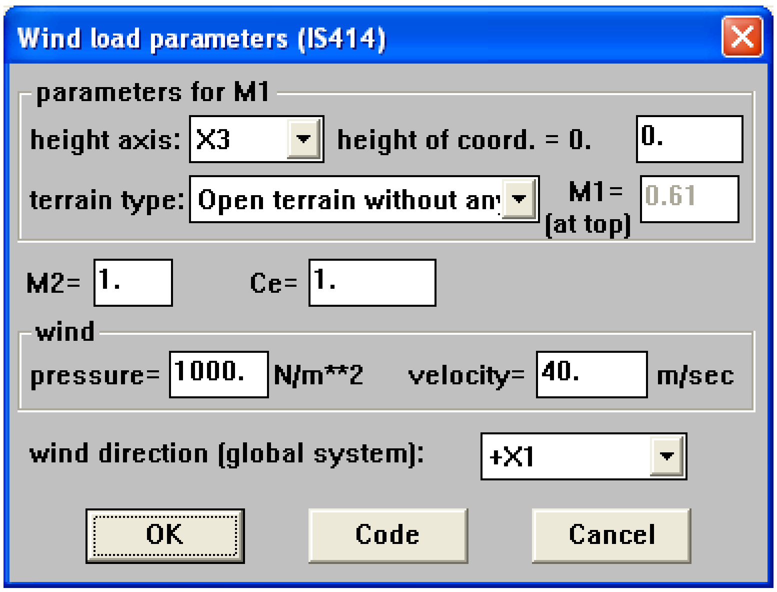
The calculation is based on IS 414 - "Characteristic loads in buildings: wind loads" (1982).

The characteristic wind pressure is calculated from:

W = q·M1·M2·Ce (Equation 1)

where:

q = basic wind pressure = **V²/1.6** N/m² (Equation 4)
V = basic wind speed (m/sec)
M_1 = terrain coefficient according to Code Table 1
M_2 = topographic coefficient according to Code Table 2
Ce = shape coefficient, according to Code section 205.



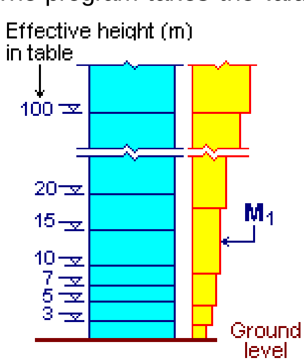
Height axis / Coordinate / Wind direction

Refer to Wind parameters - general 501

Terrain type / M1

There are three methods for specifying the value for **M1**: select one of the following options in the "**Terrain type**" list box:

- Terrain categories **Open terrain without, Open terrain with, Terrain with buildings:**
  The program takes the values of **M1** from Table 1 according to the height **H** of the panel segment.



- **User table**
  The program will use the **M1** vs. effective height table stored in file WINDUSER.DAT instead of Code Table 1 (all other factors are ignored).
- Define M1

Enter a value for **M1** in the adjoining edit box. The value will be uniform over the entire face of the

M2 / Ce

Enter values for:
- **M2** - according to Code Table 2
- **C** - according to Code section 205.

Pressure / velocity

Referring to Code Section 202:

q = 0.625·v²/1.6 (Equation 4)

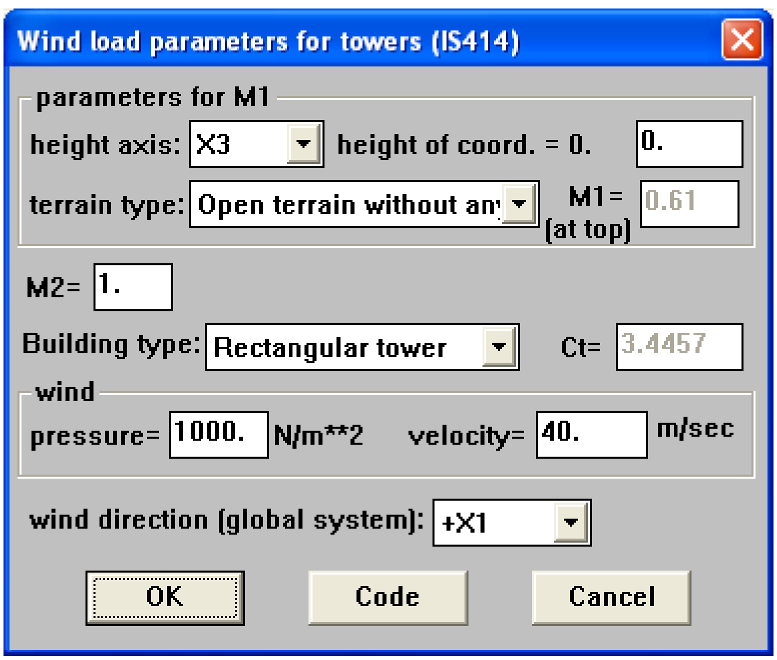
Enter **either** the wind speed, V, **or** the Wind pressure, q; the other value will automatically be updated according to the equation.

4.19.5.16 IS414 - 1982 - lattice

Wind on members in lattice structures is calculated according to IS414, equation 17:

F=q·M1·M2·An·Ct

The program determines the projected area perpendicular to the wind direction for each selected beam. The pressure on any member is constant; the program calculates the pressure using the maximum height. The H2, H3 dimensions in the beam property tables are used for this calculation and they must be defined. *Note that H2=H3=0 and hence all lattice wind loads will be zero in all models created prior to Version 9.00*.



Height axis / Coordinate / Wind direction

Refer to Wind parameters - general 501

Terrain type / M1

Refer to IS414 - Terrain type / M1 518

M2

Specify the topographic factor, **M2**, according to Table 2.

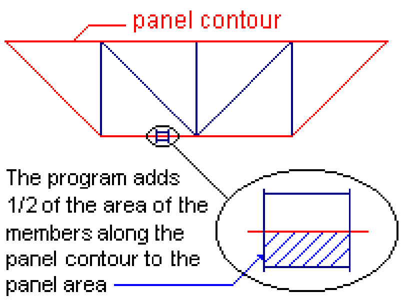
Ct

Enter the value for the shape factor, **Ct**, or specify the factor according to the building type:

• plane lattice:	Table 20 -all sections
• rectangular tower:	Table 23 -round sections (program always assumes "diagonal" direction) Table 22 -other sections (the program increases **Ct** by 1.2 according to Note 3)
• triangular tower:	Table 24 -round sections Table 22 -other sections

Note that the program automatically calculates the **DVs** value used in Tables 20, 22, 23.

The obstruction factor, ϕ, is calculated by the program as the ratio of the surface area of the members enclosed by the panel to the area of the panel:



Pressure / velocity

Refer to IS414 - pressure / velocity 519

4.19.5.17 Eurocode 1

The wind pressures are calculated according to Eurocode 1, Part 2-4: Wind actions (2005).

The wind pressure, **we**, acting on the external surface of a structure shall be obtained from:

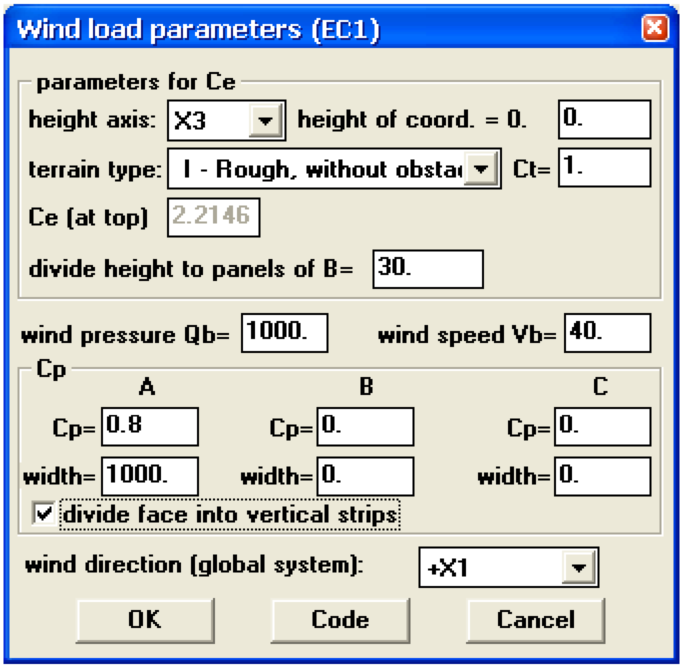
we = qb·ce·cp (5.1)

where:

qb = reference mean wind velocity pressure derived from the reference wind velocity as defined in Code section 4.

ce = exposure coefficient accounting for the terrain and height above ground (**z**), given in Code section 4.5

cp = external section pressure coefficient derived from Code section 17.



Height axis / Coordinate / Wind direction

Refer to Wind parameters - general 501

Terrain type / Co / Ce

The exposure coefficient **Ce** is determined from Code equation 4.8.

$$C_e = c_r^2 \cdot c_t^2 \left[1 + \frac{7k_T}{c_r \cdot c_t}\right]$$

where:

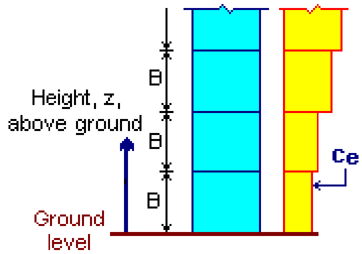
kr = terrain factor as defined in Code section 4.5
cr = roughness coefficient as defined in Code section 4.4
ct = topography coefficient as defined in the Code

There are three methods for specifying the value for **Ce**: select one of the following options in the "**Terrain type**" list box:

- Terrain categories I-IV (**Rough/Farmland/Suburban/Urban**)
  The program takes the values of **kT**, **zo**, **zmin** and ε from table 8.1 and calculates **cr** according to Code equation 8.2. **ct** is defined by the user in the edit box.
- **User table**
  The program will use the **Ce** vs. effective height table stored in file WINDUSER.DAT 500 (All other factors are ignored)
- Define Ce
  Enter a value for **Ce** in the adjoining edit box

Panels

The value of **Ce** varies along the height of the panel. The program selects the heights at which to change the value according to dimension **B** defined by the user:



q_b / v_b

Referring to Code Section 4.5:

$$q_b = 0.625 \cdot v_b^2 \qquad (4.10)$$

Enter ***either*** the wind speed **v_b** ***or*** the wind pressure, **q_b**; the other value will automatically be updated according to the equation

Cp

Refer to [BS6399 - Cp](#) 511

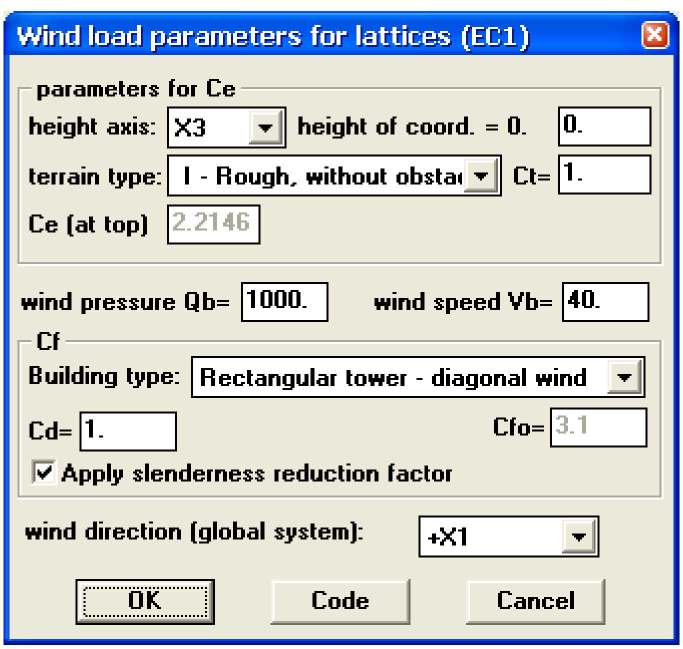
4.19.5.18 Eurocode 1 - lattice

Wind on members in lattice structures is calculated according to Eurocode 1, equation 5.3:

$$F_w = q_p \cdot c_e(z_e) \cdot c_d \cdot c_f \cdot A_{ref}$$

where **c_f** is calculated by the program according to section 7.11.

The program determines the projected area perpendicular to the wind direction for each selected beam. The pressure on any member is constant; the program calculates the pressure using the maximum height. The H2, H3 dimensions in the beam property tables are used for this calculation and they must be defined. *Note that H2=H3=0 and hence all lattice wind loads will be zero in all models created prior to Version 9.00*.



Height axis / Coordinate / Wind direction

Refer to Wind parameters - general 501

Terrain type / Ct /Ce

Refer to EC1 - Terrain type 521.

Qref / Vref

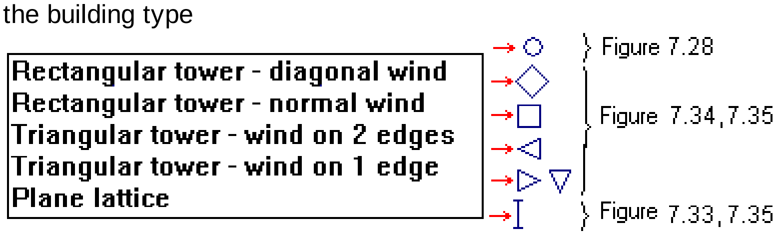
Refer to EC1 - Qref/Vref 522.

Cf

The program calculates the load on individual members of a lattice structure according to Equation (7.25):

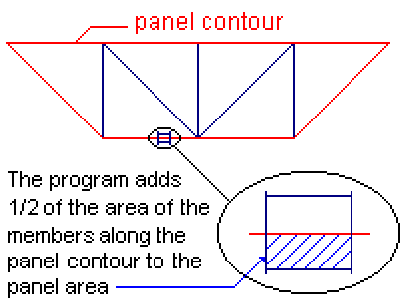
$$C_f = C_{f,o} \cdot \psi_\lambda$$

- ψ_λ, the slenderness reduction factor, is calculated by the program according to Figures 7.36, based on the max/min horizontal and vertical dimensions of the panel contour.
- **$C_{f,o}$** is calculated by the program according to Figures 7.28 and 7.33 to 7.35 based on:
  - the building type



Note: for values of φ not on the graphs of Figure 7.34, the program uses the closest line.

  - the solidity ratio, φ, calculated by the program according to Section 7.26 as the ratio of the surface area of the members enclosed by the panel to the area of the panel:



  - the Reynolds number, **Re**, required for Figure 7.28 is calculated by the program according to equation 7.15.

Cd

Enter a value for **cd**, the dynamic coefficient, according to section 9.3.

4.19.5.19 EC1 - BS N.A.

The wind pressures are calculated according to Eurocode 1, Part 2-4: Wind actions (2005) and the BS National Annex (NA).

The wind pressure, **we**, acting on the external surface of a structure shall be obtained from:

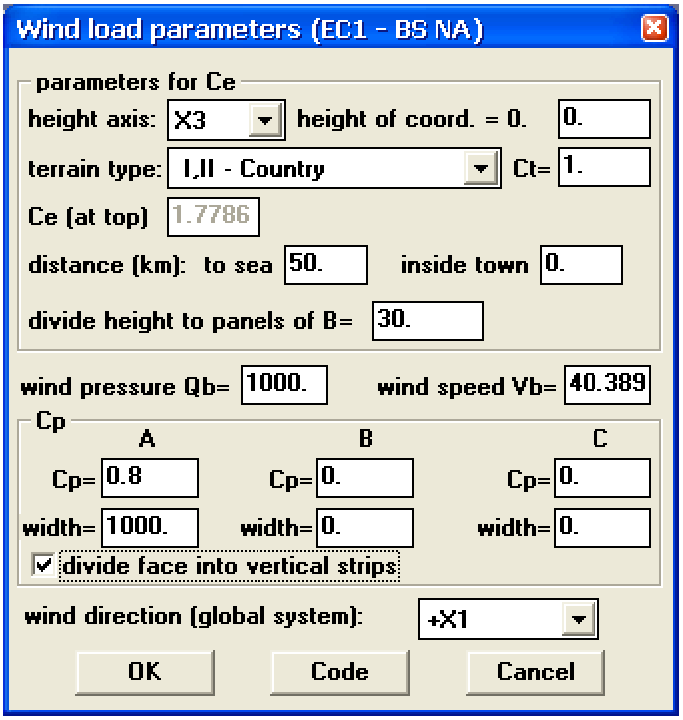
we = qb·ce·cp (5.1)

where:

qb = reference mean wind velocity pressure derived from the reference wind velocity as defined in Code section 4.

ce = exposure coefficient accounting for the terrain and height above ground (**z**), given in Code section 4.5

cp = external section pressure coefficient derived from Code section 17.



Height axis / Coordinate / Wind direction

Refer to Wind parameters - general 501

Terrain type / Co / Ce

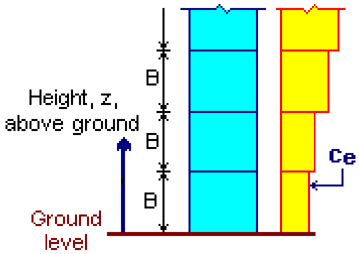
The exposure coefficient **Ce** is determined from Figure NA.7 according to the distance from the sea. For sites in "Town terrain" **Ce** is also modified by the "turbulence correction factor" $\mathbf{k_{L,T}}$ taken from Figure NA.8. **Ce** is also modified by the topography coefficient $\mathbf{c_t}$ defined by the user.

There are three methods for specifying the value for **Ce**: select one of the following options in the "**Terrain type**" list box:

- Terrain categories 0 (sea), I-II (country), III-IV (town).
  The program takes the values from Figures NA.7 and NA.8. $\mathbf{c_t}$ is defined by the user.
- **User table**
  The program will use the **Ce** vs. effective height table stored in file WINDUSER.DAT 500 (All other factors are ignored)
- Define Ce
  Enter a value for **Ce** in the adjoining edit box

Panels

The value of **Ce** varies along the height of the panel. The program selects the heights at which to change the value according to dimension **B** defined by the user:



q_b / v_b

Referring to Code Section 4.5:

$q_b = 0.613 \cdot v^2_b$ (NA.2.18)

Enter ***either*** the wind speed **v_b** ***or*** the wind pressure, **q_b**; the other value will automatically be updated according to the equation

Cp

Refer to BS6399 - Cp 511

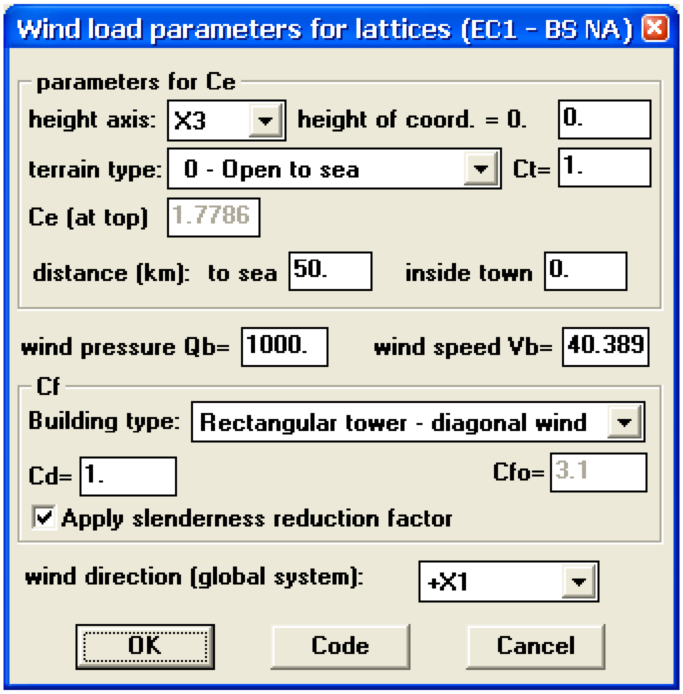
4.19.5.20 EC1 - BS N.A. - lattice

Wind on members in lattice structures is calculated according to Eurocode 1, equation 5.3 and the BS National Annex (NA):

$F_w = q_p \cdot c_e (z_e) \cdot c_d \cdot c_f \cdot A_{ref}$

where **c_f** is calculated by the program according to section 7.11.

The program determines the projected area perpendicular to the wind direction for each selected beam. The pressure on any member is constant; the program calculates the pressure using the maximum height. The H2, H3 dimensions in the beam property tables are used for this calculation and they must be defined. *Note that H2=H3=0 and hence all lattice wind loads will be zero in all models created prior to Version 9.00*.



Height axis / Coordinate / Wind direction

Refer to Wind parameters - general 501

Terrain type / Ct /Ce

Refer to EC1 - Terrain type 521.

Qref / Vref

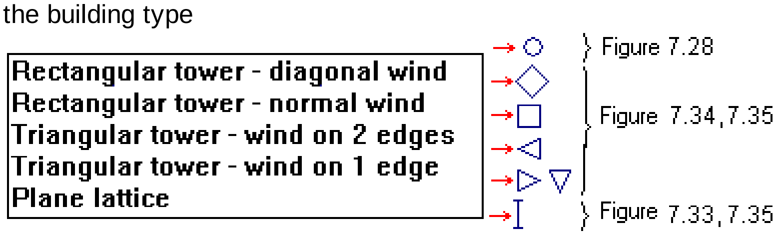
Refer to EC1 - Qref/Vref 522.

Cf

The program calculates the load on individual members of a lattice structure according to Equation (7.25):

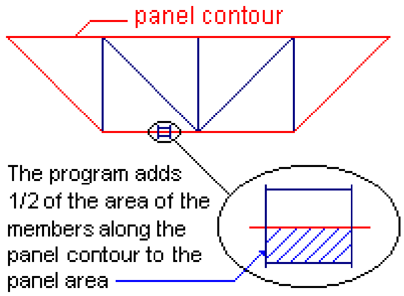
Cf = Cf,o·ψ_λ

- ψ_λ, the slenderness reduction factor, is calculated by the program according to Figures 7.36 and Table NA.6, based on the max/min horizontal and vertical dimensions of the panel contour.
- **Cf,o** is calculated by the program according to Figures 7.28 and 7.33 to 7.35 based on:
  - the building type



  Note: for values of φ not on the graphs of Figure 7.34, the program uses the closest line.

  - the solidity ratio, φ, calculated by the program according to Section 7.26 as the ratio of the surface area of the members enclosed by the panel to the area of the panel:



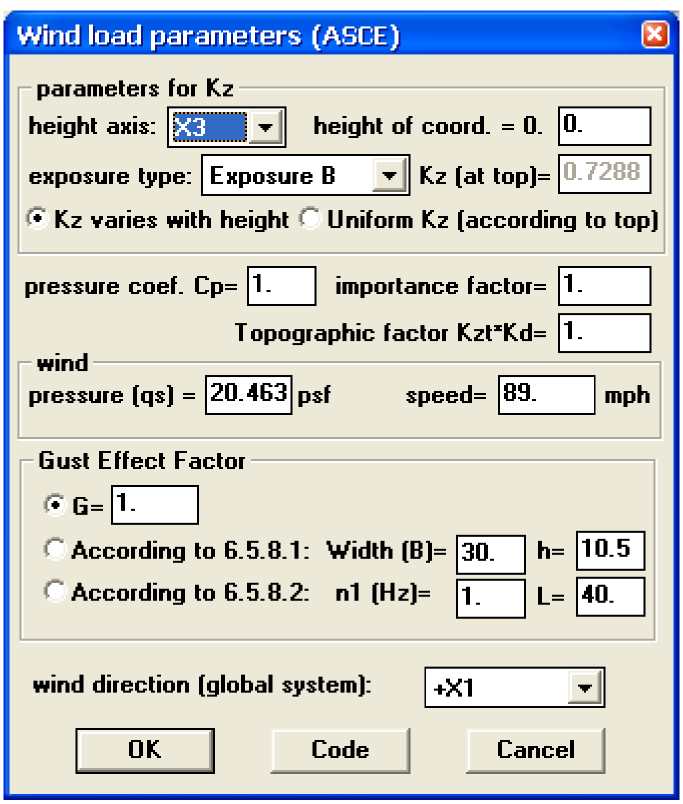
- the Reynolds number, **Re**, required for Figure 7.28 is calculated by the program according to equation 7.15.

Cd

Enter a value for **cd**, the dynamic coefficient, according to section 9.3.

4.19.5.21 ASCE

The wind pressures are calculated according to ASCE/SEI 7-10: "Minimum Design Loads for Buildings and Other Structures".



Height axis / Coordinate / Wind direction

Refer to Wind parameters - general 501

Exposure type

Select type B, C or D according to 26.7.3

Kz

Enter the value of K_z - the velocity pressure exposure coefficient - at the top of the model, according to 27.3 and Table 27.3-1. The factor can be either uniform over the height of the model or vary linearly to zero at the base.

Cp

C_p = the external pressure coefficient, according to Figures 27.4-1, 27.4-2, 27.4-3.

Importance factor

*Kzt*Kd*

K_{zt} = topographic factor. Refer to 26.8.2.

K_d = wind directionality factor. Refer to 26.6.

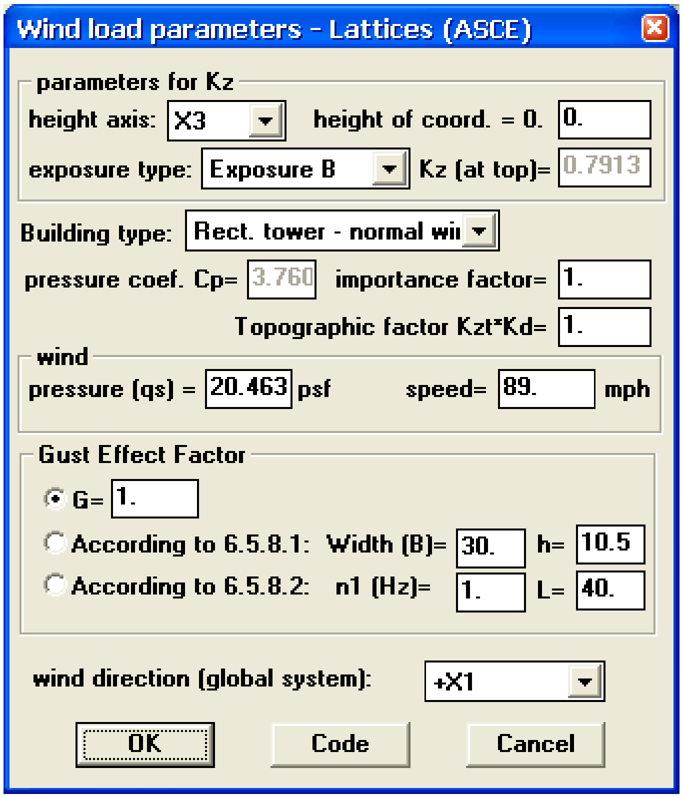
qs

Speed

Gust effect factor

4.19.5.22 ASCE - lattice

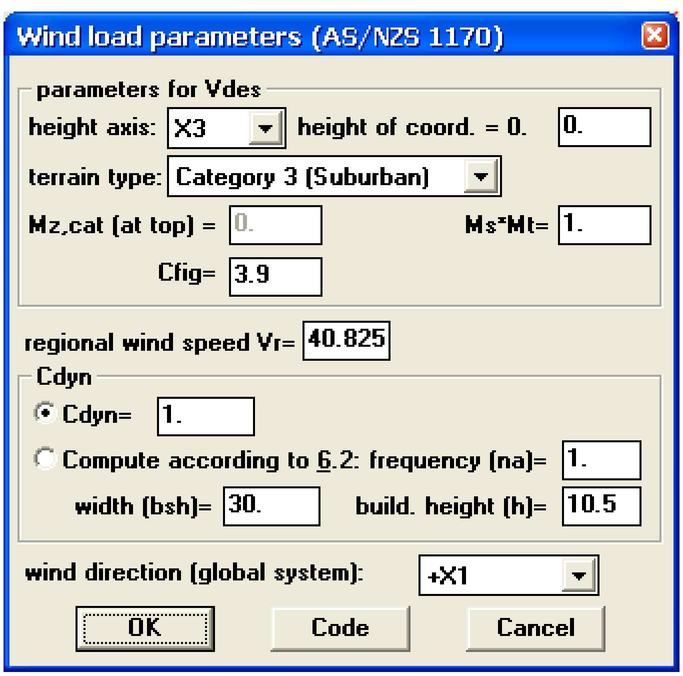
The wind pressures are calculated according to ASCE/SEI 7/05: "Minimum Design Loads for Buildings and Other Structures".



Height axis / Coordinate / Wind direction

Refer to Wind parameters - general 501

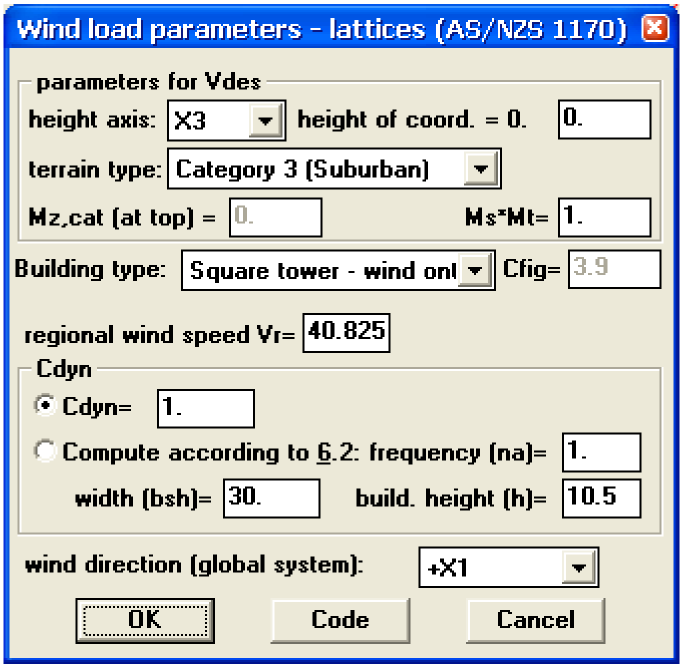
4.19.5.23 AS/NZS 1170



Height axis / Coordinate / Wind direction

Refer to Wind parameters - general 501

4.19.5.24 AS/NZS 1170 - lattice



Height axis / Coordinate / Wind direction

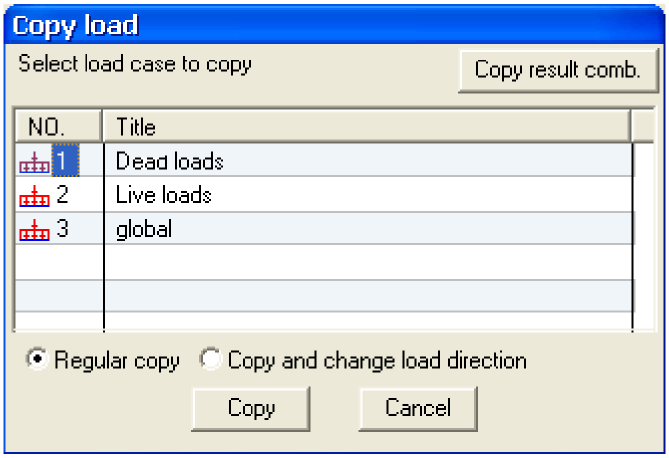
Refer to Wind parameters - general 501

4.20 Copy a load case

Copy an entire existing load case into a new load case or create a load case by copying a load combination defined in the result module.

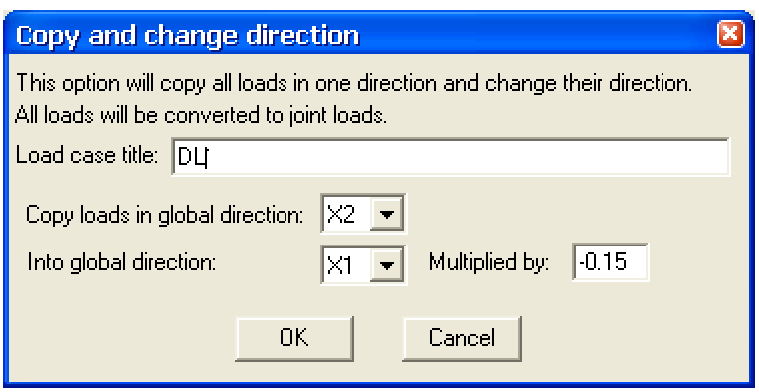
Copy an existing load case

Select the load case from the list:



Regular copy
All load commands in the selected case are copied to the new one.

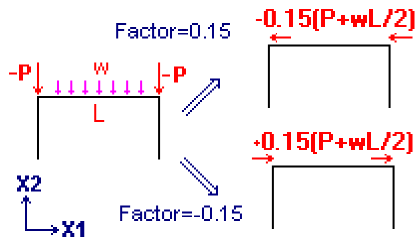
Copy and change load direction
The loads in ***only one global direction*** may be copied and are converted to joint loads in the new load case. Select the global direction to be copied and the global direction of the copied loads.



The copied loads may be multiplied by a factor. In the example above, all vertical loads are converted to horizontal loads and multiplied by -0.15

Note:

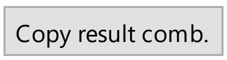
- all loads are converted to joint loads (based on the original applied loads), i.e. all beam loads are converted to joint loads.
- the program maintains the sign of the loads. For example:



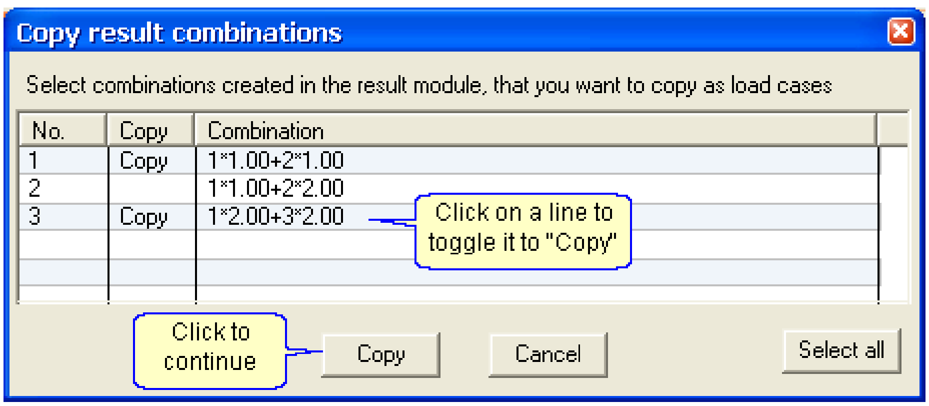
Copy a result combination

This option is identical to the Combine Loads 464 option - a new load case is created from a combination of existing load cases. The combinations are copied from the *STRAP* results module, where the definition of multiple combinations is much simpler.

To copy the combinations:

- click 

- the program displays a list of the combinations defined in the results module:

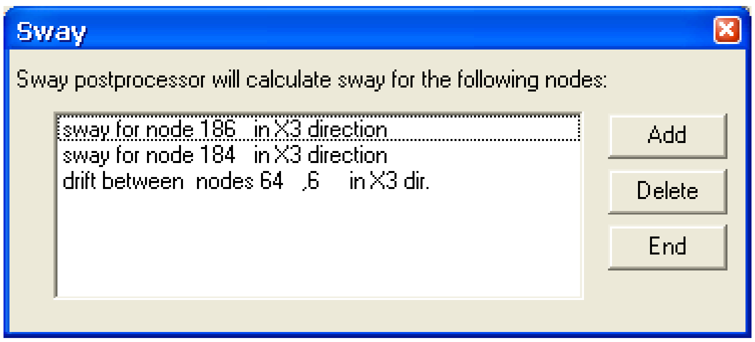


4.21 Sway

The sway/drift control module in the Steel postprocessor selects new beam sections in order to reduce the sway at specified nodes to user-defined values in any global direction. Similarly, the drift between two specified nodes may also be limited.

Specify the nodes and global directions where the Steel postprocessor will check the sway/drift (the actual sway/drift limits are defined in the Steel postprocessor).

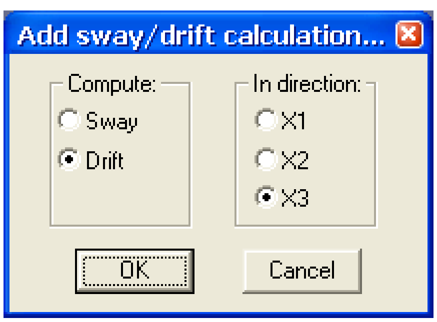
The program automatically creates unit load cases at the selected nodes (note that these load cases will not be displayed when **Revise** or **Output** are selected).



The list box at the centre of the menu displays the sway/drift nodes already defined. Select:

Add

Define new sway/drift nodes:



- ***Compute***

  Sway
  Limit the deflection at a specified node in a specified global direction

  Drift
  Limit the relative deflection between two specified nodes in a specified global direction

- ***Direction***
  Specify the global direction for the sway/drift calculation at the selected nodes

Select the sway/drift node(s) using the standard node selection option.

Delete

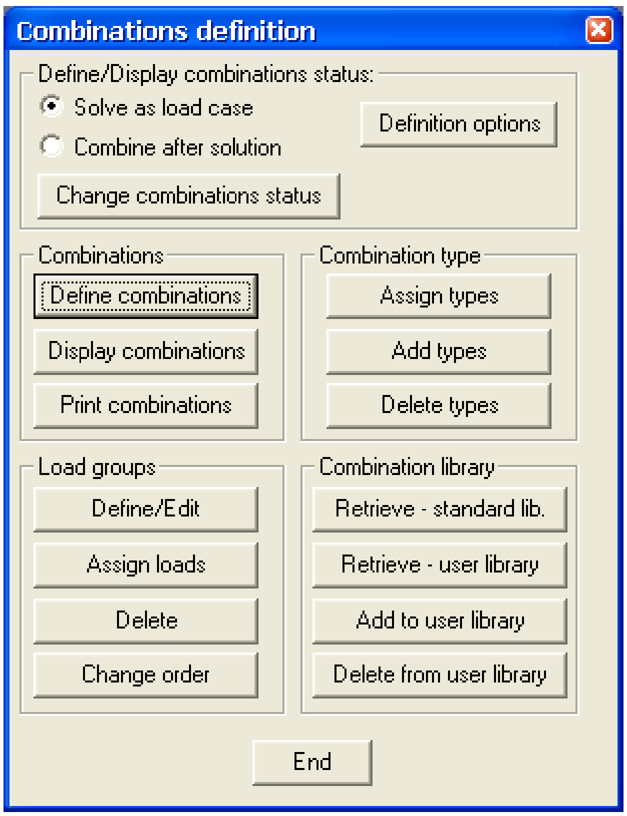
Delete a defined sway/drift check. Highlight the relevant line in the list box (single click) and click the **Delete** button; the line will be erased.

4.22 Combinations

Use this option to define combinations of individual load cases.

Note:

- combinations defined here can be calculated ***either*** during the solution ***or*** in the Results module, according to the "status 534" specified by the user in this menu.
- It is strongly recommended that combinations be calculated ***after*** the solution rather than when solving the model as results can be obtained for new or revised combinations without solving the model again, however -
- The rules of superposition do not apply for models with non-linear elements. Therefore, load combinations for models with P-Delta, tension/compression only members, unidirectional springs, etc, ***must be defined here***, rather than after the solution.
- Refer to Combinations - general for a detailed explanation on groups, types and library.

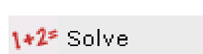


Combination status

Specify the "combination status" before selecting 

:

Solve as load case

The defined combinations will be solved when 

is selected.

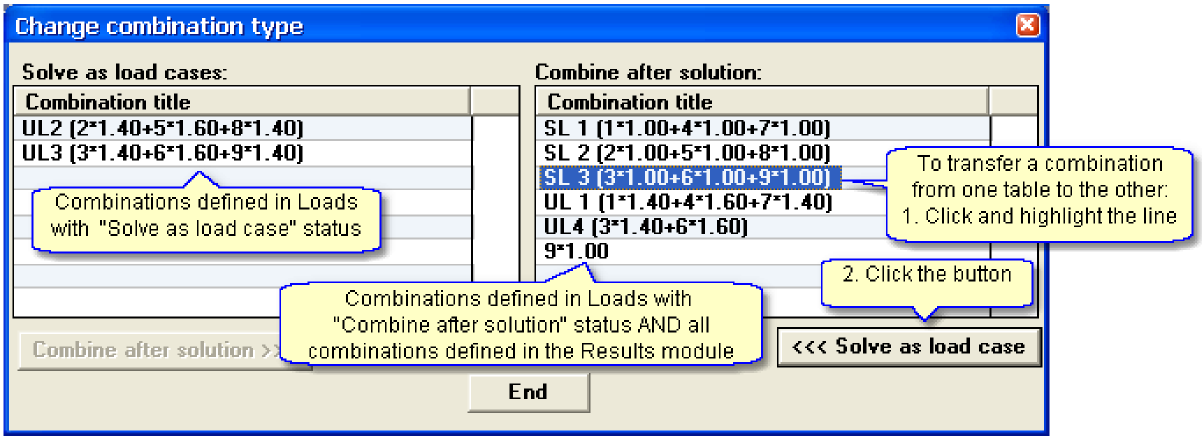
Note:

- Combinations solved as a load case can be included in combinations defined in Results by adding them to a group and then including the group in the results combination.

Combine after solution

The combinations are not solved, but are calculated in the results module from the results of the solved load cases.

Note that combinations with both types of status may be defined; click on Change combinations status to view the list of defined combinations and change the status of any combination:



4.22.1 General

To display a demo video that explains how to define combinations (in the identical option in the Results module):

- *click on* *to start the video*
- *then click on* *to enlarge the display.*

- Each "Combination" is defined as a combination of load cases, each of which is multiplied by a factor.
- "Groups" of load cases may also be defined. If a group is added to a combination definition, the program will either:
  - automatically generate a separate combination for each load in the group, or
  - add the sum of the load cases in the group to the combination.
- Combinations may also be retrieved from a "Combination Library":
  - Combinations in the Library are defined with Groups, e.g. 1.4*Dead+1.6*Live
  - load cases are assigned to these groups by the user after the combinations are retrieved from the library.
- Combinations defined for the current model may be added to the combination library for use in other models.

Note:
- For combinations defined with a Group: the program will not generate the combinations when there are no load cases in the included group.

Library:

Standard combinations need not be redefined in every model; standard combinations containing groups may be stored in the library. For example:
- 1.4*Dead + 1.6*Live
- 1.2*Dead + 1.2*Live + 1.2*Wind
- etc.

These standard combinations may be from the Library; load cases are then assigned by the user to the "Dead", "Live" and "Wind" load groups.

There are two different libraries: the "Program library" and the "User library".

- the Program library is supplied with *STRAP* and contains the standard combinations of Dead, Live, Wind and Seismic loads as listed in all major design codes. The file (COMB.DAT) is located in the program folder.
- The User library contains combinations defined by the user and saved to a library. The file is also called COMB.DAT and the files is always created in the current working folder (i.e. there can be more than one User library file, each in a different folder).
- update the Program library by manually copying the COMB.DAT file from the program folder to a working folder. Revise the library then copy the file back to the program folder.

Examples:

- **Groups**:
  The following load combination is required: *1.4*Dead + 1.6*Imposed + 1.6*Crane*

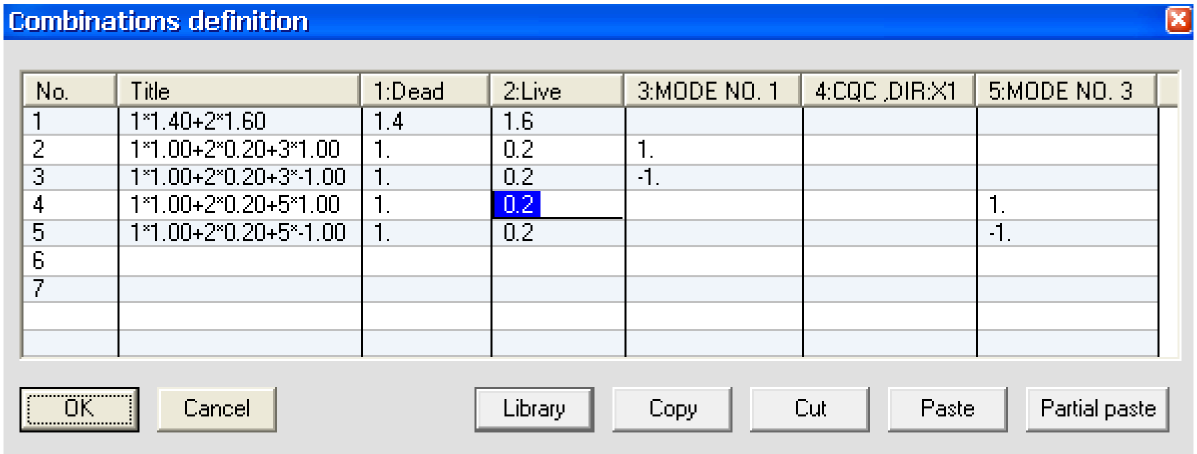
  but there are 5 different load cases with Crane loads, each corresponding to a different point of application of the same crane reaction load.

  Instead of defining 5 separate combinations, the 5 crane loading cases may be included in a Group (called "Crane"); when the combination is created the program automatically generates a separate combination for ***each*** of the five loads in the group, i.e.
  - 1.4*Dead + 1.6*Imposed + 1.6*Crane (1)
  - 1.4*Dead + 1.6*Imposed + 1.6*Crane (2)
  - etc.

4.22.2 Define / revise

There are three methods to create combinations:

- manually type in load factors for the load cases in a combination. Type the factors in the appropriate cells in the following table. There is a column for each load case and for each load group already defined (you may have to scroll horizontally if there are many cases). Combinations may also be copied from the Clipboard.
- retrieve combinations from a library. The program creates Groups from those listed in the library combinations and automatically prompts you to assign load cases to these groups. Note that combinations can only be retrieved from the Program library in this option (and not from the User library).
- copy combination definitions from the clipboard. Refer to copy/cut/paste 538



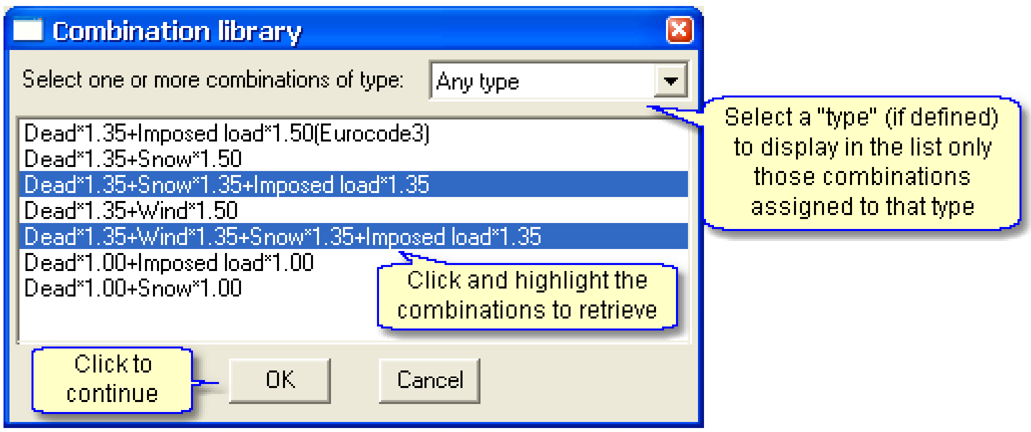
No.	Title	1:Dead	2:Live	3:MODE NO. 1	4:CQC ,DIR:X1	5:MODE NO. 3
1	1*1.40+2*1.60	1.4	1.6			
2	1*1.00+2*0.20+3*1.00	1.	0.2	1.		
3	1*1.00+2*0.20+3*-1.00	1.	0.2	-1.		
4	1*1.00+2*0.20+5*1.00	1.	0.2			1.
5	1*1.00+2*0.20+5*-1.00	1.	0.2			-1.
6						
7						

Define a combination manually:

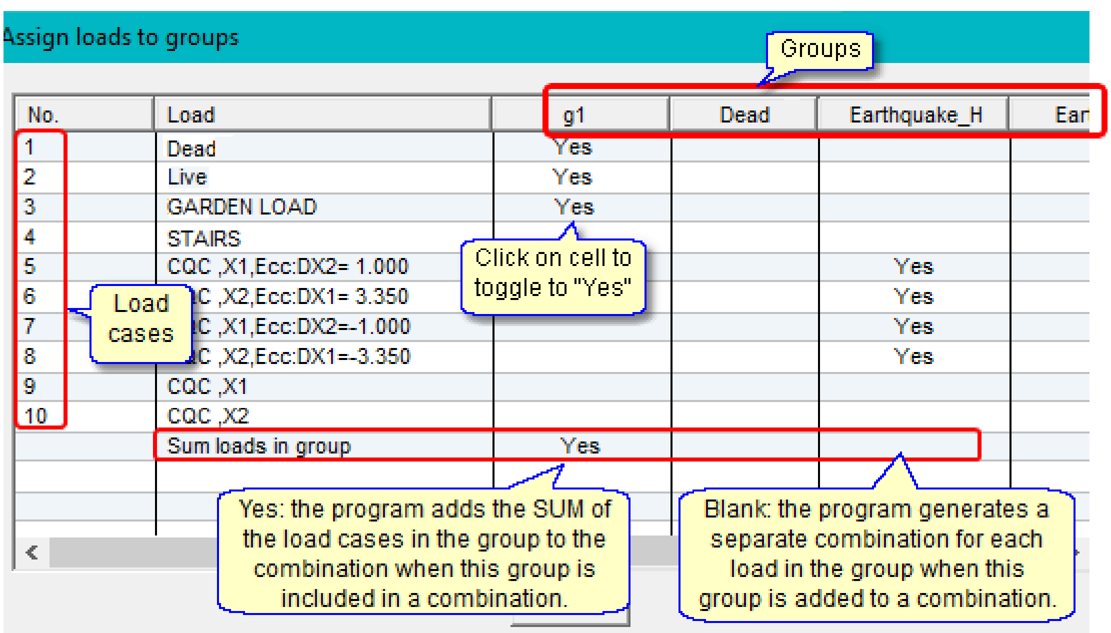
- move the arrow into the appropriate cell and click the mouse.
- type in the load factor and press [Enter]; the cursor will move to the next cell in the row.
- move to the next row after all factors have been defined. The program automatically generates a default title for the combination. The title may be edited at any time, but an edited title will not be automatically updated if the loads or factors in the combination are modified later.

Library:

- click the Library button at the bottom of the screen.
- select the combinations:



- The program checks whether the Groups in these combinations have been defined, i.e. "Dead", "Snow", "Imposed load". If not, the program prompts to assign load case to the groups:



- The program displays an updated "Combinations definition" list, with all new combinations added.

Note:

- the program automatically uses the "Generate a combination for each load in the group" option. To revise to the "Include the sum of all loads in the combination" option, use the Groups - Add/revise 541 option.
- a combination will be generated if no load cases are assigned to ***at least one*** of the groups in the combination definition, e.g. you can retrieve the snow load combinations along with the others but they will be ignored if you don't define snow load cases.
- to delete combinations, copy combinations or rearrange the combination list, refer to copy/cut/paste 538

4.22.2.1 Cut/Copy/paste

Use these options to:

- delete combinations
- copy and paste combinations
- copy selected load cases from one combination to another
- rearrange the combination list
- copy combination definitions from the clipboard

[Cut] - delete a combination:

- place the anywhere on a combination line and click the mouse
- click the Cut button

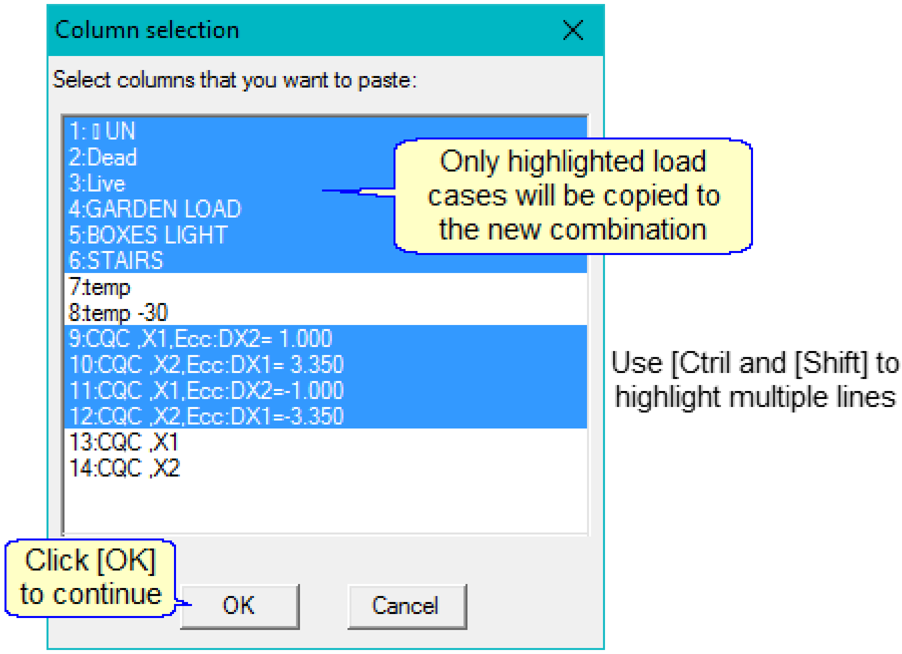
[Copy] and [Paste] a combination:

- highlight one or more combinations in the list.
- click the Copy button
- place the anywhere on the line where the copies are to be written and click the mouse (if you select a line with an existing combination, then the copies will be inserted below this line).
- click the Paste button

[Partial paste] - copy selected loads

Copy combinations but only selected loads; the loads can either create a new combination or overwrite existing combinations:

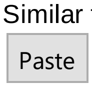
- highlight one or more combinations in the list.
- click the Copy button
- place the anywhere on the line where the copies are to be written and click the mouse
- click the Partial paste button.
- Select the loads to be included in the new or revised combinations:



Note
- if the selected location is an existing combination, the factors for the selected load cases will overwrite the existing factors.
- if the selected location is at the end of the list, new combinations (with the selected load cases only) will be generated.

Rearrange the combination list:

Similar to "Copy a combination", except click 

after selecting the combination to be moved, then 

at the new location..

Copy a combination from the clipboard:

- type the combination definition in a program such as "Notepad" in the format:

TITLE tit (optional)
lc1 f1 lc2 f2 ....lcn fn..G1 fg1 ...... Gn fgn

where:
tit = combination title string. The program will create a default title if this line is omitted
lcn = load case number
fn = factor for load case 'n'
fgn = factor for group 'n'

Example:
for a combination 1.4*load case 1 + 1.6 * load case 3 + 1.2 * group 2, titled "Dead + Live + Group 2", type:
TITLE Dead + Live & Group 2
1 1.4 3 1.6 G2 1.2

- Highlight the commands (click and hold the mouse, drag the cursor), then select "**Edit**", "**Copy**" in the menu bar
- Press [Alt][Tab] to return to *STRAP*

- place the anywhere on the line where the command is to be written and click the mouse (if you select a line with an existing combination, then the command will be inserted above the line).
- click the Paste button

Note that multiple commands may be "cut and pasted" at the same time.

4.22.3 Groups

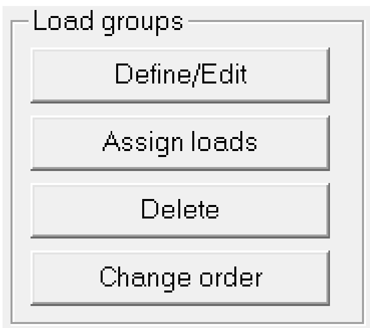
GROUPS of load cases may also be defined. Groups may be defined so that if a group is added to a combination definition:

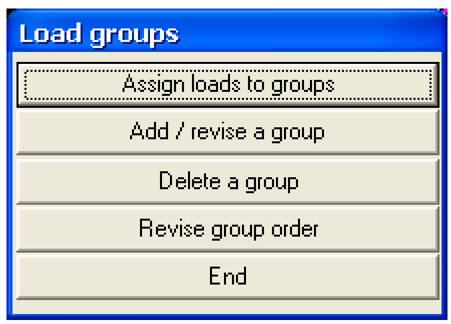
- the program automatically generates a separate combination for ***each*** load in the group, or -
- the program automatically adds the ***sum*** of the load cases in the group to the combination.

Note:

- groups are automatically created when combinations are retrieved from the combination library 543.
- Combinations solved as a load case can be included in combinations defined in Results by adding them to a group and then including the group in the results combination.

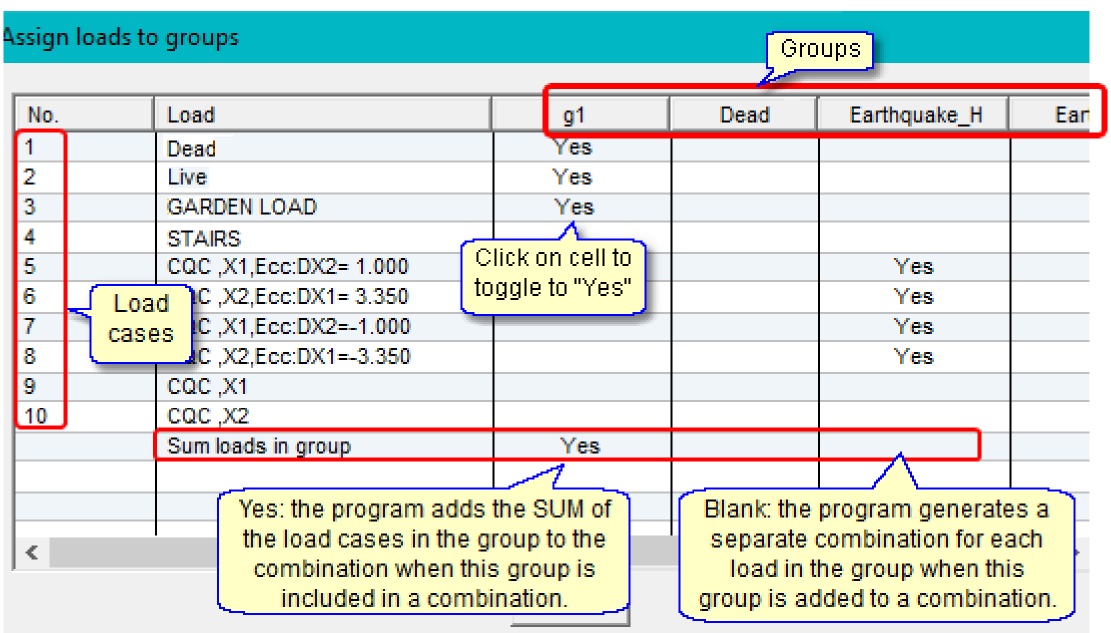
Loads: Results:





Assign loads to groups

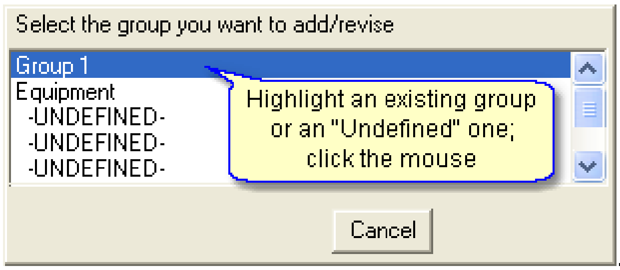
Specify which of the defined load cases belong to each of the groups:



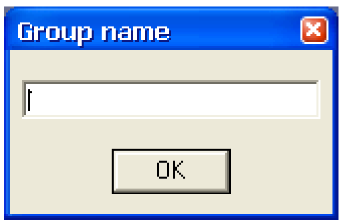
Add (define)/revise(edit) a group

Define a new group or revise an existing group.

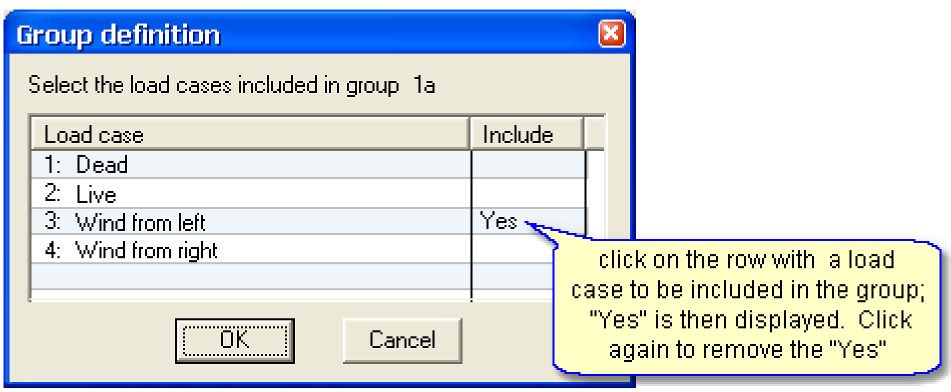
- The program displays a list of the groups. For example:



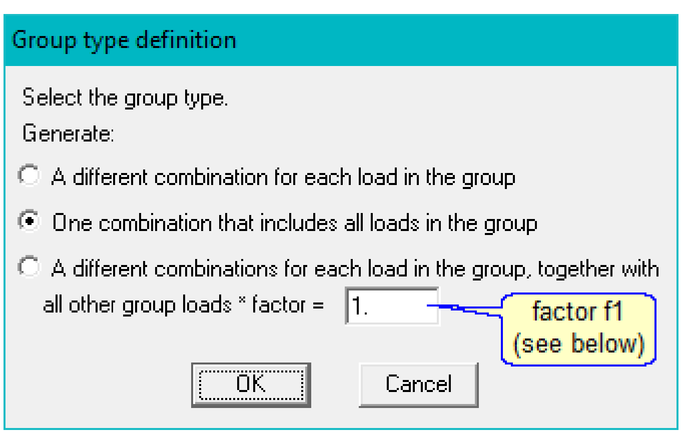
- Enter/revise the Group title.



- The program then displays a list of the load cases. For example:



- Specify the combination method for the group:



A different combination for each load in the group

the program automatically generates a separate combination for ***each*** load in the group when this group is added to a combination. If there are ***n*** load cases in the group then ***n*** combinations will be generated. The load case factor is specified when the combination is created

One combination that includes all loads in the group
the program automatically adds the ***sum*** of the load cases in the group to the combination when this group is added to a combination. Only ***one*** combination is generated. The load case factor for the sum is specified when the combination is created

A different combination for each load in the group .....
the program automatically generates ***n*** combinations if there are ***n*** load cases in the group. ***All of the load cases are included in the all of the combinations***. One of the group loads in each combination is increased by ***f2*** specified at the time the combination is created, all the others by ***f1*** specified in this menu. For example - a group with cases ***lc1, lc2, ... , lcn***; the following combinations are created:

- **lc1(*f2*)+lc2(*f1*)+lc3(*f1*)+ .... +lcn(*f1*)**
- **lc1(*f1*)+lc2(*f2*)+lc3(*f1*)+ .... +lcn(*f1*)**
- etc.

Delete a group

The program displays a list of groups; select the one to delete.

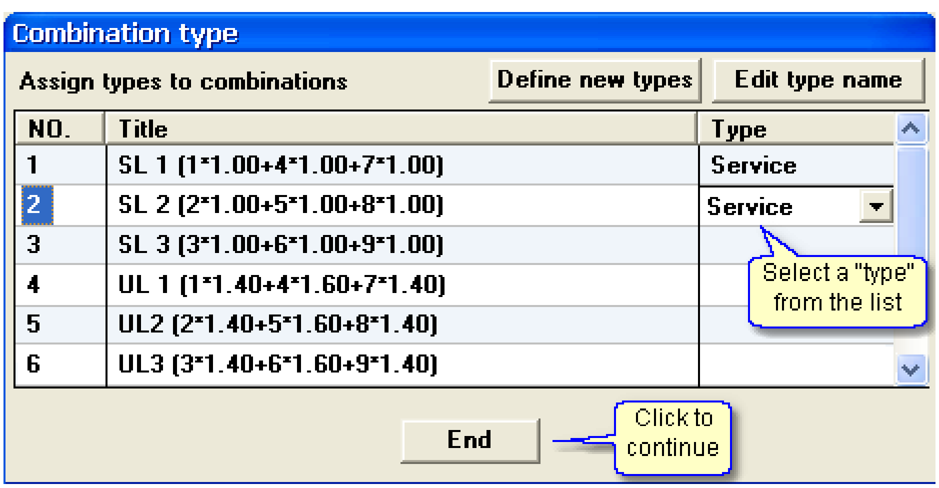
Revise group order

Rearrange the order of the groups. Use the mouse/arrow keys to highlight the group that you want to move; click the mouse. Select the new location in the list - the group will be placed ***before*** this group - and click the mouse; the program displays the group list in the revised order.

4.22.4 Types

Each combination can be assigned with a "type", e.g. Service, Factored, Seismic, etc. Result envelopes can be displayed for a selected type only. For example, display deflections only for "service" combinations.

- select Define new types and define the Type names.
- the program displays a list of combinations:



4.22.5 Library

Most models use standard load combinations listed in the Codes, e.g. **1.4*Dead+1.6*Live**. The library options allows standard combinations to be retrieved from a Library.

Two libraries are available:

- **User library** - a list of combinations created and maintained by the user. The file (COMB.DAT) is provided with the program and must be located in the current folder.
- **Program library** - a list of combinations required each of codes supported by the program, e.g. Eurocodes, ACI, CSA, etc. Cannot be edited by the user

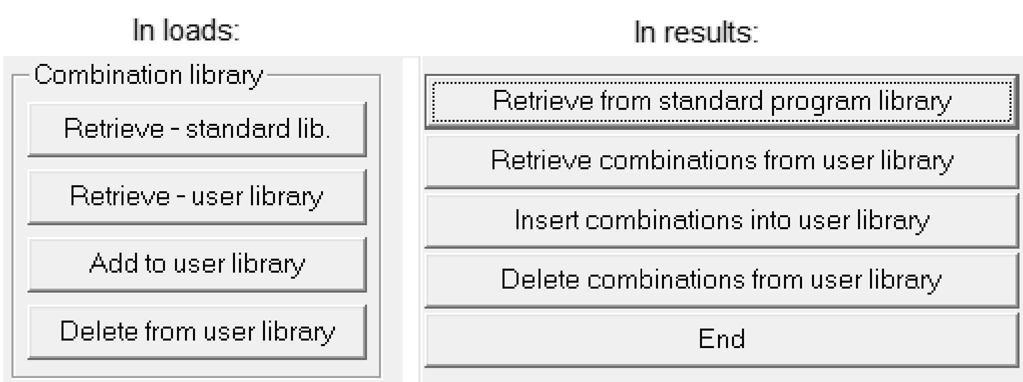
The combinations in the library are normally combinations of load ***Groups***. e.g. for "1.4*Dead+1.6*Live", ***Dead*** and ***Live*** represent load groups.

The procedure for using the library to define combinations in a particular model is as follows:

- select Retrieve ... from the Standard/program or the User library.
- the program automatically creates ***groups*** with the name in the combinations. e.g. "Dead" and "Live" in the example above.
- assign load cases to the groups

Note:

- a combination defined in any model may be saved in the User Library and may be retrieved later in another model.

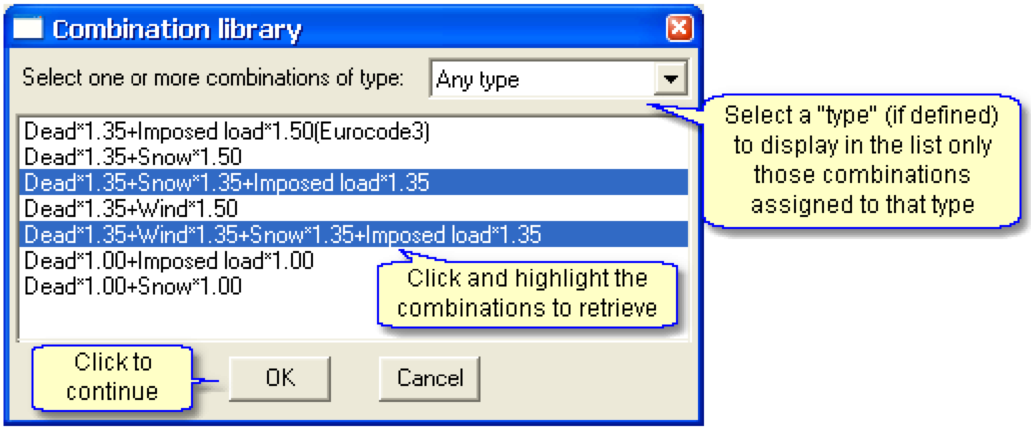


Retrieve combinations from user/standard library

Retrieve a combination from one of the Libraries:

- **User library** - the list of combinations created and maintained by the user.
- **Program library** - the list of Code combinations supplied with the program.

The program displays a list of the combinations in the library. For example:



The program automatically adds all "groups" in the selected combinations to the list of "groups" in the model.

- the program does not add a group if one with the identical name already exists
- ***new groups contain no load cases. You must*** assign load case to these groups 540.
- a combination will ***not*** be generated if no load cases are assigned to ***at least one*** of the groups in the combination definition, e.g. you can retrieve the snow load combinations in the menu above but these combinations will be ignored if you don't define "snow load" cases.

Add combinations to library

Add a combination defined in the current model to the end of the User library (in the current folder).

The combination list for the current model is displayed. Select combinations and click End when the selection is completed. The selected combinations are appended to the ***end*** of the library.

Insert combinations into library

Insert a combination defined in the current model into the User library (in the current folder).

This option is similar to **Add combinations to library**, except that the program first asks you to specify the location in the library combination list where the model combinations should be inserted.

Delete combinations

Delete a combination from the User library (in the current folder). Use the mouse/arrow keys to highlight the combination to be deleted; click the mouse.

4.22.6 Display/print

Display/print the combinations defined for the current model:

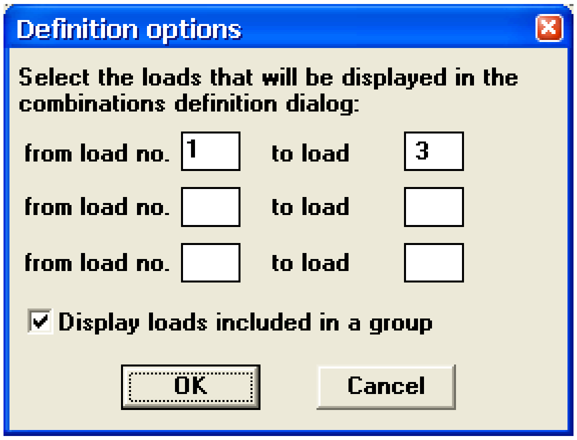
Select :

- **Combinations:**
  to display/print combinations, including those generated by groups.
- **Load groups:**
  to display/print a list of the load cases in each group.
- **Combination definitions:**
  to display/print the combination definition commands.

4.22.7 Options

The program by default displays all load cases in the Define combinations dialog box. This may be inconvenient for models with many cases, especially if some loads will never be included in a combination.

Use this option to delete specified load cases from the Define combinations dialog box:



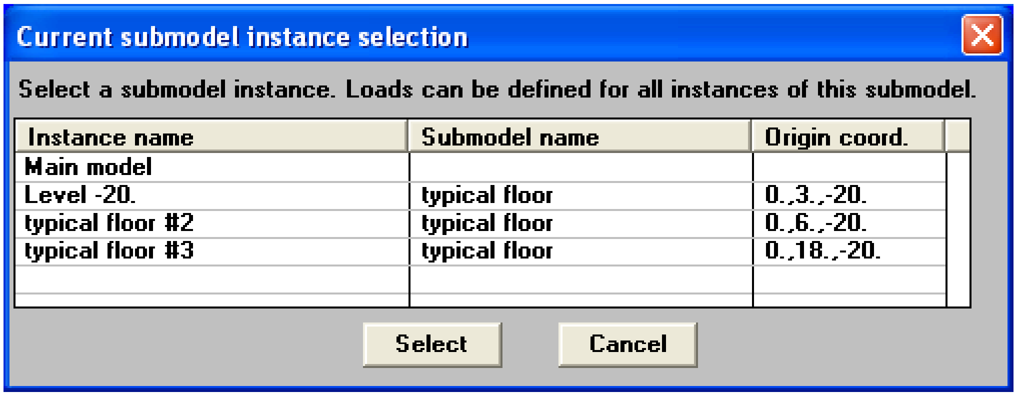
where:

☑ load cases included in a group are displayed

☐ load cases included in a group are not displayed (default)

4.23 Submodel

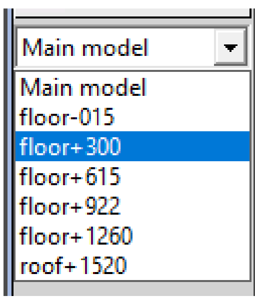
Display an existing submodel instance (or the main model):



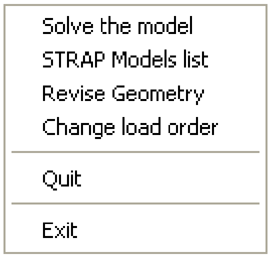
Double-click the appropriate line or highlight the line and click 

.

Note:
- loads applied to one instance of a submodel may be applied at the same time to all other instances of the same submodel.
- alternatively, select the submodel instance in the small list box:



4.24 File options



Solve the model

Save the load data and begin the solution.

STRAP models list

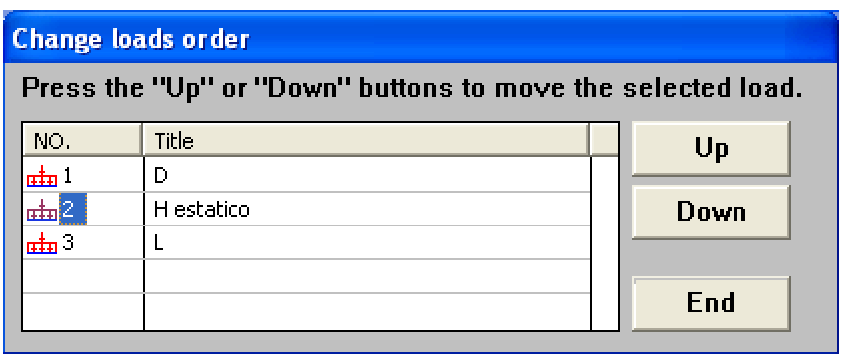
Save the load data and return to the ***STRAP*** main menu (list of models).

Revise geometry

Save the load data and return to the geometry for the current model.

Change load order

Renumber the defined load cases:



- click and highlight a load case.
- click Up or Down

Note:
- if the load cases were solved prior to rearranging the cases, the model must be solved again before viewing the results.

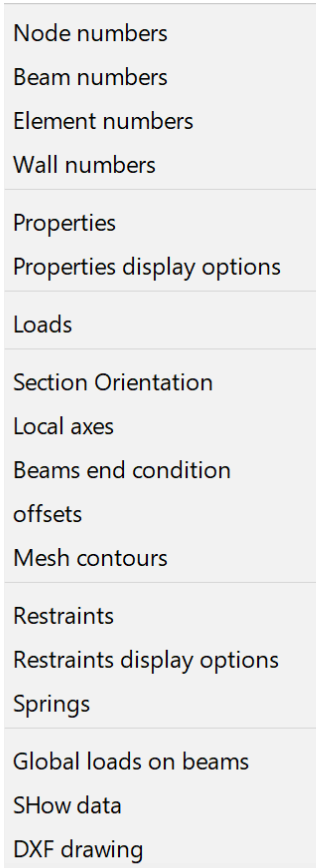
Quit

Exit ***STRAP*** without saving changes to the load data.

Exit

Save load data and leave ***STRAP***.

4.25 Display

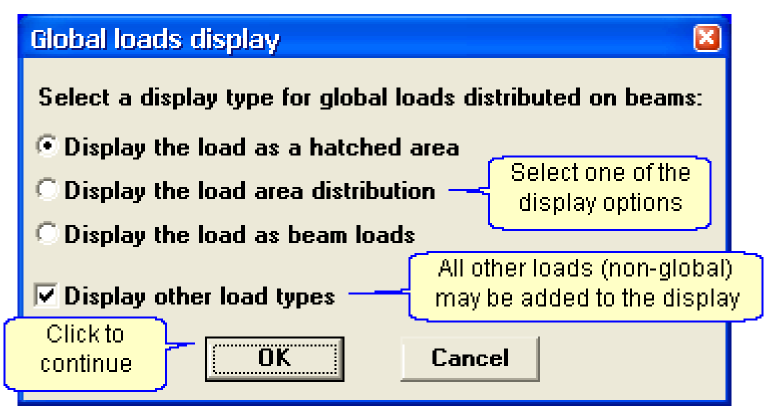


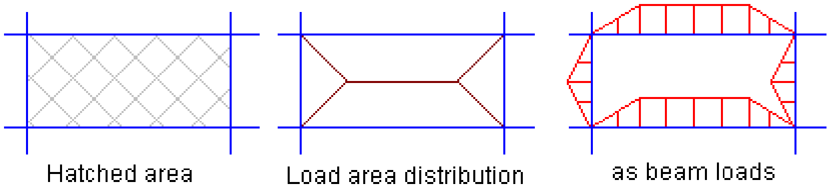
Global loads on beams

Display the distribution of global loads to beams (for global loads applied as beam loads).
- select the **Display global loads** option in the **Display** pulldown menu
- select a load case (if you are not currently editing a load case)

Example:

A global load was applied to a rectangular area bounded by beams -

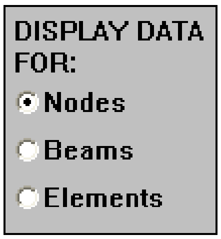




Show data

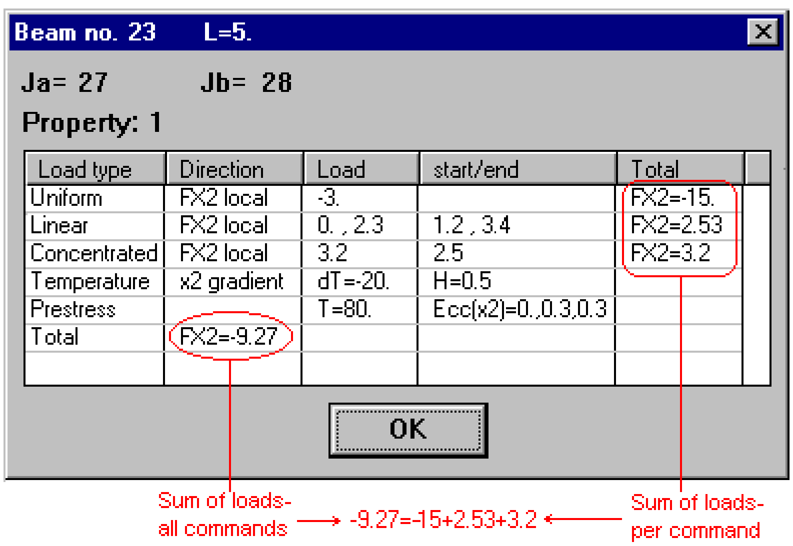
Display all load data for a selected node/beam/element.

- specify the data type option in the box at the right side of the screen:



- highlight the node/beam/element by placing the adjacent to it and click the mouse

Example: beam loads -

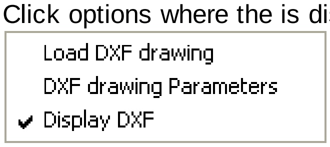


Load type	Direction	Load	start/end	Total
Uniform	FX2 local	-3.		FX2=-15.
Linear	FX2 local	0. , 2.3	1.2 , 3.4	FX2=2.53
Concentrated	FX2 local	3.2	2.5	FX2=3.2
Temperature	x2 gradient	dT=-20.	H=0.5	
Prestress		T=80.	Ecc(x2)=0.,0.3,0.3	
Total	FX2=-9.27			

DXF Drawing

Add a DXF drawing to the background display of the current model. The DXF drawing is not added to the model.
The program allows the ends of all DXF drawing lines to be selected when defining a contour of a global load 466.

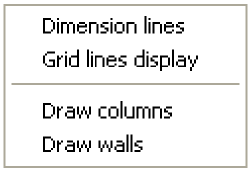
Click options where the is displayed for more detailed Help:



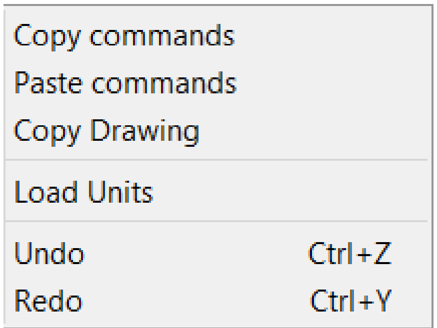
Display DXF:

- The DXF drawing may be temporarily removed from the background and later restored by clicking on the option.

4.25.1 Draw



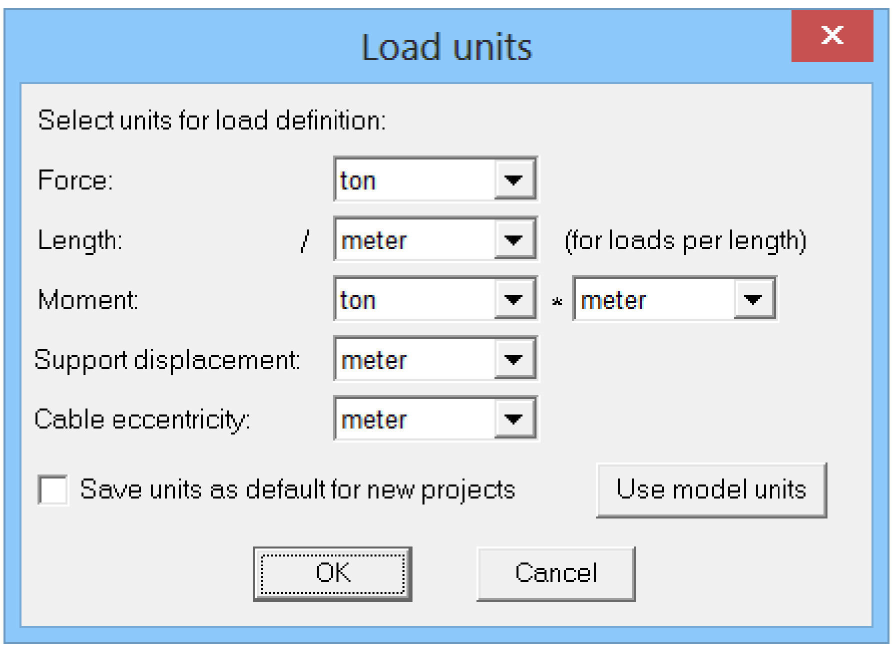
4.26 Edit



Load units

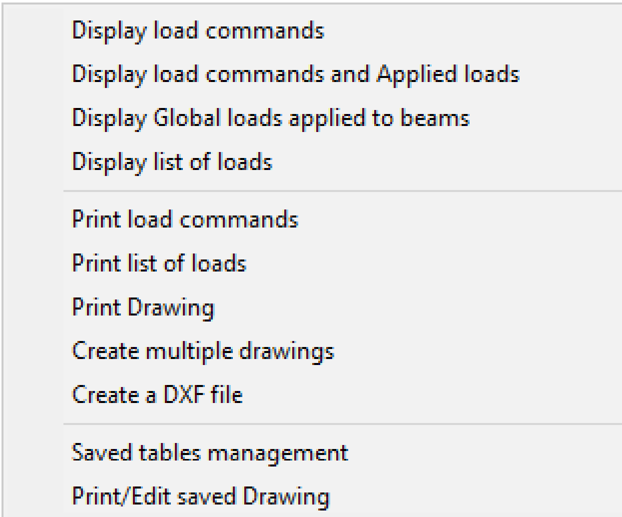
The load definition units may be revised at any time. Note that different units may be selected for the various load types:

- all existing loads are updated in dialog boxes, the graphic display and output tables according to the new units selected, however -
- the "load command" tables are always displayed according to the model load units (defined in Geometry).



- to restore the "model units" (defined in Geometry), click Use model units.
- to save the selection as the default for new projects, ☑ the box and click OK.

4.27 Output



Display load commands

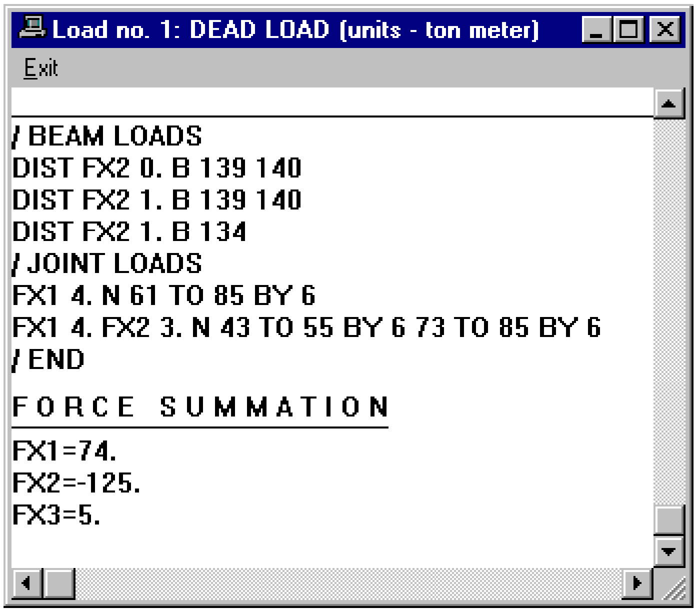
Display the defined loads for selected load cases in tabular form.

The program displays a table of the defined load cases:

- display a single load case; move the to the appropriate line and double click the mouse (or click the mouse and click the OK button)
- display multiple load cases:
  - select the first load case (single click) then select other load cases by single clicking while pressing the **Ctrl** key; click the OK button to display.
  - to display a range of cases, select the first case (single click), then select the last load case by single clicking while pressing the **Shift** key - all intermediate cases will also be highlighted; click the OK button to display.

For each loading case, the program first lists the load definition commands and then a summation of the loads in each of the three global axis directions. The commands are in the standard format; refer to the Command Mode Manual.

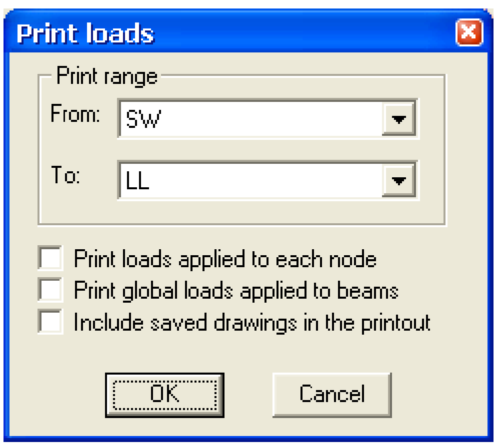
For example:



Check the input by calculating the sum of the loads on the structure in each of the global directions and comparing the result to the summation in the table displayed on the last three lines.

Print load commands

Print the loading data displayed.



- select the range of load cases to print
- the following additional information may be printed
  - Applied loads to nodes
  - Global loads applied as beam loads
  - drawings created with the **Save for "print/edit drawing"** option.

Note:

- Tables may be written to ASCII files in user-specified format using the STBatch utility.

Display load commands and applied loads

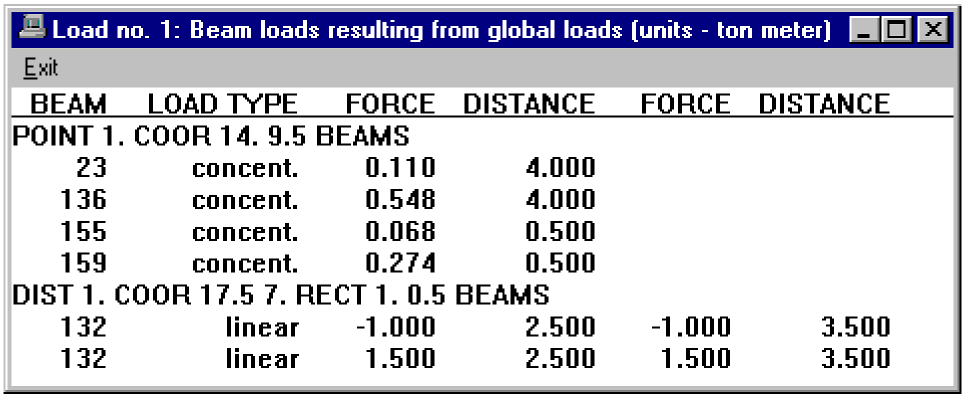
The program internally converts all defined loads into joint loads. The sum of the joint loads at each node is called the ***applied load***.

Display:

- the load commands
- the applied loads at all nodes.
- the center-of-gravity of the loads in each global direction

Display global loads applied to beams

Display the beam loads and joint loads generated by global loads applied as beam loads:

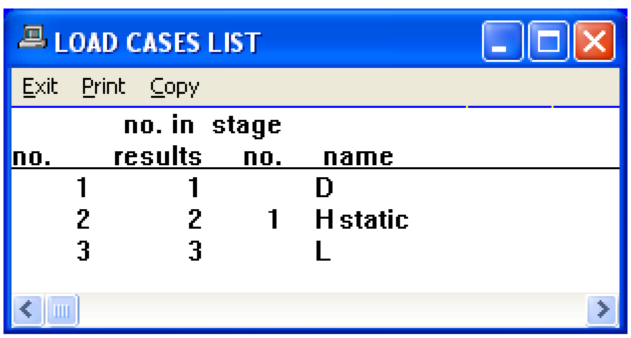


BEAM	LOAD TYPE	FORCE	DISTANCE	FORCE	DISTANCE
POINT 1. COOR 14. 9.5 BEAMS					
23	concent.	0.110	4.000		
136	concent.	0.548	4.000		
155	concent.	0.068	0.500		
159	concent.	0.274	0.500		
DIST 1. COOR 17.5 7. RECT 1. 0.5 BEAMS					
132	linear	-1.000	2.500	-1.000	3.500
132	linear	1.500	2.500	1.500	3.500

For the method the program uses to apply the global loads, refer to Global loads - Method of Application

Display/print list of loads

Display/print a list the load cases. For example:



no.	no. in results	stage no.	name
1	1		D
2	2	1	H static
3	3		L

where:

no.	=	load case number in the load definition module
no. in results	=	load case number in result module and postprocessors, where inactive loading cases are ignored in the numbering
stage. no.	=	displayed if the load case was assigned to a construction stage.

Print drawing

Print the current graphic display.

Create multiple drawings

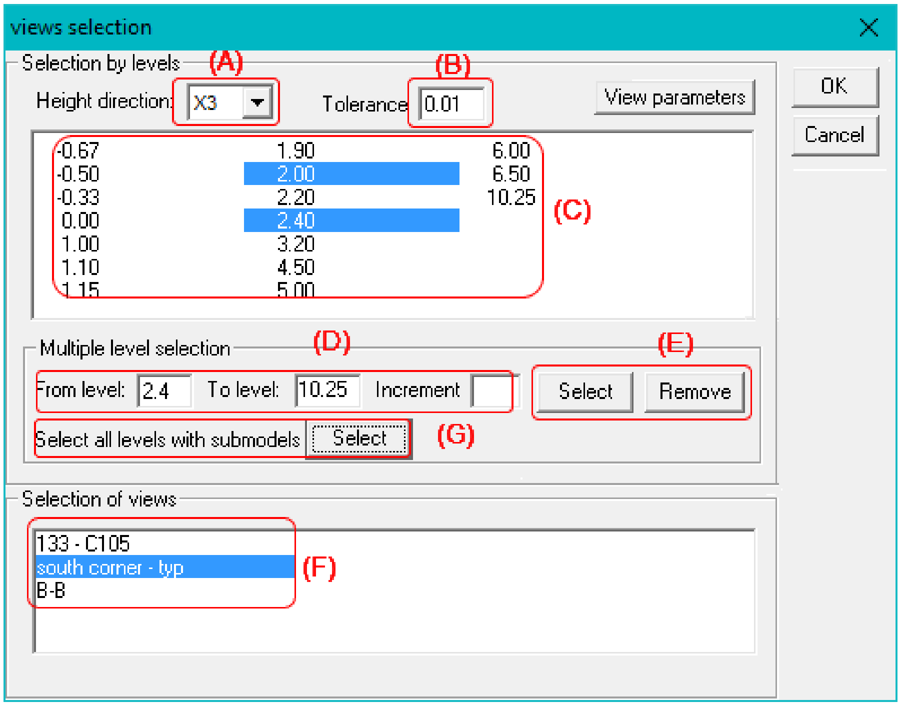
Create multiple result drawings simultaneously for the "***Print/edit a saved drawing"*** option.
A series of drawings is created for ***all combinations*** of the following selections:

- views (elevations, saved views)
- load cases

For example, if you select 4 different views and 6 load cases, a total of (4x6) = 24 drawings will be generated !

To create the drawings:

- select a series of levels and/or views:



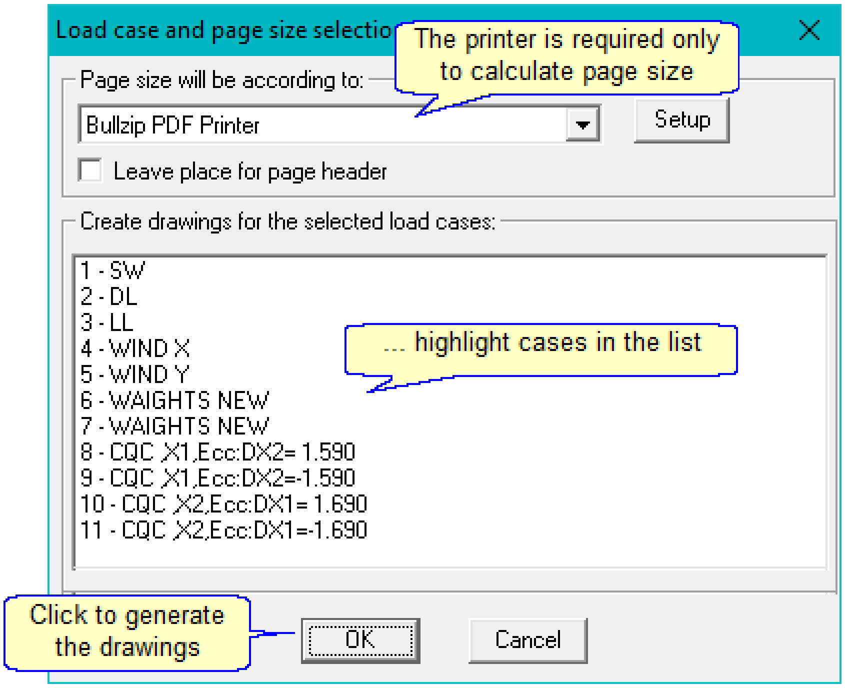
- The program displays a list of the levels parallel to one of the global axes **(C)**. A different global axis may be selected in the Height direction list box **(A)**.
- Select levels by clicking and highlighting them **(C)**.
- Equally spaced levels may be selected by entering the coordinate of the start level, end level and increment in edit boxes **(D)**.

  Click Select **(E)** to highlight these levels in the list or Remove **(E)** to remove the highlight.
- All levels with submodels can be selected **(G)**
- Similarly highlight saved Views **(F)**.
- Click OK to exit and continue.

Note:

- All nodes within the ± tolerance distance **(B)** are selected.

- select a series of load cases, combinations or envelopes:



Create a DXF file

Create a DXF format file showing the current display. Refer to Create a DXF file.

Saved output tables management

Delete tables, rearrange the order of tables or display/print tables that were saved using the **Save output for report generation** option. Refer to Saved tables management,

Print / edit a saved drawing

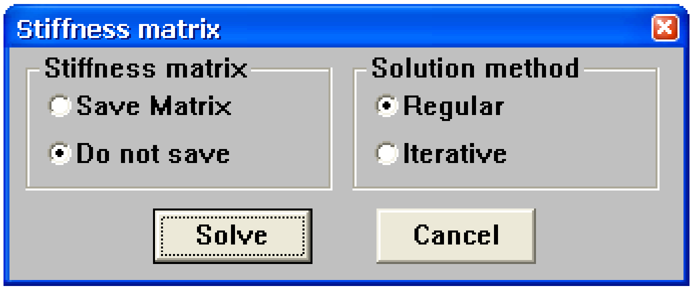
Save the the current display, add/modify text and lines; print. Refer to Print/edit drawing

Part V

Solution

5 Solution

Refer to Solution method 560 for a detailed explanation on the method used by the program to solve the model.



Stiffness matrix

Save matrix
for large models it may be advisable to save the stiffness matrix if the ***same geometry*** will be solved with different loads; the solution then skips the matrix inversion. Note that saving the matrix is recommended for models with dynamic analysis or Bridge design; both these modules will skip the matrix inversion at the start of their **Solve** option if the matrix was saved here.

Note:
- the Bridge module does not recognize the matrix saved by the Iterative method.

Do not save
The program deletes the stiffness matrix at the end of the solution phase.

Solution method

In general, solving a model involves two steps:
- creating and inverting the geometry stiffness matrix
- solving each load case using the inverted stiffness matrix

Two methods are available; comparing the solution time required by each of the methods:

Regular method
The solution is as described above, therefore -
- the matrix inversion phase is relatively slow, compared to the Iterative method.
- the load case solution is relatively fast.

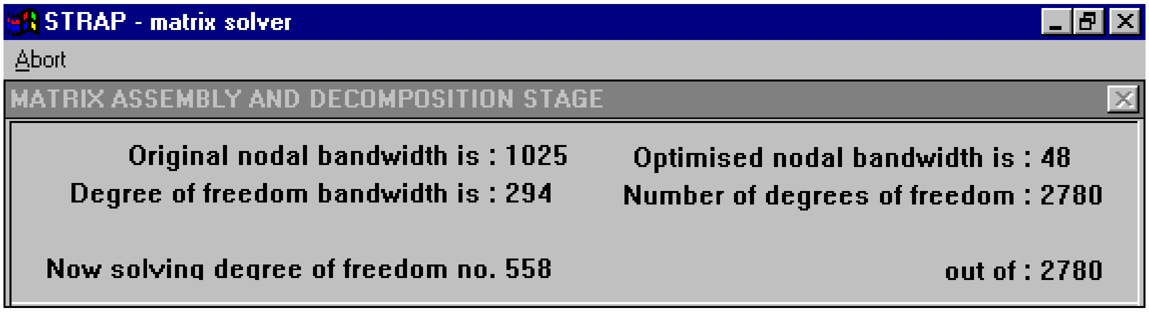
This method is recommended for:
- small and medium sized models
- models with non-linear elements (e.g. one-way springs)
- models with dynamic analysis and a large number of mode shapes
- models with a large number of load cases

Iterative method
- the matrix inversion is partial, therefore this phase is relatively fast compared to the Regular method
- the load case solution involves a large number of iterations, therefore this phase is relatively slow

This method is recommended for:
- models with a large bandwidth and a relatively small number of load cases

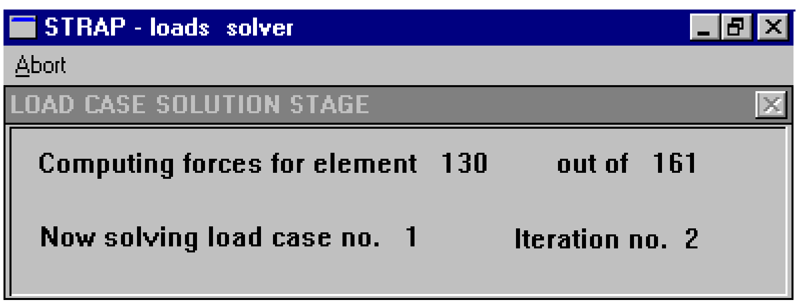
Note
- models with a bandwidth > 16,000 can be solved using this method only.
- the Bridge module does not recognize the matrix saved by the Iterative method.

The program displays the following screen when inverting the matrix:



- The original and optimized nodal bandwidths appear on the first line. The actual degree-of-freedom bandwidth and the total number of degrees-of-freedom appear in the second line of the table.
- The program displays the rate of progress of the solution on the third line; estimate the solution time by the rate-of-change of the number of degrees-of-freedom already solved.
- The program displays singularity messages if the model is unstable. Refer to Singularity 562 or Troubleshooting 563.
- The solution may be stopped at any stage by clicking **Abort** in the toolbar (see below).

After completing the matrix inversion, the program solves the load cases:



The iteration number appears only if non-linear options (P-Delta, unidirectional springs, etc) were specified.

The program copies a back-up of the solution to the disk every few minutes (refer to Setup). The solution may be stopped at any stage by clicking **Abort** in the toolbar. If the solution is aborted or the solution is interrupted by a power failure or computer malfunction, the solution can later be continued from the point of the last backup as follows:

- start the program
- select the current model
- select **Solve** on the toolbar

To restart the solution from the point of interruption, select Resume solution when the following menu is displayed:

5.1 General

The program solves the model by the stiffness method. This method solves models on the basis of joint equilibrium equations in terms of stiffness coefficients and unknown joint displacements. The method formulates the matrix equation

[K] * {d} = {P}

where:

[K] = the stiffness matrix and is a function of the geometry
{d} = the nodal displacement vector
{P} = the applied nodal forces vector

The equation is solved for the unknown node displacements and hence the internal element forces or stresses.

- The stiffness matrix is a square matrix where each side is approximately ***ndof*nj*** (where **ndof** is the number of degrees-of-freedom per joint and *nj* is the number of joints in the structure), and is symmetric about the diagonal starting in the upper-left corner of the matrix.
- Most of the values in the matrix are equal to zero; The non-zero values are usually clustered along the diagonal, forming a diagonal band. The width of this band is called the 'bandwidth'.
- The size of the bandwidth is the most important factor in determining the solution time. The bandwidth size is dependant on the way the nodes and elements in the model are numbered. For example, numbering the nodes in a multi-storey model vertically instead of horizontally increases the bandwidth size.
- However the user need not worry about the numbering method. The program ***automatically*** renumbers the model so that the bandwidth has the minimum width and so minimizes the solution time. *The renumbering is internal*, so the results always appear according to the defined node and element numbers.

The program begins the solution by inverting the stiffness matrix. The stiffness matrix is a function of the geometry only and the inversion of the matrix takes up the majority of the solution time.

- If the program finds a zero value along the diagonal when building the stiffness matrix, the program replaces it with the value 1.0E+20, and displays a ZERO STIFFNESS 562 warning.
- The program decomposes the matrix by the Cholesky method; if the program discovers a value X along the diagonal that has a value 1.0E-9*X after decomposition, the program replaces it with the value 1.0E-7*X and displays a SINGULARITY 562 message.

Beam Elements:

The program adds the shear contribution in the beam stiffness matrix and allows for a reduced shear area.

Triangular Elements:

- Bending: The program uses the HSM element (Hybrid Stress Model) as described in:
  A Study of 3-Node Triangular Plate Bending Elements
  Jean-Louis Batos
  International Jour. for Numerical Methods in Engineering , Vol. 15 1771-1812 (1980)
- Plane Stresses: The program uses a constant strain plane stress element.

Quadrilateral Elements:

- Bending:
  The program divides each rectangular element into four HSM triangular elements by creating a new node at the element centre. The program then uses matrix condensation to delete the new node.
- Plane Stresses: the program uses a linear strain plane stress element.

Refer also to References 564.

5.2 Singularity

The program may also display additional messages during the solution phase indicating that it encountered problems in solving the equation i.e. the stiffness matrix was found to be singular. The singularity messages may be printed.

Singularity indicates that the matrix determinant equals zero. The cause of singularity is instability of the model.

ALWAYS DETERMINE the REASON for SINGULARITY

There are two types of singularity:

- ***Local Singularity:***

The model as a whole is stable but there is local singularity at a node; the following message is displayed:

zero stiffness at node _ in dof ___

The program restrains the DOF and proceeds with the solution.

Several examples are:

Plane truss	:	X3 was not restrained at all nodes.
Plane frame	:	All beams connected to a node are pinned.
Space frame	:	Torsional moment of inertia (J) not defined.

Note:
- In many cases of local singularity the corrective action taken by the program will lead to the correct solution (e.g. plane truss example above). However, the solution time required will be greater.

- ***Structural Singularity:***

The entire model is unstable ; the following message is displayed:

Singularity or near singularity detected at node __

The program takes corrective action, i.e., supplies missing restraints in order to complete the solution.

Note that the program arbitrarily restrains the first joint in the model that will make the model stable. The corrected model usually does not correspond to the model that you intended to solve; display:
- the reactions in the result table to see if the program created new supports
- graphic displacements to check whether the deflected structure has the correct form.

If the reactions or deflections are not correct, ignore the results and return to geometry to revise the model.

Note:
- this singularity message may be displayed when there is a large difference between maximum and minimum moment-of-inertia values defined for property groups.

5.3 Troubleshooting

- Warning messages (solution will be completed):
  - if the model consists of two or more unconnected parts.
  - if two or more beams connect the same two nodes
  - too many beams connected to one node (node/element optimization is discontinued).

- Error messages (solution is aborted):
  - total number of degrees-of-freedom exceeds 192,2000 (6 x 32,000).
  - various errors in the geometry.
  - Not enough space on disk ; at the start of the solution the program checks if sufficient disk space is available to complete the calculations. If not, the program displays a warning message and lists the number of bytes required. Erase unnecessary files from your disk and restart the solution.

The program creates four new files during the solution phase:

File	Status	Size (bytes)
Stiffness matrix file MATRnnn.DAT	temporary or permanent	(FINAL DOF BANDWIDTH + 2) (NUMBER of DOF + 6) 4
Work file ---nnn.DAT	temporary	[(nodes ·28)+(beams+tri) ·112+(quad ·140)] (load cases) + 100
Results file RESLTnnn.DAT	permanent	(FINAL DOF BANDWIDTH)^2 · 8
Finite elements graphic results file ELRESnnn.DAT	permanent	(tri + 4 quad) · 48 · (load cases) + 50

5.4 References

Matrix Analysis of Framed Structures
Weaver and Gere
D. Van Nostrand, 1980

The Finite Element Method - A Basic Introduction for Engineers
Rockey, Evans, Griffiths and Nethercot
Granada Publishing, 1975

Finite Elements for Structural Analysis
Weaver and Robinson
Prentice Hall, 1984

A Study of Three-Node Triangular Plate Bending Elements
Jean-Louis Batos
Int. Journal for Numerical Methods in Engineering
Vol. 15 1771-1812 (1980)

Formulas for Stress and Strain
R. J. Roark
McGraw-Hill

The Finite Element Method
O.C. Zienkiewicz
McGraw-Hill 1977

Finite Element Analysis Fundamentals
R.H. Gallagher
Prentice-Hall 1975

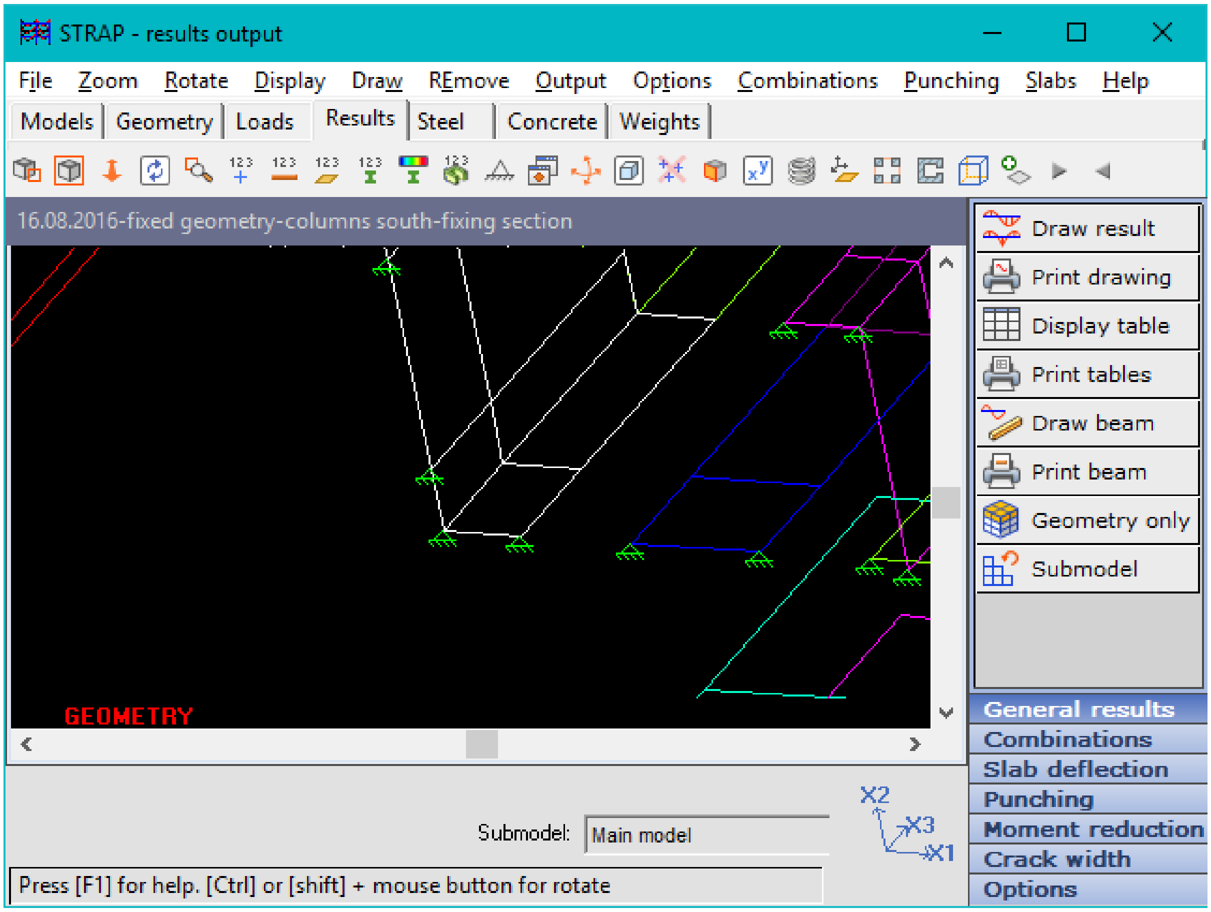
Theory of Elasticity
Timoshenko and Goodier
McGraw-Hill 1970

Strength of Materials
S. Timoshenko
Van Nostrand Reinhold

Part VI

Results

6 Results



Display graphic/tabular results:

- Define the load 

- Specify 

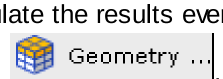
, e.g result axes
- Click:

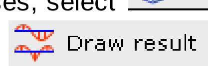
display the graphic results.

display the tabular results.

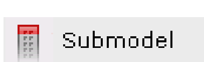
to display the results on a single beam or a line of beams.

This option redraws the current screen with geometry only and without results. This option is convenient when rotating large models with results that take a considerable amount of time to calculate and draw, e.g. contour maps. The program must recalculate the results every time one of the rotate icon is clicked.

In such cases, select 

, rotate the model (without the results), then select 

to add the results.

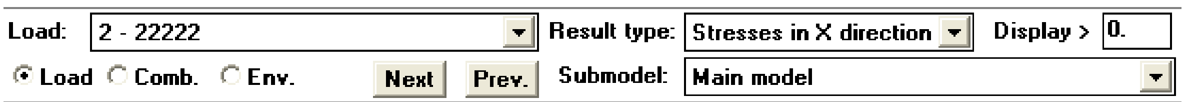


select a submodel instance to display

- Select **Output** in the menu bar to create several drawings simultaneously, to copy a drawing to the clipboard or create a DXF file.

Tabular results for selected beams, elements and nodes may also be displayed using the 'right-click' 653 option.

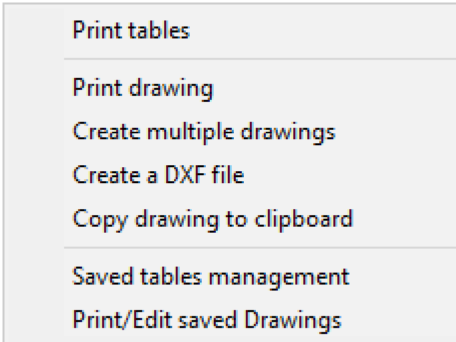
Note:
- 'Views' saved in results also retain the result type and parameters. For example, save a view when a contour map is displayed; the same contour map is displayed every time the view is recalled when in Results.
- Graphic result display may be changed without entering the main Draw result menu; the program displays the following menu at the bottom of the screen:



  Select a different load/combination, result type or submodel; the display is updated immediately.
- Results from several different models can be combined prior to displaying the results. Refer to Combine results of 2 projects.
- The following terminology is used throughout the output modules:

Load cases	Load cases defined in the Loads module (prior to solving the model). Some of these cases may in fact be "Combined load cases", but are still referred to as Cases.
Load combinations	Load case combinations defined in Loads or after solving the model using the Combinations option.

6.1 Output



Create a DXF file

Refer to Create a DXF file

Copy drawing to clipboard

Copy the current graphic display to the clipboard.

Saved output tables management

Delete tables, rearrange the order of tables or display/print tables that were saved using the **Save output for report generation** option. Refer to Saved tables management,

Print/edit saved drawing

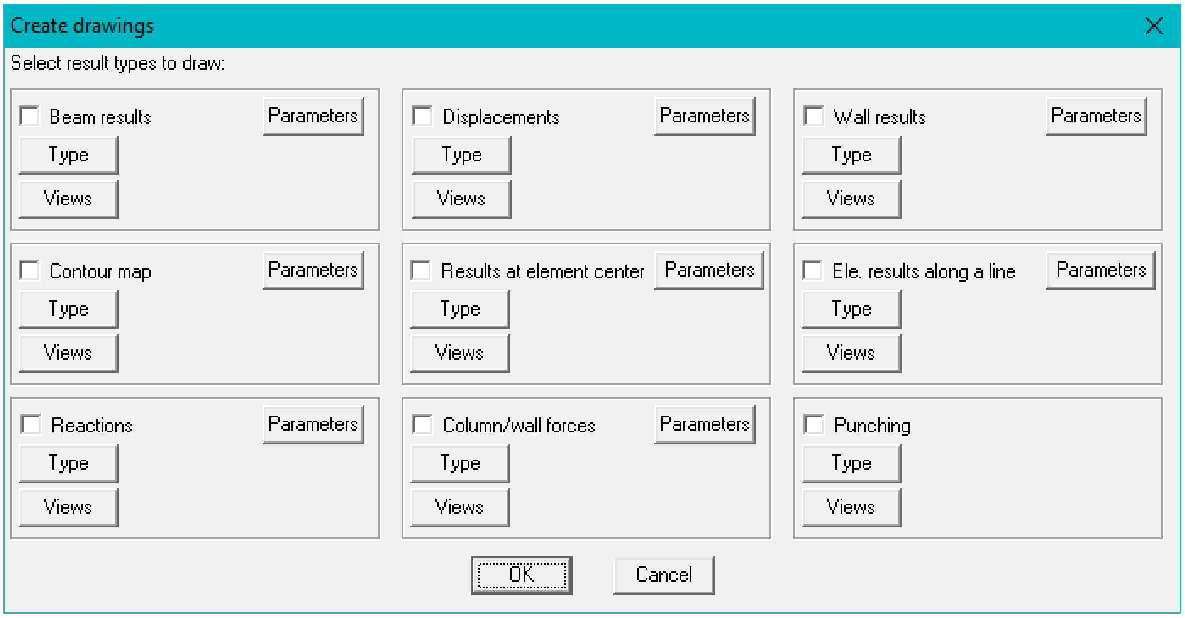
Refer to Print/edit a saved drawing.

6.1.1 Multiple drawings

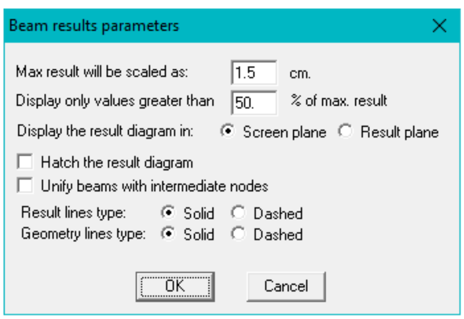
Create multiple result drawings simultaneously for the "***Print/edit a saved drawing"*** option.
A series of drawings is created for ***all combinations*** of the following selections:
- result types (beams results, element results, etc)
- result types (moments, shears, stresses, etc)
- views (elevations, saved views)
- load combination, cases and envelopes

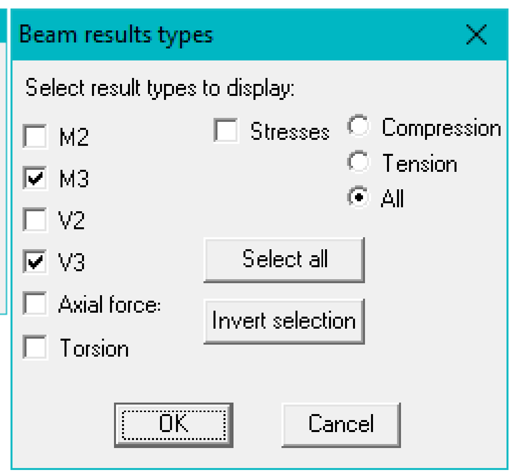
For example, if you select M2 and V3 beam results, 4 different views and 6 load combinations, a total of (2x4x6) = 48 drawings will be generated !

To create the drawings:
- select the result types, views and drawing parameters:

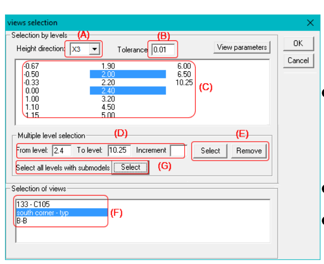


For example, the Parameters and Type options for Beam results:





- if Views is selected:

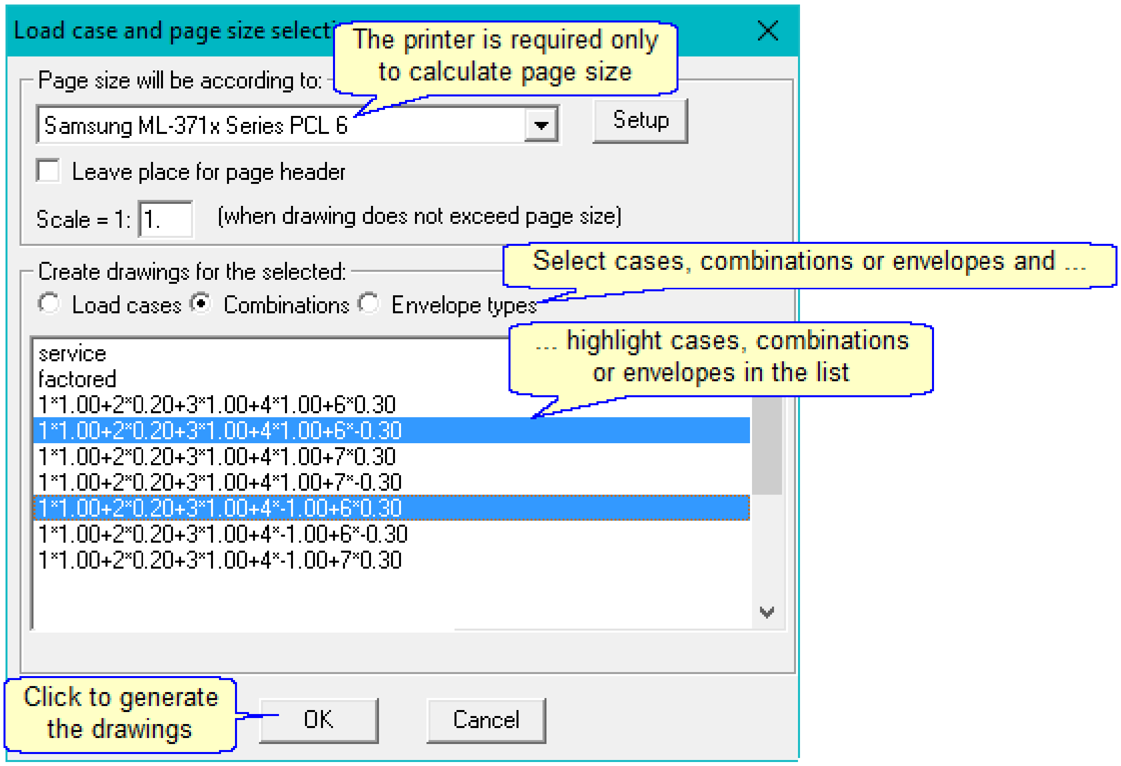


- The program displays a list of the levels parallel to one of the global axes **(C)**. A different global axis may be selected in the Height direction list box **(A)**.
- Select levels by clicking and highlighting them **(C)**.
- Equally spaced levels may be selected by entering the coordinate of the start level, end level and increment in edit boxes **(D)**. Click Select **(E)** to highlight these levels in the list or Remove **(E)** to remove the highlight.
- All levels with submodels can be selected **(G)**

- Similarly highlight saved Views **(F)**.
- Click OK to exit and continue.

Note:
- All nodes within the ± tolerance distance **(B)** are selected.

- select a series of load cases, combinations or envelopes:



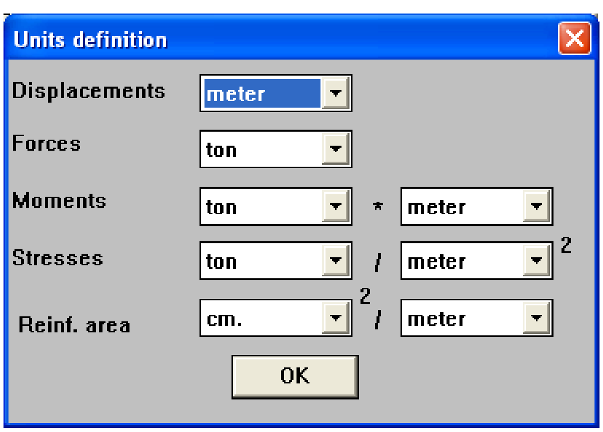
- **Scale** =
  The program automatically calculates the scale so that the drawing fits into one page. The program will use the scale defined here only if the result is that that drawing size becomes less than one page.

6.2 Options

Units	Specify the result display units 571; different units may be selected for each result type:
Output for...	Specify the no. of digits to display after the decimal point for the various result types. Refer to Output format 573.
Buckling p...	Specify parameters for axial stress calculation (buckling length, etc.). The program calculates the allowable axial stress based on design code equations. Refer to buckling parameters 574.
Coord. sy...	Specify the direction of the result coordinate system for finite elements (by default the element local coordinate system). Refer to Element result systems 577.
Reinf. ske...	Specify the skew angle between the two major reinforcement directions. Refer to Element result systems 577.
Use princi... Use geom...	Click this icon to toggle between to options this option to display beam results: • about the principal axes - Mu, Mv, etc. • the major/minor axes - Mx, My, etc. This option is relevant only for beams with unsymmetric sections,e.g. single angles.
Reactions ...	Reaction values 572 may be displayed according to the global or the "local restraint coordinate system" (if defined).
reinf. para...	Define reinforcement parameters (diameter and spacing) ***for individual elements***. The default parameters for the entire model are defined every time the display of reinforcement results is requested. Refer to Reinforcement parameters by element 580.
Display rei...	Display the current reinforcement parameter values for each element. Refer to Display reinforcement parameters 582.
Print reinf....	Print the current reinforcement parameter values for each element. Refer to Display reinforcement parameters 582.

Units

Specify the units for display of results (by default the result units are the same as the input units). Different units may be selected for each result type:

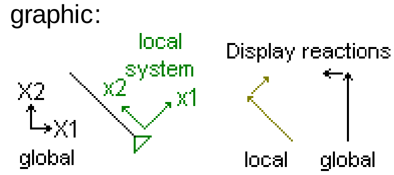


Note that the default units are not changed.

Local systems for reactions

✓ to display the reaction values according to the "local restraint coordinate system" and not the global system.

- graphic:

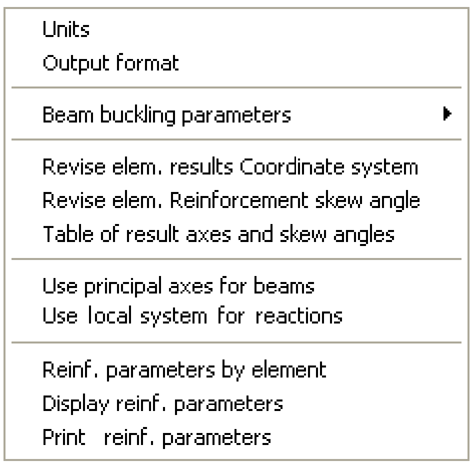


- tables:

rotated support

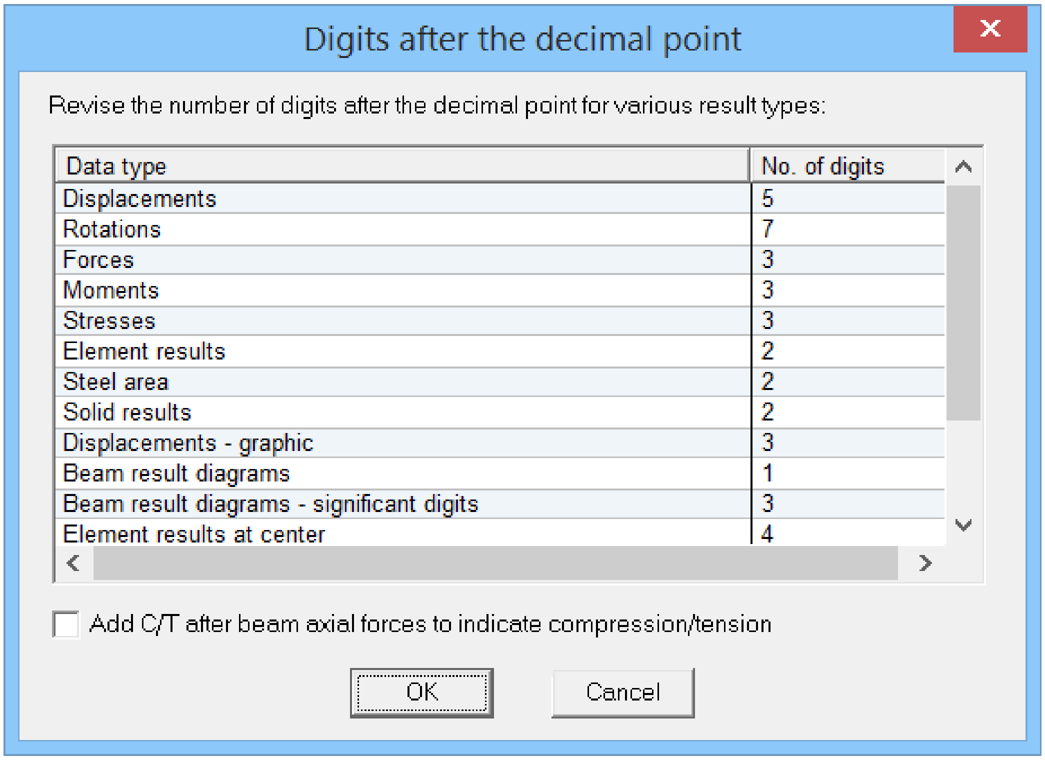
Node	X1	X2	X3	X4	X5	X6
R1	98.069	87.808	85.786	51.101	-125.717	-311.499
6	13.275	-85.580	84.157	189.161	-60.911	-310.004

If **Options** is selected in the menu bar:



6.2.1 Output format

Specify the maximum number of digits to display after the decimal point for the various result types:



No .of digits

Specify the maximum number of digits to display after the decimal point for the various result types:

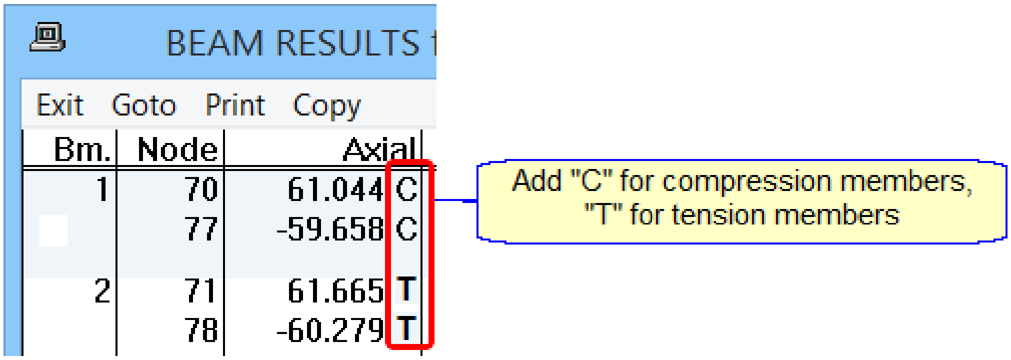
- '**Significant digits**' options:
  The program will display fewer digits after the decimal point for large values. The total number of digits displayed will equal this value (and will exceed it for larger numbers). For example, 'No of digits' = 3 and 'Significant digits' = 3:

Result	Displayed	
0.0013245	0.001	(but see note below for beam results only)
0.013245	0.013	(3 significant digits)
1.3245	1.32	"
13.245	13.2	"
132.45	132.	"
1324.5	1324.	(4 digits required for large value)

Note:

- for 'beam result diagrams, the program will always display at least two non-zero digits following the zeros after the decimal point. For example, if you specified 'No. of digits" = 3 and the result is 0.004793, the program will display 0.0048, not 0.005.

Add C/T after beam axial forces



6.2.2 Beam buckling parameters

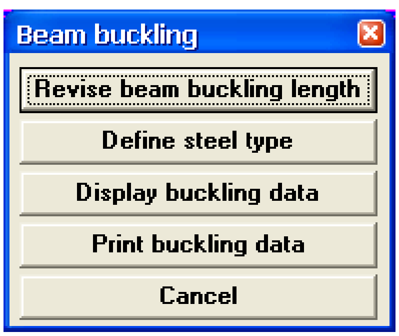
This option calculates the linear buckling effect according to the following steel design codes:

- American AISC ASD
- British BS 449
- German DIN code.

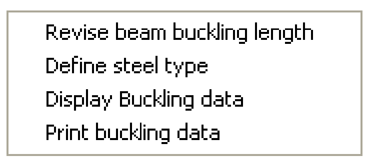
This option is suitable for trusses as bending moments and shear are ignored. (For a complete design of structural steel members, refer to Steel Postprocessor).

Axial stresses are modified by a factor which is a function of the member slenderness; the allowable stresses are reduced in the American and British codes, while the actual stresses are increased in the German code.

Select the design parameters for the design check of steel beam members under axial loads only. The parameters are effective buckling length, steel type and allowable stresses:



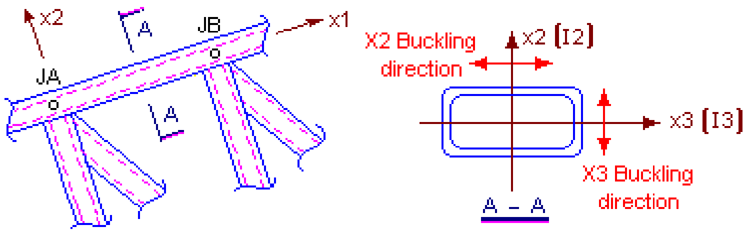
In the menu bar:



Revise beam buckling length

The buckling length of a beam is the effective length of the beam as used in the standard Euler buckling equation. The effective length is denoted in most codes by 'Le' or 'KL'.

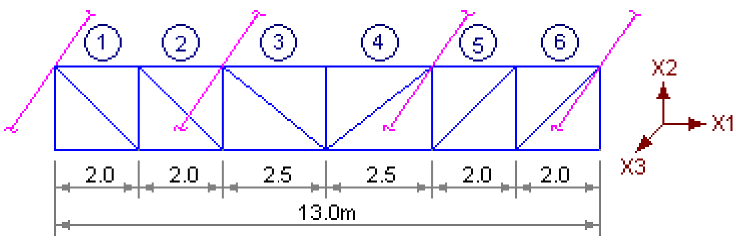
The program requires the buckling lengths about the two local axes x2 and x3 (they need not be equal). Buckling about x2 is buckling associated with the moment of inertia I2, and buckling about x3 is associated with the moment of inertia I3. In the following example, x2 buckling is perpendicular to the plane of the truss, while x3 buckling is in the plane of the truss.



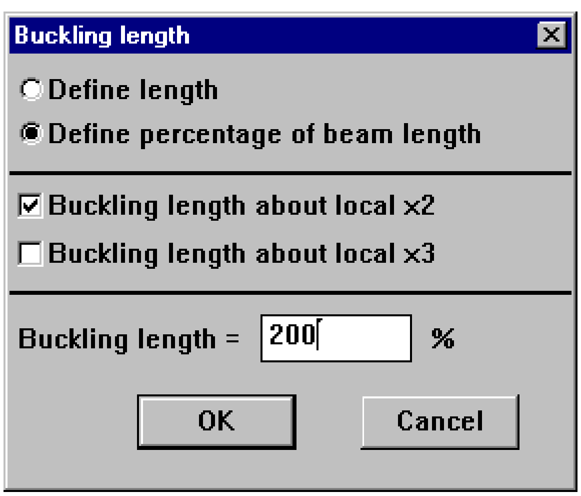
Note:

- All node restraints entered during geometry definition are ignored when calculating buckling lengths;
- The buckling length of a member may be defined as exceeding its actual length.
- If no data is entered for a beam, the program will assume the buckling length in both the x2 and x3 directions is equal to the actual length of the beam. You do not have to enter any data for such beams.
- If information for buckling length about only one axis is entered, the program assumes that the buckling length about the second axis equals the actual beam length.

Example:
For the truss in Figure below, assume that purlins at every second node provide lateral support perpendicular to the top chord of the truss, and that the in-plane buckling length = 0.9 * actual beam length.



Arrange the box as follows:

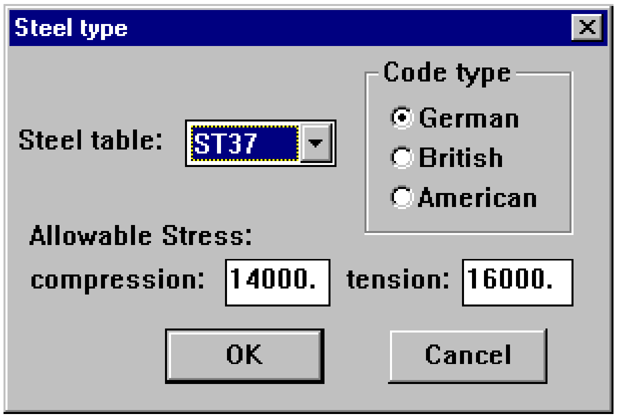


and select all top chord members using the standard beam selection option. Repeat for the x3 direction.

Define steel type

Define the steel type parameters and the design code to be used for the calculations.

After the parameters have been selected, assign them to the beams using the standard Beam Selection option.



- ***Steel table***

British code: **G43 , G50 , G55**
German code: **ST37, ST52 , PIPE37 , PIPE52**
American code: **MAIN36, BRAC36, MAIN42, BRAC42, MAIN50, BRAC50**
where the number represents the specified minimum yield point of the steel. For example: for A36 steel, enter MAIN36 or BRAC36.

- ***Code type***

Select one of the following steel design codes.
German : DIN
British : BS449
American : AISC ASD

- ***Allowable stress***

Define the allowable tension and compression stresses (Required for German code steel code only).

Display/print buckling data

Data includes effective length, radius of inertia and slenderness about each of the two buckling axes.

Examples:

British or American codes:

***** X2 DIRECTION *****

Beam no.	Length	Buckl. length	R. of inert.	Slend.	Red. Factor	Buckl. length
1	1.000	1.00	.256E-01	39.11	1.11	1.00
2	3.000	3.00	.256E-01	117.34	2.47	3.00

where:

Red. Factor:
the ratio of the allowable stress for this slenderness to the allowable stress without buckling effect.

German code:

***** X2 DIRECTION *****

Beam no.	Length	Buckl. length	R. of inert.	Slend.	omega
1	1.000	1.00	.256E-01	39.11	1.13
2	3.000	3.00	.256E-01	117.34	2.28

omega:
the factor for this value of slenderness. If the beam in compression, the program increases the actual stress by this factor.

and for all codes:
Beam no. = Beam number as listed in the geometry table.
Length = Actual length of the member.

The following values are listed separately for the x2 direction and the x3 direction:
Buckl. Length = The effective buckling length about the axis.
R. of inert = The radius of inertia of the section = sqrt(I/A).
Slenderness = Effective length divided by the radius of inertia. (Le/r)

Note:
- the factors for modifying the stresses are contained in file BCF.DAT[1240].
- beam axial stress results may be displayed if parameters were not defined in this option. In such a case the program assumes:
  - all buckling lengths equal to the beam lengths
  - the steel type is the first one listed in file BCF.DAT[1240].
- If the moment-of-inertia was not defined during the geometry definition (i.e. only the area was defined), the program is unable to calculate the radius-of-inertia and hence the slenderness. In such cases the program does not calculate the axial stresses.
- For tapered members the program uses the properties at JA.

6.2.3 Element result coordinate system

The explanation of results for finite elements necessitates the introduction of two new coordinate systems in addition to the global and local element systems. They are:

- **Result Coordinate System X , Y , Z**
  The result coordinate system is the set of axes about which the element results are calculated and displayed. In most cases the results axes are identical to the local or global systems.

  If the direction of the local x1 or x2 axes for all of the elements are not co-directional, there will be an apparent inconsistency in the results. Refer to Element result coordinate system - default[580].
- **Reinforcement Coordinate System X* ,Y* (skew angle)**
  The reinforcement coordinate system is required for reinforcement design moment calculation in concrete slab models. The reinforcement axes X* and Y* are defined as parallel to the directions of the reinforcement. The program assumes that X* is always in the same direction as the result coordinate system X axis. Y* can be at any ***skew angle*** α from X*. By default, α = 90°.

The default Element and Reinforcement Coordinate systems may be revised for each element.

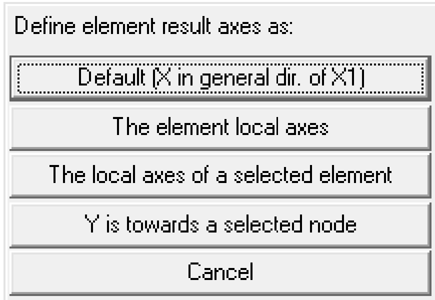
Revise elem. results Coordinate system
Revise elem. Reinforcement skew angle
Table of result axes and skew angles

6.2.3.1 Element result system

The result coordinate system is the set of axes about which the element results are calculated and displayed. The default result axes are related to the global system (refer to Element result coordinate systems - default[580]).

In certain cases the default axes may not give the required results. For example:
- reinforcement in concrete slabs is not parallel to the global axes
- x3 local axis directions are not consistent in cylindrical or spherical models

The result coordinate system may be revised for each quadrilateral and triangular finite element



Default

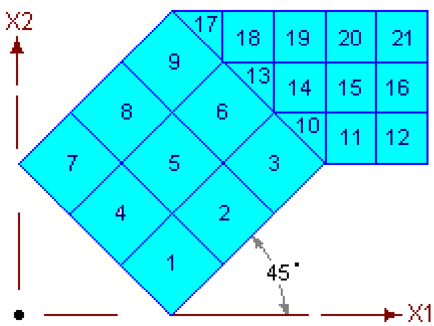
This option restores the default element result axes 580 for the selected elements.

Element local axes

The result coordinate system of each selected element will be identical to its local coordinate system, i.e. X = x1 , Y = x2 , Z = +x3

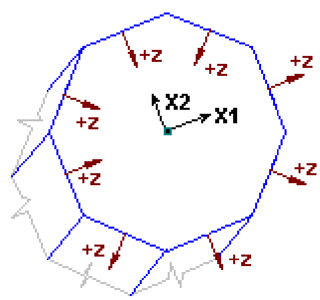
Examples:

Grid:



For elements 1 to 9, the reinforcement is rotated 45° from X1 (i.e. parallel to the element boundaries) and so the moments should be relative to the local axes. By default the results will be relative to the global axes.

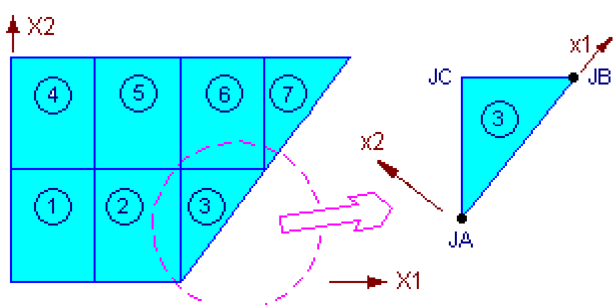
Space frame:



The default Z axis directions are displayed above; note that some point inwards towards the centre of the cylinder while the rest point outwards. Use this option to define a consistent direction for all elements.

Note that this option may lead to misleading and inconsistent results. For example:

- Triangular elements:



The results for triangular elements 3 and 7 will not be relative to the same axes as quad elements 1,2,4,5,6.

- Slab reinforcement:

Confusion will result in the reinforcement calculation if the local x3 axis directions are not consistent. The program assumes that the "TOP" face is the +Z face of the slab and the "BOTTOM" face is the -Z face. Hence, "Top" and "Bottom" reinforcement in different elements may actually be at the same face of the slab if their x3 axis directions are reversed.

Local axes of a selected element

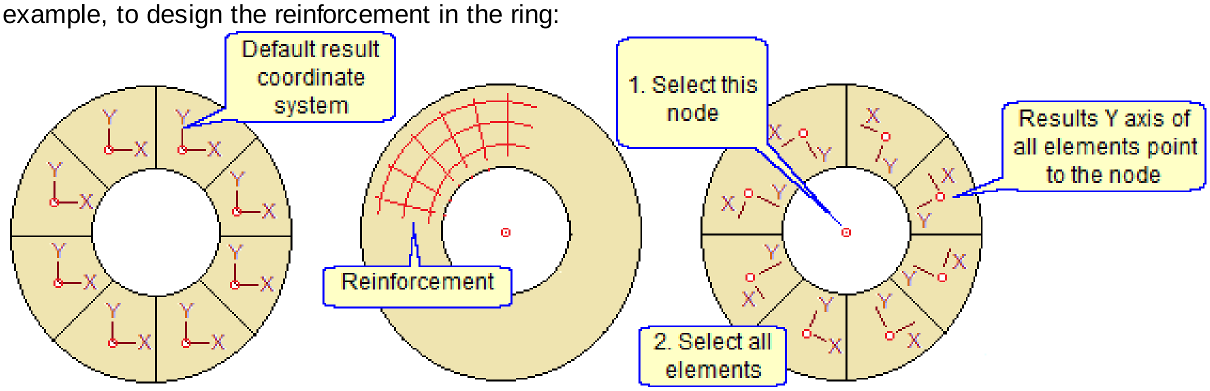
The result axes for an element may be defined as identical to the local axes of a ***different*** element. The "different" element must lie on a parallel plane.

To unify the results, specify that the results axes for all of the elements are identical to the local axes of one element.

Refer to the previous option in this menu - The element local axes 578 - for more details.

Y is towards a selected node

In certain models it is convenient to have all of the result axes point towards the same node. For example, to design the reinforcement in the ring:



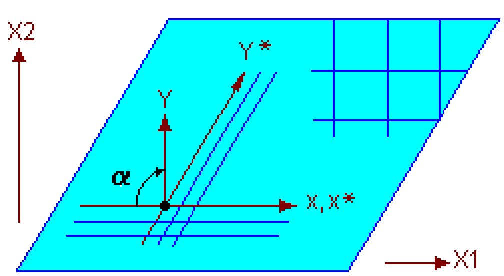
6.2.3.2 Reinforcement skew angle

The reinforcement coordinate system is required for reinforcement design calculation in concrete slab models. The reinforcement axes X* and Y* are parallel to the directions of the reinforcement.

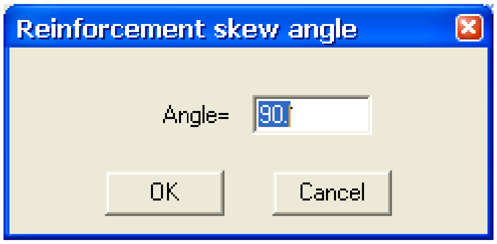
- X* is ***always*** in the same direction as the result coordinate system X axis.
- Y* can be at any ***skew angle*** α from X*. In most slabs, $\alpha = 90°$, and the program assumes this value if no other value is defined.

The design moments Mx* and My* for reinforcement design are calculated according to the Wood and Armer equations 685 in the direction of X*, Y*.

The following figure shows an example with $\alpha \neq 90°$.



Define the angle:



- Enter a value between 1° and 179°.
- Enter a list of elements using the standard Element selection option.

6.2.3.3 Display table

A table is displayed showing the following information for each element in the model:

- result coordinate system axes directions
- skew angle "α"
- deletion of element from the display.

6.2.3.4 Default system

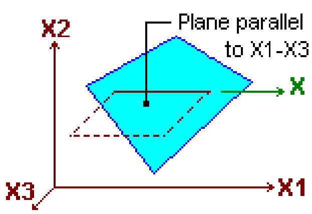
The default result X and Y axes are determined as follows:

Plane frame, plane grid:
The values plotted are those on the upper surface (+X3 Global). The X direction is always parallel to the global X1 axis and the Y direction is always parallel to the global X2 axis.

If the program discovers that the direction of the local x3 axis of an element is opposite to the direction of the global X3 axis, it reverses the sign of results for that element. This insures continuity of the lines.

Space frames:

- Elements parallel to the X1-X2 global plane:
  X = X1 , Y = X2 , Z = X3
- Elements parallel to the X1-X3 global plane:
  X = X1 , Y = -X3 , Z = X2
- Elements parallel to the X2-X3 global plane:
  X = X2 , Y = X3 , Z = X1
- Elements not parallel to a global plane:
  - X = direction parallel to the line of intersection of the element plane with a plane parallel to the X1-X3 global plane (+X in the general direction of +X1)



  - Y = perpendicular to X and in the general direction of X2. (+Y in the direction of +X2)
  - Z = perpendicular to the element and in the general direction of +X3

  If the element is perpendicular to the X1-X2 plane:
  - X = direction parallel to the line of intersection of the element plane with a plane parallel to the X1-X2 global plane.

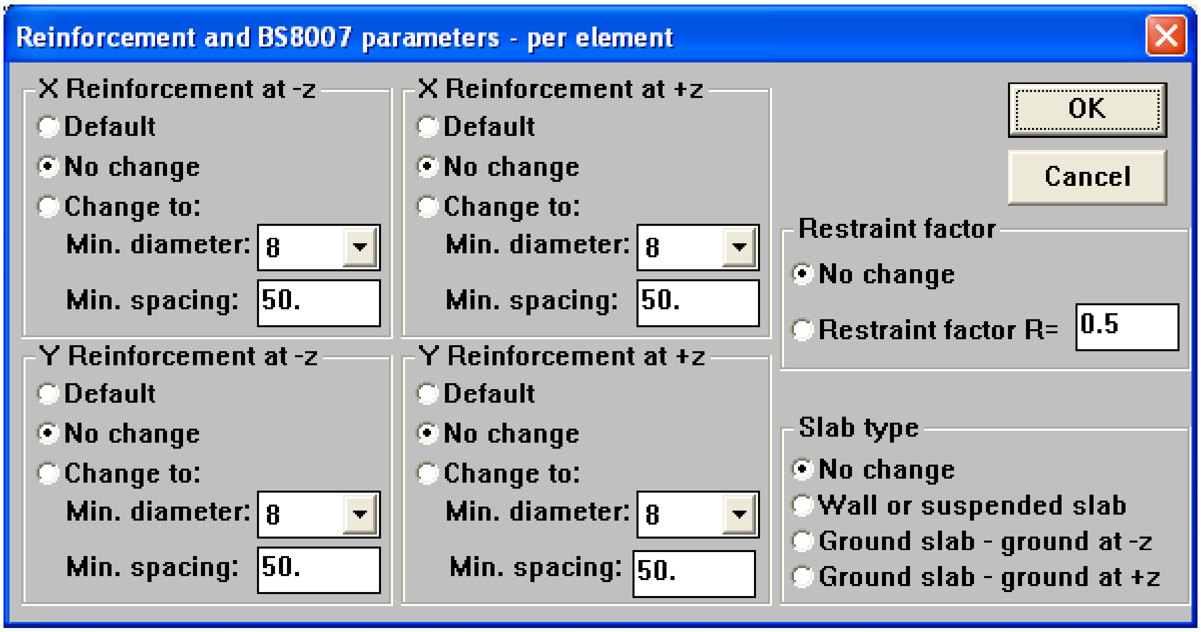
6.2.4 Reinf. parameters by element

Specify reinforcement and parameters for selected elements.

- The values are used when calculating slab deflections of BS8007 crack widths

- The default reinforcement areas and parameters for the model are specified when selecting **Solve concrete slab deflections** or **Display BS8007 results** or **Display BS8007 detailed results** options in the toolbar **Slab** menu.
- Different reinforcement parameters may be specified in both directions and at both faces.

Specify the values, then select elements using the standard element selection option.



Reinforcement at

Reinforcement (actual or minimum) may be specified in both directions and at both faces of the element. Reinforcement defined in this option overrides the default values for reinforcement specified in the **Display BS8007 results** or **Display BS8007 detailed results** options in the **Slabs** pulldown menu.

For each direction (X or Y) and each face (+x3 or -x3), select one of the following options:

Default
Use the default reinforcement values for the selected elements

No change
Use the current reinforcement values in this direction/face for the selected elements (default values or values specified previously using this option).

Change to
Select a diameter from the list box and type a spacing value in the edit box.

For all options, select elements using the standard element selection option.

Note:
- The reinforcement specified in this option will be used as follows:

Design option	*Reinforcement*
Reinf. required for moments/forces	Minimum
Reinf. required for limiting crack width	Minimum
User defined reinforcement	Actual

- Reinforcement may be checked by selecting the **Reinf. parameters by element** option in the **Options** pulldown menu.

- X/Y directions refer to the Element result coordinate system
- +x3/-x3 directions refer to the element local coordinate system.

Restraint factor

The default factor for all elements in the model is always 0.5. Refer to Code section A.3, A.5 and Figure A.3.

Select one of the following options:

Restraint factor =
Revise the restraint factor R for selected elements.

No change
Use the current factor value for the selected elements (the default value or values specified previously using this option).

Slab type

The slab type is required for the calculation of ρ, the steel ratio. the ratio value is based on the area of the "surface zone". Refer to Code section A.3 and Figure A.2.

The default slab type for all elements in the model is always "**Wall or suspended slab**".

Select one of the following options:

Wall or ground slab / Suspended slab ..
Revise the slab type for selected elements.

No change
Use the current slab type for the selected elements (the default value or values specified previously using this option).

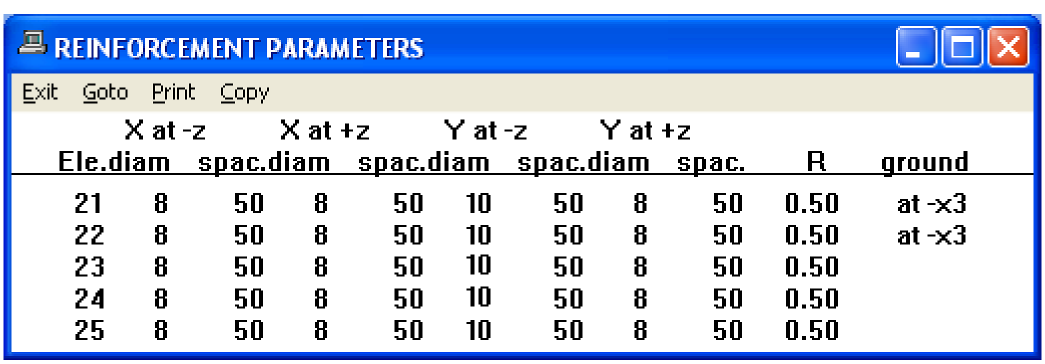
Refer to:

- BS8007 - General 623 for help on the calculation method
- How to use this module 623
- Slab deflections - method of calculation

6.2.5 Display reinf. parameters

Display the current parameter values. The parameters are those that can be defined per element using the **Reinf. parameters by element** option in the **Options** pulldown menu.

For example:



REINFORCEMENT PARAMETERS

Exit Goto Print Copy

Ele.	X at -z diam	spac.	X at +z diam	spac.	Y at -z diam	spac.	Y at +z diam	spac.	R	ground
21	8	50	8	50	10	50	8	50	0.50	at -x3
22	8	50	8	50	10	50	8	50	0.50	at -x3
23	8	50	8	50	10	50	8	50	0.50	
24	8	50	8	50	10	50	8	50	0.50	
25	8	50	8	50	10	50	8	50	0.50	

where:

R = the restraint factor. Refer to Code section A.3, A.5 and Figure A.3
ground = slab type. The field is blank for "**wall or suspended slab**". Refer to Code section A.3

and Figure A.2.

6.3 Result combinations

Use this option to define combinations of the solved load cases for the following output modules:
- tabular and graphic results
- design modules - steel, concrete, bridge, etc.

The same options are also available during Load definition, where combinations that are solved can also be defined.

Define/rev...	Define/revise load case combinations.
Groups	Define load case groups; each case in a group can then be used in a different combination.
Types	Define combination "types", e.g, Service, Factored, Seismic, etc. Each combination may be assigned to a type.
Library	Define a library of standard combinations or retrieve a combination from the library.
Display list	Display the combinations defined for the current model
Print list	Print the combinations defined for the current model
Disable all	Temporarily disable 584 all combinations (the combinations are not deleted).
Deactivate	Deactivate 585 selected combinations (the combinations are not deleted).
Options	Select load cases that are not displayed in the Define combinations option

Note:
- Refer to Combinations - General for a detailed general explanation about "Groups" and "Library".
- The same options are available in the Loads module.
- the rules of superposition do not apply for ***non-linear*** elements. Therefore, load combinations for models with tension/compression only elements, unidirectional springs, etc, ***must be defined*** in loading prior to "Solve" - and not after the solution using this option.
- the same options may also be selected in the menu bar:

Define / revise combinations
Define / revise Groups
Standard combinations library
Display combinations List
Print combinations list
Disable all combinations
Deactivate selected combinations
Definition options

6.3.1 Disable

The combinations table will be temporarily disabled (not deleted); results will then be displayed for load cases only.

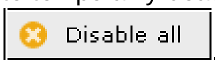
Note that this option, if selected, is ignored by all postprocessor and design modules.

To disable selected combinations in all modules, select Deactivate.

6.3.2 Deactivate

Click and highlight combinations in the list box to deactivate them; **Inactive** will be displayed next to the title.

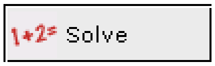
Note:

- the combinations will be deactivated for graphic/tabular results and all design postprocessors
- to temporarily deactivate all combinations in order to display load case results, select 

.

6.4 Slab deflections

Calculate slab deflections based on the effective moment-of-inertia based on the reinforcement and the cracked section properties.



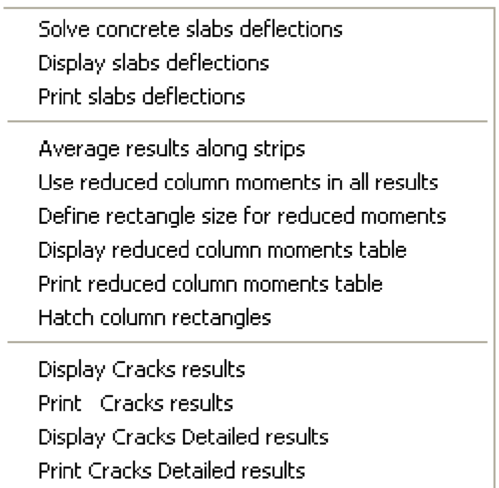
Specify the deflection parameters (combinations, reinforcement, cover, etc) and calculate the deflections 586.

Display 588 the deflection results (tabular or graphic)

Print 588 the deflection results (tabular or graphic)

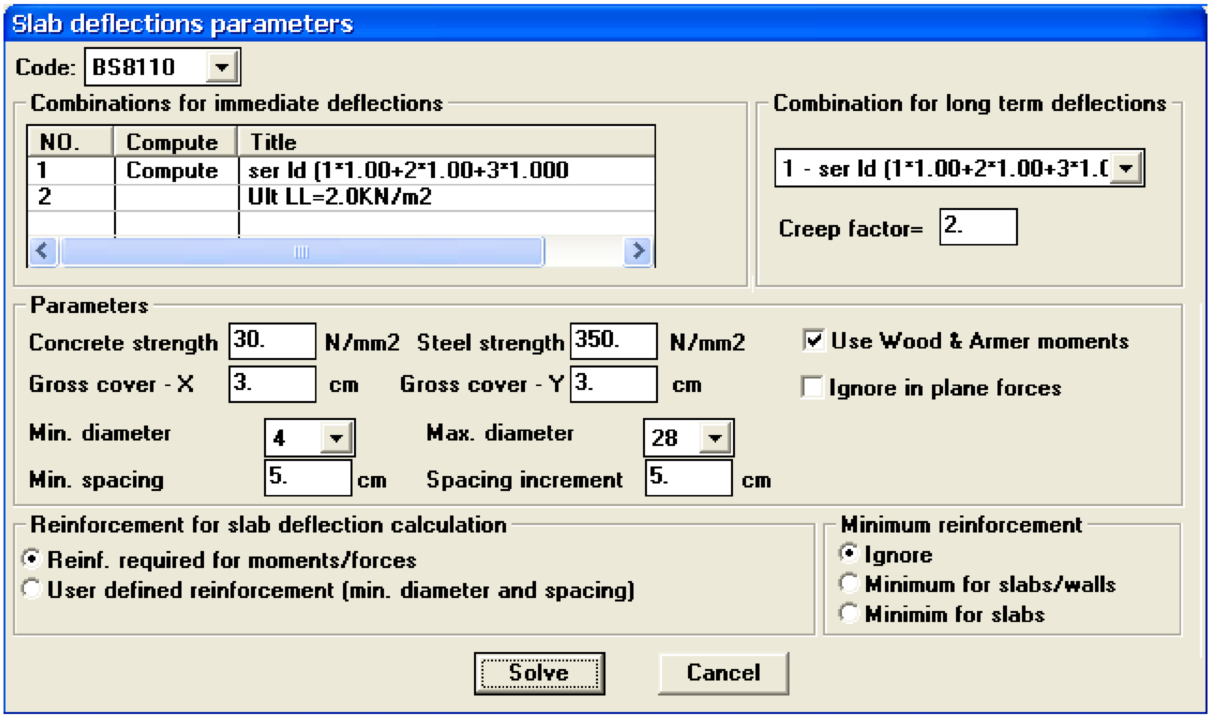
Print 89 the current graphic display

or select the **Slab** options from the menu bar:



6.4.1 Solve slab deflections

Calculate the slab deflection based on the effective moment-of-inertia based on the reinforcement and the cracked section properties. The program calculates the elastic deflections based on the revised section properties. Specify the parameters.



Code

Select the design Code from the list. Refer to Slab deflections - Method of Calculation

Combinations

Specify the combinations to be used for calculating immediate or long-term deflections:

- **Immediate deflections**

  The program displays a list of all the combinations. To select one, move the to the relevant line and click the mouse; '**Compute'** is then displayed on the line. Repeat to select more.

  If the combination is a factored one, then set **Load factors:** **Use 1.0** to ignore the factors (i.e. all factors = 1.0) during the deflection calculation.

- **Long-term deflections**

  The program displays a list of all the combinations. Move the to the relevant line and click the mouse. Only one combination may be selected.

  Note:
  - the program always uses the factors as defined.
  - the long-term deflection displayed in the results is the elastic deflection for the selected load combination modified according to the effective moment-of-inertia and multiplied by the **Creep factor**.

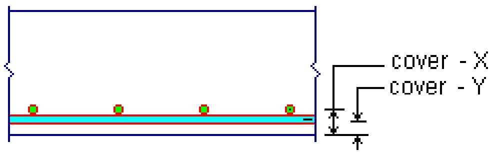
In-plane forces

Ignore in plane forces
the program calculates the reinforcement based on the bending moments only

Ignore in plane forces
the program calculates the reinforcement based on the bending moments **and** any tension/compression forces in the element.

Cover

Define the gross cover (to center of reinforcement) in both directions:



Reinforcement

Specify the slab reinforcement parameters. There are two main options:

Reinf. required for moments/forces
The program calculates the required/minimum area according to the parameters, then selects the diameter and spacing according to the min/max diameter and spacing min/increment values, as follows:

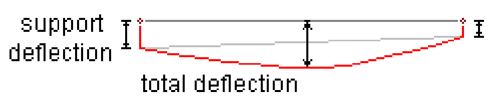
- The program initially tries minimum diameter with minimum spacing. If this combination is insufficient, it tries larger diameters with the same spacing.
- When an adequate diameter is found, the program searches for the maximum spacing (a multiple of the increment) with this diameter that provides sufficient area.

User defined reinforcement
Provide the user defined diameter and spacing at all locations in both directions. The program does not calculate required reinforcement and ignores minimum reinforcement requirements.

Note that different reinforcement areas may be specified for individual elements using the Options - Reinf. parameters by element option.

Note:

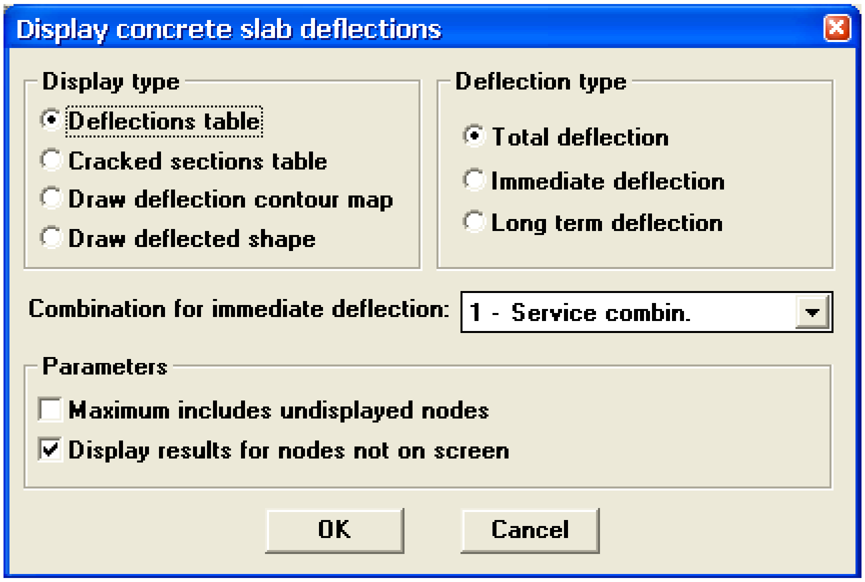
- the program recalculates the stiffness matrix for the slab elements based on the effective moment-of-inertia. Therefore, the program must solve the model again for ***every*** load combination considered for immediate or long-term deflections.
- the deflections displayed are the total deflections, i.e. support and slab deflections.



- Refer to Slab deflections - Method of Calculation

6.4.2 Display/print slab deflections

Select the display type and the deflections to display:



Menu parameters:

Deflection type

The program calculates the immediate deflection, the long-term deflection and the total deflection = immediate + long-term deflection.
Select the type.

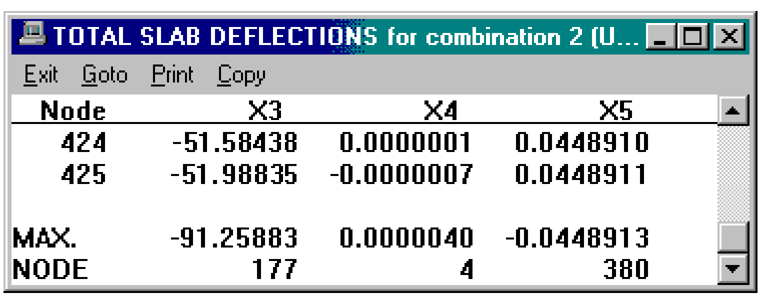
Combination for immediate deflection

Select the immediate deflection load combination (if more than one combination was selected for the immediate deflection calculation).

Output tables:

Deflection table

Display the deflections calculated according to the effective section properties:



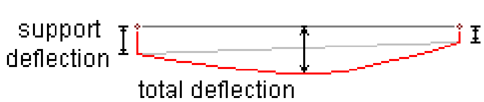
TOTAL SLAB DEFLECTIONS for combination 2 (U...

Exit Goto Print Copy

Node	X3	X4	X5
424	-51.58438	0.0000001	0.0448910
425	-51.98835	-0.0000007	0.0448911
MAX.	-91.25883	0.0000040	-0.0448913
NODE	177	4	380

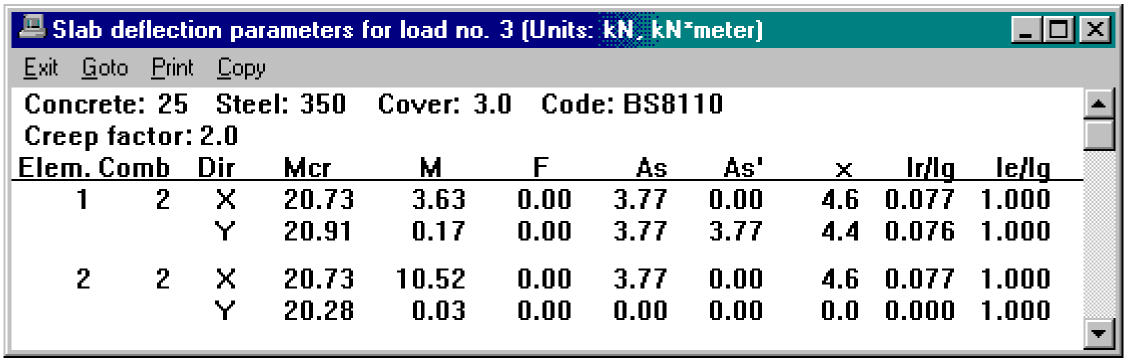
Note:

- the deflections displayed are the deflections, i.e. support and slab deflections.



Cracked section table

Display the parameters, the reinforcement and the various moments-of-inertia for each element:


Slab deflection parameters for load no. 3 (Units: kN, kN*meter)

Exit Goto Print Copy

Concrete: 25 Steel: 350 Cover: 3.0 Code: BS8110

Creep factor: 2.0

Elem.	Comb	Dir	Mcr	M	F	As	As'	x	Ir/Ig	Ie/Ig
1	2	X	20.73	3.63	0.00	3.77	0.00	4.6	0.077	1.000
		Y	20.91	0.17	0.00	3.77	3.77	4.4	0.076	1.000
2	2	X	20.73	10.52	0.00	3.77	0.00	4.6	0.077	1.000
		Y	20.28	0.03	0.00	0.00	0.00	0.0	0.000	1.000

Mcr = cracking moment
M = design moment
F = axial force
As = tension reinforcement (may be top or bottom)
As' = compression reinforcement
x = height of compression block
Ir = cracked moment-of-inertia
Ig = uncracked (gross) moment-of-inertia
Ie = effective moment-of-inertia

Refer also to:

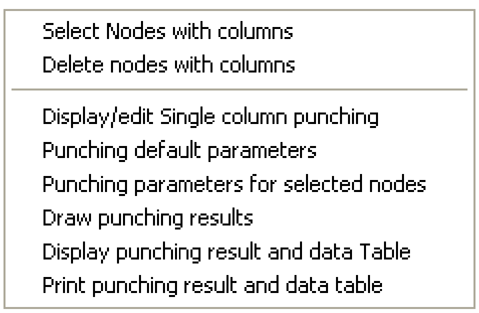
- contour map parameters 662
- deflected shape parameters 670

6.5 Punching

Calculate punching shear stresses in concrete slabs.

Select col...	Specify the location of columns 593 where punching stresses or average moments are to be calculated
Delete col...	Delete 593 the columns selected previously.
Single col...	Display/revise the punching data and display shear stress results for a single column 594.
Draw	Draw 606 the punching results superimposed on the graphic display.
Defaults	Specify the default punching parameters 601 for all columns.
Parameters	Specify punching parameters 602 for selected columns
Display ta...	Display the punching data 607 and results for all columns for the critical case/ combination or a selected case/combination
Print table	Print the punching data 607 and results.

or from the Menu bar:

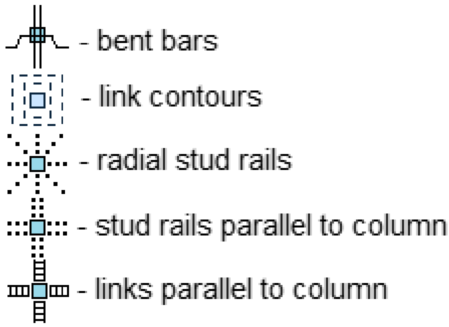


Note:
- the column location selection is common to the Punching and Average moment 613 options.
- refer also to Punching - general 591

6.5.1 General

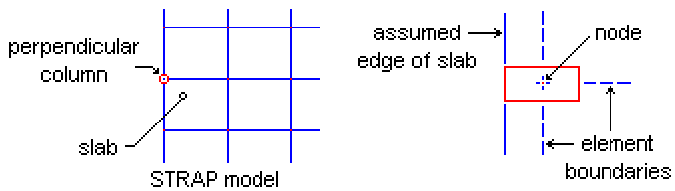
The program calculates the punching shear stress in the slab elements at all column or wall edge/corner locations according to the Code requirements and compares it to the allowable shear stress. If the actual stress exceeds the allowable, the program calculates the reinforcement required.

Five different types of punching shear reinforcement may be designed. They are:

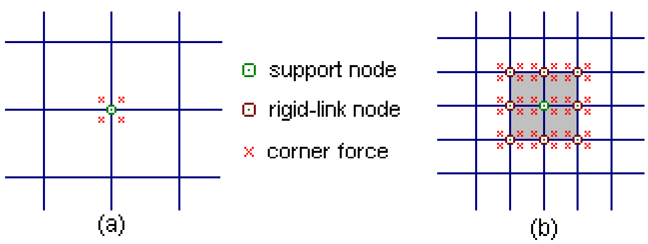


Note:

- all columns or walls edge/corner must be parallel to a global axis - the "height" axis
- when exterior columns extend the edge of the slab, the program extends the slab to the exterior face of the columns when calculating the shear perimeter

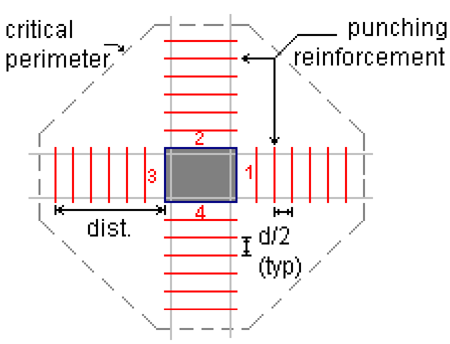


- The program calculates the shear force at the column by summing the corner forces at the support node - Figure (a). If rigid links (b) were defined at the nodes adjacent to the support to model the column - Figure (b), the program sums the corner forces of all elements attached to the linked nodes:



- The program calculates the effective punching shear at the first punching perimeter only (unless a column head is defined) and uses this shear value to compute the distance required for the reinforcement. This is a conservative approach.

The program calculates the required punching shear reinforcement.



Note:

- The area displayed in the punching stress tables 607 and the single column punching 594 screen is the ***estimated total*** area of all the links (1 to 4 in the figure above) equidistant from the columns,

assuming a spacing of d/2.

- The area displayed in the detailed results table is the ***exact*** total area based on the actual spacing.
- Distance is measured from the column face; if a column head is defined, the distance is measured from the edge of the head.

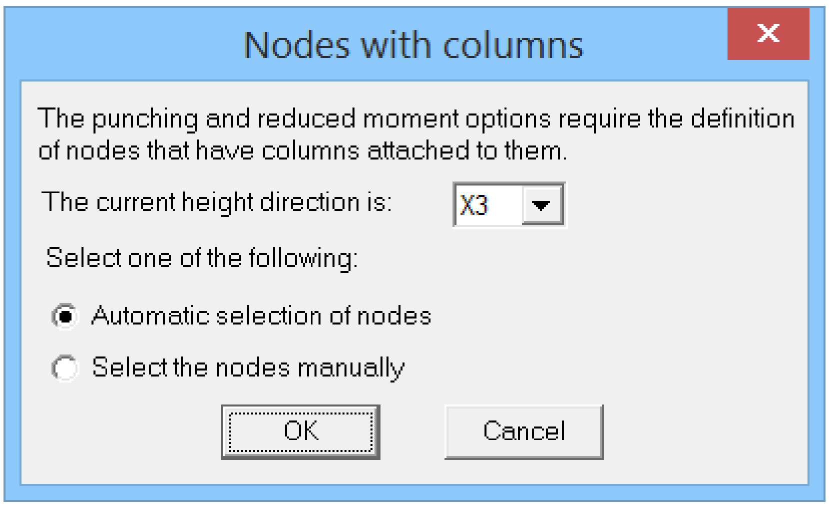
Refer to:

- Punching - Calculation method for more details.
- Punching - examples for detailed design calculations according to different Codes..

6.5.2 Select nodes with columns

Specify the location of columns where punching stresses or average moments are to be calculated (the selection is common to both options):

- ***Plane models***
  Select the nodes using the standard Node selection option
- ***Space models***



Select the global **Height direction**. The program identifies nodes at element surfaces perpendicular to this direction, including submodels. A node for punching calculation must be located on the element surface and one of the following:

- Restraint.

or

- Nodes that have beams parallel to the height axis connected to them (or < 30° from the height axis).

or

- Nodes located at wall edge or corner.

Automatic selection of nodes
The program will select all of the nodes described above.

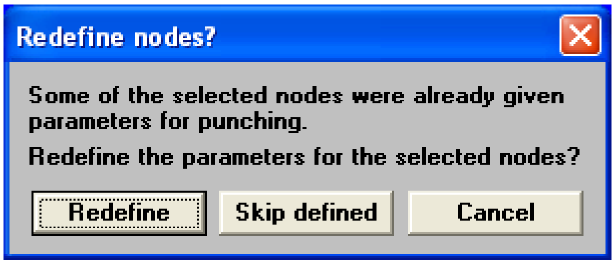
Select the nodes manually
Select the nodes using the standard Node selection option; only the nodes described above may be selected.

Note:

- For Columns:
  - The program calculates the default column type (**Centre/Edge/Corner**) and the default column dimensions when a column is defined using this option. These default values can be revised by the

user using the other options in this menu. However, the program does not compare the column punching data with the current model geometry. If the relevant data is subsequently revised in *STRAP* geometry (column dimensions, slab edge location, etc.), the column locations for the punching calculation must be selected again.
- Column locations must be defined here if reduced moments are to be used for Slab design and detailing in the Concrete postprocessor.

If some or all of the nodes you have selected already have columns defined at their location:



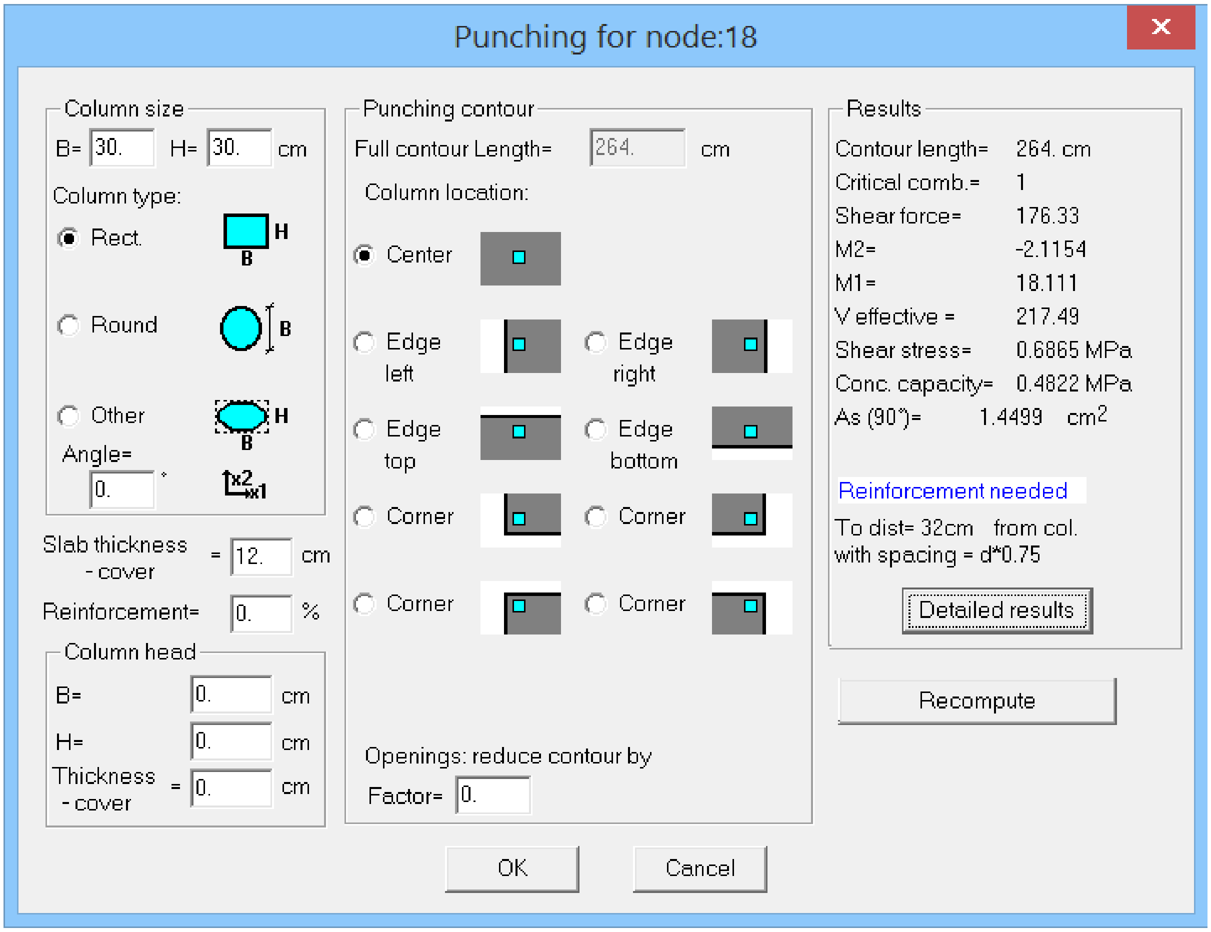
Select:

Redefine	The program recreates the columns and assign the default parameters and dimensions to them; all user-defined parameters and dimensions are erased.
Skip defined	The program creates columns only at new locations; the data for existing columns is not revised

6.5.3 Display single column punching

Display/revise the punching data and display shear stress results for a single column or wall edge/ corner.
For wall edge/corner results refer to Punching - Wall edge/corner data/results 599.

Note:
- the punching data for several columns can be revised simultaneously by selecting the Punching parameters 593 option
- the results for all columns can be displayed by selecting the Punching result and data table 607 option
- The results are compatible with the currently displayed data. If you revise the data you must click Recompute to recalculate and refresh the display.
- The reinforcement area on this screen displayed is the ***estimated total*** area of all the links/studs/etc ***assuming a spacing of d/2***. Click Detailed results to display the ***exact*** total area based on the actual spacing.

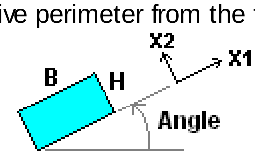


Column size

Define the column size. The program identifies the column shape and dimensions from the model geometry, if possible. The dimensions and orientation may be revised.

Note:

- the dimension **B,H** are relative to the ***global*** axes displayed at the bottom of the group box (note that the column may be rotated about these axis if a "**Beta**" angle was defined in geometry or **Angle** is defined in this option).
- The **Other** shape assumes a rectangular column when calculating **Veffective,** but unlike **Rectangular,** it allows the **Full contour length** to be revised. Note that if **Other** is selected for an edge/corner column, the **Full contour length** must be defined as if the column is **Center**; the program automatically calculates the effective perimeter from the full perimeter length
- The **Angle** value allows the column to be rotated about its axis. The program initially displays the "**Beta**" angle defined in geometry (default=0); a positive angle is counter-clockwise. The axes rotate with the section and the B,H values must be defined accordingly:

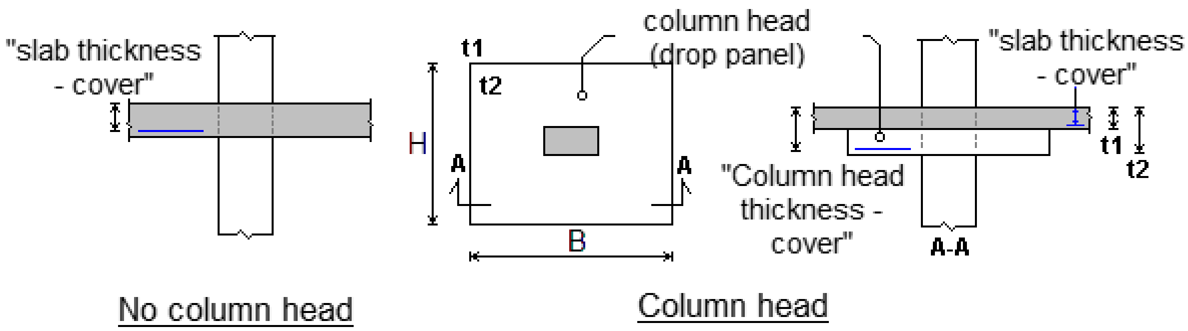


  The program modifies the **M1**, **M2** moment values required to calculate the effective punching stress according to the angle value.

Slab thickness / Column head dimensions & thickness

Define the slab thickness and the column head (drop panel) dimensions and thickness.

- the column head dimensions must be defined manually here, even if thicker elements were defined in geometry.
- a column head can be added here even if thicker elements were not defined in geometry



No column head:

- the program automatically calculates the default value for the effective depth ("thickness - cover") from the element thickness adjacent to the column and the cover value defined in Punching parameters 602.

Column head:

- if a column head is defined here, the program checks for the critical shear stress in both the head area and the surrounding slab.
- ***the program does not identify the column head dimensions or thickness even if thicker elements were defined and the dimensions must be entered manually***. The column head dimensions **B,H** are relative to the global axes and the column B,H dimensions (displayed at the bottom of the "Column size" group box). Column heads are rotated with the column.
- the default value for the effective ***slab*** thickness ("thickness - cover") is still calculated from the element thickness adjacent to the column. Therefore:
  - if thicker elements were defined:
    the default ***slab*** "thickness - cover" will be incorrect as it is calculated from **t2** adjacent to column. Copy this value to the column head "thickness - cover" and manually enter the slab "thickness - cover" calculated from **t1.**
  - if thicker elements were not defined:
    the default ***slab*** "thickness - cover" will be correct. Manually enter the column head "thickness - cover" calculated from **t2.**

Refer also to:

- Punching - general 591
- Punching - Calculation method

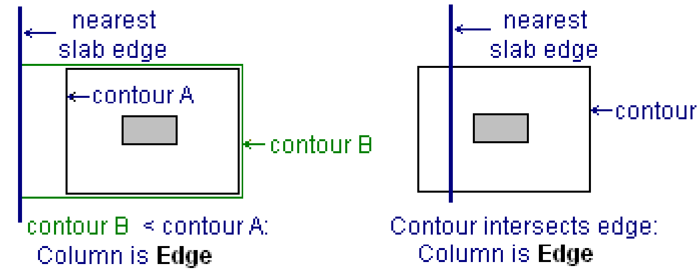
Punching contour

Define the column type - **Centre**, **edge** or **corner** - and the distance from the column centre to the slab edge.

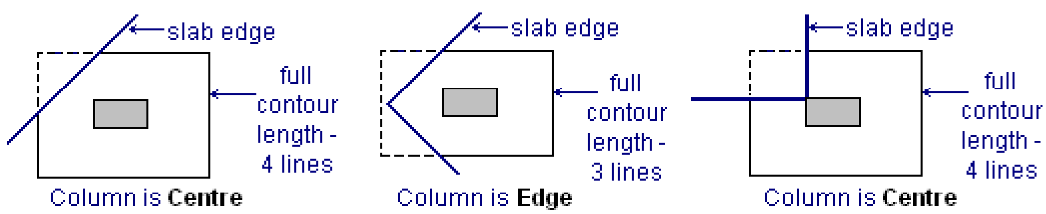
- The program uses the data to calculate the **Full contour length** at the top of the group box and the **Contour length** in the Results box.
- The side (**Left/right/top/bottom**) is important for **Edge** columns in certain codes where the sign of the moment influences the value of the effective shear
- the distance from the column centre to the slab edge is used to calculate the Effective contour perimeter.

Note:

- The initial column location type and edge distance are taken from the *STRAP* geometry. By default, the program assumes that all columns are **Center**, except in the following cases:



  and similarly for **Corner** columns
  Note that when exterior columns extend ***beyond*** the edge of the slab, the program extends the slab to the exterior face of the columns when calculating the shear perimeter (refer to Punching - General 591)
- the effective contour length when openings are present may be reduced by a **Factor**. (Factor = 0.1 will reduce the contour length by 10%).
- The **Full contour length** is always calculated automatically assuming that the column is "**Centre**" and may not be revised unless the column type is **Other**. The effective contour length is displayed in the Results box.
- If **Other** is selected for an edge/corner column, the **Full contour length** must be defined as if the column is **Centre**; the program automatically calculates the effective perimeter from the full perimeter length
- If the perimeter with four lines intersects an edge that is not parallel to a global axis, the program assumes that the column is "**Centre**" and automatically calculates the reduction factor required to reduce the contour length.



Refer also to:

- Punching - general 591
- Punching - Calculation method

Results

This box displays a result summary for the current column (data items may vary according to Code):

Contour length	-	the effective contour length used to calculate the punching stress, after all reductions, etc.
Critical comb	-	the design combination or load case
Shear force	-	sum of the corner forces (refer to Punching - general 591)
M1/M2/M3	-	the moments ***about*** the relevant global axes. **MB, MH** are displayed if the column is rotated.
Veffective	-	the effective shear force used to calculate the punching shear stress, according to the relevant Code clauses. Veffective = f(V,M)
Shear stress	-	the punching shear stress
Concrete cap.	-	the punching shear capacity, assuming no reinforcement

The program displays the calculation status:

Shear stress is OK - The shear stress is less than the capacity and no additional reinforcement is required

Reinforcement needed - The shear stress exceeds the capacity and reinforcement is needed. The required area and location are displayed.

Shear exceeds allowable - the Shear stress exceeds the maximum allowable shear stress; the slab thickness must be revised

Note:

- The reinforcement area displayed is the ***estimated total*** area of all the links/studs/etc, ***assuming a spacing of d/2***.
- The area displayed in the detailed results table 608 is the ***exact*** total area based on the actual spacing.

  Click on [Detailed results] to view.

Refer to Punching - Calculation method for more details

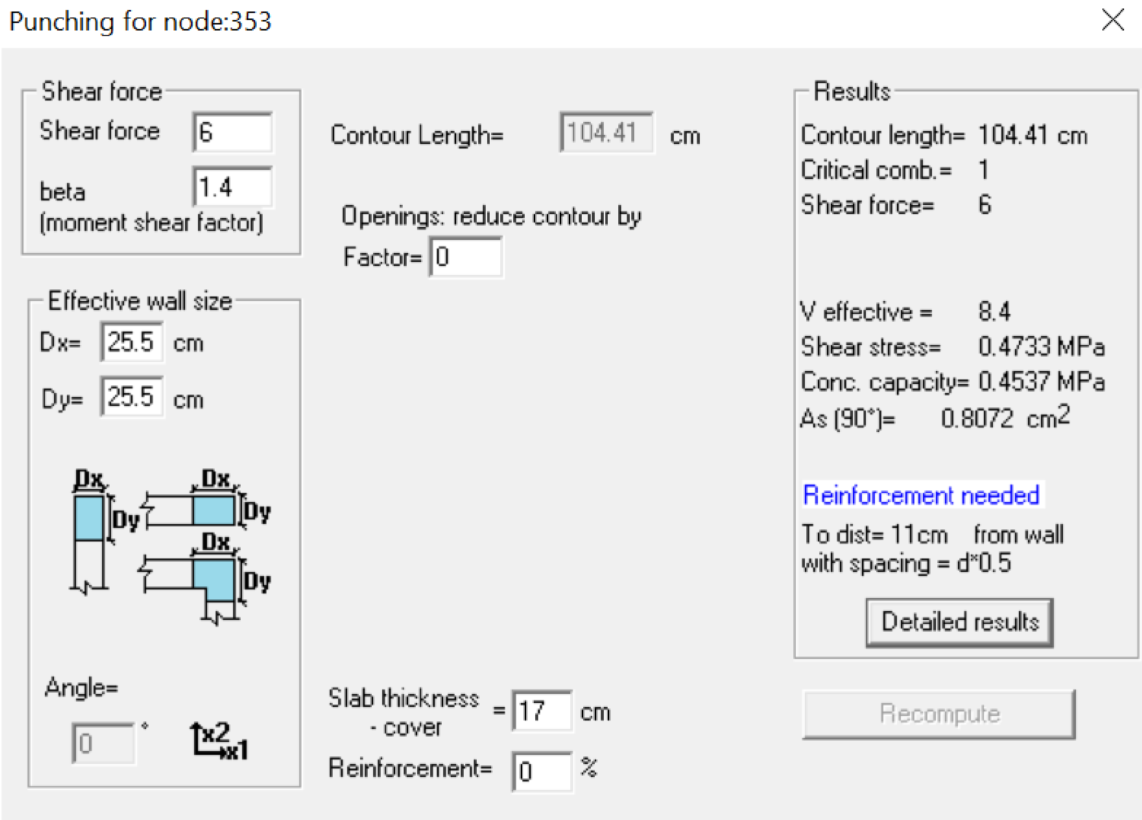
Recompute

Click this button to recalculate the punching stresses and reinforcement.

Note that the appearance of the button indicates the status of the calculation:

- [Recompute] The results are compatible with the data in the menu
- [**Recompute**] The parameters and data have been revised and the results are not compatible; click to Recompute.

6.5.3.1 Display wall edge/corner single column



Shear force

Specify the punching shear force. Include any coefficients based on code requirements.

Beta

Specify the factor beta to account for eccentric support reaction based on code requirements. Beta is multiplied by the punching shear force for design.

Effective wall size

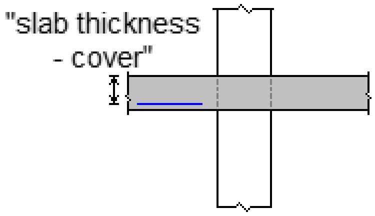
- Dx/Dy are the dimensions at the punching perimeter at the wall edge/corner faces.
- Angle = is the wall edge angle relative to the global axes.

Contour length

A display of the calculated punching contour length.

Slab thickness

Define the slab thickness minus the cover.



Note:
- the program automatically calculates the default value for the effective depth ("thickness - cover") from the element thickness adjacent to the column and the cover value defined in Punching parameters 602.

Refer also to:
- Punching - general 591
- Punching - Calculation method

Reinforcement

The allowable punching stress is generally a function of the longitudinal slab reinforcement. Enter an average reinforcement ***percentage*** = **100*(As/bd) %**. Refer to design assumptions.

Results

This box displays a result summary for the current column (data items may vary according to Code):

Contour length - the effective contour length used to calculate the punching stress, after all reductions, etc.
Critical comb - the design combination or load case.
Shear force - the inserted shear force.
Veffective - the effective shear force used to calculate the punching shear stress, according to the relevant Code clauses.
Shear stress - the punching shear stress.
Concrete cap. - the punching shear capacity, assuming no reinforcement.

The program displays the calculation status:

Shear stress is OK - The shear stress is less than the capacity and no additional reinforcement is required
Reinforcement needed - The shear stress exceeds the capacity and reinforcement is needed. The required area and location are displayed.
Shear exceeds allowable - the Shear stress exceeds the maximum allowable shear stress; the slab thickness must be revised

Note:
- The reinforcement area displayed is the ***estimated total*** area of all the links/studs/etc, ***assuming a spacing of d/2***.
- The area displayed in the detailed results table 608 is the ***exact*** total area based on the actual spacing.

  Click on [Detailed results] to view.

Refer to Punching - Calculation method for more details

Recompute

Click this button to recalculate the punching stresses and reinforcement.

Note that the appearance of the button indicates the status of the calculation:

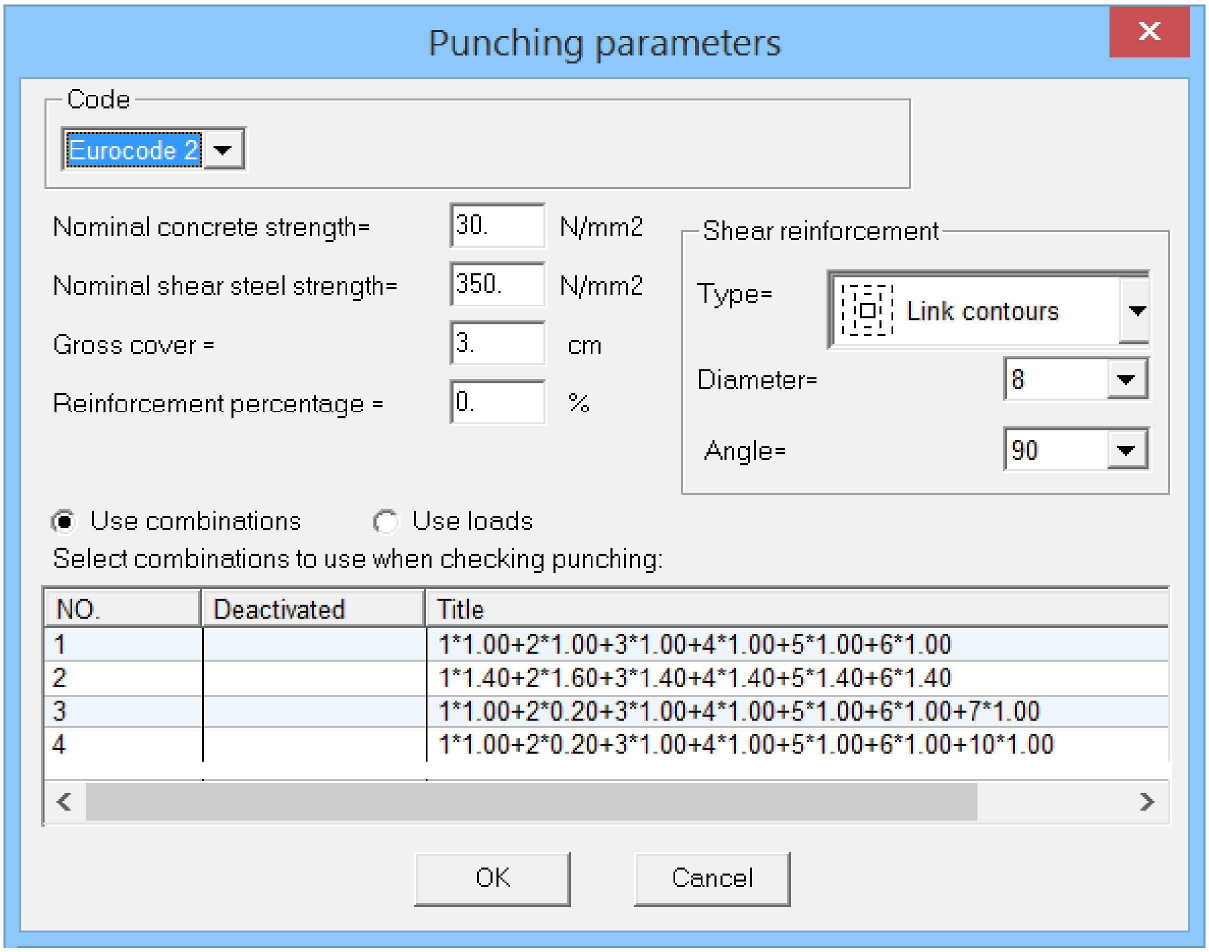
- 

The results are compatible with the data in the menu
- 

The parameters and data have been revised and the results are not compatible; click to Recompute.

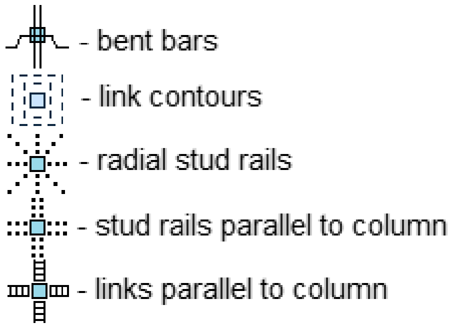
6.5.4 Punching default parameters

Define the default punching design parameters for all locations. Note that the type, diameter, angle, cover and the slab reinforcement percentage may be modified for individual column by selecting the Punching parameters for selected nodes 593 option.



Type

Select one of the following reinforcement types:



Diameter

Specify the diameter for all punching reinforcement.

Reinforcement angle

Specify the angle of the punching shear reinforcement (links/stirrups) - 45, 60 or 90 degrees

Reinforcement percentage

The allowable punching stress is generally a function of the longitudinal slab reinforcement. Enter an average reinforcement ***percentage*** = **100*(As/bd) %.** Refer to design assumptions.

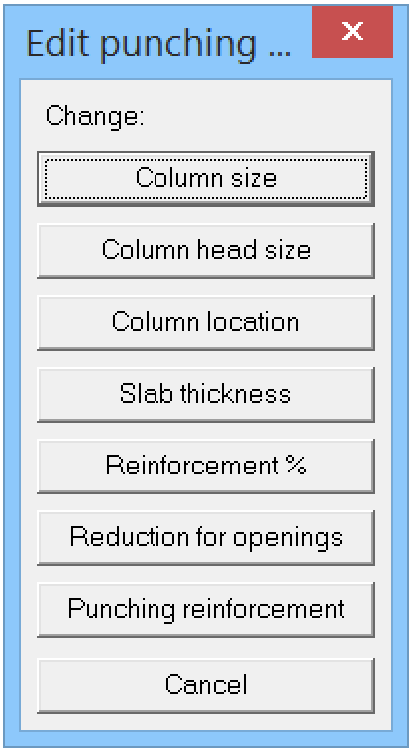
Combinations / loads - deactivate

The punching combination calculates the maximum punching shear stress for selected load CASES or COMBINATIONS.

- Select **Load cases** or **Combinations** (if defined)
- Click on any case/combination in the list box to "**Deactivate**" it or reactivate

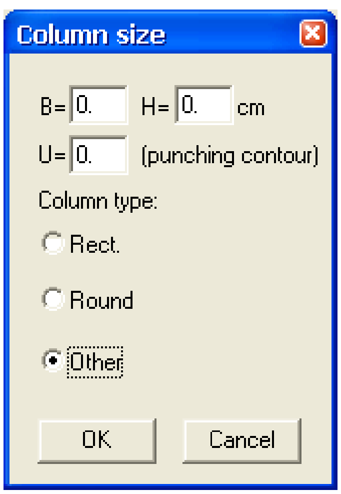
6.5.5 Punching parameters

Punching data and parameters may be revised simultaneously for several columns. Select one of the options from the following menu, define the revised data and then select the columns using the standard Node selection options.



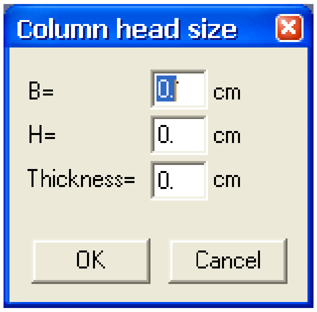
Column size

Specify the column type 595 and dimensions and then select the multiple column locations using the standard Node selection options. The option is identical to the one in Single column punching 594.



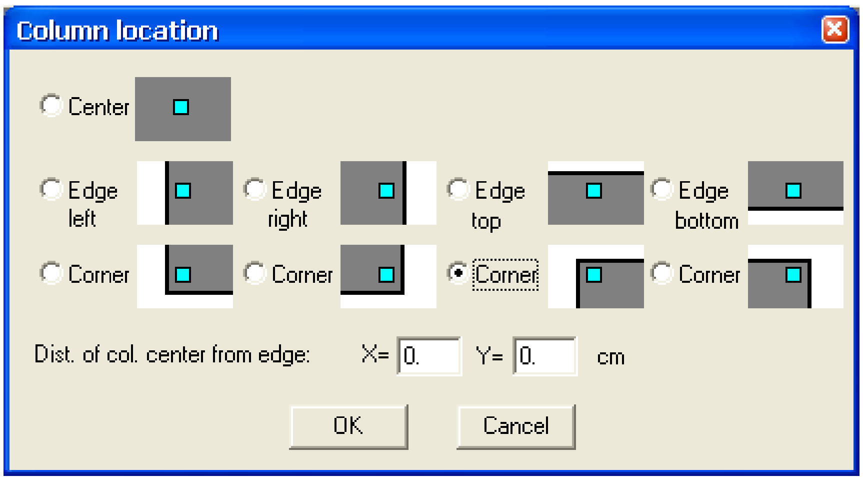
Column head size

Specify the column head 596 dimensions and then select the multiple column locations using the standard Node selection options. The option is identical to the one in Single column punching 594.



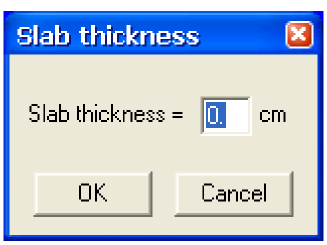
Column location

Specify the column location 595 type and then select the multiple column locations using the standard Node selection options. The option is identical to the one in Single column punching 594.



Slab thickness

Specify the slab thickness 596 dimensions and then select the multiple column locations using the standard Node selection options. The option is identical to the one in Single column punching 594.

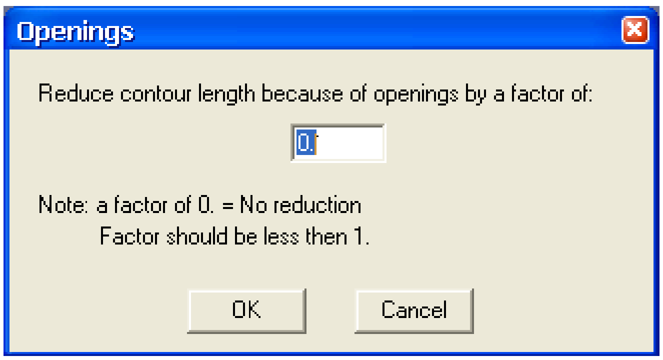


Reinforcement %

Specify the longitudinal slab reinforcement percentage and then select the multiple column locations using the standard Node selection options. The option is identical to the one in Punching parameters 604.

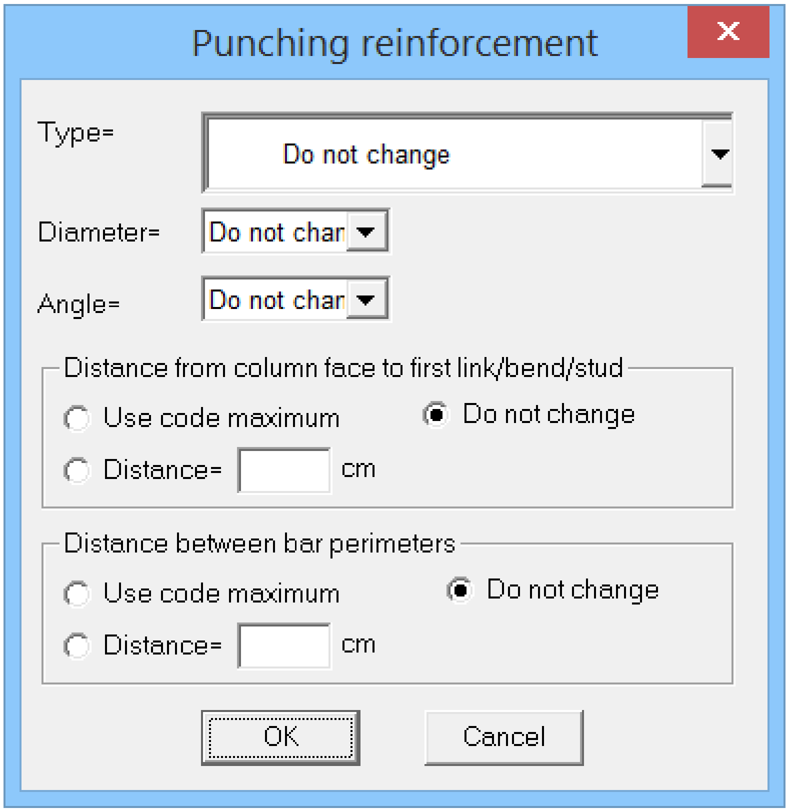
Reduction for openings

Specify the contour length reduction factor and then select the multiple column locations using the standard Node selection options. The option is identical to the one in Single column punching 596.



Punching reinforcement

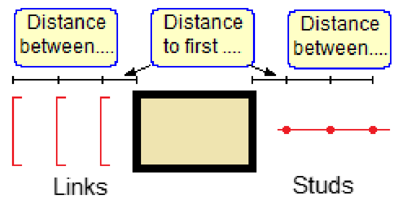
Specify punching reinforcement parameters for selected columns. These parameters override the default parameters 601



Distances:

The default distance to the first link/bend/stud and the distances between them are the Code maximum values:

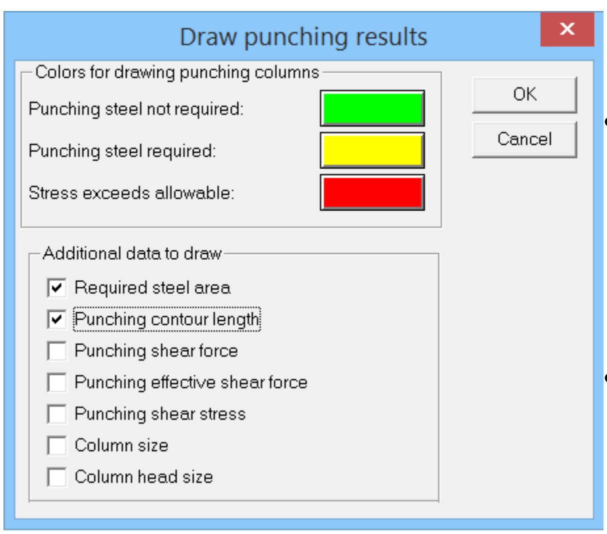
- Distance to first = 0.3d/0.5d (depending on the Code), rounded off to the nearest 50 mm.
- Distance between = 0.75d, rounded off to the nearest 50 mm.



Specify different distances for selected column. Note that the program does not check whether the distance entered complies with the Code limits.

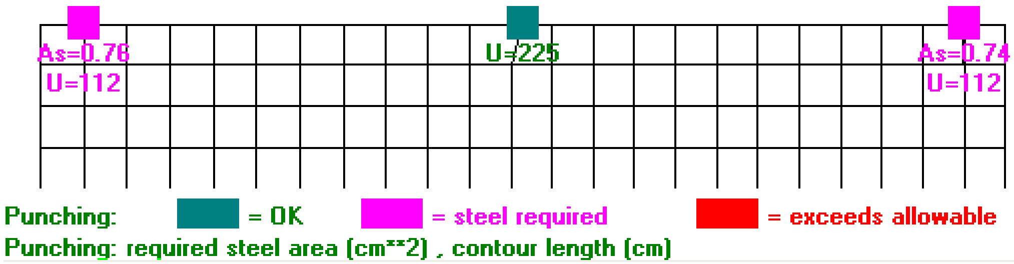
6.5.6 Draw punching results

Display the punching results superimposed on the graphic display:



- The results are colour coded according to three categories; select the colours.
- Select the results to display (more than one result type may be selected.

For example, required steel area and punching contour length:



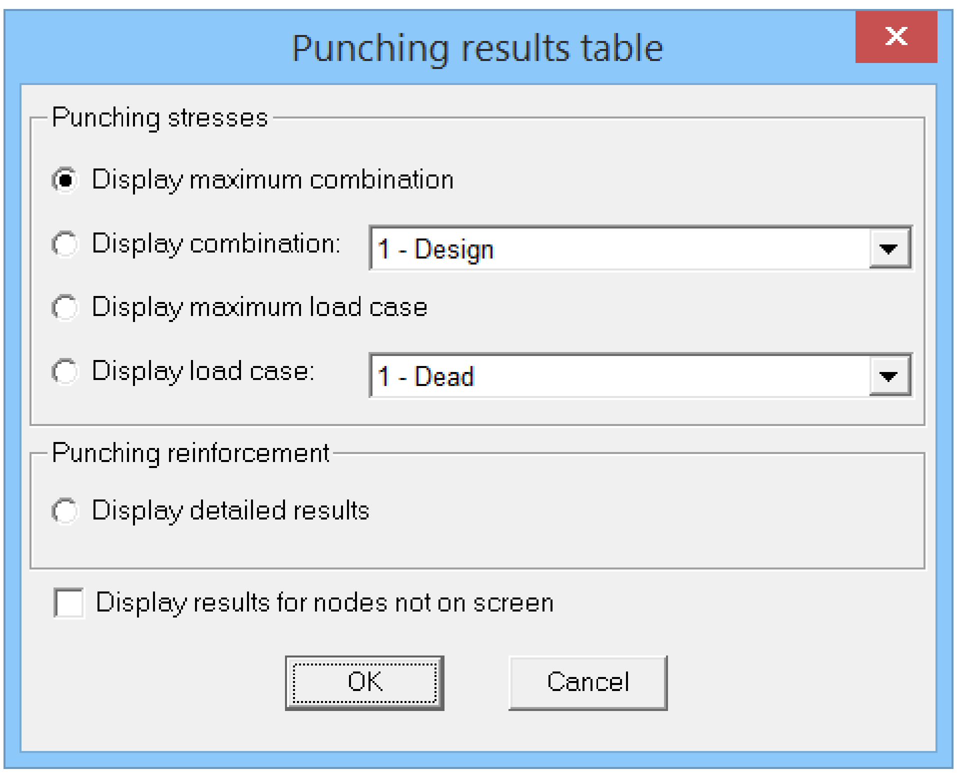
Note:

- For wall edge/corner the result will only be displayed if the shear force is specified. refer to Punching - Wall edge/corner data/results 599

6.5.7 Result & data table

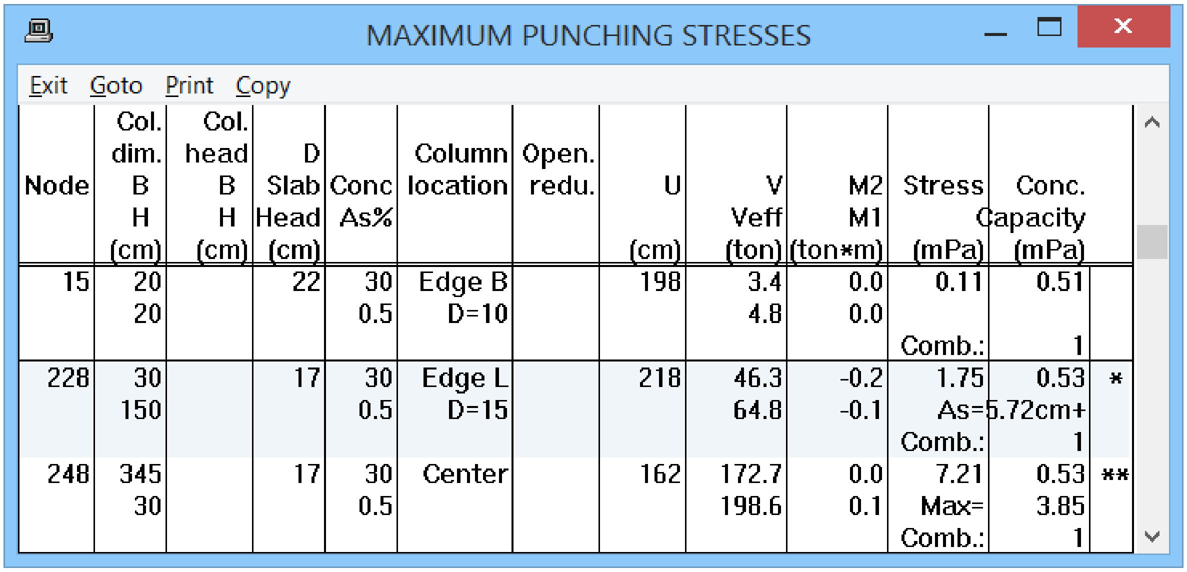
Display the punching data and results for all (displayed) columns for the critical case/combination or a selected case/combination.

- Select the case/combination (maximum or specified). Note that results may be displayed for all column locations or only those currently displayed on the screen



Punching stresses

The program displays the results and data table. For example:



MAXIMUM PUNCHING STRESSES

Exit Goto Print Copy

Node	Col. dim. B H (cm)	Col. head B H (cm)	D Slab Head (cm)	Conc As%	Column location	Open. redu.	U (cm)	V Veff (ton)	M2 M1 (ton*m)	Stress (mPa)	Conc. Capacity (mPa)	
15	20 20		22	30 0.5	Edge B D=10		198	3.4 4.8	0.0 0.0	0.11 Comb.:	0.51 1	
228	30 150		17	30 0.5	Edge L D=15		218	46.3 64.8	-0.2 -0.1	1.75 As=5.72cm+ Comb.:	0.53 1	*
248	345 30		17	30 0.5	Center		162	172.7 198.6	0.0 0.1	7.21 Max= Comb.:	0.53 3.85 1	**

where:

- **U** = effective contour length
- **column location: Centre/corner/edge**
  R = right; **L** = left; **T** = top; **B** = bottom
- **Conc. capacity**:
  ***** indicates that reinforcement is required; the area is displayed below the stress capacity
  ****** indicates that the stress exceeds the maximum allowable (i.e. the slab thickness must be increased); the max. allowable stress is displayed below the stress capacity.
- **As** = the reinforcement area = the ***estimated total*** area of all the links/studs/etc, ***assuming a spacing of d/2***.
  To display the ***exact*** total area based on the actual spacing, select Display detailed results 608.

Display detailed results

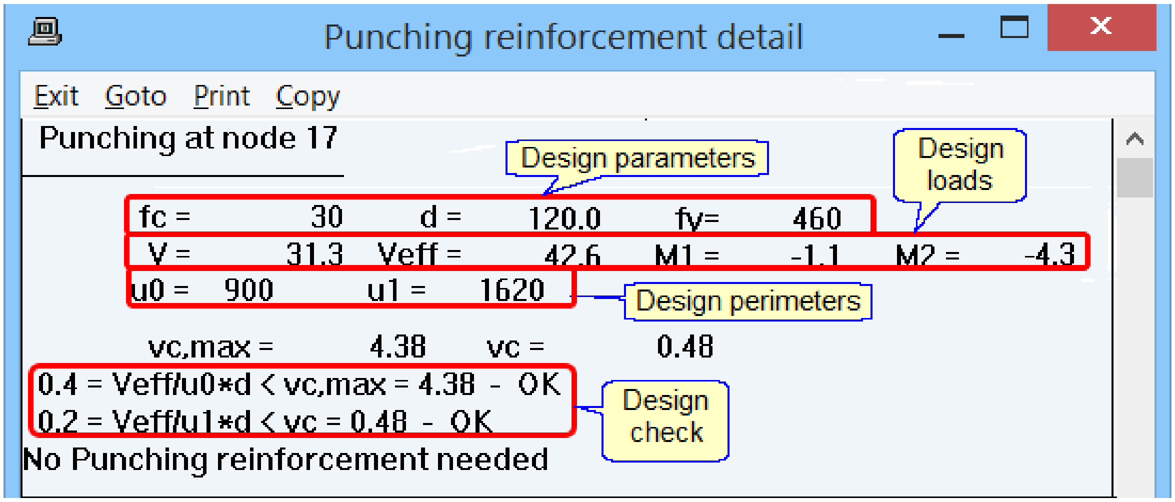
This option displays detailed punching results with a more exact calculation than in the other tables and shows a schematic arrangement of the reinforcement.

The program assumes by default a spacing of **0.75d** (the Code maximum) rounded off downwards to the nearest 50 mm. The value can be revised for selected columns in the Parameters - punching reinforcement 605 option.

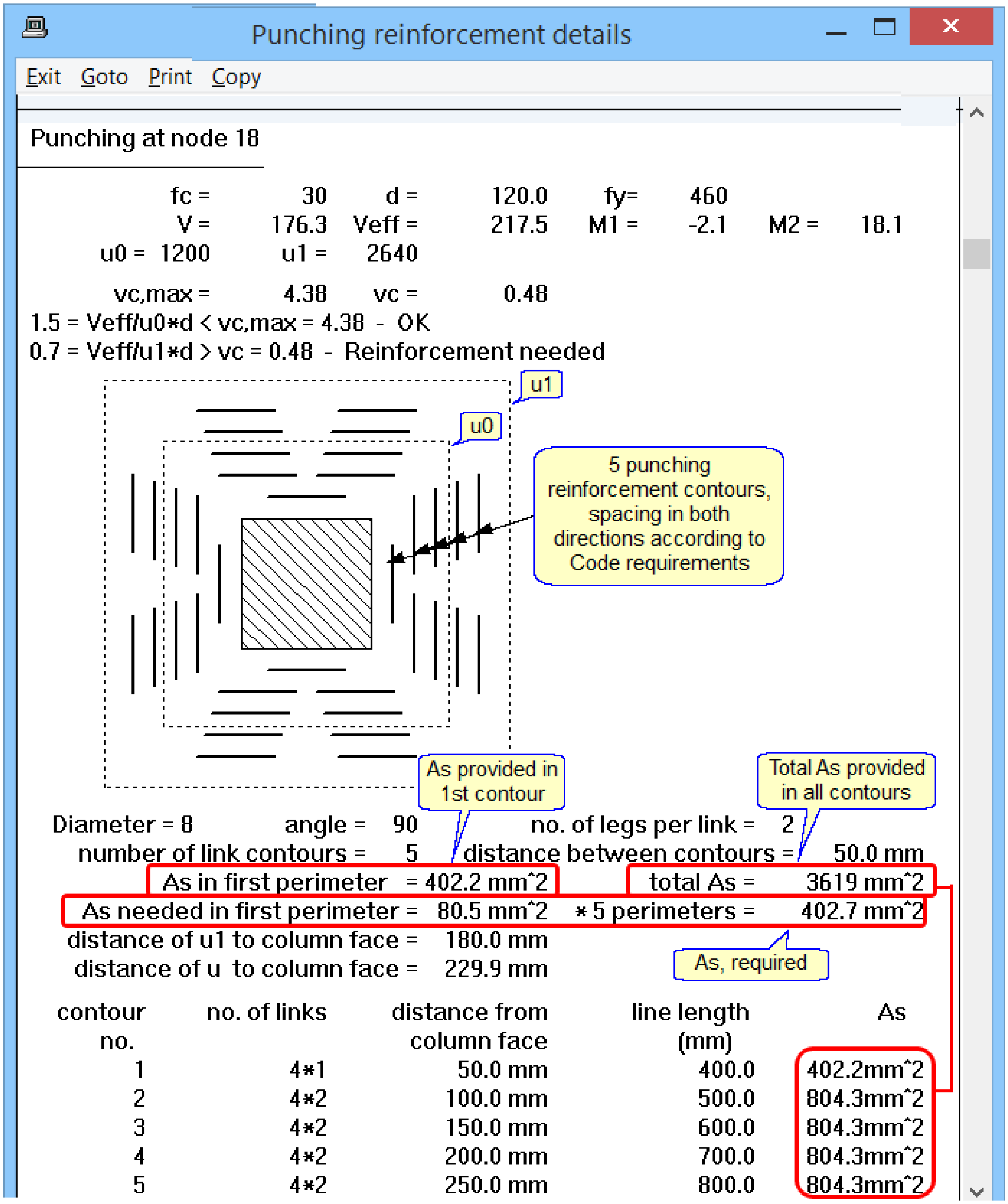
Examples:

- the examples are for slabs designed according to EC2.
- results are shown for three cases: (1) no punching reinforcement required, (2) reinforcement required, (3) no solution:

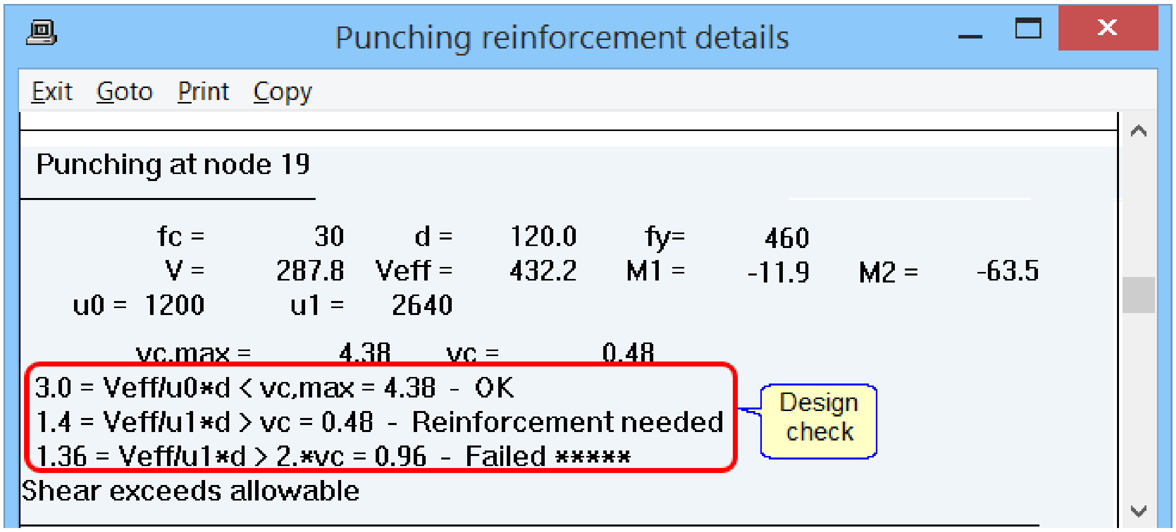
- No punching reinforcement required:



- punching reinforcement required:


Punching reinforcement details
Exit Goto Print Copy
Punching at node 18
fc = 30 d = 120.0 fy= 460
V = 176.3 Veff = 217.5 M1 = -2.1 M2 = 18.1
u0 = 1200 u1 = 2640
vc,max = 4.38 vc = 0.48
1.5 = Veff/u0*d < vc,max = 4.38 - OK
0.7 = Veff/u1*d > vc = 0.48 - Reinforcement needed
u1
u0
5 punching reinforcement contours, spacing in both directions according to Code requirements
As provided in 1st contour
Total As provided in all contour
Diameter = 8 angle = 90 no. of legs per link = 2
number of link contours = 5 distance between contours = 50.0 mm
As in first perimeter = 402.2 mm^2 total As = 3619 mm^2
As needed in first perimeter = 80.5 mm^2 * 5 perimeters = 402.7 mm^2
distance of u1 to column face = 180.0 mm
distance of u to column face = 229.9 mm
As, required
contour no. no. of links distance from column face line length (mm) As
1 4*1 50.0 mm 400.0 402.2mm^2
2 4*2 100.0 mm 500.0 804.3mm^2
3 4*2 150.0 mm 600.0 804.3mm^2
4 4*2 200.0 mm 700.0 804.3mm^2
5 4*2 250.0 mm 800.0 804.3mm^2

- No solution



6.5.8 Examples

For calculation examples according to different design codes, refer to Punching - examples.

6.6 Moment reduction

Calculate reduced moments at columns for concrete elements:

Select col...	Specify the location of columns 593 where punching stresses or reduced moments are to be calculated.
Delete col...	Delete 593 the columns selected previously.
Use average	Calculate the average of any result 612 at any point across a strip perpendicular to the relevant result axis and use this average result for all other calculations
Use reduct...	Specify the method to reduce the moments 613 adjacent to the supports.
rectangle	Define the area 614 over which the program reduces the results.
moment v...	Display 616 a table listing the rectangle dimensions and the reduced moment values for each load case.
Print mom...	Print 616 a table listing the rectangle dimensions and the reduced moment values for each load case.
Hatch rect.	click on this icon to display the defined rectangles as hatched areas on the graphic display.

or select the **Slab** options from the menu bar:

Solve concrete slabs deflections
Display slabs deflections
Print slabs deflections

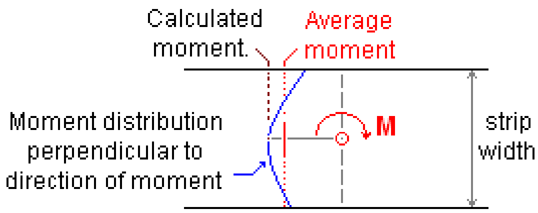
Average results along strips
Use reduced column moments in all results
Define rectangle size for reduced moments
Display reduced column moments table
Print reduced column moments table
Hatch column rectangles

Display Cracks results
Print Cracks results
Display Cracks Detailed results
Print Cracks Detailed results

6.6.1 Average results - strip

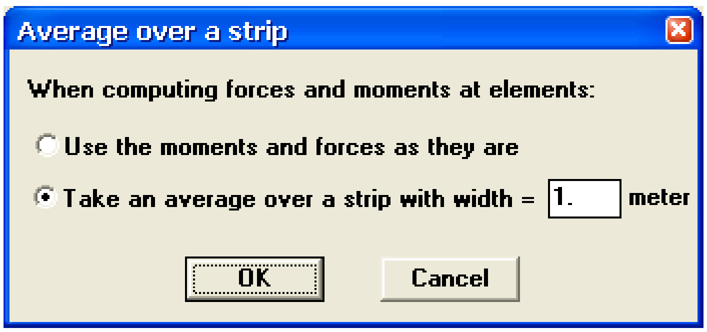
Calculate the average of any element result at any point across a strip perpendicular to the point and the use this average result for all other calculations. This option smooths out local jumps or concentrations at corners, etc, caused by the finite element method (see note below). The reduction is carried out over the entire model.

A simple example:

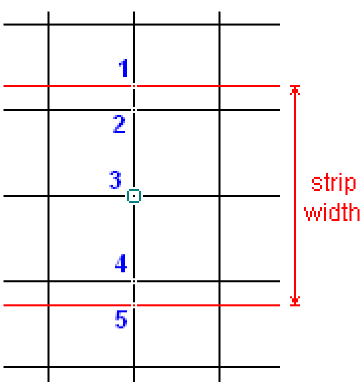


- M_x, F_x - the program averages all the results along a ***perpendicular*** line in the 'y' direction and calculates a weighted average;
- M_y,F_y - the perpendicular line is in the 'x' direction.
- M_{xy}, F_{xy} - two perpendicular lines are used, one in each direction.

Define the strip width:



For example, calculate an average result at point 3 in the following example:



The strip cuts through the two adjacent elements so the program has five result values available:

- 2,3,4 : at the element corner nodes
- 1,5 : where the strip lines cut the element edges.

The program calculates a weighted average (relative to the length associated with each point) of the results at the five points. This average is displayed in all result options.

Note:

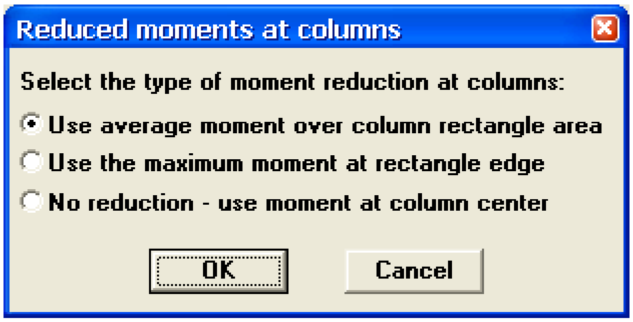
- the Reduced moments 613 option may be used together with the Average moment option; the program first calculates the Average moments at the relevant points and then calculates the Reduced moments. Refer to Results along a Line - General 669.
- this option should be used only to smooth out local jumps or concentrations caused by the calculation method that do not exist in reality; it should not be used when there are actual large variations in the results in the perpendicular direction. Care should also be taken to define a realistic strip width; a large width will always reduce the design values excessively (the influence of the strip width can easily be verified in the Contour map option).

6.6.2 Reduced moments

Finite element analysis usually gives exaggerated moment values at support nodes because the support is represented by a perpendicular line element with a zero dimension. In theory, a zero dimension support generates an infinite moment.

Use this option to 'reduce' the moments adjacent to the supports. The reduced values will be used for all output options, including "**Sum results over a strip** ..."

- select one of the following options:



Use reduced moment ...
The program calculates and uses an average moment values over the defined rectangle area

Use maximum moment ...
The program uses the maximum moment value on the defined rectangle perimeter

For a detailed explanation and examples, refer to Results along a Line - General 669.

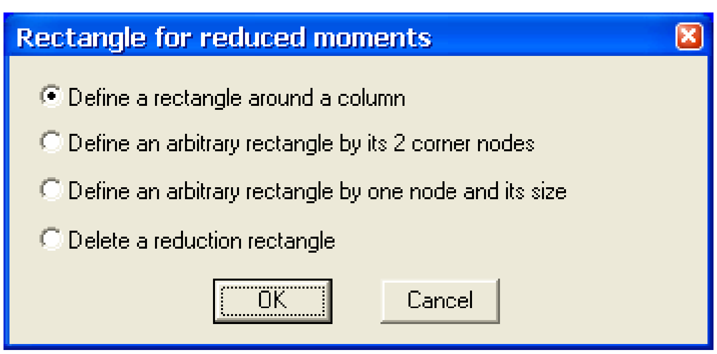
- Define a rectangular area adjacent to support nodes (select **Define rectangle size ....**)
- the Rectangle dimensions and the reduced moments can be displayed using the **Display reduced column moments table** option. Note that the reduced moments will also be displayed by all the graphic and tabular output options.

Note:

- if the ☑ **Use Wood & Armer moments** option is selected, the program first calculates separately the reduced Mx, My and Mxy moments, then calculates the reduced Wood & Armer moments from these values.

6.6.3 Define rectangle size

Define the area over which the program reduces the results:

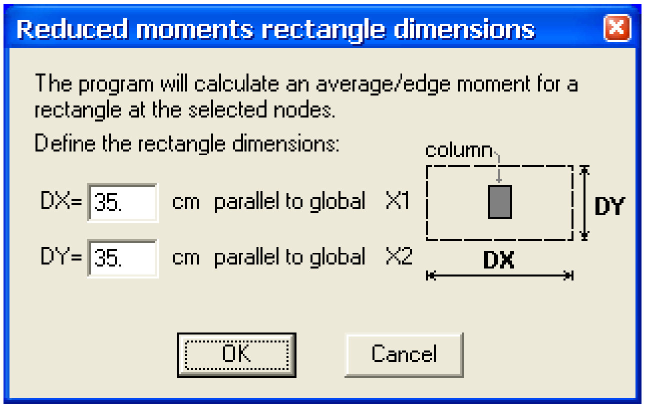


Define a rectangle around a column

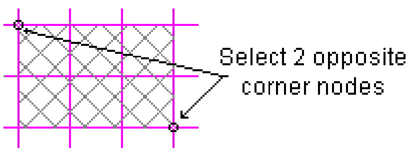
- select the column locations; note that the 'Punching' and 'Reduced moment' options use the same selected columns, i.e. any columns specified for punching will automatically be used for the Reduced moment option.
  - if no columns have been selected in the 'Punching' option, the program displays the Punching column selection 593 menu,

- otherwise, select additional individual columns using the standard node selection option
  Note that in space models only nodes with perpendicular beams or restraints can be selected

- Define the rectangle dimensions for the selected columns:

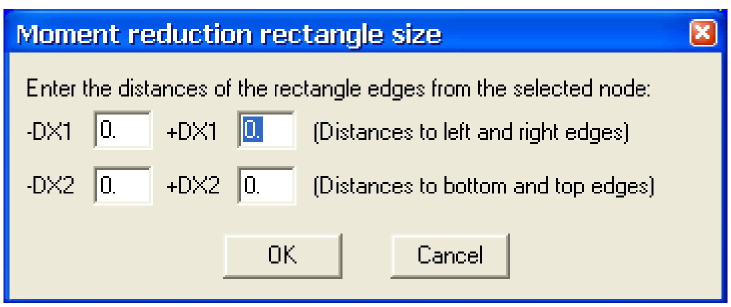


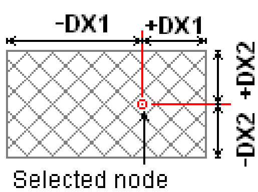
Define an arbitrary rectangle by corner nodes



Define an arbitrary rectangle by one node and its size

First select a node, then define the rectangle dimensions. Note that the rectangle does not have to be symmetric about the node:



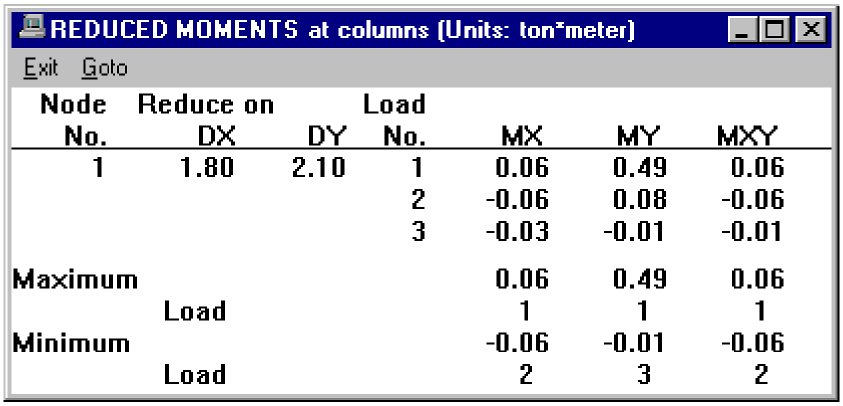


Note :
- set ✓ **Hatch column rectangles** to display the defined areas on the screen
- do not define overlapping rectangles; unexpected results may be displayed
- the global directions corresponding to DX and DY depend on the direction specified for the "Height axis":
  - height axis = X3: DX is parallel to X1; DY is parallel to X2
  - height axis = X2: DX is parallel to X3; DY is parallel to X1

- height axis = X1: DX is parallel to X2; DY is parallel to X3

6.6.4 Display reduced moments

Display the Rectangle dimensions and the reduced moment values for each load case. Note that the reduced moments are also displayed by all the graphic and tabular output options.


REDUCED MOMENTS at columns (Units: ton*meter)

Exit Goto

Node No.	Reduce on DX	DY	Load No.	MX	MY	MXY
1	1.80	2.10	1	0.06	0.49	0.06
			2	-0.06	0.08	-0.06
			3	-0.03	-0.01	-0.01
Maximum				0.06	0.49	0.06
	Load			1	1	1
Minimum				-0.06	-0.01	-0.06
	Load			2	3	2

Note:

- the global directions corresponding to DX and DY depend on the direction specified for the "Height axis":
  - height axis = X3: DX is parallel to X1; DY is parallel to X2
  - height axis = X2: DX is parallel to X3; DY is parallel to X1
  - height axis = X1: DX is parallel to X2; DY is parallel to X3'

For a detailed explanation and an example, refer to Results along a Line - General 669.

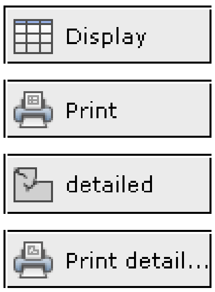
6.7 Crack width

This option checks crack widths or calculates reinforcement to limit crack widths in reinforced concrete elements according to:

- BS8007:1987 - "Design of concrete structures for retaining aqueous liquids".
- Eurocode 2: 2004.

Refer to:

- Crack widths - General 623
- How to use this module 624



Calculate 617 and display 620 crack widths and reinforcement for the elements according to the parameters specified in this option.

Calculate 617 and print 620 crack widths and reinforcement for the elements according to the parameters specified in this option.

Display 620 detailed crack widths and reinforcement calculations for for a single selected element

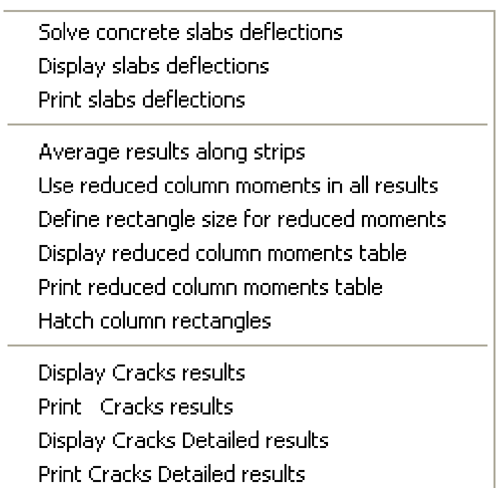
Print 620 detailed crack widths and reinforcement calculations for for a single selected element

Note:

- The **Crack width** options are displayed only if the module has been installed (contact your *STRAP* dealer).

---

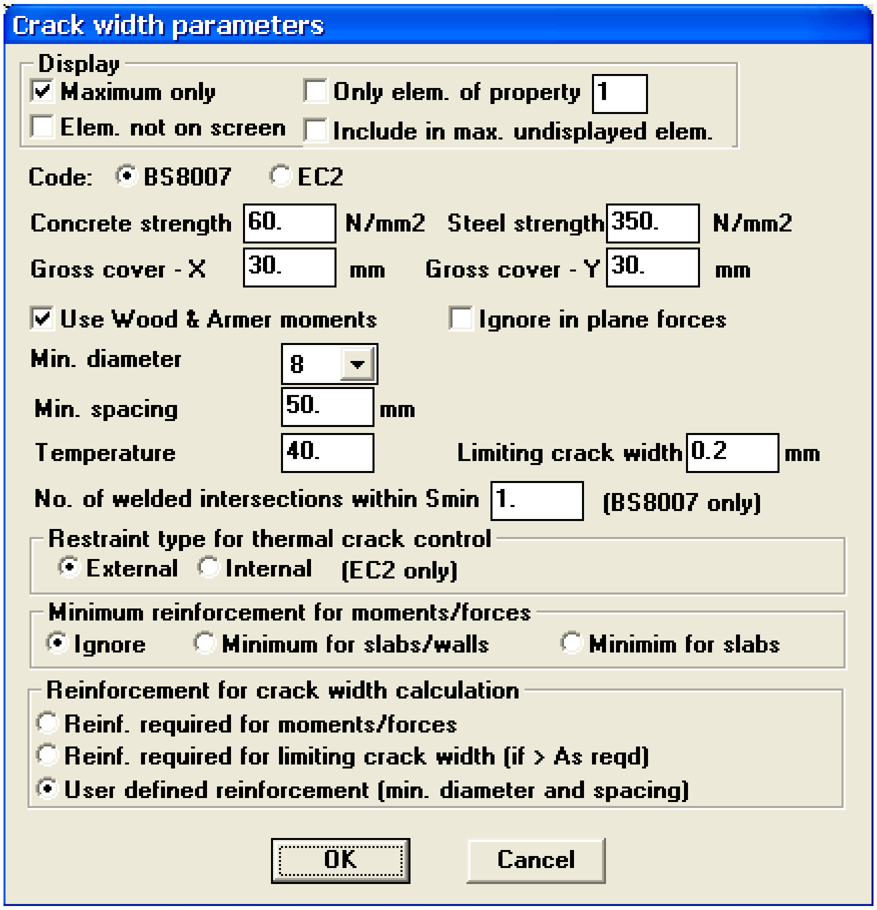
or select the **Slab** options from the menu bar:



6.7.1 Display/print

Calculate and display crack widths and reinforcement for the elements according to the parameters specified in the menu. Refer to Results table 620 for an explanation on the output table.

Note that the reinforcement diameter and spacing parameters are the default parameters for the entire model; different parameters for individual elements may be specified using the Reinf. parameters by element 580 option in the Options 571 pulldown menu.



Display

The following display options are available:

- **Maximum only**
  - ☑ for each element, display results only for the maximum combination (only if no. of combinations > 2)
  - ☐ for each element, display results separately for all combinations
- **Only element of property**
  - ☑ display only results for the property group specified in the adjacent box
  - ☐ display results for elements in all property groups
- **Elements not on screen**
  - ☑ Display results for the entire model
  - ☐ Display results only for those elements currently displayed on the screen
- **Include in maximum undisplayed elements**
  - ☑ the maximum crack widths for the entire model, displayed at the end of the table, will be calculated from the crack widths for all elements in the model.
  - ☐ the maximum crack widths will be calculated only from those elements currently displayed.

Code

Select BS8007 or EC-2

Diameter/spacing options

Specify default values for the range of reinforcement diameters and allowable reinforcement spacings. The values specified in this option will be applied by default to all elements in the model. Different values may be specified for selected elements with the **Reinf. parameters by element** option in the **Options** pulldown menu.

Define the following parameters:
- Diameter: select the minimum and maximum diameters
- Spacing: specify the minimum spacing and increment
  Example: minimum spacing = 75 mm and spacing increment = 100 mm.
  allowable spacings are 75, 100, 200, 300, ... mm.

Note:
- **Reinf. required for moments/forces**
- **Reinf. required for limiting crack width**
  The program selects reinforcement for each element from the range specified so that that the resulting crack width is as near as possible to the limiting crack width, but not less than the reinforcement required for moment and axial forces.
- **User defined reinforcement**
  The program calculates the crack width corresponding to the **Minimum diameter** and **Minimum spacing** values specified in this option. Maximum diameter and Increment values are ignored.

Cover

Specify the **gross** cover (to centre-of-gravity of the reinforcement) according to the displayed units.

Note:
- different values may be specified in the X and Y directions, where X and Y refer to the Element results coordinate system.

Temperature

Specify the total temperature drop ($T_1 + T_2$), where:
- T_1 = the fall in temperature between the hydration peak and ambient
- T_2 = additional fall in temperature due to seasonal variations

For more details, refer to:
- BS8007, Section A.3
- EC2 - CIRIA C660, Section 3.2.1

Limiting crack width

Specify the allowable crack width for the model.

Note:
- **Reinf. required for moments/forces**
- **Reinf. required for limiting crack width**
  The program calculates the reinforcement required to limit the crack width to the value specified here (but not less than the minimum reinforcement).
- **User defined reinforcement**
  This parameter is ignored by the program; the program calculates the actual crack width corresponding to the current reinforcement

Restraint type (EC-2 only)

Select "Internal" or "External" restraint. Refer to CIRIA C660, Section 3.2

Welded intersections (BS8007 only)

Specify **nw**, the number of welded intersections within the length smin (normally 1 or 2). Refer to Code section A.3.

Reinforcement for crack width calculation

Three different reinforcement design options are available:

Reinf. required for moments/forces
Calculate and detail the reinforcement required for moments and forces only. This option ignores the crack width requirements of the Code, i.e. the resulting crack width may be greater than the allowable crack width.

Reinf. required for limiting crack width
Calculate and detail the reinforcement required to limit the crack width according to the requirements of the Code. The reinforcement listed in the results will not be less than the reinforcement required for moments and forces.

User defined reinforcement
The program will calculate crack widths resulting from user defined reinforcement (minimum diameter and spacing).

Refer to:
- Results 620. for help on the results table
- Crack widths - General 623 for help on the calculation method
- How to use this module 624

6.7.1.1 Results table

The program displays results for all elements according to the options specified. The results are displayed separately for each face (+x3/-x3), each direction (X/Y) and each combination (or Max.only). For example:

REINFORCEMENT AND CRACKING BY BS8007 (Units: mm)

Exit Goto Print Copy

Concrete: 60 Steel: 350 Cover: 30.0 Code: BS8007 (Wood&Armer moments)

Allowed crack: 0.2mm Welded inters.: 1 Temperature: 40.0

		Reinforcement				Crack width				max. spacing			
ele	comb	-X	+X	-Y	+Y	-X	+X	-Y	+Y	-X	+X	-Y	+Y
984	1	8@50	8@50	8@50	8@50	0.10	0.00	0.13	0.00	44	44	44	44
	2					0.14	0.00	0.17	0.00				

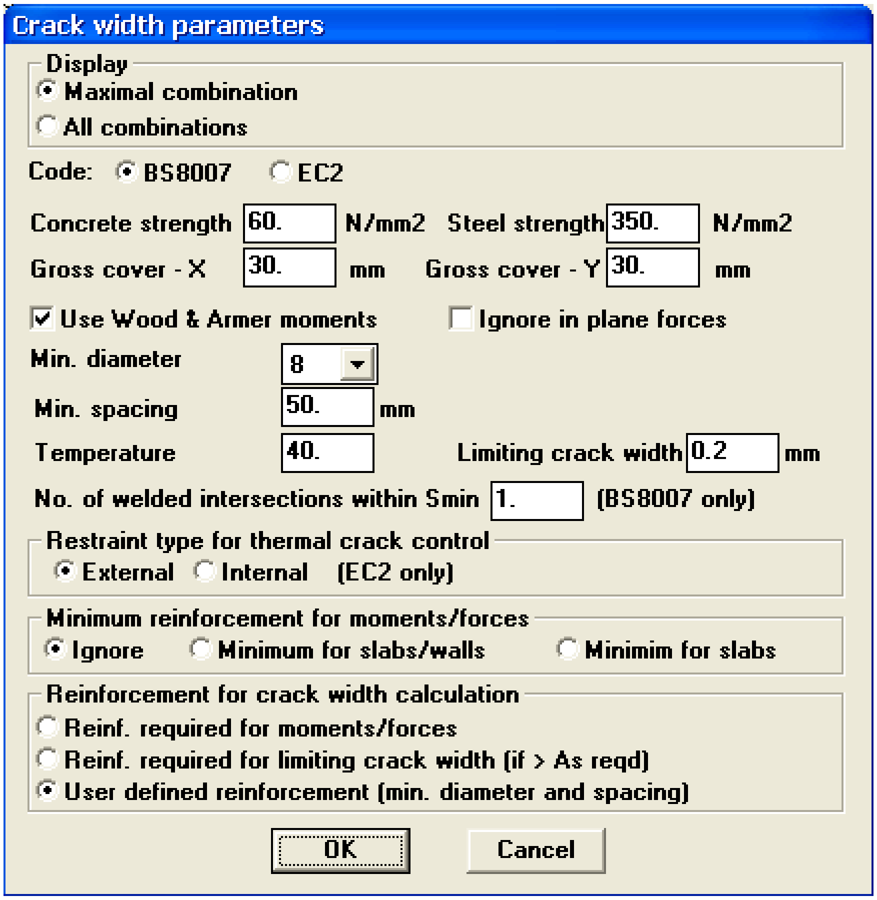
where:

Reinforcement	=	diameter and spacing of current reinforcement. In the above example, the reinforcement is governed by the moments and forces in the element and is greater than the reinforcement required to limit the crack widths to the allowable value.
Crack width	=	the crack width corresponding to the current reinforcement.
Max. spacing	=	the spacing for the current diameter that limits the crack width to the allowable value.

6.7.2 Detailed results

Display detailed crack widths and reinforcement calculations for for a single selected element. The calculations are done according to the parameters in the menu. Refer to Detailed results table 621 for an explanation on the output table.

Note that the reinforcement diameter and spacing parameters are the default parameters for the entire model; different parameters for individual elements may be specified using the Reinf. parameters by element 580 option in the Options 571 menu.



Select an element using the standard element selection option.

Display

Select one of the following:

Maximal combination
display the detailed results only for the load combination that generates the maximum crack width

All combinations
display the detailed results only for all load combinations

Refer to:

- Detailed results 621. for help on the detailed results table
- Crack widths - General 623 for help on the calculation method
- How to use this module 624

6.7.2.1 BS8007 - Detailed results

The program displays detailed calculations for a selected element. The results are displayed separately for each face (+x3/-x3) and each direction (X/Y). For example:

```
Detailed results for element no. 105
ELEMENT 105      crack at +x3 face in +X direction

LOAD 3
Mx=    -19.00                        Fx=       3.21
As Required =4773            Provided:  16@42       As=4773
     Rhocrit  =0.0043       Zone height=  100      Asmin=425        < As
       Smax =67.4                                  Wmax=0.01        < 0.20

    Service    Mx =      10.35   Fx =      -3.54   reinf. at other side 16@100
    Acr= 28.7     Cover= 30.0         H= 200.0       x=89.9
ε1= 0.001064   ε2   0.000053   εm  0.001011        Fs= 15480.48    Fc=1159.51
                                                    W=0.09       < 0.20
```

where:

Mx, Fx, etc. = factored moment and force corresponding to the relevant face/direction

As req'd = reinforcement area required to satisfy moment and force requirements. Refer to Reinforcement - method of calculation for more details

Provided = diameter and spacing (ϕ16@42 cm in above example) of reinforcement provided and corresponding area

Rhocrit = ρ **crit** = critical (minimum) ratio of steel to the gross area of the concrete section. Refer to Code section A.2.

Zone ht. = Height of 'service zone'. Refer to Code section A.2 and Figures A.1, A2.

Asmin = **As** calculated from ρ **crit** and zone height

Smax = likely maximum spacing of cracks (Code section A.3)

Wmax = estimated maximum crack width (Code section A.3)

Service = service (unfactored) moment and force corresponding to the relevant face/direction

reinf at .. = reinforcement at the opposite face in the same direction.

Referring to Appendix B:

Acr = distance from pt. considered to nearest longitudinal bar

Cover = **cmin** = cover to tension steel

H = overall depth of the member

x = depth of the neutral axis

ε**1** = strain at the level considered

ε**2** = strain due to stiffening effect

ε**m** = average strain = ε**1-** ε**2**

Fs = service stress in steel. Refer to Code section 3.2.2(c) and Table 3.1.

Fc = service stress in concrete

W = design surface crack width

6.7.2.2 EC2 - Detailed results

The program displays detailed calculations for a selected element. The results are displayed separately for each face (+x3/-x3) and each direction (X/Y). For example:

```
ELEMENT 4936     crack at +Z  face in  X direction

Comb 2    (Critical for cracks)

Mx=      7.13 Mxy=      -2.05  Mx*=       9.18  Fx=       0.00  Fxy=       0.00  Fx*=       0.00
      As Required =953              Provided:  8@50             As=1005
           Rhomin =0.0283        Zone height=  75           Asmin=2122       > As ****
             Smax =225.3                                    Wmax=0.04        < 0.20

    Service    Mx*=       9.18  Fx*=       0.00             reinf. at other side 8@50
Srmax= 233.0     Cover= 30.0        H= 350.0                     x=77.4
    εsm-εcm=         0.000933                                  Fs=    311.09   Fc=       6.61
                                                               W=0.22       > 0.20        ****
```

where:

Mx, Fx, etc. = factored moment and force corresponding to the relevant face/direction

As req'd = reinforcement area required to satisfy moment and force requirements. Refer to Reinforcement - method of calculation for more details

Provided = diameter and spacing (φ8@50 cm in above example) of reinforcement provided and corresponding area

Rhomin = minimum ratio of steel to the gross area of the concrete section. Refer to Ciria C660, Eq.(3.12).

Zone ht. = Height of 'service zone'. Refer to Ciria C660, Table 3.1.

Asmin = **As** calculated from ρ,**min** and zone height

Smax = likely maximum spacing of cracks. Refer to Ciria C660, Section 3.4

Wmax = estimated maximum crack width. Refer to Ciria C660, Section 3.5

Service = service (unfactored) moment and force corresponding to the relevant face/direction

reinf at .. = reinforcement at the opposite face in the same direction.

Referring to EC2 - Part 1-1:

Srmax = Maximum crack spacing; EC2:1-1, Eq. (7-11)

Cover = cover to tension steel

H = overall depth of the member

x = depth of the neutral axis

ε**1** = strain at the level considered

ε**2** = strain due to stiffening effect

ε**sm-**ε**cm** = strain in reinforcement - strain in concrete; EC2:1-1, Eq. (7-9)

Fs = service stress in steel = σ_s.

Fc = service stress in concrete

W = design surface crack width; Eq.(7.8)

6.7.3 General

This program module is based on either of the following Codes:

- BS8007:1987 - "Design of Concrete Structures for Retaining Aqueous Liquids".
- EC2, Part 1-1 and CIRIA C660.

The calculations enable the user to design reinforcement in concrete finite elements so that crack widths resulting from applied loading (bending and axial forces) and temperature and moisture effects are limited to acceptable values.

The module can either:

- calculate the reinforcement required to limit the crack width to a user defined value
- calculate the crack width resulting from any arrangement of reinforcement

The calculations are based on service stresses. The module calculates the service stresses from the load combinations with all factors = 1.0 (or -1.0, for negative factors). Therefore, the *STRAP* load cases must be defined with service loads.

The calculations are based on the following two code sections:

- Cracks due to temperature and shrinkage:
  - BS8007: Appendix A - "Calculation of minimum reinforcement, crack spacing and crack widths in relation to temperature and moisture effects"
  - EC2: CIRIA C660 - "Early age crack control in concrete", Section 3.
- cracking due to loads:
  - BS8007: Appendix B - "Calculation of crack widths in mature concrete"
  - EC2, Part 1-1: Section 7.3.

Crack widths are always calculated according to both temperature/shrinkage and loads and the maximum value is displayed or used to calculate reinforcement.

The calculation is carried out as follows:

- the program calculates the reinforcement required to satisfy moment/force requirements (using factored load combinations).
- the program then calculates minimum reinforcement requirements according to temperature/shrinkage (w_{max} < allowable) and revises the reinforcement, if necessary.
- the program then calculates the crack width based on the service loads. The program increases the reinforcement if the width is greater than the limiting crack width parameter specified by the user (if the option is selected by the user).

6.7.4 How to use this module

- Define default parameters for minimum reinforcement and spacing for the entire model. These parameters are displayed in the dialog box when the Display crack results 617 or Display crack detailed results 620 options are selected in the Slabs 617 pulldown menu.

- Define different minimum reinforcement parameters for selected elements, if necessary. Select the Reinf. parameters by element 580 option in the Options 571 pulldown menu.

- Calculate and display results for the model by selecting Display crack results 617 in the Slabs 617 pulldown menu:
  - Revise parameters displayed in the dialog box, if necessary (e.g. concrete type, cover). All parameters are applied to all elements in the model.
  - Select the calculation method:

    ◉ **Reinf. required for moments/forces**
    Calculate and detail the reinforcement required for moments and forces only. This option ignores the crack width requirements of the Code.

    ◉ **Reinf. required for limiting crack width**
    Calculate and detail the reinforcement required to limit the crack width according to the requirements of the Code. The reinforcement listed in the results will not be less than the reinforcement required for moments and forces.

    ◉ **User defined reinforcement**
    The program will calculate crack widths resulting from user defined reinforcement (minimum diameter and spacing).

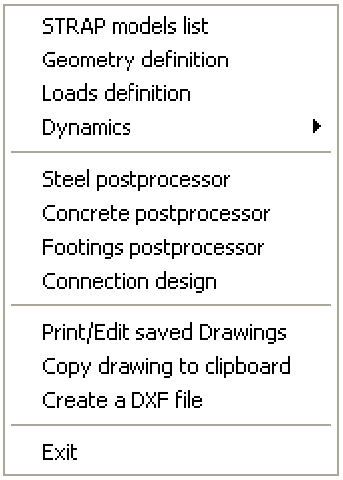
- Display detailed calculation results for selected elements by selecting the Display crack detailed results 620 option in the Slabs 617 pulldown menu. The parameters and the calculation method may be revised as in the previous step.

- Display results graphically:

- click the 

 icon in the side menu
- select **crack width** in the **Display type** list box
- select the face and the direction.
- the parameters and the calculation method may be revised as in the previous step.

6.8 File options



STRAP models list

Return to the *STRAP* main menu (list of models).

Geometry definition

Return to the geometry input for the current model.

Loads definition

Return to the load definition module for the current model.

Dynamics

Continue to the dynamic analysis module

Steel postprocessor

Go to the *STRAP* structural steel design module.

Concrete postprocessor

Go to the *STRAP* reinforced concrete module.

Footing postprocessor

Refer to Footings 627

Connection design

Go to the *STRAP* structural steel connection design postprocessor. Note that the steel memebers must be 'computed' first in the Steel postprocessor.

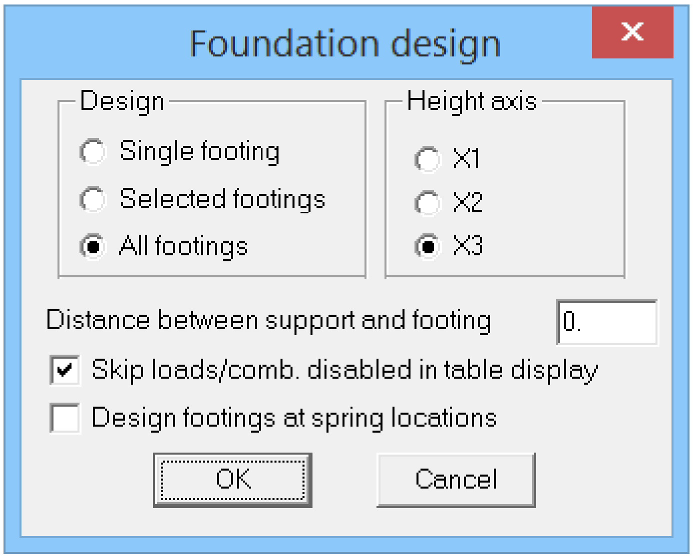
Exit

Quit *STRAP*

6.8.1 Footings

Go to the *STRAP* footing design module and design spread footings at the location of *STRAP* restraints and/or springs.

Refer to How to design footings 628.



Design

Single footing
Select a node with a restraint or spring using the standard node selection option. The footing postprocessor allows the user to define design parameters for the footing and will calculate dimensions and reinforcement accordingly.

Selected footings
Select several nodes with restraints or springs using the standard node selection option. Note that the footing postprocessor designs the selected footings as a group; parameters may be defined for the entire group but not for individual footings.

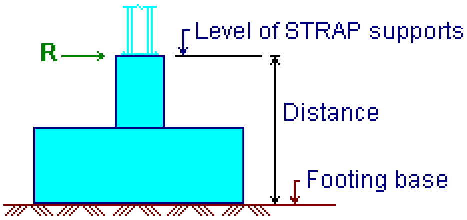
All footings
Design the footings for all nodes with restraints (or springs). Note that the footing postprocessor designs the selected footings as a group; parameters may be defined for the entire group but not for individual footings.

Height axis

Specify the height axis of the model.

The program assumes that the reaction parallel to this axis is the vertical load on the footing and will use the dimensions of the beam connected to the node lying parallel to this axis (±10°) as the column dimensions for the footing design.

Distance



The program multiplies the reactions (R) by the distance and adds the resulting moments to the loads applied to the footing.

Skip disabled loads/combinations

Load combinations may be disabled631 in the Display/print tabular results option. ☑ check this option to ignore the reactions from these combinations when transferring the reactions to the footing design. The combinations that you select should all contain either the Service load reactions or the Factored load reactions, but not both.

Spring locations

☐ Design footings only at nodes with *STRAP* restraints
☑ Design footings only at nodes with either restraints or springs

6.8.2 How to design footings

The *STRAP* footing module designs rectangular spread footings at all nodes assigned with restraints (and springs, if specified by the user).

The postprocessor automatically retrieves from *STRAP*:
- the reactions for all *STRAP* load combinations at these nodes (force and moments)
- the column dimensions (if possible) from the section of the member attached to the node.

The postprocessor works in two modes:

◉ **All/several footings**
The program automatically designs the footings for the group of supports selected. Design parameters may be defined for the entire model. Results for the entire model are displayed in tabular form. Results for footings previously designed with the Single footing option will not be overwritten.

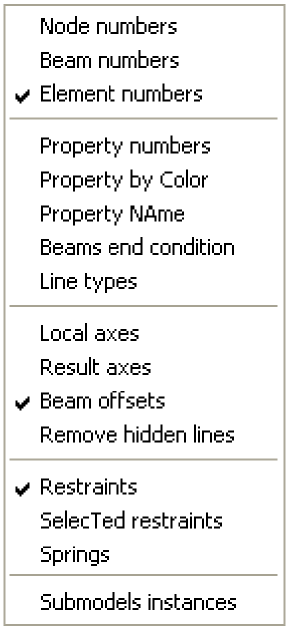
◉ **Single footing**
The program automatically designs the footing for the support selected. Design parameters may be revised for the individual footing. Load cases may be added, revised or deleted. Results for the individual footing are displayed in graphic form. Results for footings previously designed with the **All/several footings** option are overwritten.

Current results for any footing may be viewed by selecting **Single footing** and then selecting the associated node.

Note:
- footings are assumed to be oriented according to the local x2,x3 axis of the column
- the program always transfers the column dimensions as a rectangle that bounds the actual section dimensions. The loads are assumed to act in the center of this rectangle.
- for columns defined by properties (A,I), the program transfers zero dimensions and the postprocessor begins the design with a default dimension that may be revised by the user.
- for footings located at grid line junctions; the program transfers the grid lines as the footing name, e.g.

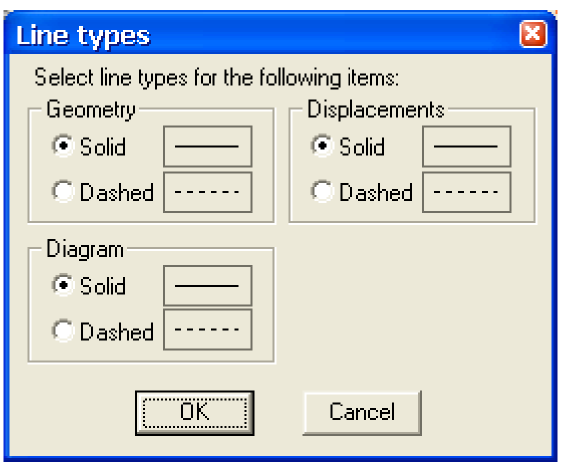
B7, G11. If there are no grid lines at a location, the *STRAP* node number becomes the footing name (may be revised in the footing design program).

6.9 Display options



Line types

Geometry, results diagrams and displacement diagrams may be displayed with either solid or dashed lines:



Result axes

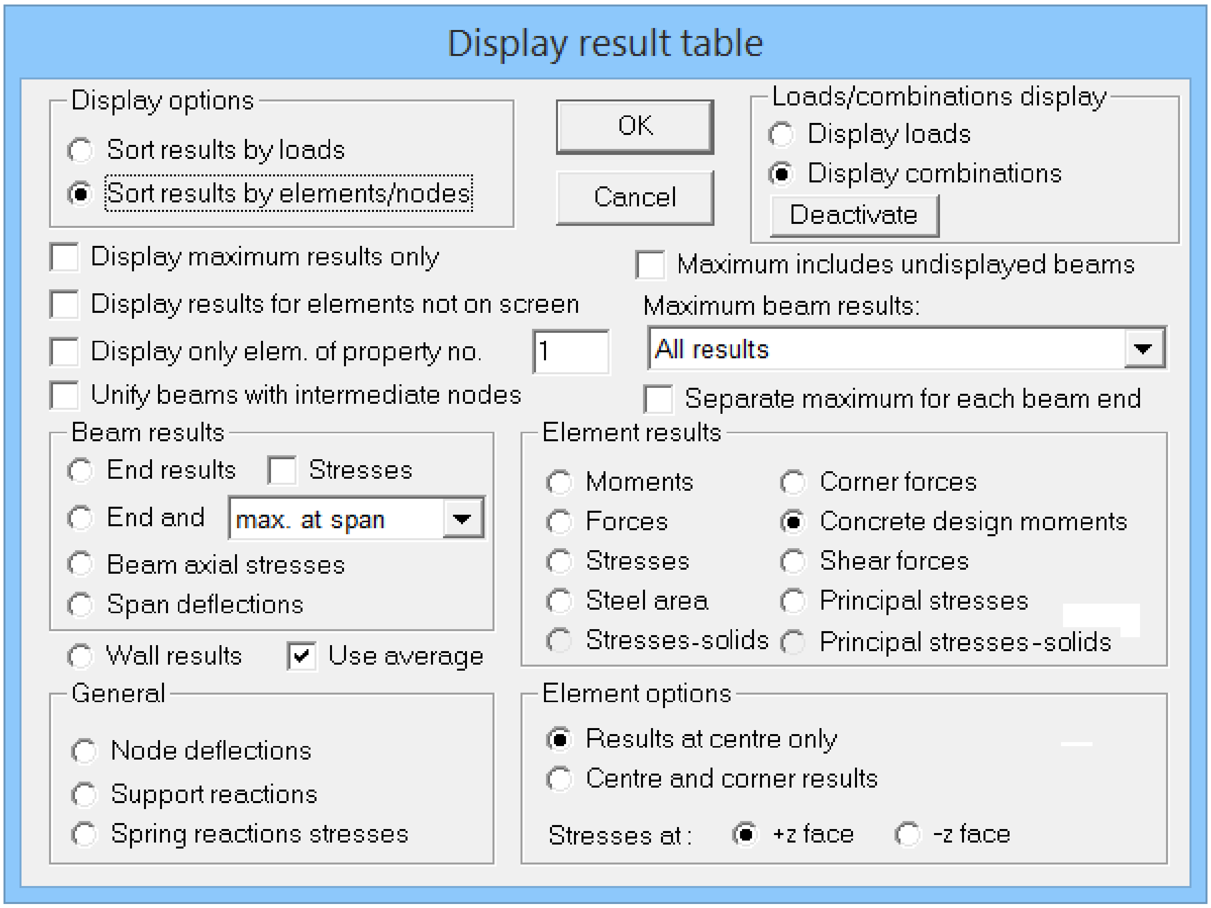
Display the X,Y result axes directions superimposed on the graphic display at the element centers.

6.10 Tabular results

The main features of the tabular results module are:

- results sorted by elements/nodes or load cases
- search for maximum/minimum results per element or node over all load cases or combinations.
- selected output.

The program displays the following menu;



For "Print":

Sort

Sort results by load

The program displays the results for load cases/combinations; Select a combination (case) from the list displayed; the program will display the specified result type for this combination.

Sort results by element/node

In this option, the results for all load combinations (cases) for a specific element/node are displayed together in one table.

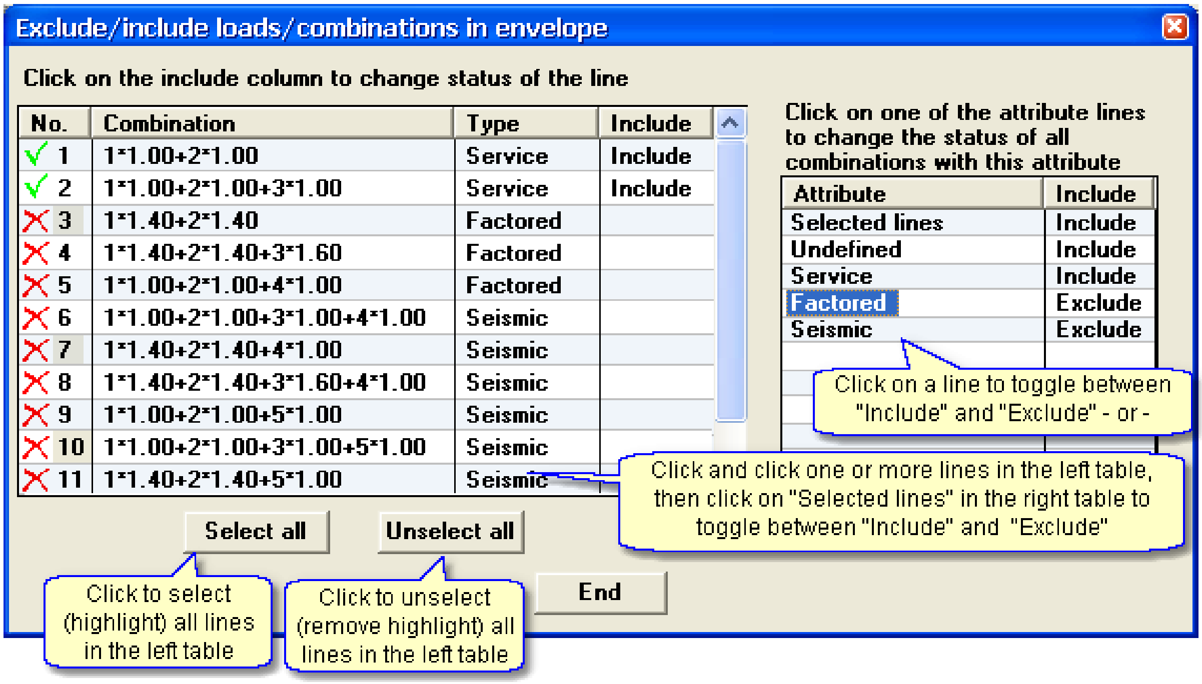
Loads / combinations - deactivate

Display the tabular results either for load cases or load combinations (if defined).

If you want the program to ignore one of the load cases/combinations, you may temporarily deactivate it using this option. The load case/combination is not deleted, and it may be reactivated at any time.

Combinations may be deactivated by selecting combination "types"

Example: display deflection results for service load combination loads only. A type named "Service" has been defined and assigned to the service load combinations (all other combinations were assigned with other Types):



- click on the "Factored" and "Seismic" lines in the right table to "Exclude" them; the left table will appear as shown.
- Alternatively, click Unselect all in the left table, click and highlight rows 1 & 2 in the left table, then click on "Selected lines" in the right table to "Include" them.

Display maximum result only

Display ***only*** the maximum results for each node/beam/element calculated over all of the load combinations (cases). The program displays the overall maximum/minimum results for all the nodes/beams/elements at the end of the table.

Note that the results are displayed ***only*** for the nodes/beams/elements currently displayed on the screen. If you want to view the overall maximum results for the entire model, select the following option.

Maximum includes undisplayed beams

For maximum results (at end of tables, or **"Display maximum results only**"):

☑ The program searches the beams/elements in the ***entire*** model for the maximum results.

☐ The program searches only the beams/elements ***displayed*** on the screen for the maximum results.

Display results for elements not on screen

The program by default displays the results only for nodes/beams/elements currently displayed on the screen. To view results for all nodes/beams/elements, or to display maximum results for the entire model, turn on this checkbox.

Maximum beam results

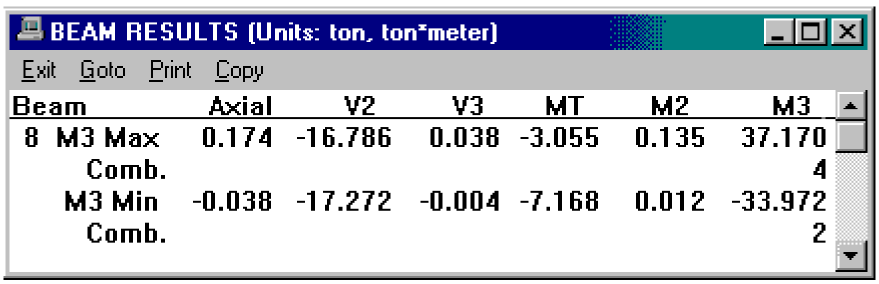
For "**Sort results by elements/nodes**":

By default the program searches for the maximum results separately for each result type and the maximum results for the different types may be from different combinations. Use this option to display the maximum result of a specified type and the corresponding results ***for the same combination*** of all other types.

Select the specified result type:

All results
Max. axial + corresponding results
Max. V2 + corresponding results
Max. V3 + corresponding results
Max. MT + corresponding results
Max. M2 + corresponding results
Max. M3 + corresponding results

For example, select "**Max. M3 + corresponding results**"



All results in the first line are from combination 4, the combination with the maximum M3 result. Similarly, all results in the second line are from combination 2.

Separate maximum

Maximum/minimum results may be displayed separately at each end of each beam. For example, a beam with end nodes 2603 and 2607:

☐ Separate maximum for each beam end

Bm.	Comb	Node	Axial	V2	V3	MT	M2
		Max	8.335	-0.001	0.764	0.000	0.361
		Comb.	2	1	3	3	3
		Min	4.780	-0.001	0.503	0.000	-0.782
		Comb.	1	2	1	2	3

☑ Separate maximum for each beam end

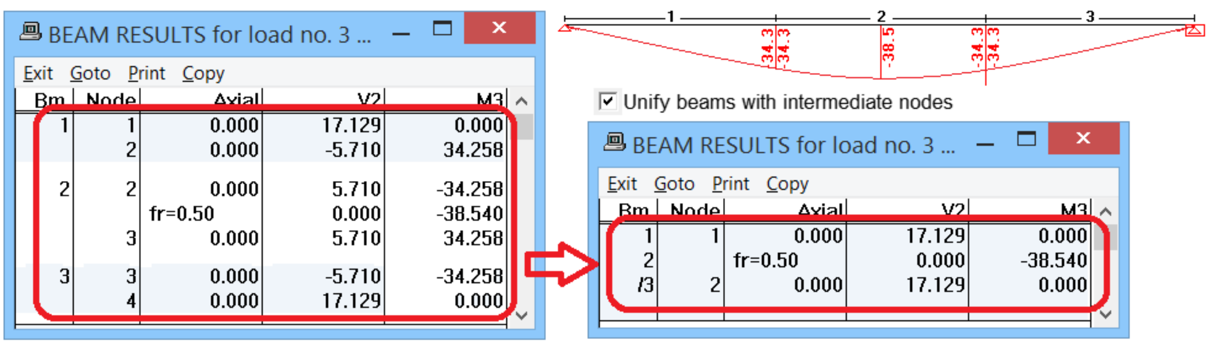
Bm.	Comb	Node	Axial	V2	V3	MT	M2
	Max	2603	8.335	-0.001	0.764	0.000	-0.528
	Comb.		2	1	3	3	1
	Min		5.909	-0.001	0.503	0.000	-0.782
	Comb.		1	2	1	2	3
	Max	2667	6.754	-0.001	0.764	0.000	0.361
	Comb.		2	1	3	3	3
	Min		4.780	-0.001	0.503	0.000	0.225
	Comb.		1	2	1	2	1

Display only elements with property ...

Display only the results for beams in a specified property group. The program also displays the maximum results for the property group.

Unify beams with intermediate nodes

Results for spans with multiple intermediate beams can be condensed to results for a single span. For example, a span with 3 intermediate beams:



Element options

- All finite element results may be displayed in two ways:
  - results at the element centre only
  - results at the element centre at the element corner nodes

  It is important to display element results at the corners in models with stress concentrations at the nodes -loads, supports, openings, etc.
- Element stress results can be viewed either at the top (+z) plane or the bottom plane (-z) of the element. This element is important only in space models.

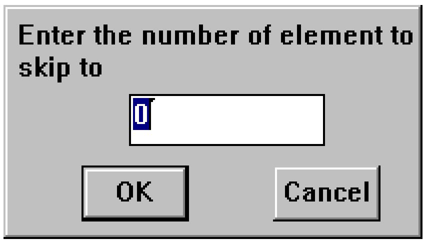
6.10.1 General

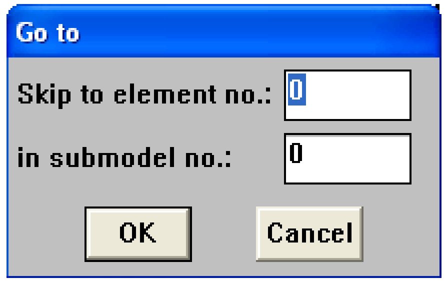
- Element results are displayed according to the Default element result coordinate system 580, unless the result system was revised for specific elements ("Options"/"Element results coord. system" in

toolbar)
- For the interpretation of the results described in this section refer to Sign conventions 649.

The program displays all results table with the following header:

- Click the **Goto** option to skip directly to the display of a specific node/beam/element:





Type in the node/beam/element number and press [Enter] or click the  button. If submodels are displayed in the table, you can skip to the submodel by entering the number of the instance according to their order in the table.
- select **Print** to print the table.
- select **Copy** to copy the current table to the 'clipboard'; the table may then be pasted into Word, Notepad or Excel documents.

The following terminology is used throughout the tabular results module:

Load cases	Load cases defined in the Loads module (prior to solving the model). Some of these cases may in fact be "Combined load cases", but are still referred to as Cases.
Load combinations	Load case combinations defined after solving the model using the Results Combinations option.

6.10.2 Beams

Refer to one of the following result types:

End results 635
Axial stress results 638
Span deflections 639

Note:
- results for selected beams may also be displayed using the right-click 653 option.

6.10.2.1 End results

Display axial force, shear force and moment values at both ends of beams and either the maximum value in the span or at specified intervals - 1/2, 1/4, 1/5, 1/10 or 1/20 of the span. All results are relative to the beam local axes.

Example:

BEAM RESULTS (Units: ton, ton*meter)

Exit Goto Print Copy

Bm.	Comb	Node	Axial	V2	V3	MT	M2	M3
4	1	5	-0.076 C	1.550	0.000	0.000	0.000	0.135
		6	0.076 C	2.450	0.000	0.000	0.000	-1.933
	2	5	+0.089 T	0.765	0.000	0.000	0.000	0.325
		6	-0.089 T	-1.235	0.000	0.000	0.000	-1.263
		MAX	0.089	2.450	0.000	0.000	0.000	1.933
		COMB	2	1	0	0	0	1
		MIN	-0.076	-1.235	0.000	0.000	0.000	0.135
		COMB	1	2	0	0	0	1

where:

Axial = axial force along beam x1 axis
- "C" = Compression, "T" = tension. To add the letters refer to Output format 573.
- for 'trusses': only the result at JA is displayed (the value at JB may be different if an intermediate axial load is applied, but the result is not displayed).

V2 = shear force parallel to the beam x2 axis
V3 = shear force parallel to the beam x3 axis
MT = torsion moment about the beam local x1 axis
M2 = bending moment about the beam local x2 axis
M3 = bending moment about the beam local x3 axis

Refer also to Sign conventions - beams 649.

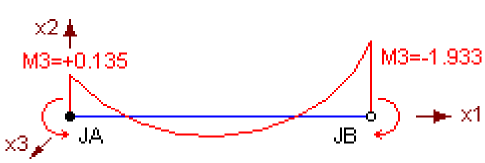
If the **End results and max in span** option is selected, the display is:

```
2      5  +0.089   0.765   0.000   0.000   0.000    0.325
        FR=0.45    0.044                           -2.325
       6  -0.089  -1.235   0.000   0.000   0.000   -1.263
```

where:
FR = fraction of span length (from JA) at which the maximum intermediate moment occurs.

Note:
- The intermediate moments are calculated at 1/20 intervals and at concentrated load locations.
- The sign of the intermediate moment and shear is relative to the sign at JA.
- The intermediate shear value displayed is at the same point as the maximum moment.
- An intermediate value is displayed only if the maximum positive moment or maximum negative moment are not at the end supports.
- All intermediate values displayed are included in the following MAX / MIN value searches.
- When calculating the MAX/MIN results, the program reverses the sign of the moments at JB:

Referring to Sign Conventions - beams 649, and to the example in the following figure, it is apparent that engineering 'negative' (hogging) moments at the two ends of the beam have opposite signs in the table.



To ensure consistency when calculating the 'maximum/minimum' results, the program ***reverses the sign of the moment at JB***. Therefore, the moment results for the beam in the above Figure are displayed as:

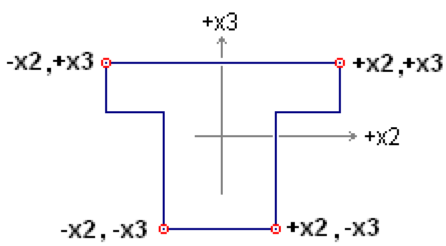
```
BM  COMB  NODE   ...    ....    M3
22     1     43   .....         +10.00
                 FR =0.45        -7.50
             44   .....  .....  -20.00   Note the sign
                                         reversal in the
              MAX .....  .....  +20.00   "MAX" table
             COMB                    1
              MIN .....  .....   -7.50
             COMB                    1
```

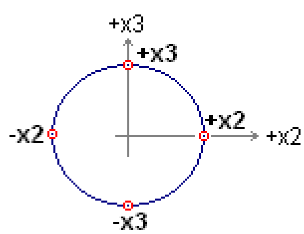
Refer to Results - tables 631 for more information on the display options.

6.10.2.2 Stresses

Display the beam stress values at both ends of beams and optionally either the maximum value in the span or at specified intervals - 1/2, 1/4, 1/5, 1/10 or 1/20 of the span. All results are relative to the beam local axes.

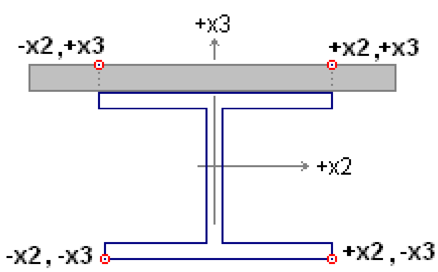
The stresses are calculated at four extreme points in the beam section. For example:





$$\sigma = \frac{P}{A} \pm \frac{M_{x2}(x3)}{I_{x2}} \pm \frac{M_{x3}(x2)}{I_{x3}}$$

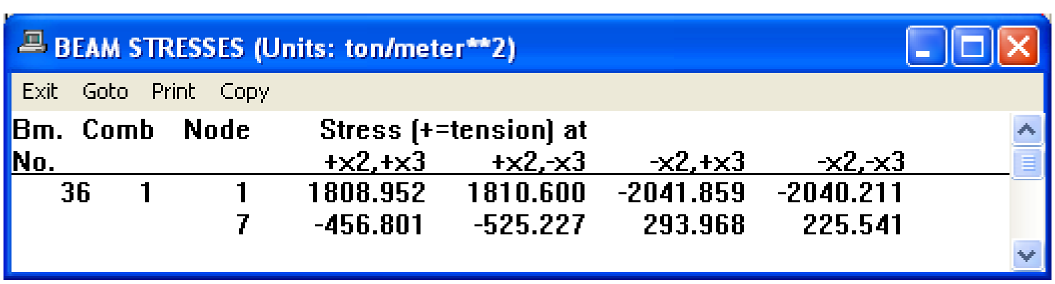
For composite sections:



where the stresses are calculated from the transformed section.

Note that for symmetric sections the topping is always located at the +x flange face.

Example:


BEAM STRESSES (Units: ton/meter**2)

Exit Goto Print Copy

Bm. No.	Comb	Node	Stress (+=tension) at +x2,+x3	+x2,-x3	-x2,+x3	-x2,-x3
36	1	1	1808.952	1810.600	-2041.859	-2040.211
		7	-456.801	-525.227	293.968	225.541

6.10.2.3 Axial stress results

Display the beam stresses due to axial forces **only**, based on the Buckling parameters 574.

Note that the use of this option requires that the moments-of-inertia for all relevant beam elements be entered during the geometry definition.

BS 449 and AISC - ASD Codes:

The max./min. stresses are given for each member along with the calculated allowable stresses.

Note that the maximum stress is the greatest tension stress amongst all the loading combinations. If the beam is always in compression, the value represents the smallest compression stress.

Similarly, the minimum stress is the largest compression stress amongst all of the combinations, or the smallest tension stress when there is no compression.

Example:

BEAM NO.	JA	JB	COMB NO.	AXIAL P	P/A	** ALLOW. STRESS ** X2 DIR.	X3 DIR	% OF ALLOW	
10	2	3	1	-15.32	-10213.	13456.	9855.	104	**
			2	3.17	2113.	15500.	15500.	14	
			3	18.33	12220.	15500.	15500.	78	
	MAX.COMB		3	-15.52	-10213.	13456.	9855.	104	**
	MIN.COMB		1	18.33	12220.	15500.	15500.	78	

where:

AXIAL P	=	The axial force in the beam.
P/A	=	The axial stress corresponding to that force
ALLOW. STRESS	=	The allowable stress about the two buckling axes. For compression, the value is a function of the slenderness of the beam.
% OF ALLOW	=	The ratio of the actual stress to the allowable. Note that if the ratio exceeds 100%, a ** flag is added to the end of the line.

German DIN Code:

The maximum and minimum stresses are given for each member.

Note that the maximum stress is the greatest tension stress amongst all the loading combinations. If the beam is always in compression, the value represents the smallest compression stress.

Similarly, the minimum stress is the largest compression stress amongst all of the combinations, or the smallest tension stress when there is no compression.

Example:

BEAM NO.	JA	JB	COMB NO.	AXIAL P	P/A	** P/A * OMEGA ** X2 DIR	X3 DIR	MAX
10	2	3	1	-15.32	-10213.	15156.	12855.	**
			2	3.17	2113.	2113.	2113.	
			3	18.33	12200.	12200.	12200.	
			4	-9.54	-6360.	9438.	8005.	
	MAX.COMB		3	18.33	12200.	12200.	12200.	12200
	MIN.COMB		1	-15.32	-10213.	15156.	12855.	15146

where:

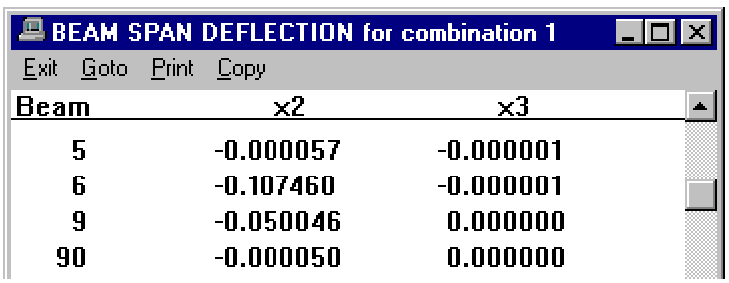
AXIAL P = The axial force in the beam.
P/A = The axial stress corresponding to that force.
P/A * OMEGA = The actual axial stress multiplied by the corresponding omega factor. If the beam is in tension, omega = 1. If this stress exceeds the allowable, a ** flag is displayed at the end of the line.

Refer to Results - tables 631 for more information on the display options.

6.10.2.4 Span deflections

Display the maximum span deflection. The deflection is relative to the final location of the nodes, not the joint coordinates.

For example:

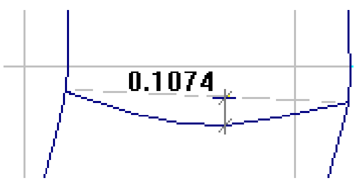


BEAM SPAN DEFLECTION for combination 1

Exit Goto Print Copy

Beam	x2	x3
5	-0.000057	-0.000001
6	-0.107460	-0.000001
9	-0.050046	0.000000
90	-0.000050	0.000000

For beam 6:



Refer to Results - tables 631 for more information on the display options.

6.10.3 Elements

Refer to one of the following result types:

Moments, forces & stresses 639
Principal stresses 640
Concrete design moments 641
Solids - stresses 641
Solids - principal stresses 642
Shear forces 642
Corner forces 643

Note:
- results for selected elements may also be displayed using the right-click 653 option.

6.10.3.1 Moment, force & stress tables

Display :

Stresses:

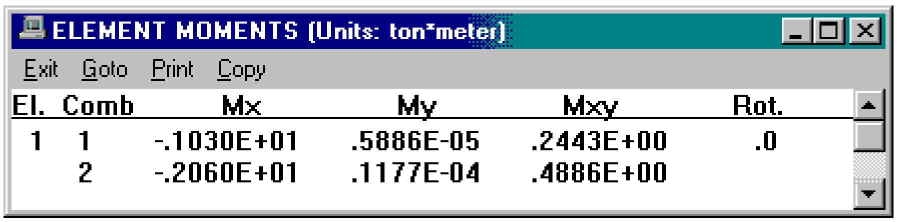
±SX = stress in element result X direction on the ±Z surface.
±SY = stress in element result Y direction on the ±Z surface.

±SXY = shear stress on ±Z surface.

Forces, moments = stress resultants in the result coordinate system:

Note:
- the moments are per unit width, i.e. MX = 50 indicates 50 t m/m (kN m/m, ft kip/ft, etc.)
- MX is the moment ***in the direction of the element result X axis*** and not the moment about the X axis. Therefore, the stress SX corresponds to the moment MX. Similarly, MY is the moment ***in the direction of the element result Y axis*** and ***not*** the moment about the x2 axis.
- a stress distribution with tension on the +z face results in a ***positive*** bending moment; refer to Element sign conventions 650.
- For the equations relating moments and forces to stresses, refer to Element sign conventions 650.

Example (moments sorted by combinations):


ELEMENT MOMENTS (Units: ton*meter)

Exit Goto Print Copy

El.	Comb	Mx	My	Mxy	Rot.
1	1	-.1030E+01	.5886E-05	.2443E+00	.0
	2	-.2060E+01	.1177E-04	.4886E+00	

Maximum values are calculated from all of the result points.

Rot indicates the angle between the local x1 axis of the element and the result X axis.

The result axes may be defined in the Element result coordinate system 577 option. If no specific axes are defined, the program uses the default result axes, as follows:

- **Plane model**

  The X direction is always parallel to the global X1 axis and the Y direction is always parallel to the global X2 axis.

  If the program discovers that the direction of the local x3 axis of an element is opposite to the direction of the global X3 axis, it reverses the sign of the results for that element.

- **Space model**

  - Elements parallel to the X1-X2 global plane : X=X1, Y=X2
  - Elements parallel to the X1-X3 global plane : X=X1, Y=-X3
  - Elements parallel to the X2-X3 global plane : X=X2, Y=X3
  - Elements not parallel to a global plane : X=direction parallel to X1
    Y=perpendicular to X and in the direction of X3

  ****** Warning ******

  In space models the local x3 axis direction is not revised as in plane models. The sign of the results will be inconsistent if the local x3 axes of adjacent elements point in opposite directions and their interpolation over the element surface will be incorrect.

Refer to Results - tables 631 for more information on the display options.

6.10.3.2 Principal stresses

Display the principal maximum and minimum element stresses. The principal stresses are displayed at the element centre and corners.

Refer to:
- Sign conventions - elements 650 for more details on the sign conventions.

- Results - tables 631 for more information on the display options.

6.10.3.3 Concrete design moments

To display the reinforcement design moments for slab design - Mx* and My*. The calculations are based on the Wood & Armer equations 685. Note that the slab reinforcement can also be designed, detailed and draw in the Concrete design module.

The moments are calculated separately for the +Z and the -Z surfaces. Note that +Z is referred to as the "TOP" surface in the tables.

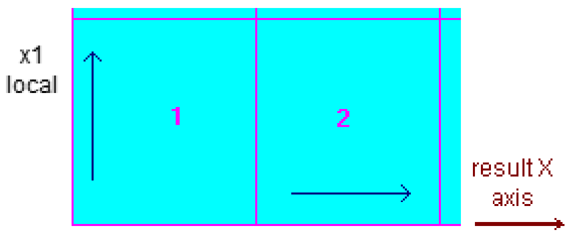
Example:

ELEMENT DESIGN MOMENTS (Wood & Armer) (Units: ton*meter)

Exit Goto Print Copy

		BOTTOM		TOP			
El.	Comb	Mx*	My*	Mx*	My*	Alpha	Rot
1	1	.1274E+01	.2443E+00	.0000E+00	-.5797E-01	90.0	−90.0
	2	.2548E+01	.4886E+00	.0000E+00	-.1159E+00		

The angle required to rotate the element local x1 axis to the element result X axis is denoted by **Rot,** where counterclockwise is positive. For element 1 in the above example:



The reinforcement coordinate system angle 579 **Alpha** is listed adjacent to **Rot.**

The moments at the element corners may also be displayed.

Refer to:
- Sign conventions - elements 650 for more details on the sign conventions.
- Results - tables 631 for more information on the display options.

6.10.3.4 Solids - stresses

Stresses may be displayed at the element centre or the element corners. All stresses are relative to the global coordinate system.

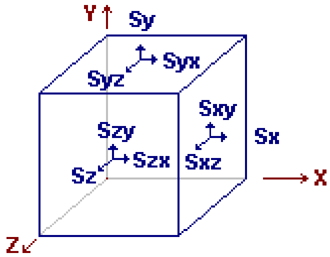
Example:

ELEMENT RESULTS for combination 1 (Units: ton/meter**2)

Exit Goto Print Copy

Elem	Sx	Sy	Sz	Sxy	Syz	Szx
35	231.93	41.24	18.35	120.87	13.19	83.91
38	-87.13	-0.41	-2.37	-37.50	-3.79	23.99
39	-70.56	-10.70	-12.55	-27.39	16.93	20.33

where:



6.10.3.5 Solids - principal stresses

The principal stress table displays two different cases:

- principal stresses; the stresses on the principal planes, where the shearing stresses **Sxy = Syz = Szx = 0**
- maximum shearing stresses

For example:

PRINCIPAL STRESSES for combination 1 (Units: ton/meter**2)

Exit Goto Print Copy

Elem	Sp1	Sp2	Sp3	Tau1	Tau2	Tau3
8	35.55	-13.99	-313.08	149.55	-174.31	24.77
angles=	61.4,130.8, 54.1				VM=326.69	
9	7.53	-5.54	-91.99	43.22	-49.76	6.54
angles=	99.4, 43.8, 47.7				VM=93.67	

where:

Sp1, Sp2, Sp3 = the three principal stresses, determined as follows:

$$S^3 - (S_x+S_y+S_z)S^2 + (S_xS_y+S_yS_z+S_zS_x-S_{yz}^2-S_{xz}^2-S_{xy}^2)S - (S_xS_yS_z+2S_{yz}S_{xz}S_{xy}-S_xS_{yz}^2-S_yS_{xz}^2-S_zS_{xy}^2)=0$$

The three roots of this equation give the values of the three principal stresses **Sp1**, **Sp2** and **Sp3**.

angles = angles to rotate the X1-X2-X3 global axes to the principal axes; the first angle represents the rotation about X1, etc.

VM = Von Mises stress

$$= \sqrt{0.5 * [(Sp1 - Sp2)^2 + (Sp2 - Sp3)^2 + (Sp3 - Sp1)^2]}$$

Tau1, Tau2, Tau3 = maximum shearing stress. These stresses act on the plane bisecting the angle between the largest and smallest principal stresses and is equal to half the difference between these two principal stresses:

Tau1 = 1/2**(Sp2-Sp3)**
Tau2 = 1/2**(Sp3-Sp1)**
Tau3 = 1/2**(Sp1-Sp2)**

(Reference: "Theory of Elasticity" by Timoshenko & Goodier, 3rd Edition, p. 219-226)

6.10.3.6 Shear force results

Display the transverse shear forces Qx, Qy at the element centre.

$$Q_x = \frac{dMx}{dx} + \frac{dMxy}{dy} \qquad Qy = \frac{dMy}{dy} + \frac{dMxy}{dx}$$

- Tabular results:
  The shear values are calculated from the slope of the moment diagram at the element centre only.
- Contour map:
  The program calculates the corner values for Qx, Qy based on an estimated 2nd derivative of the Mx,

My,Mxy results at the centre and uses the averaged estimated values to draw the contours.

It is obvious that Qx, Qy will be less accurate than Mx, My, Mxy for the same elements because of the inaccuracy of the corner results. The accuracy of the shear results are more sensitive to changes in the density of the mesh.

Refer to:
Sign conventions - elements 650 for more details on the sign conventions.
Results - tables 631 for more information on the display options.

6.10.3.7 Corner force results

These are the forces and the moments in the six degrees of freedom at each corner of an element, and represent the 'reactions' at the element 'supports' - i.e. at the connection points to the adjacent elements.

Note that the corner forces are always relative to the global coordinate system.

Refer to:
- Sign conventions - elements 650 for more details on the sign conventions.
- Results - tables 631 for more information on the display options.

6.10.4 Nodes

Refer to one of the following result types:

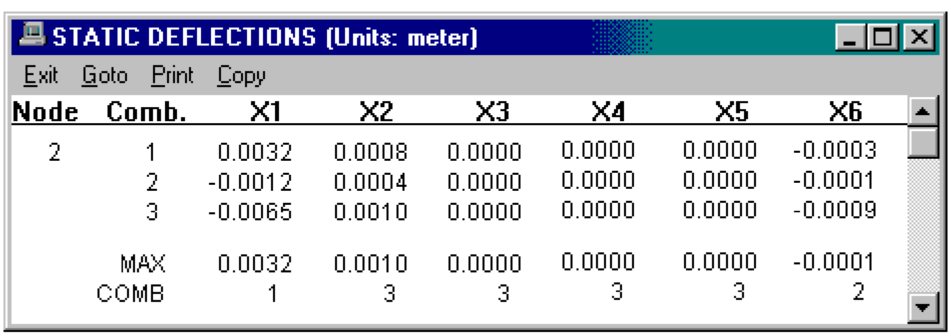
Deflections 643
Spring reaction stresses 644
Reactions 644

Note:
- results for selected nodes may also be displayed using the right-click 653 option.

6.10.4.1 Deflections

Display the global deflections and rotations at the nodes in the model.

Example: for a model with three loading combinations:



where:

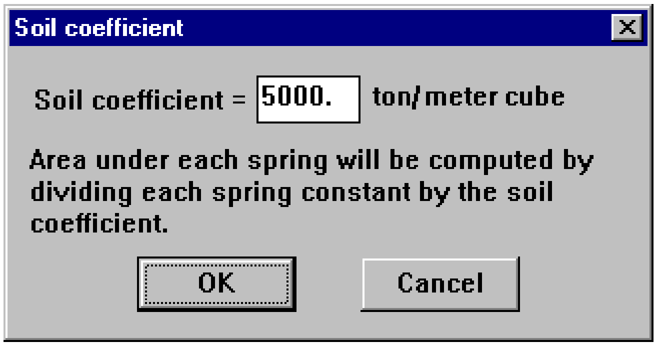
X1, X2, X3	=	translations parallel to the global axes. A positive deflection is in the positive direction of the axis.
X4, X5, X6	=	rotations ***about*** X1, X2, X3, respectively, in ***radians***. For the positive direction of rotation, refer to Sign conventions 649

Refer to Results - tables 631 for more information on the display options.

6.10.4.2 Spring reaction stresses

Calculate the pressure (= **R/A**) under the springs, i.e. the soil pressure for elastic foundations, where:

R = *STRAP* result reaction
A = **K/M**
K = spring constant defined in *STRAP* geometry
M = soil coefficient (subgrade modulus) defined here:



Note:
- For models with rigid links with springs defined at the slave nodes: The program sums all of the reactions from the slave nodes at the master node but uses the tributary area of the master node only.

6.10.4.3 Reactions

The program sums at each node all of the applied loads and element result end forces of the attached elements for all degrees-of-freedom. The sum for degrees-of-freedom at nodes that were not defined as restraints should be equal to zero.

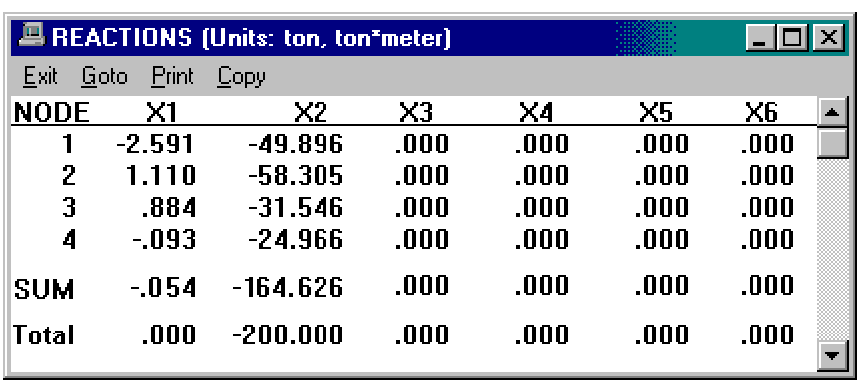
For submodels: the program display the reactions at the connection points to the main model (only when the submodel is the current display).

The reactions are:

X1, X2, X3 = reaction force parallel to the global axes. A positive force is in the positive direction of the axis.
X4, X5, X6 = moments ***about*** X1, X2, X3, respectively. For the positive direction of the moment, refer to Sign conventions 649

The results displayed are in effect the "actions" on the support.

- **Results Sorted by Loads:**
  The program displays the reactions only for nodes in the current display.. For example:


REACTIONS (Units: ton, ton*meter)

Exit Goto Print Copy

NODE	X1	X2	X3	X4	X5	X6
1	-2.591	-49.896	.000	.000	.000	.000
2	1.110	-58.305	.000	.000	.000	.000
3	.884	-31.546	.000	.000	.000	.000
4	-.093	-24.966	.000	.000	.000	.000
SUM	-.054	-164.626	.000	.000	.000	.000
Total	.000	-200.000	.000	.000	.000	.000

where:
SUM = sum of reactions for nodes displayed in the table
Total = The sum of the reactions for all support nodes in the model; the values should equal the sum of the loads applied in that loading case.

Due to the limited numerical precision of personal computers, non-zero values will occasionally appear in the Reactions for unrestrained degree-of-freedoms (i.e. nodes that were not defined as supports). Usually, these values will be negligible in comparison to the internal forces at these degree-of-freedoms (approximately N*10E-5, where N are the internal forces at a DOF). However in certain instances, the numbers could be much greater.

In such cases, the user should check for the following causes:
- Singularity messages during the solution stage.
- Too large a difference between the values of the largest and smallest moment-of-inertia (or area) was defined for the model; The ratio between the largest and smallest values should not exceed 1:10E8.
- In plate bending elements, the element thickness is large relative to the element dimensions (note that only the Reactions are affected - moments and deflections are accurate)

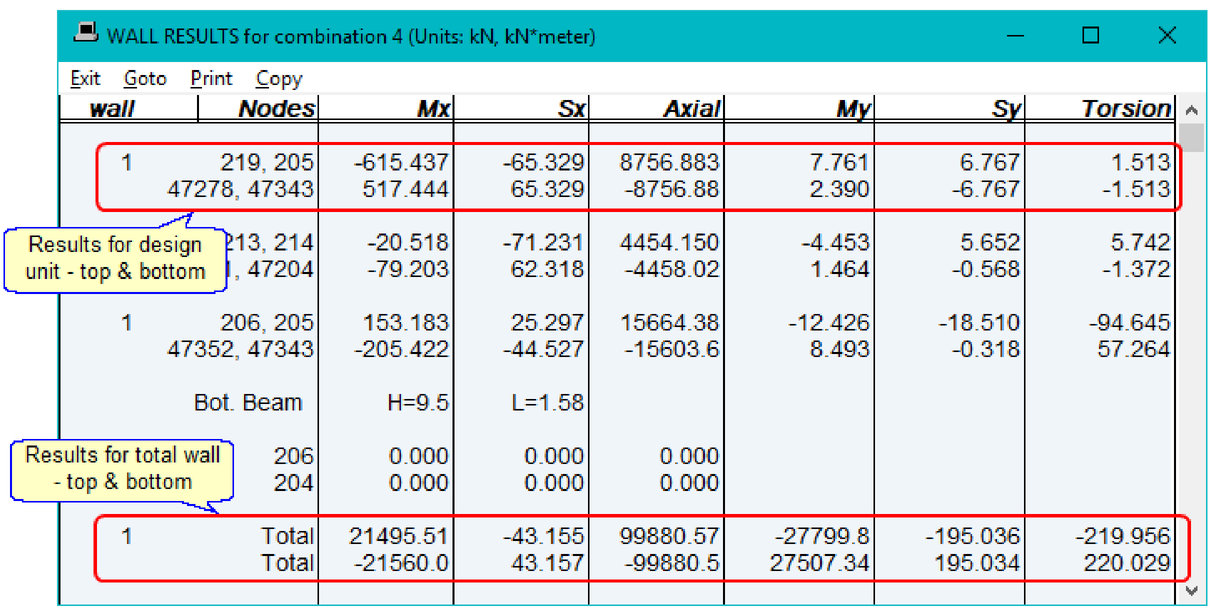
- **Results Sorted by Elements/Nodes**
  The reactions are printed ***only*** at nodes that were defined as restraints and so the inaccuracies described above will not be apparent.

  Note that the max/min reactions are the maximum and minimum ***numerical*** reactions at the supports.

Refer to Results - tables 631 for more information on the display options.

6.10.5 Walls

Wall results are displayed for each segment in a wall element, including beams formed by openings,as well as the total wall results. For example:



wall	Nodes	Mx	Sx	Axial	My	Sy	Torsion
1	219, 205	-615.437	-65.329	8756.883	7.761	6.767	1.513
	47278, 47343	517.444	65.329	-8756.88	2.390	-6.767	-1.513
	213, 214	-20.518	-71.231	4454.150	-4.453	5.652	5.742
	[illegible], 47204	-79.203	62.318	-4458.02	1.464	-0.568	-1.372
1	206, 205	153.183	25.297	15664.38	-12.426	-18.510	-94.645
	47352, 47343	-205.422	-44.527	-15603.6	8.493	-0.318	57.264
	Bot. Beam	H=9.5	L=1.58				
	206	0.000	0.000	0.000			
	204	0.000	0.000	0.000			
1	Total	21495.51	-43.155	99880.57	-27799.8	-195.036	-219.956
	Total	-21560.0	43.157	-99880.5	27507.34	195.034	220.029

The results are displayed in the form of beam results and values are shown for each result type at the top and bottom of the wall (along the height axis).
- Design units:
  The **Mx** and **Sx** values are design unit major axis results; **My** and **Sy** are the minor axis results and usually will be relatively small. Refer to Wall elements - sign conventions 652.

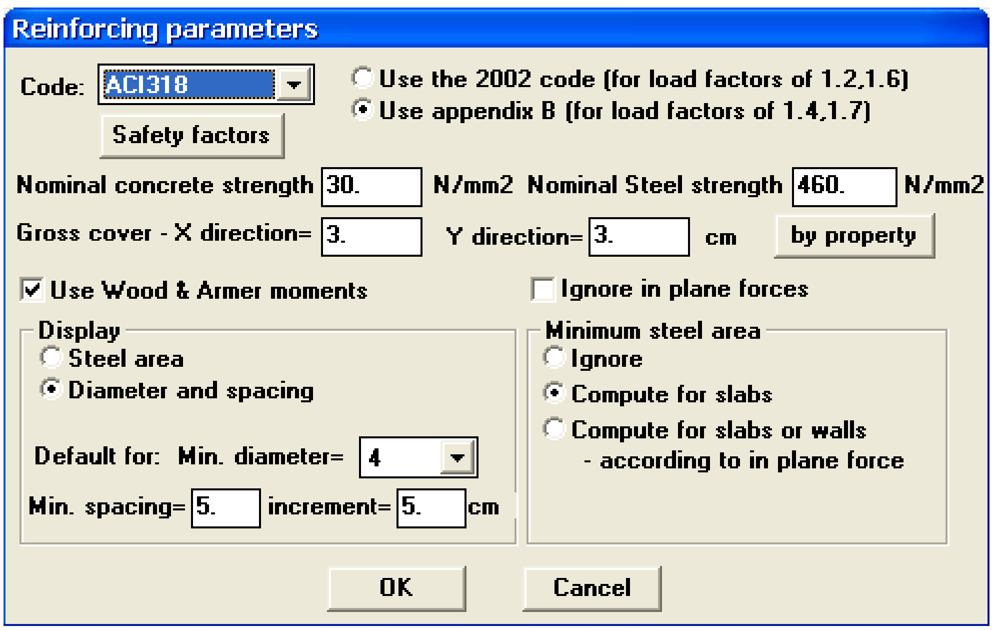
- Total wall:
  the X axis is the horizontal axis in the wall section definition menu.
- Beams formed by openings:
  Results are identical to regular beams, where **Mx** = **M3, My** = **M2, Sx** = **V2** and **Sy** = **V3.**

Note:
- the result X,Y axes (for design units and the total wall) may be displayed on the model using the Display - local axes option.

6.10.6 Element reinforcement area

Calculate the area of reinforcement required for bending moment and axial force - top and bottom - in both directions.

Specify the default parameters for the entire model (different parameters for selected elements may be defined by selecting the "Options - Reinf. parameters by element" option in the toolbar).



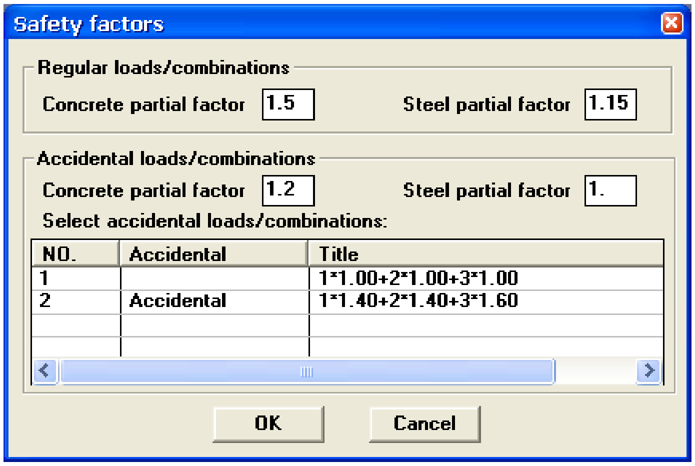
Code

Select the design code; if the Code you require does not appear in the list, please contact your ***STRAP*** dealer.

Note:
- for ACI318 shear and compression calculations, select the ϕ coefficient - those specified in chapter 9 of the Code, or those specified in Appendix B. ***Note that you must define the load combinations with the load factors corresponding to the ϕ coefficient that you select***.

Safety factors

Specify the following data for Eurocode 2:
- identify the 'Accidental' loads/combinations
- specify γ_c = the concrete partial factor and γ_s = steel partial factor for regular and accidental load combinations.



To specify a combination as **Accidental**, place the on the relevant line and click the mouse.

Concrete

Specify the nominal concrete strength:

- BS8110- fcu
- Eurocode 2 - fck
- ACI318 - f'c
- CSA A23.3 - f'c

Steel

Specify the nominal steel strength:

- BS8110- fy
- Eurocode 2 - fyk
- ACI318 - fy
- CSA A23.3 - fy

Cover

Specify the **gross** cover (to centre-of-gravity of the reinforcement) according to the displayed units.

- different cover values may be specified in the X and Y directions (element result coordinate system 577). These are the default values.
- define values other than the default for selected property groups; click by property and enter the X and/or Y cover values in the table.

Wood & Armer

☑ Use the Wood & Armer design moments to calculate the reinforcement area
☐ Use the ***STRAP*** M_x and M_y moments to calculate the reinforcement area (ignore the influence of M_{xy})

In-plane forces

Space frame models only: Calculate the reinforcement area required for the Mx. My, Mxy moments only and ignore the Fx, Fy, Fxy forces

Display / Min diameter & spacing

Select one of the following:

◉ **Steel area**
The reinforcement results will be displayed as area/unit width, e.g. mm²/m, in²/ft, etc.
The program does not check minimum and maximum reinforcement percentages.

◉ **Diameter and spacing**
The reinforcement results will be displayed as diameter and spacing, e.g. ϕ10@200, #6@10, etc.
The program requires the following parameters in order to calculate the results:

- minimum diameter
- minimum spacing
- spacing increment

The program initially tries minimum diameter with minimum spacing. If this combination is insufficient, it tries larger diameters with the same spacing. When an adequate diameter is found, the program searches for the maximum spacing (a multiple of the increment) with this diameter that provides sufficient area.

The program checks minimum reinforcement percentage but does not check maximum reinforcement percentage

Note:

- a uniform steel area is calculated for each element according to the maximum area required in the element, i.e. only one value or one colour is displayed for each element.

Minimum steel area

Select one of the following options:

◉ **Ignore**
Ignore Code minimum reinforcement requirements and provide only area required

◉ **Compute for slabs**
Compute and provide the minimum area according to the Code sections for ***slabs***

◉ **Compute for walls or slabs - according to in-plane forces**

- if the wall is ***entirely*** in compression:
  compute and provide the minimum area according to the Code sections for ***walls***.
- otherwise:
  compute and provide the minimum area according to the Code sections for ***slabs***.

Note:

- the program calculates compression reinforcement if the moment exceeds the Code limit for tension reinforcement
- ***the program does not check maximum reinforcement percentages***

For more details, refer to:

- Reinforcement - method of calculation.
- Wood and Armer Equations 685

Example:

BENDING STEEL AREA for combination 1 (Units: mm**2 per meter)

Exit Goto Print Copy

Concrete: 30 Steel: 350 Cover: 3.0 Code: EURO 2 (Wood&Armer moments)

	BOTTOM		TOP			
Ele.	Asx	Asy	Asx	Asy	Alpha	Rot.
50	364*	0	364*	396	90.0	0.0
51	528	364*	0	0	90.0	0.0
52	448	364*	0	0	90.0	0.0
53	0	0	508	410	90.0	0.0

where

- minimum reinforcement area is denoted by a *

Rot = element result coordinate system 577 angle
Alpha = reinforcement skew angle 579

6.10.7 Tables - sign conventions

The tabular results use a mathematical sign convention as opposed to the standard engineering sign conventions; the sign of the results may often be opposite to what is expected and so the user must understand the conventions in order to correctly interpret the results.

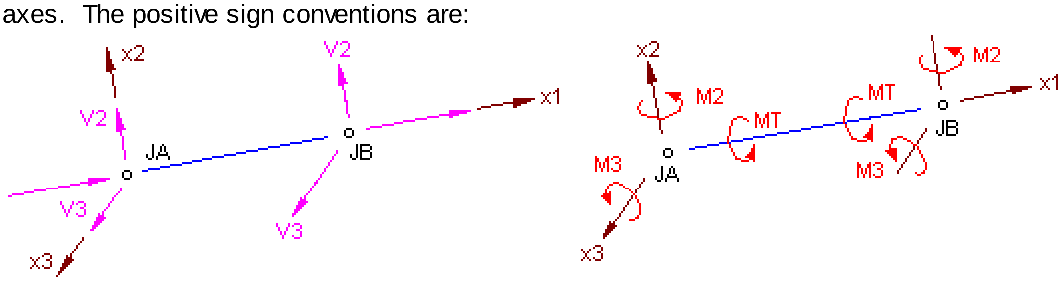
Beams 649
Elements 650
Walls 652
Reactions 644

Note:

- Element results are displayed according to the Default element result coordinate system 580, unless the result system was revised for specific elements ("Options"/"Element results coord. system" in Menu bar)

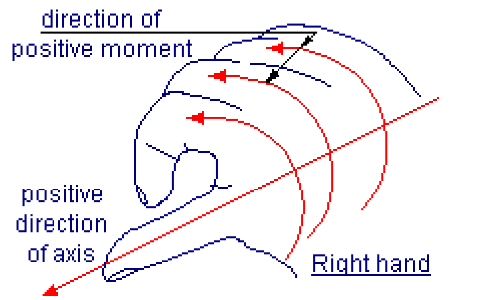
6.10.7.1 Beams

Member results are listed at nodes JA, JB of each element. Results are relative to the local coordinate axes. The positive sign conventions are:

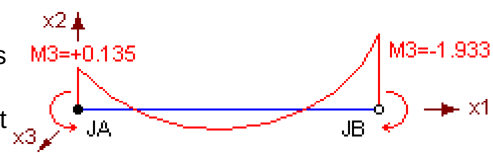


- **Moments:** (M2, M3, MT)

The direction of a positive moment is determined by a right-hand rule. The thumb points in the positive direction of the local axis about which the moment acts, and the other fingers all curl in the direction of the positive moment.



In the following example ***engineering*** negative moments act at ***both*** ends of the beam, however the sign of the moment at JA will be positive in the tables.



- **Shear:** (V2, V3)
  A positive shear force acts in the positive direction of the local axis.
- **Axial:**
  The sign of the axial force is positive in the +x1 direction of the beam. A positive axial force value at JA (along with a negative value at JB) always indicates that the beam is in compression. For trusses, only the result at JA is displayed, i.e. a positive value indicates compression! (the value at JB may be different if an intermediate axial load is applied, but the result will not be displayed).

6.10.7.2 Elements

The following is an explanation of the finite element results:

Note: The results are relative to the element result coordinate system 577.

STRESSES:

+SX = stress in element result X direction on the +Z surface.
+SY = stress in element result Y direction on the +Z surface.
+SXY = shear stress on +Z surface.

-SX stress in element result X direction on the -Z surface.
-SY stress in element result Y direction on the -Z surface.
-SXY shear stress on -Z surface.

FORCES:

The forces are per unit width. i.e, FX = 50.2 means 50.2 ton/m (kN/m, kip/ft, etc.).

The element forces are computed directly from the element stresses:

$$\begin{bmatrix} FX \\ FY \\ FXY \end{bmatrix} = \frac{T}{2} x \left(\begin{bmatrix} SX \\ SY \\ SXY \end{bmatrix}_{+Z} + \begin{bmatrix} SX \\ SY \\ SXY \end{bmatrix}_{-Z} \right)$$

PRINCIPAL STRESSES:

The principal stress at each face derived from the Mohr's circle equations:

$$MAX = \frac{SX+SY}{2} + \sqrt{\left(\frac{SX-SY}{2}\right)^2 + SXY^2}$$

$$MIN = \frac{SX+SY}{2} - \sqrt{\left(\frac{SX-SY}{2}\right)^2 + SXY^2}$$

where:

- MAX and MIN are the algebraic maximum and minimum, not the absolute.
- S. MAX = $S.MAX = \frac{MAX - MIN}{2}$ = maximum shear
- ANGLE = the amount (in degrees) the element X axis must rotate counterclockwise about the Z axis to coincide with the principal stress axis.

$$= \frac{\pi}{2} \tan^{-1}\left[\frac{2 * SXY}{SX - SY}\right]$$

Note:

- when the X axis coincides with the maximum stress axis, Y coincides with the minimum stress axis.
- SHEAR occurs on a plane offset 45° from the principal axis.
- Shear is zero in the principal stress directions.

The positive sign conventions for all stresses and forces are shown in the figures below.

MOMENTS:

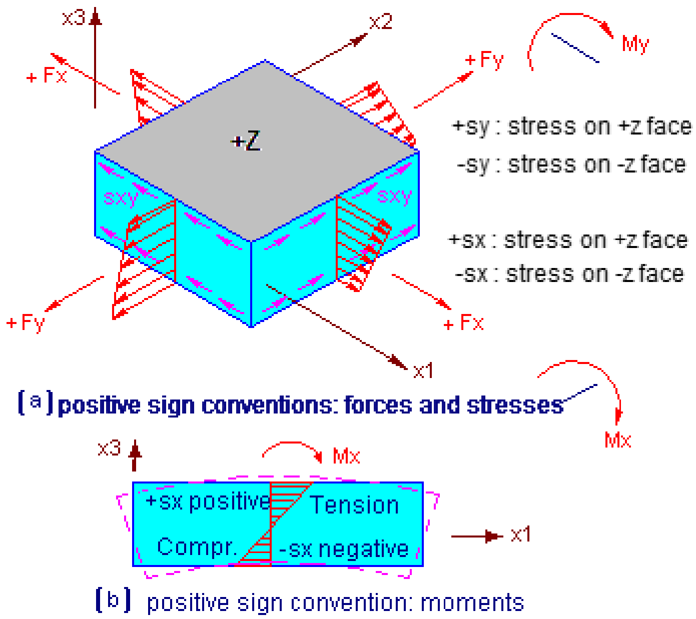
Moments relative to the result coordinate system at the centre of the element. The moments are computed directly from the stresses by:

$$\begin{bmatrix} MX \\ MY \\ MXY \end{bmatrix} = \frac{T^2}{12} x \left(\begin{bmatrix} SX \\ SY \\ SXY \end{bmatrix}_{+Z} + \begin{bmatrix} SX \\ SY \\ SXY \end{bmatrix}_{-Z} \right)$$

Note:

- The sign convention for moments is illustrated in Figure (b) below: referring to the equation above for calculating MX, it is obvious that a stress distribution with tension on the +z face results in a **positive** bending moment.
- MX is the moment ***in the direction of the element result X axis*** and ***not*** the moment about the X axis (see Figure below). Therefore, the stress SX corresponds to the moment MX. Similarly, MY is the moment ***in the direction of the element result Y axis*** and ***not*** the moment about the Y axis.
- The moments are per unit width, i.e. MX = 50 indicates 50 t m/m (kN m/m, ft kip/ft, etc.)

The following are the positive element results sign conventions:



Referring to Figure (a) above, approximately equal and opposite forces act on opposite faces of the element. The sign of the results displayed are for the forces and stresses on the face in the positive direction of the result axis. Therefore:

- if FX is positive, the element is in tension in the direction of X.
- if +SX is positive, the top face is in tension.

Note:

- If results are displayed graphically, the program may modify the sign of the results in order to ensure consistency, etc.

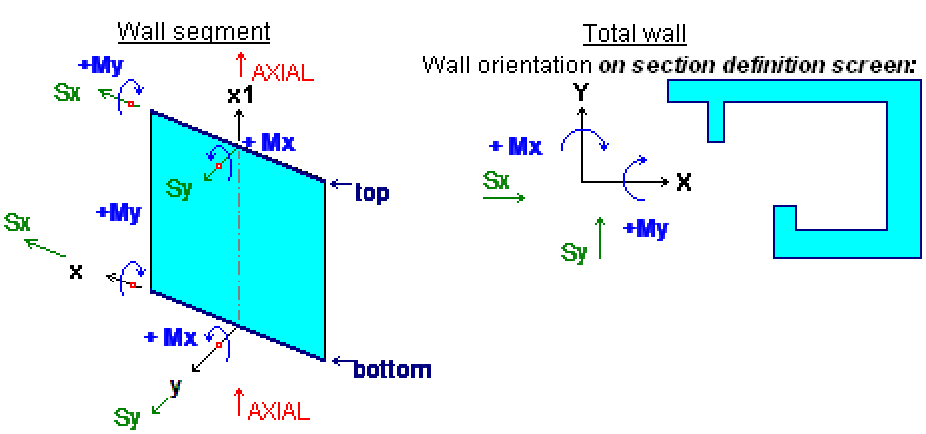
6.10.7.3 Walls

Member results are listed at the top and bottom of each element design unit (identified by the corner nodes) or for the total wall. Results are relative to the wall local coordinate axes.

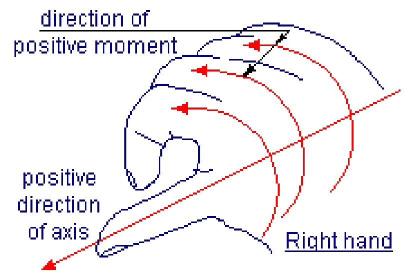
Note:

- the result X,Y axes (for design units and the total wall) may be displayed on the model using the Display - local axes option.

The positive sign conventions are:

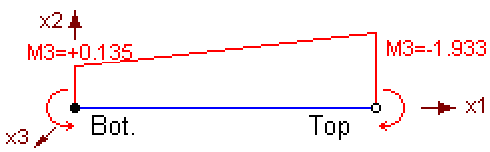


- **Moments:**



The direction of a positive moment is determined by a right-hand rule. The thumb points in the positive direction of the local axis about which the moment acts, and the other fingers all curl in the direction of the positive moment.

In the following example the wall is in single curvature, however the sign of the moment at the bottom will be positive in the tables.



- **Shear:**

A positive shear force acts in the positive direction of the local axis.

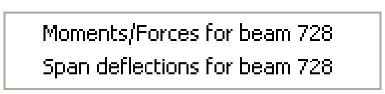
- **Axial:**

  The sign of the axial force is positive in the +x1 direction of the wall. A positive axial force value at the bottom (along with a negative value at the top) always indicates that the wall is in compression.

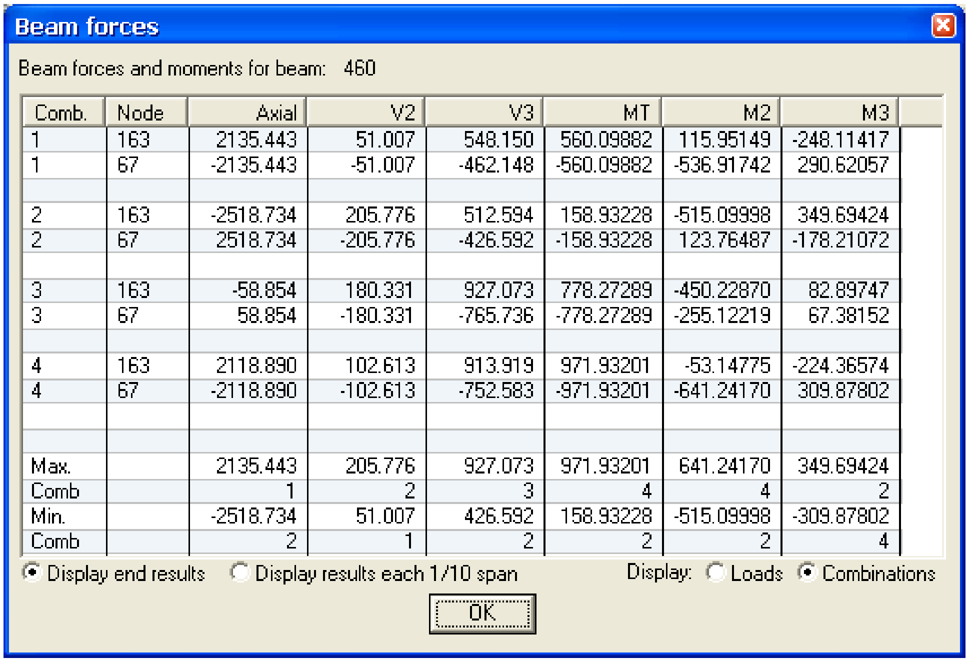
6.10.8 Right click

Tabular results for selected beams, elements and node may also be displayed using the 'right-click' 653 option. Results may be displayed either for load combinations or load cases.

Beams:

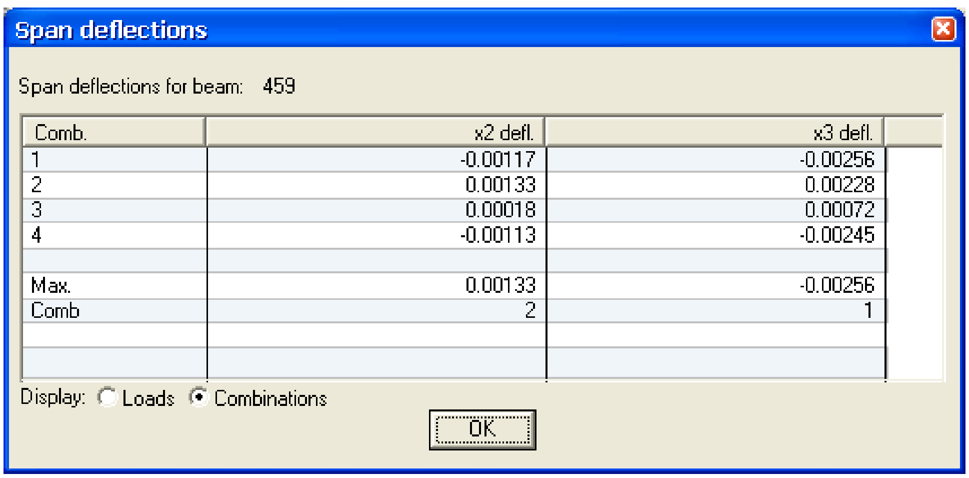


- Moments/forces



Comb.	Node	Axial	V2	V3	MT	M2	M3
1	163	2135.443	51.007	548.150	560.09882	115.95149	-248.11417
1	67	-2135.443	-51.007	-462.148	-560.09882	-536.91742	290.62057
2	163	-2518.734	205.776	512.594	158.93228	-515.09998	349.69424
2	67	2518.734	-205.776	-426.592	-158.93228	123.76487	-178.21072
3	163	-58.854	180.331	927.073	778.27289	-450.22870	82.89747
3	67	58.854	-180.331	-765.736	-778.27289	-255.12219	67.38152
4	163	2118.890	102.613	913.919	971.93201	-53.14775	-224.36574
4	67	-2118.890	-102.613	-752.583	-971.93201	-641.24170	309.87802
Max.		2135.443	205.776	927.073	971.93201	641.24170	349.69424
Comb		1	2	3	4	4	2
Min.		-2518.734	51.007	426.592	158.93228	-515.09998	-309.87802
Comb		2	1	2	2	2	4

- Deflections



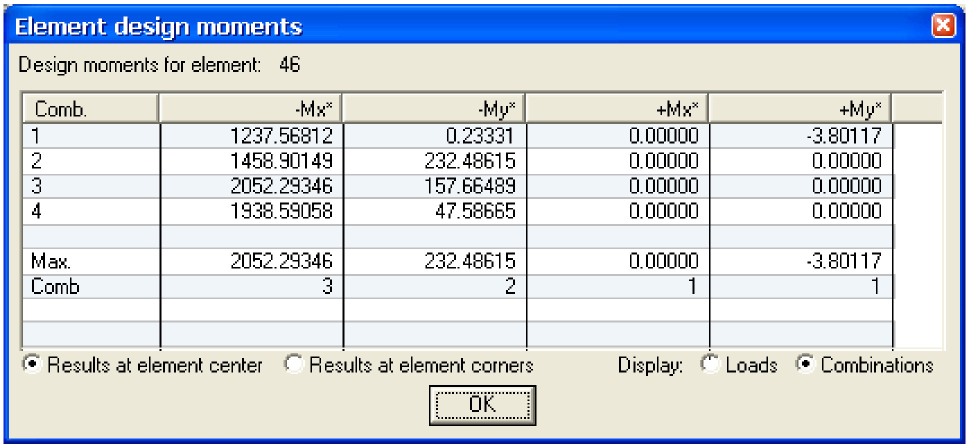
Comb.	x2 defl.	x3 defl.
1	-0.00117	-0.00256
2	0.00133	0.00228
3	0.00018	0.00072
4	-0.00113	-0.00245
Max.	0.00133	-0.00256
Comb	2	1

Elements:

Select one of the following:

Moments for element 6080
Forces for element 6080
Stresses at +Z for element 6080
Stresses at -Z for element 6080
Principal Stresses at +Z - ele. 6080
Principal Stresses at -Z - ele. 6080
Design moments for element 6080
Shear forces for element 6080
Steel areas for element 6080

For example, moments:

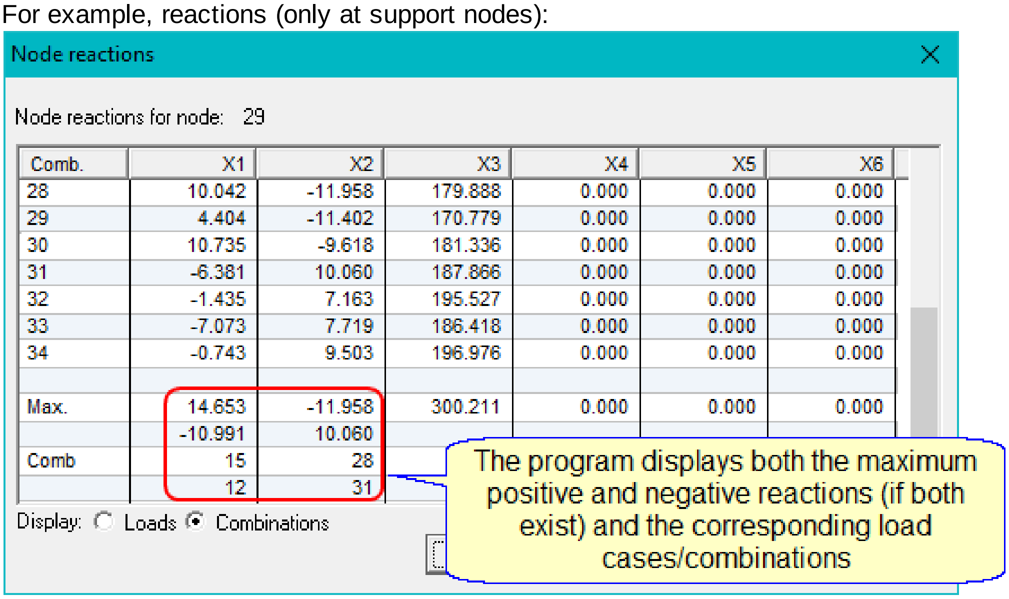


Nodes:

Select one of the following:

Displacements for node 1
Reactions for node 1

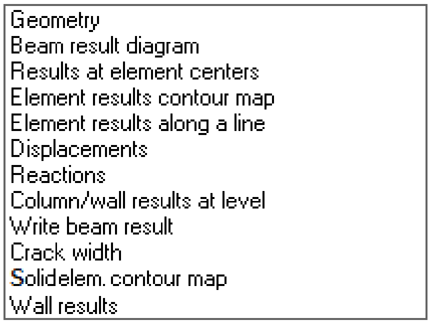
For example, reactions (only at support nodes):



6.11 Graphic results

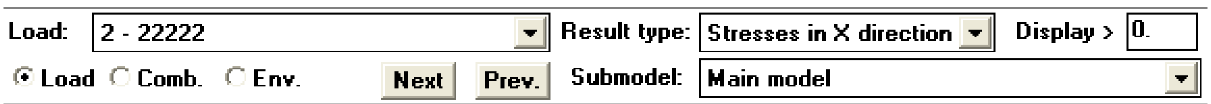
All result types - moments, shear and axial forces, stresses and deflections - for beam elements and finite elements may be displayed graphically on the screen.

The following options are available when **Display types** is selected in the menu:



The result display may be changed without entering the side menu 

option; the program displays the following menu at the bottom of the screen:



Select a different load/combination, result type or submodel; the display is updated immediately.

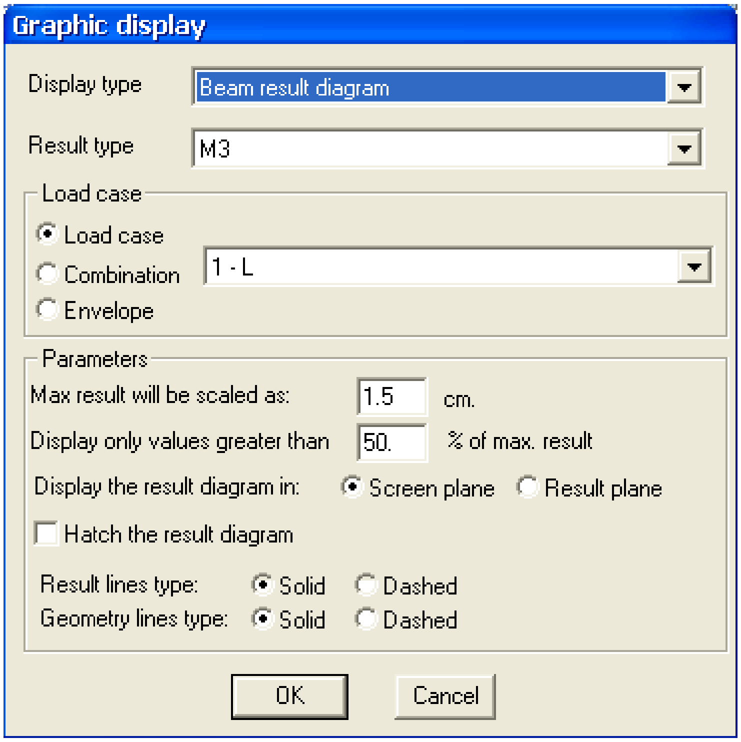
Note:

- the following terminology is used throughout the graphic results module:

Load cases	Load cases defined in the Loads module (prior to solving the model). Some of these cases may in fact be "Combined load cases", but are still referred to as Cases.
Load combinations	Load case combinations defined after solving the model using the Results Combinations option.

6.11.1 Beams

Result values are displayed at the beam ends and at the maximum span result, but only when the result value exceeds the percentage of the maximum specified in the display options below.



Result type

All result types may be displayed on the screen.

Axial	=	the axial force acting on the beam. The values are according to the sign at **JA**: • tension = negative • compression = positive
M2	=	moments about the local x2 axis.
M3	=	moments about the local x3 axis.
Torsion	=	the torsion moment about the local x1 axis.
V2	=	shear parallel to the local x2 axis
V3	=	shear parallel to the local x3 axis
Stresses	=	select compression, tension or "all" (both) at the extreme fibre = P/A ± M/S

Case / combination

Refer to General parameters 680.

Max result scale

Refer to General parameters 682.

Display values greater than

Refer to General parameters 682.

Display in plane

The program displays the beam result diagrams plotted on the model geometry. Select:

Screen plane

The result diagram is plotted to its full size on the screen plane without considering the direction of the local axis of the beam. This is the default option. The size of the result diagram will not change if the model is rotated, however the sign of the result may be reversed.

Result plane

The result diagram is plotted on the relevant beam local axis plane. Therefore, the diagram will appear smaller as the model is rotated and will appear as a line if the local axis is perpendicular to the screen. The moments are drawn on the tension side of the beam

Result values will be displayed at the beam ends and at the maximum span result, if the result value exceeds the percentage of the maximum specified in the display options.

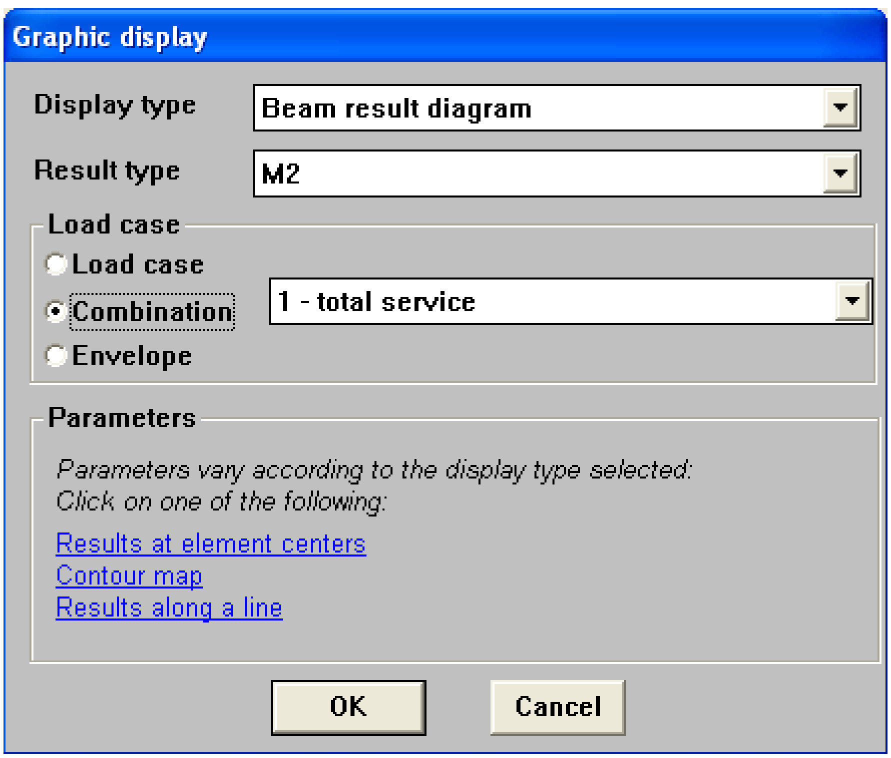
Hatch

Refer to General parameters 683.

Line type

Refer to General parameters 683.

6.11.2 Elements

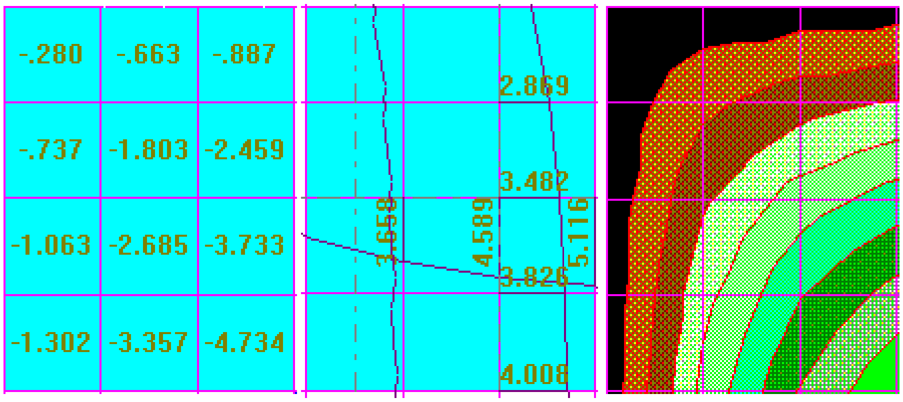


Display types

Select one of the three finite element display options:

- Results at element centres 660
- Result contour map 661
- Results along a line 665

For example:



Results at Element centres | Results along a line | Contour map

Result types

Select the result type to be displayed on the screen.

Quad and triangle elements:

Moment in X direction
Moment in Y direction
Moment XY
Forces in X direction
Forces in Y direction
Forces XY
Stresses in X at +z face
Stresses in Y at +z face
Stresses XY at +z face
Stresses in X at -z face
Stresses in Y at -z face
Stresses XY at -z face
Min. principal stress at +z face
Max. principal stress at +z face
Min. principal stress at -z face
Max. principal stress at -z face
Mx* Wood & Armer moment at +z face
Mx* Wood & Armer moment at -z face
My* Wood & Armer moment at +z face
My* Wood & Armer moment at -z face
Reinforcement in x dir. at +z face
Reinforcement in x dir. at -z face
Reinforcement in y dir. at +z face
Reinforcement in y dir. at -z face
Qx Shear force
Qy Shear force
Deflection (absolute value)
Deflection - perp. to element
Von Mises stresses at +z face
Von Mises stresses at -z face
Spring stresses

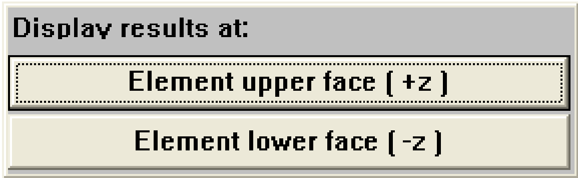
Solid elements:

Sx stresses
Sy stresses
Sz stresses
Sxy stresses
Syz stresses
Szx stresses
Maximum principal stresses
Minimum principal stresses

Note:

- The list of options differs slightly for Results along a line/Contour map/Results at element centre

- The definition of X,Y, Z axes varies with the result type selected. Refer to:
  Results at element centres 660
  Contour map 661
  Results along a line 665
- Stress results may be displayed on the upper or lower surface:



- Refer also to Sign Conventions - Elements

Case / combination

Refer to General parameters 680.

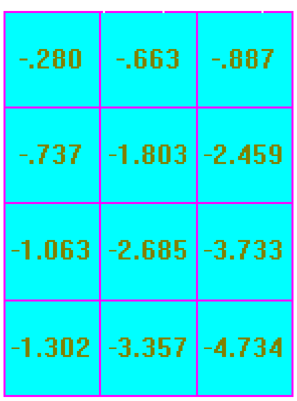
Parameters

Refer to:
Parameters - results at element centers 661
Parameters - contour map 662
Parameters - results along a line 666

Refer also to:
- Sign Conventions - Elements 650

6.11.2.1 Results at element centres

Display the geometry with the numerical result at the element center and/or corners written superimposed on the element. For example:

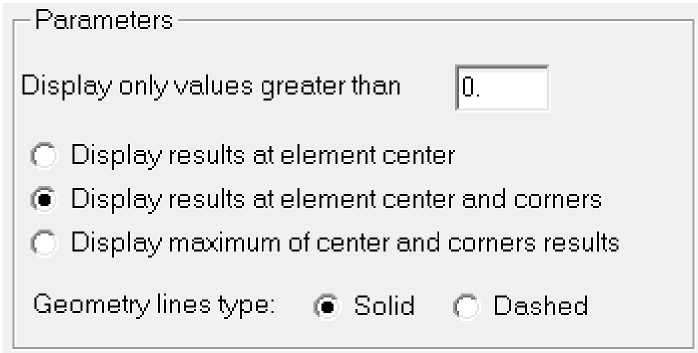


Note:
- Results are displayed according to the Default element result coordinate system 580, unless the result system was revised for specific elements (Options - Element results coord. system 577)
- the program does not change the sign of the results, ***even if the directions of the local axes are inconsistent***.
- to display local stress or moment concentrations at element corners, e.g. from a joint load applied at a node or a support reaction, select "Display results at element center and corners" in the Parameters 661 option. (Contour map 661 or Results along a line 665 will provide a more complete picture in these cases).

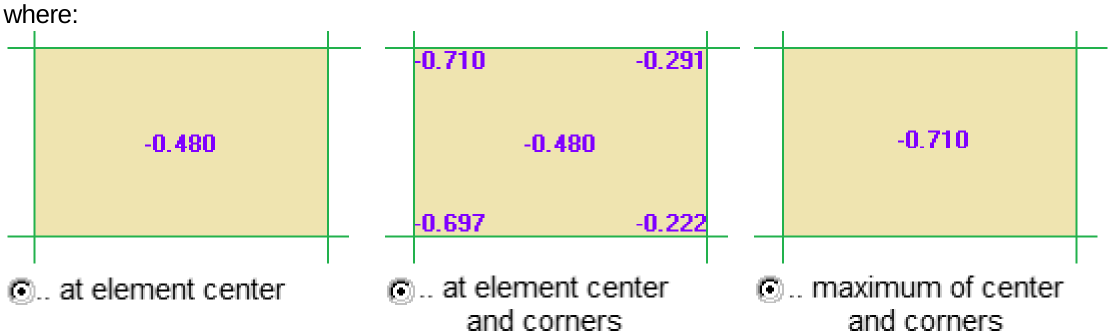
Refer to Graphic results - elements 658 for more information on the display options.

Refer to the following topics for more information on the result types:
- Moments, forces and stresses 639
- Principal stresses 640
- Shear forces 642
- Corner forces 643
- Design moments 641
- Reinforcement - method of calculation

6.11.2.1.1 Parameters



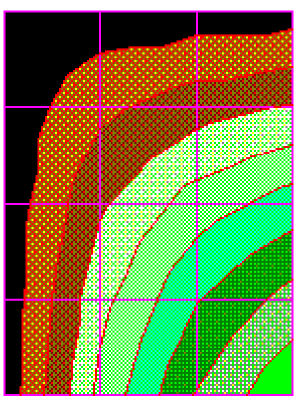
where:



Note:
- "maximum of center and corners" = maximum absolute value.

6.11.2.2 Contour map

Display the model geometry with a contour map of the results superimposed. Each line of the contour map gives the location of a specified value of the result. For example:



This option creates a contour plot from the element centre and corner results. In order to produce smooth and continuous contours, the program averages the exact corner results from all the elements connected to a particular node, as well as along the edges of two adjacent elements.

Note:
- Results are not averaged along a line where two planes meet, e.g. a wall is connected to a slab (results are averaged when the angle between the element planes in less than 10°)
- Stresses and reinforcement area results are ***not*** averaged along a line where two property groups meet, i.e. at locations where the element thickness changes.
- It is obvious that the averaging of results will create a discrepancy between the corner results in the tables and the results at element corners in the contour map. Refer also to the explanation for Results along a Line 665,
- Results are displayed according to the Default element result coordinate system 580, unless the result

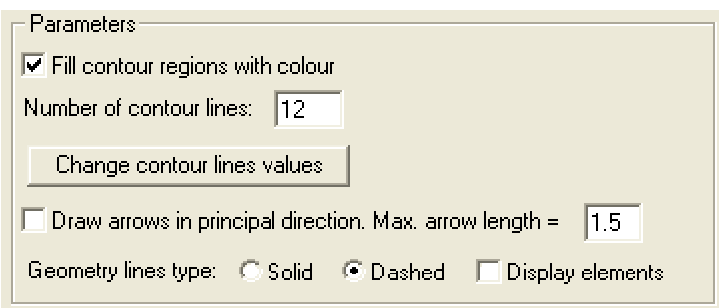
system was revised for specific elements ("Options"/"Element results coordinate system 577" in Menu bar)

- Reinforcement by diameter and spacing:
  a uniform steel area is calculated for each element according to the maximum area required in the element, i.e. only one colour is displayed in each element.
- Deflections:
  the following result types may be selected:
  - **Deflection - absolute value**
    The program displays the vector sum of the deflections in the three global directions, i.e. $\sqrt{(dX1^2 + dX2^2 + dX3^2)}$.
  - **Deflection - perp to element**
    The program displays the deflection perpendicular to the element (including the corner node deflections).
- Solid elements:
  The contour map displays the stresses on the surface of the elements. The surface may be either planar or cylindrical and is defined by pointing to three nodes. Refer to Solids - define surface 663.

Refer to Graphic results - Elements 658 for more information on the display options.

Refer to the following topics for more information on the result types:
Moments, forces and stresses 639
Principal stresses 640
Shear forces 642
Corner forces 643
Design moments 641
Steel area 646

6.11.2.2.1 Parameters



Fill with colour

Set the box to ☑ to fill the areas between contour lines with colour. The colour intensity will reflect the magnitude of the result values.

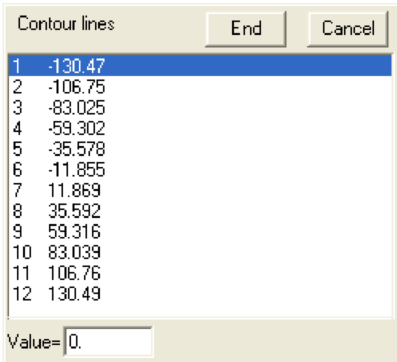
Number of contour lines

A maximum of 20 contour lines can be specified.

Change line values

The program scans the results to find the maximum and minimum values, and divides the resulting range of results into equally spaced steps for contour lines. These are the default values for the contour lines and initially "Equally spaced" is displayed for this option.

The values may be revised. For example, if twelve contour lines were specified, the program displays a table:



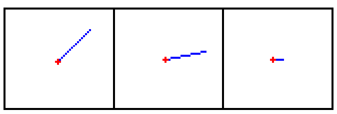
To revise a value:

- using the mouse/arrow keys highlight the line displayed the contour line to be revised; click the mouse.
- type in the new value in the **Value** text box.
- repeat for additional contour lines.
- click End to end revisions.

Note that program automatically recalculates the remaining contour line values so that the steps between defined values are equally spaced.

Draw arrows

Draw vectors representing the maximum/minimum principal stresses at the centre of each element. For example:



The direction of the vector is in the direction of the "Angle" in the tabular results (refer to Sign convention - elements 650) and the length is proportional to the maximum value of principal stress in the displayed elements.

Geometry line types

The contour map is superimposed on the geometry lines representing the beams, elements, etc. Select:

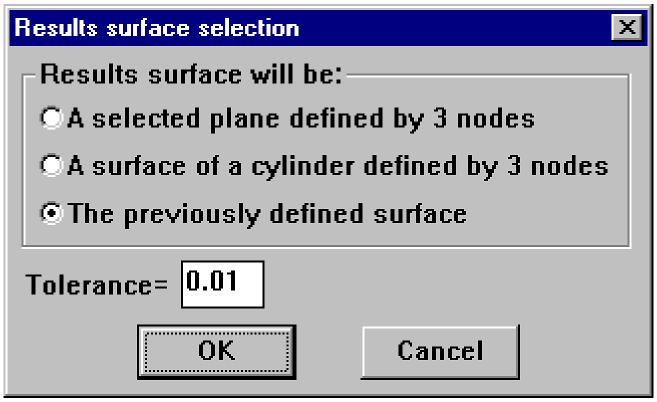
Solid
All displayed geometry lines are drawn as solid lines

Dashed
All displayed geometry lines are drawn as dashed lines

Display elements
Uncheck the box to suppress the element boundary lines; lines representing the beam are drawn.

6.11.2.2.2 Solids - define surface

The contour map for solid elements displays the stresses on the surface of the elements. The surface may be either planar or cylindrical and is defined by pointing to three nodes.



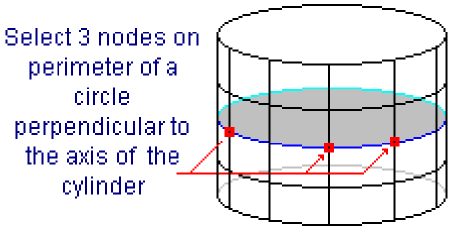
A selected plane

Display a contour map of stresses on a plane defined by three nodes. The program identifies all element surfaces lying on the plane and draws the contour map on them.

Note:
- the ***surface*** must lie on the ***face*** of solids. No results will be drawn if the plane cuts through an element.

Surface of cylinder

Display a contour map of stresses on the surface of a cylinder defined by three nodes:



The program identifies all element surfaces perpendicular to the perimeter of the circle and draws the contour map on them.

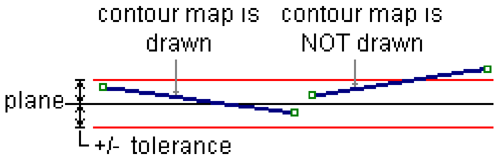
Note:
- the ***surface*** must lie on the cylinder defined by the projection of the circle. No results will be drawn if the cylinder cuts through the element.

Previous surface

Display the results on the previously defined surface.

Tolerance

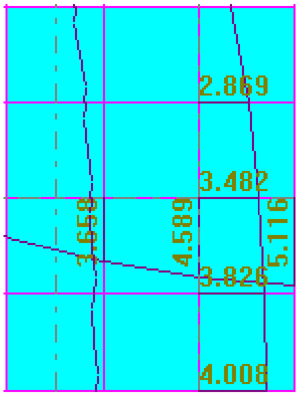
All corner nodes of the surface must be located at a distance less than the "tolerance" from the plane/ cylinder for the contour map to be drawn on it.



The tolerance is defined according to the model geometry units.

6.11.2.3 Results along a line

Display the results plotted along a section through the model. For example:



This option calculates the results at all points along any line arbitrarily drawn through the model, using the linear stress distribution assumed in each element. Therefore, this option will show a local stress or moment concentrations at element corners and the results correspond to those in the tables.

If a section is plotted along an element boundary, the program uses the results of one of the adjacent elements and does not average the values of all the adjacent elements.

The user selects the plot of the result type ***along the line*** or ***perpendicular to the line.***

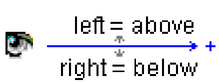
The result coordinate axes are defined as follows:

- **X *always*** refers to the axis along the line of the section.
- **Y *always*** refers to the axis perpendicular to X (in the element plane).
- **Z** axis is perpendicular to the plane of the element:
  - Plane models:
    The positive direction of the Z axis of ***all*** elements is the positive direction of the global X3 axis, i.e. if the local x3 axis of adjacent elements point in opposite directions, the result diagram will still be continuous.
  - Space models:
    The positive direction of the Z axis of each element is the positive direction of the local x3 axis of the element; if the local x3 axis of adjacent elements point in opposite directions, the result diagram will ***not*** be continuous.

The results are drawn 'above' and 'below' the line as follows:

- stresses: compression stresses are negative and drawn below the line.
- moments: moments are positive if they create compression stresses 'above' the element and are drawn below the line.
- deflections: relative to the global axes.

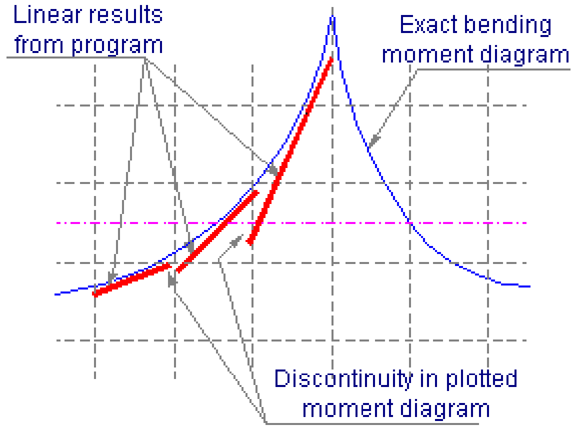
The 'above' side is always to the ***left*** of the line when looking in its ***positive*** direction:



The positive direction is determined as follows:

- line parallel to a global axis (within 5° of the axis):
  the positive direction of the axis
- plane model - line defined by 2 points:
  from the 1st point to the 2nd point
- space model - general case:
  in the general direction of +X1, except -
  - if perpendicular to X1: in the general direction of +X2
  - if perpendicular to X1-X2: in the general direction of +X3

Note that while the stress distribution within each element is assumed linear, the actual stress distribution is usually non-linear. This discrepancy results in discontinuities in the result diagram as shown below. Using a finer mesh in areas where the slope of the result diagram varies significantly will improve the accuracy of the results. An indication of the inaccuracy can be obtained by comparing results at the same node for adjacent elements - theoretically they should be identical but in general they differ.



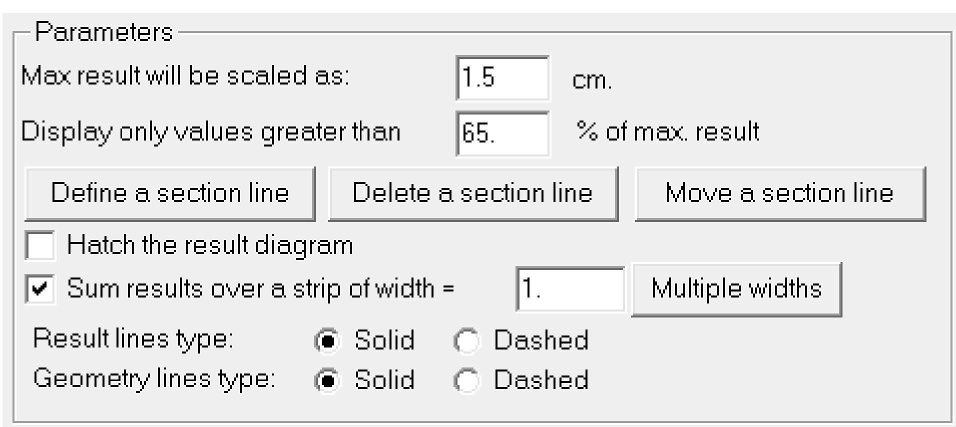
Refer to Graphic results - Elements 658 for more information on the display options.

Refer to the following topics for more information on the result types:
Moments, forces and stresses 639
Principal stresses 640
Shear forces 642
Corner forces 643
Design moments 641
Steel area 646
For a beam section modeled as a strip of elements - the program can calculate equivalent bending moments and shear from the element in-plane forces. Refer to Strip moments 668

6.11.2.3.1 Parameters

To display a demo video that explains how to use the "Results along a line" option:

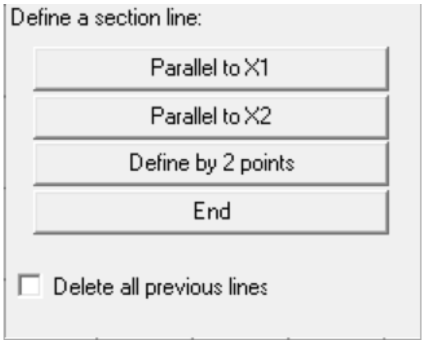
- *click on*  *to start the video*
- *then click on*  *to enlarge the display.*



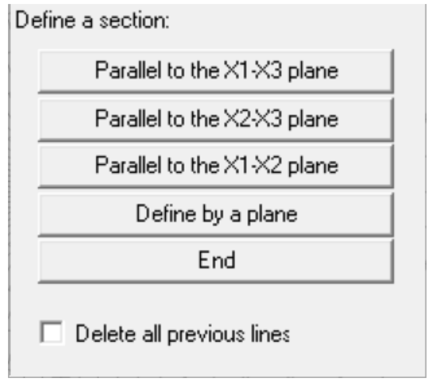
Define a section line

Plot any of the results types along a section line drawn through the model in any direction.

- Plane models: the section is defined by a line drawn in any direction on the surface of the model.

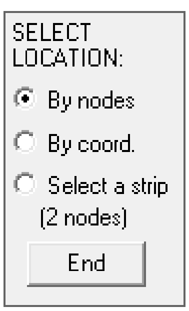


- Space models: the section is defined by the intersection of a plane drawn in any direction and the model surface.



- ***Parallel to axis***
  Select the axis and then define the coordinate on the perpendicular axis
- ***Define by 2 points***
  Use this option to draw a section in any arbitrary direction in a plane model.
- ***Parallel to plane***
  Select the global plane and then define the coordinate on the perpendicular global axis.
- ***Define by plane***
  In space models, locate the section line by three points which define a plane cutting through the model. If the display plane is parallel to one of the Global Planes, the program assumes that the section plane is perpendicular to the Global Plane and requests two points only.
- ***Delete all previous lines***
  Check the box to delete all current section lines. By default, the program saves all previously defined section lines.

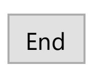
For all options:



By nodes:
the cursor highlights nodes on the model; select one

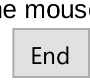
By coord:
Move the cursor to any location on the model.

Select a strip:
Select 2 points defining the strip; the section line is drawn ***between*** the two points selected. This option is identical to the Sum results over a strip width 668 option in the parameters box

Repeat for multiple section line locations. Click 

when the selection is complete

Delete a section line

Use the mouse/arrow keys to highlight one or more section line to be deleted with the rectangular blip ■ ; Click 

when the selection is complete

Delete a section line

Move a section line

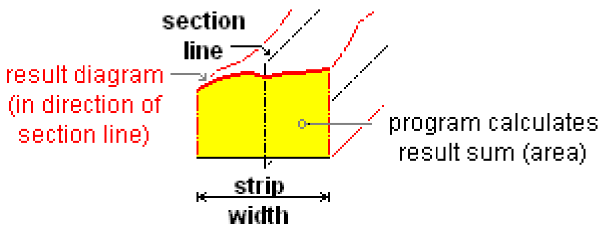
- move the adjacent to the centre of the section line so that it is highlighted with a ■; click the mouse
- move the to the new location of the section line.

Hatch results diagram

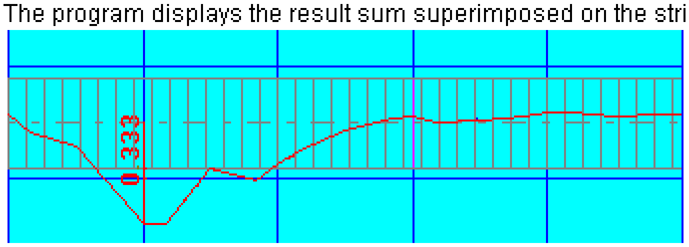
Refer to General parameters 683.

Sum results over a strip

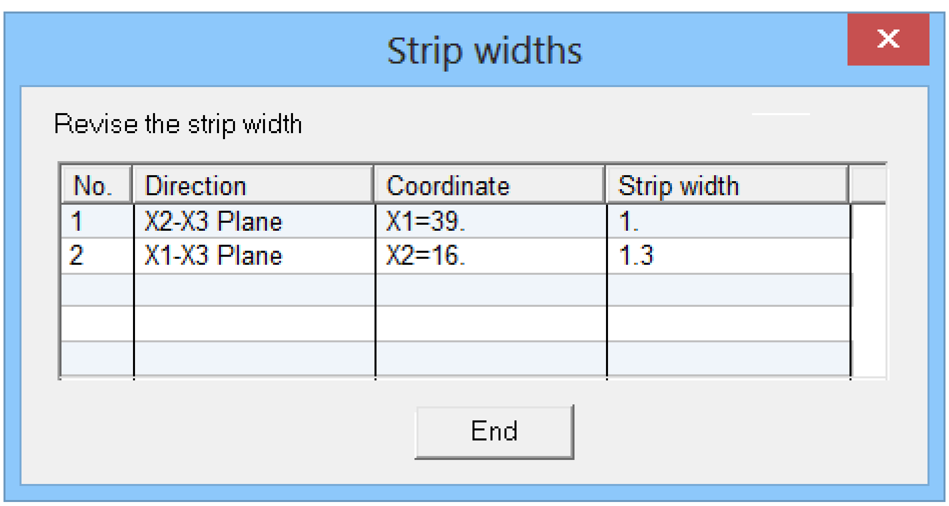
Display the ***total*** result across the width of a strip defined along the section line.



The program displays the result sum superimposed on the strip:



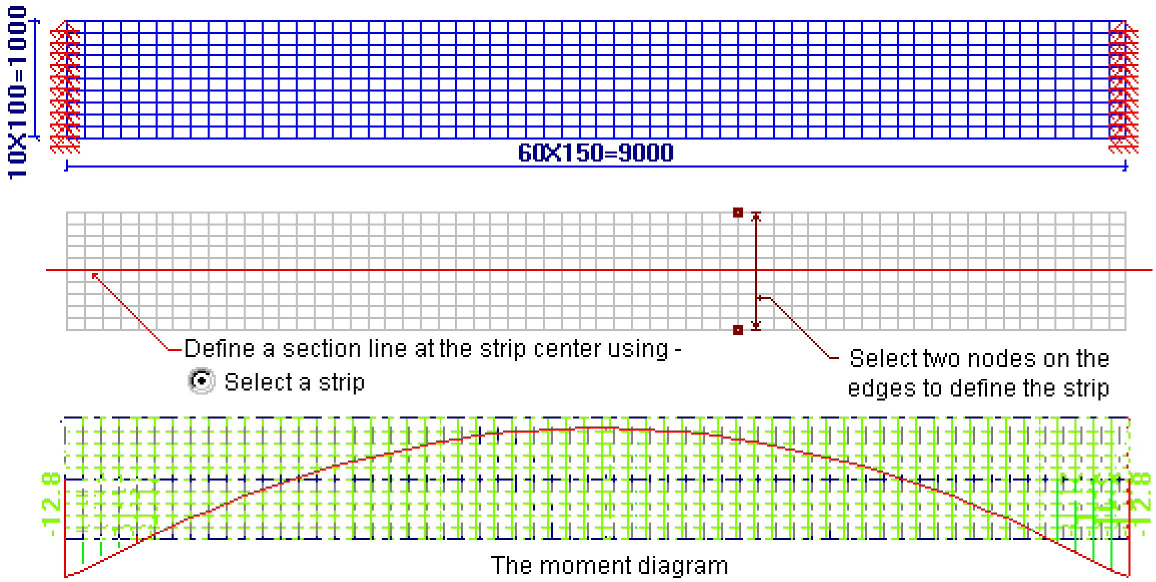
A different strip width may be defined for each line; click Multiple widths and specify a different width for each defined line:



No.	Direction	Coordinate	Strip width
1	X2-X3 Plane	X1=39.	1.
2	X1-X3 Plane	X2=16.	1.3

6.11.2.3.2 Strip moment

For a beam section modeled as a strip of elements - the program can calculate equivalent bending moments and shear from the element in-plane forces. For example:



Note:
A moment envelope should ***not*** be requested. For example - the maximum moment envelope for beam with two load cases:

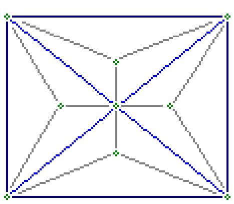
- case 1 : stresses = +10 at the top and -10 at the bottom.
- case 2 : stresses = +5 at the top and -5 at the bottom.

The program searches for the maximum stresses at top and bottom and uses them to calculates the maximum moment. In this example, these stresses are +10 at the top and -5 at the bottom, which will give a moment that is less than the maximum (from case 1).

6.11.2.3.3 General

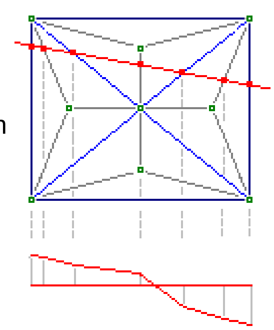
In general, the program calculates the results at any point on the surface of a rectangular element as follows:

- Each element consists of 4 triangular elements; the program calculates results at the centre of the rectangle and at the centre-of-gravity of each triangle, i.e., results are available at the 4 corners and 5 internal points:



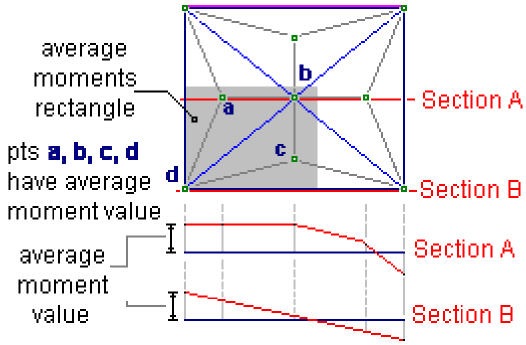
The program can linearly interpolate results along any of the lines shown in the figure.

- when the user defines a section line cutting through the element, the program calculates the result values at the intersection points of the section line with the internal lines, for example:



- if the "Average moments" option is selected, the program first calculates the average values at each of the nine points and then interpolates these values as described above.

- If the "Reduced moment" option is selected, the program assigns the Reduced moment value to any of the 9 result points that lie within the rectangle defined by the user. The program then interpolates and draws the moment diagram as explained above. For example:

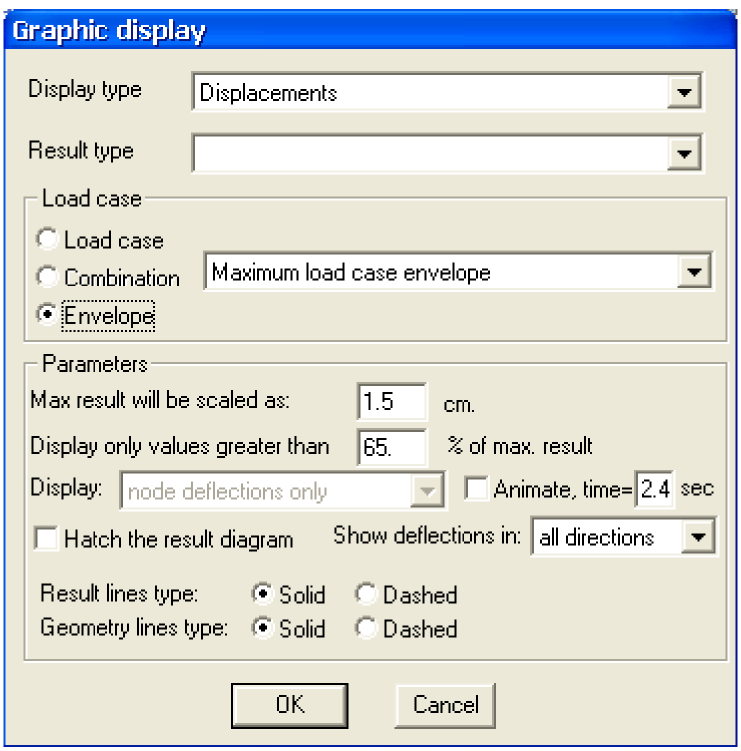


Note that in section B, the "reduced moment" value is drawn only at point **d** because the section line intersects only one other internal line, at the bottom-right corner.

- If both the "Average moments" option and the "Reduced moment" option are selected, the program first 'averages' the moments, then 'reduces' them.

6.11.3 Deflections

:



Load case / combination

Refer to General parameters 680.

Max result scale

Refer to General parameters 682.

Display values greater than

Refer to General parameters 682.

Display: deflection type

This option is relevant for beam elements only.

- **Node and beam deflections**
  The complete deflected structure will be plotted. (i.e. the deflections of the two following options are combined).
- **Node deflections**
  The program will plot the nodes in their deflected location and connect them with straight lines representing the beams. This is the fastest option.
- **Beam deflections**
  Only deflections due to beam loads are plotted; the beam ends remain at their original locations.

Note:

- If **Envelope** is selected, only **Node deflections** may be displayed.

Animate

Set this option to ☑ to animate the deflections.

- The model will deflect to its full displacement in five equal steps during the **time =** interval specified.
- The animation will continue until the **End animation** button at the bottom of the screen is clicked.

Deflection direction

Display deflections and values in one of the following directions:

- **Total of all directions**
  Display only the vector sum of the deflections in the three global directions, i.e. sqrt $(dX1^2 + dX2^2 + dX3^2)$.
- **Global X1/X2/X3 directions**
  Display the deflection value for one of the global directions only.

Deflections - values:

The program displays the numerical value of the deflection in the form ddd / 10^n, where ddd is written adjacent to the beam and the factor 10^n appears at the bottom of the screen.

Examples:

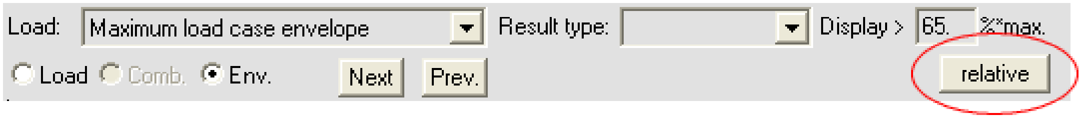
- **51** written adjacent to the element; **Values are * 10^3** at the bottom of the screen.
  deflection = 51/1000 = 0.051 (current length units).
- **272** written adjacent to the element; **Values are * 10^2** at the bottom of the screen.
  deflection = 272/100 = 2.72 (current length units).

Note:

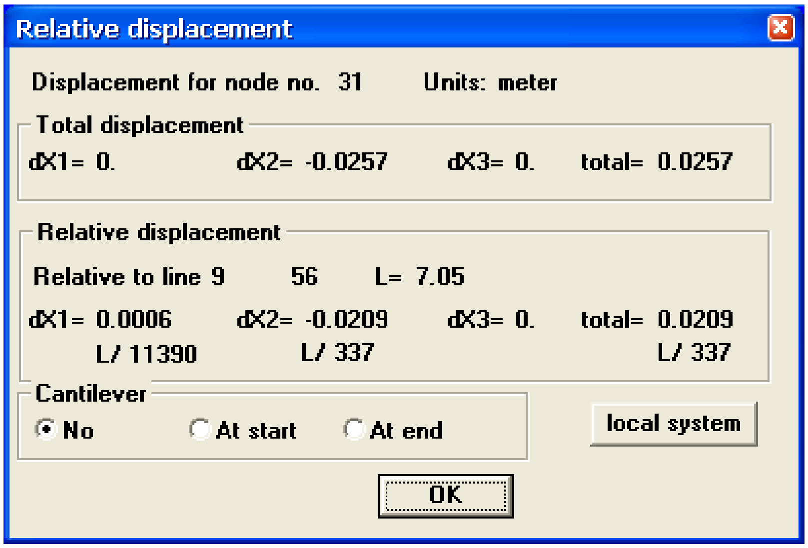
- only flexural deflections are displayed; ***shear deflections are not displayed*.**

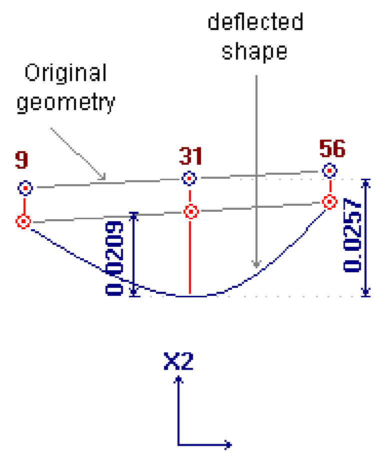
The deflection of a node relative to any other two nodes may also be displayed.

- click Relative

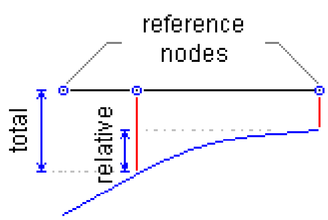


- select the node whose deflection is to be displayed
- select the two reference nodes.
- The program calculates and displays the relative deflection. For example:

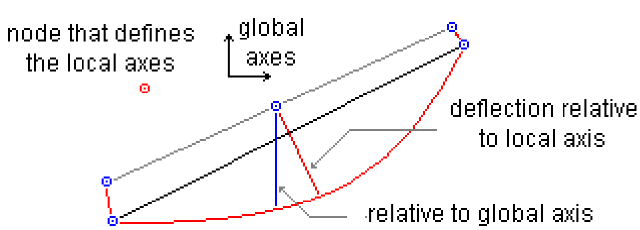




- if the beam is a cantilever select **At start** or **At end**. For example - "at start":



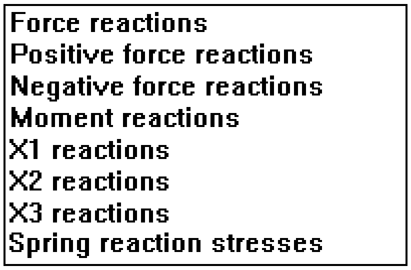
- select local system to display the deflection perpendicular to the line joining the reference nodes. Select any node to define a plane. For example:



6.11.4 Reactions

Display the force and moment reactions at the supports.
For submodels: the program display the reactions at the connection points to the main model.

Select one of the following:



- ***Reactions***
  - Envelope:
    The program calculates the maximum positive and negative reactions:

Force reactions	=	the absolute maximum of the "positive" and "negative" force reactions
Positive reactions	=	the maximum positive value is displayed; nothing is displayed if all values are negative
Negative reactions	=	the maximum negative value is displayed; nothing is displayed if all values are positive.
X1/X2/X3 reactions	=	the same as "Force reactions", but in the specified direction only
Moment reactions	=	the absolute maximum moment

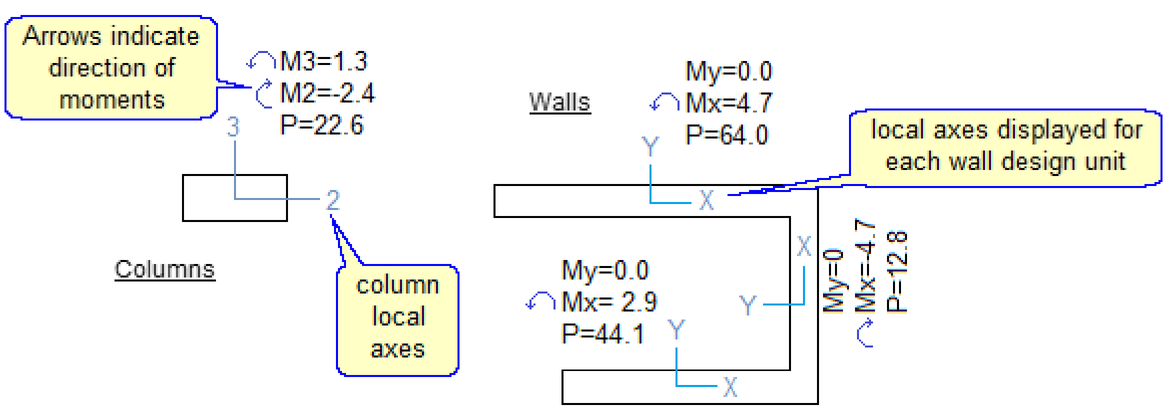
  - Load case/combination
    All force options display the same values
- ***Spring reaction stresses***

  The program calculates the absolute maximum soil pressure under nodes with springs. Refer to Spring reaction stresses 644.

Refer to Graphic results - beams 656 for information on the display options.

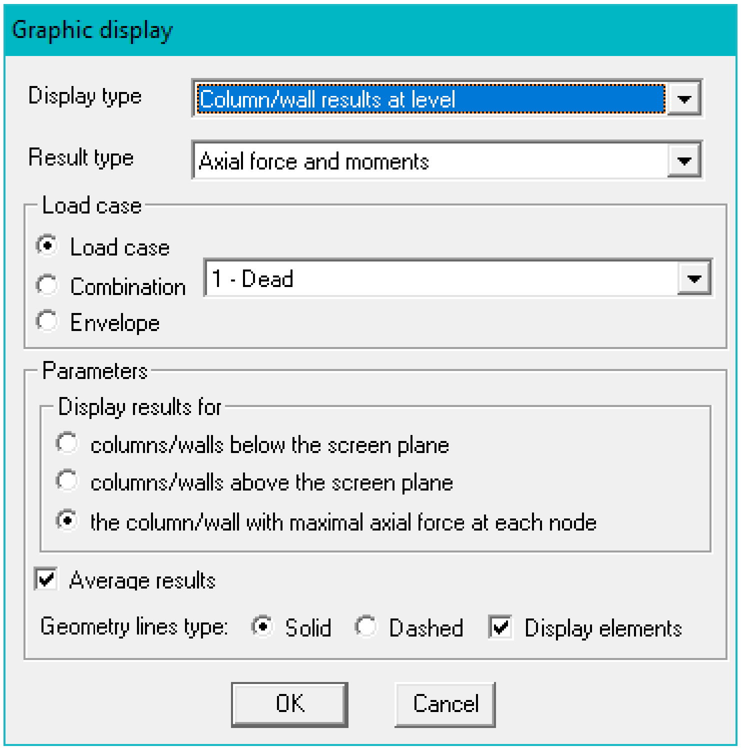
6.11.5 Column/wall results at level

For a selected plane, display the results from the columns and walls that are attached to it, either above or below the plane. For example:



Note:
- display a single plane before selecting this option.
- select "Draw" - "Draw columns" and "Draw walls" to add the colum/wall sections to the display.
- the wall results are displayed for each "design unit"
- the directions of the moments are indicated by the arrows adjacent to the moment values.

- ▫ Columns: M2 and M3 are the moments ***about*** the x2 and X3 local axes, respectively.
- ▫ Walls: Mx and My are the moments ***in the direction of*** the X and Y local axes, respectively
- the local axes and the arrows for columns and walls are displayed only if the **Display - local axes** option is selected and ☑ **x2** and/or ☑ **x3** are checked.



Result type

All result types may be displayed on the screen. Select one of the following:

Axial force	=	Display P results only
Axial force and moments	=	Display P, M2 and M3 results
Shear	=	Display V2 and V3 results
All forces	=	Display P, M2, M3, V2 and V3 results

Case / combination / Envelope

Refer to General parameters 680.

Parameters

Display one of the following:

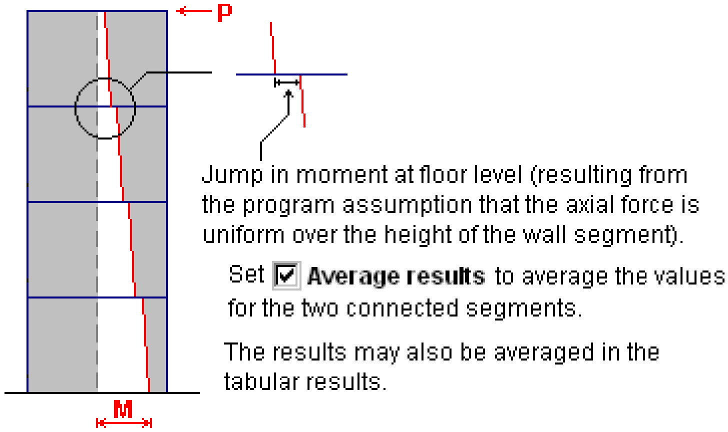
- the results of the column/walls that are attached ***below*** the selected level
- the results of the column/walls that are attached ***above*** the selected level
- at each column/wall location: the result from above or below that has the maximum axial force.

Note:

- for wall results, there is usually only a small difference between the results 'above' and 'below' a plane. Displaying 'average results' (above or below) is generally sufficient.

Average results

For wall results only:



Geometry line types

The contour map is superimposed on the geometry lines representing the beams, elements, etc. Select:

Solid
All displayed geometry lines are drawn as solid lines

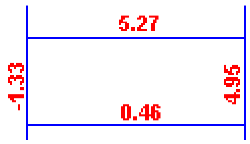
Dashed
All displayed geometry lines are drawn as dashed lines

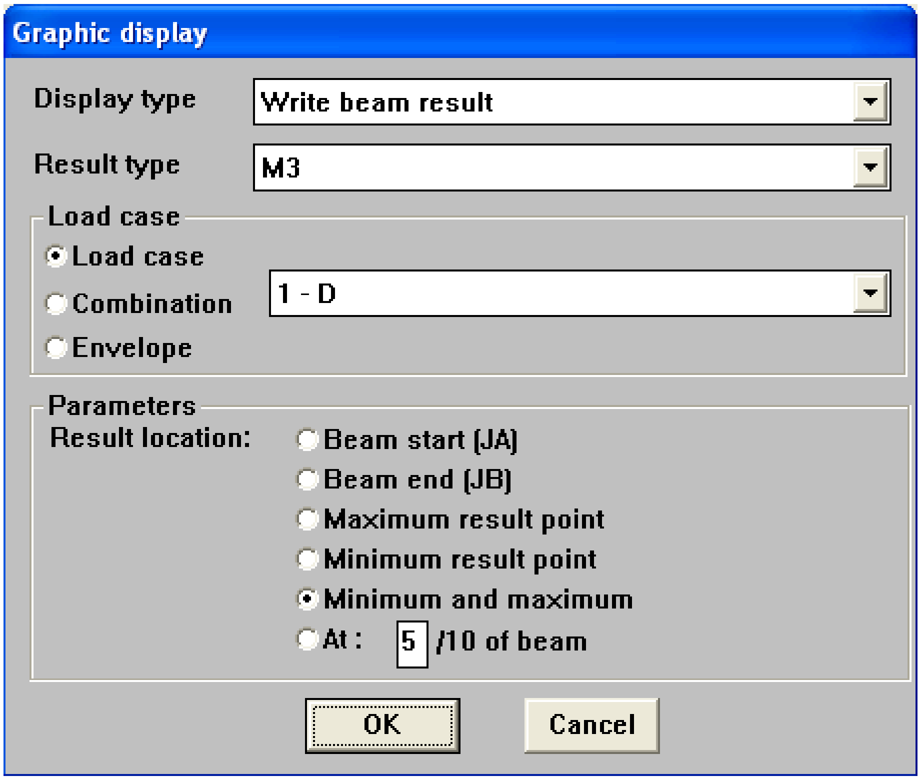
Display elements
Uncheck the box to suppress the element boundary lines; lines representing the beam are drawn.

6.11.6 Write beam results

Write the value of a selected result adjacent to the beam. Note that the value will be written at the midpoint of the beam span (even if you select the result at the beam end).

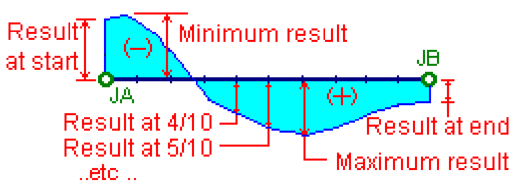
Example:





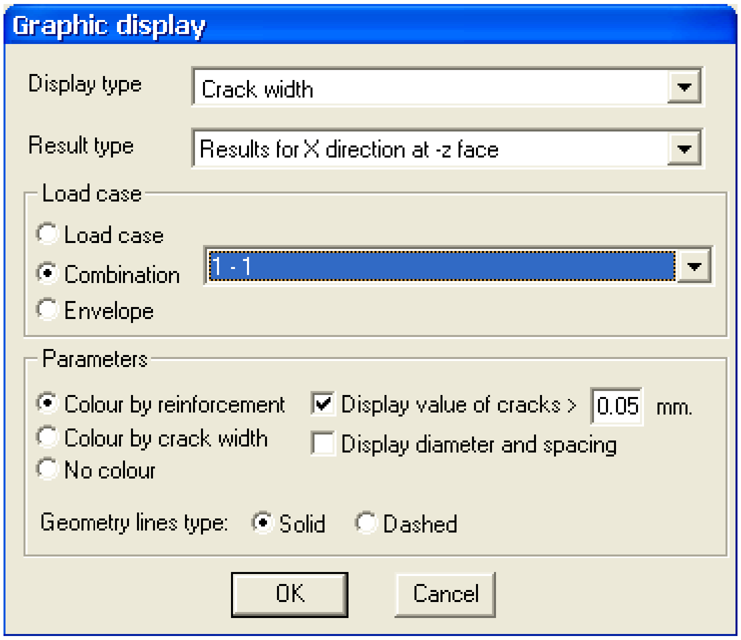
Parameters

Select the beam result to display:



6.11.7 Crack widths

Display crack widths for current reinforcement or reinforcement required to limit crack widths to a specified value according to BS8007 or EC2.



Result location

Crack width results may be displayed for either direction and for either face.

Select one of the following combinations:

Results for X direction at +x3 face
Results for X direction at -x3 face
Results for Y direction at +x3 face
Results for Y direction at -x3 face

Note:
- X/Y refers to the Element result coordinate system
- +x3/-x3 refers to the local element coordinate system

Parameters - color coding

Specify the colour coding.

by reinforcement
The elements are filled with colour according to reinforcement group (diameter and spacing)

by crack width
The elements are filled with colour according to crack width values.

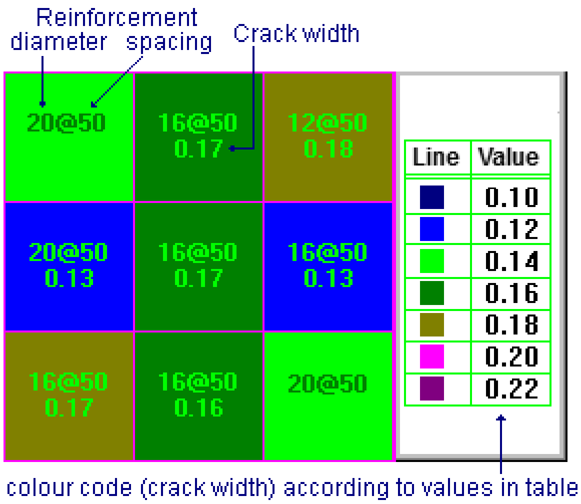
Parameters - result type

The result types may be displayed simultaneously:

cracks
Display crack width values at the element centre. Small values may be removed from the display by entering a minimum display width in the adjacent text box.

diameter and spacing
Display diameter and spacing values at the element centre

For example:



6.11.8 Walls

Display the wall results graphically at the centre line of each element segment or for the entire wall. Result values at the wall ends and any max/min intermediate result are displayed, if the result value exceeds the percentage of the maximum specified in the display options.

To display a demo video that explains how to display and interpret wall results:

- *click on*  *to start the video*
- *then click on*  *to enlarge the display.*

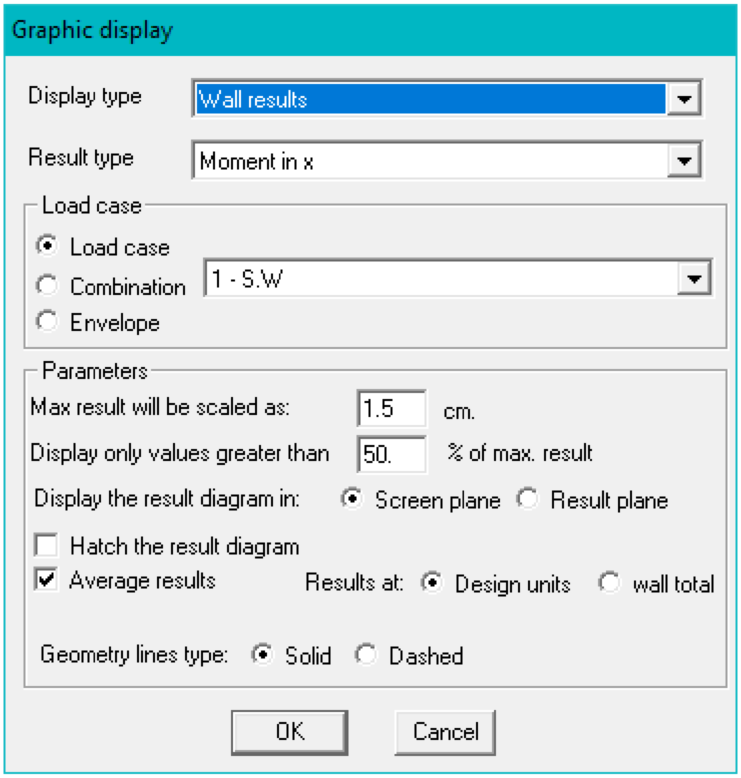
Note:

- *this video includes an explanation on "design units" and wall drawings*

Axial result values are according to the sign at bottom (smaller height axis coordinate):

- tension = negative
- compression = positive

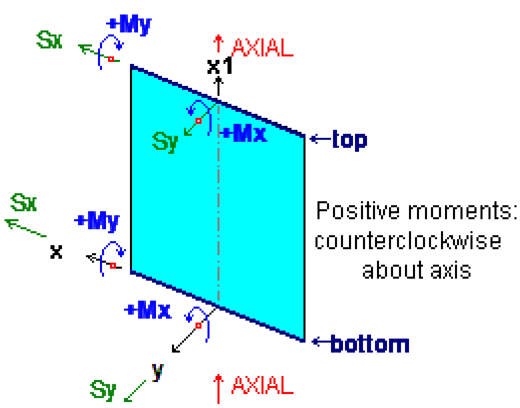
The following menu is displayed when **Wall results** is selected as the result type:



Result type

All result types may be displayed on the screen for each individual wall design unit or for the entire wall.

Axial = the axial force acting on the wall segment.
Mx = moments in the direction of the x axis.
My = moments in the direction of the y axis.
Torsion = the torsion moment about the x1 axis.
Sx = shear parallel to the x axis
Sy = shear parallel to the y axis



Note:

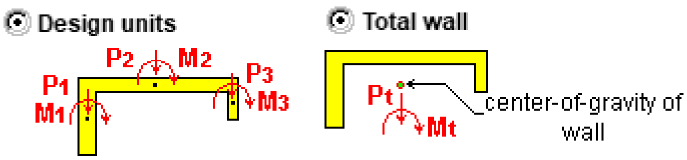
- the result X,Y axes (for design units and the total wall) may be displayed on the model using the Display - local axes option.

Display result diagram in plane

Refer to Beams - display plane 657.

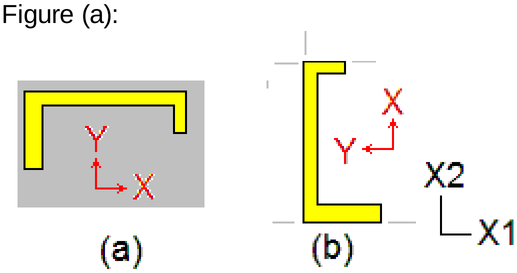
Results at:

Display the result value for each "design unit" of the total for the entire wall. For example, a wall with three segments, where each segment is a design unit or all three are combined to one unit:



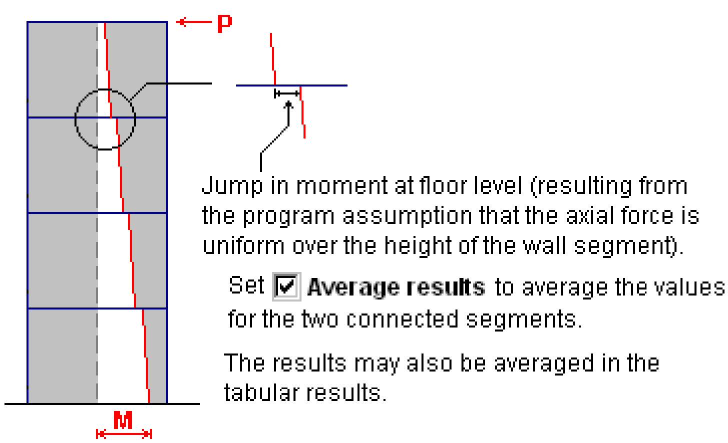
for **Wall total**:

- **$P_t = P_1 + P_2 + P_3$**
- stresses in each segment due to **Pt** and **Mt** are identical to those due to **Mn, Pn.**
- The moment axes X,Y are the horizontal and vertical axes in the ***geometry*** wall section definition - Figure (a):



The result axes remain unchanged no matter how the section is rotated when added to the model - Figure (b).

Use average results



6.11.9 General parameters

Load cases / combinations

Select the load case or load combination to be displayed. You may select a single load case or combination or an envelope of maximum/minimum results for all of the load cases / combinations.

Select one of the following options and then select the case/combination in the list box.

Load case
The program displays a list of the load cases in the box at the centre; select one.

Combination
The program displays a list of the combinations in the box at the centre; select one.

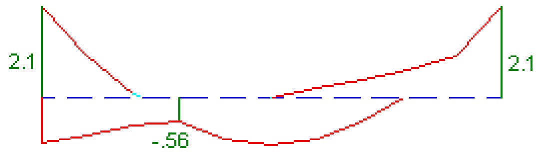
Envelope

- **Beams / Reactions / BS8007**:
  Select one of the following options in the box at the centre:

  Load case envelope
  Combination envelope

  The program searches all load cases/combinations for the maximum/minimum results.

  The following is an example of an envelope for bending moments:



- **Elements / Displacements**:
  For all element result types, select a "maximum" result envelope or a "minimum" result envelope; the program cannot display maximum and minimum results simultaneously.

  Maximum load case envelope
  Minimum load case envelope
  Maximum combination envelope
  Minimum combination envelope
  Maximum of selected loads
  Minimum of selected loads
  Maximum of selected combinations
  Minimum of selected combinations

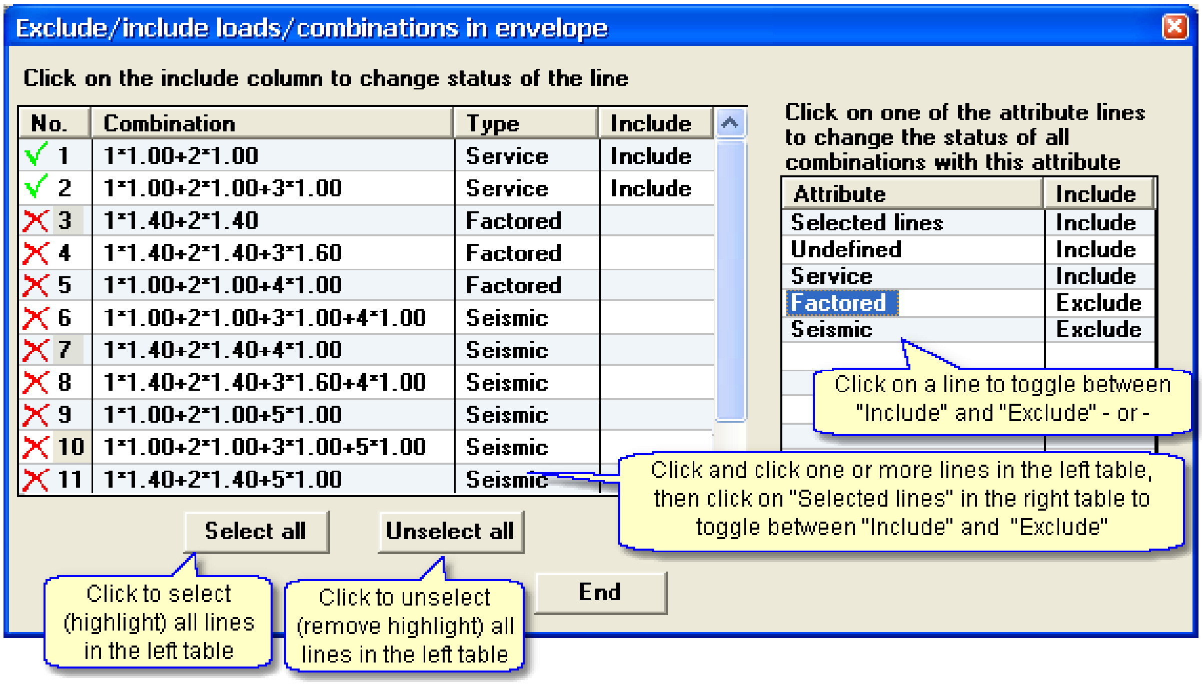
  - ***Element Centre***:
    maximum = largest positive result or smallest negative result
    minimum = largest negative result or smallest positive result
  - ***Contour Map:***
    maximum = positive results only
    minimum = negative results only
  - ***Results along a line:***
    maximum = largest positive result or smallest negative result
    minimum = largest negative result or smallest positive result

For "selected loads" or "selected combinations":

If you want the program to ignore one of the load cases/combinations, you may temporarily deactivate it using the following menu. The load case/combination is not deleted, and it may be reactivated at any time.

Combinations may be deactivated by selecting combination "types".

Example: display deflection results for service load combination loads only. A type named "Service" has been defined and assigned to the service load combinations (all other combinations were assigned with other Types):



- click on the "Factored" and "Seismic" lines in the right table to "Exclude" them; the left table will appear as shown.
- Alternatively, click Unselect all in the left table, click and highlight rows 1 & 2 in the left table, then click on "Selected lines" in the right table to "Include" them.

Display only values greater than:

For clarity, part of the numerical values may be deleted from screen (the entire geometry and result diagram are plotted). All values less than a given fraction (default = 0.5) of the maximum result will not be displayed.

Move the into the text box, type a new percentage and press [Enter].

Example:
- Maximum bending moment = 12 kN m and fraction = 0.5 : Only numbers greater than 6 kN m will be displayed on the screen.

Note:
- the value is an absolute one. For example, if you specify 4.50, then -2.5 and +2.5 will not be written on the screen but both -7.2 and +7.2 will be.

Maximum result will be scaled as:

The bending moment diagrams, deflections, etc. are displayed relative to a scale chosen as follows:

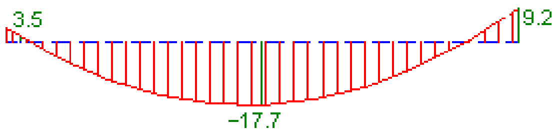
The program searches for the maximum result in the plot area and plots it on the screen as the dimension listed above - the default value is 1.5 cm (0.6 in.). All other results are plotted in proportion to this value.

Move the into the text box, type a new dimension in cm. and press [Enter].

Example: bending moment diagram:
- Maximum moment = 12 kn m is drawn as 1.5 cm. on plotted diagram;
- Moment = 4 kn m will be drawn as 0.5 cm.

Hatch

Set the box to ☑ to hatch the results diagram. For example:



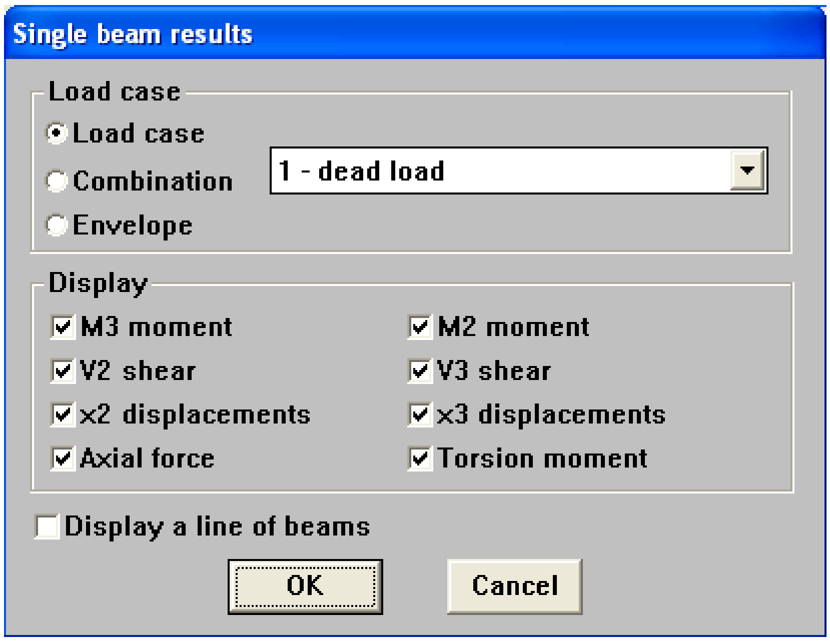
Line type

Select ⊙ **Solid** or ⊙ **Dashed** for the geometry and results lines

6.12 Single beam results

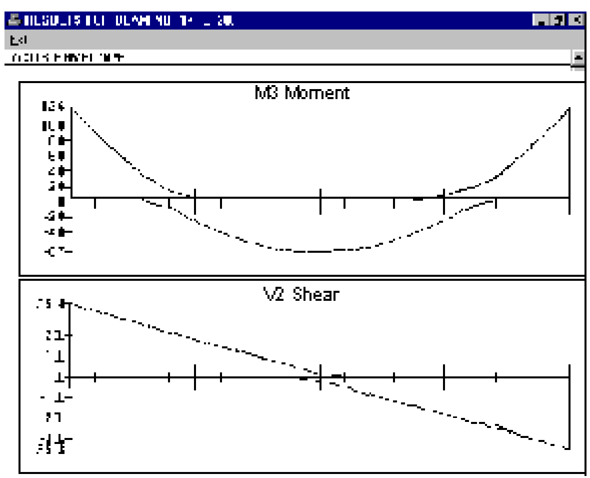
To display all graphic results simultaneously for a selected beam or a line of beams.

- select the load case/combination and the result types to be displayed:



- select any beam, or set ☑ **Display a line of beams** and specify the first and last beams in a continuous line of *STRAP* members
- scroll through the results.

Example:



6.13 Wood & Armer Equations

The ***STRAP*** output tables display the elastic bending and torsional moments at the centre of each element. (M_x, M_y and M_{xy}). For reinforced concrete plates, these moments must be translated into equivalent design moments M_x* and M_y* which take into account not only the bending moments M_x and M_y but also the torsional moment M_{xy}. These design moments are then used to calculate the required reinforcement steel.

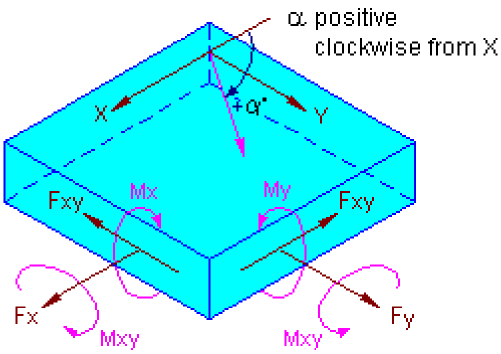
The calculation of the design moments M_x* and M_y* is based on the Wood & Armer equations.

The postprocessor first calculates the element moments M_x, M_y and M_{xy} relative to any orthogonal coordinate system chosen (the "result coordinate system"). The design moment calculation assumes that the reinforcement X* axis is parallel to the X axis of the result coordinate system and that the Y* reinforcement axis is skewed at an angle alpha (usually 90°).

A similar calculation must be carried out to derive the design forces F_x* and F_y* from the *STRAP* results F_x, F_y and F_{xy}.

Refer also to Element coordinate systems 577 for a more detailed explanation on the result and reinforcement coordinate systems.

The sign convention for the design moment equations is shown in the following Figure:



The equations are:

Moments:

Reinforcement:

Orthogonal (X,Y)

a. Bottom

$$M_x^+ = M_x + |M_{xy}|$$

$$M_y^+ = M_y + |M_{xy}|$$

If $M_x^+ < 0$

$$M_x^+ = 0$$

$$M_y^+ = M_y + \left|\frac{M_{xy}^2}{M_x}\right|$$

If $M_y^+ < 0$

$$M_y^+ = 0$$

$$M_x^+ = M_x + \left|\frac{M_{xy}^2}{M_y}\right|$$

b. Top

$$M_x^+ = M_x - |M_{xy}|$$

$$M_y^+ = M_y - |M_{xy}|$$

If $M_x^+ > 0$

$$M_x^+ = 0$$

$$M_y^+ = M_y - \left|\frac{M_{xy}^2}{M_x}\right|$$

If $M_y^+ > 0$

$$M_y^+ = 0$$

$$M_x^+ = M_x - \left|\frac{M_{xy}^2}{M_y}\right|$$

Skew (X,α)

a. Bottom

$$M_x^+ = M_x + 2\,M_{xy}\cot\alpha + M_y\cot^2\alpha + \left|\frac{M_{xy} + M_y\cot\alpha}{\sin\alpha}\right|$$

$$M_\alpha^+ = \frac{M_y}{\sin^2\alpha} + \left|\frac{M_{xy} + M_y\cot\alpha}{\sin\alpha}\right|$$

If $M_x^+ < 0$

$$M_x^+ = 0$$

$$M_\alpha^+ = \frac{1}{\sin^2\alpha}\left(M_y + \left|\frac{(M_{xy} + M_y\cot\alpha)^2}{M_x + 2\,M_{xy}\cot\alpha + M_y\cot^2\alpha}\right|\right)$$

If $M_\alpha^+ < 0$

$$M_\alpha^+ = 0$$

$$M_x^+ = M_x + 2\,M_{xy}\cot\alpha + M_y\cot^2\alpha + \left|\frac{(M_{xy} + M_y\cot\alpha)^2}{M_y}\right|$$

b. Top

$$M_x^+ = M_x + 2\,M_{xy}\cot\alpha + M_y\cot^2\alpha - \left|\frac{M_{xy} + M_y\cot\alpha}{\sin\alpha}\right|$$

$$M_\alpha^+ = \frac{M_y}{\sin^2\alpha} - \left|\frac{M_{xy} + M_y\cot\alpha}{\sin\alpha}\right|$$

If $M_x^+ > 0$

$$M_x^+ = 0$$

$$M_\alpha^+ = \frac{1}{\sin^2\alpha}\left(M_y - \left|\frac{(M_{xy} + M_y\cot\alpha)^2}{M_x + 2\,M_{xy}\cot\alpha + M_y\cot^2\alpha}\right|\right)$$

If $M_\alpha^+ > 0$

$$M_\alpha^+ = 0$$

$$M_x^+ = M_x + 2\,M_{xy}\cot\alpha + M_y\cot^2\alpha - \left|\frac{(M_{xy} + M_y\cot\alpha)^2}{M_y}\right|$$

In-plane forces:

Reinforcement:

Orthogonal (X,Y)

$$F_x^+ = F_x + |F_{xy}|$$

$$F_y^+ = F_y + |F_{xy}|$$

If $F_x^+ < 0$

$$F_x^+ = 0$$

$$F_y^+ = F_y + \left|\frac{F_{xy}^2}{F_x}\right|$$

If $F_y^+ < 0$

$$F_y^+ = 0$$

$$F_x^+ = F_x + \left|\frac{F_{xy}^2}{F_y}\right|$$

Skew (X,α)

$$F_x^+ = F_x + 2F_{xy}\cot\alpha + F_y\cot^2\alpha + \left|\frac{F_{xy} + F_y\cot\alpha}{\sin\alpha}\right|$$

$$F_\alpha^+ = \frac{F_y}{\sin^2\alpha} + \left|\frac{F_{xy} + F_y\cot\alpha}{F\sin\alpha}\right|$$

If $F_x^+ < 0$

$$F_x^+ = 0$$

$$F_\alpha^+ = \frac{1}{\sin^2\alpha}\left(F_y + \left|\frac{(F_{xy} + F_y\cot\alpha)^2}{F_x + 2F_{xy}\cot\alpha + F_y\cot^2\alpha}\right|\right)$$

If $F_\alpha^+ < 0$

$$F_\alpha^+ = 0$$

$$F_x^+ = F_x + 2F_{xy}\cot\alpha + F_y\cot^2\alpha + \left|\frac{(F_{xy} + F_y\cot\alpha)^2}{F_y}\right|$$

Combined forces:

From examination of the equations above, it is obvious that for the general case

Mx ± |Mxy| and **Fx ± |Fxy|**

where the worst case is used for each calculation.

When reinforcement is calculated for combined forces, four different combinations of moment and in-plane forces must be checked to determine the worst condition, i.e.

Mx ± |Mxy| combined with **Fx ± |Fxy|**

For example, bottom X reinforcement:

- Moment only:
  Mx + |Mxy| will always be the governing case
- Moment and in-plane force:
  For a small negative moment and large tension force, tension reinforcement will be required. **Mx - |Mxy|** will reduce the design negative moment and hence minimize the bottom compression stress.. Therefore, **Mx - |Mxy|** (min. compression) combined with **Fx + |Fxy|** (max. tension) will be the worst case condition for maximum bottom tension.

Part VII

Steel design

7 Steel design

For general information on the steel design module, refer to Steel Design - General 691.

Section ta...	Specify general section table 696 information. • define and revise section groups 696 • define and revise built-up sections 699 • replace model table
Defaults	Revise the default parameter 702 values displayed at the bottom of the screen.
Sections	For a specified member, limit the section selection 711 to • a section type • a section group • a specified section
Identical s...	Specify that the identical section 713 must be selected for a series of members.
Major/minor	Define the orientation of the section major/minor 715 axes relative to the section local x2/x3 axes, i.e. to rotate the section by 90° about the section x1 axis.
Parameters	Specify different parameters for individual beams 717 (if a parameter is not defined for a specific beam, the default parameter is used).
Supports	Define intermediate supports 729 along a member to allow the program to calculate effective lengths. Intermediate supports may be defined for both bending and compression design: • lateral-torsional buckling: define the location of intermediate supports on the top and/or bottom faces of the section. • buckling: define intermediate supports about the major and/or minor axes.
End condit...	Define the location of cantilevers, and: BS5950, IS:800 - Define the "Conditions at Support" at the ends of beams and cantilevers. AISC - Specify main/secondary members. Eurocode 3 - Define the effective length factor for LTB.
Combine ...	Use this option to combine members 738 into a single design unit.
Copy	Copy 741 parameters, intermediate supports and combined beams from one member to another.
Compute	Start the design 744 and section selection process, according to the parameters specified in the previous options.
Submodel	• select a submodel 742 to display • specify that the same member in two or more instances of a submodel must be identical (similar to ***Same section***)

From the menu bar:

File Zoom Rotate Display Draw REmove ReSults Data Tables Loads Sway Help

File 746
Leave the steel design module. Note that the ***STRAP*** geometry may be updated with the selected sections.

Display 750
Display input data ***graphically***.

Results 756

- display and/or print the ***tabular*** results
- graphically display the load/capacity ratios

Data Tables 763

Display tables showing the input data (design parameters, constraints and supports).

Loads 768

- axial load reduction: specify load cases as "live" and define reduction factor
- composite beams: specify load cases as applied to steel or to composite beam
- deflections: define different allowable deflections for different load combinations

Sway 772

Select new sections that limit the sway/drift at specified nodes to user-defined values.

7.1 General

The ***STRAP*** Steel Post-processor is a program for the design of structural steel buildings:

- The program selects for each member the lightest section which meets the Code requirements. The sections are chosen from a specified table. Section constraints, design data and intermediate supports may be defined.
- Alternatively, the program may be instructed to check the suitability of a section specified by the user.
- A concrete topping may be specified for the steel beam; in such cases the program selects the steel beam based on composite section design.
- The program automatically searches for the critical load combination, and checks the section for bending moments, lateral-torsional buckling, shear, axial forces and combined stresses as well as deflections and slenderness.

The program may be instructed to select sections according to the methods outlined in any one of the following structural steel design codes. For more information, refer to:

- British Standard BS5950:
  - Part 1 : 2000, "Structural Use of Steelwork in Buildings".
  - Part 3, Section 3.1 - "Design of Simple and Continuous Composite Beams"
- Eurocode:
  - Eurocode 3 - Design of Steel Structures - Part 1 , 2005
  - Eurocode 4 - Design of Composite Steel and Concrete Structures - Part 1, 1990
- AISC - LRFD & ASD - 2016
- CSA/CAN S16-14 - Limit States Design of Steel Structures - 2014.
- AASHTO - LRFD Bridge Design Specifications - 2012
  AASHTO - Standard Specifications for Highway Bridges (ASD) - 1996
- SABS 0162-2: 1993
- India:
  - IS:800 - Code of Practice for General Construction in Steel - 1984
  - IS:800 - Code of Practice for General Construction in Steel - 2007 (LRFD)
  - IS:802 - Use of Structural Steel in Overhead Transmission Line Towers (1995)
- NBr 8800
- GBJ17-88 (China)
- AS4100 (Australia)
-

Cold formed:

- AISI Specification for the Design of Cold-Formed Steel Structural Members:
  The 2016 Edition with Supplement 1 of the AISI-S100 Standard “North American Specification for the Design of Cold-Formed Steel Structural Members” .
- CSA S136-1994
- Eurocode 3 - Part 1.3- 2009 - “Supplementary Rules for Cold-formed Thin Gauge Members & Sheeting".
- BS5950 - Part 5 - 1998 - "Code of practice for design of cold-formed thin gauge sections"
- AZ/NZS 4600:2005 - "Cold-formed steel structures"

The program strives to design the lightest structure possible; the section chosen is the one with the least self-weight that satisfies all of the design criteria for all loading combinations and meets the Code requirements.

For all Limit States Codes (all Codes except AISC/AASHTO/AISI - ASD and IS:800):

The factored combinations may be defined in *STRAP* loading or in the "Combinations" option after the solution. The choice does not affect any of the design calculations except for deflections which are based on service loads. The postprocessor assumes that load ***cases*** are unfactored, i.e. the combinations were defined after the solution. If the factored combinations were defined in Loading, the deflection results for the load cases will also be factored. The allowable deflection limit should then be increased proportionally. Refer to Allowable deflection 702 for an example.

Steel section tables and selection 692
Creating a steel structure from the STRAP model 693
Design example 778

7.1.1 Sections

The program contains complete section tables - the "master" table.

- British sections (CONSTRADO tables)
- American sections (ASTM)
- European sections (Euronorm)
- Cold-formed sections
- Canadian
- South African
- Indian

Cold-formed sections may be added to the cold-formed table; refer to Steel section table.

In addition:

- A "***user***" steel table may be created. This table can contain sections from any or all of the tables above, or user-defined rolled/cold-formed sections. Refer to Steel section table 696 for instructions.
- additional "built-up" sections" 699 may be defined by specifying dimensions; the program assumes that these sections are welded shapes.
- Combined 697 (rolled) sections may be defined (2L, I+[,][, etc).

The following terminology is used throughout the program:

- **Master table**: The initial complete table of steel sections (British, American, European, etc. **or** User).
- **Model table**: The section table for the current model, containing the sections from the selected master table(s).
- **Section type**: Section classification according to shape, e.g. CHANNEL, RHS, Z+LIPS, etc.
- **Group**: A user-defined list of sections which may contain sections from several types.

The program either selects a suitable section from a list or checks a specified section.

In the following cases, the program by default checks the section defined in ***STRAP*** geometry:

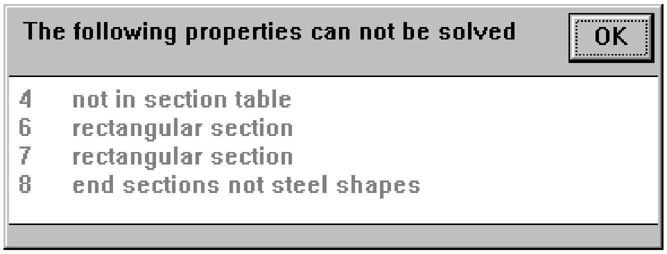
- if the section properties of the member were defined as a steel section using the **Steel table** option.
- "Combined" sections, defined in ***STRAP*** geometry or in the steel design module
- if the section properties of the member were defined using the **Define the section dimensions** option, the program assumes that the member has a Built-up welded section with the same dimensions.
- if a "tapered section" was defined in ***STRAP*** and the properties at both ends of the member were defined using the **Define the section dimensions** option, the program checks the tapered section.

For all other ***STRAP*** properties, the program ignores the ***STRAP*** properties and must be instructed to select the lightest suitable section from a user-defined list or to check a specified section. The cases are:

- "dimension" section types - "Rectangular" and unsymmetric "[shape", "I shape" and "T shape" sections.
- properties defined with the section constants "**A=,I=**" option

- tapered sections defined with properties other than acceptable "built-up" shapes.

A warning is displayed at the start of the program. For example:



When selecting a suitable section, the program checks sections from a specified list, beginning from the lightest one, until it finds the first adequate section. If the program had to search the complete "master" table when designing each member in the model, the selection process would be very time consuming.

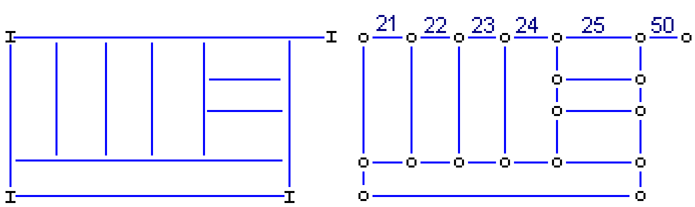
To limit the number of sections that the program is allowed to check, instruct the program to select the section from a single type or from a predefined group of sections only.

Refer to Sections 711 for complete details.

7.1.2 Defining a steel structure from a STRAP model

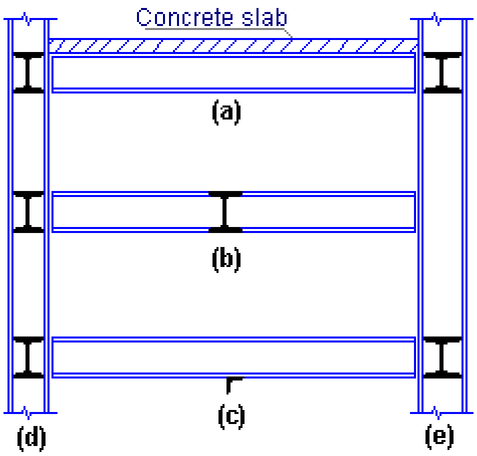
Member selection is automatic; the program designs all beams in sequence without prompting for information. Therefore, all parameters relevant to the design of the members as required by the Code must be specified before member selection begins. In many cases the model geometry as defined in ***STRAP*** does not provide sufficient information for the Code to carry out an accurate design.

For example, Figure (a) shows a typical steel floor plan. Figure (b) shows the same floor as analysed in ***STRAP***. It is obvious that the program is unable to determine which ***STRAP*** members form continuous beams, i.e. which chains of members must be designed as a single unit. In this example members 21-22-23-24-25-50 form a single beam.



(a) (b)

Another important item that must be defined is the location of "intermediate supports". The support locations are required by the program to automatically determine the unsupported length for beams and the design length for columns.



- beam (a) has a continuous support on its upper flange, and is unsupported along the entire bottom flange.
- beam (b) has a single support on its upper and lower flanges.
- beam (c) is unsupported on its upper flange and has a single support on its lower flange.

- the section required for each of the beams (a),(b),(c) will be different if the upper flange is entirely in compression. This support data is not available from the ***STRAP*** geometry and must be defined here.
- similarly, column (e) has an unsupported length about its minor axis double that of column (d).

Other data items required for design are:
- allowable maximum deflection
- allowable maximum slenderness (tension and compression)
- beam end support conditions
- column effective length factors
- section orientation
- steel grade

Additional design constraints for section selection may also be specified:
- minimum and/or maximum section dimensions for each member.
- a series of members may be defined as "identical", i.e. the same section will be selected for all members in the series.

7.1.3 Design assumptions

- ***AISC - LRFD & ASD***
  Beams/columns
  Composite beams
  Composite columns
- ***AASHTO - LRFD***
  Beams/columns
  Composite beams
- ***AASHTO - ASD***
  Beams/columns
  Composite beams
- ***BS5950***
  Beams/columns
  Composite beams
  Composite columns
- ***Eurocode***
  Beams/columns (EC3)
  Composite beams (EC4)
- ***IS800-2007***
  Beams/columns
  Composite beams
  Composite columns
- ***AS4100***
  Beams/column
  Composite beams
  Composite columns
- IS802
- SABS 0162-2

Cold-formed Codes:

- AISI - ASD & LRFD
- BS5950 - Part 5
- Eurocode 3

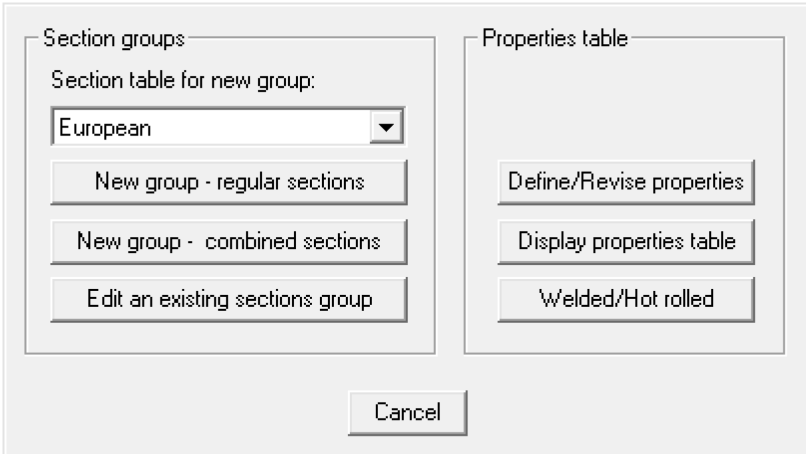
Composite columns (EC4)

- ***CSA S16-14***
  Beams/columns
  Composite beams
  Composite columns
- ***NBr 6800***
  Beams/columns
  Composite beams
  Composite columns
-

- CSA S136

- Other codes

7.2 Section table



7.2.1 Section groups

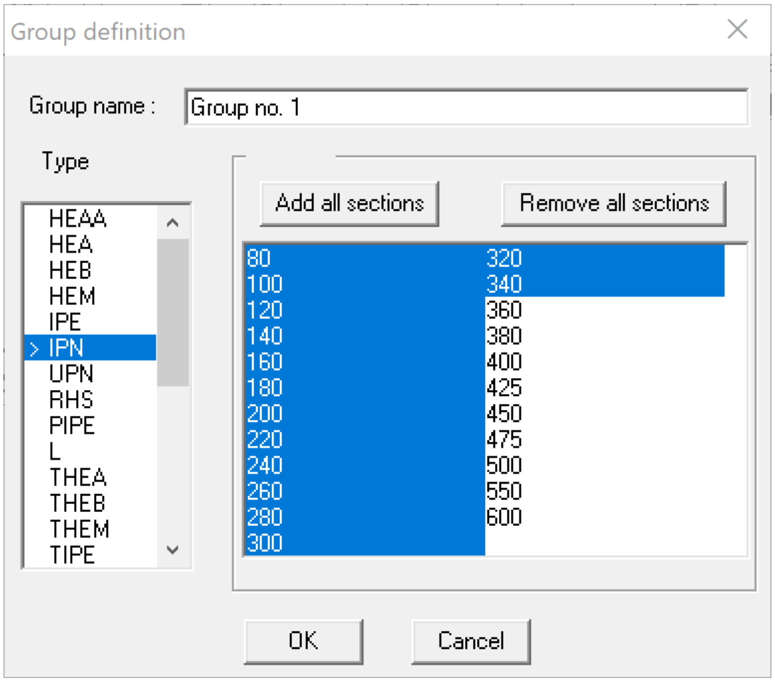
A group is a subset of the current model table, and the program can be instructed to select the section for a particular member from a specified group.

Examples: groups may be defined to contain:

- economical sections only
- available sections only
- all angles and channels with a limited range of flange widths
- etc.

It is recommended that groups be defined for every model because the time required for member selection is reduced as the number of possible sections for each member is reduced.

The program displays a list of the section types and section names that are in the model table. The names of section types currently included in the group appear highlighted. For example:



Then select the sections that are to be included in the group:

- click one of the section types in the left list box; the sections in this type are displayed in the right list box.
- only highlighted sections are included in the group; click the section name to include it in the group or to remove it from the group
- use the horizontal scroll bar at the bottom of the list box to display the continuation of the section list.
- similarly select sections for all section types
- click OK to end the group definition.

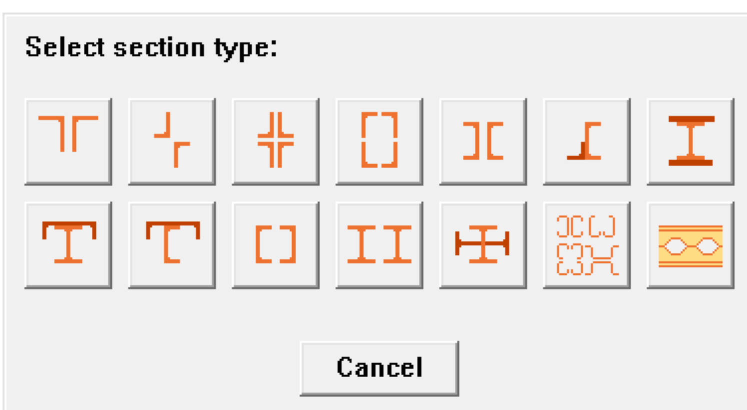
Note:

- a section group may contain a maximum of 640 sections
- built-up sections cannot be included in a group.

7.2.2 Combined sections group

Define a group of combined sections.

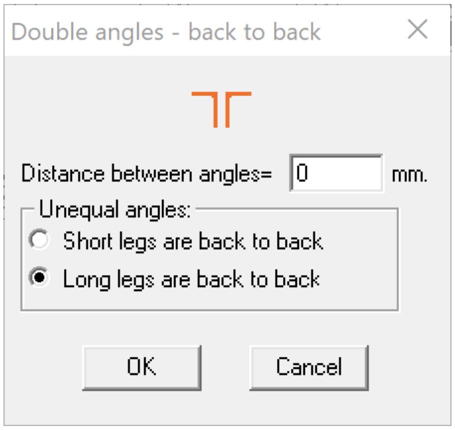
- Select a section type from the list in the menu:



- specify section types, spacing, orientation, etc. The types may be divided into several categories

Identical sections are combined:

For example, double angles, back-to back:

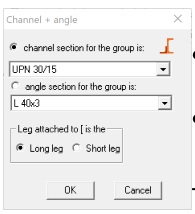


- the program displays a list of angles; highlight sections in the list to select them.

The group that is created contains double angle sections made from the single angles selected from the master list, ***all with the same angle leg attachment option (long/short) and all with the same spacing***.

Different sections are combined:

For example, channel and angle:



- select either a specific channel section or a specific angle section.

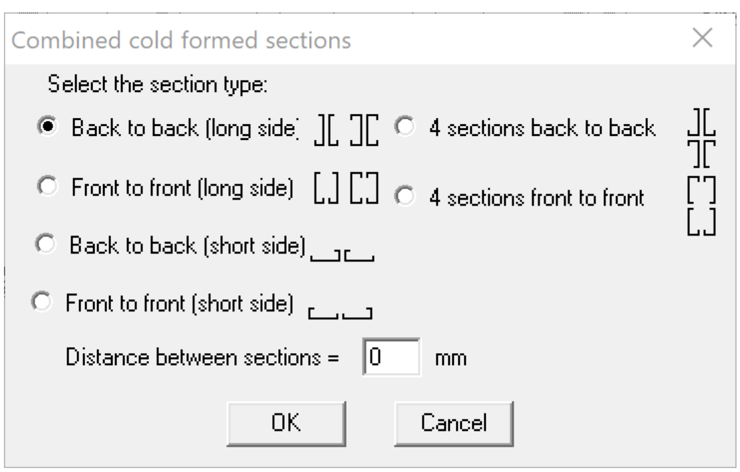
- if you select a specific channel section the program displays a list of angles; highlight sections in the list to select them.

The group that is created contains the selected channel section together with all angles selected from the master list, ***all with the same angle leg attachment option (long/short)***.

- similarly, if you select a specific angle section the program displays a list of channels; highlight sections in the list to select them.

Combined cold-formed sections:

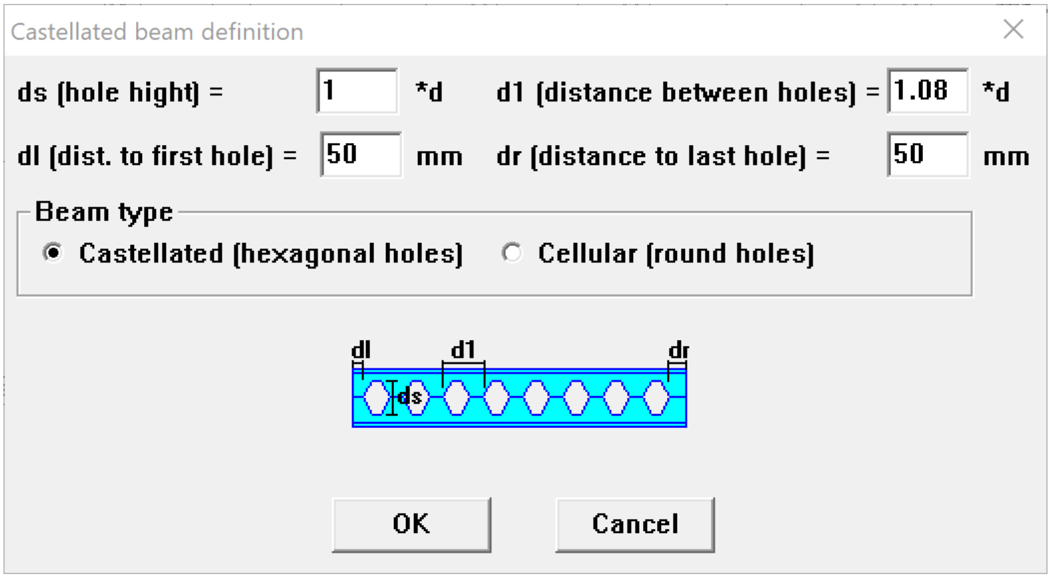
- Select the cold-formed section configuration:



- select types and sections (more than one type may be added to the group)

Castellated sections:

- specify the hole type (castellated/cellular) and the hole dimensions (vary according to the beam height, **d**)



- select I-types and sections (more than one type may be added to the group)

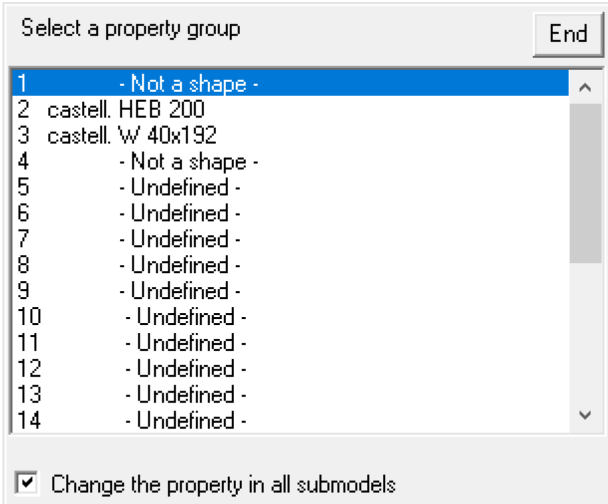
7.2.3 Built-up sections

Use this option to define:

- ***Single built-up sections.***
  I, U, T, L, round/hollow tubes or tapered sections may be defined by entering the section dimensions; The program assumes that these defined sections are ***welded*** with reference to various clauses in the Code. All required properties are calculated automatically by the program from the dimensions. No rounded corners are assumed.

  Note:
  - Built-up sections cannot be selected by the program during automatic design; the user may request that such a section be specifically checked by the program (Refer to Sections 711).
  - The section defined here ***replaces*** the section defined in geometry, therefore the Section - Check section from geometry 711 option will not work if a new built-up section is defined here.

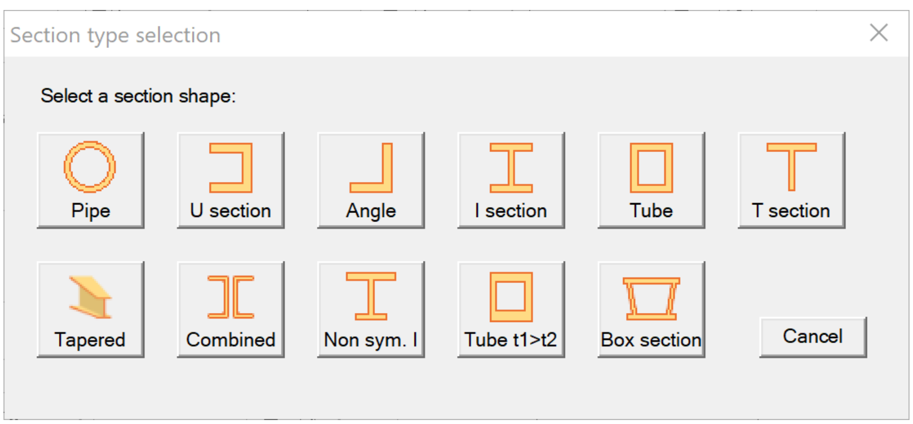
Select the property to be defined:



Change the property in all submodels

If the current model contains submodels they may have their own property tables that are different from the Main model property table (refer to Submodel - new for more details). If, for example, you revise property 7 and the ☑ is checked, the same section is assigned to property 7 in ***all*** of the submodel property tables.

Enter the section number; to define a new section, select an "Undefined" section. A list of the possible section types is displayed:



Select a type.

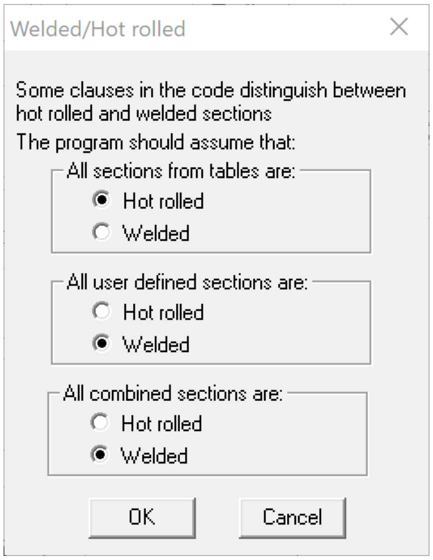
Note:

- for welded single sections, define the section dimensions according to the diagram displayed.
- for combined sections, select the shape and define additional properties (e.g. spacing)
- **Unsym I** has different flange thicknesses
- **Unsym Tube** may have different thicknesses on all four sides
- **Box section** defines typical sections for box-girder bridges.
- **Tapered** sections can be defined only with built-up sections at both ends. Tapered sections with shapes at either end must be defined in geometry.
- The default section orientation is identical to that explained in Major / minor 715.
- if the default dimension in ***STRAP*** are "feet" or "inch", define the built-up section dimensions in **inches**. For all other default length dimensions, define the built-up section dimensions in **millimeters**.

7.2.4 Welded/hot rolled

The program, by default, assumes that all sections from the tables are hot-rolled and that all user-defined sections are welded and designs them according to the relevant code provisions.

Use this option to change the default assumption:



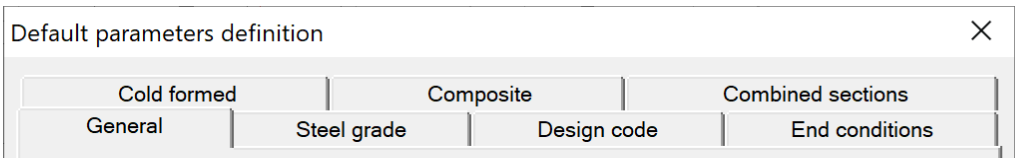
7.3 Default parameters

The default parameters are design codes (hot-rolled and cold-formed), maximum slenderness (compression and tension members), maximum deflection, steel grade, net tension area factor, end connection types (BS5950, IS:800 only) and building type (AISC, CSA and IS:800).

The default parameters are used when design parameters have not been defined for individual members using the Parameters 717 option.

Note:

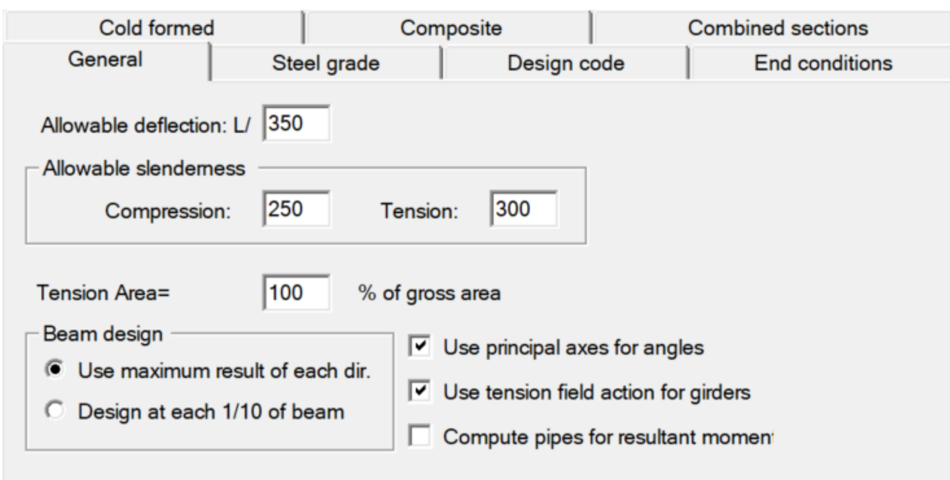
- the effective length factors for columns, kx and ky, are assumed by default to be 1.0. This default value cannot be revised. To specify a different value for selected members, refer to Parameters 717.
- the maximum allowable tension slenderness cannot be modified for individual members.



Note:

- the current default parameter values are displayed at the bottom of the screen.
- default parameters are always revised for both the main model and all submodels.

7.3.1 General



Allowable deflection

The maximum allowable deflection expressed as L/def. e.g. to specify a maximum deflection of L/300, enter "300".

Note that deflections must be calculated from ***service*** loads. As the program assumes that the ***STRAP*** loading cases contain service loads, the program ***sets all the load factors equal to 1.00*** when calculating the deflection for each load combination.

If the factored combinations were defined in the ***STRAP*** load definition module and not in the Combinations option, the ***STRAP*** deflections will also be factored, so the allowable deflection should be increased by 1.5.

Allowable slenderness

The maximum Kl/r value allowed (about any axis) for a member. Different values may be defined for tension and compression.

Tension area

A factor to calculate the net area for tension members. For example, if $A_n = 0.9A_g$, enter 90%.

Note:
- IS:800 - the effective section area computed by the program is multiplied by this factor
- This option is applicable for hot-rolled sections only. For cold-formed sections, the net area is calculated by deducting the diameter of the holes in the web (defined in Parameters 717).

Beam design

The program calculates various capacities for combined forces - major and minor axis moments, moment and axial load, moment and shear.

Two options are available:

◉ **Use maximum result in each direction**
The program combines the maximum result from each type, even though they may not be at the same location. For example,in a typical beam, the maximum shear is at the support while the maximum moment is at the mid-span. This approach is conservative, but faster. The program uses the worst classification for non-symmetric sections.

◉ **Design at 1/10 of beam**
The program calculates the combined forces at 11 points along the length of the beam, using the actual forces at each point, and uses the worst case for design. This approach is more accurate, but slower. For non-symmetric sections, the program calculates the classification at each point according to the sign of the moment.

Principal axes

Two options are available for the design of angles, and user-defined doubly unsymmetric sections (defined in the Section editor - *CROSEC* utility):

☑ The program uses the principal axis properties, **Iu**, **Iv**, etc. - for designing these sections
☐ The program uses the major/minor axis properties, **Ix**, **Iy**, etc. - for designing these sections

(In "Parameters - allowable 717", The initial status of the checkbox - ☑ - indicates '**No change**' for the selected members).

Note:
- this option applies only to LTB, bending and and deflection calculations (but not axial compression, where principal axes are always considered in the calculation).

Tension field

The program calculates vertical stiffeners for slender webs. Use the following design methods:

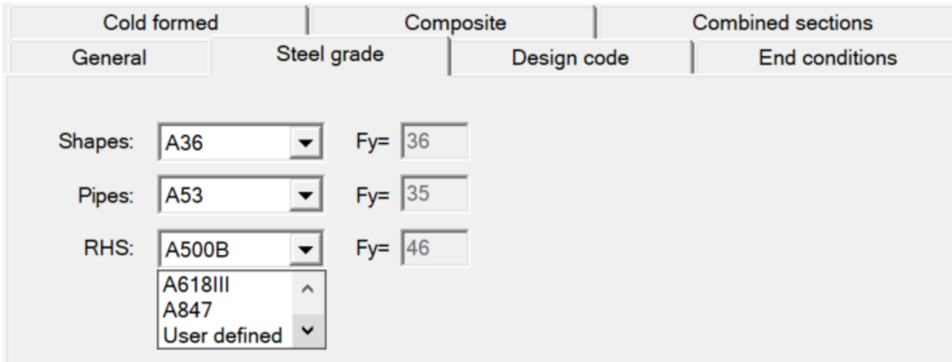
☐ the program uses the regular method only.
☑ the program uses the tension field method to reduce the shear.

Compute pipes for result moment

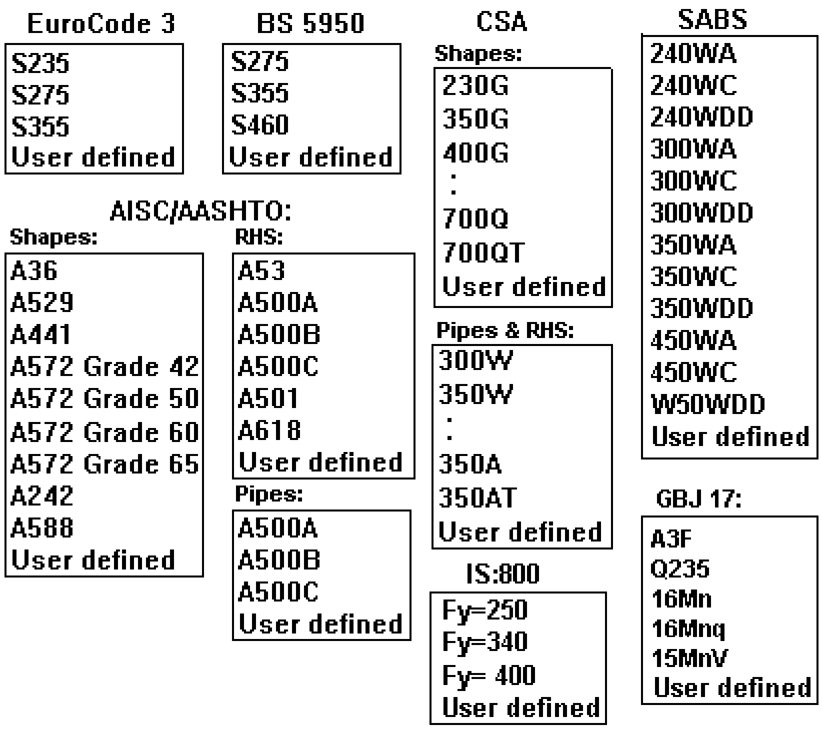
For round pipe sections:

☐ compute separately for Mx and My
☑ compute for a single resultant moment **$M=v(Mx^2+My^2)$**

7.3.2 Steel grade



Select the steel grade or define your own:



User defined: define the yield strength.

- For AISC/AASHTO Codes: F_y must be entered in **ksi**.
- For other codes: p_y must be entered in **N/mm²**.

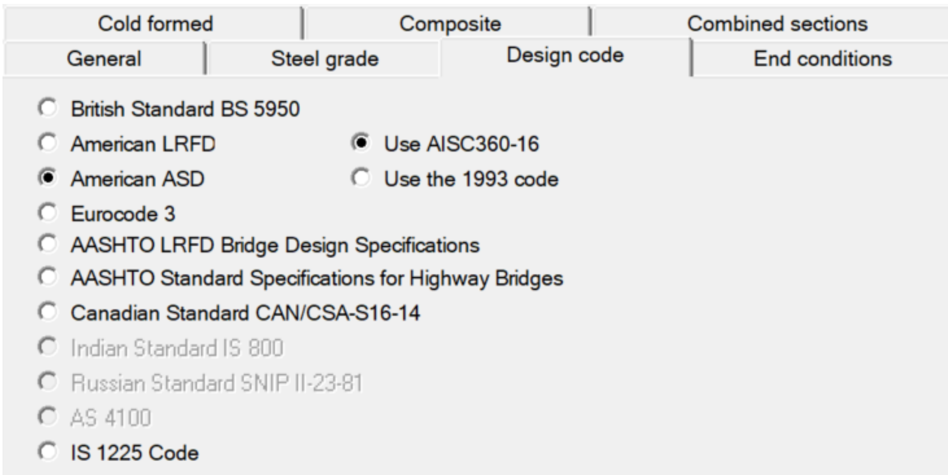
Additional options

CSA, Hollow structural sections may be designated as CSA Standard G40.20, Class C or Class H.

EC3: Hollow structural sections may be designated as Hot-rolled or cold formed

7.3.3 Design code

Use this option to specify the design codes for hot-rolled and built-up sections. The Codes available are:



Indian Standard:

- set the first checkbox to ☐ to use the IS800:1984 (ASD) code.
- set the second checkbox to ☑ to calculate axial strength according to IS802 (all other strength values are calculated according to the selected IS:800 Code).

Eurocode 3:

The design may be carried out with the modifications specified in the UK National Annex; refer to Setup - Code factors.

Note:

- refer to Design assumptions for detailed explanations on the Code equations used by the program.
- all code modules are purchased separately ; contact your ***STRAP*** dealer for further information.

7.3.4 End conditions

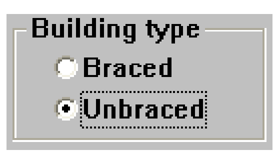
Define the default end conditions of beams:

Values selected in the 

option override the default values.

AISC / AASHTO / CSA / IS:800-84

The building must be classified as ***Braced*** or ***Unbraced***.



This parameter is required for:

AISC-ASD: **C_b** (Section F - Moment - non-compact): **C_b** = 1.0 for braced buildings
C_m (Section H - Combined Stresses): **C_m** = 0.85 for unbraced buildings

AISC-LRFD: Equation C1-1: braced frames: **$M_u = B_1 M_{nt}$**
unbraced frames: **$M_u = B_2 M_{lt}$**

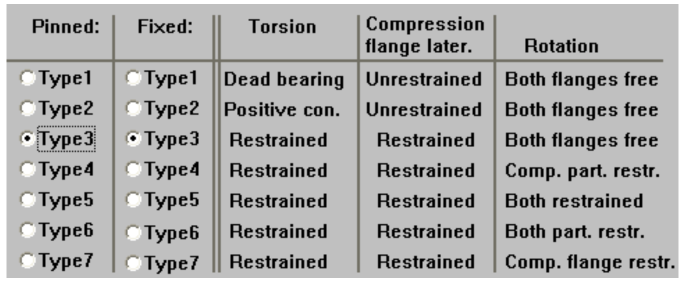
AASHTO-ASD **C_b** (Section 10.32 and Table 10.32.1A): same assumption as for AISC-ASD
C_m (Section 10.36 - Combined Stresses): Value from Table 10.36a).

AASHTO-LRFD: Equation 4.5.3.2.2b-1: braced frames: **$M_c = \delta_b M_{2b}$**

	unbraced frames:	**Mc =** δ**sM2s**
CSA S16-14:	**U1x** and **U1y**, according to Clause 13.8.2 (a)(b)(c)	
IS:800-84:	**Cm** = 0.85 for unbraced buildings (7.1.1)	
	Cm = 1.00 for braced buildings	

BS5950 / IS800-2007

Define the "Conditions at Support" referred to in BS5950 - Tables 13 or IS800 - Table 15 in order to calculate the effective length Le for lateral-torsional buckling.



Pinned:	Fixed:	Torsion	Compression flange later.	Rotation
Type1	Type1	Dead bearing	Unrestrained	Both flanges free
Type2	Type2	Positive con.	Unrestrained	Both flanges free
Type3	Type3	Restrained	Restrained	Both flanges free
Type4	Type4	Restrained	Restrained	Comp. part. restr.
Type5	Type5	Restrained	Restrained	Both restrained
Type6	Type6	Restrained	Restrained	Both part. restr.
Type7	Type7	Restrained	Restrained	Comp. flange restr.

Note:

- values are for fixed/pinned beams only; for cantilevers, define the restraint conditions at both ends using the 

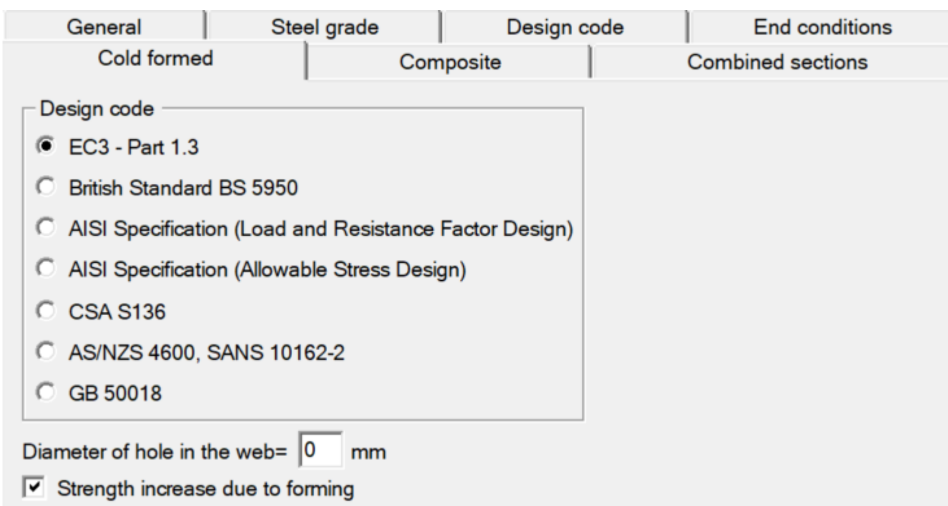
option

Eurocode 3

Define the parameters in the 

option

7.3.5 Cold formed



Design code

Select one of the Cold-formed design codes displayed in the menu.

Note:

- the selected cold-formed Code applies only to members limited to a cold-formed type or section in the Sections 711 option.
- refer to Design assumptions for detailed explanations on the Code equations used by the program.
- All code modules are purchased separately ; contact your ***STRAP*** dealer for further information.

Diameter of hole

Specify the diameter of the web holes; the program calculates the reduced section properties according to AISI - Section B2.2.

Note:
- the area is removed from ***each*** web element in the section
- all stiffened elements are assumed to be webs

For example: the specified area is removed ***twice*** in back-to-back channels.

Strength increase

☑ The program calculates the increase in strength due to cold forming according to Section A7.2 in the AISI Code; F_{ya} is calculated according to Eq. A7.2-1 and is substituted for F_y when calculating axial capacity, flexural capacity and for the combined stress checks.

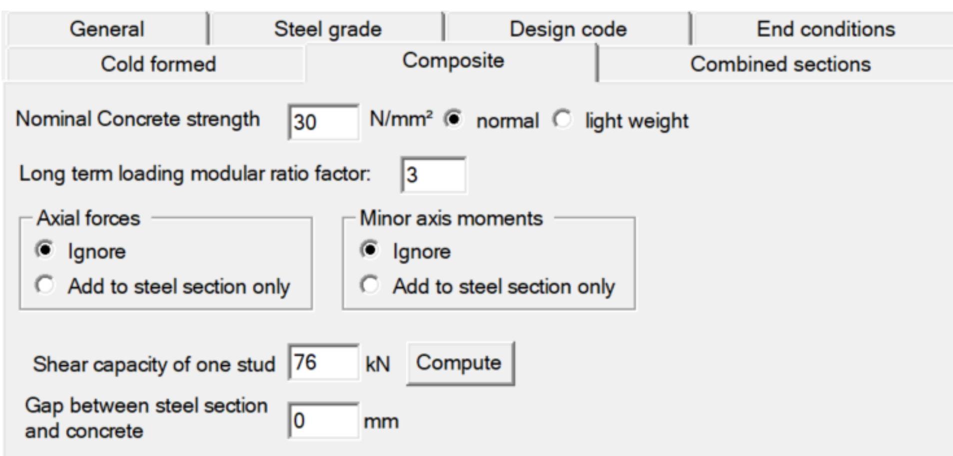
☐ Strength increase from cold forming is not calculated.

Note that this option is specified for the entire model and is not a parameter for specific members.

7.3.6 Composite

Specify the default parameters and dimensions for composite beams.

To design a composite beam:
- specify the default parameters (Composite and/or Composite - additional)
- specify Parameters for individual beams if necessary (Composite and/or Composite - additional)
- specify ☑ **Composite** in the Sections[711] option for all composite beams



Concrete strength

Enter the ***nominal*** concrete strength used in the current design code; the program automatically reduces the strength by the appropriate factors.

Normal / lightweight

The calculation of the modulus-of-elasticity, $\mathbf{E_c}$, (and hence the modular ratio, **n**) is revised for lightweight, as follows:

AISC-ASD:AISC-LRFD: AASHTO-LRFD: AASHTO-ASD	normal: w = 145 psf lightweight: w = 110 psf $E_c = w^{1.5} 33\sqrt{f'_c}$ (ACI 318 - 8.5.1) $\mathbf{E_c}$ and **n** are revised only for the calculation of deflections and 2-2)
CSA S16:	lightweight: γ_c = 1750 kg/m^3

$$E_c = (3300\sqrt{f'_c} + 6900)\left(\frac{\gamma_c}{2300}\right)^{1.5}$$ } (CSA A23.3 - Eq. 8-6)

normal: $E_c = 4500\sqrt{f'_c}$

BS5950: normal: α = 6/18 (BS5950 - part 3 - Table 1)
lightweight: α = 10/25

IS:800: modular ratio **m = Es/Ec**
Es = 200000
$E_c = 5000\sqrt{f_{ck}}$ (N/mm^2)

Long-term modular ratio

The program converts the concrete slab to an equivalent steel section by reducing the slab width according to the modular ratio (Es/Eb).

Many codes specify a different modular ratio for transient (short-term) and long-term loading; specify the relationship between the long-term and short-term modular ratios:

- BS5950 - Part 3
  Refer to Section 4.1, Table 1
- AASHTO - LRFD
  Refer to Section 6.10.5.1.1b
- AASHTO - ASD
  Refer to Section 10.38.1.4

All other Codes do not refer to this factor. However, the program will use the factor for all Codes if long-term and short-term loads are specified.

Axial forces

Select one of the following options for design of axial forces in composite sections:

◉ **Ignore**
The program ignores completely the axial forces present in all load cases.

◉ **Add to steel section only**
The program assumes that the entire axial force is taken by the steel section and that no axial force is present in the concrete slab.

Minor axis moments

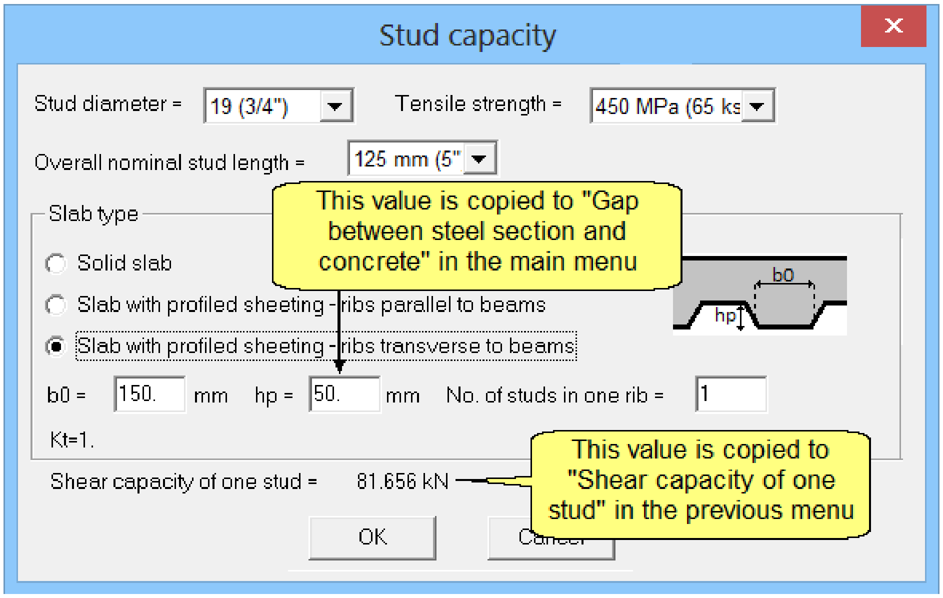
Select one of the following options for design of minor axis moments in composite sections:

◉ **Ignore**
The program ignores completely the minor axis moments present in all load cases.

◉ **Add to steel section only**
The program assumes that the entire minor axis moment is taken by the steel section and that the slab does not contribute the minor axis strength.

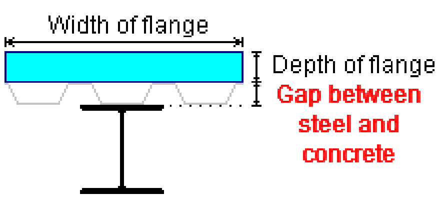
Stud shear capacity

Specify the shear capacity of a single stud or click [Compute] to calculate the capacity according to the Code. For example, EC3:



Gap

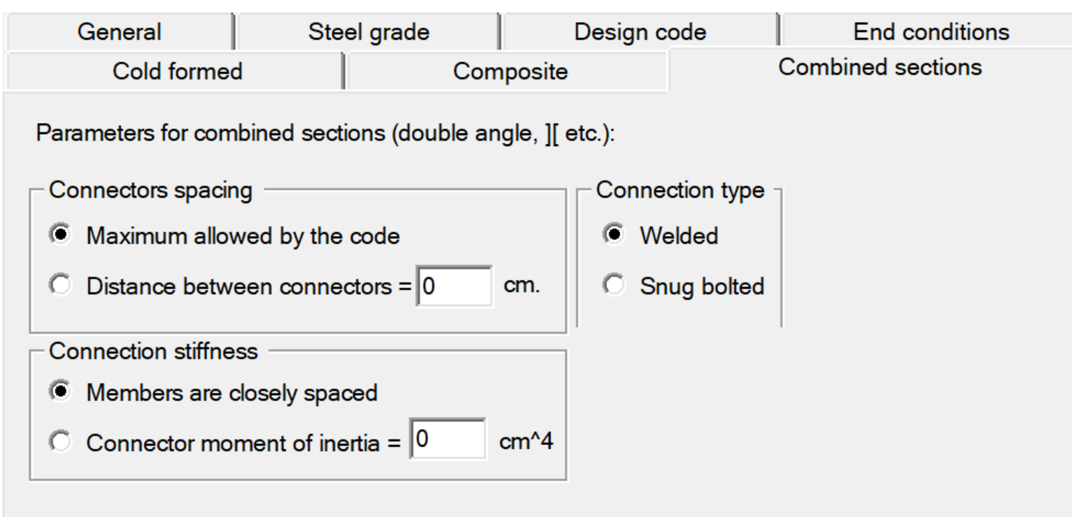
Specify the gap between the steel section and concrete:



If you select Compute for the stud capacity, the value of "Gap between steel and concrete" is automatically copied from the "h_p" value used to calculate the capacity.

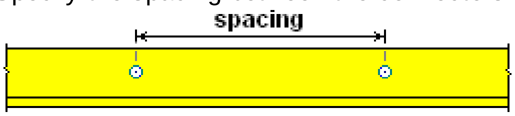
7.3.7 Combined section

Define additional default data for members fabricated from several sections. The program uses this data when calculating the slenderness of the section.



Connectors spacing

Specify the spacing between the connectors:



Maximum allowed by the code
The program calculates the maximum spacing allowed by the Code and uses this value.

Distance =
Specify any other distance

Note:
- the program uses this value to calculate the axial buckling of each component of the combined section between the connector points. This calculation is possible only if the combined sections were defined using the "Properties - Combined - cold-formed" option in *STRAP* Geometry. The program cannot identify the component parts of the combined section if the section was defined using *CROSEC* or the "Utilities - Create/edit a Steel section table" option in the STRAP main menu.

Connection stiffness

For some codes, e.g. Eurocode - select:

Members are closely spaced
Referring to Code Figure 6.12, these are standard built-up section (not battened) such as double angles back-to-back, etc.

Connector moment-of-inertia =
For battened sections, specify **Ib** = the in-plane second moment of area of one batten. Refer to Section 6.4.3 in the Code

Connection type

For American codes only:
- specify the default connection type, **Welded** or **Snug bolted**.
- for members that are not in contact, always select **Snug bolted**.

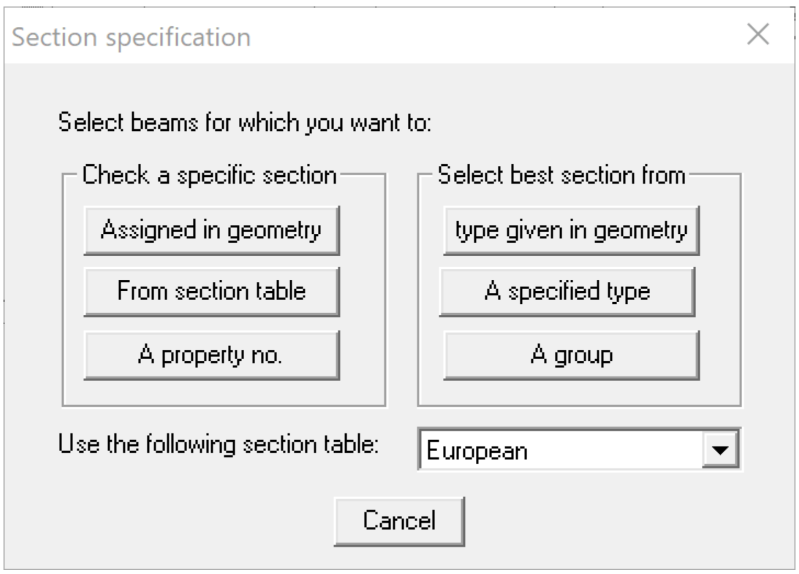
7.4 Sections

For each member, there are two options -

- Select best section from - the program searches through a user-specified section list (not the entire steel table) in order to find the lightest adequate section, or -
- Check a specific section - the program checks a specified section (not rolled, cold-formed or built-up).

The beam may also be designed as a:

- composite beam 707 (steel section + concrete topping)
- composite column 725 (encased steel section or filled section)
- steel joist 745 (American steel table only)



A different option may be selected for each member; after the option is chosen, select beams using the standard Beam Selection option.

Assigned in geometry (Check for specific section)

Check the section currently in the ***STRAP*** geometry file for the selected members. This is the default option.

Note:

- the geometry section property must be a steel section included in the model table.
- for built-up sections defined in Geometry:
  - the program checks the section for the property group currently listed in *STRAP* geometry.
  - if the dimensions in this property were revised in the Section table 696 option (or if a new built-up section type was defined), the program will check the revised property, not the one currently in Geometry.

From section table (Check for specific section)

Instruct the program to check a specified section only. Select the section type and section name and assign this selection to specified beams using the standard Beam Selection option..

If a 'cold-formed' section is selected, the program automatically checks the section capacity according to the Cold-formed Code selected in Defaults 706.

Note: The available sections correspond to the selected table in Use the following section table.

A property no. *(Check for specific section)*

Instruct the program to check an existing built-up section only. Select the section from the list and assign this selection to specified beams using the standard Beam Selection option.

Note: built-up sections can only be checked; they cannot be added to groups.

Use geometry section type *(Select best section from)*

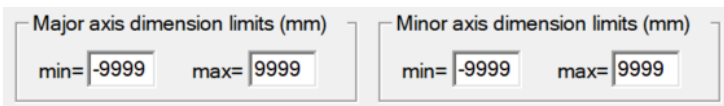
Similar to the previous option; the program checks all sections of the type specified in Geometry - Properties, e.g. if the property assigned to the beam in geometry was IPE240, the program searches for the optimum IPE section.

Note: The available sections correspond to the selected table in Use the following section table.

A specified type *(Select best section from)*

Instruct the program to check only the sections of one type in the model table.

- you may further limit the selection by specifying upper and lower limits for the section's Major/Minor dimensions -



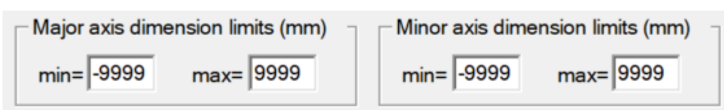
- If a 'cold-formed' section type is selected, the program automatically checks the section capacity according to the Cold-formed Code selected in Defaults 706.

Units:
inch - if the ***STRAP*** default length unit is "feet" or "inch"
mm - for all other ***STRAP*** default units.

A group *(Select best section from)*

Select one of the existing groups from the list.

- You may further limit the selection by specifying upper and lower limits for the section's Major/Minor dimensions -



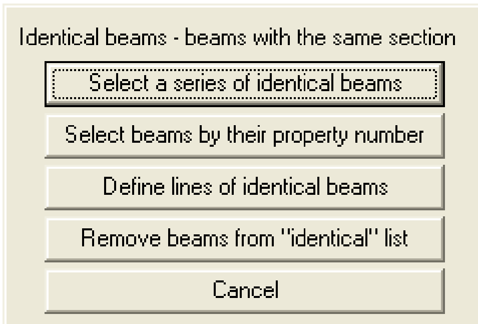
Use the following section table

Select a steel table (European, American, etc.) for the following options:

- From section table (Check for specific section).
- A specified type (Select best section from).

7.5 Identical section

Specify a list of members that must have identical sections.

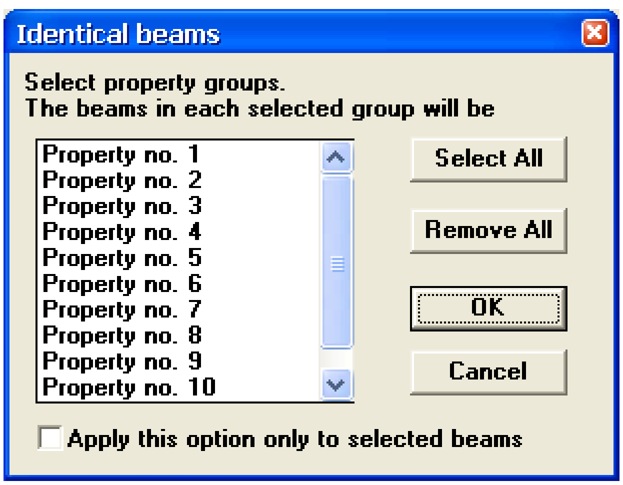


Select a series of identical beams

Use the standard beam selection option to select members which are to be "Identical".

Select beams by their property number

Specify the elements in 'same section' groups according to geometry property groups:



- Click property groups in the list box; each highlighted property number creates one 'same section' group, or -
- Click "**Select all**' to select all of the property groups in the list box.
- if you check the "**Apply this option ...**" checkbox, the program displays the standard beam selection option; select elements - only the specified elements that are also in highlighted geometry property groups are included in the 'same section' groups.

Note:
- If a member selected already belongs to an identical list, the two lists are combined.

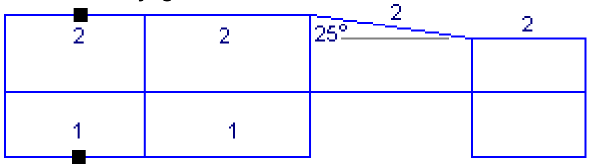
Define lines of identical beams

Select a member located on a line of continuous members; the program automatically creates an "identical" list that includes the specified member and all preceding and following ones.

Note that the program adds preceding/following members to the list only if the angle between the two adjacent members is less than 30°.

Select beams using the standard beam selection option.

Example:
The two beams highlighted with the rectangular ■ blip are selected; the two "identical" series are automatically generated.



Remove beams from "identical" list

Remove beams from identical groups using the standard beam selection option.

Note:
- If different section types 711 were specified for the various members in an identical list, the program uses the section type of the the ***lowest numbered member*** in the list for all of the members.
- If Check a specified section 711 was selected for a member in an identical list, the ***identical command is ignored***.
- If a member selected already belongs to an identical list, the two lists are combined.
- if a identical beams are defined in a submodel instance, the program automatically create the same identical beams in all other instances of the same submodel.
- When you **Compute** a beam in an identical group you must either select **All beams** or display all of the identical beams in order to calculate the section correctly.

7.6 Major/minor

The program assumes by default that the section orientation of a member is as **currently** defined in ***STRAP*** geometry. The orientation may be revised using this option:

- the section may be aligned so that its major axis resists M2 moments or M3 moments.
- for sections not symmetric about one axis, flange location may also be specified.

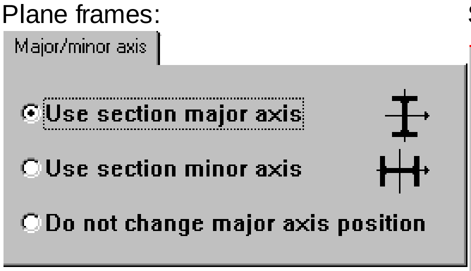
The default axis resisting moment is determined as follows:

STRAP geometry	I3 > I2	I2 > I3	I2 = I3
Plane frame	Major	Minor	Minor
Plane grid	Minor	Major	Major
Space frame	Major resists M3	Major resists M2	Major resists M3

Major / minor axis

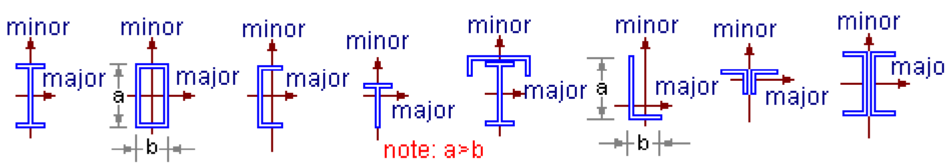
Select one of the following options:

Plane frames:



Space frames:

The major/minor axes for the standard steel sections are:

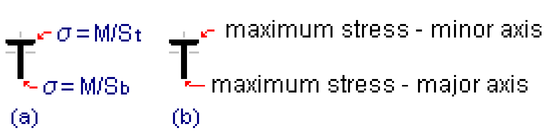


Note that these assumptions for orientation of major/minor axes remain unchanged even if the minor axis moment-of-inertia is greater than that of the major axis.

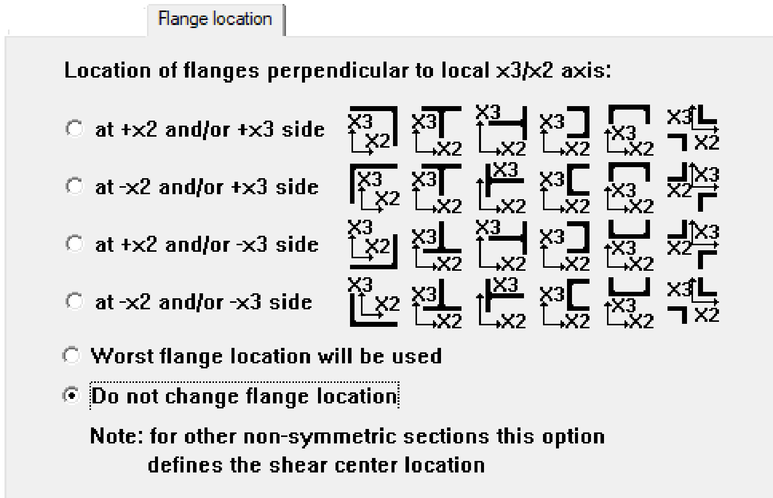
Flange location

The design of non-symmetric sections is dependent on the location of the flange:

- stresses at opposite faces of the section are not equal - Figure (a)
- maximum stresses for major/minor axes are not at the same point - Figure (b)



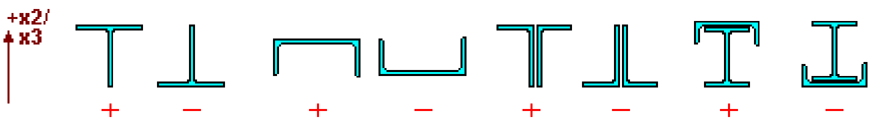
The program, by default, combines the maximum stresses, even if they are not at the same point (**Worst flange location** in the following menu). This option allows the exact orientation of the section to be specified, relative to the local axes:



When calculating lateral-torsional buckling, the program must know whether the flange is in tension or compression.

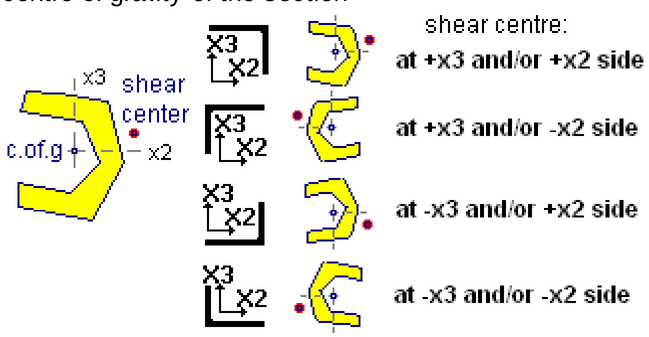
Worst
the program assumes that the flange is in tension for all moments, regardless of the sign.

+/- x2/x3 local side
the program assumes that the flange is located on the side in the specified direction of the relevant local axis. In the following figure, the beam is part of a plane frame and was defined as "Use section major axis". The relevant local axis for flange location is x2.



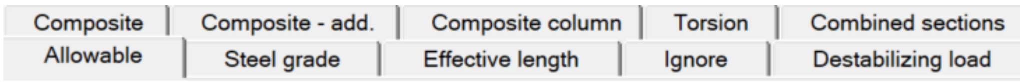
Note:

- this option applies only to sections defined using the elastic modulus (all working stress - ASD - codes and certain sections in ultimate stress - LRFD - codes)
- for all other non-symmetric sections, this option defines the location of the shear center relative to the centre-of gravity of the section



7.7 Parameters

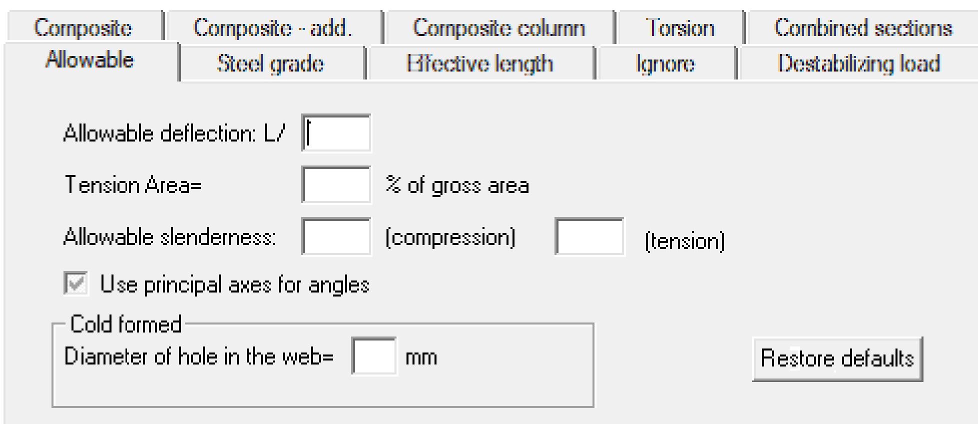
Define the design parameters ***for specified members***.
If a parameter is not defined for a specific member using this option, the program uses the default 702 value when designing the member.



Only parameters with defined values are used; all other parameters are not revised for the selected members.

Note:
- if a submodel instance is currently displayed on the screen, the parameters are revised only for the instance displayed or all instances of the same submodel.
- if the main model is currently displayed, the parameters are revised only for the main model unless **Select beams in submodels** is specified in the standard beam selection option.
- Default parameters can be restored to selected members. Refer to Parameters - allowable 717.

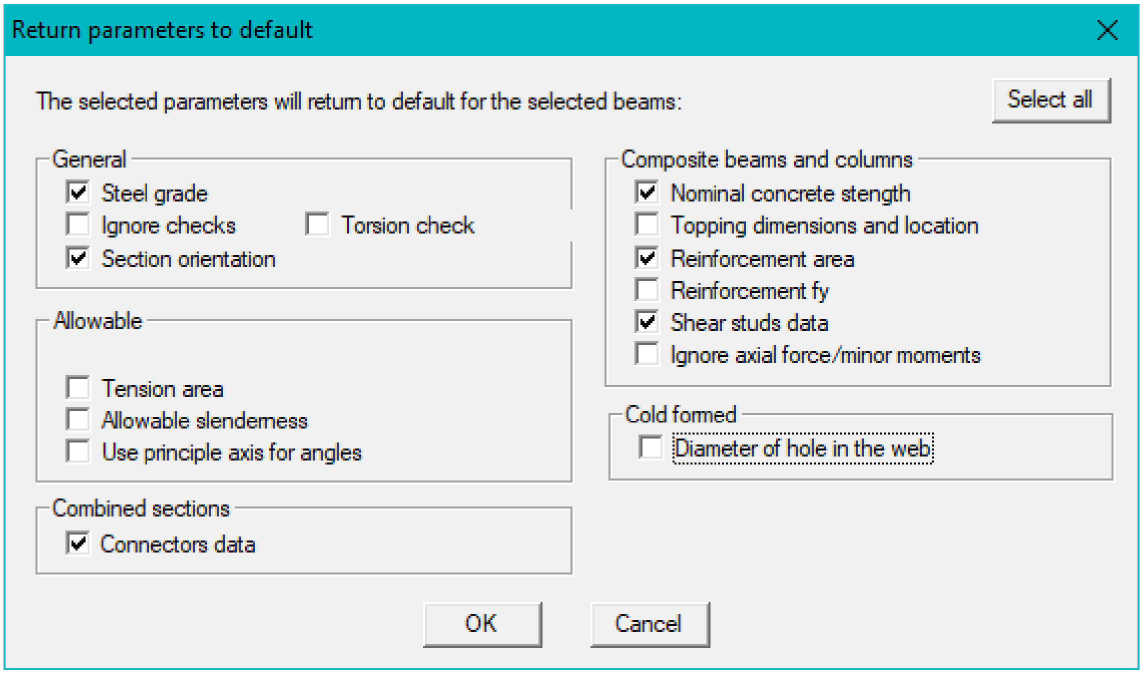
7.7.1 Allowable



For all parameters, refer to:
- Default - general 702.
- Default - cold-formed 706

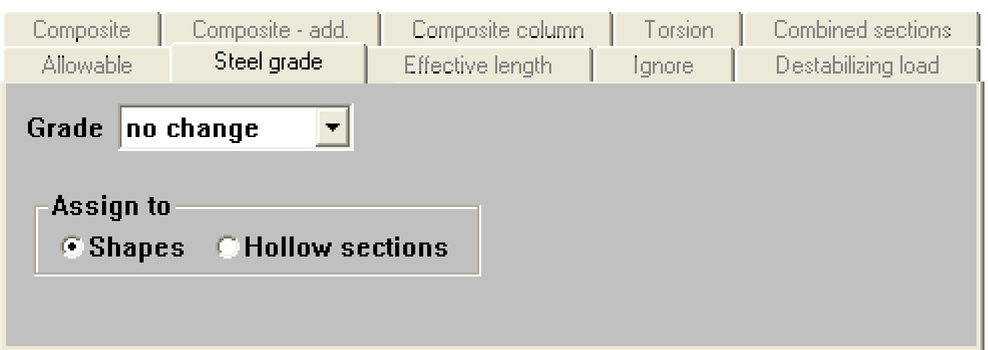
Restore defaults

Restore the default parameters to selected beams:



- select the parameters to be restored
- select the members using the standard beam selection option.

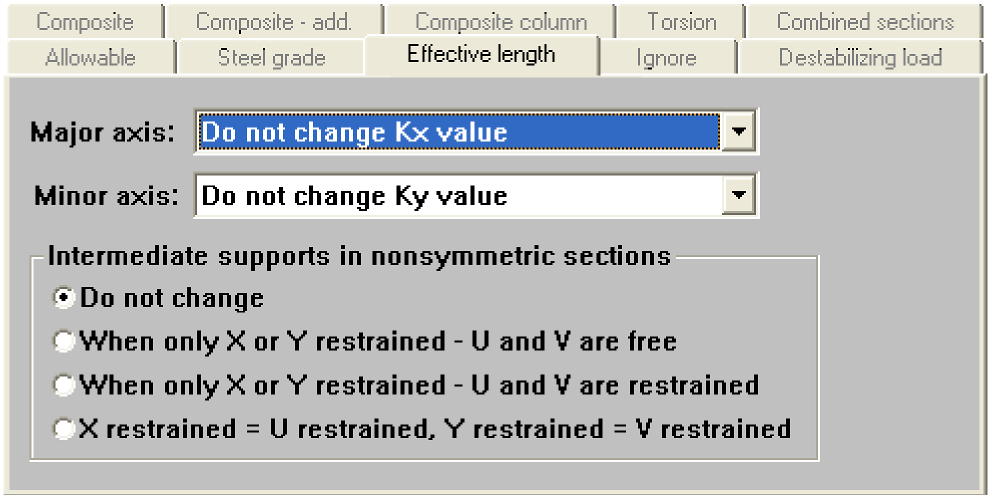
7.7.2 Steel grade



Refer to Defaults - steel grade 704

7.7.3 Effective length

Define the effective length of columns:

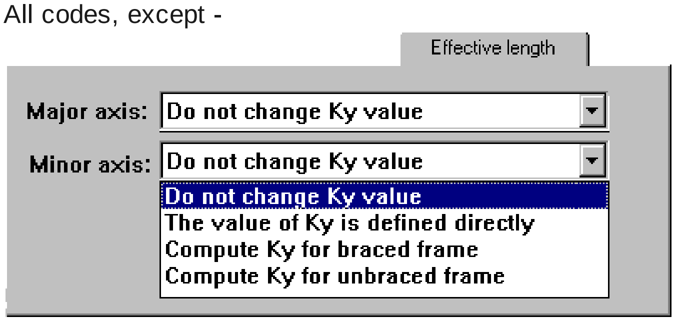


Major minor axis

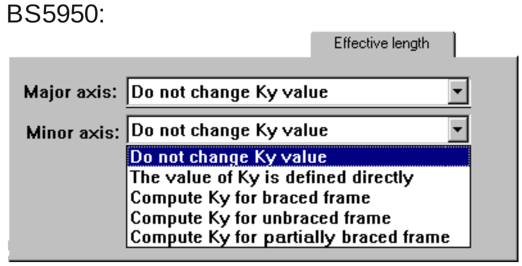
Specify the effective length ratio L_e/L about both axes.

Select one of the following from the list box:

All codes, except -



BS5950:



- ***Define directly***
  enter the value for kx / ky
- ***Compute***
  Instruct the program to compute the factors according to the Code.

AISC, AASHTO, CSA: The program calculates the k-factors according to the standard alignment charts, such as Figure C-C2.2 in the Code commentary.

The stiffness of members attached to "Restraints" (pinned or fixed supports) defined in ***STRAP*** geometry are calculated according to the note in Fig. C-C2.2, as follows:
- pinned: G_B = 10.
- fixed: G_B = 1.

Eurocode: The program calculates the k-factors according to Figures E.2.1 and E.2.2

IS:800: The program calculates the k-factors according to Figures C-1 and C-2

BS5950: Referring to BS5950 Appendix E, § 4.7.3 and Figures E1, E2, E4 and E5, instruct the program to calculate the relative stiffnesses of the members framing into the end nodes.

Braced frame - the program uses Figure E1.
Unbraced frame - the program uses Figure E2.
Partial bracing - the program uses Figures E4 and E5 and requests the value of "k3".

The stiffness of members attached to "Restraints" (pinned or fixed supports) defined in ***STRAP*** geometry are calculated according to § 5.1.3, as follows:

- pinned: $k_2 = 1.0/1.1$
- fixed: $k_2 = 0.5$

SNIP II-23: Referring to SNIP, Section 6.10:

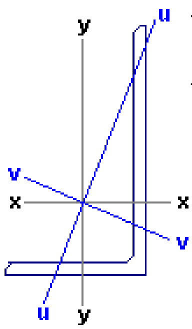
- Braced frames - the program uses Eq. 70B.
- Unbraced frame - the program uses Eq. 70a, 70b.

Note:

- the program always calculates the factors using the **sections defined in *STRAP* geometry**, not the sections selected by the program, i.e. the factors do not change when new sections are selected or specified.
- if the beam is part of a "combined" beam, the program calculates kx/ky for the combined beam.
- intermediate supports and "combined" beam end conditions are ignored by the program.
- in plane models, the program calculates the factor for the in-plane axis only.

Non-symmetric sections

This option apples only to the axial compression calculation (not bending, LTB or deflection).



The program calculates buckling about both the geometric axes and the principal axes (u,v). Supports are defined only about the geometric axes (x,y). To apply supports to the principal axes, specify that certain geometric axis supports also act in one or more principal axis directions:

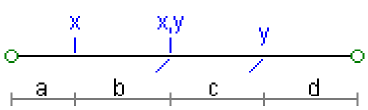
When only X or Y are restrained - U and V are free
when there is only one geometric axis restraint defined (either X or Y) - ***both*** U and V are assumed free

When only X or Y are restrained - U and V are restrained
when there is only one geometric axis restraint defined (either X or Y) - U and V are assumed restrained

X restrained = U restrained; Y restrained = V restrained
the U restraint is identical to the X restraint; the V restraint is identical to the Y restraint

Note that all points supported in both X and Y are automatically supported in both U and V.

Example: the following column has three supports defined as shown, creating four design segments a, b,c,d -



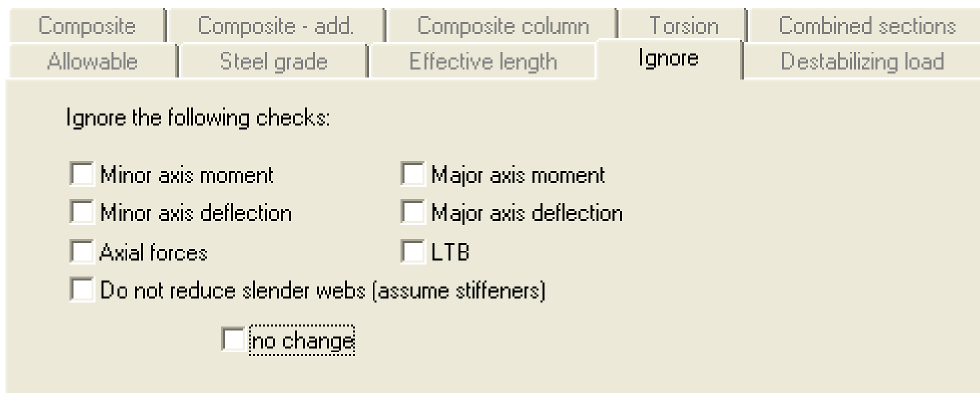
Option	Segments checked:
X or Y restrained - U,V free	(a + b) - r_v (c + d) - r_v
X or Y restrained - U,V restrained	a, b, c, d - for r_v (a + b), c, d - for r_y
X=U restrained,	(a + b), c, d - for r_v

Y=V restrained	a, b, (c+ d) - for r_u

Note that only the potentially critical checks are listed in the table.

7.7.4 Ignore

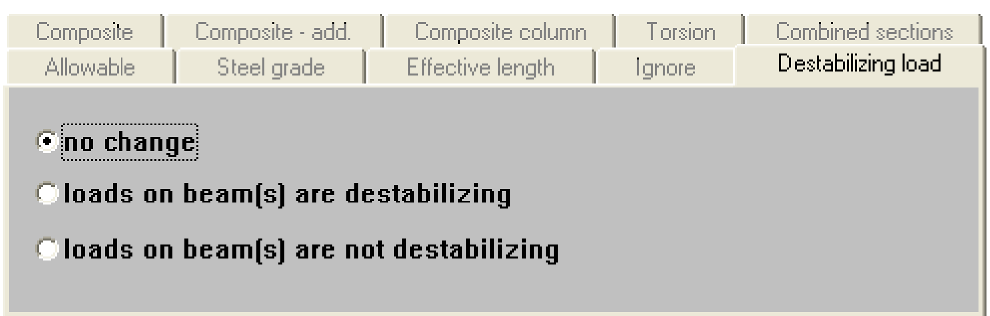
In space models, analysis results may include relatively small values for moments, deflections, etc. which may be ignored for certain members, e.g. a floor slab restrains minor axis bending. These small values may however cause the program to select a larger section.



This option instructs the program to ignore specified design checks when selecting members:

- moment, deflection, etc:
  these design items are not calculated and results are not displayed.
- slender webs:
  the program does all the calculations but assumes that the user will detail suitable horizontal stiffeners and so does not reduce the allowable stress. Note that ***the stiffeners are not designed by the program***.

7.7.5 Destabilising



BS5950:	Specify that a destabilizing load as defined in § 4.3.4 acts on selected members.
Eurocode:	Destabilizing loads: when using equation F.2 to calculate M_{cr} for lateral-torsional buckling for "loaded" beams, the program calculates Z_g as the maximum distance from the shear centre to the face of the beam; otherwise $Z_g = 0$.
GBJ 17:	For calculation of factor β_b for lateral-torsional buckling, a destabilizing load indicates a transverse load acting on the compression flange.

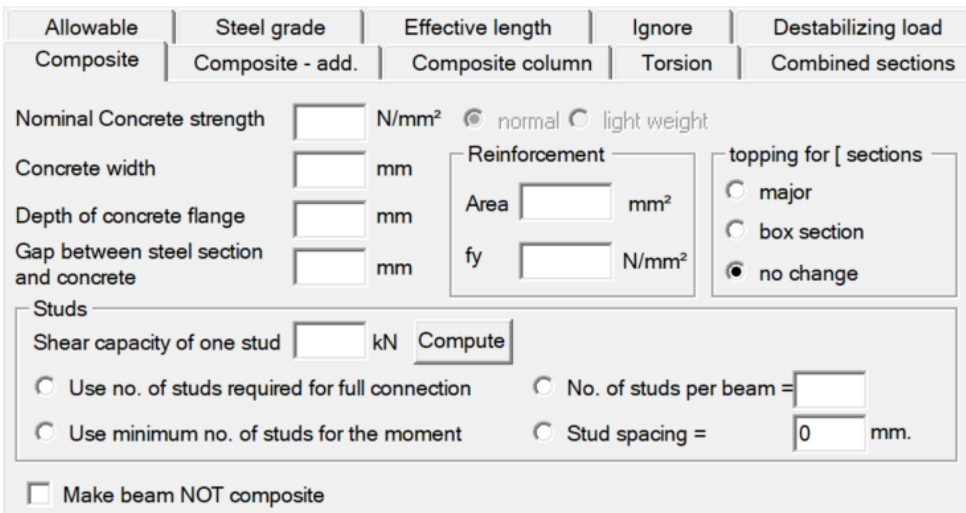
IS:800-07: Destabilizing loads: when using the equation in E.1.2 to calculate M_{cr} for lateral-torsional buckling for "loaded" beams, the program calculates y_g as the maximum distance from the shear centre to the face of the beam; otherwise y_g = 0.

7.7.6 Composite

Specify the parameters and dimensions for composite beams.

To design a composite beam:

- specify the Default parameters (Composite 707 and/or Composite - additional 724)
- specify Parameters for individual beams if necessary (Composite and/or Composite - additional)



Concrete strength

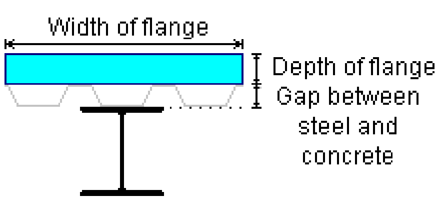
Refer to Default - composite 707

Normal / lightweight

Refer to Default - composite 707

Depth of concrete flange / Width / Gap

The following dimensions are required:



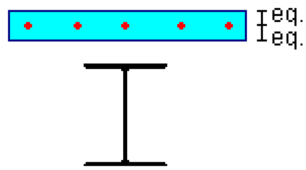
If you select Compute for the stud capacity, the value of "Gap between steel and concrete" is automatically copied from the "h_p" value used to calculate the capacity.

Reinforcement

The reinforcement in the slab increases the section capacity in regions of negative (hogging) moment, i. e. when the slab is in tension.

The program assumes that the reinforcement is located in the center of the concrete slab.

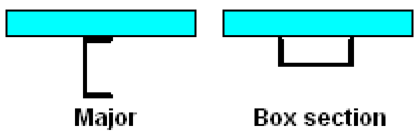
Enter:
- the **total** reinforcement area
- the nominal value of fy



Note:
- The number of shear connectors in negative moment regions cannot be specified. If reinforcement in the topping is specified, the program assumes full shear connection in the negative moment region; hence the user should not specify topping reinforcement if the required number of connectors cannot be provided.

Topping for [-section

Select one of the following options:



Studs

There are two options for stud design:

Use no. of studs required for full connection:
Compute the total number of studs required for full capacity.

Notes:
- For positive moment - the program calculates the total number of studs within the positive moment area. The user must place half of the studs amount at each side of the maximum value of the positive moment.
- For negative moment - the total amount of studs within the negative moment area is determined based on the concrete reinforcement.

Use minimum no. of studs the moment:
Compute the optimal stud spacing for the design moment. i.e partial connection.

Note: The program calculates the stud spacing based on the maximum design moment. Depending on the moment diagram and values, in some cases the default calculation based on maximum moment can be inaccurate. For these cases it is necessary to set the beam design for each 1/10th of a beam. Refer to Default - Design each 1/10th of a beam 703.

No. of studs required per beam=:
Specify the number of studs in the ***positive moment region*** of the beam.

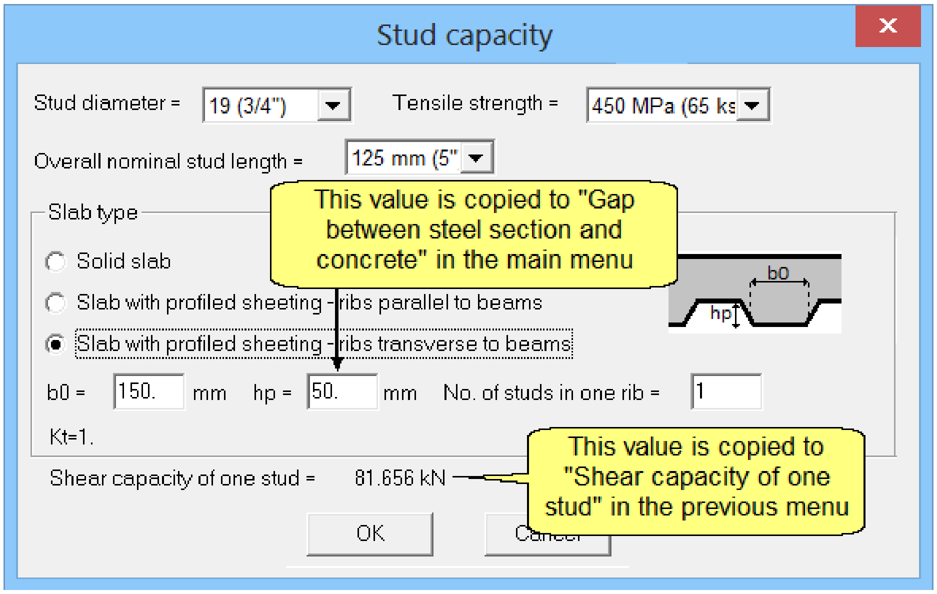
If the user specifies the number of connectors in the positive moment region and the number is less than the number required to develop the positive moment capacity of the section, the program designates the beam as having a "partial shear connection" and reduces the Bending capacity accordingly (by reducing the effective slab width).

The number of connectors in negative moment regions cannot be specified. If reinforcement in the topping is specified, the program assumes full shear connection in the negative moment region; hence the user should not specify topping reinforcement if the required number of connectors cannot be provided.

Stud spacing=:
Specify the stud spacing along the beam. The program calculates the capacity based on the actual number of studs in the required regions along the beam.

For both options, specify the shear capacity of a single stud or click 

to calculate the capacity according to the Code. For example, EC3:



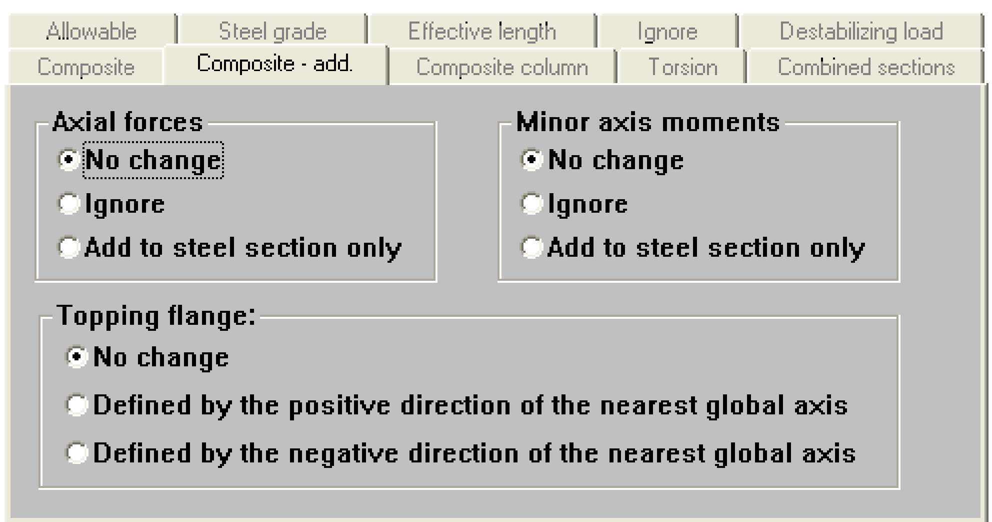
Note:

- for BS5950 only - the program uses 80% of the capacity specified.

Make beam NOT composite

 Make a selected beam non-composite. Apply only when instructing the program to select a new section. Refer to Sections 711.

7.7.7 Composite - additional



Axial forces

Select one of the following options for design of axial forces in composite sections:

 **Ignore**

The program ignores completely the axial forces present in all load cases.

Add to steel section only
The program assumes that the entire axial force is taken by the steel section and that no axial force is present in the concrete slab.

Minor axis moments

Select one of the following options for design of minor axis moments in composite sections:

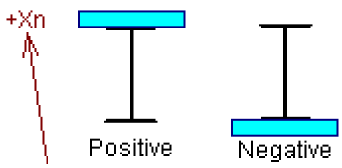
Ignore
The program ignores completely the minor axis moments present in all load cases.

Add to steel section only
The program assumes that the entire minor axis moment is taken by the steel section and that the slab does not contribute the minor axis strength.

Topping flange

The program, by default, assumes that the concrete topping is located on the side of the steel beam in the positive direction of the closest global axis.

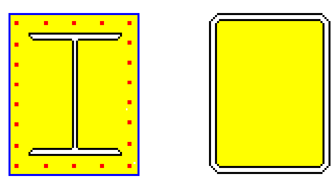
Select one of the following directions:

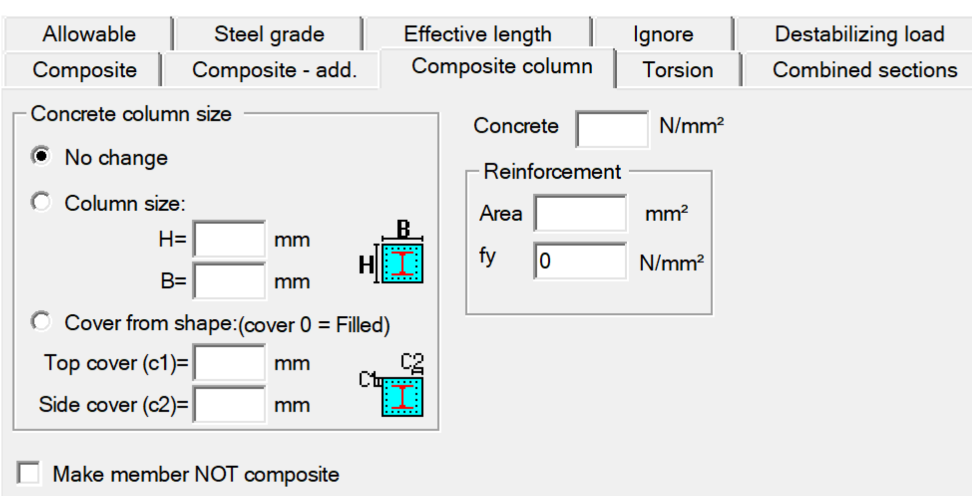


7.7.8 Composite column

Specify parameters for specific composite columns. The program can calculate the following columns:

- all sections encased in concrete, including pipes and RHS
- pipes and RHS filled with concrete


Encased Filled



Concrete

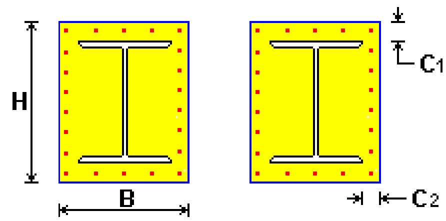
Enter the **nominal** concrete strength used in the current design code; the program automatically reduces the strength by the appropriate factors.

Concrete column size

Two options are available:

Column size
The concrete column size (B,H) is specified, the program searches for the smallest steel section that fits into the concrete rectangle

Cover from shape
The concrete cover (C1, C2) is specified; the program determines the corresponding concrete rectangle dimensions each time it selects a steel section to be checked.



Note:
- The dimensions (B,H or C_1,C_2) are relative to the major/minor axes of the steel section.

Reinforcement

Add longitudinal reinforcement to composite columns. Specify -
- the total area
- the design stress of the reinforcement

Note:
- the additional reinforcement is used to calculate axial capacity only and its location is not relevant. The user must ensure that the specified cover value is sufficient to accommodate this reinforcement.

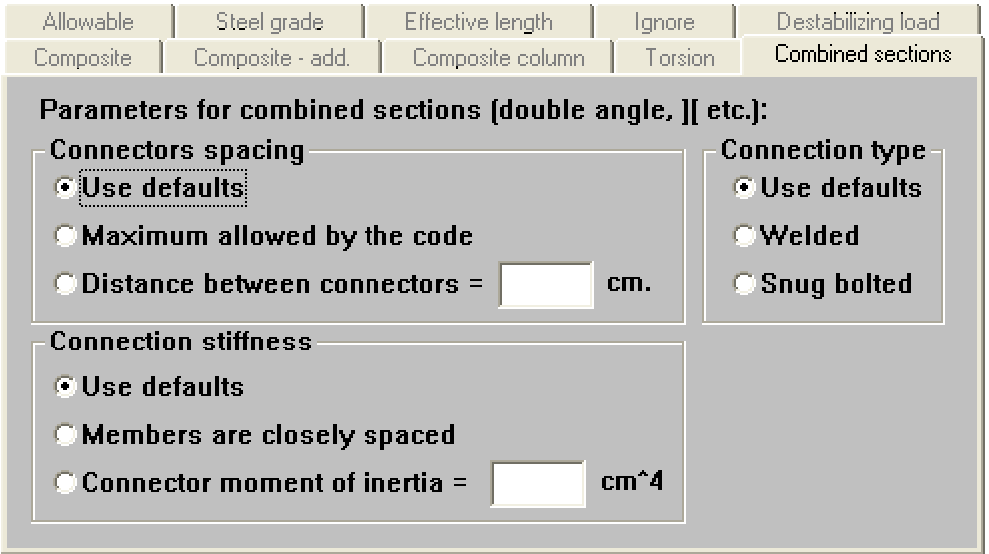
Note:
- the steel section alone is used to calculate bending capacity

Make member NOT composite

Make a selected column non-composite. Apply only when instructing the program to select a new section. Refer to Sections 711.

7.7.9 Combined section

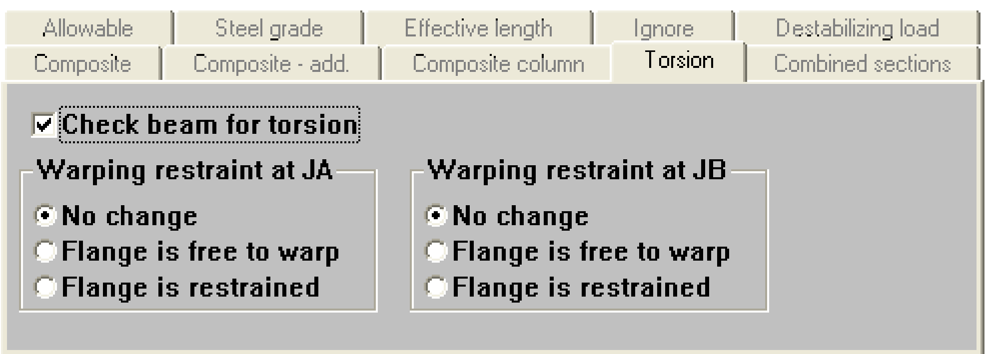
Define additional data for members fabricated from several sections; the parameters defined here override the corresponding default parameters. The program uses this data when calculating the slenderness of the section.



Refer to Default - combined section 709.

7.7.10 Torsion

Specify the torsion parameters for the selected beams:



Check beam for torsion:

☑ include torsional moments in the design check
☐ ignore torsional moments

Warping restraint:

The ends of the beam may be either free or restrained for warping.

Note:
- warping stresses are generated only when ends are restrained
- when a beam is free at one end and restrained at the other, the program calculates the warping stresses along the length of the beam according to the solution for a beam subjected to a linearly varying torsional moment.

Select one of the following options:

◉ **No change**
Do not revise the warping restraint detail for the selected beams

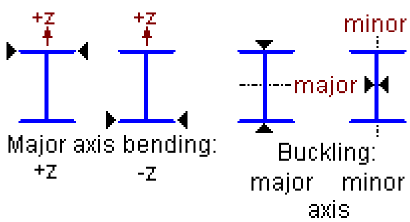
- **Flange is free to warp**
  warping stresses are not generated in the selected beams
- **Flange is restrained**
  Warping stresses are generated in the selected beams

Refer also to Torsion - general 781.

7.8 Supports

Define intermediate supports along the length of a member to allow the program to calculate effective lengths.

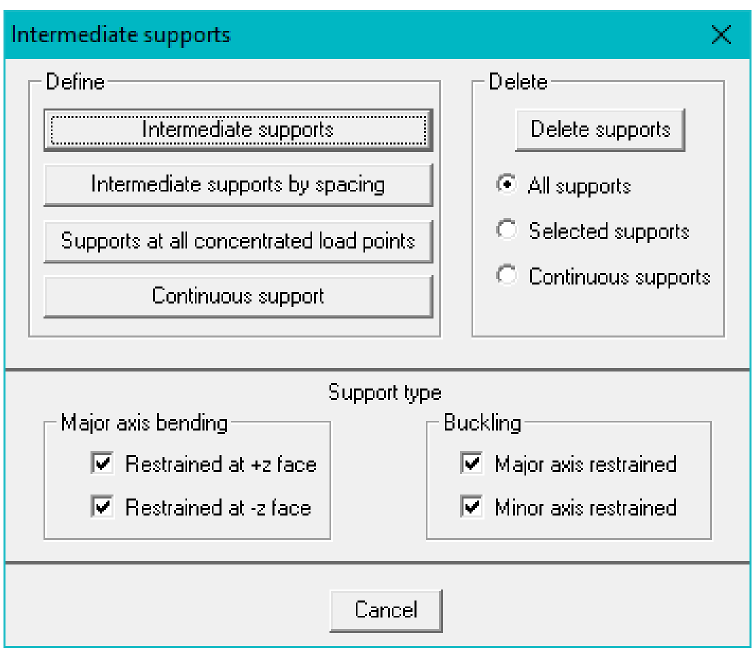
- supports may be defined at any location along the span of a member or as continuous along the entire span. The maximum number of individual supports that may be defined for a single member is 10.
- intermediate supports may be defined for both bending and compression buckling design:



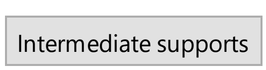
- **Bending**:
  For lateral-torsional buckling check, define the supports at the top and/or bottom faces of the section. The program calculates lateral-torsional buckling separately for the top and bottom flanges according to the supports defined and the sign of the bending moment.
- **Compression buckling**:
  For buckling check, define intermediate supports about the major and/or minor section axes.

Note:

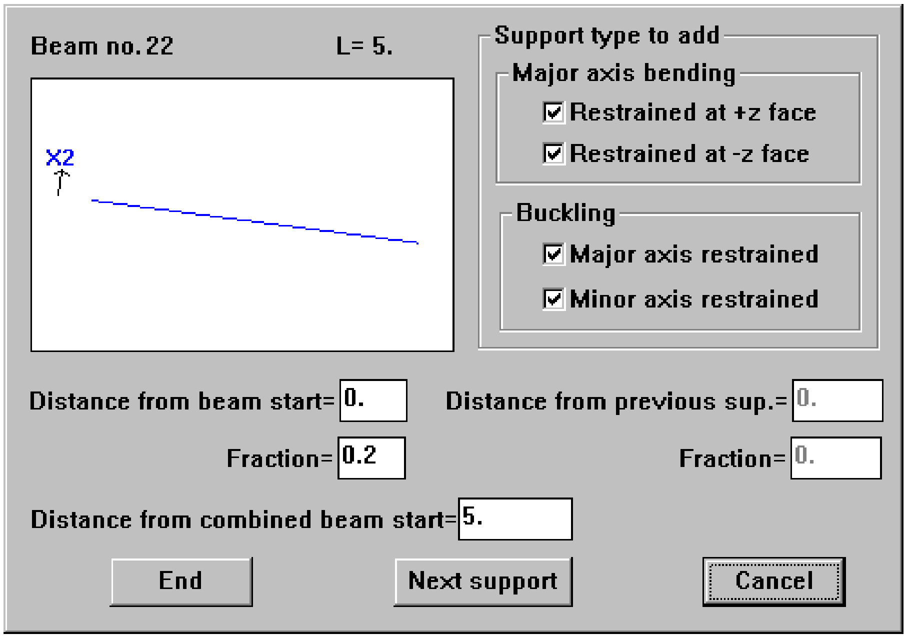
- compression supports do not influence the effective length for lateral-torsional buckling and vice-versa.



Define

- 

  Define support locations along selected members:



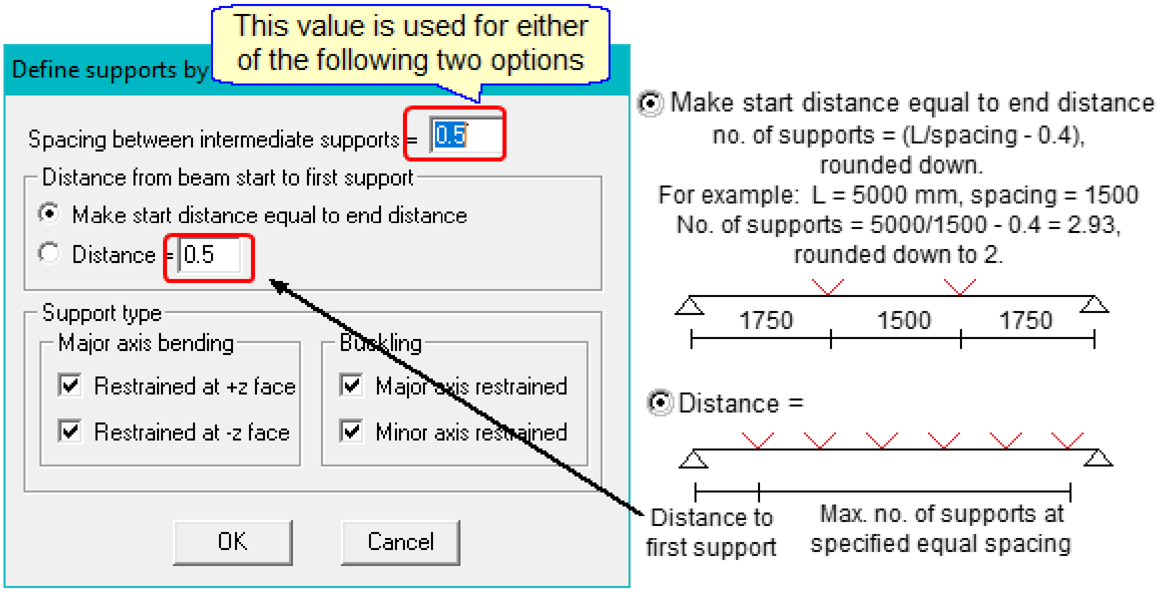
- Set each of the support types to ☐ or ☑ according to the support conditions.
- Define the coordinate of the support along the length of the beam, either as a length or a fraction of the beam length. If a combined beam is selected, the location may be specified from the start of the combined beam.
- The program displays the support on the sketch at the centre of the screen. (for an explanation of the symbols used to display the support, refer to Display - supports 753.)
- Select one of the following:

End	no more supports are located on this beam
Next support	define another support on the same beam. The same support conditions apply. The distance can be defined from the beam start **or** from the previous support (length or fraction).

Select the beams where the support is located using the standard Beam Selection option

- Intermediate supports by spacing

Define multiple supports at equal spacings along the length of the beam:



Select the beams where the support is located using the standard Beam Selection option.

- Supports at concentated load points

  Continuous supports

Define support locations along selected members:

- Set each of the supports types to ☐ or ☑ according to the support conditions.
- Select:
  - **Supports at all concentrated load points**
    add supports at all locations where concentrated beam loads were defined (in any load case). After they are defined, these supports are assumed by the program to be "intermediate supports".
  - **Continuous support**
    Define a continuous support along the ***entire*** span of a member.

Select the beams where the support is located using the standard Beam Selection option.

Note:

- for "Combined beams", the supports are added on the entire length of the combined beam, even if only the first component beam is selected.

Revise/delete

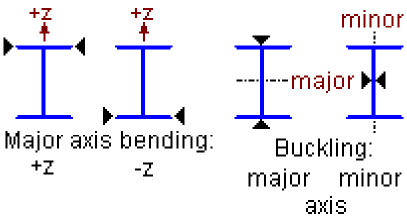
Revise or delete a support already defined.

Select:

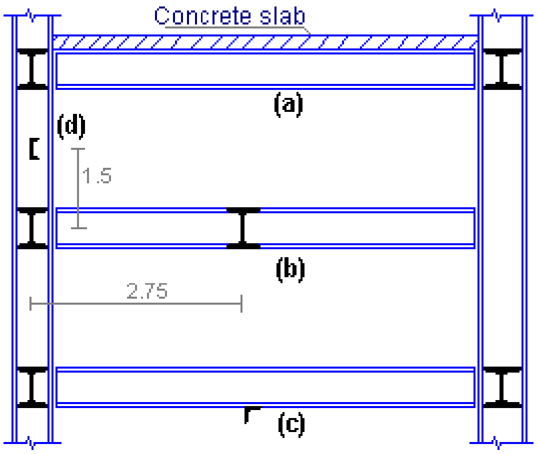
all supports	-	revise/delete **all** supports (intermediate or continuous) on selected spans.
continuous supports	-	revise/delete continuous supports on selected spans.
selected supports	-	revise/delete intermediate supports on selected spans.

- Set each of the supports types to be deleted to ☑.
- Click the **Delete supports** button.
- select the beams where the supports are located using the standard Beam Selection option.
- for **Intermediate supports**, the program displays a sketch of the beam showing the support locations. Move the to the supports to be deleted/revised and click the mouse.

Support type



Examples:



- ***Member (a):***
  The slab restrains the top flange along its entire length:
  - Select "Continuous support".
  - Set "Major axis bending: at +z face" to "Restrained".

  Note that the slab most likely restrains the beam also against buckling; if so, set "Major axis buckling" and "Minor axis buckling" to "Restrained" as well.
- ***Member (b):***
  The perpendicular beam supports the top and bottom flanges:
  - Select "Intermediate supports".
  - Set "Major axis bending: at +z face" and "Major axis bending: at -z face" to "Restrained".
  - Define "Distance from beam start" as 2.75 m.

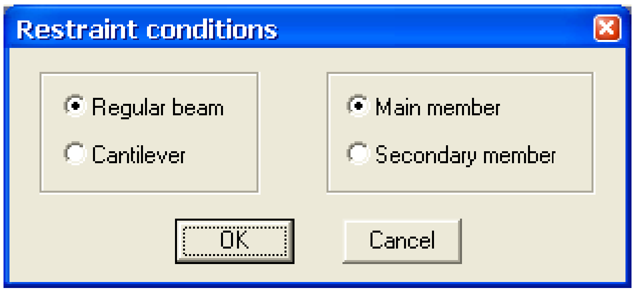
  Note: The perpendicular beam most likely restrains the beam also for "Minor axis buckling".
- ***Member (c):***
  The perpendicular angle supports only the bottom flange:
  - Select "Intermediate supports".
  - Set "Major axis bending: at -z face" to "Restrained".
  - Define "Distance from beam start" as 2.75 m.
- ***Member (d):***
  The channel prevents minor axis buckling:
  - Select "Intermediate supports".
  - Set "Minor axis buckling" to "Restrained".
  - Define "Distance from beam start" as 1.5 m.

Note:
- compression supports do not influence the effective length for lateral-torsional buckling and vice-versa.
- For single angles - compression buckling check: If the user restrains the angle in both local axis directions, the program restrains the angle in **both** principal axis directions. If the user restrains the angle in one local axis direction only, the program assumes that the angle is ***unrestrained*** in **both** principal axis directions. To restrain angles in only one of the principal axis directions, refer to Parameters - effective length 718.
- if an intermediate support is defined in a submodel instance, the program automatically create the same support in all other instances of the same submodel.

7.9 End conditions

Define the end conditions of beams:

7.9.1 AISC, AASHTO, CSA



Regular / cantilever

Beams defined as cantilevers are designed with Cb = 1.00. In addition, cantilevers must be identified so that their maximum deflection will be calculated properly (refer to Calculation method - deflections).

Select the cantilever beams using the standard Beam Selection option.

Main / secondary

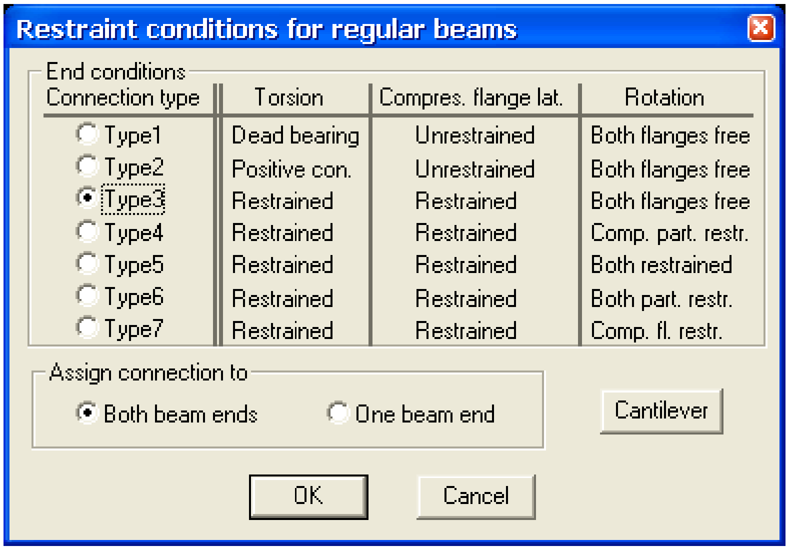
Applicable only for design according to the ASD code - 1978 version. For beams with l/r > 120 and specified as "secondary", Fa is modified by a factor: **1.6 - l/200r**. Note that this clause is not present in the 1989 version of the ASD code.

Select the main/secondary members using the standard Beam Selection option.

7.9.2 BS5950, IS800-07

Calculate the effective length Le for lateral-torsional buckling. Define:
- for fixed/pinned beams, define the "Conditions at Support" referred to in BS5950 - Tables 13 or IS800 - Table 15.
- for cantilevers, define the restraint conditions at both ends according to BS5950 - Tables 14 or IS800 - Table 16. Cantilevers must be identified so that their maximum deflection will be calculated properly (refer to Calculation method - deflections).

Values selected in this option override the default values.



End conditions

The **End conditions** refer to BS5950 - Tables 13 or IS800 - Table 15:

Assign to

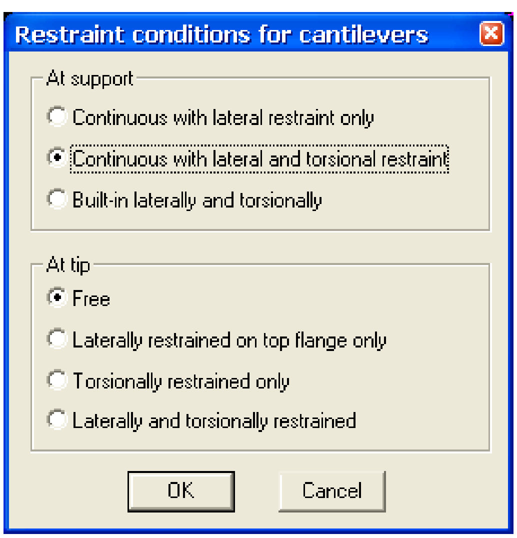
Both beam ends
select members using the standard beam selection option

One beam end
beams must be selected individually; move the crosshair to the end of the beam to be released; note that the blip can appear at either end of the beam.

Cantilever

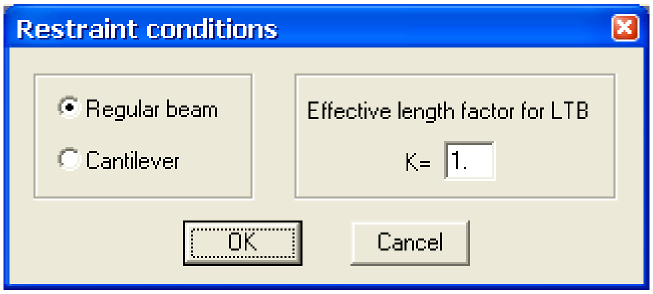
Referring to BS5950 - Tables 14 or IS800 - Table 16 -

- select the restraint condition at both ends of the cantilever:



- select the ***free end*** (tip) of the cantilever. Beams must be selected individually; move the crosshair to the end of the beam to be released; note that the blip can appear at either end of the beam.

7.9.3 Eurocode 3



Regular / cantilever

Cantilevers must be identified so that their maximum deflection will be calculated properly (refer to Calculation method - deflections).

Select the cantilever beams using the standard Beam Selection option.

Effective length factor

Define the value of the "k" factor for specified beams, as explained in Appendix F of the Code (§ F.1.2). The factor refers to end rotation on plan; if not defined, it is assumed by default to be equal to 1.0.

Define a value for "k" and assign it to members using the standard Beam Selection option.

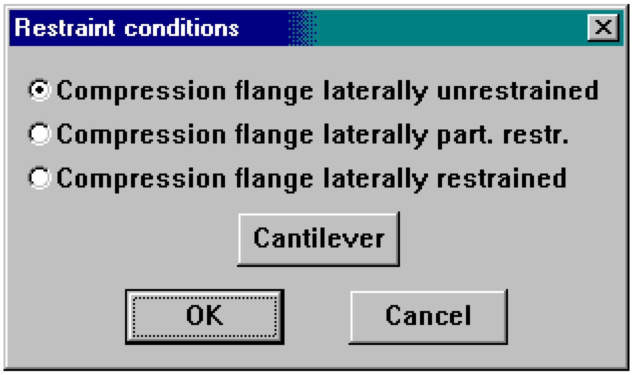
7.9.4 IS:800 - 84

To calculate the effective length L_e for lateral-torsional buckling, define:

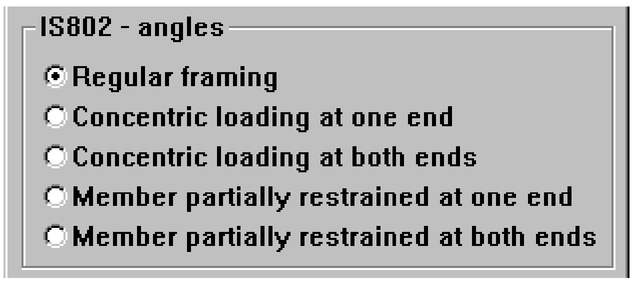
- for fixed/pinned beams, define the support conditions according to § 6.6.1 of the Code.
- for cantilevers, define the restraint conditions at both ends according to § 6.6.3.
  cantilevers must be identified so that their maximum deflection will be calculated properly (refer to Calculation method - deflections).

Values selected in the 

option override the default values



when IS:802 is used:

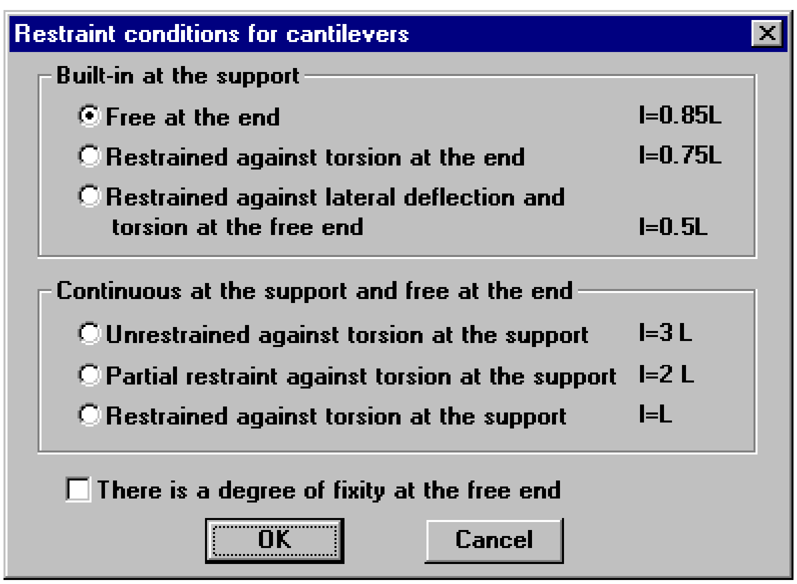


Restraint conditions

The three options correspond to items a), b) and c) in § 6.6.1.

Cantilever

Referring to § 6.6.3 in IS:800, select the restraint condition at both ends of the cantilever; the six options correspond to items (a) to (f) in § 6.6.3:



Then select the ***free end*** (tip) of the cantilever.

IS:8002 - angles

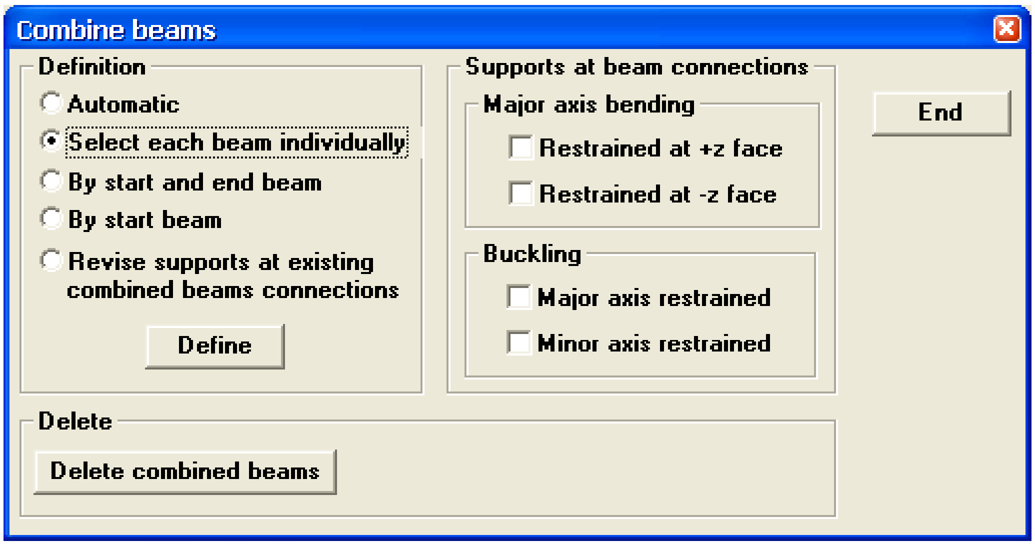
Select the connection conditions according to IS802 (Part 1/Sec 2) - Clause 6.1 and Annex B; the end conditions are used to determine the Slenderness ratios for the compression members.

7.10 Combine beams

The program, by default, designs each member defined in geometry as a separate unit. In certain cases, this results in an incorrect design. For example -

- frames: dummy nodes along a beam
- grids: intermediate node on a main beam where it supports a secondary beam.
  nodes where grid of elements is connected

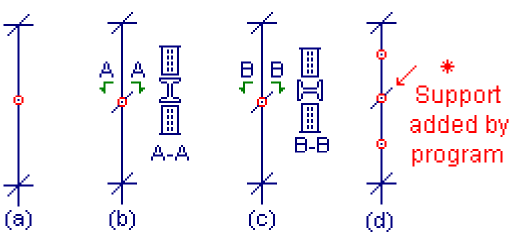
Use this option to combine members into a single design unit; select a series of beams which have common nodes and specify the bending/buckling restraints at the connection.



Automatic

The program identifies connected beams on the same line that have no perpendicular beams or columns at their common node in one or both directions. The program automatically combines the beams and applies the relevant restraints.

For example:



Restraints applied at o :

		(a)	(b)	(c)	(d)*
Buckling	major		✔		
	minor			✔	
Bending	+z			✔	
	-z			✔	

∗ Example(d):
The specified restraints are always identical at all common nodes in a combined beam. Therefore the program applies no restraints at all three nodes and adds a "Support" at the middle node (with restraints according to the orientation of the column - A-A or B-B). Note that the Support remains if the Combined beam is deleted.

Individually

Select a series of members using the standard Beam Selection option. The program checks that the beams are connected.

Note:
- Connected beams with x1 axis pointing in opposite directions may be combined (in this option only)
- Connected beams not forming a straight line (e.g. an arc) may be combined (in this option only)

Beam start and end

Select the first and last members only of the combined beam; the program automatically identifies all intermediate members. The order of selection is important - the program only "combines" members when the JA node of the member is the same as the JB node of the previous one.

Beam start

Select the first member only of the combined beam; the program automatically identifies all following members. Note that the program adds the following member to the beam only if the angle between it and the previous member is less than 30°.

Revise at existing

Revise the "combined" directions for a combined beam:
- specify the new bucking and bending restraints
- click Revise
- select the Combined beams using the standard beam selection options.

Supports at connections

The program assumes that the combined beam was defined as a single member, i.e. the common nodes do not exist. You may add intermediate supports ***at the common nodes***:

This menu is identical to the one in Supports 729.

These support conditions will be defined at the common nodes in the Combined beam. If the support conditions are not identical at all the common nodes:
- select the ***minimum*** conditions in this option
- return to the Supports 729 option and define the missing supports at the relevant member start/end.

Delete

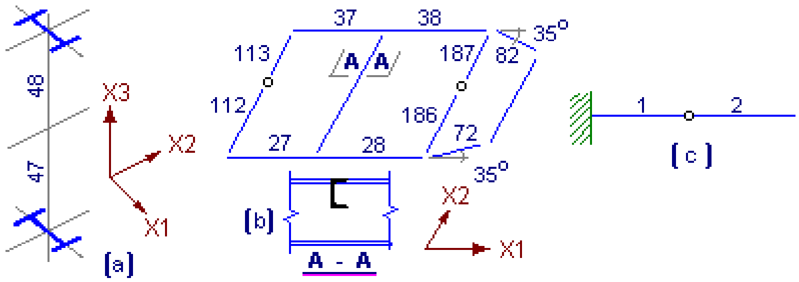
Select a member to be deleted from a combined beam; all the following members will also be deleted from the combined beam list.

Note:
- A member cannot appear in two different "Combined" beams.
- A maximum of 50 members may be combined in one beam.
- All members in a Combined beam are designed according to the current parameters for the ***first*** member.
- When "Checking" sections for Combined beams, different sections may be specified for the different members in the list (but all other parameters will be those of the first beam in the list).
- if a combined beam is defined in a submodel instance, the program automatically defines the same combined beam in all other instances of the same submodel.
- The deflection check is calculated for the length of the ***combined*** beam. If there is a deflection support at a common node, i.e. the beam deflects only on part of its length, the allowable deflection parameter should be adjusted accordingly. For example: two members of equal length are combined; enter deflection limit of L/150 instead of L/300.
- If the axial force is not constant in a Combined beam, the program uses the largest (absolute) value for

all the design checks throughout the length of the beam.

- A beam with a hot-rolled type or section cannot be combined with a beam having a cold-formed type or section.

Examples:



Example (a):

- Combine members 47 and 48.
- Define a "Minor axis buckling" support at the common node

Example (b):

- Combine members 27 and 28, 37 and 38.
  Define a "Major axis bending: +z face" support at the common node.
  If the beams are defined with "select start beam only" option, beams 72, 82 will not be included because the angle is > 30 .
- Combine members 112 and 113, 186 and 187
  As the common nodes are dummy nodes, set all support conditions at the common nodes to "unrestrained".

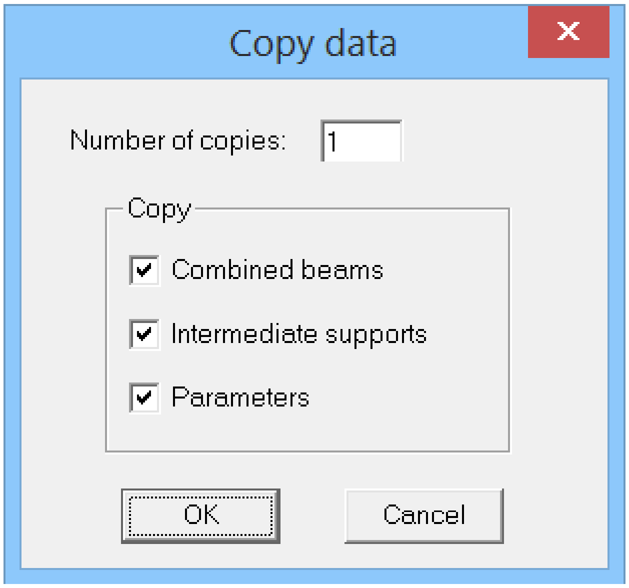
Example (c):

- Combine members 1,2 to form the cantilever; all supports should be "unrestrained". The cantilever must be defined in ***member 1***.

7.11 Copy

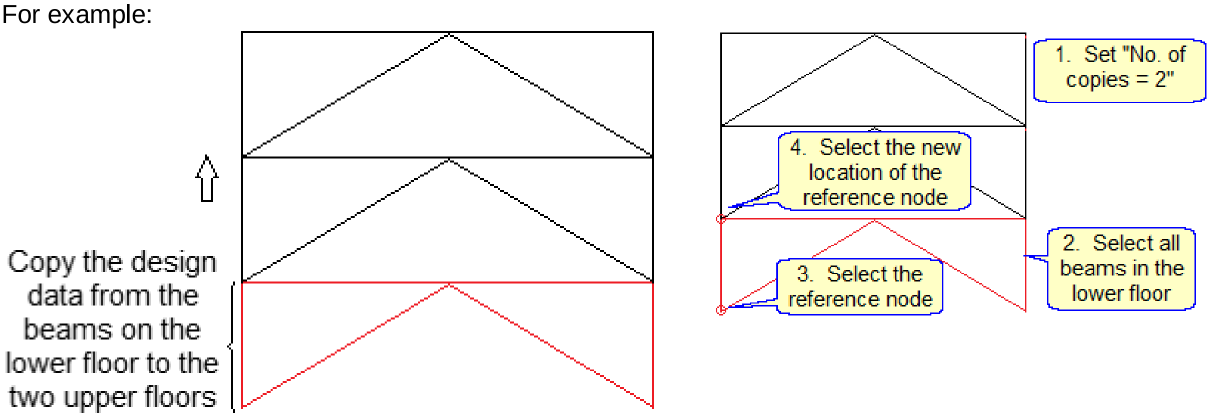
Copy parameters, combined beams and intermediate supports from one group of beams to another:

- select the design items to copy and the number of copies:



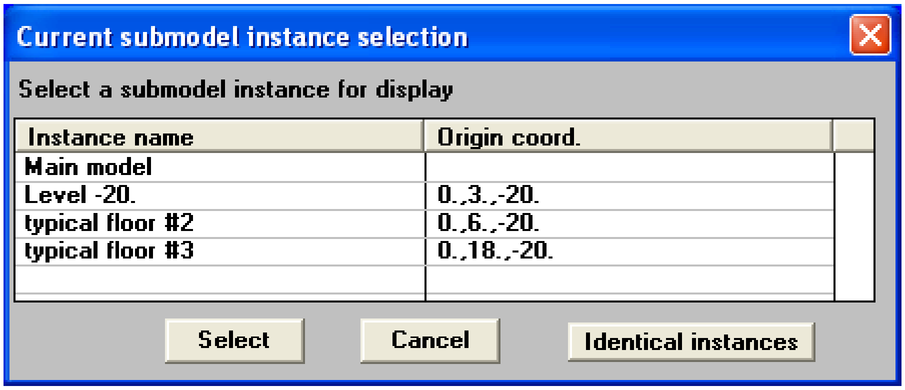
- select one or more beams using the standard beam selection option.
- select a reference node
- select a new location of the reference node.

For example:



7.12 Submodel

Select the submodel to display on the screen:



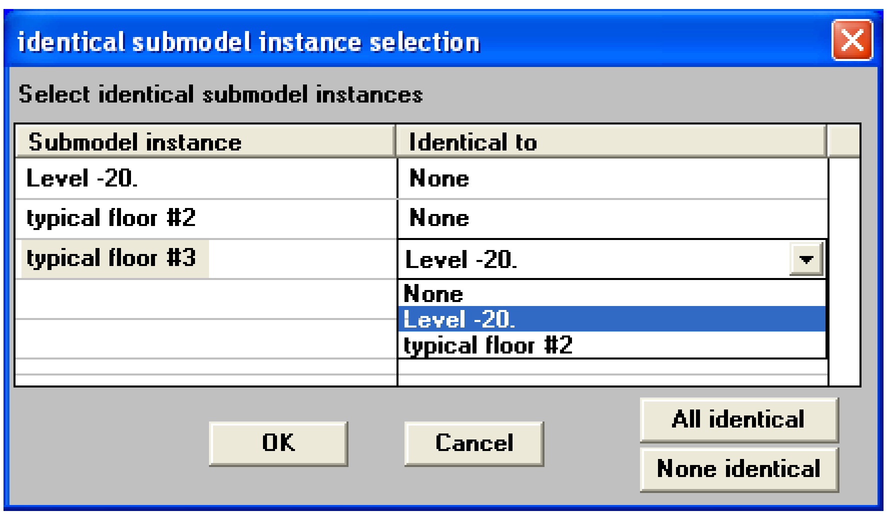
- double-click the appropriate row (or click and highlight an instance and click **Select**)

Note:

- parameters may be assigned to individual members in a submodel only when the submodel is displayed. The same parameters are always assigned to ***all*** instances of the submodel.
- click Identical instances to specify that the same section must be selected for the same member in two or more instances. Note that the instances are selected, not individual members.

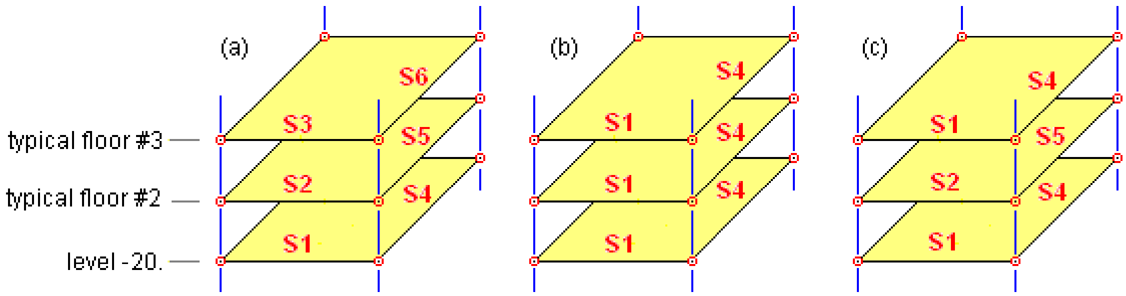
Identical instances

Specify that the same section must be selected for the same member in two or more instances:



Select two or more instances that must be identical;

- click on a line
- click on the arrow in the **Identical to** column
- select another instance that the highlighted instance is identical to. For the example above refer to Figure (c) below.



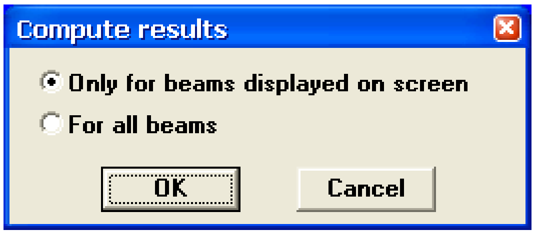
- click None identical if ***different*** members may be selected for corresponding members in the instances of a submodel; refer to Figure (a) above. This is the default option.
- click All identical if identical members ***must*** be selected for corresponding members in ***all*** the instances of a submodel; refer to Figure (b) above.

Note:
- default parameters are always revised for both the main model and all submodels.
- if the main model is currently displayed, the parameters are revised only for the main model unless ☑ **Select beams in submodels** is specified in the standard beam selection option.
- if a combined beam is defined in a submodel instance, the program automatically defines the same combined beam in all other instances of the same submodel.
- if a identical beams are defined in a submodel instance, the program automatically create the same identical beams in all other instances of the same submodel.
- if an intermediate support is defined in a submodel instance, the program automatically create the same support in all other instances of the same submodel.
- When you ◉ **Compute** a beam in an identical instance you must either select ◉ **All beams** or display all of the identical instances in order to calculate the section correctly.

7.13 Compute

When the option "Compute" is selected in the Main Menu, the program begins automatic member selection.

First, select the beams to be computed:

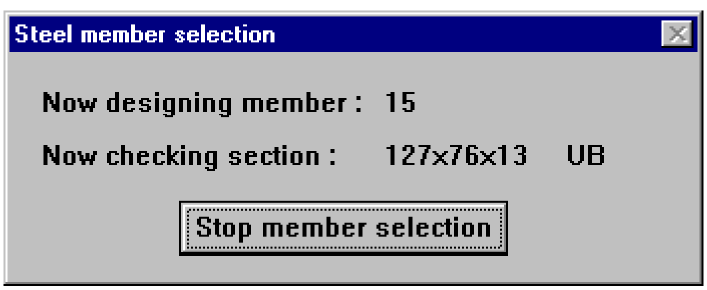


Only for beams displayed on screen
The program computes results for all beams at least partially displayed on the screen.
Delete beams from the screen using the "Remove" or "Zoom" options in "Display".

All beams
The program computes results for the entire model, even if beams were deleted from the display using the "Remove" or "Zoom" options.

Note:
- If a beam is in an identical list or it is in a submodel that has identical instances you must either select **All beams** or display all of the identical beams/submodels in order to calculate the section correctly.

The progress of the design is displayed on the screen:



The program begins the design by checking the lightest section available. If the check fails (because of inadequate capacity or failure to meet user-defined constraints), the program selects the next heaviest member in the list and begins the check again. The process is continued until an adequate section is found.

When "Compute" is completed, the program automatically displays the Result Summary 756.

Note:
- if the type assigned to a member in the Section 711 option is a cold-formed section, the program automatically designs the member according to the specified cold-formed Code.
- If more than one type is allowed (i.e. if there is more than one type in a group), the program searches for an adequate section independently for all of the types and then selects the overall lightest adequate section.
- if a member is part of an identical list, the program begins the design check from the section selected for the previous member in the list. If a larger section is required for the current member, the program rechecks all of the previous members in the list for the new section.

- refer to Design assumptions for detailed explanations on the Code equations used by the program.
- refer to Joists 745 for a detailed explanation on the method used by the program to select steel joists (American steel table only).
- For combined forces, the program may be instructed to either calculate the combined forces at 11 points along the length of the beam, using the actual forces at each point, and then use the worst case for design, or to combine the maximum result from each type, even though they may not be at the same location. Refer to Default - general 702 for more details
- Two options are available for the design of angles, ┐└ and user-defined doubly unsymmetric sections (defined in the Section editor - *CROSEC* utility): the program uses the principal axis properties, **Iu**, **Iv** or the program will use the major/minor axis properties, **Ix**, **Iy**.
- For general non-symmetric sections (e.g. T-sections), the program by default combines the maximum stresses even if they are not at the same point unless the exact flange orientation is specified. Refer to Major - minor 715 for more details.

7.13.1 Joists

The steel design module checks and selects the standard open-web steel joists listed in the Steel Joist Institute (SJI) Tables (1994).

All of the Steel design module options are available for Joists, including min/max height, same section, etc. Note that the program does not select a joist having a maximum fabrication span length (listed in the Tables) less than the actual beam length.

The following joist types are available:
- K-series
- LH/DLH
- KCS

K-series, LH/DLH:

The tables list the safe uniform load carrying capacities. In order to check the adequacy of a joist for a particular load case, the program calculates the following equivalent loads and compares them to the values in the SJI tables:

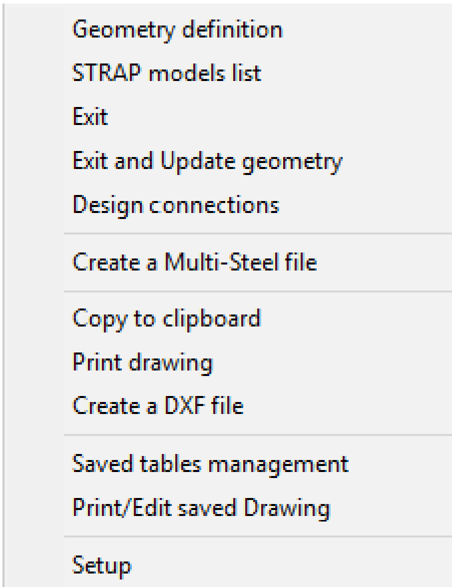
- black figures in table (moments, shear):
  $w1 = 8\ M_{max} / l^2$
  $w2 = 2\ V_{max} / l$
  max**(w1,w2) < W black**
- red figures in table (deflections):
  w1 < w red (360/k)
  where:
  - **L/k** is the allowable deflection value specified by the user (may be different for every combination).

For Limit States (LRFD) design codes, the program increases the uniform load values in the tables by the factor $\phi \times 1.65 = 0.9 \times 1.65 = 1.485$

KCS:

- the maximum moments and shears in the joist are compared directly to the values in the table
- the actual deflection is calculated using the Gross moment of inertia values in the Table, using unfactored loads.
- for Limit States (LRFD) design codes, the program increases the moment and shear values in the tables by the factor $\phi \times 1.65 = 0.9 \times 1.65 = 1.485$

7.14 File options



Geometry

Exit the steel design module and return to the Geometry definition for the same model.

Strap models list

Exit the steel processor and return to the ***STRAP*** main menu to select another model.

Exit

Exit the steel processor and ***STRAP***.

Exit and update geometry

Select this option to instruct the program to rewrite the ***STRAP*** geometry file with the steel sections selected in this module. You may then solve the model again with the new properties in order to obtain more accurate results.

Note:

- the program automatically erases all of the ***STRAP*** geometry and result files are not compatible. As the steel module reads data from the result files, you have to solve the model again before you reenter this module.
- the program erases properties that are not used from the ***STRAP*** geometry file.
- if a section is aligned so that the major axis properties are used in some members and the minor axis properties of the same section are used in others, the program creates two different properties (with the same section) in the ***STRAP*** geometry file.
- the steel design module does not erase the definition of the result combinations.
- For composite sections the program creates a new geometry material for the topping named TOP1.

A warning is displayed if the new number of property groups exceeds the program limit. Two options are available:

the program does not update *STRAP* geometry.

OK the program automatically reduces the number of property groups to the program limit. Sections are replaced by larger existing sections.

Note:
- a 'larger' section has greater values for A, I2 and I3
- built-up sections are not replaced.
- the sections are revised so that there is a minimum increase in the self-weight of the model.

The program displays the increase in self-weight resulting from the reduction in the number of property groups.

Design connections

Design beam-column and beam-beam connections. Refer to Connections.

Create a multi-steel file

Create a "Multi-Steel" ***.SXF** format file. Please contact your *STRAP* dealer for more information.

Copy to clipboard

Copy the current graphic display to the clipboard.

Print drawing

Print the current graphic display. Refer to Print drawing.

Create a DXF file

Refer to Create a DXF file

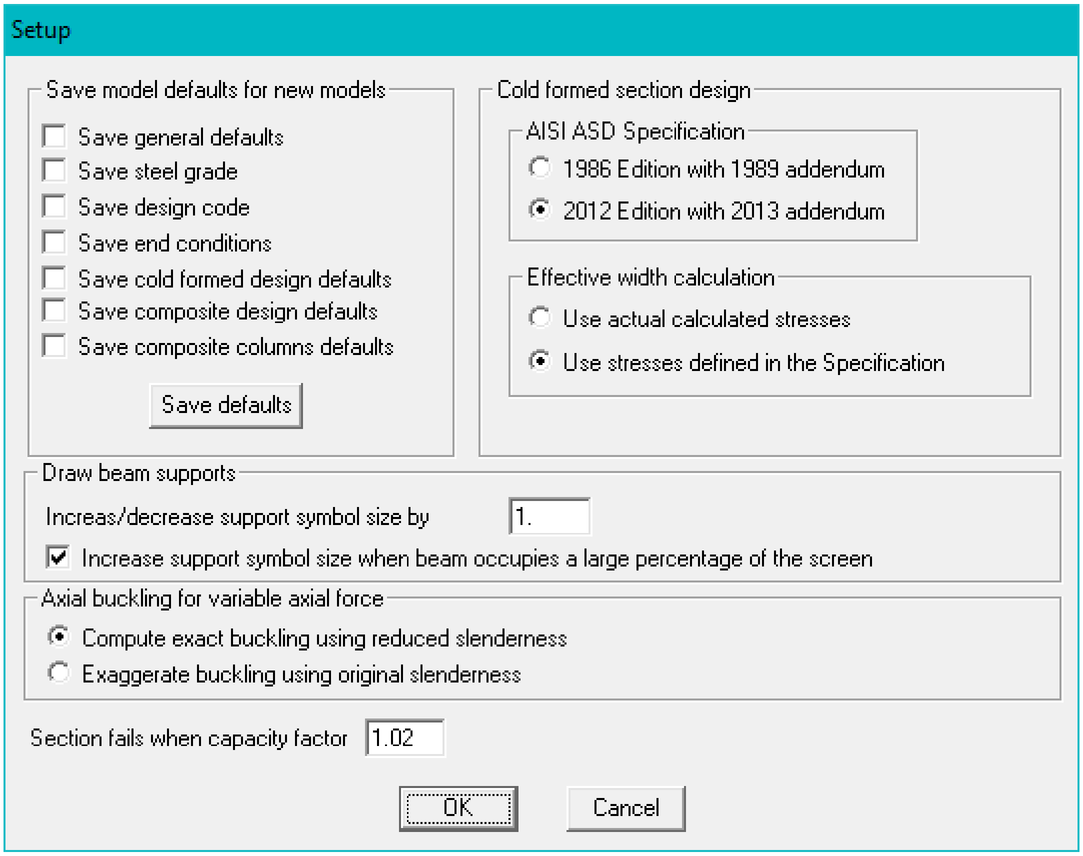
Saved output tables management

Delete tables, rearrange the order of tables or display/print tables that were saved using the **Save output for report generation** option. Refer to Saved tables management,

Print/edit saved drawing

Refer to Print/edit a saved drawing.

Setup

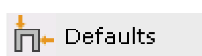


- ***Save model defaults***

  The default parameters specified in the current model can be designated as the default parameters for all subsequent models. Each ☑ option refers to one of the tabs in the Default parameters 702 menu.

- ***AISI - ASD***

  Select one of the editions of the AISI **ASD** Code. The version selected applies to **all** models in all directories if they are checked according to the ASD Code. This option does ***not*** apply to the AISI - LRFD Code.

  To specify AISI ASD or LRFD, click the 

icon in the side menu.

- ***Effective width calculation***

  Specify the stresses to be used for the effective width calculation.

  Use actual calculated stresses
  the effective section properties are based on the actual compressive stress. The properties are calculated using an iterative procedure.

  Use stresses defined in the specification
  the program uses a compressive stress defined in the Code, i.e. the stress is independent of the actual load. The properties are calculated using an iterative procedure.

  The option selected applies to ***all*** models in all directories, and for ***both*** editions of the Code.

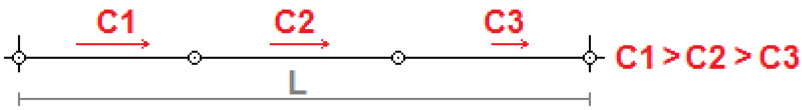
- ***Beam supports***

  - Increase the length of the lines representing beam supports and combined beams 753 by a factor.
  - ☑ the second option to further increase the length of the lines on beams that occupy a large

percentage of the screen (i.e. the length of the lines will increase proportionately when you zoom in on a portion of the model).

- ***Axial buckling***

In many cases a beam/column with axial compression is made of two or more *STRAP* members, each with a different axial force:



The program always designs the member with the maximum axial force (C1 in the example above), but this is overly conservative. The program can do a more accurate calculation by increasing the capacity through a reduction in the slenderness:

Compute exact bucking using reduced slenderness
the program uses the maximum axial compression force together with a reduced slenderness.

Exaggerate buckling using original slenderness
the program uses the maximum axial compression force together with the actual slenderness of the member..

- ***Capacity factor:***

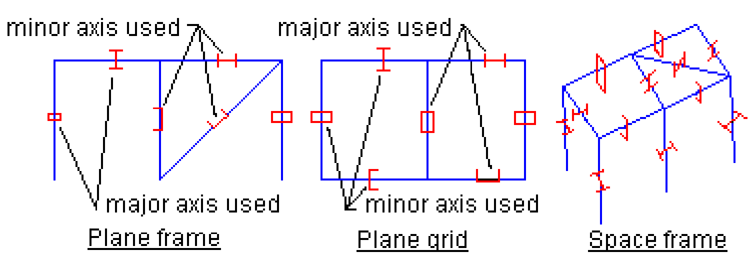
Specify a capacity factor other than 1.00 (the program uses this factor when deciding whether a section is acceptable).

7.15 Display options

Node numbers	Ctrl+N
Beam numbers	Ctrl+B
Property numbers	Ctrl+P
Restraints	Ctrl+R
Springs	Ctrl+S
Beam releases	
Beam offsets	
Render selected sections	
Rendering parameters	
Section Orientation	
Local axes	
General arrangement drawing	
General arrangement parameters	
Submodels instances	

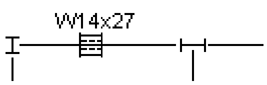
Section orientation

The program displays the section shape and orientation for each beam on the graphic display. For example:



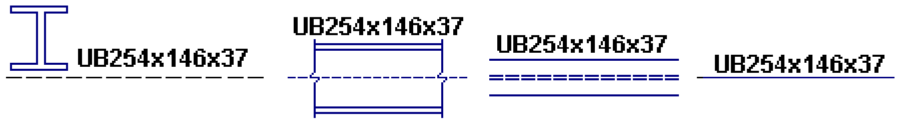
General arrangement drawing

Create a "General arrangement" drawing (Line diagram) for any plane in the model. For example:



- the drawing may be generated on any plane, e.g. plans or elevations. If more than one plane is displayed on the screen when this option is selected, the program prompts the user to select a plane defined by three nodes.
- the program writes the section name adjacent to each member. The name is written only once for a string of identical sections. The text size is specified by the user.
- The program differentiates between primary and secondary beams and terminates the line of the secondary beams before the intersection with the primary beams
- The section shape and name may be imposed on the beam line using one of the following four

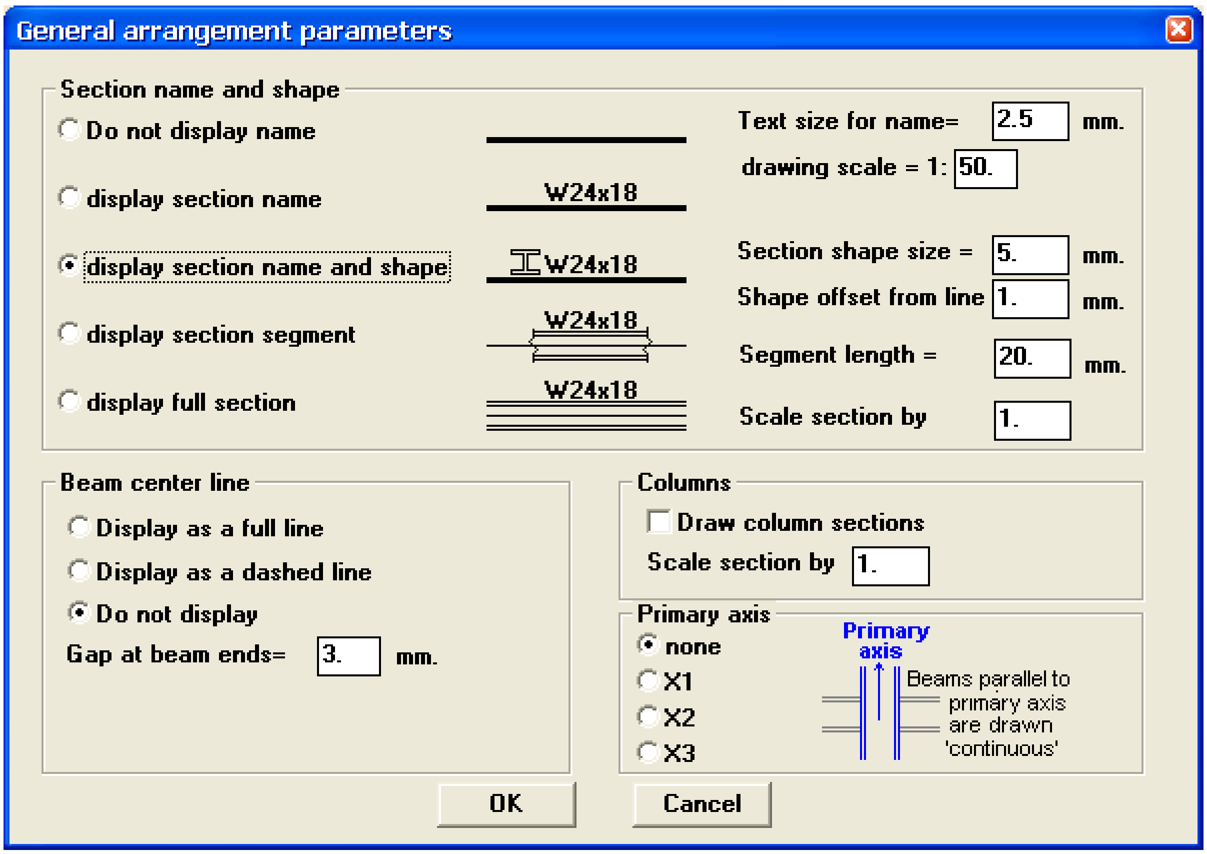
methods:



For more details, refer to General arrangement - parameters751.

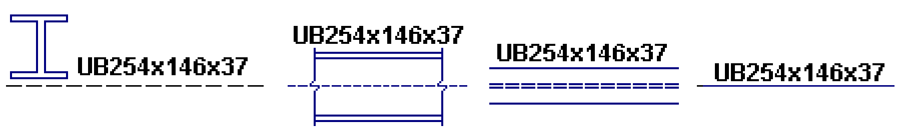
7.15.1 General arrangement - parameters

Specify the parameters for General arrangement drawings. Note that any changes to the parameters also revises existing arrangement drawings (saved as views).



Section name and shape

The section shape and name may be imposed on the beam line using one of the following four methods:



The name is written only once for a string of identical sections.

- The section by default is drawn according to the same scale as the drawing. Modify **Scale section by** to increase/decrease the size of the sections on the drawing.

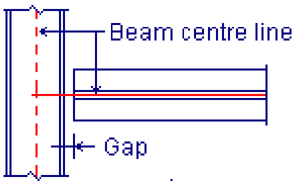
Drawing scale / text size

Specify the drawing scale and the text size. The drawing scale is required to determine the size of the text on the screen display.

Note that the text is printed with this size only if the scale specified when printing is the same as the scale specified in this option; otherwise the text size is modified according to the ratio of the scales. For example, a scale of 1:50 is specified here but a scale of 1:100 is specified when printing: the actual text size will be one-half (50/100) of the size selected in this option.

Beam center line

Specify the beam center line type and the "gap distance":



The centre line may be deleted when **Display full section** is selected.

Columns

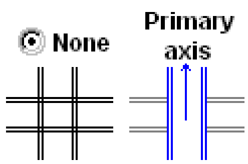
"Columns" are members perpendicular to the displayed plane, i.e. for an elevation, "columns" are in fact the perpendicular beams. The sections are drawn to scale but may be increased/ reduced by the scale factor.

Primary axis

The program differentiates between primary and secondary beams and terminates the secondary beams at the intersection with the primary beams that are drawn as continuous. By default, beams with releases are always secondary beams

Similarly if a "column" (a member perpendicular to the plane) is drawn at an intersection, then all members on the plane connected to that intersection are terminated.

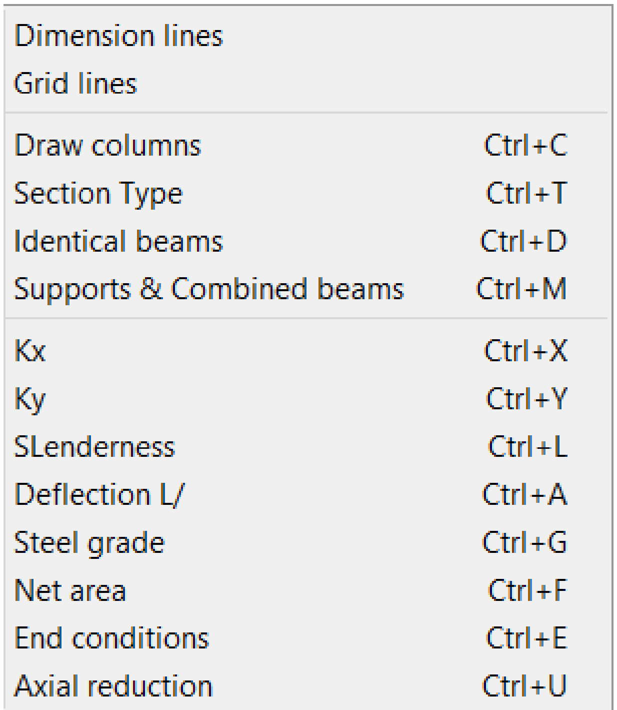
For all other intersections, specify the axis that has continuous members:



If **None** is selected, all beams are terminated at the intersection points.

7.16 Draw options

Display parameters adjacent to the member



Section type/group/check

Referring to Sections 711, the program must be instructed by the user whether to check a specific section (from the tables or a built-up section), or to select a section from a specified type or group, for each member in the model.

Item Displayed:	***Explanation:***
section name:	program checks this section
section type:	program selects a section from this type
group number:	program selects a section from this group
property number:	program checks this built-up section
blank:	"illegal" property assigned to this member; program ignores it.

Draw columns

Display the column sections perpendicular to the plane on the screen (you must first select a plane to display using the Limit display to a plane 55 option).

Identical beams

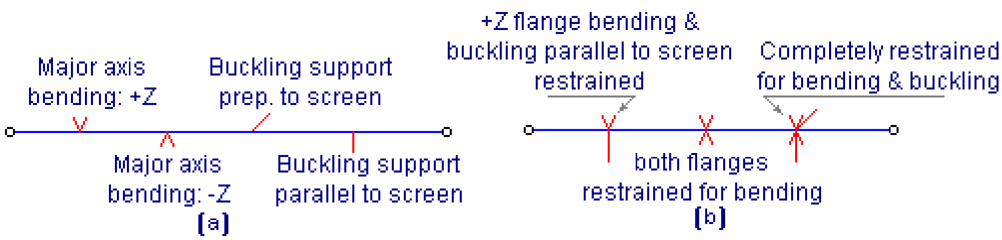
The program displays the number of the identical series that the member belongs to; all members with the same number belong to one identical series.

Supports and combined beams

- ***Supports***

  The program displays the support conditions on the beam for each of the four supports that may be defined: major axis bending: top and bottom flange ; major and minor axis buckling

  The symbols used by the program are:

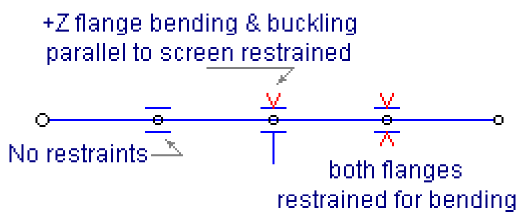


Note that the symbols are displayed relative to the ***screen***:

- Bending:
  The supports are always displayed relative to the direction of the local axis parallel to (or closest to) the plane of the screen.
- Buckling:
  The symbols are for supports perpendicular/parallel to the plane of the screen. Note that if you revise the major/minor definition after a buckling support is defined, the display is also revised.

If more than one support is defined at a location on a member, the program displays the relevant symbols superimposed.

- ***Combined beams***
  The program displays a pair of parallel lines at the common node of combined beams. The support conditions at the common node are superimposed. For example:



Kx/ Ky

Effective length factors

Slenderness

l_e/r (effective length / radius-of-inertia) or **l_e/h** for certain codes.

Deflection

Allowable deflection/span length

Steel grade

Structural steel type

Net area

Net tension area.

End conditions

BS5950/IS800-07:	fixed and pinned end conditions; cantilevers
AISC/AASHTO/CSA:	cantilevers
EC 3:	K factor; cantilevers
IS:800-84:	cantilever end conditions

Axial reduction

Live load reduction factor

7.17 Results

Use this option to:

- display or print all tabular result tables
- display graphically the sections selected or the result capacity

Display Result summary
Display Detailed results
Display Sections summary

Print rEsult summary
Print deTailed results
Print seCtions summary

✔ Include beams not displayed

Display seLected sections
Display caPacity

7.17.1 Summary

The program displays the capacity ratio (Actual/Allowable) for each of the design criteria for each design member. For example:

					CAPACITY					
Beam	Section	Com	Defl L/	Slen	Axial	Dir	Shear	Mom	LTB	Combined Axial+Mom
3	IPE 330	5	1411	100	-0.17	MJ	0.18	0.57	0.66	0.90
						MI	0.01	0.24	0.00	

< 1.00 : section is adequate

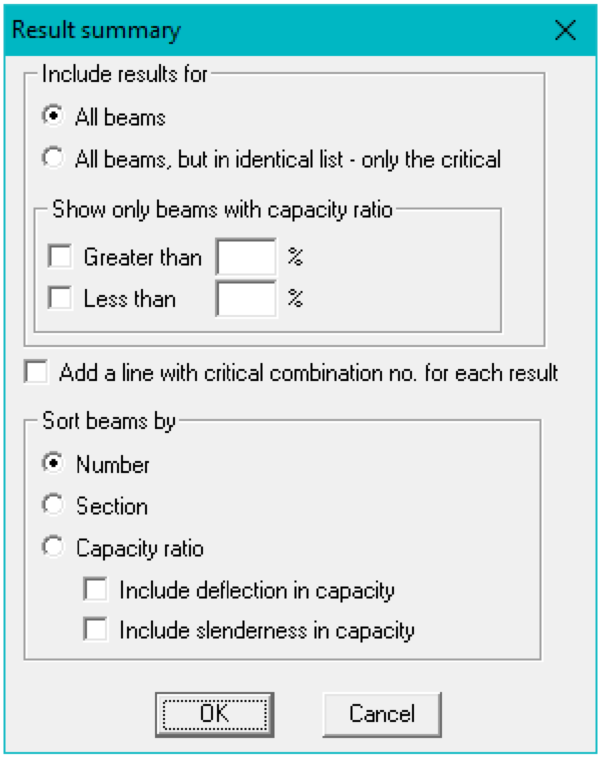
If the program calculates that web stiffeners are needed, the required spacing is displayed:

					CAPACITY					
Beam	Section	Com	Defl L/	Slen	Axial	Dir	Shear	Mom	LTB	Combined Axial+Mom
3	IPE 330	5	1411	100	-0.17	MJ	1.00	0.57	0.66	0.90
Transverse stiffeners needed at spacing = 1523 mm.										

=1.00 : stiffeners are adequate
> 1.00: stiiffeners not adequate

Note: the program displays 1.00 for every value < 1.00 when stiffeners are added (the correct value is shown in detailed results)

Select one of the following options:



Include results for

All beams
Include all beams in the model in the results summary table. Note that only the first member in a combined beam (showing the results for the entire beam) is displayed.

All beams - ... - only the critical
If lists of "identical" beams were defined, the program includes only one beam from each list - the critical beam that determined the section selected.

Show only beams with capacity ratio:

Select an upper limit, a lower limit, or both.

Add a line with the critical combination number

☐ Add a line with ...

Combination corresponds to maximum capacity factor

Beam	Section	Com	Defl L/	Slen	CAPACITY Axial	Dir	Shear	Mom	LTB	Combined Axial+Mom
3	IPE 330	5	1411	100	-0.17	MJ	0.18	0.57	0.66	0.90
						MI	0.01	0.24	0.00	

☑ Add a line with ...

The critical combination is listed for each result type

Beam	Section	Com	Defl L/	Slen	CAPACITY Axial	Dir	Shear	Mom	LTB	Combined Axial+Mom
3	IPE 330	5	1411	100	-0.17	MJ	0.18	0.57	0.66	0.90
			(12)		(6)		(5)	(5)	(5)	(5)
						MI	0.01	0.24	0.00	
							(10)	(11)		

Sort beams by

- The results can be sorted by beam number (the default), section property or capacity ratio.
- if results are sorted by capacity ratio, the deflection and/or slenderness ratios can be ignored.

Examples:

AISC/AASHTO/CSA/SABS/IS800-84:

							CAPACITY			
			Defl			Dir				Combined
Beam	Section	Com	L/	Slen	Axial		Shear	Mom	LTB	Axial+Mom

Section = section selected or checked (note that the program may shorten the type name if the section name is long)
Com = design combination that governs member selection.
Defl/L = maximum deflection in terms of beam length (L/?).
Slen = actual slenderness.
Dir = the section axis which the moment is acting about:
MJ: Major axis
MI: Minor axis

Ratio of actual forces/moments to allowable forces/moments:

Axial = axial force; a negative value indicates compression.
Shear = shear force.
Mom = bending moment.
LTB = lateral-torsional buckling moment.
Combined = Combined axial and moment
IS:800 - maximum of combined axial & moment, combined shear & moment

BS5950, EC3, IS:800-07:

										Combined	
					Pc(t)		Fv	M	M		
			Defl		—	Dir	—	—	—	Axial+Mom	
Beam	Section	Com	L/	Slen	APc(y)		Pv	Mc	Mb	Loc.	Over.

Section = section selected or checked (note that the program may shorten the type name if the section name is long)
Com = design combination that governs member selection.
Defl/L = maximum deflection in terms of beam length (L/?).
Slen = actual slenderness.
Dir = the section axis which the moment is acting about:
MJ: Major axis
MI: Minor axis

Ratio of actual forces/moments to allowable forces/moments:

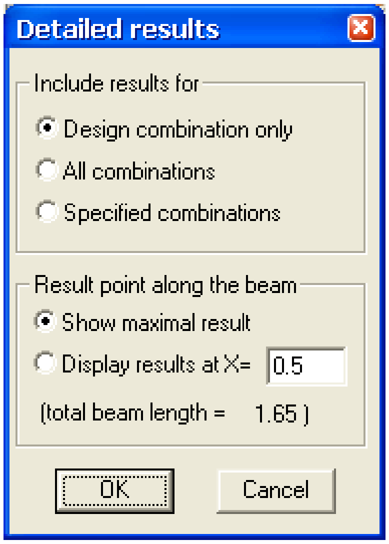
Pc(t)/APc(y) = axial force; a negative value indicates compression.
Fv/Pv = shear force.
M/Mc = bending moment.
M/Mb = lateral-torsional buckling moment.
Loc = Local capacity check, Combined axial and moment
Over = Overall buckling check, Combined axial and moment

For more details refer to:
- Detailed results 759

- Steel codes - calculation method

7.17.2 Detailed results

Select the combination and the location along the beam:



Design combination only
display detailed results for the design combination only.

All combinations
display the results for all combinations sequentially

Specified combinations
the program display a list of all combinations; select one.

Result point:

Show maximum result
The default option; the program searches for the critical point along the beam and displays the results for that location.

Display results at:
Enter any distance from the start of the beam; the program calculates the results and capacity at that point.

For Joists, refer to Detailed results - joists 760

The detailed results displayed for each beam are:
- beam geometry and orientation.
- summary of all design options, intermediate supports, etc.
- summary of analysis results (moments, shear and axial force) for the combination displayed.
- the section selected and section properties.
- the section classification and classification limits (**R**=axial stress/Fy)
- the results of the design check. For example:

DESIGN	EQUATION	Factors	Values	Result
Axial Force (E2-1)	$\frac{Pu}{0.85AgFcr} < 1.00$	(kL/r)x =68 (kL/r)y =197 λc = 2.21 QS*QA =1.00	Pu = 430.1 Ag = 16.8 Fcr = 45.72	.66

The design check information is displayed in five columns:

Design : the strength or serviceability state being checked (e.g. Shear, Moments, Deflection, etc.) and the relevant Code clauses.
Equation : the governing Code equation.
Factors : the value of the variables required to calculate the terms in "Equation".
Value : the value of the terms in "Equation".
Result : the ratio of the actual moment/shear/deflection/etc. to the member capacity for the same state.
In the example above, the axial force is 430.1 and the capacity of the member is 652.9; "**Result**" = 430.1/652.9 = 0.66

Note:
- Slenderness:
  The program calculates an equivalent (reduced) slenderness for members with a variable axial force

and displays an additional line:

Slender. reduct. x 0.*xx*

- Classification:
  The program displays the worst classification in the case of unsymmetric sections subject to both positive and negative moments, even if the section was designed at every $^{1}/_{10}$ of the span.

- Shear:
  The program designs shear stiffeners if the result for shear is < 1.00.

Units:

- Section data:
  inch - if ***STRAP*** default length unit = feet or inch
  centimeter - for all other ***STRAP*** default length units.
- Forces and moments: according to ***STRAP*** default length and force units.
- Stresses:
  ksi (kips/in²) - if ***STRAP*** default length unit = feet or inch
  N/mm² - for all other ***STRAP*** default length units.

Refer also to Steel codes - calculation method.

7.17.2.1 Joists

K-series, LH/DLH joists:

Results for design combination

$W = \max(8M/L^2, 2V/L)$

W_{dl} = black value in SJI table (×0.9×1.65 for LRFD)

DESIGN	EQUATION	Factors	Values	Result
Moment	W / Wdl < 1	M = 6.75 V = 4.50	W = 1500.00 Wdl = 1582.15	0.95
Deflection	d < L / 300 (W / Wll < 1)	for L/360 Wll = 1250.88	W = 1500.00 Wll = 1501.05	1.00

W_{ll} = red value in SJI table (×0.9×1.65 for LRFD)

$W = 8M/L^2$

W_{ll} = Wll in "Factors" column × 360/(allowable defl) = 360/300

KCS joists:

DESIGN	EQUATION	Factors	Values	Result
Moment	M / Mcap < 1	M = 6.75	Mcap= 10.72	0.63
Shear	V / Vcap < 1	V = 4.50	Vcap= 4.65	0.97
Deflection	d < L / 300	d = L/ 994	d = 0.00603	0.30

actual unfactored deflection

Values in SJI table (×0.9×1.65 for LRFD)

7.17.3 Section summary

The program sums the length and weight of each section selected. For example:

Section	Total length	Weight
UB 127x76x13	123.10	1.601
UB 152x89x16	10.04	0.161
UB 178x102x19	7.00	0.133
18 beams were not assigned a section		
Total weight:		1.896

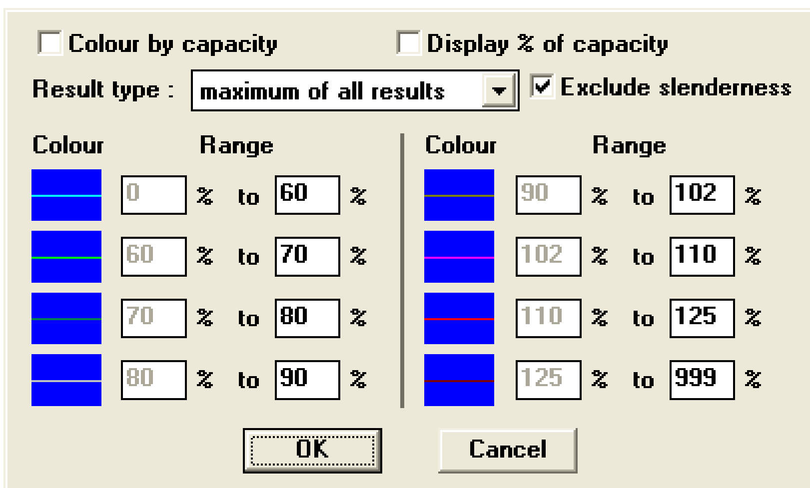
7.17.4 Selected sections

The program writes the name of the steel section selected adjacent to the member.

7.17.5 Display capacity

Use this option to display the computational results graphically on each member in the model.

The results are displayed in the form of the % ratio of the load to the member capacity. The percentage may be written alongside the member and the member and the percentage are displayed with a colour that represents a specified range of capacity percentage.



Colour by capacity

Display the member and the capacity percentage according to a colour code.

Display % of capacity

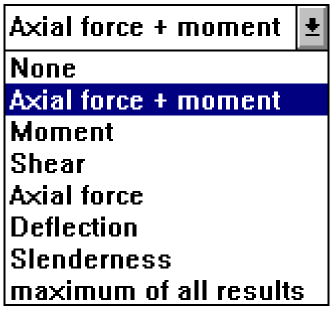
Display the ratio of the actual force/moment/etc. to the design capacity.

For example, if the design axial load on a member is 31 kN and the computed compression resistance, Pc, equals 50 kN, the program displays "62%" alongside that member.

The ratio is displayed for the result type selected in the following option.

Result type

Select the result type to be displayed:



Note:

- for **Maximum of all results**, the slenderness ratio may be excluded.
- for a detailed explanation of each type, refer to Result summary 756.

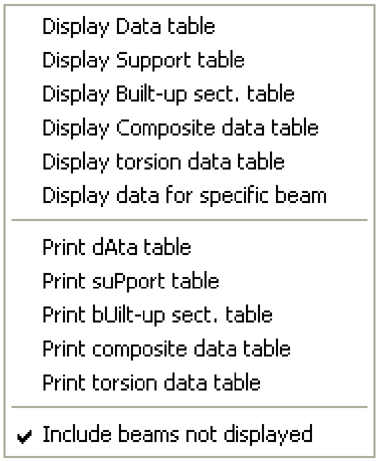
Colour / range

This option allows you to specify a range of result percentages for each colours. The colours are specified in the Setup option of the ***STRAP*** main menu.

Select a range; enter the new upper value. The program automatically updates the lower boundary of the following range.

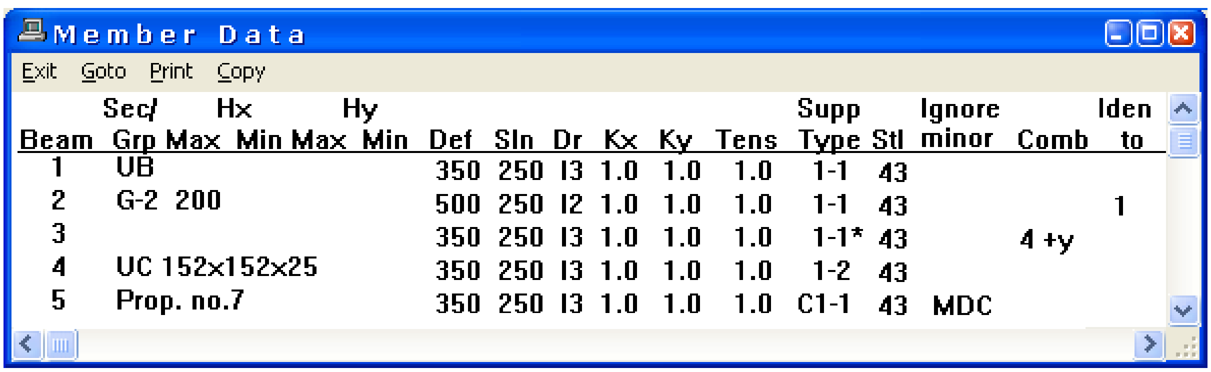
7.18 Data tables

This option displays all input data in tabular form. To display the data in graphic form, use the Display 750 option.



7.18.1 Data table

To display all input data except supports. For example (BS5950):



Beam	Sec/ Grp	Hx Max	Hx Min	Hy Max	Hy Min	Def	Sln	Dr	Kx	Ky	Tens	Supp Type	Stl	Ignore minor	Comb	Iden to
1	UB					350	250	I3	1.0	1.0	1.0	1-1	43			
2	G-2	200				500	250	I2	1.0	1.0	1.0	1-1	43			1
3						350	250	I3	1.0	1.0	1.0	1-1*	43		4 +y	
4	UC 152x152x25					350	250	I3	1.0	1.0	1.0	1-2	43			
5	Prop. no.7					350	250	I3	1.0	1.0	1.0	C1-1	43	MDC		

where:

Sec/Grp : Indicates the limitations imposed on the section selection.

Hx / Hy : minimum/maximum allowable section dimensions (mm.) - OR - name of section to be checked.
In the table above -
Beam 1 : selection is limited to UB sections in the model table; no dimension limits.
Beam 2 : selection is limited to all sections included in Group 2; maximum HX = 200 mm.
Beam 3 : blank: beam is not designed as its property is not a shape or built-up section.
Beam 4 : the program checks UC 152x152x25.
Beam 5 : the program checks built-up section no. 7.

Def : Allowable deflection expressed as L/"DEF". The allowable deflection for beam 2 is L/500.

Sln: Allowable compression slenderness (kl/r)

Dr: **I2** or **I3** indicate the major axis direction and the **+/-** preceding them indicate the flange location.

Referring to the Major/minor - Flange location 715 options -

I2 or **I3:** no +/-, indicates the default option - **Worst flange location -** in both directions

+**I2** or+**I3 :** location of flanges at +x2 and +x3

-**I2** or -**I3** : location of flanges at -x2 and -x3

+-**2** or +-**3**: location of flanges at +x2 and -x3 (2/ 3 indicates the major axis direction - I2/I3)

-+**2** or -+**3**: location of flanges at -x2 and +x3 (2/ 3 indicates the major axis direction - I2/I3)

Kx, Ky: Effective length factor for compression buckling.

Tens: Net tension area factor (Ae = Ag * Tens)

Supp Type: End support conditions

BS5950: C = cantilever, e.g. BEAM 5 above
* = destabilizing load, e.g. BEAM 3 above.
fixed,pinned : 1st number = condition at JA; 2nd number = condition at JB.
cantilever: A/B = location of free end (JA/JB)
1st number = condition at support;
2nd number = condition at tip.

Eurocode, IS800-07: K = 1.00C - K value for LTB = 1.00; cantilever beam
K=0.7 - K value for LTB = 0.7
* = destabilizing load, e.g. BEAM 3 above.

AISC - ASD: CANT - cantilever beam

AISC - LRFD: MAIN - main member
MAINC - main member; cantilever
SECOND - secondary member
SECNDC - secondary member; cantilever

IS:800-84: C = cantilever, e.g. BEAM 5 above
* = destabilizing load, e.g. BEAM 3 above.
fixed,pinned : number = restraint condition number
cantilever: a/b = location of free end (JA/JB)
number = restraint condition number for cantilever
f = degree of fixity at the free end.

IS:802: number = restraint condition number (1-5) according to the program options

Steel = Steel grade (dependent on Code selected).

Ignore Minor Displays which minor axis results are to be ignored when designing a space model member, where:
M = ignore minor axis moment
D = ignore minor axis deflection
C = ignore minor axis when calculating section classification.
In beam 5, all three items are ignored.

Comb= The member which this member is combined with to form a single design unit.
+/-/X/Y = *combined* directions (not the support directions).

For example, beams 3 and 4 are combined in the top flange bending and minor axis buckling (+Y) directions, i.e. supports were defined for bottom flange bending and major axis buckling.

Iden to = The member which this member must be identical to.
For example, beams 1 and 2 must have the same section.

7.18.2 Support table

Display all of the defined intermediate supports (bending or lateral-torsional buckling). For example:

Support/Restraint Data (units=meter)

Beam no.	Continuous	Additional Supports 1	2	3	4	5	6
4	+xy						
5	+xy						
6	+xy						
16		2.50 y					
18		2.50 +xy					
20		4.30 xy					
22		4.30 xy					
28		1.99 +xy					

Intermediate support points are defined at -
Continuous = continuous along entire length of beam.
Additional = at any user defined distances from JA

The directions the support is applied are indicated by **+/-/X/Y.** Refer to Supports and combined beams 729. for a detailed explanation.

7.18.3 Built-up section table

Display / print the built-up section table, including section dimensions and properties.

7.18.4 Composite data table

Composite sections data table

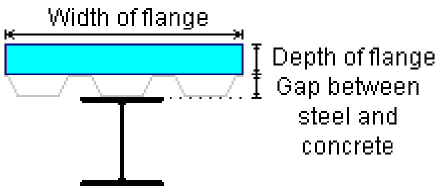
beam	composite	conc. heig. (mm.)	conc. width (mm.)	conc. dist. (mm.)	stud force (kN)	number of studs	conc. fc (mPa)	steel area (mm.)	steel fy (mPa)	top. side	P	My
11	col.	+30	+30				30.0	0	340.0			
15	col.	340	300				30.0	0	340.0			
16	col.	filled	secti				30.0	0	340.0			
20	beam	120	3000	0	76.0	Compute	30.0	0	340.0	+X2	No	No

Height / width / distance

For composite columns, the table displays

- **Encased columns**:
  - cover defined, e.g. Column 11, where cover = 30
  - size defined, eg. Column 15, where size = 340x300
- **Filled section**
  - e.g. Column 16

For composite beams:



Studs

The total number of studs required for full capacity.

Concrete

Nominal concrete strength.

Steel

- total reinforcement area in the slab
- nominal reinforcement steel strength

Topping

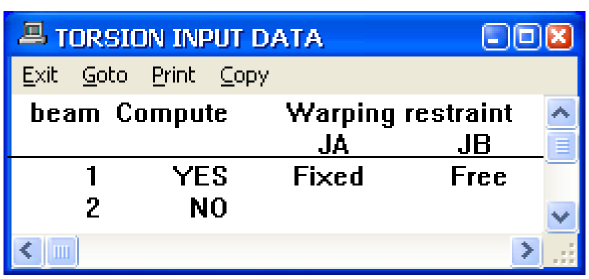
Topping location

P / My

No = axial / minor axis moments are ignored.

7.18.5 Torsion data table

Display the current torsion parameters. For example:


TORSION INPUT DATA

Exit Goto Print Copy

beam	Compute	Warping restraint JA	JB
1	YES	Fixed	Free
2	NO		

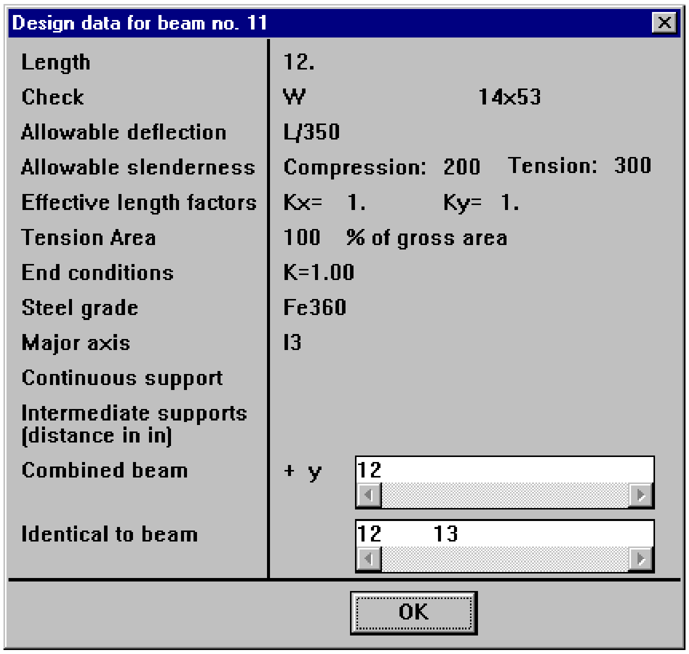
where:

- **Compute**
  YES : Compute torsion/warping for the beam
  NO : Ignore torsion moment for the beam
- **Warping restraint JA/JB**
  Fixed : End is fixed for warping, i.e. normal stresses are generated
  Free : End is free to warp, i.e. normal stresses are ***not*** generated

Refer also to Torsion - general 781

7.18.6 Data for specific beam

Use this option to display ***all*** of the data for a specific member; to select the member, move the crosshair until the member is highlighted with the rectangular blip; click the mouse. The following table is displayed on the screen:

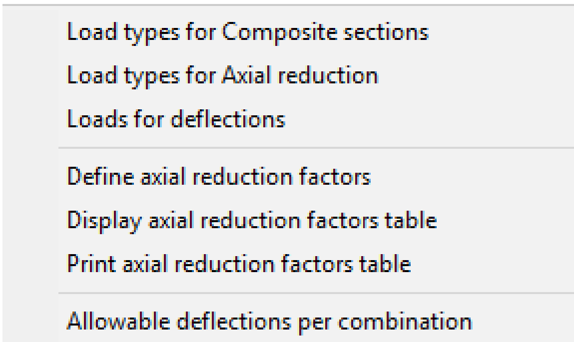


For detailed explanations on the items in the tables, refer to Display data table 763 and Display supports table 765 .

7.19 Loads

Certain design parameters are dependent on the load case/combination that is computed by the program:

- axial load reduction factor: "live" load cases must be identified
- composite beams: load cases may be applied to the steel beam or to the composite beam
- deflections: different allowable deflections may be required for different load combinations



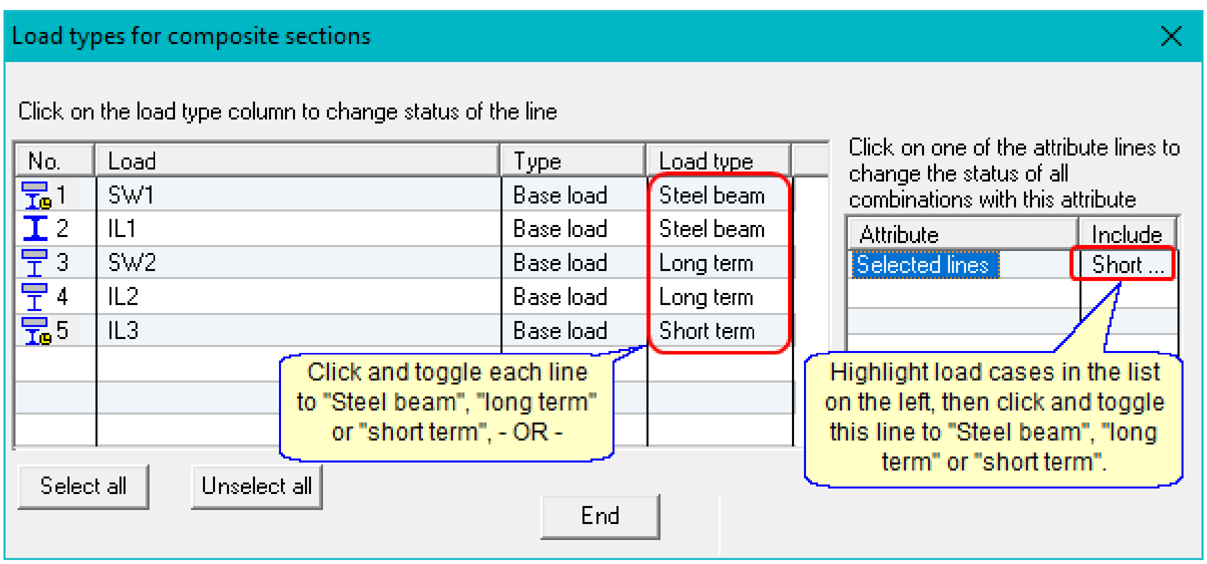
Load types for composite section

Loads on composite beam may be applied at two different stages:

- to the non-composite steel beam
- to the composite steel and topping beam, either as a short-term load or as a long term load

The load application stage may be specified for each load case.

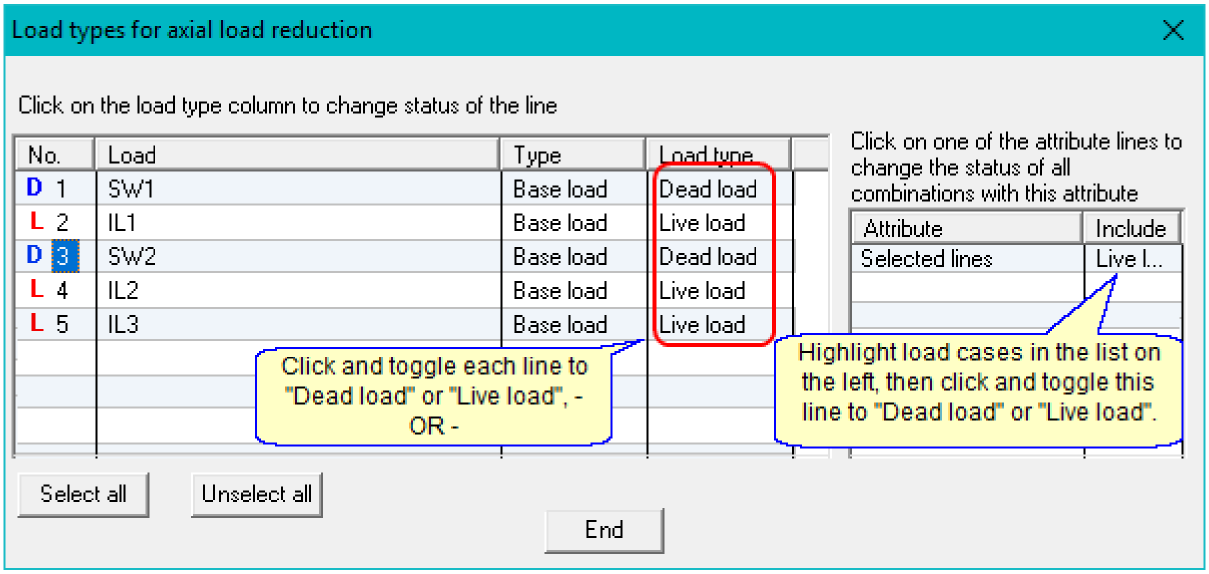
This information is required for all calculations where the elastic modulus (I/y) is used, deflection calculations, etc.



Load types for axial reduction

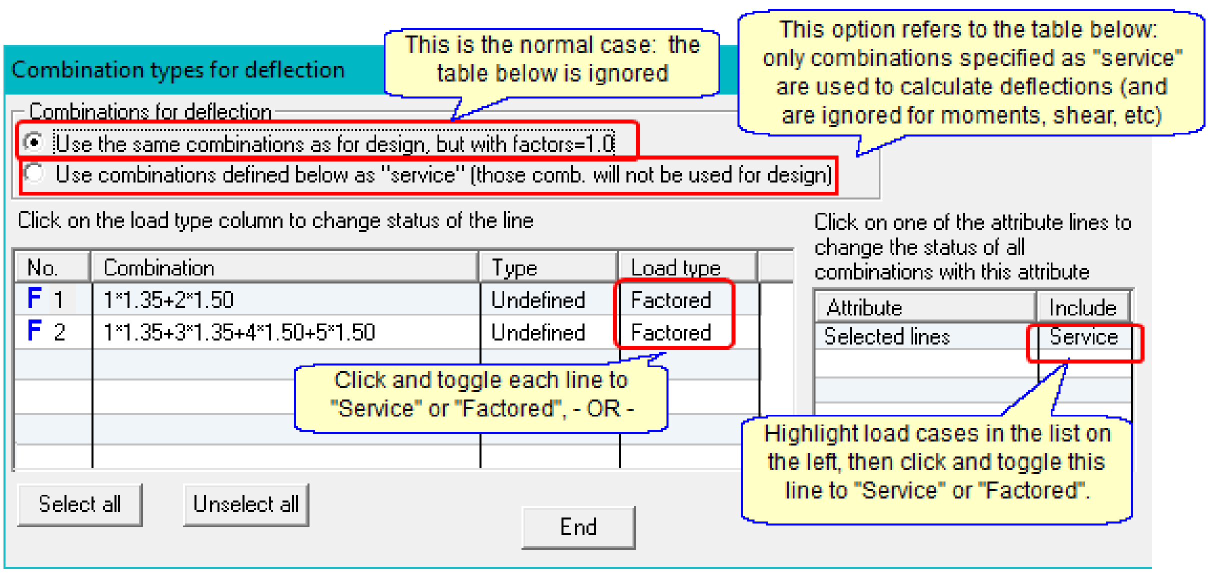
Most design codes allow the axial live load in a column to be reduced if the column supports a large area.

Use this option to identify the live load cases:



Loads for deflections

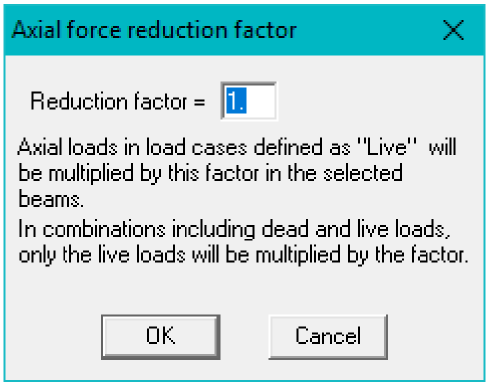
Specify which combinations are to be used to calculate deflections. When there are no non-linear cases (P-Delta, etc), it is usual sufficient to use the design combinations with a factor of 1.0 for all load cases. However when non-linear calculations are done and the load cases are already Factored, different load combinations are usually defined to calculate the deflections.



Axial reduction factor

Define the live load reduction factor for selected columns:

- select columns using the standard Beam selection option
- define the factor (normally < 1.00)

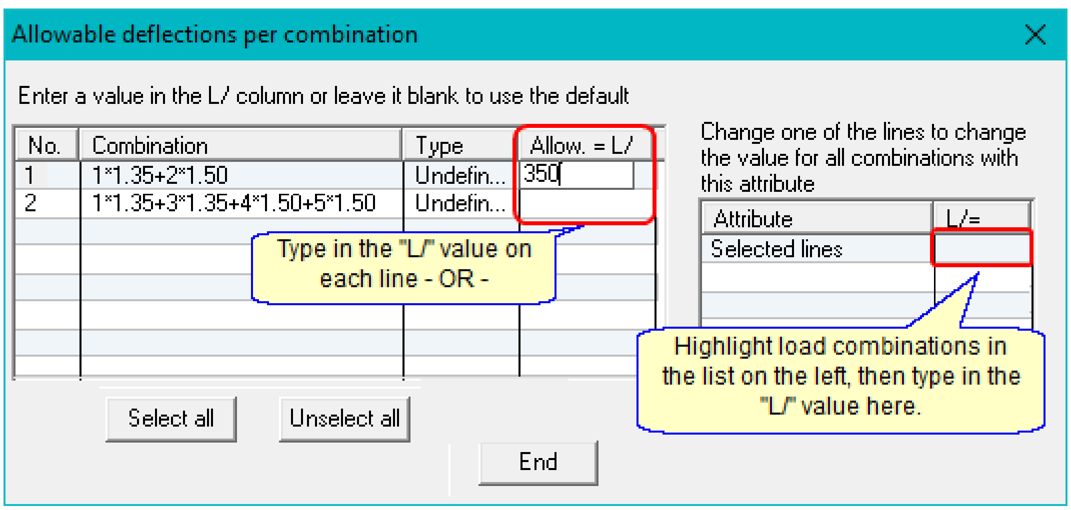


The live load cases are specified in the "**Load types for axial reduction**" option.

Allowable deflections per combinations

Different allowable deflections may be required for different load combinations, depending on the type of loading. The Default 702 and Parameters 717 options specify the same allowable deflection for all load combinations.

Specify the allowable deflection for each combination:



Note:

- if the allowable deflection for a combination is not defined in this option, then the program uses the value defined in Default 702 or Parameters 717.
- if the allowable deflection for a combination is defined here and an allowable deflection for a specific member was defined in Parameters 717, the program multiplies the value defined here by (parameter/ default).

For example:

- default deflection = L/300
- parameters deflection for beam *n* = L/500
- allowable deflection for load combination *k* specified in this option = L/400

The deflection check is carried out as follows:

- for all beams other than *n*:
  - for all combinations other than *k*: allowable deflection = L/300
  - for combination *k*: allowable deflection = L/400.
- for beam *n*:

- for all combinations other than *k*: allowable deflection = L/500
- for combination *k*: allowable deflection = L/(400*500/300)=L/667

7.20 Sway

This option is active only if :
- sway loads were defined and solved (Refer to Loads - sway.)
- the Compute 744 option was selected in the Steel design module.

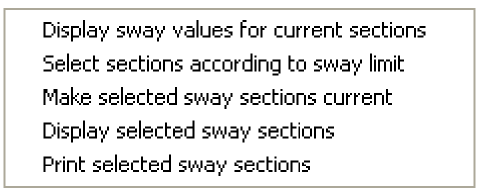
The sway option revises the section properties of beams in the model in order to reduce the sway at selected nodes (or the drift between two selected nodes) to user defined limits.

The program searches for the member that most significantly decrease the sway/drift, enlarges the section and recalculates the sway/drift. The calculation is repeated until the sway/drift is reduced to the limiting value.

The nodes at which the sway/drift is calculated must be specified during load definition; unit load cases necessary for the algorithm are defined at the relevant nodes. Refer to Loads - sway.

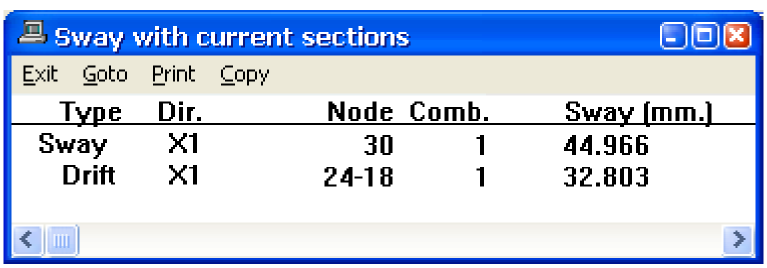
Note the following terminology:
- **Current sections:**
  sections selected by the steel design module **Compute** option
- **Sway sections**:
  sections selected by this option.

For the recommended procedure for using the Sway option and more detailed information, refer to Sway - general 773.



Display sway values for current sections

The deflections at the sway/drift nodes in the relevant global directions are displayed:



Note:
- The deflections are for the current sections, not the sway sections, i.e. the sections selected or checked by the **Compute** option of the Steel design module or the sway steel sections transferred to the current sections. All other sections are taken from the *STRAP* geometry.
- The program automatically recalculates the sway values when new current sections are selected. The calculation uses an algorithm based on the results obtained from the original *STRAP* geometry and hence the new values will be slightly inaccurate. If there is a significant relative difference between the original *STRAP* geometry sections and the current sections, then the user should consider solving the model again using the new sections.

Select sections according to sway limit

Specify the sway/drift limits and start the calculation. Refer to Select sections 775.

Make selected sway section current

This option sets the steel design module current sections equal to the sway sections. The sway sections calculated by the sway/drift option are saved by the program but are not used by the Steel module until this option is selected. i.e.

- all steel module result options display results for the sway sections
  - o the sway sections can be transferred to the *STRAP* geometry

Note that this option applies only to steel sections selected according to the **Automatic section selection** option.

Display/print selected sway sections

Display the results of the sway calculation. For example:

- steel sections

Section selection for sway

Exit Goto

Beam	Old section	New section	volume addition	addition *cost
26	IPE500	IPE600	0.012	0.012
27	IPE550	IPE600	0.007	0.007
28	IPE500	IPE600	0.012	0.012
29	IPE400	IPE600	0.021	0.021

- sections defined by properties:

Beam	Old section		New section	volume addition	addition *cost
Property 1	A=0.3340E-02	I2=0.2050E-05	I3=0.2770E-04		
Optimal:	A=0.8440E-02	I2=0.5180E-05	I3=0.4470E-03	0.510	2.550
Property 2	A=0.2500E+00	I2=0.5208E-02	I3=0.5208E-02		
Optimal:	A=0.2500E+00	I2=0.5208E-02	I3=0.5208E-02	0.000	0.000
TOTAL				0.510	2.550

where:

volume addition = length of member (or sum of lengths of members in property group) multiplied by the section area

addition*cost = volume addition multiplied by the cost factor defined for the property group.

7.20.1 General

Sway is a module that allows the engineer to design a structure for sway, drift and/or deflections.

The sway option can automatically revise the section properties of beams and columns to reduce the sway at selected nodes and/or the drift between two selected nodes to user defined limits. Sway can be the controlling factor even in buildings of a moderate height (as little as 6-8 floors). To have the ability to optimize your structure for sway and drift automatically using the least weight of steel or volume of concrete can produce substantial savings even on a medium size project.

The sway option can also be used to reduce the deflection at any node to a specified value. For example, a truss is designed and all stresses are found to be within the Code requirements. However the deflection of the truss needs to be reduced. The Sway module provides information as to which members can be enlarged most economically. If this is a steel truss (light gauge, rolled sections or both), the Sway module optimizes all sections automatically, if requested by the engineer.

The program searches for the member that most significantly contributes to the sway/drift/deflection, enlarges its section and recalculates the sway/drift/deflection. The calculation is repeated until the sway/drift/deflection is reduced to the limiting value.

The Sway option is part of the Steel design module:
- the user can immediately check the effect of section selection on sway/drift values
- sections selected by the sway option can be immediately checked by the Steel design module.
- the dimensions of concrete sections may be enlarged by the sway options, i.e. this module may be used to limit the sway in concrete models or mixed models consisting of steel and concrete members.

Recommended procedure for using the the sway option:

STRAP:
- Define loads and combinations as usual.
- specify the nodes at which the sway is to be checked (the program applies unit loads at these nodes automatically)
- Solve the model.

Steel design module:
- First complete the design of beams and columns for stresses, member deflections, and other parameters. Define sections groups, sections to select from, identical sections, intermediate supports for lateral-torsional buckling, etc.
- select the **Compute** option (the Sway option is grayed if the model has not been Computed).

After you complete the design of the beams and columns to Code you can proceed to the Sway option.
- select **Select sections according to sway limit** in the **Sway** pulldown menu
- define the sway limits and specify additional parameters (refer to "Parameters" below)
- click the **OK** button to start the calculation
- select **Display selected sway sections** in the **Sway** pulldown menu; review revised sections.
- revise parameters and recalculate, if necessary, until satisfactory results are obtained.

Note that although larger steel sections are selected by this option, there is no guarantee that the new sections comply with all of the steel Code requirements. To check the new sections:
- select **Make selected sway sections current** in the **Sway** pulldown menu
- select **Compute** in the Steel design module side menu to check the new sections

or:
- set ☑ **Check new selected sections for stress** in the Sway parameters menu and select sway sections again. Sections that do not comply with Code requirements are automatically enlarged.

If the section properties were significantly revised by the Sway option you may want to solve the model again:
- select **transfer new sections to *STRAP* geometry**
- return to the *STRAP* main menu and solve the model again
- return to the Steel design module and repeat the procedure outlined above.

Parameters:

- Selection method:
  The user can specify for members in each property group:

  ◉ **Automatic section selection:**

The program enlarges steel sections using the following steel design module data:

- only members with a **type** or **group** assigned to them are considered. All other members (**check** or non-steel section property) are ignored.
- the program considers the following parameters when determining the sway section: **same sections**, **major/minor axis dimension limits.**

Calculate optimum area
The program ignores all data defined in the Steel design module (type, group, identical, etc) and increases the section properties for all members in the property group. This option is also used for concrete sections.

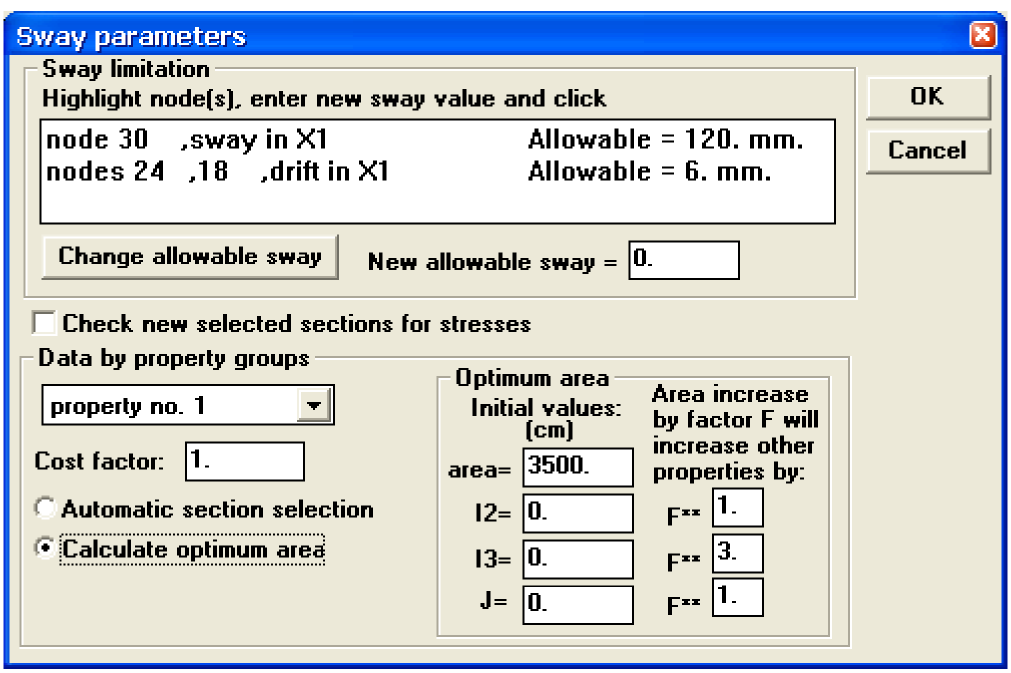
- Cost factor:
The user can define a cost factor for each property group in the model. If cost factors are defined the solution represents the minimum "cost factor * volume increase" for the model, i.e. the most economical solution. Therefore members in property groups with a low factor have priority in the calculation and will more likely be enlarged. For example, you can assign different cost factors to beams and columns. Conversely, define an arbitrarily large factor for a property group that you do not want to enlarge under any circumstances.

Additional information:

- Selected sway sections are stored separately from the sections selected by the Steel design module until they are made “current” by this option. Once the sway sections are current the user can display the detailed results for them or transfer them to the *STRAP* geometry.
- The calculation is approximate as it is calculated by modifying the solved deflection results, i.e. based on the section properties specified in *STRAP* geometry. If the enlarged sections are significantly different from the original *STRAP* sections, transfer the sections to the geometry, solve the model again and repeat the process in order to obtain the exact deflection results.

7.20.2 Select sections

Specify the sway/drift limits and start the calculation. Specify the following parameters and click the OK button to start the sway calculation.



Sway limitation

Define the new sway/drift limits at the nodes:

- highlight line(s) in the list box with the data for the relevant node(s)
- Type in a value in the **New allowable sway** edit box.
- Click the **Change allowable sway** button

The program updates the list box.

Check new selected sections for stresses

Select one of the following options:

☐ The program selects the sections based solely on the sway/drift limits. The selected sections are not checked for compliance with the steel Code design requirements (moments, shear, LTB, etc) as in the **Compute** option. This is the faster option.

☑ The program checks all sections selected according to the sway/drift requirements for compliance with the Code design requirements. This is the slower option and the calculation time is significantly increased for large models with many load combinations.

Note that sway sections selected using this option automatically become the current sections.

Recommended procedure:

- set this option to ☐ and select sway sections. Check results, revise parameters, repeat selection, etc.
- set sway sections to "current sections" and display "Result summary" table; if any of the new sections do not comply with the Code design checks, then:
- set this option to ☑ and select the final sway sections

Property group

Define the selection mode for each property group:

Select the *STRAP* property group from the pull-down list box and specify the option:

◉ **Automatic section selection:**
The program enlarges the section according the the steel design module data:

- only members with a **type** or **group** assigned to them are considered. All other members (**check** or non-steel section property) are ignored.
- the program uses the following parameters when determining the sway section: **same sections**, **major/minor axis dimension limits.**

◉ **Calculate optimum area**
The program ignores all data defined in the Steel design module (type, group, identical, etc) and increases the section properties for all members in the property group. Refer to optimum area 777 for more details.

Cost factor

Define a cost factor for each property group.

If cost factors are defined the solution represents the minimum (cost factor * volume increase) for the model. Therefore members in property groups with a low factor have priority in the calculation and are more likely be enlarged.

Conversely, define an arbitrarily large factor for a group that you do not want to enlarge under any circumstances.

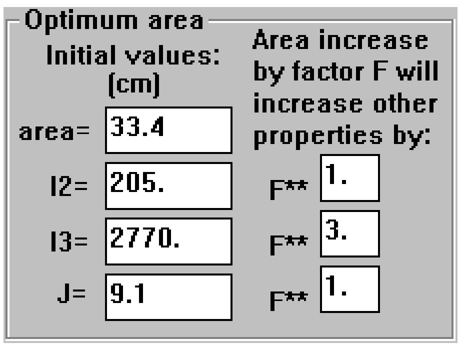
Example:
The cost of enlarging a column is 1.5 times greater than the cost of enlarging a beam and a cost factor of 1.5 is defined for column property group:

- the program calculates the change in sway resulting from a unit weight added to the beams and from a unit weight added to the columns.
- if the change in sway resulting from the enlarged column is more than 1.5 times the change due to the enlarged beam, then the program changes the column section. Otherwise the beam is enlarged.

Optimum area

These options are displayed only if **Calculate optimum area** is selected.

The program ignores all data defined in the Steel design module (type, group, identical, etc) and increases the section properties for all members in the property group. New values of area and moment-of-inertia are calculated by the program.



- ***Initial values***:
  The program displays the current property values (area, moments-of-inertia) for the selected property group. The program starts the sway calculation from the values in these edit boxes, i.e. smaller sections will not be selected; different values may be entered.

- ***Area***:
  The program increases the section **area** by incremental values when calculating new section properties to limit the sway. Specify the increase in the moments-of-inertia when the area is increased by a factor '**F**'.

  The program assumes a rectangular section by default, i.e. **I = bh^3/12.** Therefore, **I3** is increased by **F** ^{3} and **I2** by **F.**

Note:

- **Current sections:**
  sections selected by the steel-design module Compute 744 option
- **Sway sections**:
  Section selected by this option.

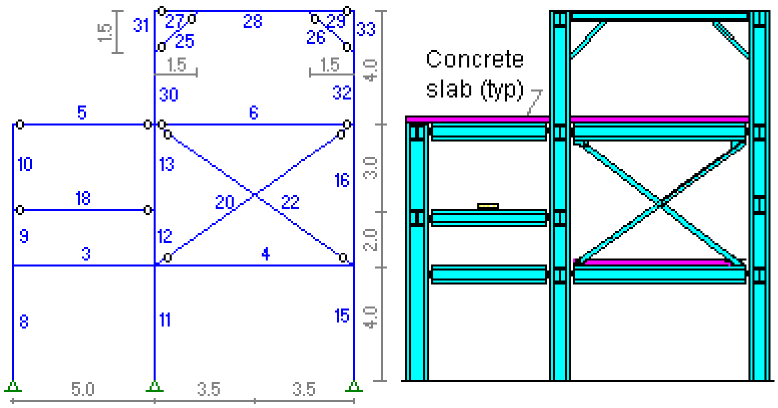
7.21 Example

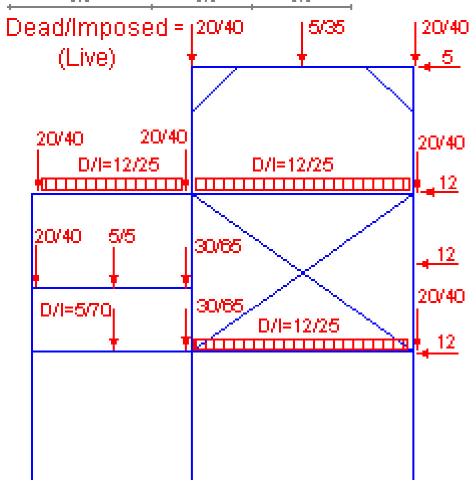
Design the following plane frame.

Note:

- This example illustrates the method of application of several options. The example is intended as an aid to learning the proper use of the program options, ***and is not intended as a guide to proper engineering judgment in the construction of a model for design***.

The ***STRAP*** geometry, loads and combinations are:





Load cases

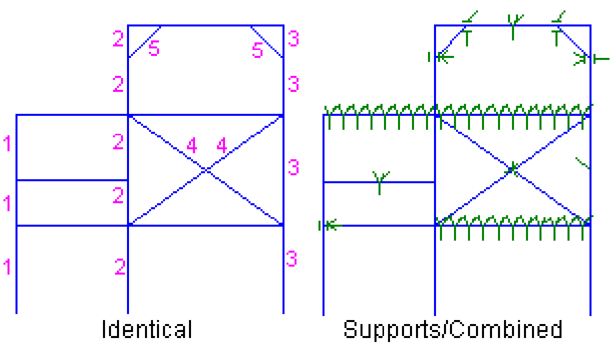
1 Dead

2 Live (imposed)

3 Wind (all horiz.)

Load combinations	for AISC-ASD
1=1.4D+1.6L	1=1.0D+1.0L
2=1.2D+1.2L+1.2W	2=0.75(D+L+W)
3=1.2D+1.2L-1.2W	3=0.75(D+L-W)
4=1.0D+1.4W	4=0.75(D+W)
5=1.0D-1.4W	5=0.75(D-W)

After the geometry and loads were defined in *STRAP* and the model was solved, the following data was defined using the Steel design module options:



***Section*:**

Limit selection to:

Code	BS5950	Eurocode 3	AISC
Columns	UC	HEA	W
Beams	UB	IPE	W
Diag. bracing	EQ.D.ANGLE	L	2L
Haunch beams	CHANNEL	UNP	C

Same section:

- Column 8-9-10 specified as an identical group
- Column 11-12-13-30-31 specified as an identical group
- Column 15-16-32-33 specified as an identical group
- Bracing 20 and 22 specified as an identical group
- Haunch beams 25 and 26 specified as an identical group

Supports:

- Members 4,5,6: "Continuous" support for +z major axis bending, major and minor axis buckling. The support is provided by the floor slab. The continuous buckling support for both axes cancels the axial force check.
- Member 16: Minor axis buckling support at mid-span. The support is provided by the beam perpendicular to the plane of the frame.
- Member 18: Support at midspan for +z major axis bending, major and minor axis buckling. The support is provided by the walkway passing over the beam. This support may be defined using the option "Define supports at concentrated load points".
- Members 20,22: Support at the midspan of the bracing for major and minor axis buckling (assumes that the bracings are attached).
- Member 28: Support at midspan for +z major axis bending, major and minor axis buckling. This support may be defined using the option "Define supports at concentrated load points".

Combined sections:

- Members 8,9: Beam 3 provides buckling support only for the major axis of this column, i.e. members 8 and 9 act as a single unit for minor axis buckling. Define major axis buckling and -z bending supports at the common node.
- Member 27,28,29: These three members form a single beam. Define major axis buckling and -z bending supports at the common node.
- Member 30,31: Haunch beam 25 provides buckling support only for the major axis of this column, i.e. members 30 and 31 act as a single unit for minor axis buckling. Define major axis buckling and -z bending supports at the common node.
- Members 32,33: Similar to 30, 31. Define major axis buckling and +z bending supports at the common node.

Data tables:

The data tables for AISC - LRFD for the above input are: (the tables for the other Codes will be slightly different).

Member Data

Exit Goto

Beam	Sec/ Grp	Hx Max	Hx Min	Hy Max	Hy Min	Def	Sln	Dr	Kx	Ky	Tens	Supp Type	Steel	Comb	Iden to
3	W					360	200	I3	1.00	1.00	1.00		A36		
4	W					360	200	I3	1.00	1.00	1.00		A36		
5	W					360	200	I3	1.00	1.00	1.00		A36		
6	W					360	200	I3	1.00	1.00	1.00		A36		
8	W					360	200	I3	1.00	1.00	1.00		A36	9+y	
9	W					360	200	I3	1.00	1.00	1.00		A36		8
10	W					360	200	I3	1.00	1.00	1.00		A36		9
11	W					360	200	I3	1.00	1.00	1.00		A36		
12	W					360	200	I3	1.00	1.00	1.00		A36		11
13	W					360	200	I3	1.00	1.00	1.00		A36		12
15	W					360	200	I3	1.00	1.00	1.00		A36		
16	W					360	200	I3	1.00	1.00	1.00		A36		15
18	W					360	200	I3	1.00	1.00	1.00		A36		
20	2L					360	200	I2	1.00	1.00	1.00		A36		
22	2L					360	200	I2	1.00	1.00	1.00		A36		20
25	C					360	200	I3	1.00	1.00	1.00		A36		
26	C					360	200	I3	1.00	1.00	1.00		A36		25
27	W					360	200	I3	1.00	1.00	1.00		A36	28+	
28	W					360	200	I3	1.00	1.00	1.00		A36	29+	
29	W					360	200	I3	1.00	1.00	1.00		A36		
30	W					360	200	I3	1.00	1.00	1.00		A36	31+y	13
31	W					360	200	I3	1.00	1.00	1.00		A36		30
32	W					360	200	I3	1.00	1.00	1.00		A36	33-y	16
33	W					360	200	I3	1.00	1.00	1.00		A36		32

Support/Restraint Data (units=meter)

Exit Goto Print Copy

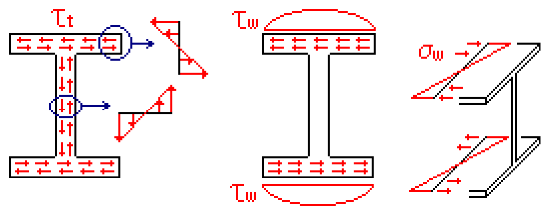
Beam no.	Continuous	Additional Supports 1	2	3	4	5	6
4	+xy						
5	+xy						
6	+xy						
16		2.50 y					
18		2.50 +xy					
20		4.30 xy					
22		4.30 xy					
28		1.99 +xy					

7.22 Torsion - general

The calculation of torsion for hot-rolled structural steel sections is based on the following publication:

AISC - American Institute of Steel Construction
Steel Design Guide Series - 9
"Torsional Analysis of Structural Steel Members"
by P.A. Seaburg and C.J. Carter, 1997

Pure torsion generates shear stresses in the section, while warping generates shear and normal stresses. For example, an I-section:



(a) Pure torsion (b) Warping
Shear stresses Shear stresses Normal stresses

The program carries out the following design checks:

- Shear stresses:

$\tau_{bx} + \tau_{by} + \tau_t + \tau_w = \tau_{allowable}$

where:

τ_{bx}, τ_{by}	=	major/minor axis bending shear stresses. Note that the exact shear stress is calculated at all levels from the equation $\tau_b = V \cdot Q / I \cdot t$ (the average shear stress V/A_v is use for the shear design check)
τ_t, τ_w	=	pure torsion and warping shear stresses, as shown above.
$\tau_{allowable}$	=	the allowable stress for shear.

- Normal stresses:

$$\frac{\sigma_a}{\sigma_{a,all}} + \frac{\sigma_{bx}}{\sigma_{bx,all}} + \frac{\sigma_{by}}{\sigma_{by,all}} + \frac{\sigma_w}{\sigma_{w,all}} \leq 1.0$$

where:

σ_a	=	axial stress
σ_{bx}, σ_{by}	=	major/minor axis bending stresses
σ_w	=	warping normal stress, as shown above.
σ_{all}	=	allowable stresses for axial, bending and warping. Note that the program calculates both the 'local' and 'overall' (second order effects) values for axial and bending and uses the worst value.

For example, AISC-LRFD:

Local:

$$\frac{\sigma_a}{0.85F_{cr}} + \frac{\sigma_{bx}}{\phi_b F_{cr}} + \frac{\sigma_{by}}{0.9F_y} + \frac{\sigma_w}{0.9F_y} \leq 1.0$$

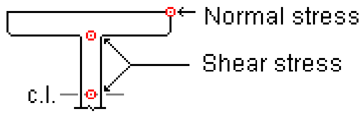
Overall:

$$\frac{\sigma_a}{0.85F_{cr}} + \frac{\sigma_{bx}}{\left(1-\frac{P_u}{P_{ex}}\right)\phi_b F_{cr}} + \frac{\sigma_{by}}{\left(1-\frac{P_u}{P_{ey}}\right)0.9F_y} + \frac{\sigma_w}{\left(1-\frac{P_u}{P_{ey}}\right)0.9F_y} \leq 1.0$$

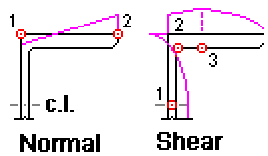
The program carries at the design checks at every 0.10L, i.e. 11 equidistant points along the span length (or each component member of a combined beam).

The stresses are also checked at several points on the cross section:

- I-sections:



- [-sections



  Note:
  - Normal: pt.2 will always govern if "worst case" is specified for flange orientation.
  - Torsion: pt.3 is at point of maximum torsion shear stress

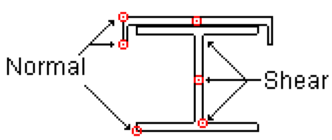
- RHS
  - Plane sections remain plane, i.e. no warping is present.
  - The program assumes that torsional stresses are uniformly distributed over the face of the section; the stresses are checked at the point of maximum τ_{bx}, τ_{by}

- T-sections
  - warping stresses in T-sections are usually negligible and are ignored by the program.
  - shear stresses are checked at the point of maximum τ_{bx}, τ_{by}

- Angles
  - Warping stresses in angles are usually negligible and are ignored by the program.
  - shear stresses are checked at the point of maximum τ_{bx}, τ_{by}

- Pipes
  - Plane sections remain plane, i.e. no warping is present.
  - Torsional stresses are uniformly distributed over the face of the section; the stresses are checked at the point of maximum τ_{bx}, τ_{by}.

- I+[



- Cold-formed and user-defined sections

  The program checks the stresses at both ends of all segments and at any point of maximum τ_b or τ_w along the length of any segment. ***Torsion and warping are not calculated for sections with a closed portion***.

Note:
- Tapered beam / combined beam composed of different sections:
  The calculation of warping and torsion is not exact. When calculating the rotation function θ at any point along the length of the beam, the program assumes that the entire beam has the section properties found at that point.
- Refer also to Design assumptions for each Code.

Part VIII

Concrete design

8 Concrete design

For general information on the Concrete design module, select:
- General [786]
- Creating a concrete structure from a STRAP model [797]
- Design procedure [791]
- Design assumptions [790]

Beams
Columns
Walls
Slabs
Drawings

Select "Beam [786]", "Column [786]", "Wall [786]", "Slab [786]" or "Drawing" mode.
Note:
- the program designs beams, columns, walls and slabs ***independently***
- "Walls" is displayed only if there are wall elements in the current model; "Slabs" is displayed only if there are plate elements in the model.
- the Drawing option currently deals only with slab wall and foundation drawings.

Defaults – Revise the default drawing and design parameters (displayed at the bottom of the screen), including column detailing parameters.

Define – Define continuous beams [786] or columns [786] consisting of a series of *STRAP* members [786]; revise supports and section orientation for these beams/columns; define beam/column names.

Properties – Revise the *STRAP* cross-section properties or define new sections; for columns, define corner bars and reinforcement groups for *STRAP* 'Solid sections'

Parameters – Specify different design parameters for individual beams, columns, slabs or walls (if a parameter is not defined for a specific item, the default parameter is used).

Compute – Design beams/columns/walls/slabs according to the parameters specified in the previous options. ***Only "Defined" beams and columns may be computed***.

Identical – Specify that selected beams/columns/walls must have identical reinforcement.

Specify bars – Specify actual reinforcement in beams/columns/walls. This option is used to calculate the ***capacity*** of these elements and the Capacity/Demand ratio.

For columns only:

Draw – Create column tables or drawings, including elevations, section and bar schedules, according to the column detailing parameters.

Detailing p... – Specify different detailing parameters for individual columns (if a parameter is not defined for a specific column the default parameter is used).

For slabs only:

Design – Calculate and detail reinforcement in elements; create reinforcement drawings according to the user parameters. Refer to Draw slabs - general [794]

For drawings only:

Draw – Create a drawing with slabs, wall sections or foundation plans according to the drawing parameters.

From the menu bar:

File Zoom Rotate Display REmove Draw ReSults Data Display Data Tables Help

Display [976] - display *STRAP* geometry data ***graphically***; display General arrangement [974] drawings.

Results 926	- display and/or print the results. For columns, a different reinforcement arrangement may be specified.
Data display 977	- display concrete parameters and result data graphically.
Data table 980	- display tables showing the input data (design parameters and supports).

8.1 General

The *STRAP* Concrete design module is a program for the design of reinforced concrete buildings.

The program designs beams, columns and walls according to the methods outlined in any one of the following reinforced concrete design codes:

- British Standard BS 8110 - Part 1: 1997, "Structural Use of Concrete".
- Eurocode 2 (EC2) - "Design of Concrete Structures" - Part 1 , 2004.
  Eurocode 8 (EC8) - “Design Provisions for Earthquake Resistance of Structures” - 2003
- ACI 318-02 - "Building Code Requirements for Reinforced Concrete"
- CSA A23.3-94 (Canada) - "Design of Concrete Structures"
- IS:456-2000 (India) - "Code of Practice for Plain and Reinforced Concrete"
  IS:13920 - 1993 - “Ductile Detailing of Reinforced Concrete Structures Subjected to Seismic Forces”
- NBr 6118-2014 (Brazil)
- AS3600 - 2009 (Australia)

Refer to Design assumptions 790 for more details on each code.

Note that the manual uses the following terminology:

MEMBER - a *STRAP* beam element

BEAM - a continuous beam consisting of a series of connected ***members*** defined in this module. "***Beams***" are defined by the user.

COLUMN - a column consisting of a series of connected ***members*** defined in this module. "***Columns***" are defined by the user.

WALL - refers ***only*** to a wall defined using the *STRAP* "Wall" option; ***Walls*** are identified automatically by the program. Quad and triangular elements cannot be designed as walls.

SPAN - a span (between supports) in a **BEAM** or **COLUMN** that may be comprised of more than one ***member*.**

SLAB - a plane of quad and/or triangular elements; ***Slabs*** are identified automatically by the program.

The terms **STIRRUPS** (American) and **LINKS** (British) are identical.

8.2 Seismic - general

This section explains the general principles of seismic design for reinforced concrete frames common to all design Codes. For detailed information pertaining to a specific Code, refer to Design assumptions.

In general, seismic design must insure minimum levels of ductility in the beams and columns and so has more stringent requirements for minimum reinforcement. In addition, much of the design is based on the moment ***capacity*** of the members rather than the design forces calculated by the analysis. The moment capacity must be calculated from the actual reinforcement in the beams/columns and so the program allows the user to increase the theoretical areas to the actual areas as detailed.

The design procedure is summarized as follows:

- beam longitudinal reinforcement is calculated from the design forces, but not less than the minimum specified by the Code
- beam shear reinforcement is calculated from shear forces derived from the moment capacity of the beam (but not less than the design forces).
- column longitudinal reinforcement is calculated from the design forces, but may be increased to ensure that the sum of the column moment capacities at a joint exceeds the sum of the beam moment capacities.
- column shear reinforcement is calculated from shear forces derived from the moment capacity of the beams connected to the ends of the columns (but not less than the design forces).

This method ensures a hierarchy of strengths of the different members. Note that three different moment capacities are calculated by the program:

- factored : normal capacity for non-seismic members
- nominal : capacity calculated using concrete and steel strengths not reduced by Code factors
- probable : capacity calculated using increased steel strength, i.e. actual conditions

and each calculation uses the appropriate capacity.

The design of both longitudinal and transverse reinforcement in columns is dependent on the capacity of the beams. Therefore, it is mandatory to compute the beams prior to computing the columns. Refer to Design procedure - seismic 793 for more details.

Beams:

Moment:

The beams are designed for all load combinations as defined by the user.

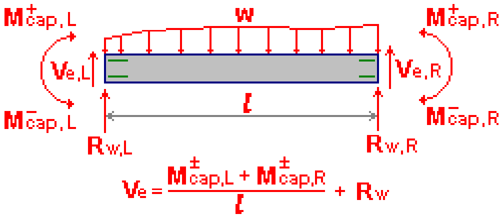
In addition, the program complies with the following requirements found in all Codes:

- at support, the positive moment capacity is not less than a specified percentage of the negative moment capacity
- at any point along the beam, the positive and negative moment capacity is not less than a specified percentage of the negative moment capacity at the support

The program adds additional top/bottom reinforcement as required.

Shear:

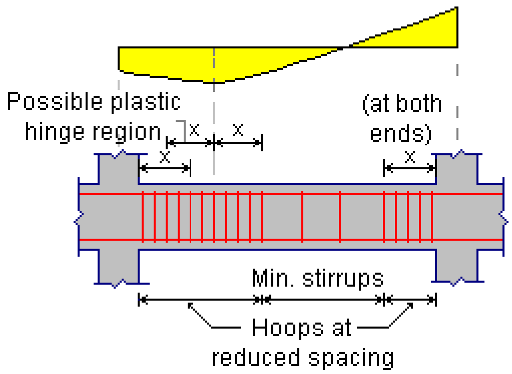
The seismic design shear forces (Ve) are calculated from the probable moment strength of the beam together with the factored beam loads:



The program also checks the beam for the design shear from all load combinations.

Note that **Mcap** is calculated from the actual reinforcement; it is important that the user increase the theoretical values to reflect the actual detailing.

Stirrups (links) are calculated from **Ve** , subject to the minimum requirements in the Code. In general, closed hoops at reduced spacing are required at all locations where plastic hinges may form (and to a distance x' beyond):



Columns:

Moment and axial load:

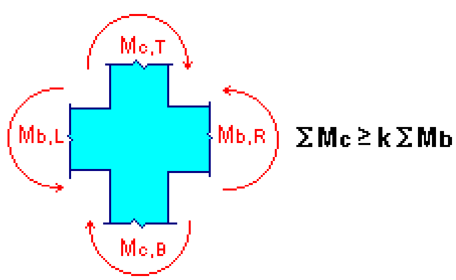
The columns are designed for all load combinations defined by the user. Column areas are not 'reduced' for lightly loaded columns.

In addition, all codes require that the sum of the column flexural strengths at a column-beam joint exceed the sum of the nominal beam flexural strengths (strong column - weak beam):

Σ**Mc = k** Σ**Mb**

where

- **Mb** is calculated from the ***actual*** beam reinforcement at the support (may be increased by the user).
- **k** is specified by the Code.



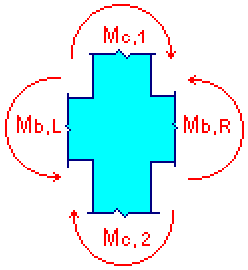
The program ensures that the columns comply with this requirement using the following procedure:

- Σ**Mb** is calculated at the joint
- the moment is then apportioned to the columns above and below according to their relative stiffness
- the resulting moment is then applied as a separate load 'case' at the column top/bottom along with the factored axial load; additional reinforcement is automatically added to the column if this requirement governs.

These load cases are marked as 'seismic' in the column extended detailed results.

Shear:

Similar to beams, the seismic design shear forces (Ve) are calculated from the probable moment strengths of the beams framing into the columns. The sum of the beam moment capacities at a joint are apportioned to the columns above and below according to their stiffnesses:



$$M_{c,1} = \Sigma M_b \times \frac{\text{column stiffness,1}}{\Sigma\,\text{column stiffnesses}}$$

and $\mathbf{V_e = (M_{c,T} + M_{c,B})/L}$

Stirrups (links) are calculated from $\mathbf{V_e}$, subject to the minimum requirements in the Code. In general, closed hoops at reduced spacing are required at all locations where plastic hinges may form (and to a distance x' beyond).

At the base of the column, the program calculates **Mc** based on the column capacity because no beams frame into the node:

Mc = Md * capacity factor * ϕye/ϕy

where:

Mc * capacity factor = approximate column capacity based on design loads

ϕy = steel strength reduction factor for normal design calculations

ϕye = steel strength factor for seismic capacity calculations

For example, a column at the base was designed for a moment = 46.2 kn-m and the capacity factor for the actual reinforcement = 1.07. Steel strength used for regular design = 0.87fy while 1.25fy is used for seismic capacity probable strength:

Mc = 46.2 * 1.07 * (1.25/0.87) = 71.0 kN-m

8.3 Design assumption - manual

Refer to the "Appendix" manual.

8.4 Design procedure

- For beams and columns designed for seismic loads, refer also to Design procedure - seismic 793
- for slab design and detailing, refer to Draw slabs - general 794

For beams, columns and walls not designed for seismic forces:

- Define all continuous beams/columns including support locations and support widths. Walls are identified automatically be the program *(STRAP* wall elements)
- Define all properties not specified in *STRAP* geometry by dimensions.
  Beams: The program designs rectangular, tee, inverted tee, L, inverted L, and I shaped sections.
  Columns: The program designs rectangular, L-shaped, round, symmetric U and T shaped sections and any 'solid section' created in the *CROSEC* section generator.
  Note that other shapes are converted automatically by the program or ignored; refer to Define 836.
- Specify the default parameters for all beams/columns/walls.
- Define parameters for specific beams/columns, if different than the default parameters. Examples of parameters that may be defined:
  Beams: reinforcement and concrete type, stirrup parameters, moment redistribution percentage, shear reduction, etc.
  Columns: reinforcement and concrete type
  Walls: effective length factors, allowable bar diameters, structure type: braced/unbraced, etc.
- Compute the results:
  The beams/columns/walls are designed in sequence without any prompts for information by the program. Therefore all design data and parameters must be defined before the design begins.

  The program carries out the following calculations:

  Beams:
  - calculation of moment and shear envelopes from all load combinations.
  - automatic moment redistribution (optional).
  - shear reduction at span supports (optional).
  - calculation of reinforcement steel required at all supports and spans.
  - automatic stirrup detailing with variable spacing.
  - deflection check:
    - span/depth ratio (BS8110, EC2, IS456, NBr)
    - deflection based on effective moment-of-inertia (ACI, CSA)

  Columns:
  - determination of the critical *STRAP* load combination
  - calculation of magnified (additional) moments for slender columns/walls.
  - selection of reinforcement arrangement that is able to withstand applied bending moments and axial force for all load combinations.

  Walls:
  - The capacity is calculated separately for *each* segment in the wall -
  - determination of the critical *STRAP* load combination
  - calculation of magnified (additional) moments for about the weak axis and minimum moments .
  - selection of reinforcement arrangement which is able to withstand applied bending moments and axial force for all load combinations, subject to minimum Code requirements. The reinforcement is distributed equally on both wall faces.

- Revise parameters, properties, etc. and compute again.
- Create column drawings and tables
- Print results.

To display a demo video that explains how to *create, design & detail BEAMS and add the beam detailing to drawings:*

- *click on*  *to start the video*
- *then click on*  *to enlarge the display.*

To display a demo video that explains how to create, design & detail WALLS and add the wall detailing to drawings:

- *click on*  *to start the video*
- *then click on*  *to enlarge the display.*

Note:

- *this video includes an explanation on "design units" and result interpretation*

8.5 Design procedure - seismic

The design procedure is more rigorous for models designed for seismic loads:

- Define all continuous beams/columns including support locations and support widths.
- Define all properties not specified in *STRAP* geometry by dimensions.
- Specify the default parameters for all beams/columns. In particular, specify the seismic frame type and identify the seismic load cases (click the Seismic tab).
- Define parameters for specific beams/columns, if different than the default parameters.

The Codes specify the following hierarchy for the calculation:

- beam longitudinal reinforcement is calculated from the design forces, but not less than the minimum specified by the Code
- beam shear reinforcement is calculated from shear forces derived from the moment capacity of the beam, based on the actual reinforcement area (may be modified by the user), but not less than the design forces.
- column longitudinal reinforcement is calculated from the design forces, but may be increased to ensure that the sum of the column moment capacities at a joint exceeds the sum of the beam moment capacities ($\Sigma M_c/\Sigma M_b > k$).
- column shear reinforcement is calculated from shear forces derived from the moment capacity of the beams connected to the ends of the columns (but not less than the design forces).

Note that beams must be computed prior to columns as the column capacity is dependent on the end moment capacities of the connecting beams.

The design procedure is as follows:

- Compute the beams - select Compute
- Check results, revise parameters, properties, etc, if necessary, and compute again.
- Specify the exact (increased) reinforcement at all beam ends, top and bottom;select Defaults or Parameters and click the Modify Reinf. tab. To check, click Data tables 980 and select **Display reinforcement table**.
- Display the reinforcement areas and the corresponding moment and shear capacities: click Data tables 980 and select **Display seismic capacity table**.
- Compute columns - select Compute
- To display data used to calculate links/stirrups, click Data tables 980 and select **Display column shear table**.
- Check results, revise parameters, properties, etc, if necessary, and compute again.

8.6 Slabs - design procedure

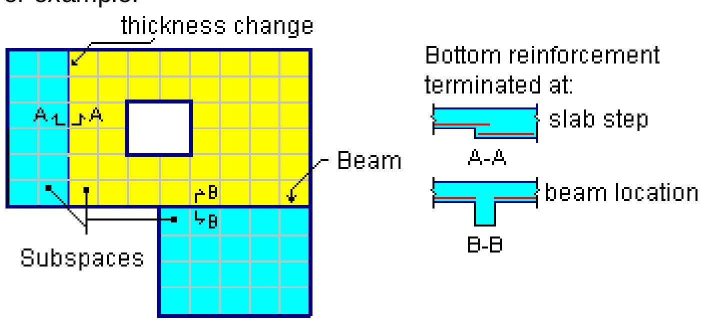
The program calculates and details the slab reinforcement, either as an arrangement of individual bars, a pattern of prefabricated meshes, or a combination of both.

To calculate, detail and draw slab reinforcement:

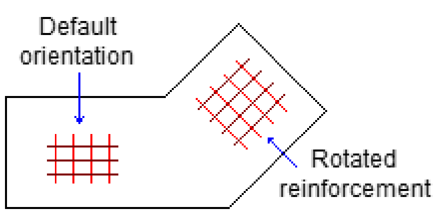
- select Define to define subspaces:
  Subspaces are defined areas on a slab:
  - reinforcement is terminated at subspace boundaries (optional lap into adjacent subspace)
  - each subspace can be assigned with different detailing parameters.

  By default, each level is a separate subspace and this is generally sufficient for most slabs.
  - additional user-defined subspaces may be created along element boundaries, at beam locations and at changes of slab thickness.
  - separate subspaces may be defined for top and bottom reinforcement

  For example:



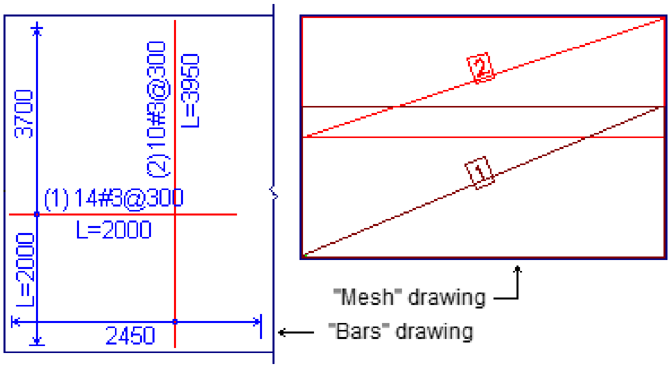
- select Defaults to specify the default parameters for the entire model:
  - reinforcement type: bars or prefabricated meshes
  - steel type, concrete type, cover, etc.
  - bars/mesh parameters: size, min/max diameter, spacing, length, etc.
  - drawing parameters: text size, titles, etc.
- select Parameters to revise any of the above parameters for selected subspaces. Use this option to place bars at a different angle in a subspace. For example:



- select Design to create drawings and add objects to them
  - select a drawing and click Add
  - add slab reinforcement drawing, bar schedules, mesh schedules or mesh details.
- check the reinforcement arrangement and revise parameters if necessary
- select Edit to manually edit the drawing: add/delete bars, revise diameter/spacing/ length, etc.
- select Check det... to check that the revised reinforcement provides sufficient area and

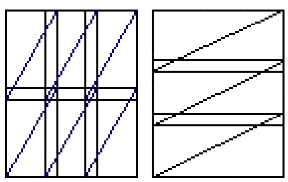
anchorage.

Examples:



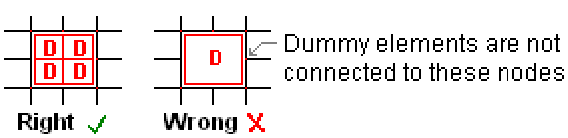
Reinforcement selection method:

- As,req'd is calculated at the center and four corners of each element; the maximum of these values is used for the entire element.
- Bars or meshes are selected according to the parameters.
- The program arranges the reinforcement in two patterns, each with the longer bars in a different direction, and selects the pattern with the smaller weight of steel.



Hints and Suggestions:

- for the "Bars" option: If you decided to place bars only where required but the program details bars over most of the slab area:
  - select **Results** in the toolbar and then **Display detailed results**; note the total weight of reinforcement in the slab.
  - select "**Defaults**" (or "**Parameters**") and set the option to **Put bars also where As = 0**.
  - display the revised drawing and check the total weight of reinforcement again; the second solution is preferable if the weight increase is relatively small because the bar arrangement is simpler and more uniform.
- if you selected **User defined bars/mesh and additional bars/mesh**:
  - click the "**Edit**" icon and select **Display only additional bars**
  - if the additional bars/mesh have been detailed over a relatively large area, decrease the spacing and/or increase the diameter of the fixed bars/mesh so that the additional bars are placed only over a relatively small area.
- if there are small openings in the slab, it may be more practical to detail the slab with bars placed over the opening (to be cut away on site) rather than trying to arrange the bars around the opening. Define "dummy elements" over the openings (the area required is always zero even if minimum reinforcement is requested). Note that the dummy elements must be connected to all nodes on the opening perimeter:



Note:

- in general, the scale should be 1:50. If the drawing size is reduced (e.g. 1:100), the text (bars and dimensions) will overlap and make reading the drawing very difficult. It will probably be necessary to reduce the text size in such cases.

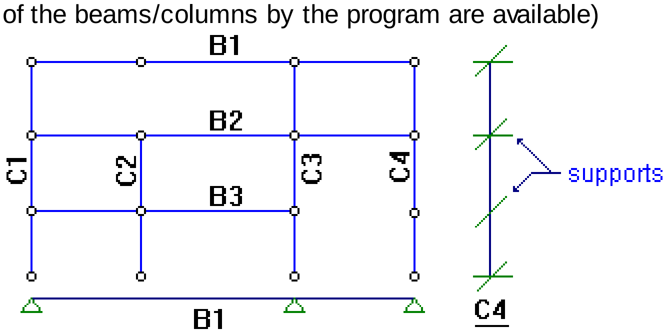
8.7 Create a concrete structure from a STRAP Model

The Concrete design module must design beams, columns and walls separately because the design methods and reinforcement calculation methods are totally different.

- All continuous beams and columns must be created by the user; the program uses the information from *STRAP* geometry for determining support locations and widths and section dimensions, if possible. The program then designs the beams and columns according to user specified parameters.
- Walls are identified automatically by the program from *STRAP* "wall" elements; quad and triangular elements cannot be designed as walls.

Beams & columns:

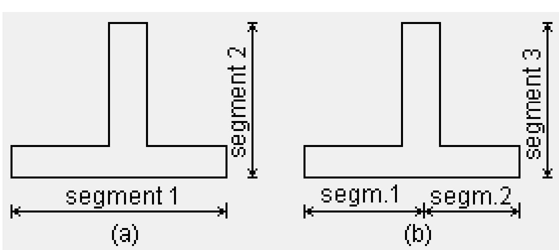
The following figure shows a simple plane frame. In order for the program to design the beams and columns beams B1-B3 and columns C1-C4 must be defined by the user (options for automatic definition of the beams/columns by the program are available)



- Beam B1 must be defined ignoring the dummy node.
- Column C4 must be defined with a support in one direction at the dummy node.

Walls:

The program designs separately each segment of every wall. By default, the program automatically combines colinear segments of equal width to create a single design segment, e.g. segment 1 in Figure (a):



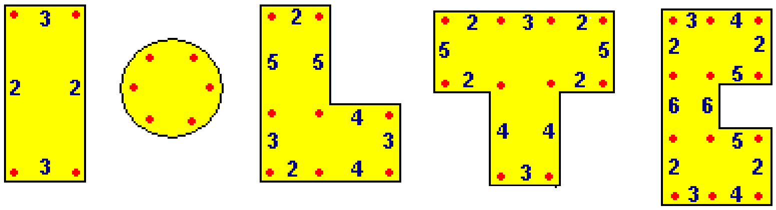
"Design units" with multiple segments can be defined in STRAP geometry. for example. all three segments in the wall above can be combined into a single design unit.

8.8 Column bar groups

The program arranges reinforcement along the entire section perimeter as individual bars and not as a total area. In order to ensure a logical arrangement of bars, the reinforcement is arranged in "groups". These groups are arranged symmetrically in the section.

In all section types, Group 1 represents the corner bars; these bars are ***always*** present and only their diameter varies as required.

Default group arrangement for the various section types:



For example, in a rectangular section Group 1 consists of the four corner bars. Groups 2 and 3 are arranged along the faces of the section between the corner bars. Bars will be located in these groups only if required by the calculation or in order to limit the distance between adjacent bars.

Note:

- non-symmetric group arrangements may also be selected for rectangular sections. Refer to Properties - define/revise 853.
- Round sections contain only Group 1 bars; both the number and the diameter may be revised.

8.9 Defaults

The default parameters are used when design parameters have not been defined for individual members using the Parameters 861 option.

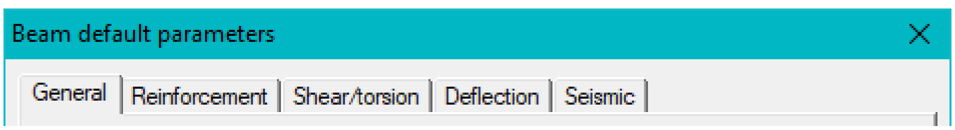
The current default parameter values are displayed at the bottom of the screen.

Beam default parameters 799

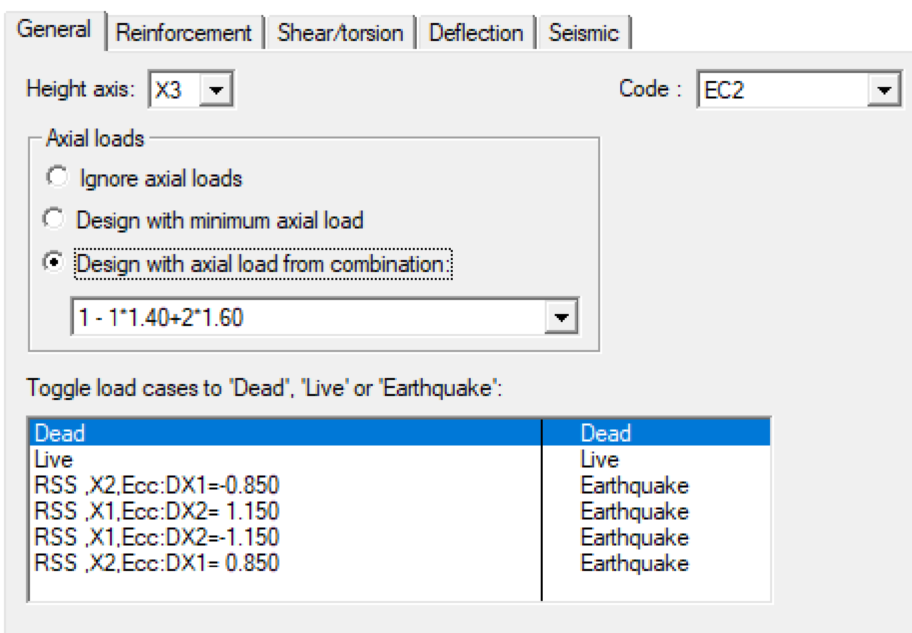
- Column default parameters 809
- Wall default parameters 816
- Slab default parameters 822

8.9.1 Beam default parameters

Specify the default beam design parameters. Values specified here are used for the all beams in the model unless different values were defined for specific beams using the Parameters 861 option.



8.9.1.1 General



Height axis

Define the vertical axis of the model:

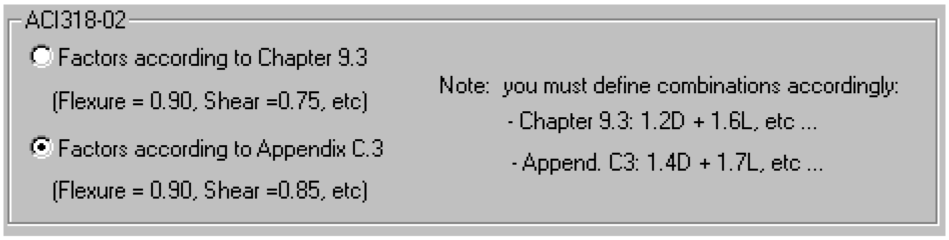
- all columns are assumed to lie parallel to this axis and all beams are assumed to be perpendicular to it when continuous beams and columns are defined automatically.
- slabs are assumed to be perpendicular to the height axis

Code

Select the design code from the list displayed.

Note:

- if the Code is changed after continuous beams and columns have been defined, the program automatically adjusts design parameters assigned to them.
- ACI 318:
  The strength reduction factor, ϕ, may be taken from Chapter 9 or from Appendix C:



Note:

- ***the combinations defined in the STRAP results module must use the load factors corresponding to the strength reduction factors specified***

Axial load

The program designs beams for a combination of moments and axial force. However because the program calculates a bending moment envelope it cannot check each combination separately for combined moment and axial force.

Select one of the following options:

Design with minimum axial load
The program calculates the reinforcement in any span using the moment values from the envelope combined with the minimum axial load (minimum compression or maximum tension) calculated from all of the combinations.

Design with axial load from combination
The program calculates the reinforcement in any span using the moment values from the envelope combined with the axial load from a selected combination.

Ignore axial load
Axial loads are ignored.

Note that a different option may be selected for individual spans; refer to Beam Parameters - general 861.

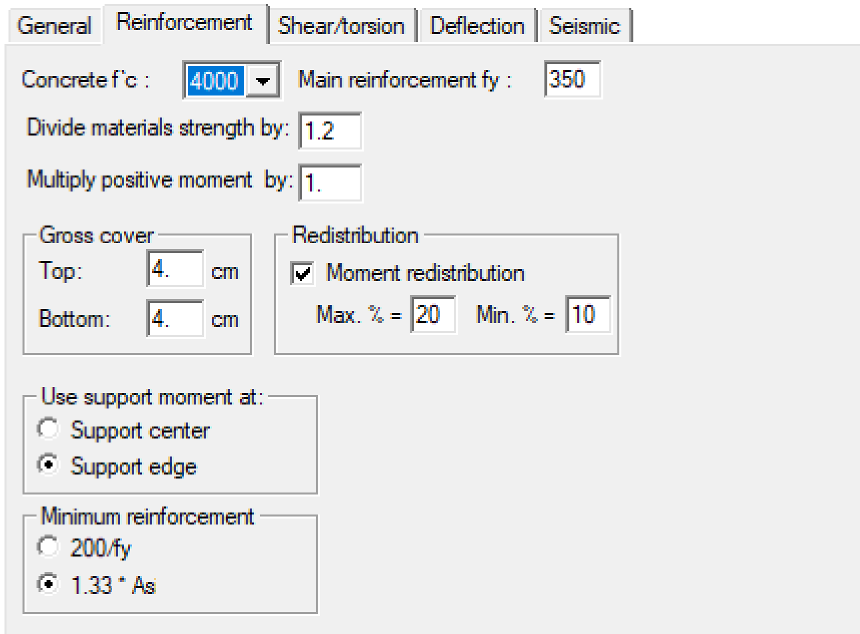
Load cases

Toggle each load case (place the over the title and click the mouse) and specify it as a **Dead**, **Live, Wind** or **Earthquake** load case.

Note:

- seismic design is done only when one or more combinations include an earthquake load case.
- the live load reduction factor is applied only to axial column loads in load cases specified as "Live".
- deflections are calculated only for load combinations that do ***not*** include earthquake load cases. If all load cases in non-seismic combinations are specified as 'Dead' or 'Live', the program can calculate the service load deflections from the cases; otherwise, the calculation is based on factored load deflections and are less accurate. Refer to Deflection - calculation 806.
- *CSA and ACI only*; When designing slender columns, the program uses this data to calculate the value of ßd, where ßd = the ratio of maximum factored axial dead load to maximum factored axial total load. If dead load cases are not specified, the program assumes ßd = 0.40 for all columns.

8.9.1.2 Reinforcement



Concrete

Select the concrete grade. Note that different default concrete grades may be defined for beams, columns, slabs and walls.

Reinforcement

Specify the steel grade for reinforcement. Note that a different steel grade can be defined for beams, columns, slabs and walls, etc.

Divide material strength by

For the beam capacity calculation (specify reinforcement 882), certain codes specify a reduction in the material strength (steel and concrete) when checking existing structures.
Note:

- the program ***divides*** the material strengths by this factor, i.e. enter a factor >1.
- the reduced strength values are used to calculate both the moment and shear capacities.

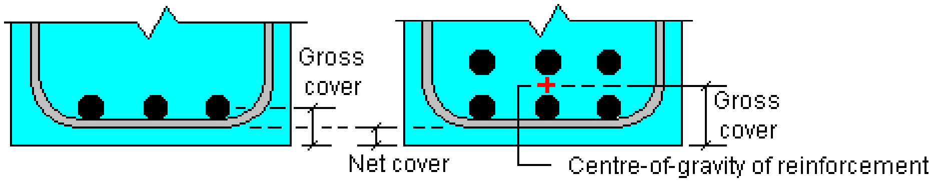
Multiply positive moment by

Increase the positive span moment by a factor when calculating the reinforcement. Values can be check in the Data tables - strength reduction table 985.

Cover

Define the gross concrete cover - the distance from the center-of-gravity of the reinforcement steel to the face of the beam, column or wall.

The gross concrete cover = net concrete cover + stirrup diameter + ½·main reinforcement diameter.



Redistribution

Redistribution is carried out on the individual load cases. The moments in each load case are adjusted so that -

- The support moments in the envelope are reduced up to the maximum percentage specified by the user, but not less than the minimum percentage specified.
- The maximum span moments in the envelope remain constant or are decreased (unless the minimum redistribution requirement forces an increase in the span moment, which generally occurs in exterior spans with fixed supports or columns).
- The shear forces in the spans are adjusted so as to maintain equilibrium of forces and moments.
- For beams with columns, the moment transferred by the beam into the column before and after redistribution is constant. This prevents redistribution in the columns. Therefore, there is no redistribution at exterior column supports.
- There is no redistribution at supports of cantilevers.

The net result is that the negative moments at the supports are reduced without a corresponding increase in the positive span moments.

Set the checkbox to ☑ and specify the maximum and minimum percentages:

Max % - specify the maximum redistribution percentage that the program can carry out at any support.

Min % - specify the minimum percent of redistribution at all support; the program reduces the moments by at least this percent even if the envelope span moments increase as a result.

Note:

- the program carries out redistribution at ***defined*** supports; it cannot identify support locations from the shape of the moment envelope diagram.
- for the BS8110, EC2 and IS:456 Codes, the envelope after redistribution is checked to ensure that it lies within the "70% line". If the maximum redistribution percentage specified is less than 10%, the program uses a 90% line.

Minimum reinforcement (ACI 318/CSA A23.3 only)

The program can calculate minimum beam reinforcement according to either of two methods:

ACI 318:

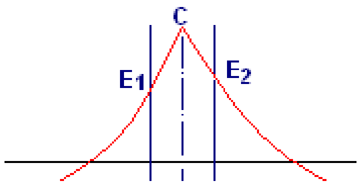
- $200/f_y \cdot bd$
- $1.33*A_s$ required (but not greater than $200/f_y*bd$)

CSA A23.3:

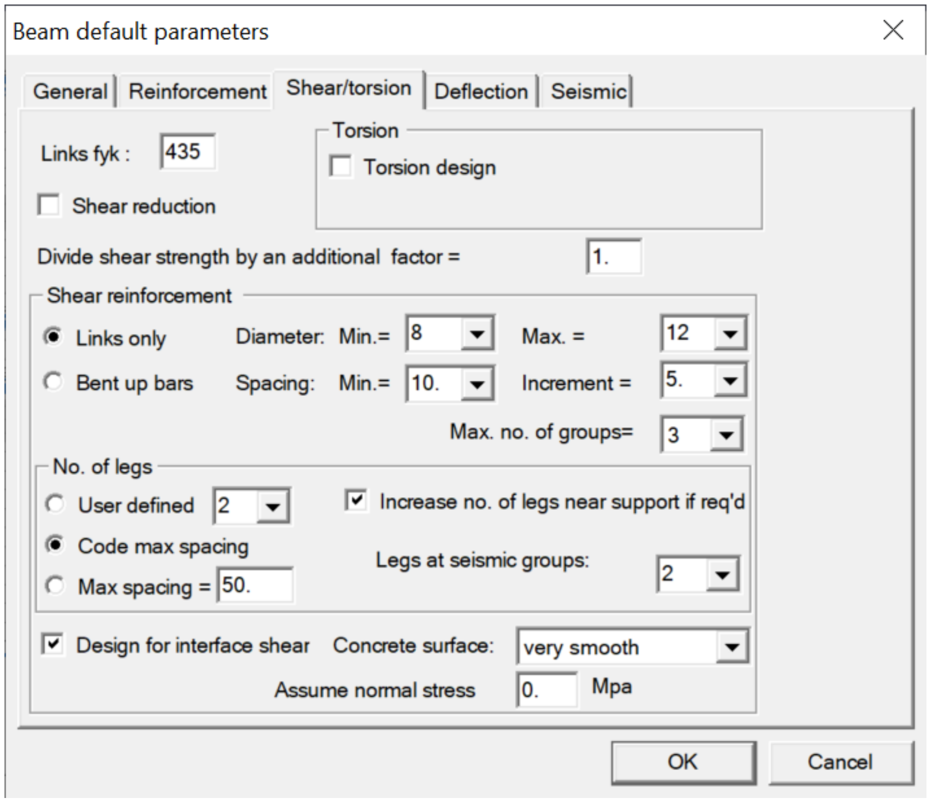
- $0.2v(f'_c)\ bh/f_y$
- $1.33*A_s$ required (but not greater than $200/f_y*bd$)

Support moment

Design the reinforcement at the support using either the moment at the support center (**C**) are the maximum calculated from the two support edge moments (**E_1, E_2**):



8.9.1.3 Shear



Links fy

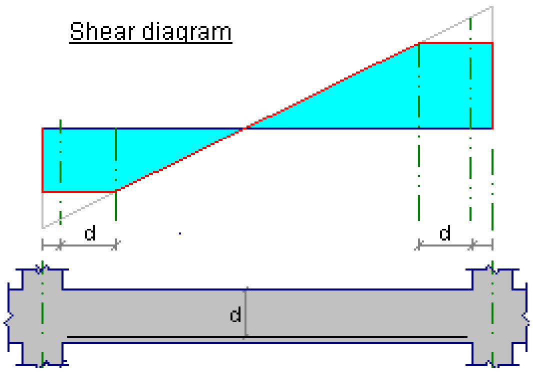
Specify f_y for the links/stirrups. Note that a different f_y for small diameter links can be defined in the Setup 963 option.

Torsion

☑ Design all beams for torsion according to the relevant Code clauses.
☐ Suppress torsion design for all beams.

Shear reduction

Set this option to ☑ to instruct the program to reduce the shear stress at the supports; the shear from the face of the support to a distance '**d**' from the face of the support will have a constant value.



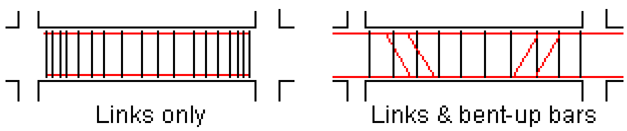
Divide shear strength by

For the beam shear capacity calculation (specify reinforcement 882), certain codes specify a reduction in the shear capacity.
Note:

- the program ***divides*** the capacity by this factor, i.e. enter a factor >1.
- This reduction is ***in addition*** to the reduction generated by the Divide material strength 801 factor.

Reinforcement type

Shear stresses may be taken by stirrups only or by a combination of stirrups and bent-up bars (bent-up bars may not be selected for seismic design):



Links (stirrups) only

The parameters are minimum and maximum stirrup diameter, the minimum spacing, the spacing step, the number of stirrup legs and an alternate number, and the maximum number of stirrup groups per span.

The program assumes the minimum number of legs and checks which diameter in the range specified provides spacing greater or equal to the minimum specified (the program will use the smallest diameter possible). If the minimum number of legs with maximum diameter and minimum spacing does not provide a solution, the program tries again after increasing the number of legs to the alternate.

The program then varies the spacing along the length of the span according to the shear stress. If the number of groups in a span is greater than that specified, the program uses an iterative process to delete the groups with the fewest number of stirrups

Bentup bars

Specify details of stirrups with **constant** spacing to be provided. The program calculates the area of bent-up bars required ***in addition*** to the stirrups provided. The stirrup details required are diameter, spacing and number of legs.

Diameter

Specify the link diameter.

Links only

Specify a range of diameters. The program assumes the minimum number of legs and determines which diameter in the range specified provides spacing greater or equal to the minimum specified (the program uses the smallest diameter possible). Note that diameter specified for the **first** span is used for **all** spans in the beam.

- **Links and bent-up**
  Specify the diameter for the uniform links; the program does not revise this value.

Spacing / increment

Specify the allowable values for link spacing:

- **Links only**
  A range of spacings must be specified as the program calculates variable spacing along the length of the span. Specify the minimum spacing and increment. For example, if you specify an initial spacing of 75 mm and an increment of 50 mm, the allowable spacings are 75, 100, 150, 200, etc.
- **Links and bent-up**
  Select a uniform spacing for the entire span from the list box. Note that you can also type in a value.

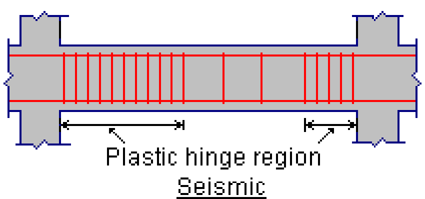
No. of legs (in section)

Specify the number of link legs. For a standard rectangular link, the number of legs = 2.

- **Stirrups/Links and bent-up**
  The number of legs must be specified for the entire span.
- **Stirrups/Links only**
  The following options are available:
  - **User defined**
    Specify the number of legs. The program does not check Code min/max spacing limits.
  - **Code max. spacing**
    The program calculates the number of legs required to limit the spacing to the maximum allowed by the code.
  - **Maximum spacing =**
    The program limits the spacing to the value specified by the user, even if it exceeds the Code limit.

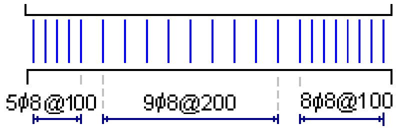
☑ **Increase no. of legs near support where required**
If the combination of [max. diameter + min spacing + specified no. of legs] is not sufficient, this option allows the program to increase the number of legs in the ***first/last*** groups in the span. The program calculates the number of legs required.

- **Legs at seismic groups**:
  Specify the number of legs for the hoops within the plastic hinge length adjacent to the supports. Note that there is always only one group within this length.



Groups

A link group is a series of equally spaced links. For example, the following beam has three link groups.



Specify the maximum number of groups in the current span. Note that the program may design fewer groups than the maximum specified.

Design for interface shear

Set this option to ☑ to instruct the program to design for interface shear. The program will calculate the interface shear

Concrete surface

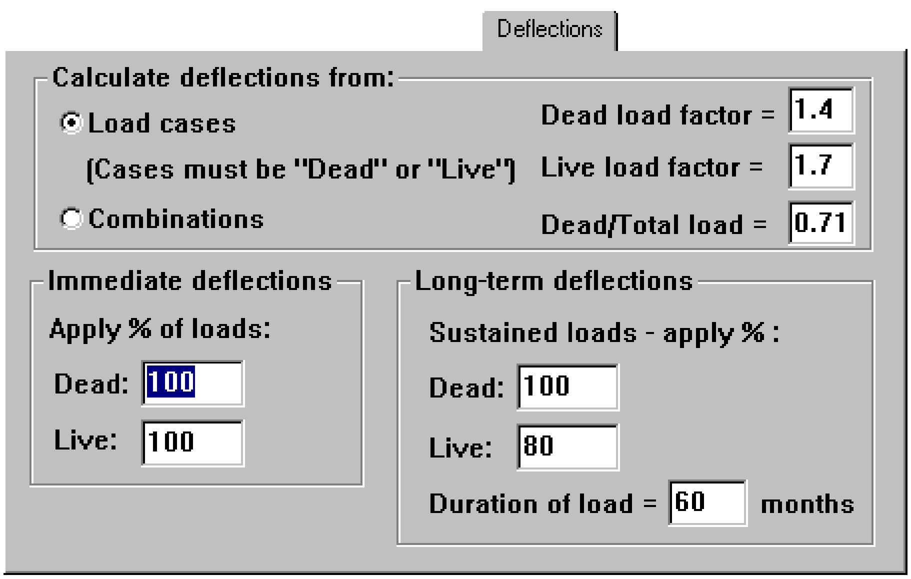
For the calculations of interface shear specify the concrete surface definition according to the selected code.

Assume normal stress

Insert the assumed normal stress. The normal stress is the perpendicular to the interface surface. Compression - positive value, Tension - negative value.

8.9.1.4 Deflection

ACI/CSA type codes only:



Calculate deflections from

Load cases
This option is available if the load cases all non-seismic combinations -
- are identified as "Dead" or "Live"
- contain service loads.

The program can determine the dead and live elastic service deflections for each combination and proceed to calculate the dead, live and sustained load deflections, based on effective moments-of-inertia, according to the Code.

Combinations
When the "Dead" and "Live" load cases are not identified, the program can only calculate the

factored deflections for the ***combinations***. In order to estimate the service deflections for dead and live loads the program requires:

- The ratio of the dead load to the total load to convert the total factored load to separate dead and live factored loads.
- The dead and live load factors to convert the factored loads to service loads.

Note:

- the program checks that all load are identified as "Dead" or "Live", but does ***not*** check that the loads in these cases are 'service'

Immediate deflections

The program by default applies all dead and live loads to the beam when calculating the immediate deflection, a_i. Enter different percentages if you want to apply only a portion of the loads.

For example, to calculate the immediate deflection excluding the self-weight of the beam when the self-weight is approximately 35% of the dead load, revise the values to:

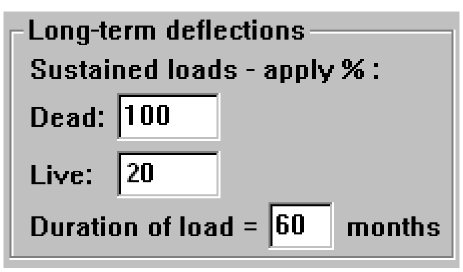
Note that these values are not used for the calculation of the long-term deflection, a_t.

Long-term deflections

These options influence the long term deflection calculation only and specifically refer to Table 9.5(b) in the Code, which requires the calculation of 'the sum of the long-term deflection due to all sustained loads and the immediate deflection due to any additional live load'.

Specify the percentages of the dead and live loads that should be applied for the long term deflection calculation and the duration of the load, required to calculate the value of λ (Eq. 9-11)

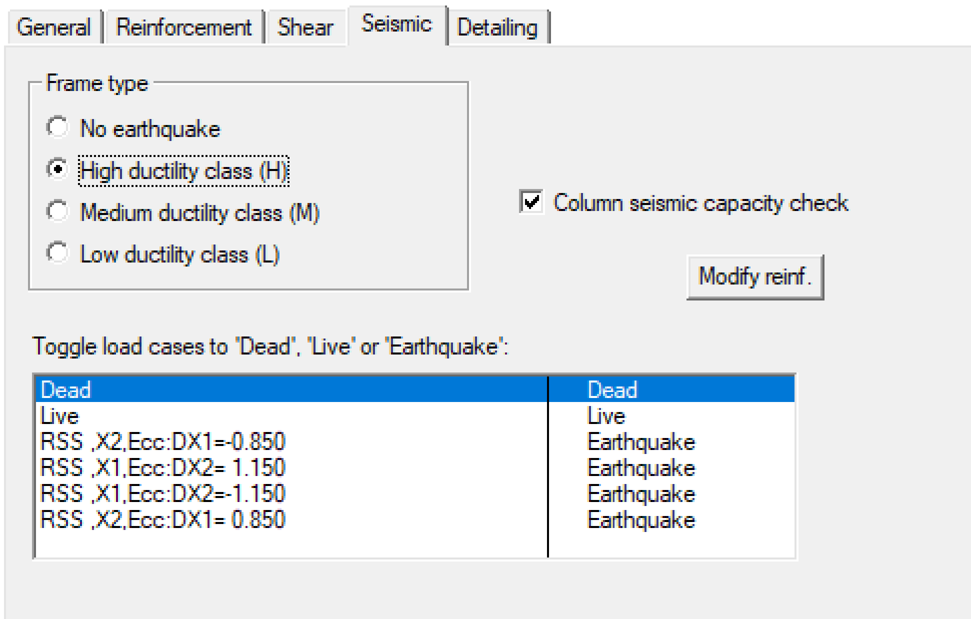
Example:
The sustained load includes the dead load and 20% of the live load . The duration of the load is 5 yrs (60 months) Set the values in the dialog box to:



Note:

- parameters in this dialog box do not affect the value of the immediate deflection.

8.9.1.5 Seismic



(the options in the menu vary according to the Code)

Frame type

Specify the seismic frame type according to the Code classification.

- The default is **no earthquake**, i.e. the frames in this model are not designed according to the seismic requirements of the Code.
- Refer to Design assumptions for more details
- different frame types may be defined for specific members using the Parameters - General 869 option

Load cases

Refer to Beams - General - Load cases 800

Column seismic capacity check (columns only)

The program will ***not*** add the column seismic capacity check load cases ($\Sigma M_{column} = k\Sigma M_{beam}$) but will design and detail the columns according to all other Code requirements for seismic columns, e.g. shear capacity, detailing, etc.

8.9.1.5.1 Modify reinforcement

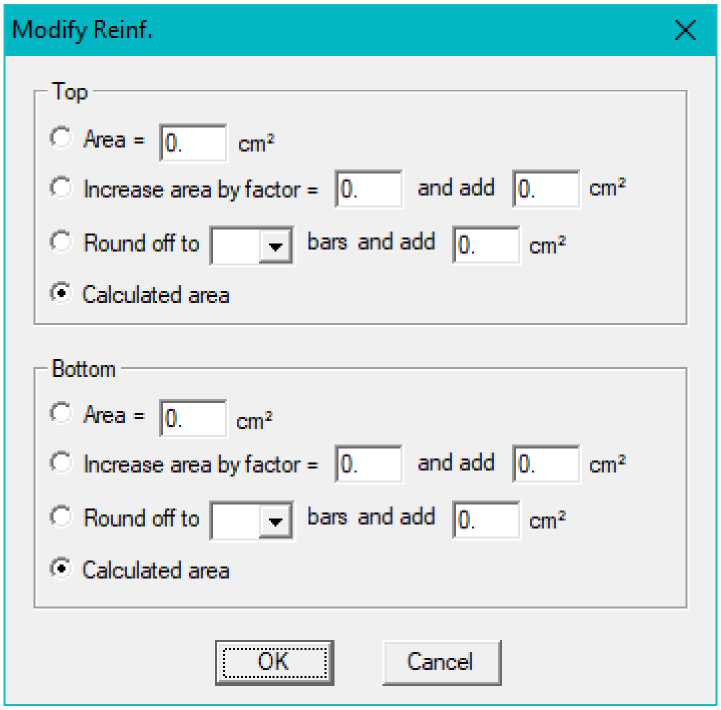
For seismic design, the Codes specify that:

- the beam design shear forces are based on the beam end moment capacity
- the sum of the column moment capacities at a joint must exceed the sum of the beam capacities at the same joint.

Hence, it is necessary to calculate the beam end moment capacities (positive and negative) based on the ***actual*** end reinforcement rather than the theoretical reinforcement.

Note:

- the reinforcement areas are modified ***only at beam supports***.



This option allows you to specify an -

Area =
Specify an increased area according to the units displayed

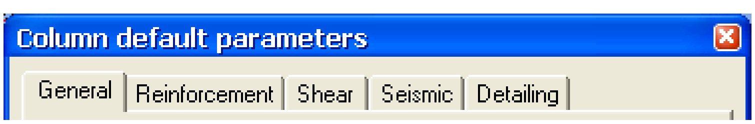
Increase area by factor **and add**
Example: calculated area = 765 mm², factor = 1.1; add = 100 mm²
Increased area = 1.1(765)+100 = 941 mm²

Round off to **bars and add**
Example: calculated area = 765 mm², bars = 15 mm (177 mm²); add = 100 mm²
765/177 = 4.32: round off to 5 bars
Increased area = 5(177)+100 = 985 mm²

Calculated area =
Restore the calculated area.

8.9.2 Column default parameters

Specify the default column design parameters. Values specified here are used for the all columns in the model unless different values were defined for specific columns using the Parameters 865 option.

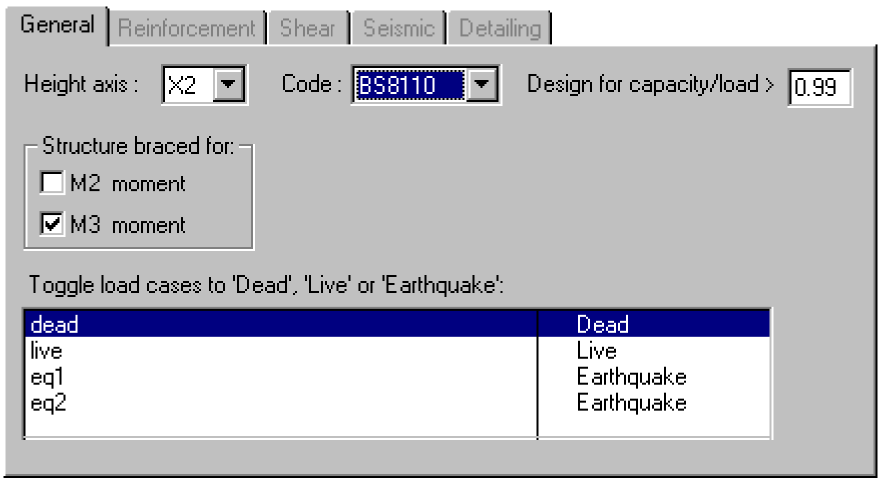


Refer to:
General 810
Reinforcement 811
Shear 812
Seismic 808 (beams)
Detailing 814

Note:

- the effective length factors for columns, k_x and k_y, are assumed by default to be 1.0. This default value cannot be revised. To specify a different value for selected members, refer to Parameters 867

8.9.2.1 General



Height axis

Refer to Beam defaults - General - Height axis 799

Code

Refer to Beam defaults - General - Code 799

Design for capacity/load

Specify the design (Capacity/Load) ratio.

- The default value is generally 0.98 - 1.00 (the program uses 0.99). Higher values give more conservative results.
- A new default value may be specified and different values may be assigned to individual columns in the Parameters - Design 867 option.

Refer also to Compute 874.

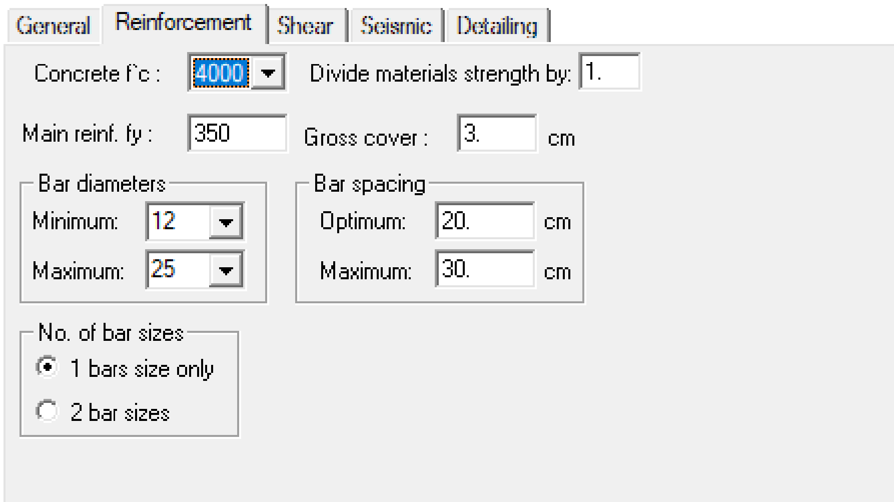
Structure braced for

Specify the structure type to be assumed when calculating the magnified (additional) moments for slender columns: ***BRACED*** or ***UNBRACED***. Note that the column may be braced in one direction and unbraced in the other.

Load classification

Refer to Beam defaults - General - Loads 800

8.9.2.2 Reinforcement



Concrete

Select the concrete grade. Note that different default concrete grades may be defined for beams, columns, slabs and walls.

Reinforcement

Specify the steel grade for reinforcement. Note that a different steel grade can be defined for beams, columns, slabs and walls, etc.

Divide material strength by

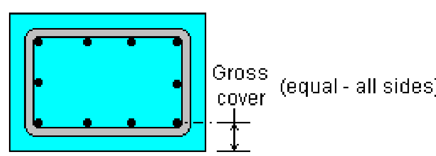
For the column capacity calculation (specify reinforcement 882), certain codes specify a reduction in the material strength (steel and concrete) when checking existing structures.
Note:

- the program ***divides*** the strengths by this factor, i.e. enter a factor >1.
- the reduced strength values are used to calculate both the moment and shear capacities.

Cover

Define the gross concrete cover - the distance from the center-of-gravity of the reinforcement steel to the face of the column.

The gross concrete cover = net concrete cover + stirrup diameter + ½·main reinforcement diameter



Diameters

Select the allowable range of main reinforcement bar sizes.

Bar spacing

- ***Optimum:***
  The program selects the diameter (in the range defined above) which gives a spacing between bars not less than the value defined here. However, if there is no solution using the maximum diameter and the optimal spacing, ***the program automatically uses a spacing less than the optimal specified in this option***, but not less than the absolute minimum spacing allowed between bars by the Code. A

warning message is displayed.

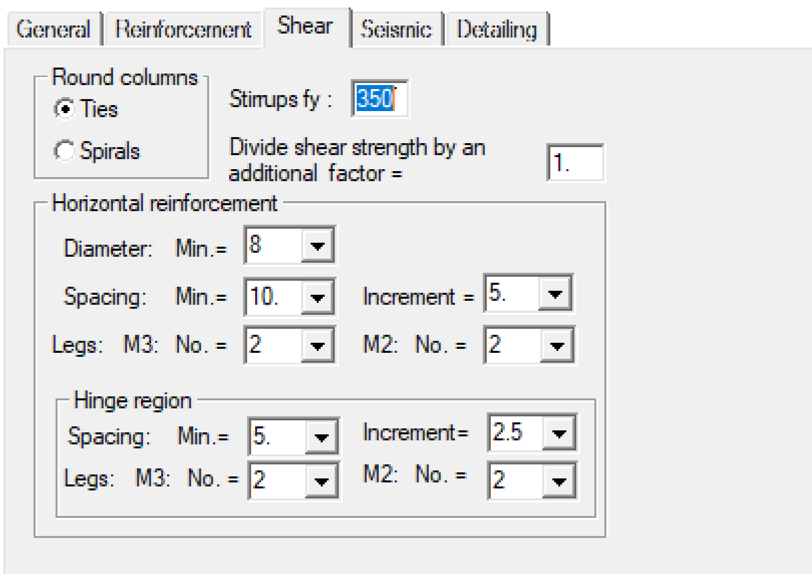
- ***Maximum:***
  Define the maximum allowable spacing between adjacent bars.

No. of bar sizes

Select:

◉ **1 bar size only**:	All bars in a column will have the same diameter.
◉ **2 bar sizes** :	The corner bars may be one bar size larger than the bars along the faces of the column.

8.9.2.3 Shear

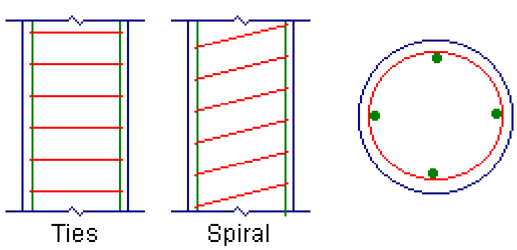


Links fy

Specify f_y for the links/stirrups. Note that a different f_y for small diameter links can be defined in the Setup 963 option.

Round columns

Specify the transverse reinforcement type for ***ROUND*** columns:



This option affects the stirrup design only for the ACI318 and CSA A23.3 Codes. The relevant clauses are:

ACI:

- 7.10.4 - Spirals
  7.10.5 - Ties

- 9.3.2.2 - ϕ value for columns
- 10.3.5.1 - ϕPn,max for columns (refer to ACI - columns)
- 10.9.4 - Equation (10-6) - spirals
- 21.4.4.1 - Seismic requirements - spirals (refer to ACI - seismic- columns)

CSA:

- 7.6.4 - Spirals
  7.6.5 - Ties
- 10.9.4 - Equation (10-7) - spirals
- 21.4.4.2 - Seismic requirements - spirals (refer to CSA - seismic- columns)

Links/Stirrups steel

Specify the steel grade for links (stirrups). Note that a different steel grade cannot be defined for beams and columns.

Divide shear strength by

For the column shear capacity calculation (specify reinforcement 882), certain codes specify a reduction in the shear capacity.
Note:

- the program ***divides*** the capacity by this factor, i.e. enter a factor >1.
- This reduction is ***in addition*** to the reduction generated by the Divide material strength 811 factor.

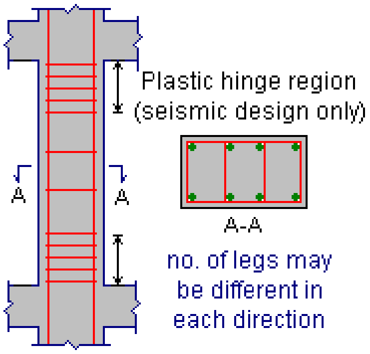
Transverse reinforcement

Specify the parameters for column links (stirrups).

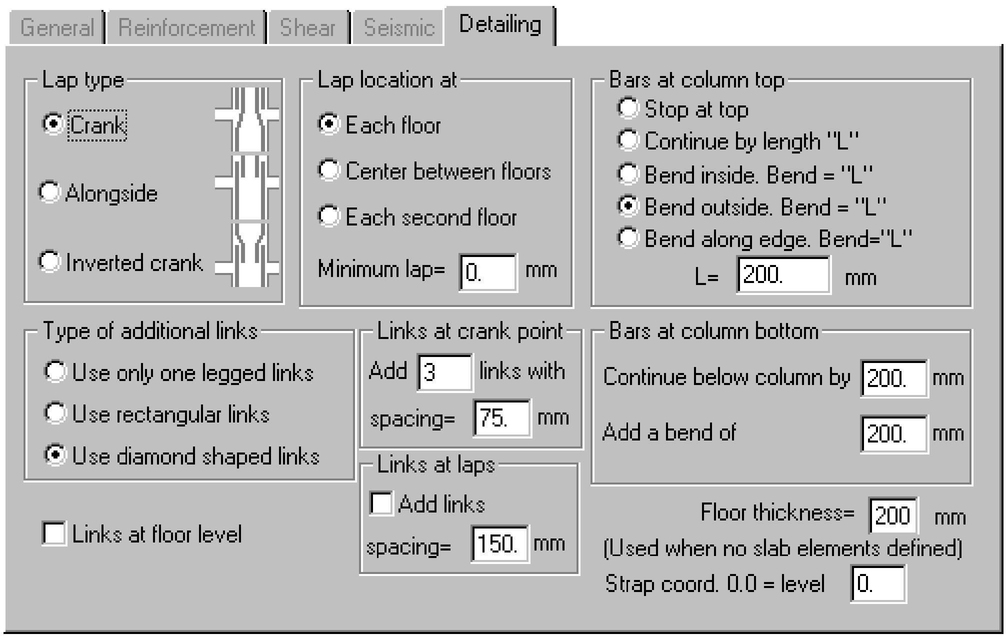
Diameter	:	specify the minimum diameter only
Spacing	:	specify the minimum spacing and the spacing increment
Increment	:	The increment for allowable spacing values. For example, if you specify an minimum spacing of 75 mm and an increment of 50 mm, the allowable spacings that may be selected by the program are 100, 150, 200, etc (75 mm will not be in the list).
Legs	:	A different number of legs may be specified in each design direction

The program initially searches for the maximum spacing that is adequate with the minimum diameter. In no case will the selected spacing exceed the maximum value specified by the Code. If the minimum spacing in the list is inadequate, the program will increase the diameter until it finds a suitable combination of diameter and minimum spacing.

Note that for seismic design different parameters may be specified in the plastic hinge regions adjacent to the column ends:



8.9.2.4 Detailing



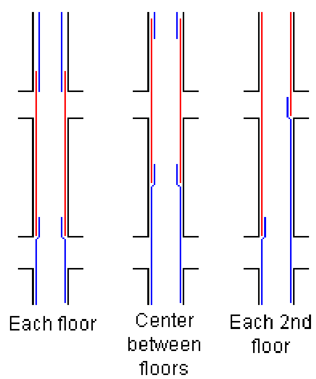
Lap type

Specify the default lap type.

- to use a different lap type for specific columns, select the detailing parameters - lap type 886 option in the side menu.

Lap location at

- Specify the default lap arrangement:



Each floor

Center between floors
the laps are placed at every floor. To combine bars, select the detailing parameters - lap type 886 option in the side menu.

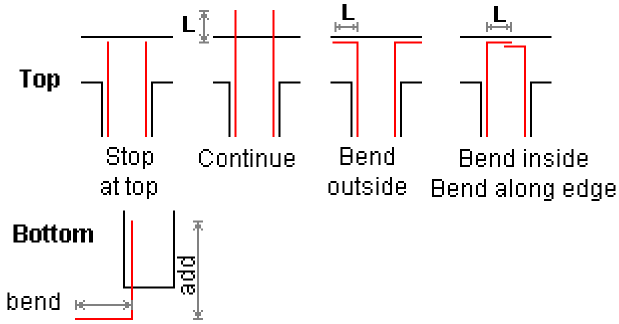
Each second floor
All bars are two stories long, but the laps are staggered at alternate floors, i.e. half the bars are terminated at each floor.

To combine bars over two or more stories, select the Detailing parameters - Lap type 886 option.

- Define the minimum lap length
Note that the program calculates the lap length according to the Code requirements. The lap length used is the maximum of the required lap length and the minimum lap length defined here.

Bars at column top/bottom

Specify the default reinforcing detail at the column top/bottom and the minimum hook lengths

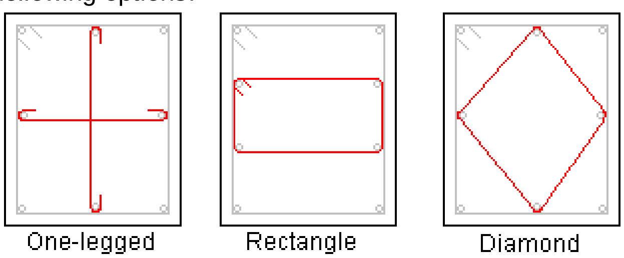


Note:

- this option also applies at intermediate locations where a step in the column face does not allow the bars to be extended to the column above.
- To use a different detail for specific columns, select the detailing parameters - bends at bar ends 886 option.

Type of additional links

Specify the type of additional links to use to tie bars placed between corner bars. Select one of the following options:

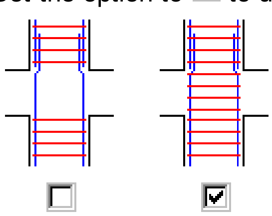


Note:

- Rectangle and diamond links are used only when there are sufficient intermediate bars and internal link angles comply with the Code requirements. Otherwise one-legged ties are used.
- To use a different link type for specific columns, select the detailing parameters - link types 886 option in the side menu.

Links at floor level

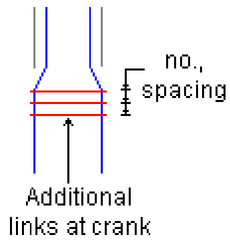
Set the option to ☑ to detail links in the area of the floor and beam:



This is a general option and cannot be modified for individual columns.

Links at crank points

Add additional links at crank points:



This is a general option and cannot be modified for individual columns.

Links at laps

Define the maximum spacing for links within the lap length. The spacing used within the lap length is the minimum of that specified here and the spacing calculated according to the Code requirements.

To use a different detail for specific columns, select the detailing parameters - links at laps 886 option in the side menu.

Floor thickness

Specify:

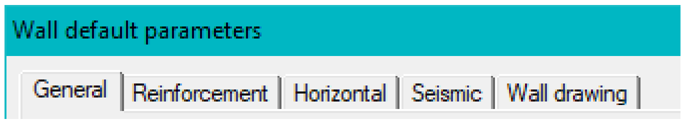
- the default floor thickness used when no slab elements were defined at a floor level. To define a different floor thickness for specific columns, select the detailing parameters - floors - thickness 886 option in the side menu.
- The drawing coordinate equivalent to the *STRAP* 0.00 coordinate. The value defined here added to the *STRAP* node coordinate is the elevation value displayed on the column drawing.

Note:

- the default parameters for the drawing/table are assigned to the column when the columns are created; the parameters can be subsequently revised only by using the “Drawing parameters” option in the side menu. Creating the column again restores the default parameters.

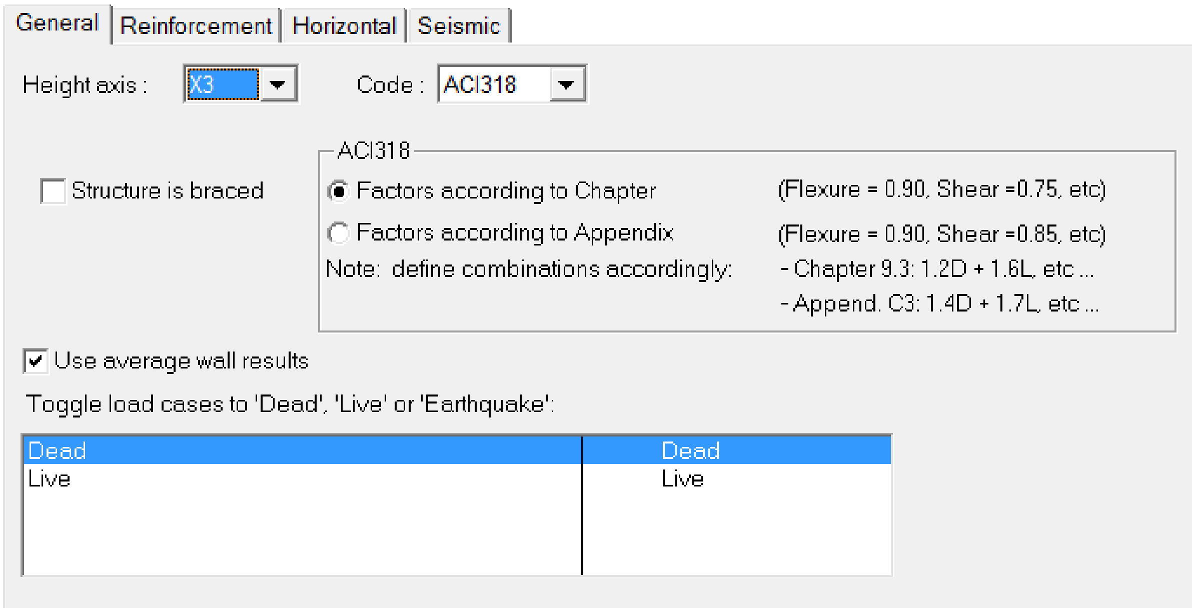
8.9.3 Wall default parameters

Specify the default wall design parameters. Values specified here are used for the all columns in the model unless different values were defined for specific walls using the Parameters 872 option.



Refer to:
General 817
Reinforcement 818
Horizontal 819
Seismic 820
Drawing 821

8.9.3.1 General



Height axis

Refer to <u>Beam defaults - General - Height axis</u> 799

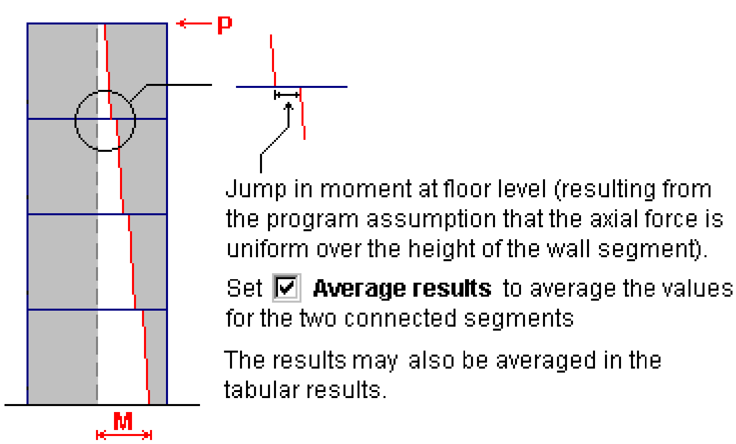
Code

Refer to <u>Beam defaults - General - Code</u> 799

Braced

This option refers to the design of moments about the weak axis.

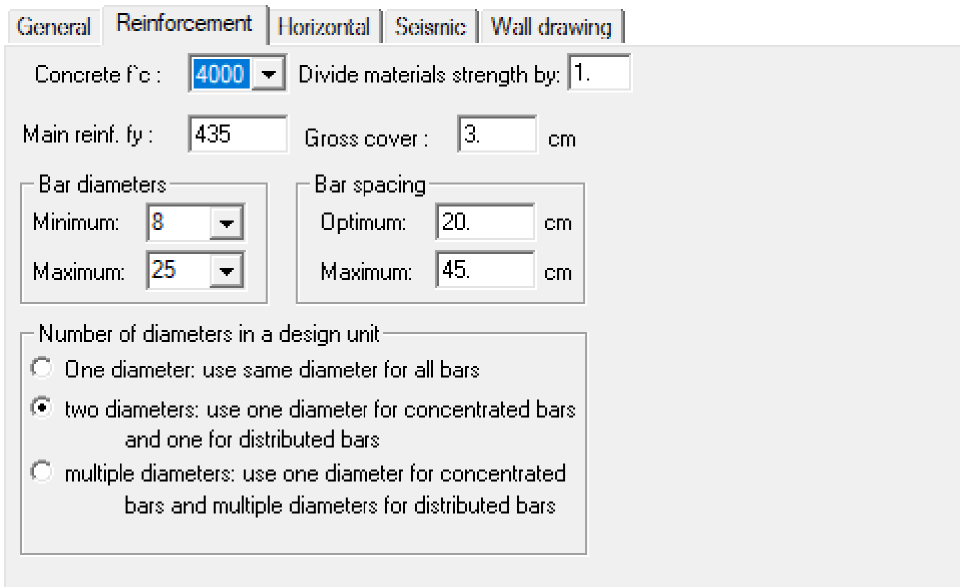
Use average results



Load cases

Refer to <u>Beam defaults - general - load cases</u> 800

8.9.3.2 Reinforcement



No. of bar sizes

Select:

one diameter	All bars - corner and distributed - in a wall design unit will have the same diameter.
two diameters	All corner bars will have one diameter; all distributed bars will have a different diameter
multiple diameters	when there is more than one segment in a design unit: • all corner bars in the unit will have the same diameter • each segment can have a different diameter for the distributed bar, as required.

Note:

- for seismic walls, the program always uses **two diameters** when **one diameter** is selected

Divide material strength by

For the wall capacity calculation (specify reinforcement 882), certain codes specify a reduction in the material strength (steel and concrete) when checking existing structures.

Note:

- the program ***divides*** the strengths by this factor, i.e. enter a factor >1.
- the reduced strength values are used to calculate both the moment and shear capacities.

Concrete

Refer to Column defaults - concrete 811.

Reinforcement

Refer to Column defaults - reinforcement 811.

Cover

Refer to Column defaults - cover 811.

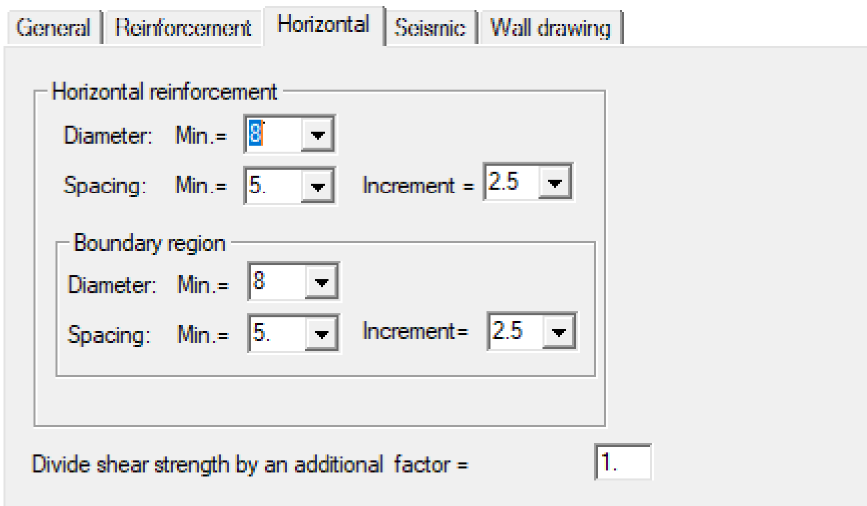
Bar diameters

Refer to Column defaults - diameters 811.

Bar spacing

Refer to Column defaults - bar spacing 811.

8.9.3.3 Horizontal



Horizontal reinforcement

Specify the parameters for horizontal reinforcement in walls.

- the program assumes there are two bars, one at each face.
- different parameters may be specified for the distributed reinforcement and the additional reinforcement inthe boundary regions.

Diameter : specify the minimum diameter only
Spacing : specify the minimum spacing and the spacing increment
Increment : The increment for allowable spacing values. For example, if you specify a minimum spacing of 75 mm and an increment of 50 mm, the allowable spacings that may be selected by the program are 100, 150, 200, etc (75 mm will not be in the list).

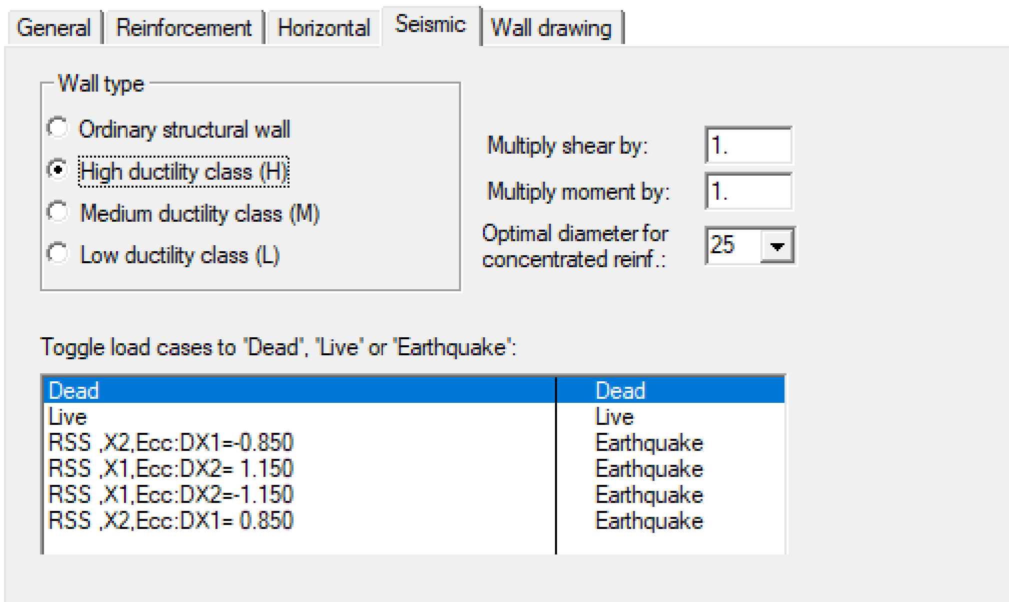
The program initially searches for the maximum spacing that is adequate with the minimum diameter. In no case will the selected spacing exceed the maximum value specified by the Code. If the minimum spacing in the list is inadequate, the program will increase the diameter until it finds a suitable combination of diameter and minimum spacing.

Divide shear strength by factor

For the wall shear capacity calculation (specify reinforcement 882), certain codes specify a reduction in the shear capacity.
Note:

- the program ***divides*** the capacity by this factor, i.e. enter a factor >1.
- This reduction is ***in addition*** to the reduction generated by the Divide material strength 811 factor.

8.9.3.4 Seismic



Wall type

Refer to Beam defaults - Seismic - frame type 808

Multiply shear by

In certain Codes, the design shear force must be increased by a factor. For example, Eurocode 8, section 2.11.1.3:

$\mathbf{V_{sd}} = \varepsilon\ \mathbf{V_{sd}}$ (2.51)

ε = magnification factor (2.52)

Multiply moment by

In certain Codes, the design moment must be increased by a factor. For example, Eurocode 8, section 5.4.2.4 and Figure 5.3.:

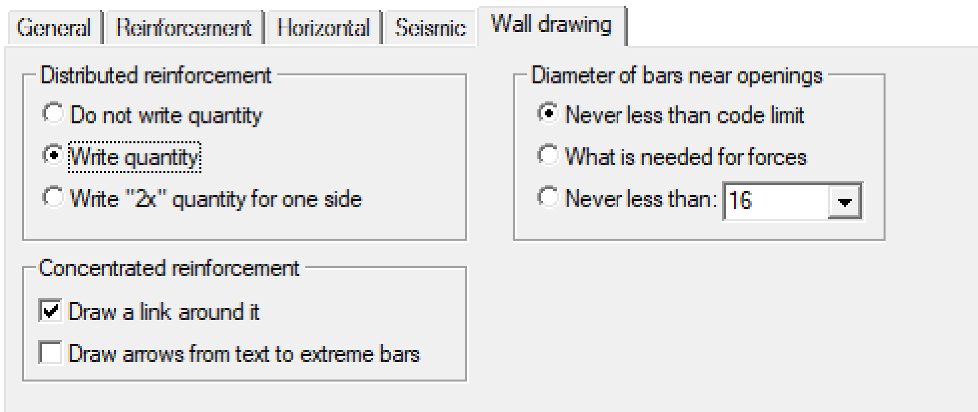
Optimal diameter

Select the optimum diameter for the concentrated reinforcement at the ends of the walls.

Load cases

Refer to Beam defaults - general - load cases 800

8.9.3.5 Drawing



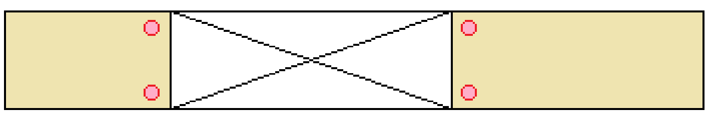
Distributed reinforcement

There are three options for drawing the bar label for the distributed reinforcement.

Example: if there are 16 ϕ12 distributed bars at 200 mm spacing on ***each*** face of a segment, the program will write the label as follows:

Do not write quantity	ϕ12 @ 200
Write quantity	32 ϕ12 @ 200
Write "2x" quantity for one side	2x16 ϕ12 @ 200

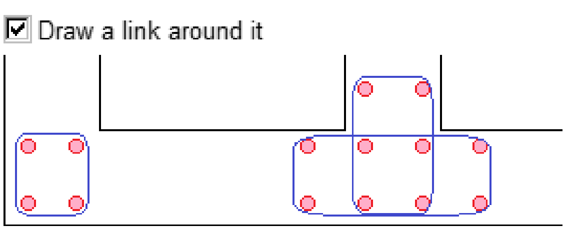
Diameter of bars near opening

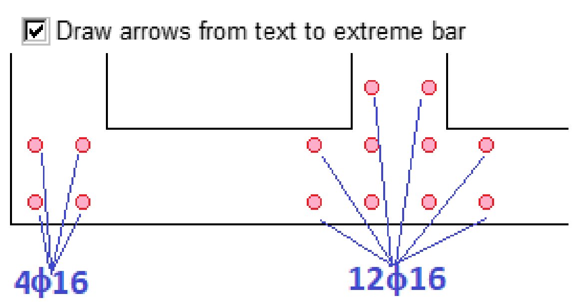


There are three options for selecting the diameter of the reinforcement adjacent to openings

Never less than code limit	use the minimum reinforcement specified by the code (where relevant)
What is needed for forces	ignore the Code requirement and use the diameter calculated by the program
Never less than __	use the maximum of the diameter specified here and the diameter calculated by the program

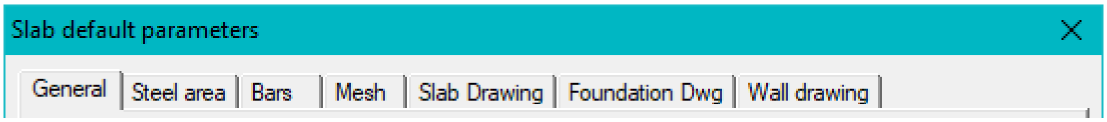
Concentrated reinforcement





8.9.4 Slab default parameters

Specify the default slab design and detailing parameters. Values specified here are used for the all slabs in the model unless different values were defined for specific slabs using the Parameters 869 option.

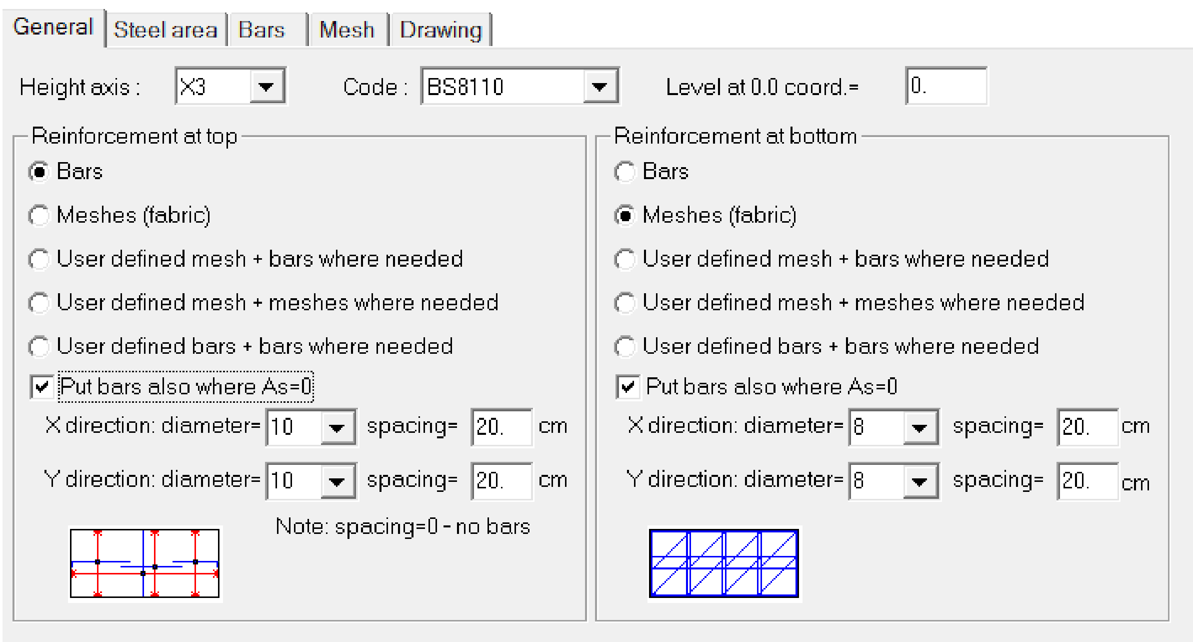


Refer to:
General 822
Steel area 824
Bars 826
Mesh 828
Slab drawing 831
Foundation drawing 833
Wall drawing 821

8.9.4.1 General

Specify the default type for slab reinforcement. There are two main types:

- Bars : the program details the diameter and spacing required in both directions in a series of overlapping rectangles that cover the slab surface
- Meshes : prefabricated fabric; the program calculates the number of bars required in either direction.



Height axis

Refer to Beam defaults - General - Height axis 799

Code

Refer to Beam defaults - General - Code 799

Level at 0.0 coordinate

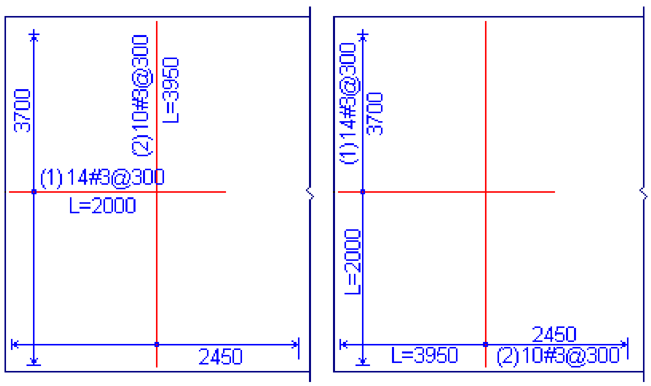
A constant dimension added to the STRAP height coordinate when the slab level is added to the drawing. Refer to Draw slabs - edit - parameters 895.

Reinforcement

There are two slab reinforcement options:

Bars : the program details the diameter and spacing required in both directions in a series of overlapping rectangles that cover the slab surface

Meshes : prefabricated fabric; the program calculates the number of bars required in either direction.



The options are:

- **Bars**
  The bars are selected and detailed according to the Steel area 824 and Bars 826 parameters
- **Meshes**
  The bars are selected and detailed according to the Steel area 824 and Mesh 828 parameters
- **User defined mesh + bars where needed**
  Specify the details of a mesh (diameter and spacing); the program details additional bars if the required area is greater than the area provided by the mesh.
- **User defined mesh + additional meshes**
  Specify the details of a mesh (diameter and spacing); the program details additional meshes if the required area is greater than the area provided by the mesh.
- **User defined bars + bars where needed**
  Specify the constant bar details (diameter and spacing); the program details additional bars if the required area is greater than the area provided by the bars.

For the last three options, the user-defined meshes/bars are placed over the ***entire*** subspace area. The following option is available for the **Bars** and **Meshes** options:

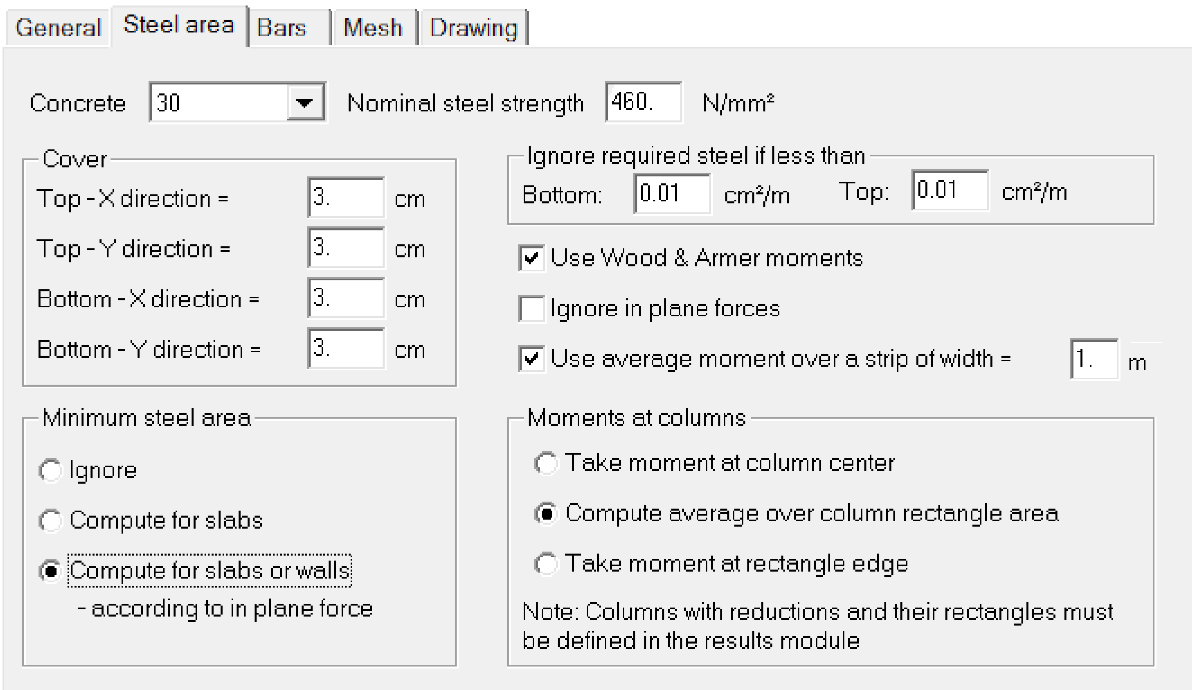
- [ ] **Put bars also where As = 0**
  No reinforcement is placed where not required.
- [x] **Put bars also where As = 0**
  Reinforcement with the specified diameter and spacing is placed at all locations where not required. Enter spacing = 0 to cancel this reinforcement in the X or Y direction.

Note:

- different values may be defined for specific subspaces using the Parameters - General 869 option.

8.9.4.2 Steel area

Specify the default parameters for calculating the area required:



Concrete

Select the concrete grade. Note that different default concrete grades may be defined for beams, columns, slabs and walls.

Steel strength

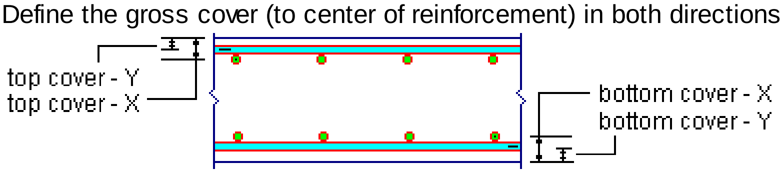
The steel grade can be designed in three different locations in the Slab default menu:

- **Steel area** tab : this is the default value for bars and meshes
- **Bars** tab : grade for bars
- **Mesh** tab : grade for meshes

A change to the value in the **Steel area** tab is applied to the **Bars** and **Mesh** values only if they were not previously revised.

Cover

Define the gross cover (to center of reinforcement) in both directions:



Wood & Armer

☑ Use the Wood & Armer design moments to calculate the reinforcement area

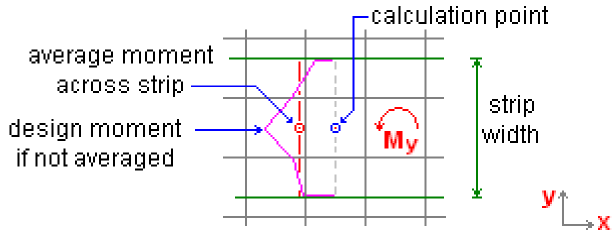
☐ Use the ***STRAP*** M_x and M_y moments to calculate the reinforcement area (ignore the influence of M_{xy})

In-plane forces

For space models only, set the option to to ignore the axial forces when calculating the reinforcement area, i.e. areas are calculated from moments only.

Use average moment

Calculate the design moment at every point by using the average moment across a strip perpendicular to the direction of the moment. This option reduces the design moment at points with high moment concentrations. For example:



Ignore required steel

Use this option to suppress top and/or bottom reinforcement where the amount required is very small. Enter cut-off values; the program revises all smaller areas to zero (and hence does not provide minimum steel area).

Minimum steel area

The calculated steel area at any location may be less than the minimum steel area specified by the Code.

Select one of the following options:

Ignore
Provide only the required reinforcement and ignore the minimum area requirements in the Code.

Compute for slabs
Provide the minimum area specified by the Code requirements for slabs.

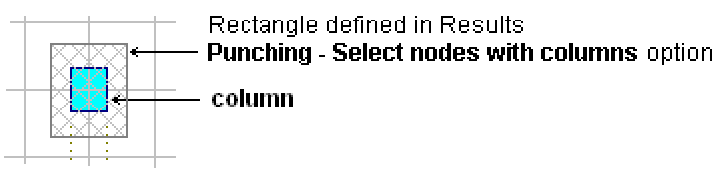
Compute for slabs or walls
Provide the minimum area specified by the Code requirements for slabs or walls (the maximum of the two).

Refer to Design assumptions for the relevant code.

Moments at column

The program can design the slab adjacent to columns using either the moment at the column center or a reduced moment value calculated at a specified distance from the face of the column

The column location must be defined in the Results - Punching - Select nodes with columns option; otherwise the program uses the moment value at the column center.
The rectangle size is defined in the Results - moment reduction option.



Select one of the following options:

No reduction ...

The program uses the moment value at the column center.

Use average moment ...

The program calculates and uses an average moment values over the defined rectangle area

Use maximum moment ...

The program uses the maximum moment value on the defined rectangle perimeter

For a detailed explanation on reduced and examples, refer to Results along a Line - General.

Note:

- if the **Use Wood & Armer moments** option is selected, the program first calculates separately the reduced Mx, My and Mxy moments, then calculates the reduced Wood & Armer moments from these values.

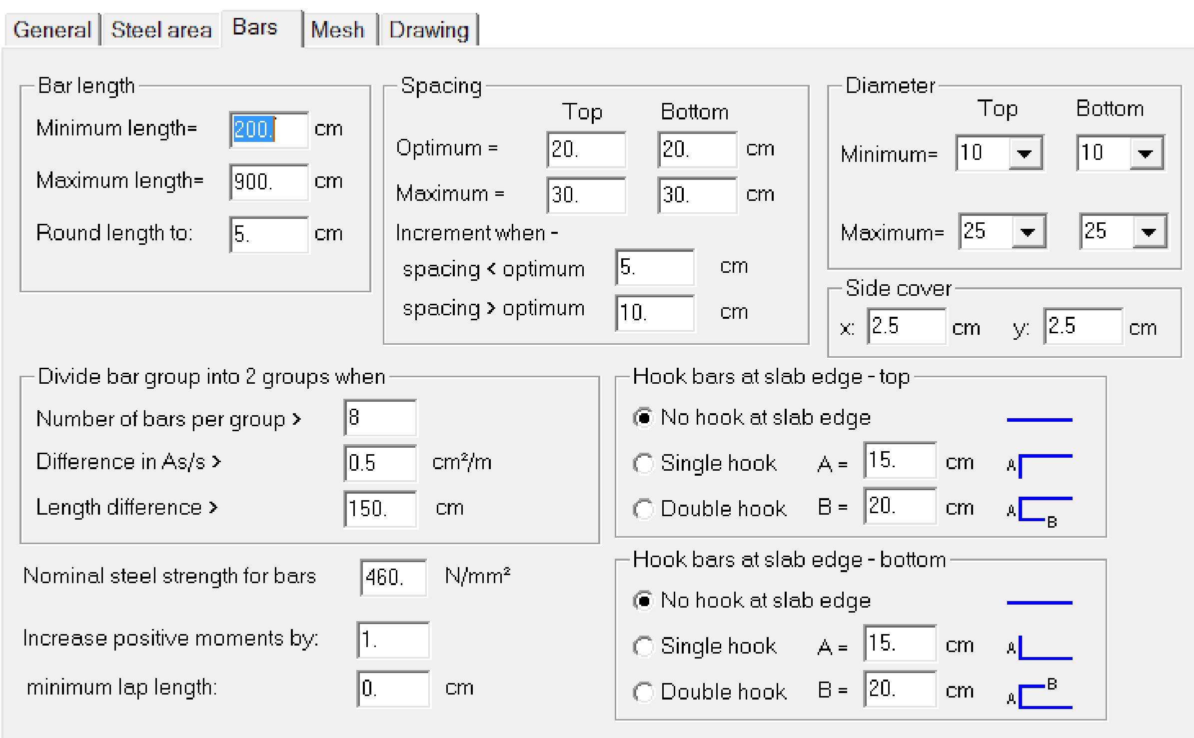
Note:

- different values may be defined for specific subspaces using the Parameters - Steel area 870 option.

8.9.4.3 Bars

Specify the parameters for detailing the bar grids in all subspaces.

- the program selects bars within the diameter range as close as possible to the optimal spacing
- a grid is terminated and a new one started when either
  - the maximum bar length is exceeded
  - a smaller diameter/larger spacing is adequate or a larger diameter/smaller spacing is required.
- hooks may be detailed at the slab edges



Bar length

Specify the following parameters for bar length:

- minimum and maximum bar length
- bar length round-off value

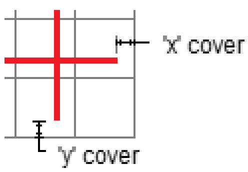
Spacing/diameter

Specify the following parameters for bar spacing and diameter:
- minimum and maximum diameter (top and bottom)
- optimal spacing and maximum spacing (top and bottom)
- spacing increment when spacing is less than optimal and when greater than optimal.

The program selects diameter and spacing as follows:
- the program calculates the diameter required for optimum spacing
- if the minimum diameter is selected, the program tries to increase the spacing (increment when spacing greater than optimal)
- if the maximum diameter is selected and the optimal spacing is not adequate, the program decreases the spacing by the regular increment.

Side Cover

Define the cover from the end of the bar to the perpendicular slab edge, in both directions:



Divide into 2 groups

The program calculates a uniform bar grid over an area of the slab, based on the maximum reinforcement area required in that grid. At a certain point the program terminates the grid and begins another one with larger/small spacing or diameter.

There are three criteria for starting a new group; the new group is created when the 1st criteria and either the 2nd or the 3rd are met, i.e.
- Number of bars in the group exceed the maximum specified

- AND -
- Current reinforcement area exceeds the area required by the specified value

- OR -
- Current reinforcement length exceeds the length required by the specified value.

Hook bars at edge

Specify the following parameters for hooks at the end of the slabs:
- hook type
- hook length

Different types and lengths may be specified for top and bottom bars.

Steel strength

The steel grade can be designed in three different locations in the Slab default menu:
- **Steel area** tab : this is the default value for bars and meshes
- **Bars** tab : grade for bars
- **Mesh** tab : grade for meshes

A change to the value in the **Steel area** tab is applied to the **Bars** and **Mesh** values only if they were not previously revised.

Increase positive moments by

Enter a factor; the program will increase all positive moments in the slab by this factor when calculating the bottom reinforcement.

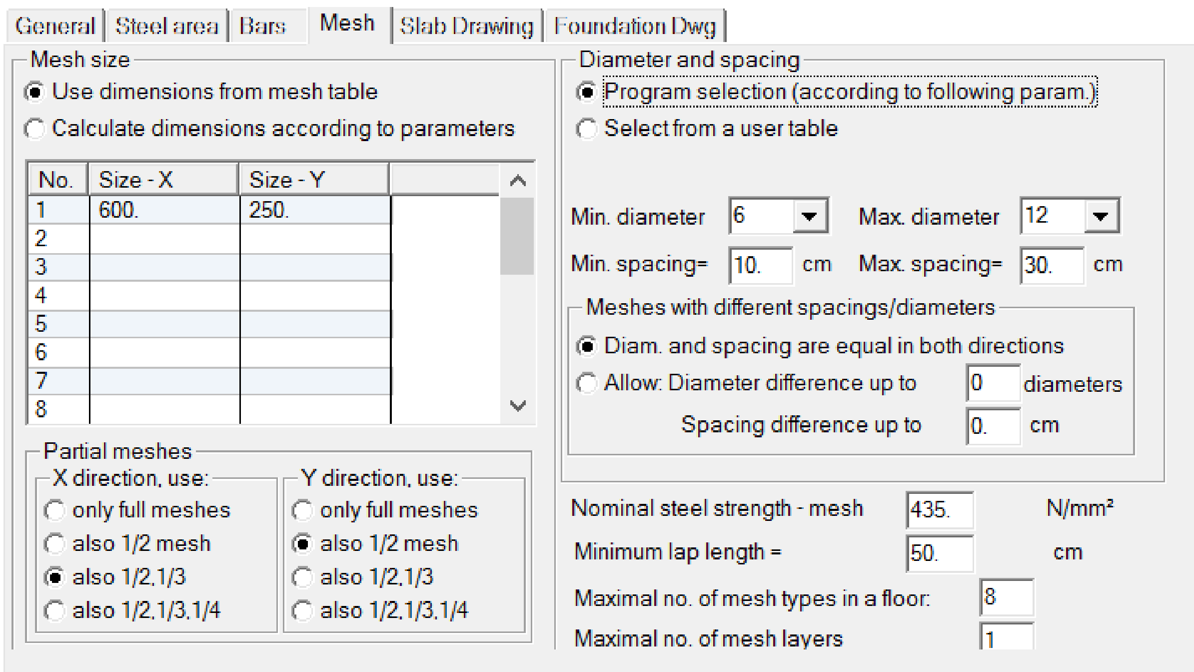
Minimum lap length

Specify a minimum lap length for all diameters.

Note:
- different values may be defined for selected subspaces using the Parameters - Bars 871 option.

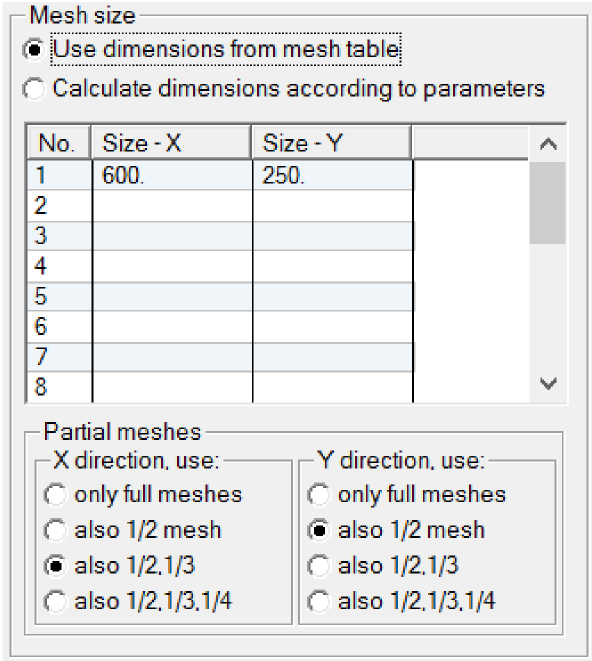
8.9.4.4 Mesh

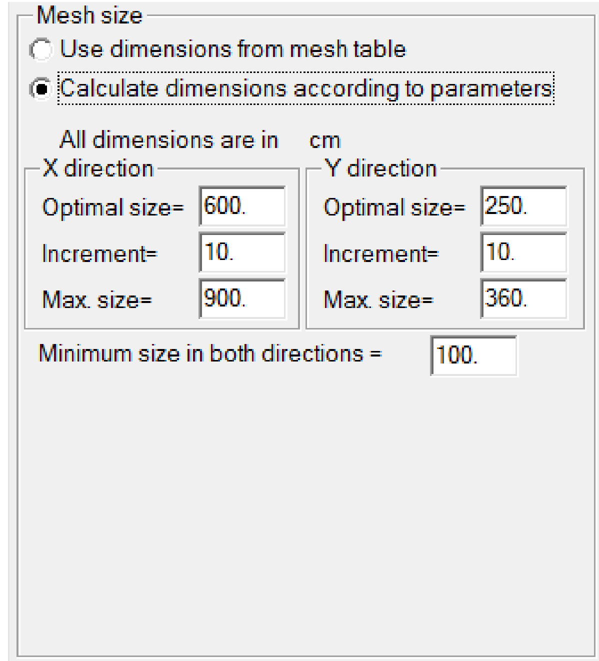
Specify the mesh parameters:
- permissible dimensions and dimension parameters
- reinforcement diameter and spacing parameters



Mesh size

The size of the prefabricated meshes are selected according to the parameters specified here. There are two options:





Specify:

- the permissible mesh X,Y dimensions in the table; all meshes in the slab are selected from the table.
- partial meshes may also be used.

The program arranges meshes in the spaces only with the specified sizes; overlap is adjusted so that the meshes align with the boundaries.

Specify:

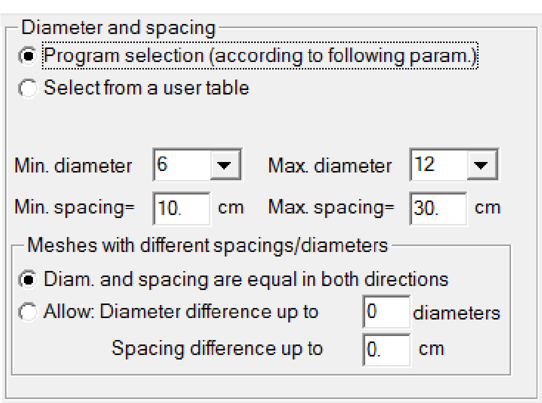
- the optimal size for all meshes.
- the maximum and minimum size
- the size increment

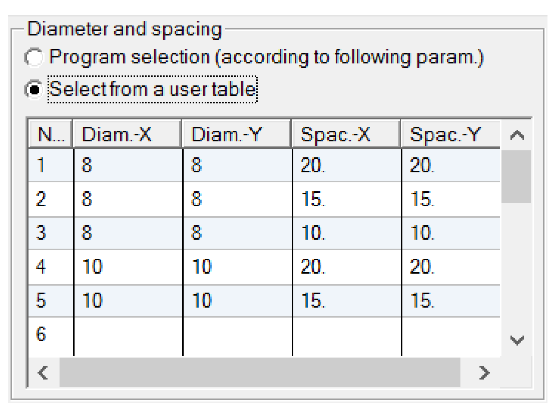
The program arranges meshes with the optimal size. If the last mesh projects from the space boundary, the program decreases its dimension by the 'increment' value until it is aligned with the boundary.

If the gap from the last mesh to the boundary is small, the program increases the dimension by the increment up to 'Max size'.

Diameter and spacing

There are two options:





Specify the reinforcement parameters:
- minimum and maximum diameter
- minimum and maximum spacing

The difference between the diameter and spacing in the two directions may be limited by the user:

◉ **Diam and spacing are equal in both directions**
The program selects the diameter and number of bars according to the largest As,req'd and applies the identical detailing in the other direction.

◉ **Allow: diameter/spacing difference up to ....**
The program provides less reinforcement in the smaller As,req'd direction, but limits the spacing and diameter difference between the two directions.

Note:
- different min/max diameters for the top and bottom reinforcement can be defined in Mesh - parameters 871.

The program selects only meshes that are defined in this table.

Specify the diameter and spacing in both directions for each mesh.

Up to 20 different meshes may be defined.

Nominal steel strength

The steel grade can be designed in three different locations in the Slab default menu:
- **Steel area** tab: this is the default value for bars and meshes
- **Bars** tab: grade for bars
- **Mesh** tab: grade for meshes

A change to the value in the **Steel area** tab is applied to the **Bars** and **Mesh** values only if they were not previously revised.

Minimum lap length

Specify a minimum lap length for meshes. The program uses the maximum of the value specified here and the Code value.

Maximal no. of mesh types

Specify the maximum number of different mesh types that may be detailed in this model.

Note that the 'type' refers only to the diameter and spacing; meshes with the same diameter and spacing but different bar lengths are considered to be the same 'type'.

Maximal number of mesh layers

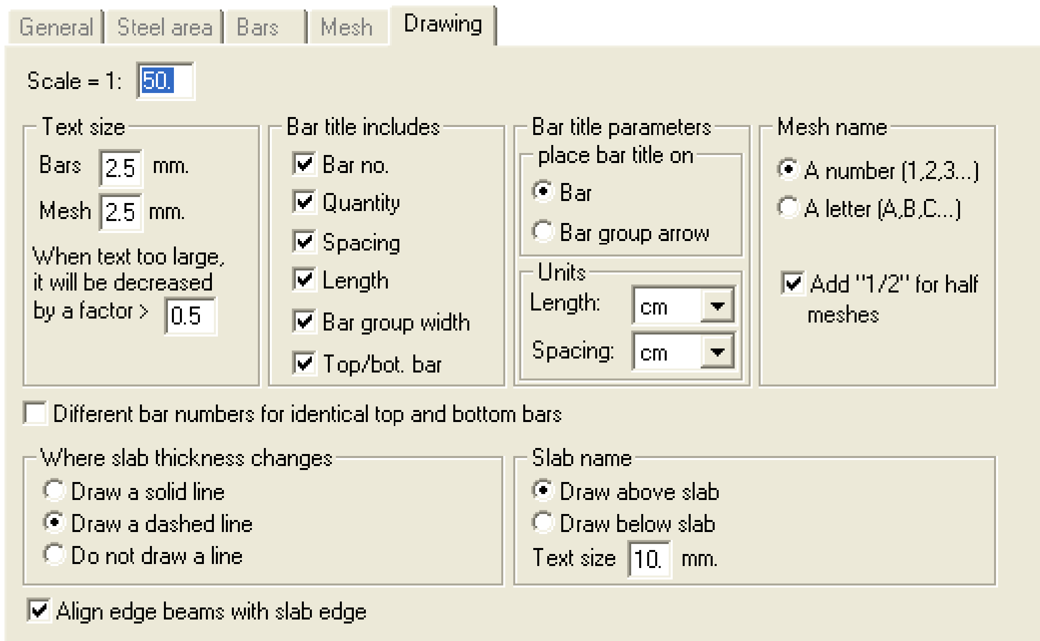
if the largest permissible mesh does not provide a sufficient area of reinforcement, the program places additional layers of meshes, up to the maximum no. of layers specified. The program then adds regular bars if the area is still insufficient.

The program selects the reinforcement as follows:
- the program tries to arrange meshes over the required area (or the entire area, if specified by the user) according to the various parameters
- the program avoids, as much as possible, arranging the meshes so that they overlap in areas of maximum moment (e.g. adjacent to columns for the upper mesh).
- the program initially details as many different meshes as required, but if the number of different types exceeds that maximum specified by the user, the program eliminates types so that the additional reinforcement area is minimum.
- The program adds regular bars if the area provided by the allowed meshes is still insufficient.

Note:
- different values may be defined for specific subspaces using the Parameters - Mesh 871 option.

8.9.4.5 Slab drawing

Specify default parameters for the drawing: text size, title location, bar label style, etc:

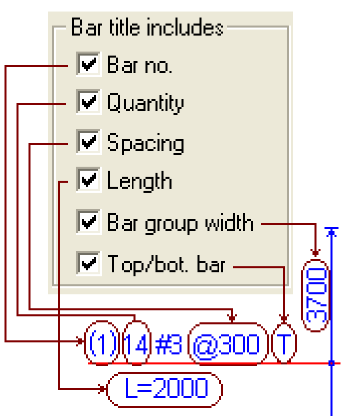


Text size

Specify the text size for drawings with bars and drawings with meshes.

If a specific text is too long for the bar/mesh/etc, the program reduces the text size by a factor not less than the fraction specified in this option.

Bar title



Note:
- if quantity and spacing are requested but there are fewer than 4 bars in the group, the program does not display the spacing.
- the characters displayed for the **Top/bot. bar** option may be revised in the file **STRAP.INI**, section **[CONC_PARAM]**:

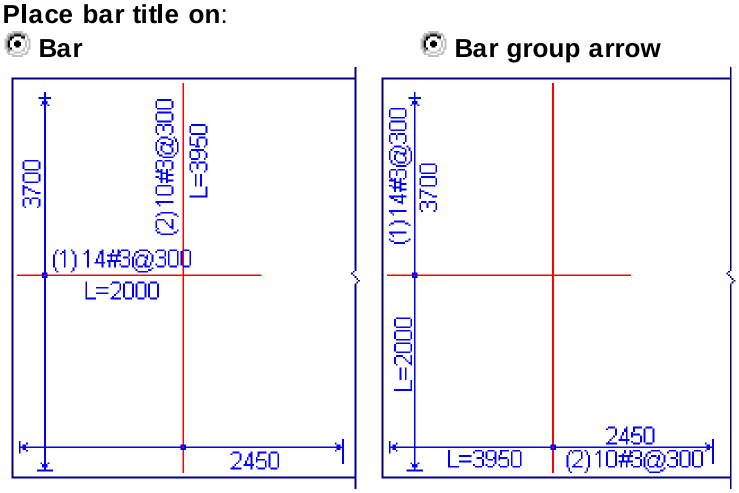
TopBarString=	(normally T)
BotBarString=	(normally B)
TopTopBarString=	a different string may be used for the exterior top/bot bars where reinforcement is
BotTopBarString=	placed in both directions. If not defined the program uses the same strings defined for TopBarString= and BotBarString=

The strings may have a maximum of four characters.

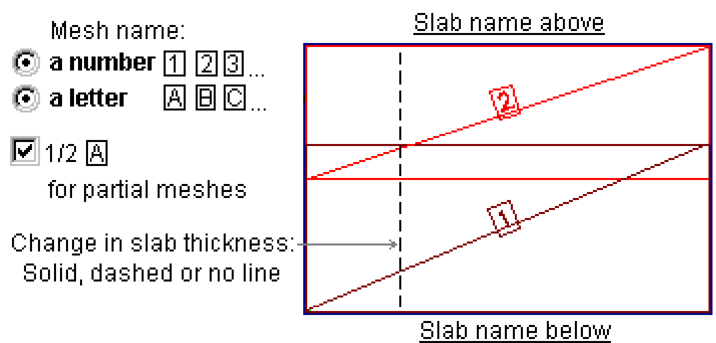
Bar title parameters

Two options are available:

Place bar title on:



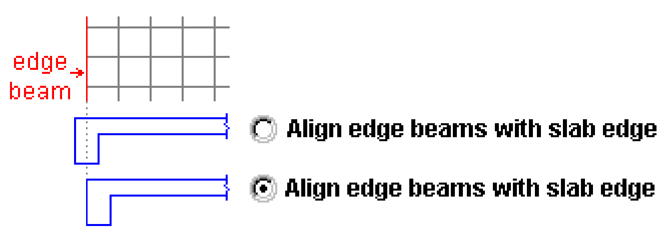
Mesh name / Slab name / Slab thickness change



Different bar numbers

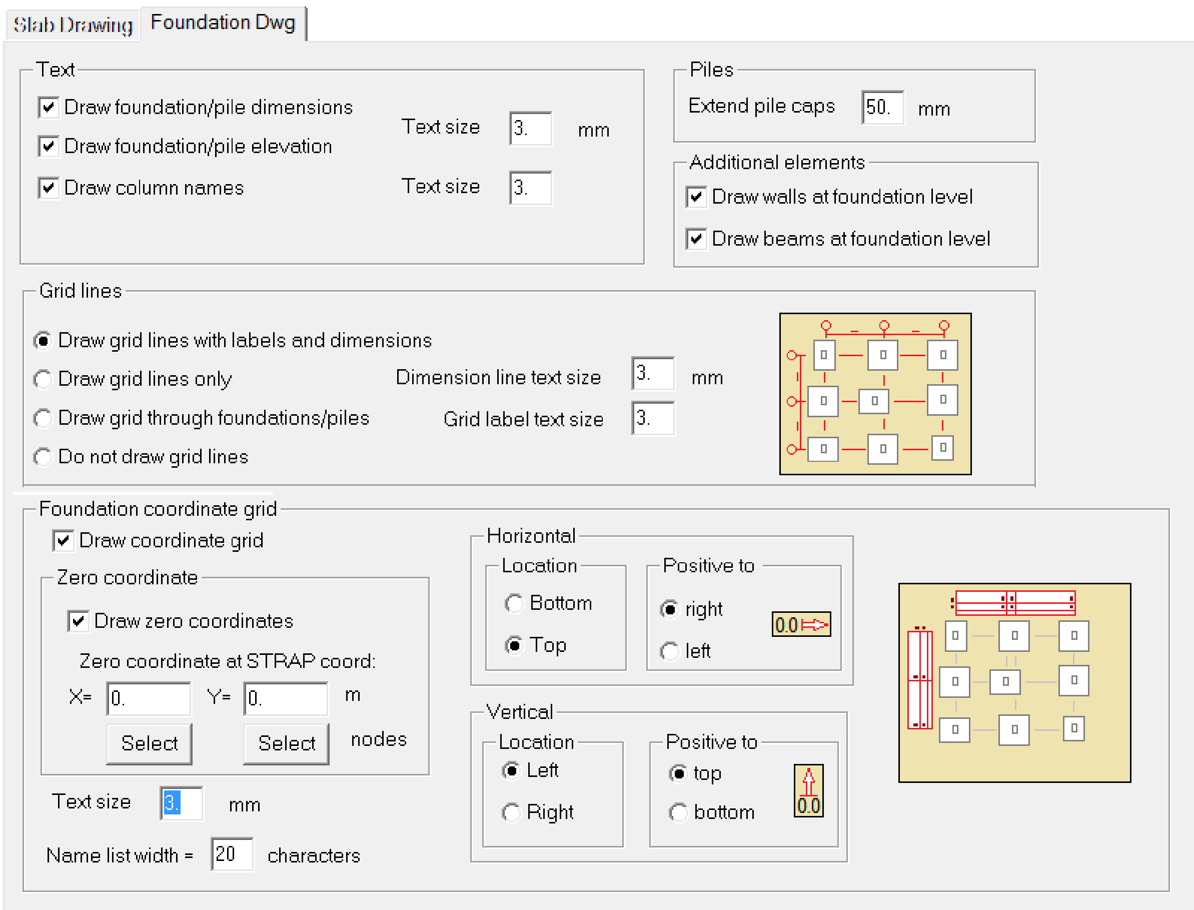
☑ Different bar numbers will be assigned to identical top and bottom bars in the same section.
☐ The same bar number is assigned to identical bars.

Align edge beams



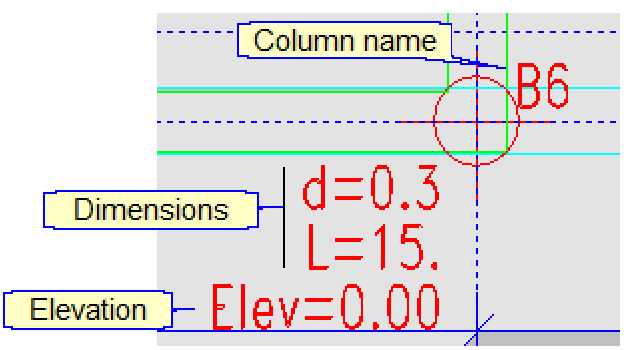
8.9.4.6 Foundation drawing

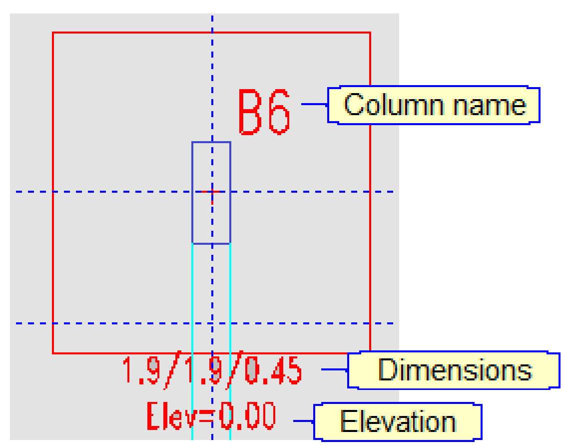
Specify the default foundation drawing parameters.



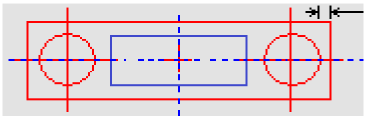
Text

The following footing/pile texts may be added to the drawing; specify the text size.

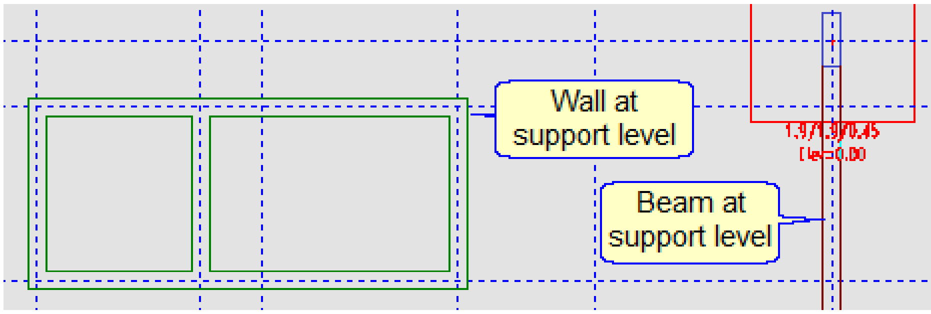




Piles



Additional elements



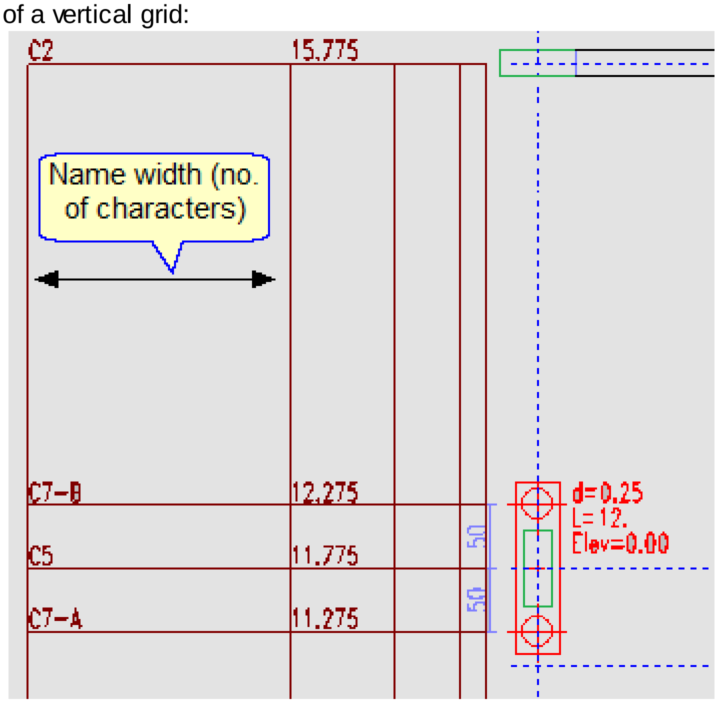
Grid lines

- If grid lines were defined in *STRAP* geometry:
  The grid lines may be added to the drawing, with or without the labels and dimensions

- if grid lines were not defined:
  Dashed lines may be drawn through the centre of every footing and pile.

Coordinate grid

Add a vertical and horizontal grid showing the coordinates of the pile/footing centers. For example, part of a vertical grid:



The options are:

Location:

- Vertical grid can be placed at the left or right side
- Horizontal grid may be placed at the top or bottom

Positive direction:

The positive direction of X,Y coordinates are normally to the right and upwards but may be reversed.

Name width:

The width (no. of characters) of the "name" column.

Zero coordinates:

The "zero" coordinates are by default the zero *STRAP* X,Y coordinates but they may be revised:

- enter the STRAP coordinates at the grid "zero coordinate", or
- click Select and select a node at the zero coordinate location.

Check ☑ **Display zero coordinate** to add 0.00 to the grid even if there is not a pile/footing at that location.

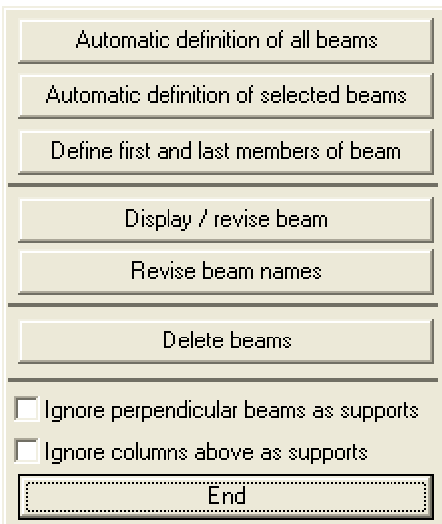
8.10 Define

Define:

- beams and columns
  - beams and columns are defined as continuous chains of *STRAP* members.
  - **only members that are part of a continuous beam or column defined by this option will be designed**.
  - **beams and columns with undefined properties cannot be defined.**
- slabs
  define 'subspaces' in each slab; different slab detailing parameters may be assigned to each subspace.

Define beams 836
Define columns 842
Define slabs 846

8.10.1 Define/display beams



Note:

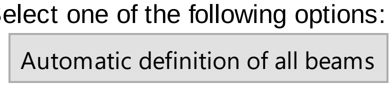
- If no beams have been defined when the **Compute** columns option is selected, the program will automatically define all beams prior to computing.

To display a demo video that explains how to create, design & detail BEAMS and add the beam detailing to drawings:

- *click on*  *to start the video*
- *then click on*  *to enlarge the display.*

8.10.1.1 Define a beam

Select one of the following options:

- 

  The program automatically searches for chains of members perpendicular to the Height axis 799 (as defined in Defaults 799) and defines them as continuous beams.

- Automatic definition of selected beams

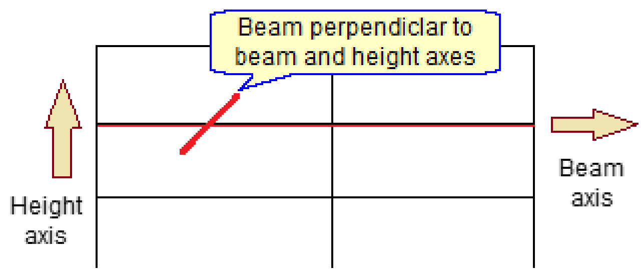
  Select members using the standard Beam Selection option; the program automatically identifies the following members in the chain and defines continuous beams (only members perpendicular to the height axis may be selected).
- Define first and last members of beam

  Select the first and last members in a chain using the standard Beam Selection option; the program automatically identifies the intermediate members in the chain and defines continuous beams (members parallel to the height axis may also be selected).

For all options, check and revise the definition by selecting the Display/revise beams option.

Note:

- Up to 80 *STRAP* members may be included in a single beam.
- the program automatically identifies continuous chains of members (a member is considered as a continuation of the previous member if the angle between the x1 axes of the two members is less than 30°).
- supports are automatically defined at nodes where members parallel to the height axis are connected (a member is parallel if the angle between its x1 axis and the height axis is > 60° and < 120°). The support widths are calculated from the dimensions of the perpendicular member.
- supports may be defined automatically at nodes where members that are ***not*** parallel to the height axis are connected, i.e. perpendiculat to both the beam axis and the height axis. Refer to Ignore supports at perpendicular members 841



- walls at beam ends are also considered as supports
- if no support members are found at the common node between adjacent members, the program automatically "combines" the adjacent members to form a single design span.
- x1 "Offsets" are deducted from the span length.
- the program converts the *STRAP* sections into design section as follows:
  - Beams with Structural Steel sections are ignored by the program, i.e. when automatically creating beams the program assumes that these members are not in the model.
  - Sections defined by properties are considered as "Undefined"; ***the program does not design beams or columns with undefined properties in any of the component spans***.
  - the program automatically creates T-sections or L-sections from rectangular beams defined with an offset from the slab using the Geometry-Properties **Include element strip in beams** option. The flange width for the entire span is the ***minimum effective width*** in proximity to the maximum positive moment.
  - Rectangular and L-sections: dimensions are maintained; geometry flange orientation is maintained
  - Tubes and pipes are considered as "Undefined".
  - T and I-sections:

    major axis - converted as defined; geometry flange orientation is maintained.

    minor axis - converted to rectangular section with similar dimensions
  - U sections:

    major axis - converted to rectangular section with similar dimensions

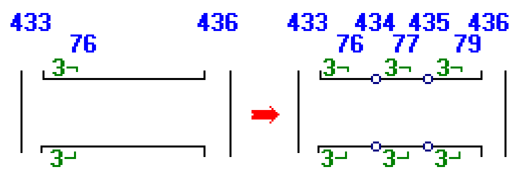
    minor axis - converted to I-section

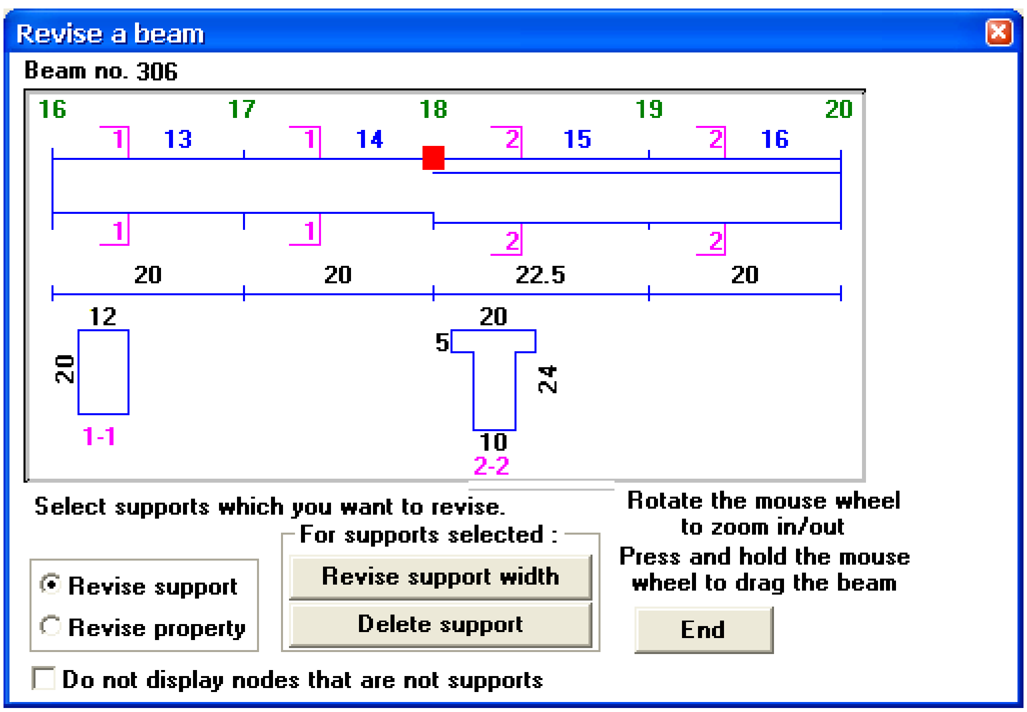
- If the *STRAP* section is not symmetric (T, U and I-sections), the module creates a symmetric section using the minimum dimensions.

8.10.1.2 Display a beam

Use this option to check the defined beams and to -
- revise support widths or delete supports
- invert the section orientation or assign a different property.

Note:
- For clarity, the program does not display the intermediate nodes in spans with multiple members. To display them, set ☐ **Do not display nodes that are not supports**. For example:





To revise the beam:
- select ◉ **Revise support** or ◉ **Revise property**
- Select the spans or supports - move the cursor adjacent to the span/support and click the mouse. Multiple spans/supports may be selected.
- click on one of the option buttons, e.g. Revise support width.

Refer to:
Revise support 845
Revise property 846

Note:
- rotate the mouse wheel to zoom in/out; press & hold the mouse wheel and move the mouse to pan.

8.10.1.3 Revise beam names

Define the beam title:

Titles are automatically assigned to each beam when it is created.

There are two methods for defining the name:

- **Support names**; the beam title is automatically generated by the program from the grid lies or column names (this is usually the default method).
  - The user can specify the format for the title (see below)
  - the format is saved in the registry and is used for all subsequent model.
- **Span names**; the beam title is automatically generated by the program from the span names (Contact your *STRAP* dealer if you prefer this method). The format cannot be revised.

Support names:

There are several general format available:

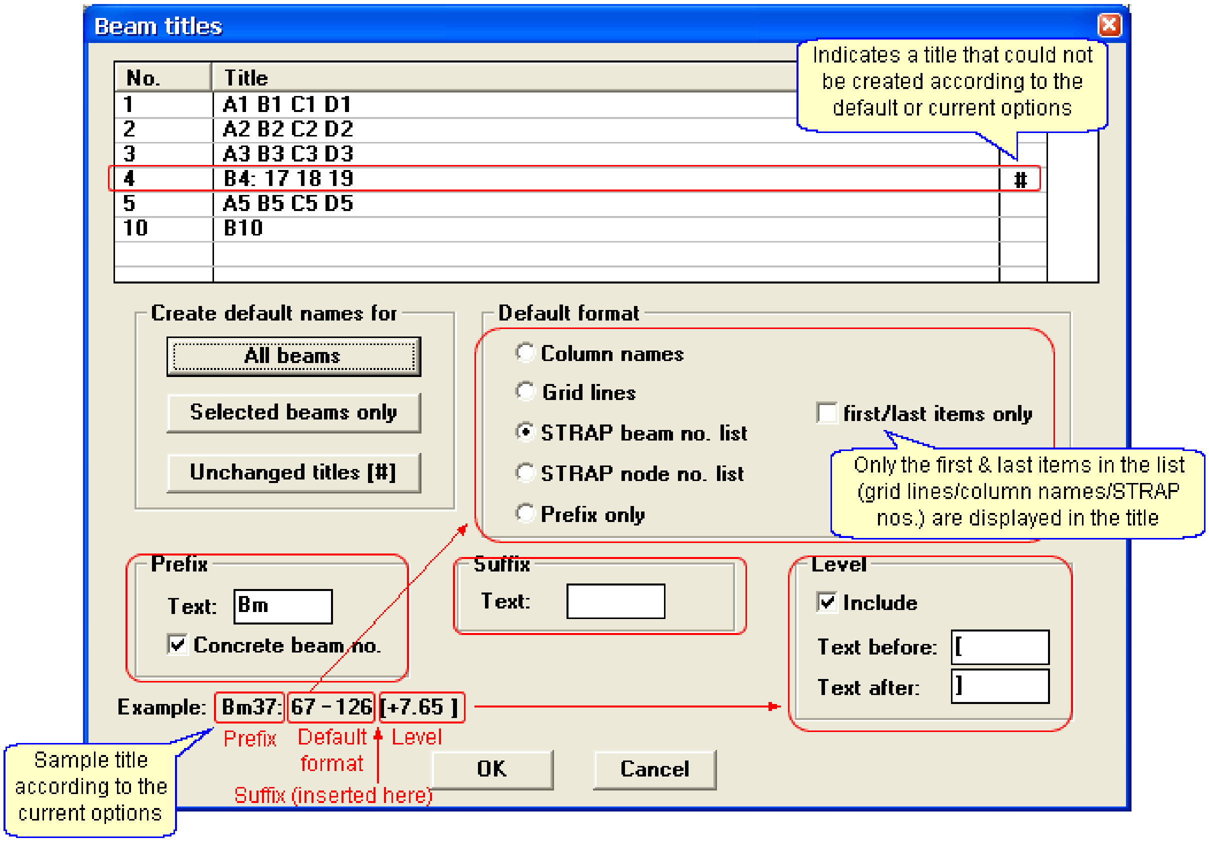
- a list of column names (or grid lines if a column name was not defined at a support location)
- a list of grid lines (or column names if a grid line was not defined at a support location)
- a list of *STRAP* members; this is the default beam name if there are no grid lines and columns.
- a list of *STRAP* node numbers
- A user-defined default text (called the "prefix")

To revise beam names:

- click on a line corresponding to the relevant beam and type in a new name, or -

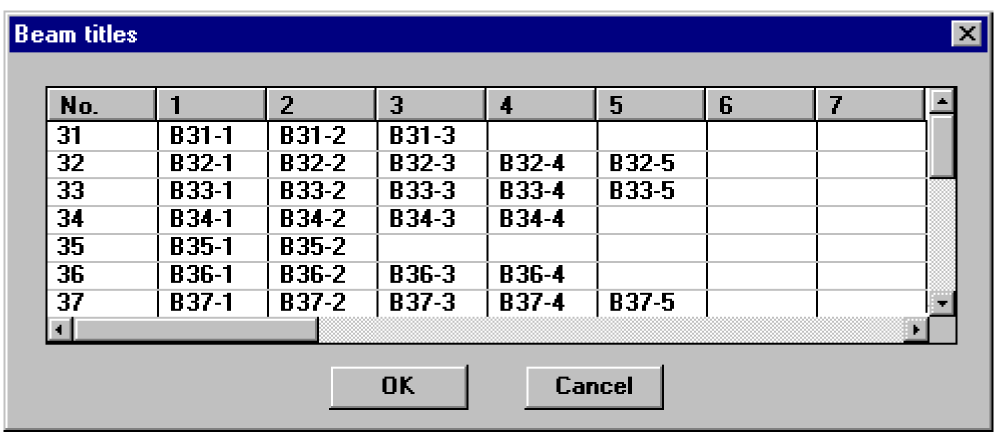
To recreate one or more beams according to the current default format:

- click and highlight several beams in the list, then click [Selected beams only], or -
- click [All beams] to recreate the names for all beams in the model.
- click [Unchanged titles [#]] to create titles for all beams marked with a #. These are titles that could not be generated according to the current format either when the beam was created or when [All beams] was selected in this option.



Span names:

- Default span names are "Bnn-1", "Bnn-2", "Bnn-3", etc. Click on the cell corresponding to the relevant beam and span and type in a new name or revise the existing one
- The default beam name is the beam number followed by the list of *STRAP* members and cannot be revised here.



No.	1	2	3	4	5	6	7
31	B31-1	B31-2	B31-3				
32	B32-1	B32-2	B32-3	B32-4	B32-5		
33	B33-1	B33-2	B33-3	B33-4	B33-5		
34	B34-1	B34-2	B34-3	B34-4			
35	B35-1	B35-2					
36	B36-1	B36-2	B36-3	B36-4			
37	B37-1	B37-2	B37-3	B37-4	B37-5		

Note:
- the beam name is also transferred to the *BEAMD* program.

8.10.1.4 Delete beam/column

Delete a continuous beam or a column select ***any*** member included in the beam/column

8.10.1.5 Ignore supports at perpendicular beams

Ignore perpendicular beams

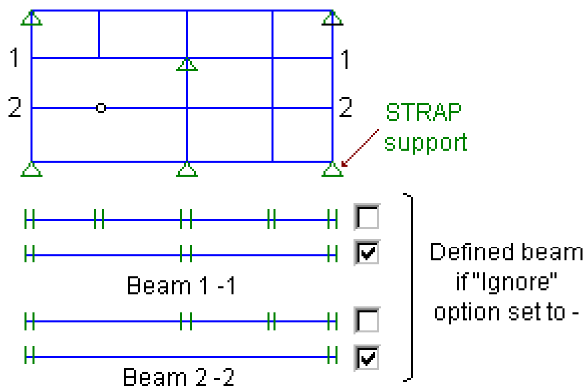
This option is relevant for space frames and grids only.

☐ the program assumes that there are supports wherever *STRAP* "Restraints" were defined

☑ the program assumes that supports are located only where Restraints were defined or where members parallel to the height axis are connected; members not parallel to the height axis are ignored. If no supports (members or restraints) are found, the program automatically add supports at the beam ends.

A member is considered to be perpendicular if the angle between its x1 axis and the height axis > 60° and < 120°).

For example, define beams 1-1 and 2-2 in the following grid:



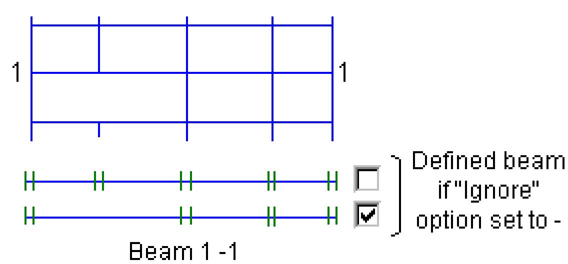
Ignore columns above

This option is relevant for plane frames and space frames only

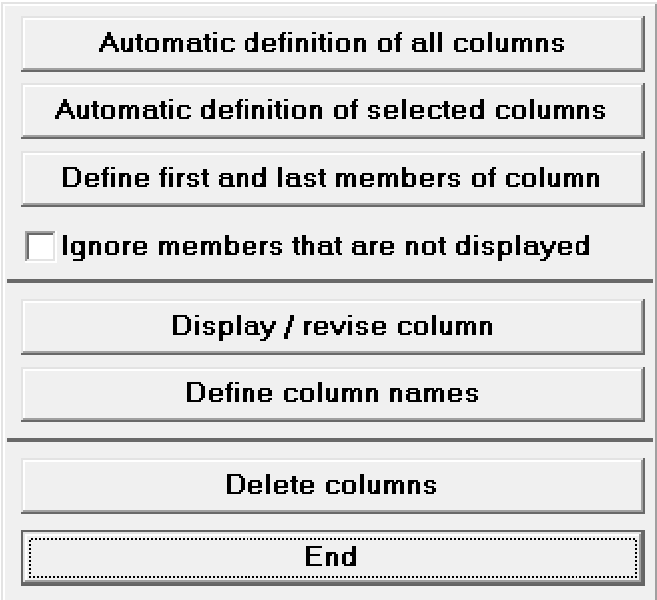
☐ the program assumes that columns above a beam provide a support

☑ the program assumes that columns above a beam are not supports (i.e. the beam supports the column).

For example, define Beam 1-1 in the following frame:



8.10.2 Define/display columns



Note:

- If no columns have been defined when the **Compute** columns option is selected, the program will automatically define all columns prior to computing.

8.10.2.1 Define a column

- Automatic definition of all columns

  The program automatically searches for chains of members parallel to the "Height axis" (as defined in Defaults 809) in the entire model and defines them as continuous columns.

- Automatic defintion of selected columns

  Select members using the standard Beam Selection option; for each member selected the program automatically identifies the previous/following members in the chain and defines a continuous column.

- Define first and last members of a column

  Select the first and last members in a chain using the standard Beam Selection option; the program automatically identifies the intermediate members in the chain and defines a continuous column.

For all options, check and revise the definition by selecting the Display/revise columns option.

Note:

- Up to 80 *STRAP* members may be included in a single column.
- the program automatically identifies continuous chains of members (a member is considered as a continuation of the previous member if the angle between the x1 axes of the two members is less than 30°).
- the program will create columns for members that are not displayed and include them in the member chains unless **Ignore members that are not displayed** is checked.
- supports are automatically defined at nodes where members perpendicular to the height axis are connected (a member is perpendicular if the angle between its x1 axis and the height axis > 60° and < 120°). The support widths are calculated from the dimensions of the perpendicular member.

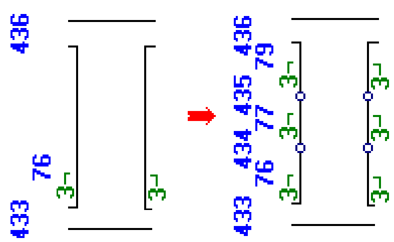
- supports are automatically defined at planes of elements.
- if no support members are found at the common node between adjacent members, the program automatically "combines" the adjacent members to form a single design span.
- x1 "Offsets" are deducted from the span length.
- the program converts the *STRAP* sections into design sections as follows:
  - Rectangular, round, and L-sections: dimensions are maintained.
  - T and U-sections; If the *STRAP* section is not symmetric, the design module creates a symmetric section using the minimum dimensions.
  - Tubes, pipes and I-sections are considered as "Undefined".
  - Solid sections created in *CROSEC* (section generator) are accepted and the dimensions are maintained.
  - the program automatically creates T-sections or L-sections from rectangular beams defined with an offset from the slab using the Geometry-Properties **Include element strip in beams** option. The flange width is the ***effective width defined by the user*** and not the effective width calculated by the program.
- Steel sections, sections defined by properties and tapered sections are considered as "Undefined"; ***the program does not design with undefined properties in any of the component spans***.
- Columns with Structural Steel sections are ignored by the program, i.e. when automatically creating columns the program assumes that these members are not in the model.
- If the sections are different in the members found in a design segment of a column (between supports), the program always uses the section of the ***first*** member in the segment.

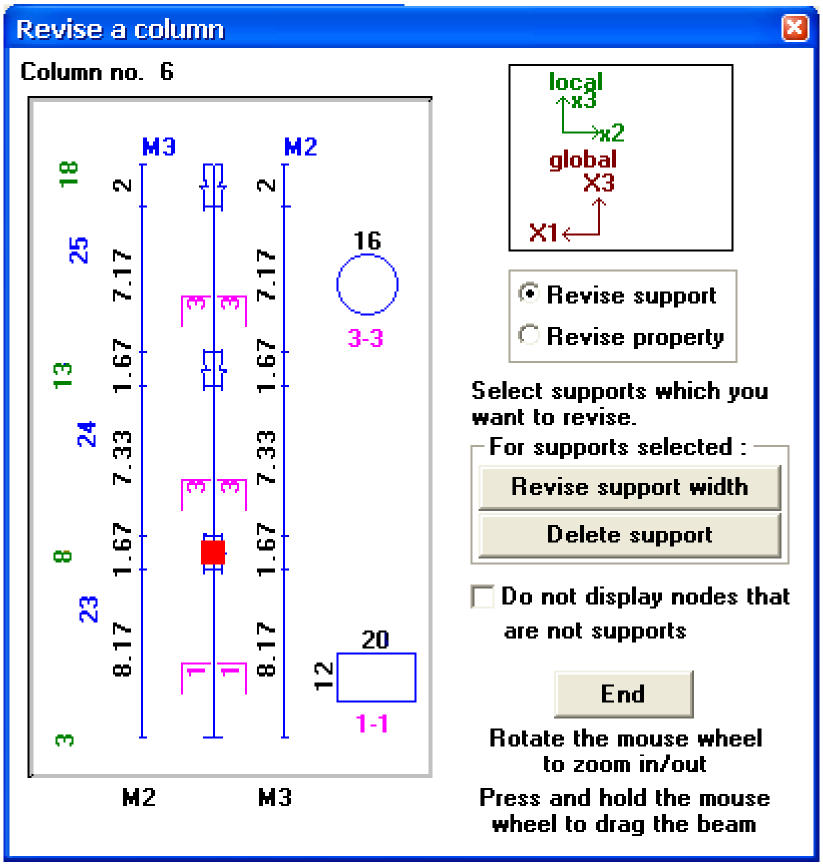
8.10.2.2 Display a column

Use this option to check the defined columns and to -

- revise support widths or delete supports
- invert the section orientation or assign a different property.

- For clarity, the program does not display the intermediate nodes in spans with multiple members. To display them, set ☐ **Do not display nodes that are not supports**. For example:





To revise the column:

- select **Revise support** or **Revise property**
- Select the spans or supports - move the cursor adjacent to the span/support and click the mouse. Multiple spans/supports may be selected.
- click on one of the option buttons, e.g. Revise support width.

Refer to:

[Revise support](845)
[Revise property](846)

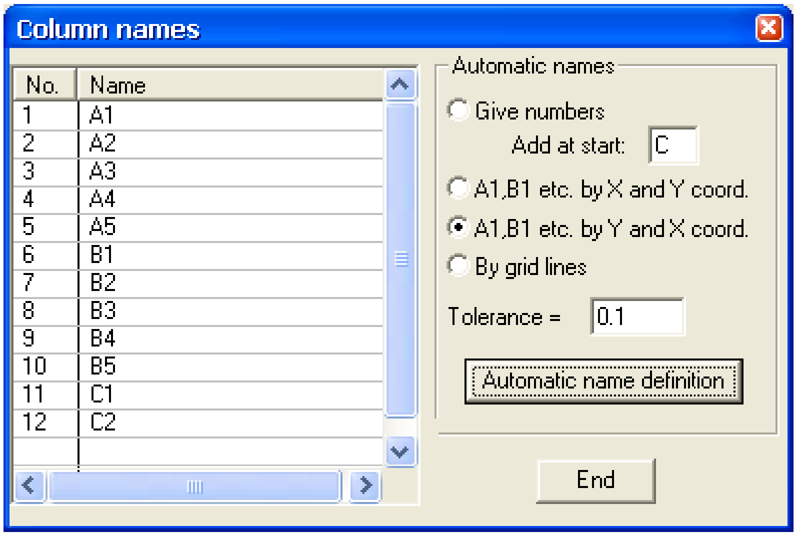
Note:

- rotate the mouse wheel to zoom in/out; press & hold the mouse wheel and move the mouse to pan.
- For clarity, set ☑ **Do not display nodes that are not supports**. For example:

8.10.2.3 Column names

Define names for all columns in the model.

- The program automatically assigns names to the columns and the names may be revised at any time by the user
- The column names are displayed in the column drawings and tables as well as the result tables.



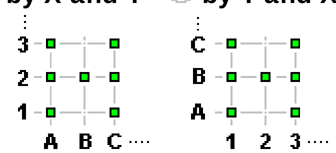
To define the names:

- Select one of the following options:

  Give numbers

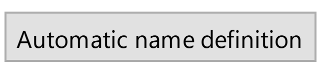
  The program assigns numbers **1,2,3**,.... to the columns in the order of the column numbers along with a user-defined prefix, e.g. C1, C2, etc.

  by X and Y **by Y and X**



  By grid lines

  The program uses the grid line names defined by the user in the Display - grid lines option

- click 

  The program updates all names in the table at the left
- to revise a name, click on the cell and edit the text

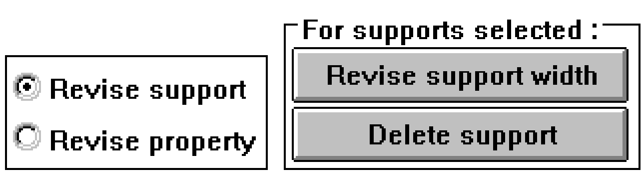
Note:

- two column with a coordinate difference less than the **Tolerance** value are assumed to be on the same line

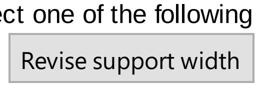
8.10.3 Revise support

Use this option to revise support widths or to delete supports at intermediate nodes.

- select nodes using the standard Node Selection option
- select **Revise support** in the menu at the bottom of the screen:

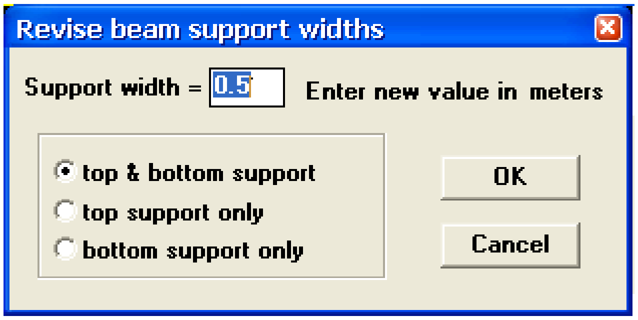


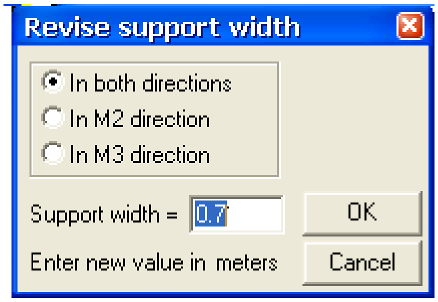
- Select one of the following options:

revise the support width or to change an intermediate node into a support.

For beams, different widths may be defined above and blow the beam. For columns, a different support width may be defined in the M2 and M3 directions:





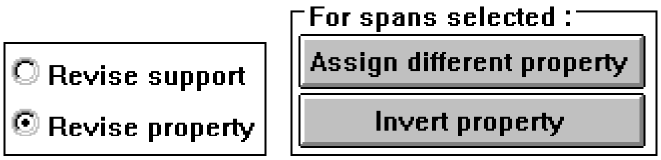
- Delete support

  change a support to an intermediate node.

8.10.4 Revise property

Use this option to assign a different property group to spans or to switch the flange locations in the current property group.

- select members using the standard Beam Selection option
- select **Revise property** in the menu at the bottom of the screen:



- Select:
  - Assign different property

    assign a different property group to a span. Note that if you want to revise the property groups for many spans, it may be more convenient to select Properties in the main menu.
    To define new property groups, to revise dimensions or switch major/minor axes, refer to Properties.

- Invert property

  reverse the flange location for T,L and U-sections. Note that this option does not switch the major and minor axes.

  This option is equivalent to the "flange orientation" option in Parameters.

8.10.5 Define slabs

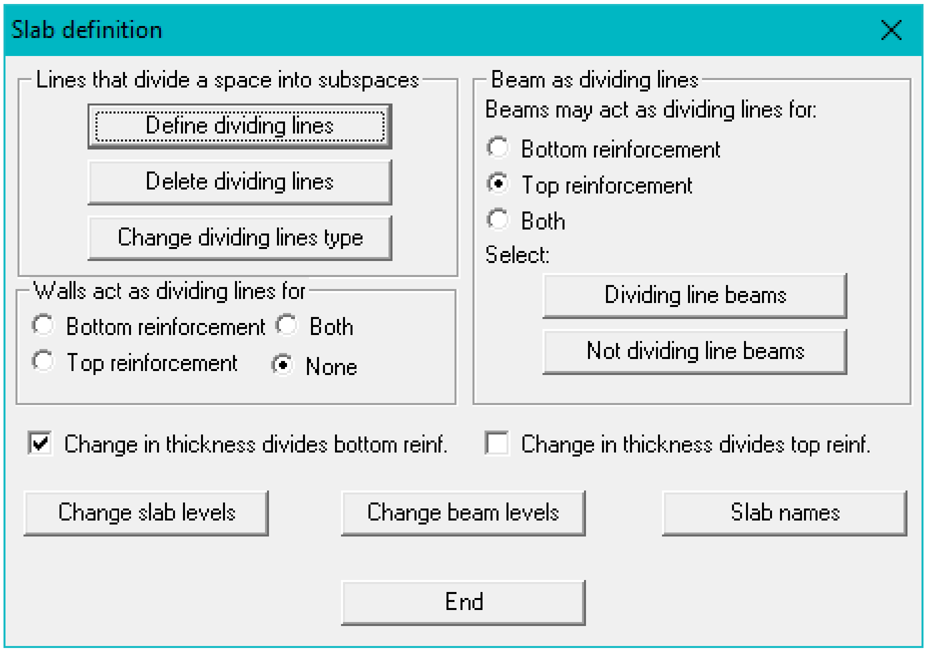
The slab can be divided into several subspaces by 'dividing lines'. Each dividing line represents a boundary for slab reinforcement

- different dividing lines may be defined for top and bottom reinforcement

- different parameters may be defined for each space
- reinforcement terminates at a dividing line (with optional overlap length)
- dividing lines may be defined along element boundaries or at beam locations

Note:
- by default, each level is a different space.
- dividing lines that do not extend at both ends to a slab contour or another dividing line are ignored.

Refer to Draw slabs - general 794

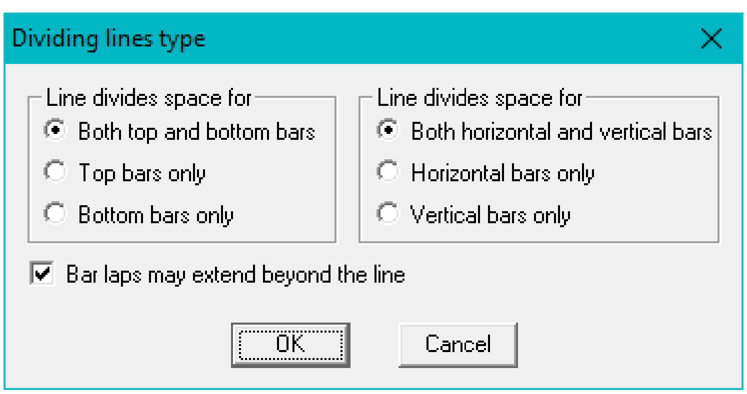


Define / change dividing lines

Divide a slab into several subspaces by defining dividing lines:
- different parameters may be assigned to each subspace.
- subspaces can divide top reinforcement only, bottom reinforcement only or both.
- subspaces can divide vertical (on plan) reinforcement only, horizontal reinforcement only, or both.

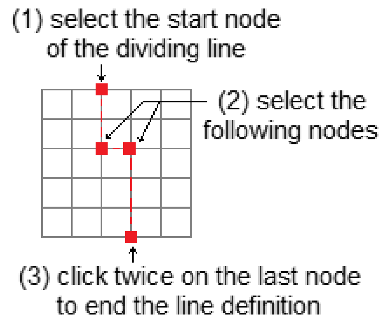
When revising an existing line:
- select 851 one line or several lines

To define/revise the line:
- select the dividing line type - it can divide top and/or bottom bars, bars in the X and/or Y direction:

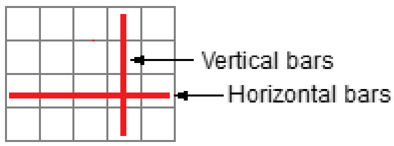


When defining a line:

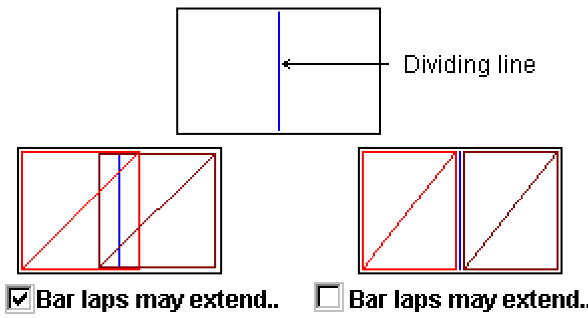
- select two nodes defining the dividing line or define a chain of lines to create an irregular space. Click on the last node twice to end the selection.



- "Horizontal" and "vertical" refer to the display (in plan):



- Bar laps may extend beyond the dividing line:



Note:

- dividing lines that do not extend at both ends to a slab contour or another dividing line are ignored.

Delete dividing lines

To delete dividing lines:

- select 851 one or more dividing line

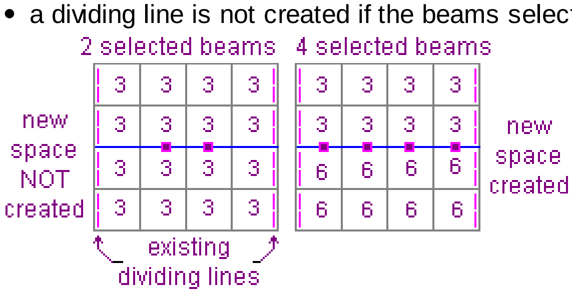
Beam dividing lines

Beams may be defined as dividing lines for top and/or bottom reinforcement. Use these options to create or remove beam dividing lines

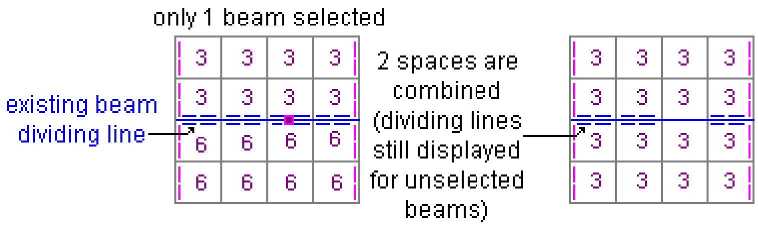
- select beams using the standard beam selection option

Note:

- a dividing line is not created if the beams selected do not extend to other dividing lines. For example:



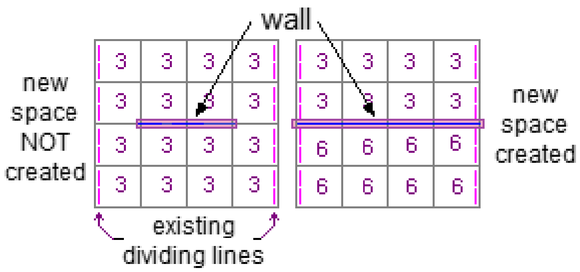
- removing a single beam from a line of beams is sufficient to combine two spaces. For example:



Wall dividing lines

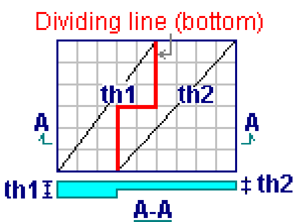
Walls may be defined as dividing lines for top and/or bottom reinforcement. ***All*** walls are used as dividing lines if this option is activated (individual walls cannot be selected).

Note:
- a dividing line is not created if the wall does not extend to other dividing lines. For example:



Change in thickness

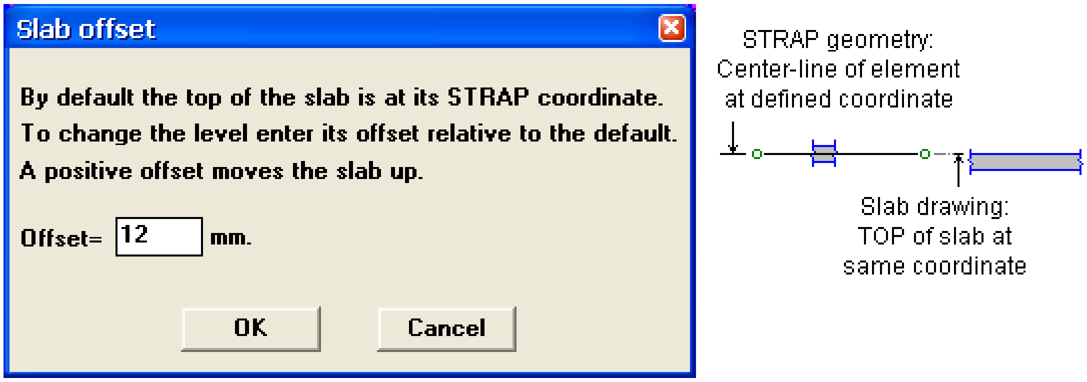
Create dividing lines for top or bottom reinforcement at element boundaries where the thickness changes. For example:



Note:
- all locations with thickness change are dividing lines
- dividing lines resulting created by this option are not displayed by the **Data display | Slab** dividing lines option in the toolbar

Change slab levels

Revise the elevation of the top of the slab in a subspace. By default the program draws the top of the slab at the element coordinate:

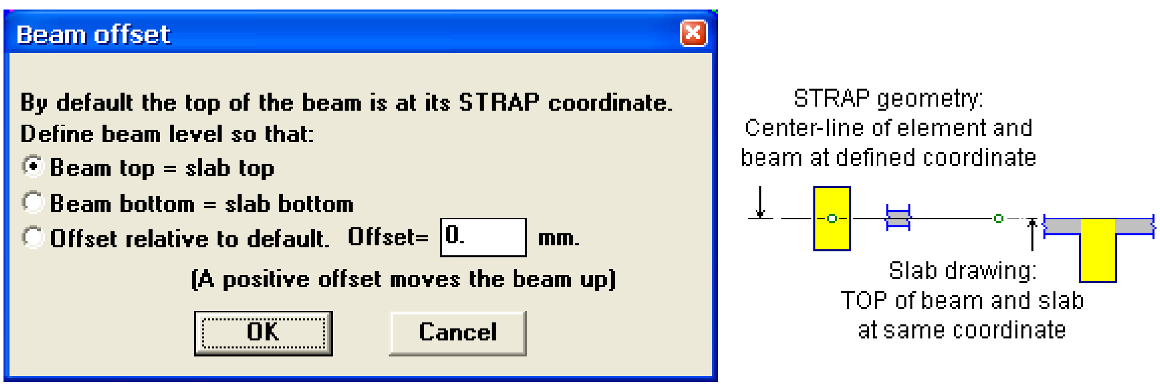


Select elements using the standard element selection option:

- only one element per subspace has to be selected.
- a positive offset is in the same direction as the positive direction of the relevant global axis.
- the level is used in the following options:
  - slab section 906 drawings
  - display slab level 895 on drawing

Change beam levels

Revise the elevation of the top of the beam. By default the program draws the top of the beam at the beam coordinate thereby aligning it with the top of the slab:



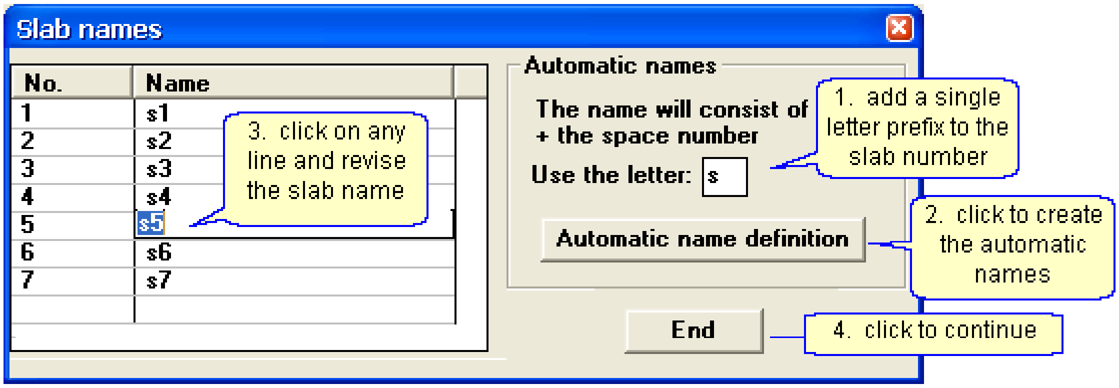
Align the beam bottom with the slab bottom or specify any vertical offset from the default location:

- a positive offset is in the same direction as the positive direction of the relevant global axis.
- if the two adjacent slabs are not at the same level, the beam top is aligned with the higher slab top; alternatively, the beam bottom will be aligned with the lower slab bottom.
- the offset is used when drawing beam sections.

Select beams using the standard beam selection option.

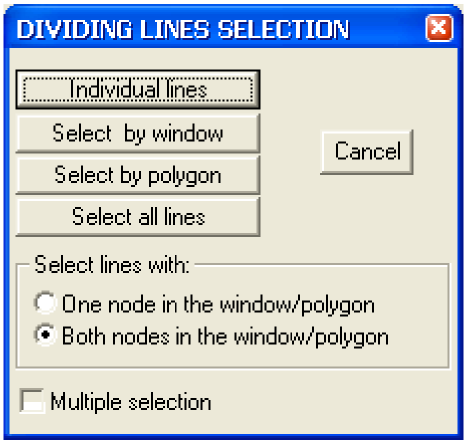
Slab names

Assign a name to the slab subspaces. The names may be displayed on the slab drawings (refer to Slab - edit - parameters 895).



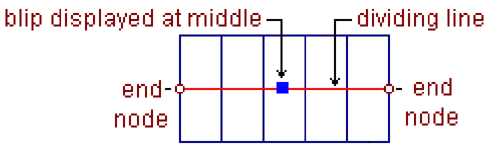
8.10.5.1 Select dividing lines

The dividing line selection options are similar to the standard beam selection options:



Note:

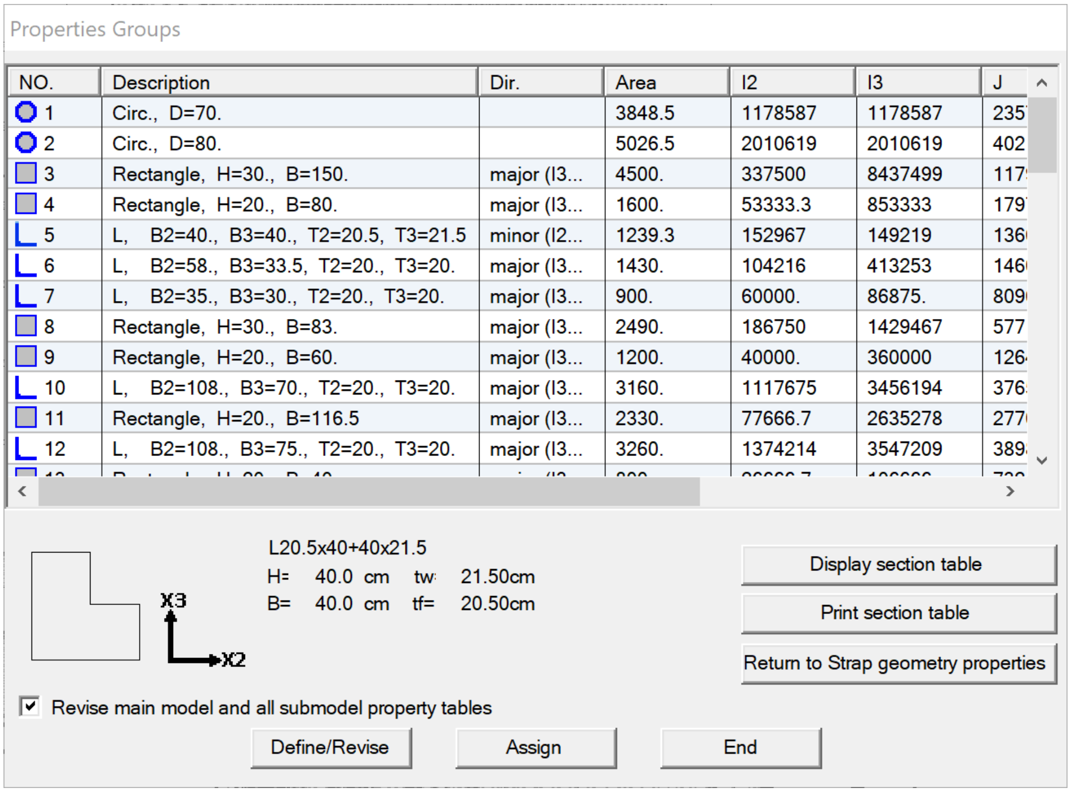
- the ■ is displayed at the centre of the dividing line; 'nodes' refers to the end nodes selected to define the line. For example:



8.11 Properties

Use this option to:

- Revise the section dimensions defined in *STRAP* geometry.
- Define new sections.
- Assign section types to selected members.
- Define corner bars and reinforcement groups for *STRAP* solid sections



Define/revise section

Refer to Properties - Define/revise 853

Display print section table

Display/print a list of the property groups and the section dimensions.

Assign sections to beams

Select a property group number that you want to assign to selected members. The program displays a list of the property groups (defined and undefined).

Select beams that this property is to be assigned to using the standard Beam selection option.

Note:

- An ***-Undefined-*** property may be assigned to beams/columns; the section properties may be defined later. However the program does not compute beams/columns with a member having an undefined property.
- If the sections are different in the members found in a design segment of a column (between supports), the program always uses the section of the first member in the segment.

Return to STRAP properties

Restore ***all*** dimensions and all property group assignments to those in the current *STRAP* geometry. New sections defined in the concrete module are not deleted from the property table.

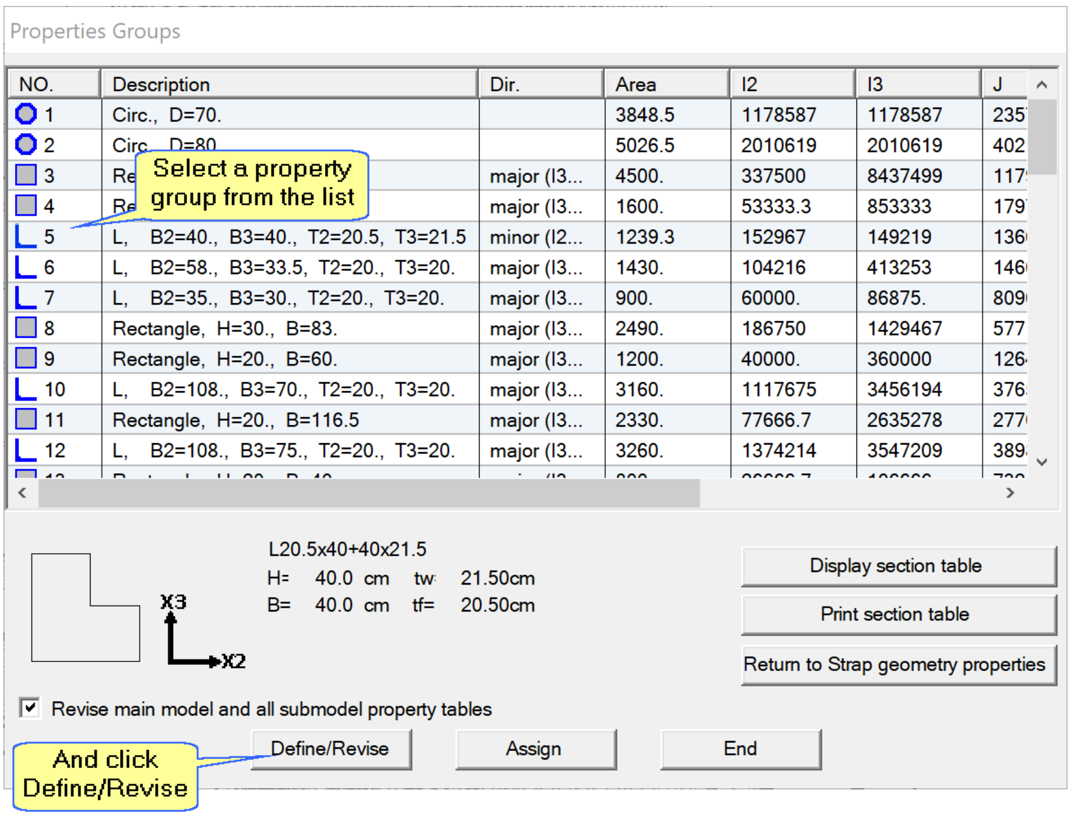
STRAP solid sections

Refer to Properties - solid sections 854.

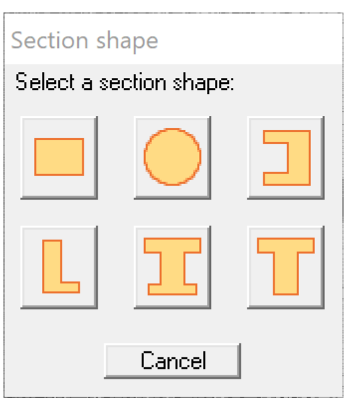
8.11.1 Define /revise

To define or revise a property:

- select a property group from the list displayed on the screen and click Define/revise:

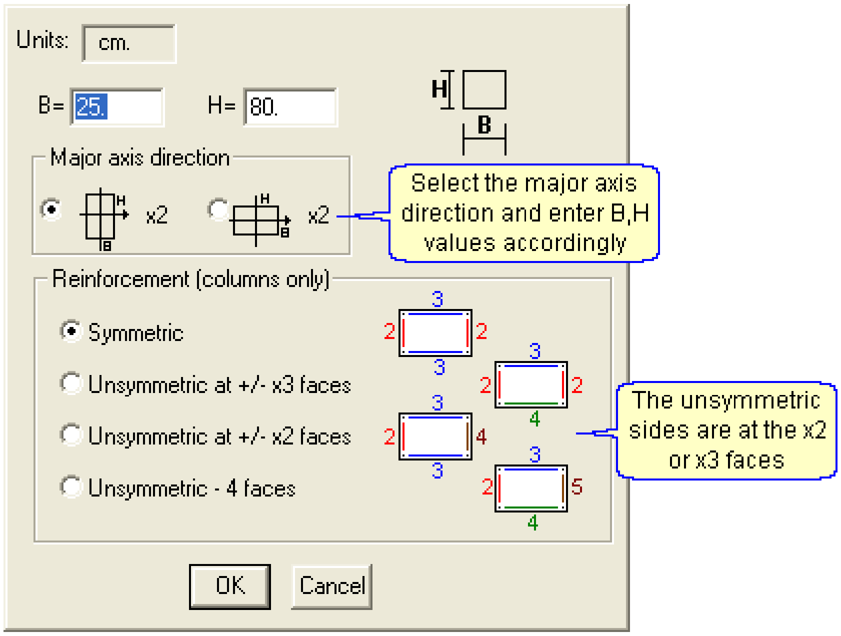


- select a section type:



- define the dimensions:

  The reinforcement group pattern 798 can be specified for rectangular sections, either the default symmetric arrangement or various non-symmetric arrangements.



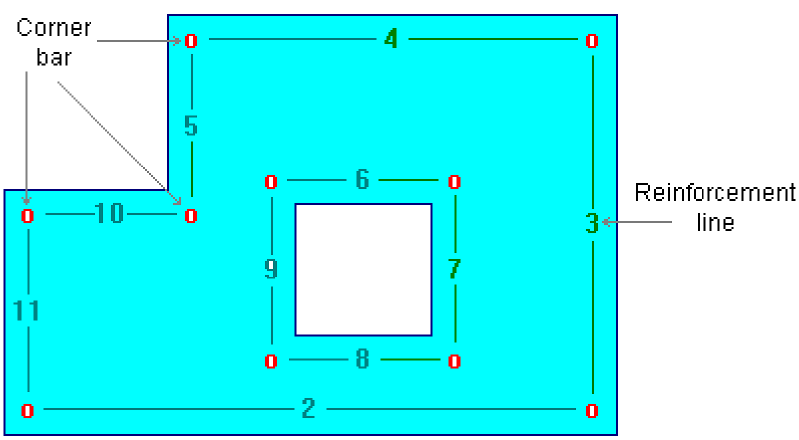
8.11.2 Solid sections

Define corner bars and reinforcement groups for *STRAP* solid sections.

Note:
- these solids sections can only be defined in *STRAP* geometry using the Section generator (*CROSEC*) option.

The program calculates a 'default' bar arrangement for each solid section which may be modified by the user:
- a corner bar is attached to each corner; all corner bars are in Group 1.
- a reinforcement line, each in a different group, is placed between each pair of adjacent corner bars.

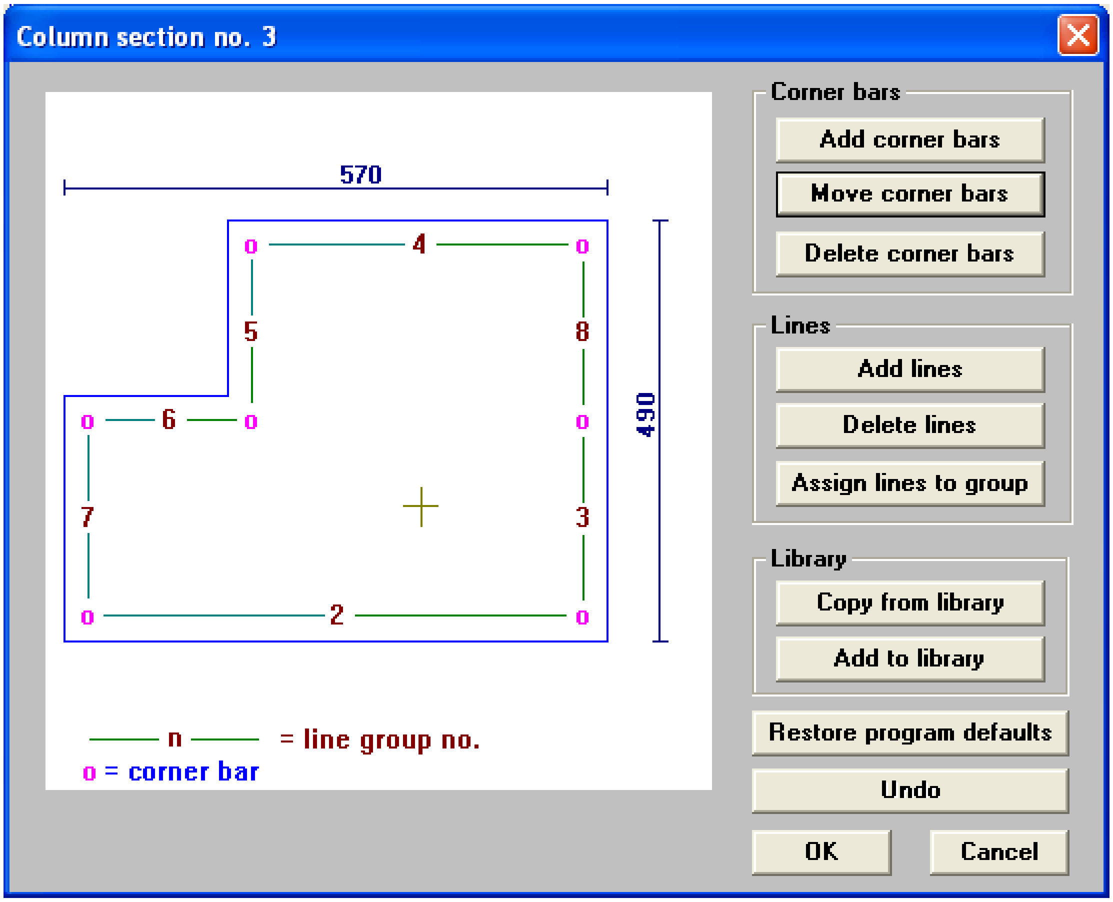
For example:



The following modifications may be made:
- Corner bars — add or delete bars
- Lines — add or delete lines, assign lines to a group

The arrangement may be saved in a the 'solid section library'; the arrangement may be retrieved for a similarly shaped column from the section library.

Select a **-Solid section-** from the list displayed.

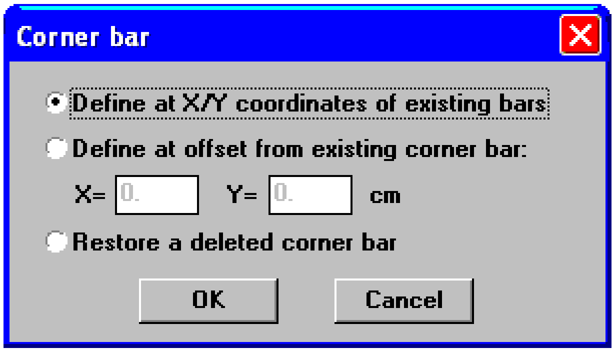


If the arrangement is retrieved from the Section library, Try again may be displayed at the bottom of the screen, indicating that another similar arrangement is available; refer to Copy from a library 858.

Refer also to Column reinforcement - groups 798.

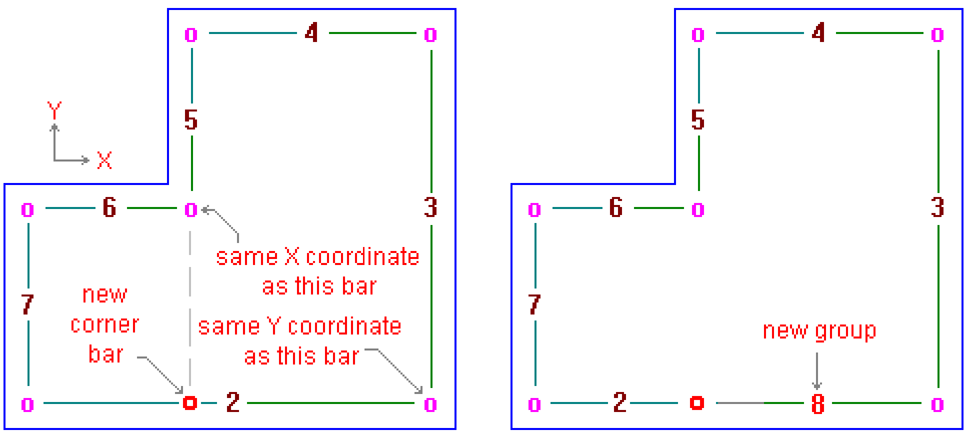
8.11.2.1 Add corner bars

Define a new corner bar or restore a deleted default corner bar.



Define at X/Y coordinates

New corners are defined relative to existing corner bars; the X and Y coordinates are defined separately. For example:



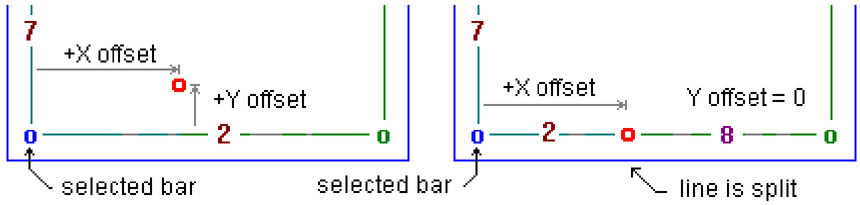
- Select **Define at X/Y coordinates of existing bars**
- click OK
- move the ■ to a corner with the same X coordinate and click the mouse
- move the ■ to another corner with the same Y coordinate and click the mouse
- The program adds the corner bar at the selected location.

Note:
- if the new corner bar lies on an existing line, the program automatically divides that line into two separate ones (see example above).

Define at offset

Define a new corner bar off set from any existing corner bar:
- enter the X and Y off set values (see example below for positive direction)
- select a corner bar

Note:
- if the new corner bar lies on an existing line, the program automatically divides that line into two separate ones



- the program does not allow bars to be created outside the section.

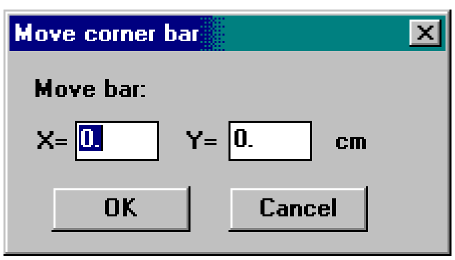
Restore a deleted bar

Restore a deleted corner bar:
- Select **Restore a bar at corner**
- click OK
- move the ■ to a corner where the bar was deleted and click the mouse
- The program adds the corner bar at the default location.

Note:
- the program does not restore any deleted reinforcement lines.

8.11.2.2 Move corner bars

Define an offset for any existing corner bars:



- Define the offsets and click OK
- The program redraws the bar and all attached reinforcement lines in their new locations

Note:
- the program does not allow bars to be moved outside the section.

8.11.2.3 Delete corner bars

Delete any corner bar:
- move the ■ to any existing corner bar and click the mouse.
- the program deletes the bar and any connected reinforcement lines.

8.11.2.4 Lines

Add lines

Add a new reinforcement line between any two existing corner bars:
- move the ■ to any existing corner bar and click the mouse.
- move the ■ to any other existing corner bar and click the mouse.

The program draws a new reinforcement line between the two corner bars.

Note:
- the program checks whether the line is outside the section

- the program does not check whether two lines intersect.

Delete lines

Delete a reinforcement line.

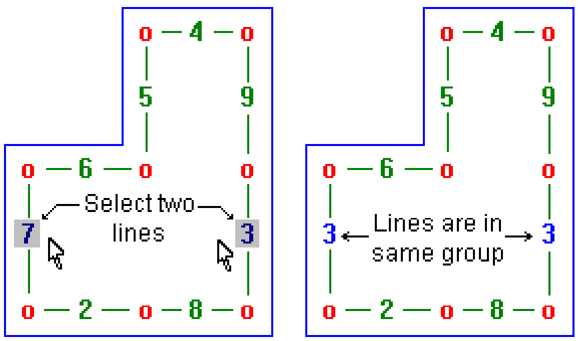
- move the ■ to any existing reinforcement line and click the mouse.

Note:

- A section must contain at least one line, i.e. cannot consist only of corner bars. Any bars placed by the program in an unnecessary line can be deleted by the user in the Specify reinforcement 953 option.

Assign lines

Combine several reinforcement lines into a single group. For example:



S

Note:

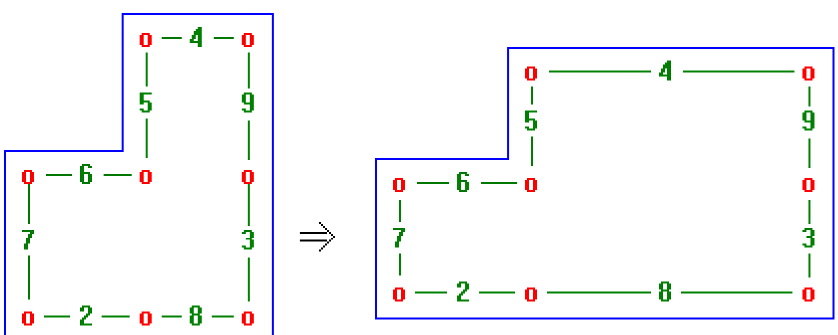
- the selected lines must be of equal length.

8.11.2.5 Library

Copy from library

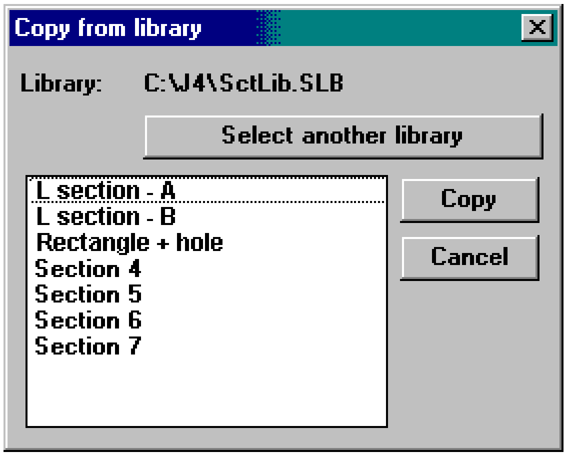
Retrieve a saved reinforcement pattern from the library.

The library section must be similar to the current section, i.e. it must have the same number of corners and the same pattern of internal angles. The program copies the arrangement of corner bars and reinforcement lines from the library section to the current section. For example:



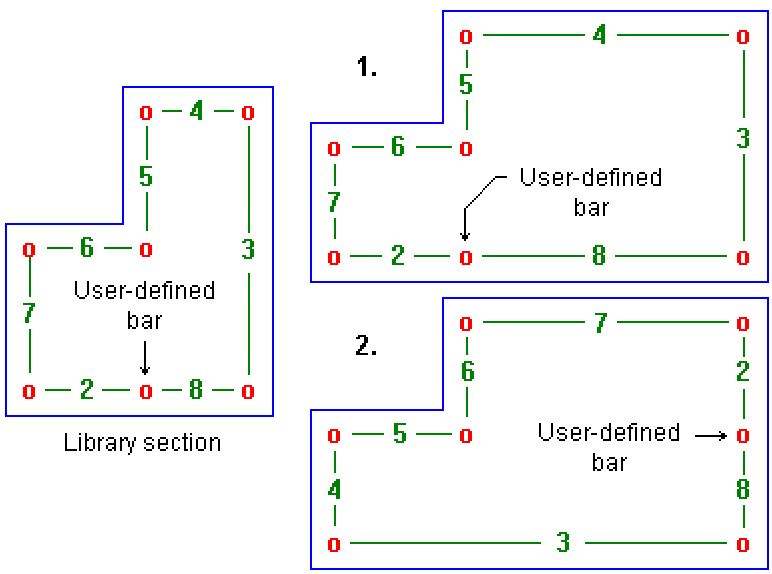
Library section

Current section



- the default library is **SctLib.SLB** in the current directory; click Select another library to choose a different file.
- Click and highlight the section, then click Copy to retrieve the section from the list

Note:
- More than one configuration may be retrieved from a particular library section; for example:

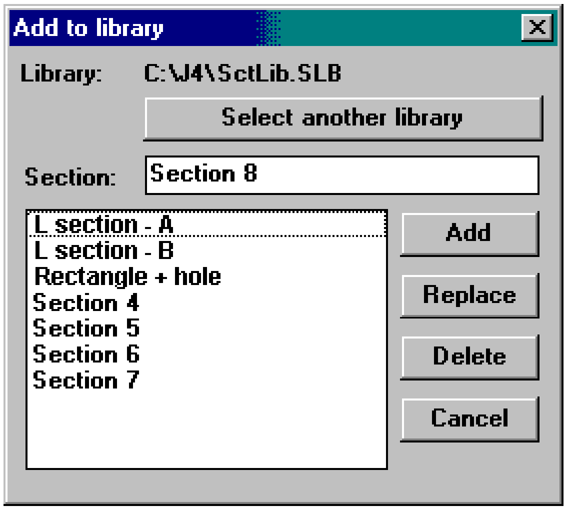


Library section

If the first configuration displayed is not the correct one, click Try again at the bottom of the dialog box.

Add to library

Save the existing pattern of corner bars and reinforcement lines; the pattern may be recalled later and used for a similar solid section (same general shape but different dimensions). Refer to Copy from library 858 for more details.



- the default library is **SctLib.SLB** in the current directory; click Select another library to choose a different file.
- revise the section name (the default name is '**Section *n***')
- Click Add to append the section to the list, or -
- to replace an existing section, click and highlight that section, then click Replace.
- to delete an existing section, click and highlight that section, then click Delete.

8.11.2.6 Restore / undo

Restore program defaults

Erase all of the changes made to the reinforcement in this section and restore the initial program defaults.

Undo

Undo the previous revision to the reinforcement in this section.

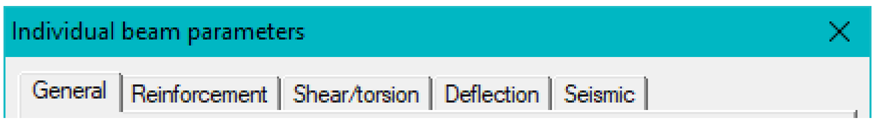
8.12 Parameters

Define the design parameters ***for specified members, elements***, etc. If a parameter is not defined for a specific member using this option, the program uses the default value when designing the member.

Beam parameters 861
- Column parameters 865
- Wall parameters 872
- Slab parameters 869

8.12.1 Beam parameters

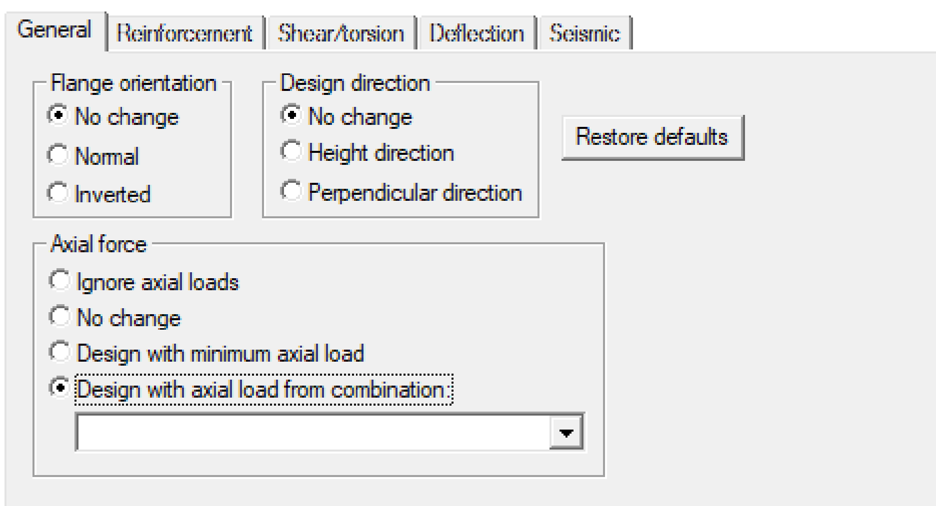
Specify ***different*** parameters for specific beams. The program uses the Default parameters for ***all*** beams except those with different parameters specified using this option.



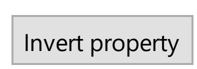
General 861
Reinforcement 862
Shear 863
Deflection 864
Modify reinforcement 864
Seismic 820

8.12.1.1 General

Define beam design parameters for selected beams. Values specified here override the model default parameters specified in the Default 799 option.



Flange orientation



To reverse the flange location for T,L and U-sections. This option is equivalent to the options in "**Display/revise/beam**".

Note that this option does not switch the major and minor axes, i.e. 0 may be revised to 2 , but not to 1.

To switch the axes, use the 

option in the main menu.

Design direction

The program design beams for uniaxial bending only.

Specify the design direction - M2 or M3; moments in the other direction are ignored. Beams with significant moments in both directions should be defined as columns.

Note:

- For "**Design direction**", the parameter specified for the ***first*** member in the beam are used for the entire beam.

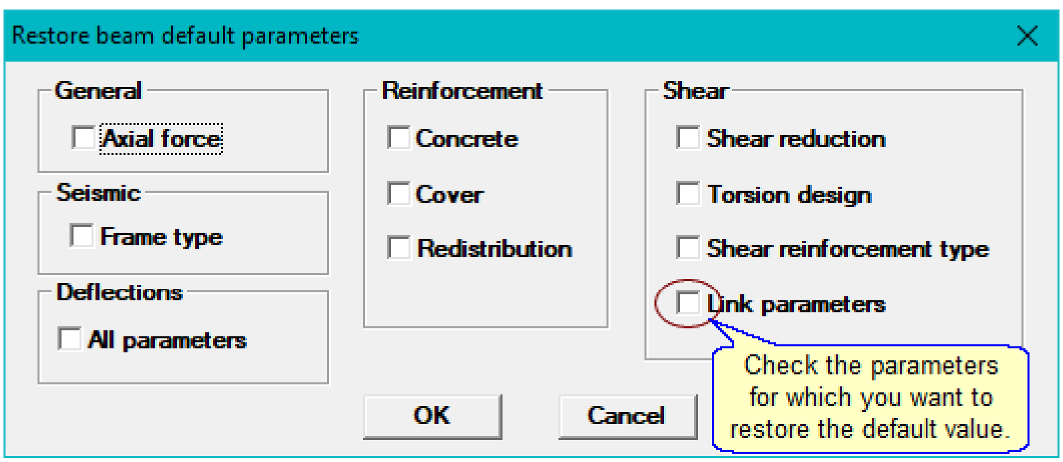
Axial loads

The program designs beams for a combination of moments and axial force. However because the program calculates a bending moment envelope it cannot check each combination separately for combined moment and axial force.

Refer to Default parameters - axial 800

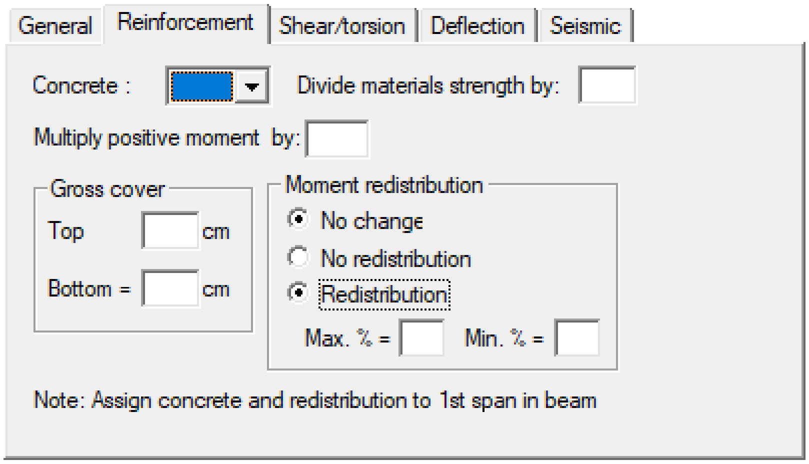
Restore defaults

Restore the current default values 799 to selected beams. Values may be restored for selected parameters:



8.12.1.2 Reinforcement

Define beam design parameters for selected beams. Values specified here override the model default parameters specified in the Default 799 option.



Concrete

Refer to Beam default - concrete 801

Divide materials strength by

Refer to Beam default - divide 801

Multiply positive moment

Refer to Beam default - multiply positive 801

Cover

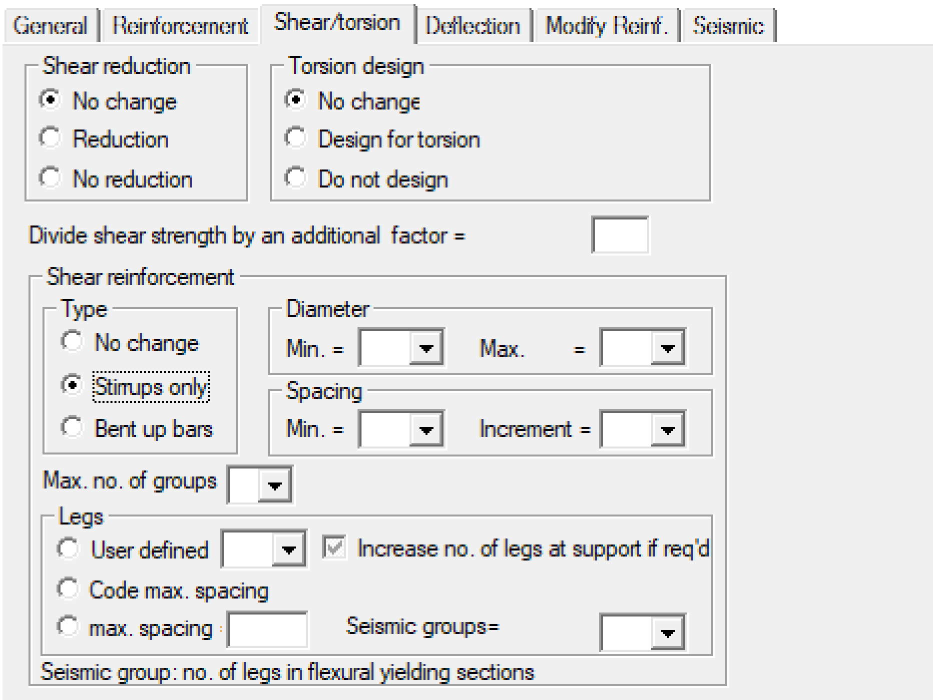
Refer to Beam default - cover 801

Moment redistribution

Refer to Beam default - redistribution 801.

8.12.1.3 Shear

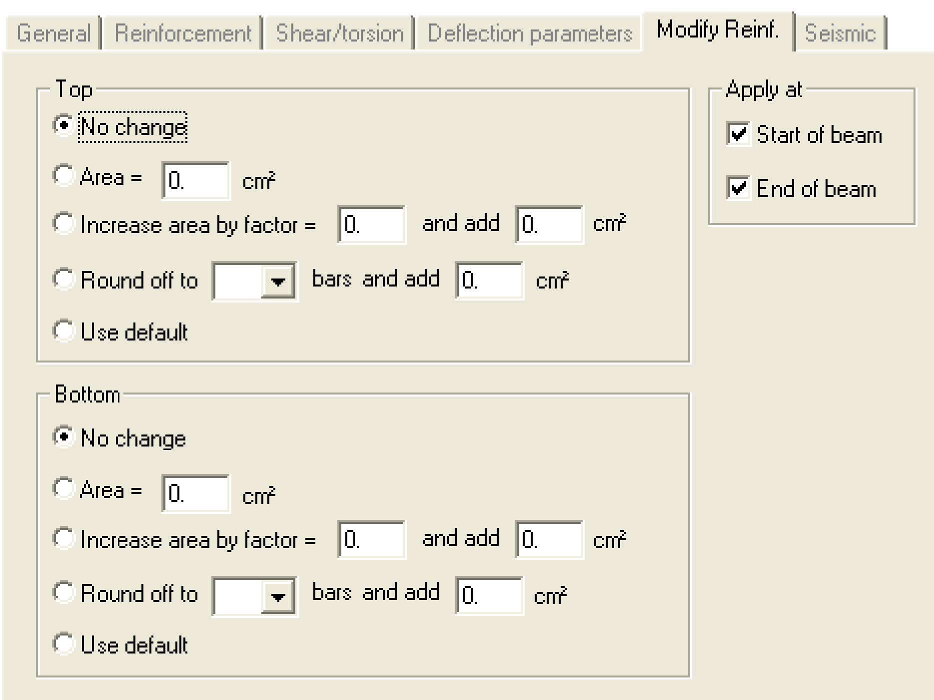
Define beam design parameters for selected beams. Values specified here override the model default parameters specified in the Default 799 option.



8.12.1.4 Deflection

8.12.1.5 Modify reinforcement

This option allows you to specify an reinforcement area at either end of any beam, top or bottom:



- Select :

Start/end of beam
Select the end (JA/JB) nodes at which the reinforcement is to be modified.

- select a reinforcement option:

Area =
Specify the increased area according to the units displayed

Increase area by factor and **add**
Example: area required = 765 mm², factor = 1.1; add = 100 m²
Increased area = 1.1(765)+100 = 941 mm²

Round off to **bars and add**
Example: area required = 765 mm², bars = 15 mm (177 mm²); add = 100 m²
765/177 = 4.32: round off to 5 bars
Increased area = 5(177)+100 = 985 mm²

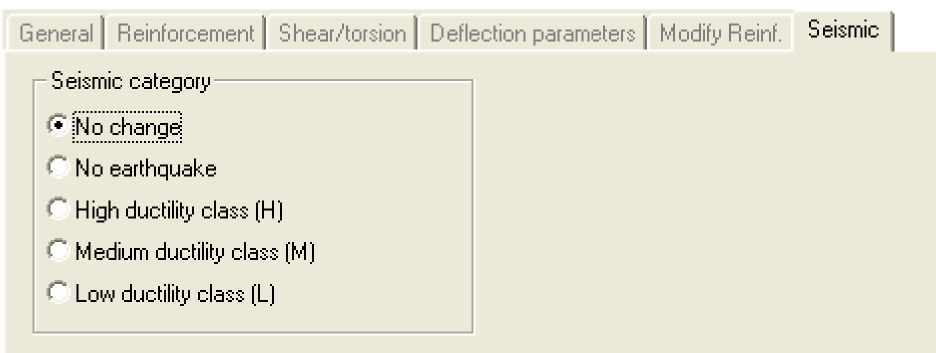
No change
Do not revise the current modified reinforcement value at the selected end or/or face of the beam.

Use default
Use the area specified in the "Default" option.

- click OK and select the beams.

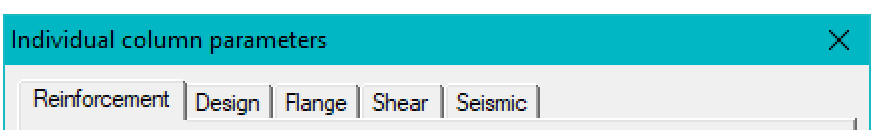
8.12.1.6 Seismic

Specify the frame type for selected beams. Values specified here override the model default type specified in the Default 808 option.



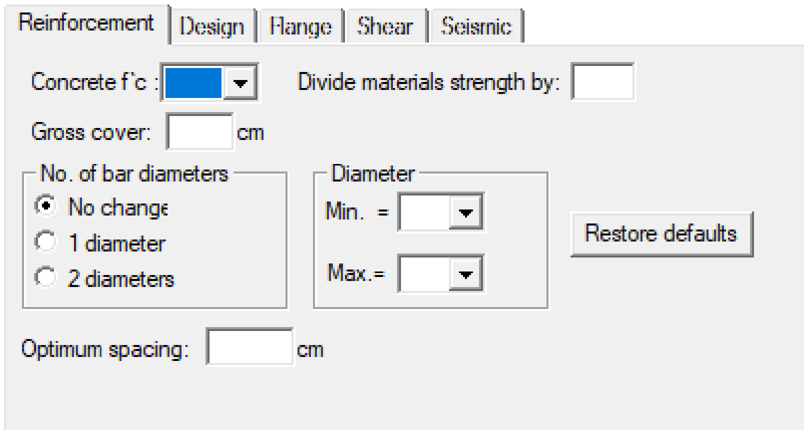
8.12.2 Column parameters

Specify ***different*** parameters for specific columns. The program uses the Default parameters for ***all*** columns except those with different parameters specified using this option.



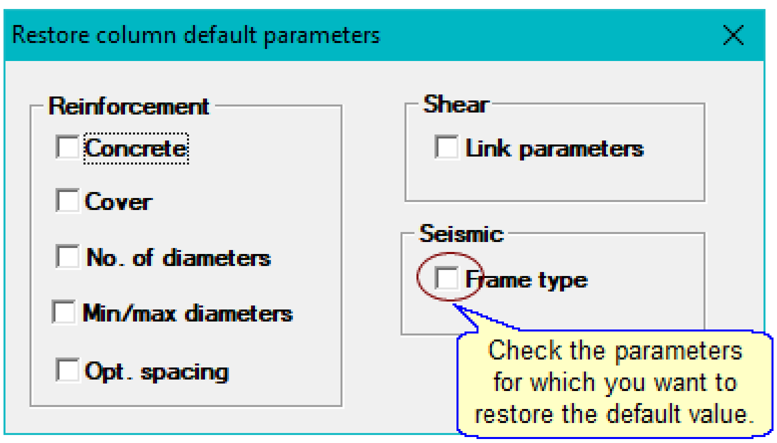
8.12.2.1 Reinforcement

Define column design parameters for selected columns. Values specified here override the model default parameters specified in the Default 809 option.

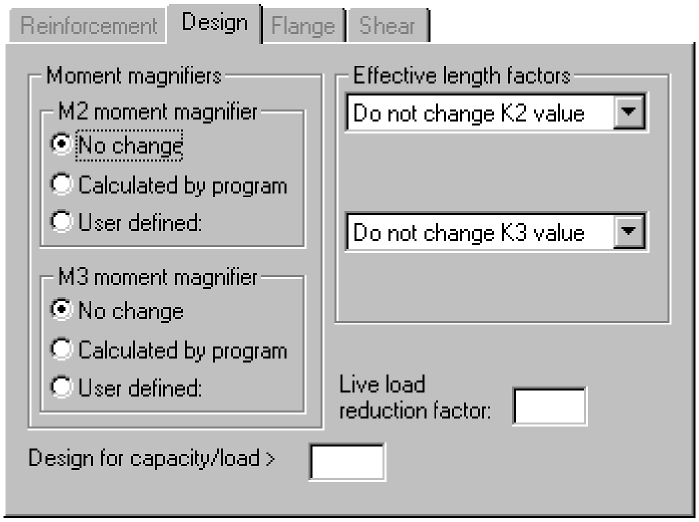


Restore defaults

Restore the current default values 809 to selected columns. Values may be restored for selected parameters:



8.12.2.2 Design



Moment magnifiers

Specify the method to calculate the additional/magnified moment for specific slender columns and walls.

Calculated by program

User defined

Define the moment magnifier or additional moment for slender columns or walls in terms of a factor δ to multiply the relevant factored moment, i.e.

- braced: $\mathbf{M_d = \delta \cdot (0.6 \cdot M_2 + 0.4 \cdot M_1)}$
- unbraced: $\mathbf{M_d = \delta \cdot M_{end}}$

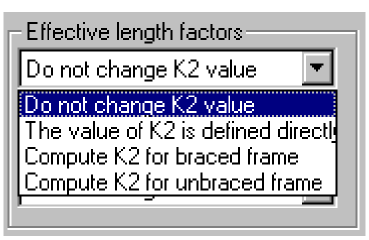
Note:

- the program **automatically** calculates the magnified (additional) moments according to the Code unless a **User defined** factor is specified for a column.

Refer to Design assumptions for columns.

Effective length factors

To specify the effective length ratio **K = Le/L** for both axes. Select one of the following options:



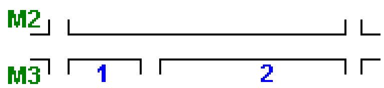
- **Compute**:

  The program calculates the effective length ratio based on the relative stiffnesses of the beam and columns framing into both ends of the column. For example, refer to ACI318-08, Section 10.10 (Figure R10.10.1.1) or to IS456 - Annex E.

  Note

  - different values are calculated for braced (nonsway) and unbraced (sway) frames.
  - the program ignores slabs and slabs combined with offset beams when calculating the "beam" stiffnesses.

- **Defined directly**:
  If several members are combined in a design direction, the relevant K value used for all members is the one defined for the first member. For example:



  Members 1 and 2 are computed separately, but are combined in the M2 direction; the K2 value used when computing member 2 is the value specified for member 1.

Note:
- if K2,K3 are not defined here, the program always uses a default value = 1.00.
- K2 is associated with M2; K3 is associated with M3

Live load reduction factor

Most design codes allow the axial load in a column or wall to be reduced if it supports a large area. Enter a factor and select the columns/walls.

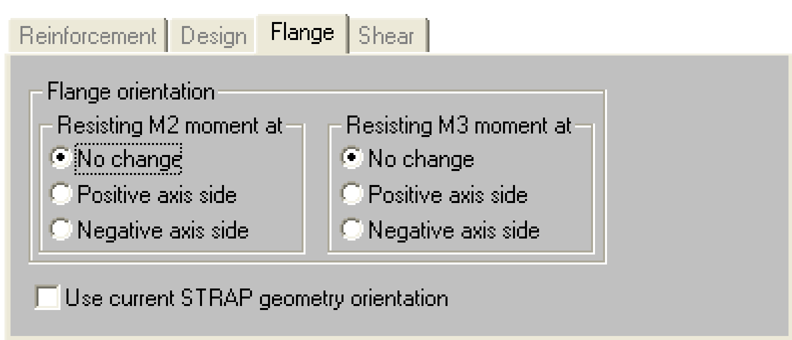
Note:
- the program multiplies the live load by the factor, e.g. enter 0.8 to reduce the live load by 20%.
- the default factor for all columns and walls is always 1.00 and cannot be revised
- the factor is applied only to the axial load in load cases specified as **Live**, i.e. the factor is ignored if there are no **Live** load cases.

Design for capacity/load

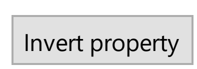
Specify the design (Capacity/Load) ratio.
- The default value is generally 0.98 - 1.00 (the program uses 0.99). Higher values give more conservative results.
- A new default value may be specified and different values may be assigned to individual columns in the Parameters - Design option.

Refer also to Compute 874.

8.12.2.3 Flange



Flange orientation



To reverse the flange location for T,L and U-sections. This option is equivalent to the options in "**Display/revise column**".

Note that this option does not switch the major and minor axes, i.e. 0 may be revised to 2 , but not to 1.

To switch the axes, use the 

option in the main menu.

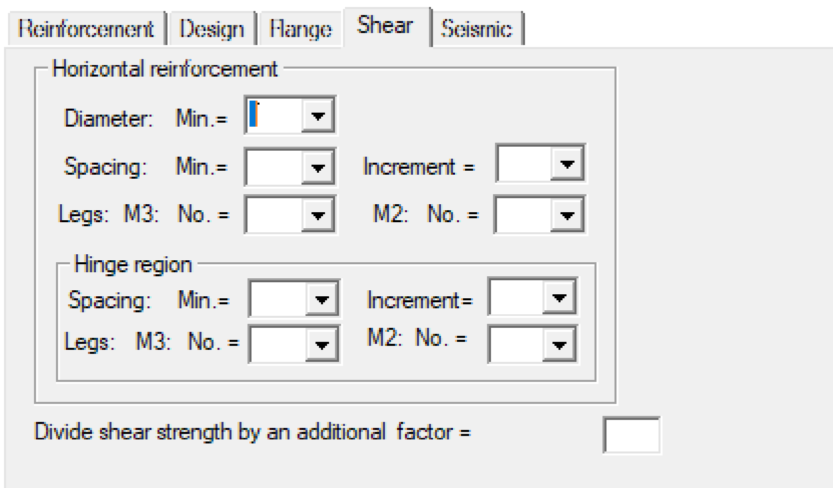
Use current STRAP geometry orientation

If a non-symmetric section is rotated in *STRAP* geometry after a column has been created, the program does not automatically rotate the column section in this module.

Select this option to rotate sections to the current orientation in geometry.

8.12.2.4 Shear

Define column shear parameters for selected columns. Values specified here overrides the model default parameters specified in the Default 812 option.

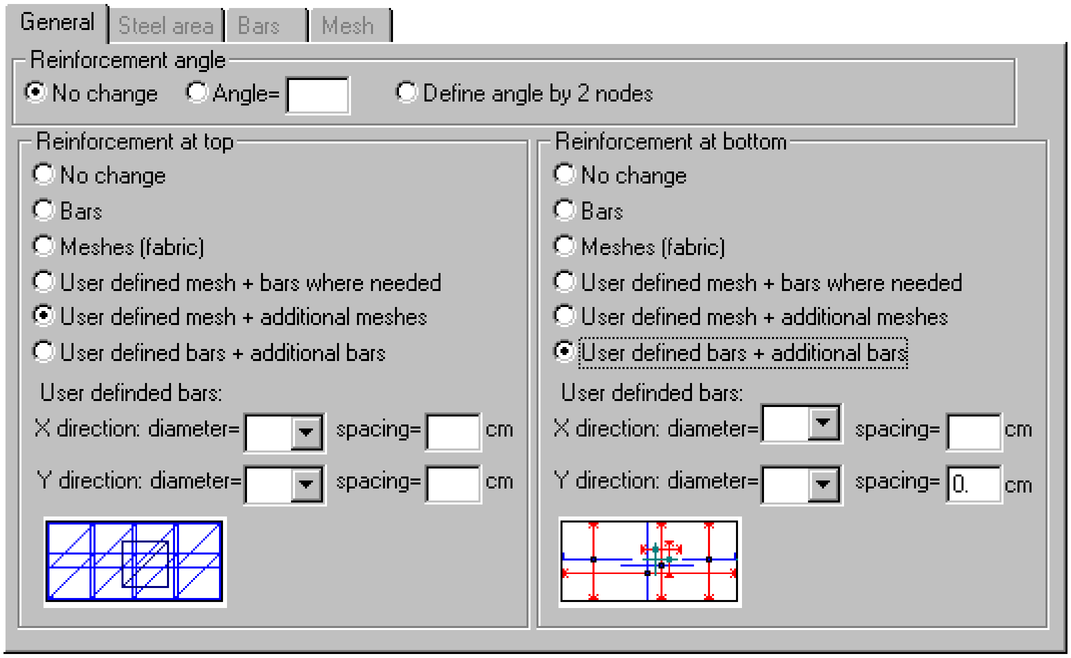


8.12.3 Slab parameters

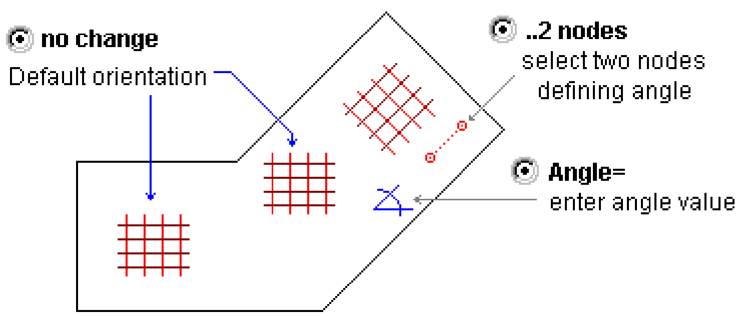
Slab parameters - general 869
Slab parameters - steel area 870
Slab parameters - bars 871
Slab parameters - mesh 871

8.12.3.1 General

Define the general parameters for selected slabs spaces. Values specified here overrides the slab default parameters specified in the Default 822 option.



Angle

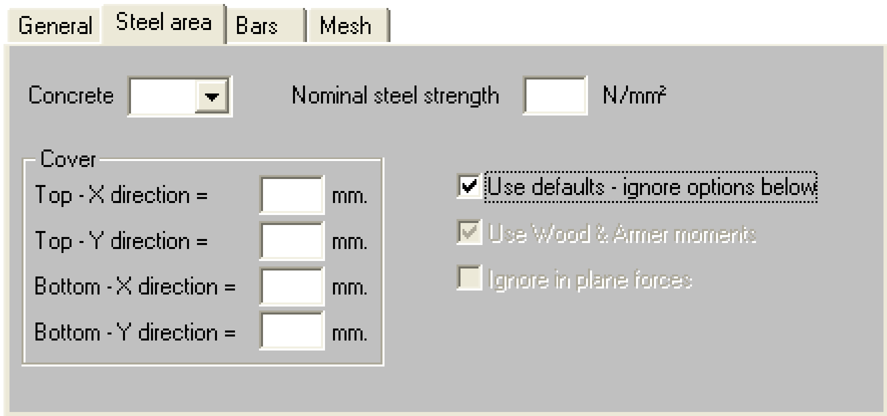


Reinforcement

Refer to Slab default parameters - General 823.

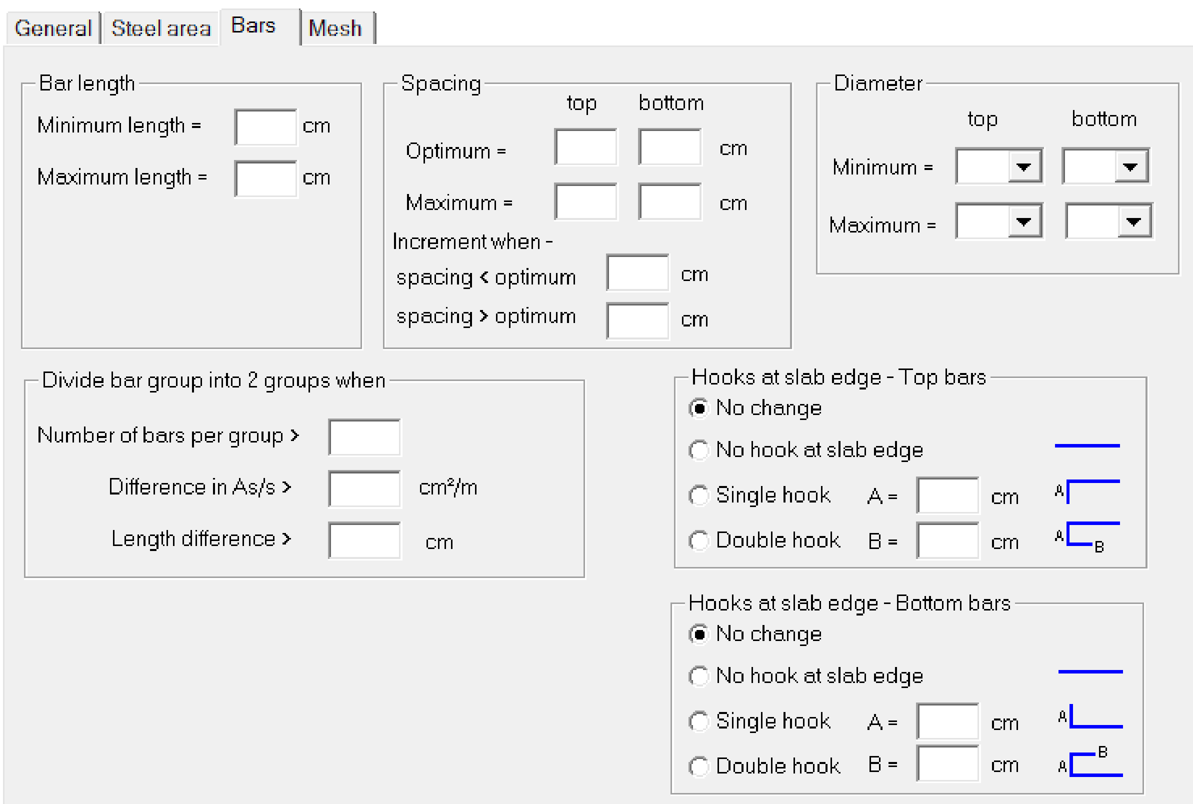
8.12.3.2 Steel area

Define the parameters for selected slabs for calculating the reinforcement area. Values specified here overrides the slab default parameters specified in the Default 824 option.



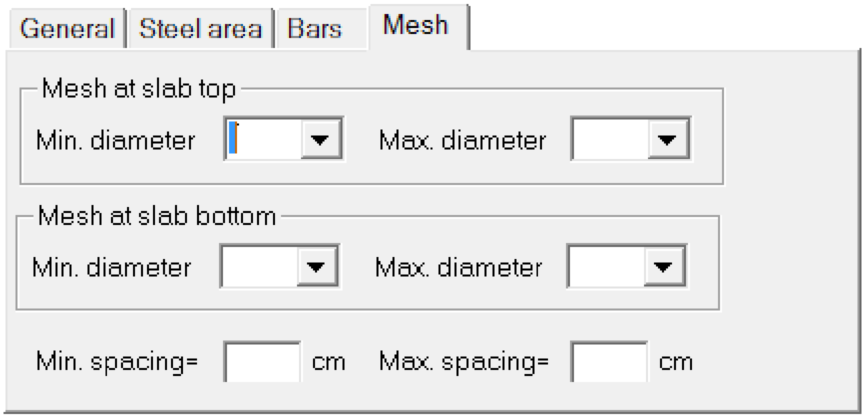
8.12.3.3 Bars

Define the parameters in selected subspaces for selecting the reinforcement bars. Values specified here overrides the slab default parameters specified in the Default 826 option.



8.12.3.4 Mesh

Define the parameters for selected slabs for selecting meshes. Values specified here overrides the slab default parameters specified in the Default 828 option.



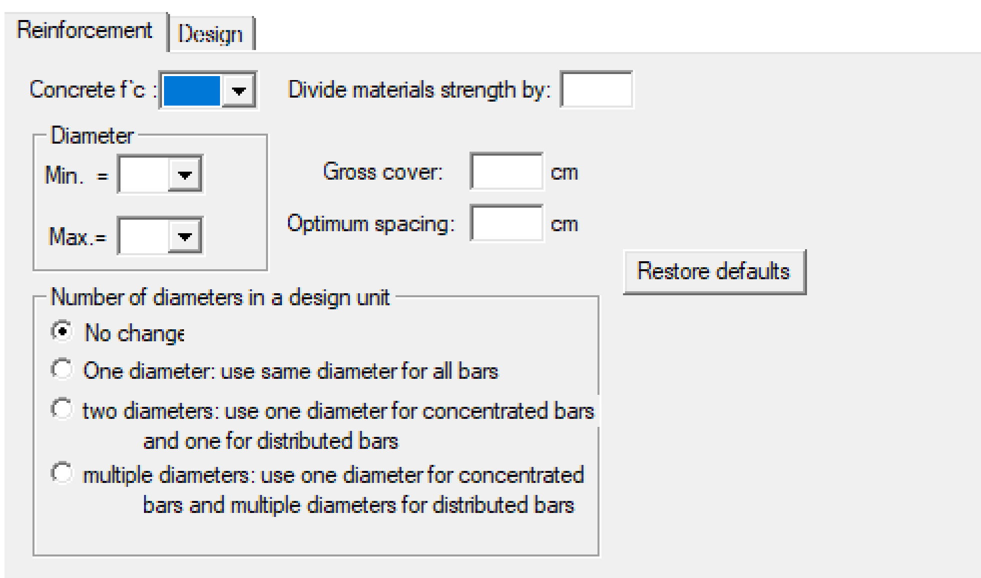
8.12.4 Wall parameters

Define different parameters for selected walls. Values specified here overrides the wall default parameters specified in the Default 816 option.

Wall parameters - reinforcement 872
Wall parameters - design 872

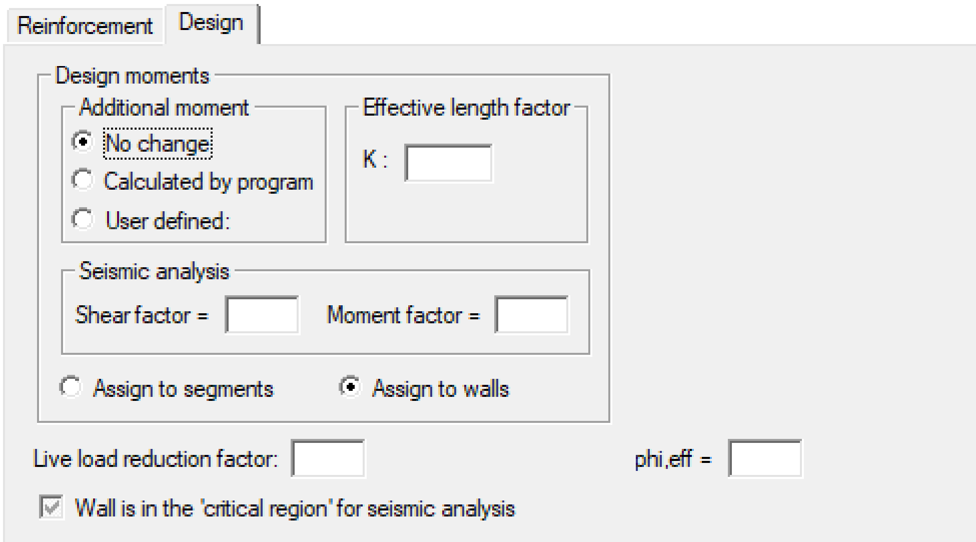
8.12.4.1 Reinforcement

Define the general parameters for selected walls. Values specified here override the wall default parameters specified in the Default 818 option.



8.12.4.2 Design

Define the general parameters for selected walls. Values specified here override the wall default parameters specified in the Default 818 option. Note the the 'Design moment parameters can be assigned to an entire wall or to individual segments.



Effective length factor

Specify the effective length factor for the wall in the weak direction.

Note:

- the k-factor may be assigned to a wall or to individual segments in the wall by selecting either
  - **Assign to walls**
    Select walls; the factor is assigned to all segments in the walls selected
  - **Assign to segments**
    Select segments; different factors may be assigned to different segments in the same wall.

Critical region

The program assumes by default that all walls are *not* in the critical/hinge region;

Select:

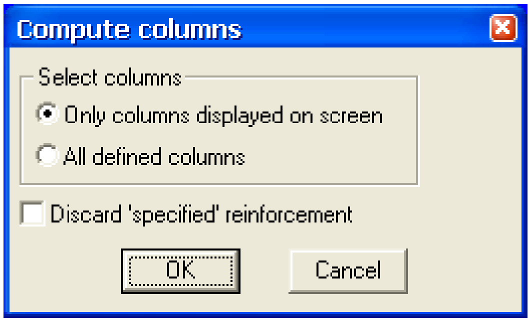
The selected wall segments are located in the plastic hinge/critical region.

No change in option for selected wall segments

The selected wall segments are not located in the plastic hinge/critical region.

8.13 Compute

Design beams, columns and walls according to the current defaults and parameters. Note:
- if no beams beams/columns have been defined, the program will automatically create them.

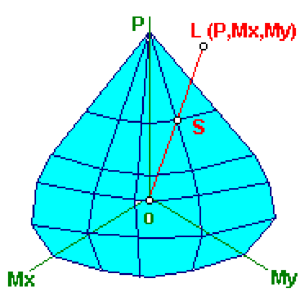


- **Select beams/columns/walls**
  To compute specific beams/columns/walls only: use the "**Zoom**" or "**Remove**" options to isolate the required beams/columns, or select an appropriate saved "**View**" and specify **Only for ..... displayed on screen**.
- **Discard 'specified' reinforcement**
  The user can specify reinforcement for selected column/wall members using the Results - Specify reinforcement 953 option.
  ☑ recalculate the columns/walls discarding any specified reinforcement, i.e. the specified reinforcement is deleted
  ☐ do not recalculate, i.e. maintain the specified reinforcement. ***Note that such columns that are part of an identical list will not be designed as identical.***

The program designs the selected beams/columns/walls in series and automatically displays the Result Summary table 926 on the screen.

For a detailed explanation of the theory and methods used by the program with reference to the design Codes, refer to Design assumptions.

The following is a general explanation of the algorithm for column/wall design:
- The program calculates the section capacity for a specified reinforcement pattern and compares it to the external forces.



Referring to the figure, the program calculates the location of point "S" on the interaction diagram and checks the ratio OS/OL. If the ratio OS/OL > 1, the reinforcement is adequate for the external forces.

As the program can only check the capacity of sections, the design procedure is iterative:
- the program arranges minimum reinforcement in the section and calculates the capacity.
- if the section capacity is inadequate, the program increases the reinforcement and calculates the new capacity.
- the process continues until the capacity is greater than the external forces.

- The program stops the iteration when the calculation is greater than 99% convergent. Similarly, a ratio of capacity/load > 0.99 is considered acceptable. These two points explain minor discrepancies in the results.
- The iteration also stops if the reinforcement percentage exceeds the allowable value in the Code or if no more bars can be placed because of spacing limitations. Warnings are displayed in such cases.

- The algorithm includes a search for the face of the column where the addition of more reinforcement is most beneficial. Bars with the current diameter are added until the optimal spacing is reached; the program then increases the bar diameter by one size.
- The iterations are internal to the program and not visible to the user.

If two different bar sizes are allowed -
- the corner bars only have the larger bar size
- there is one size difference between the corner bars and the face bars.

Application Point of Load:
The program applies the load at the centre-of-gravity of the concrete section. Note that for flanged sections, this point may not be identical to the center of resistance of the section, the centre-of-gravity calculated from the concrete ***and*** reinforcement. In such cases, an additional internal moment is generated.

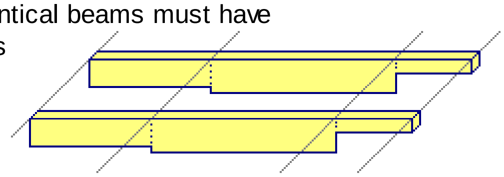
8.14 Identical

Specify, prior to the start of the calculation, that the reinforcement in several beams/columns/walls must be identical. The program calculates the reinforcement required for each one and assign the heaviest reinforcement to all beams/columns/walls in the "identical list".

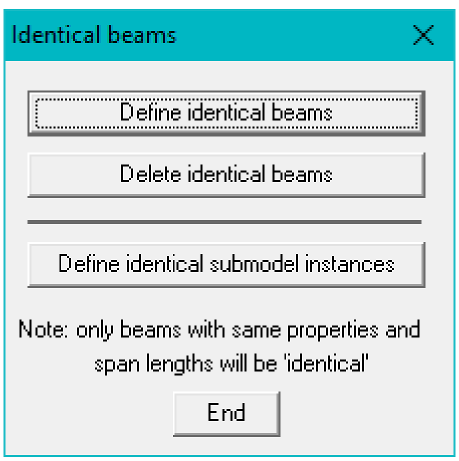
Select one of the following:
Identical beams 876
Identical columns 877
Identical walls 880

8.14.1 Identical beams

Define identical beams; identical beams must have
- identical number of spans
- identical spans length
- identical sections



The program combines the moment/shear envelopes from all of the selected beam to create one design envelope for all of the beams and so all beams will have the same reinforcement.



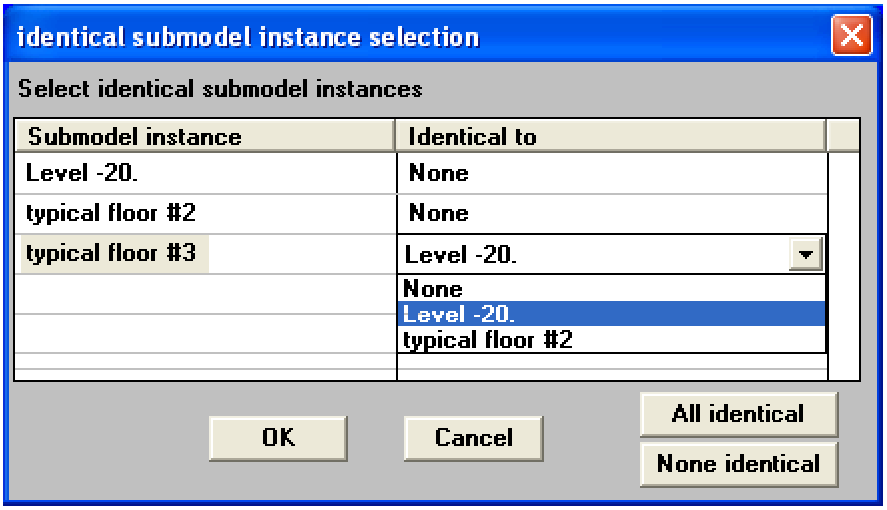
Define/delete identical beams

For both options, select existing beams using the standard beam selection option.

Note:
- if identical beams are defined in one instance of a submodel and identical instances were defined for this instance, then the envelopes for all of the beams in ***all of the instances*** are combined.
- for the *STRAP* geometry "Include element strip in beams" option, the program checks only whether the ***beams*** (without the added slab) are identical. For each span along the beam - the program calculates the slab widths for that span in each of the beams in the identical group and uses the ***minimum*** width for calculating the reinforcement in that span for ***all*** of the beams..

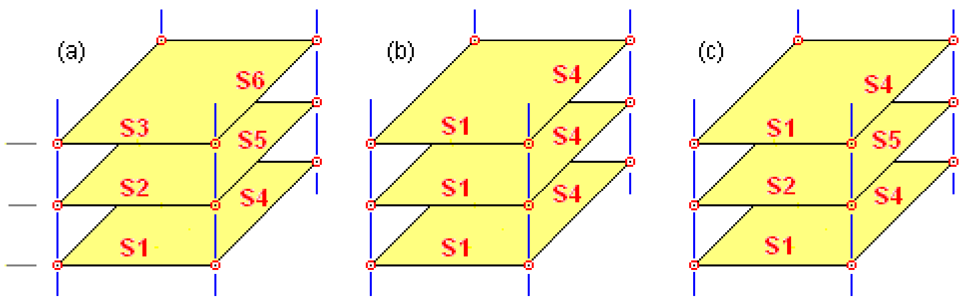
Identical instances

Specify that the same reinforcement must be selected for the same member in two or more instances:



Select two or more instances that must be identical;

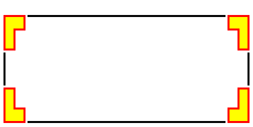
- click on a line
- click on the ▼ arrow in the **Identical to** column
- select another instance that the highlighted instance is identical to. For the example above refer to Figure (c) below.



- click None identical if ***different*** reinforcement may be selected for corresponding members in the instances of a submodel; refer to Figure (a) above. This is the default option.
- click All identical if identical reinforcement ***must*** be selected for corresponding members in ***all*** the instances of a submodel; refer to Figure (b) above.

8.14.2 Identical columns

Specify, prior to the start of the calculation, that the reinforcement in several columns must be identical. The program calculates the reinforcement required for each column and assign the heaviest reinforcement to all columns in the "identical list".

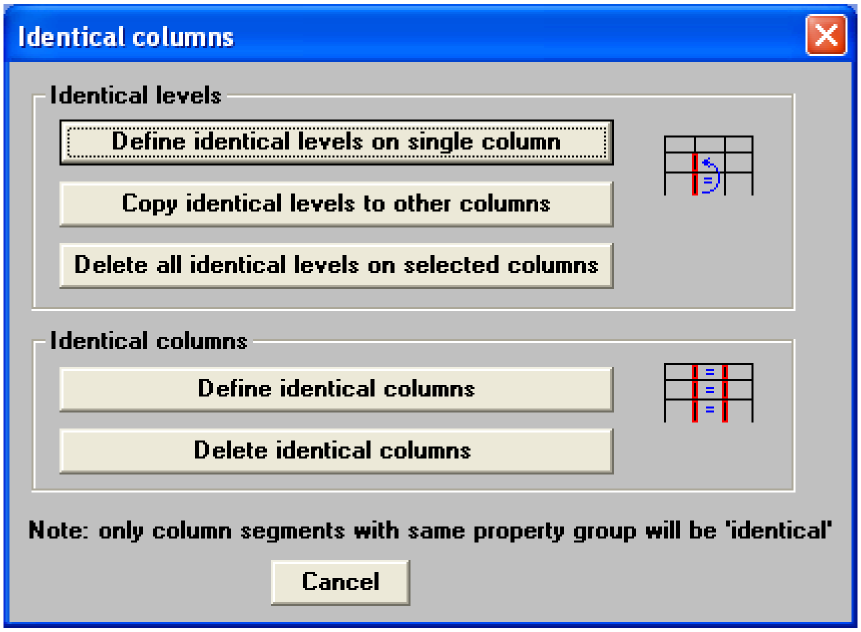


Note that only columns with ***identical section dimensions*** may be defined as "identical". Orientation of the section is not important - all of the columns in the this example may be defined as identical.

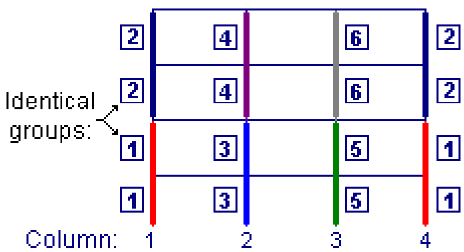
There are two main options:

- **Identical levels** : specify that different levels on a single column are identical
- **Identical columns** : specify that the same levels on two or more columns are identical

The program combines the results of both options. Refer to the example below.

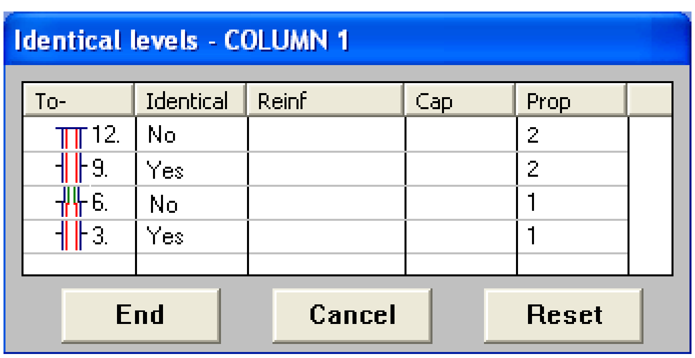


Example: create the following identical groups:



- Create two different identical groups in column 1:

  Select Define identical levels on single column
  select column 1 and arrange the menu as shown:



- Create the same two groups in column 4:

  Select Define identical columns

  Select columns 1 and 4 using the standard beam selection option.

- Copy the same division into groups along the height to column 2 and 3, but not the same groups.

  Select Copy identical levels to other columns

Select column 1 as the base column, then select columns 2 and 3.

8.14.2.1 Identical levels

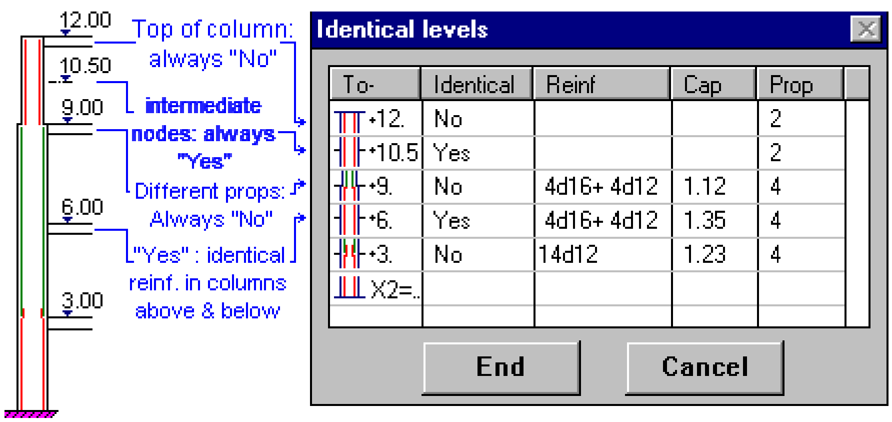
Define

Specify that the reinforcement in adjacent levels must be identical. This is convenient when the same bars are used over several storeys.

The following screen shows an example for a column in a four storey building; the "Identical" column in each row describes the situation at the floor level between two adjacent columns, where:

- **Yes** - the reinforcement in the columns above and below will be identical
- **No** - the reinforcement may be different

For columns that have already been designed, the program displays the current reinforcement and capacity factor.



To change the status, click on the relevant row; the program toggles the **Yes**/**No** value.

Note that in the above example, the status can be revised at the bottom two levels only (3.00 and 6.00):

- **Yes** cannot be specified at levels where the property changes (+9.00)
- **No** cannot be specified at a level that is a *STRAP* intermediate node (+10.5)
- The top level is shown for information only.

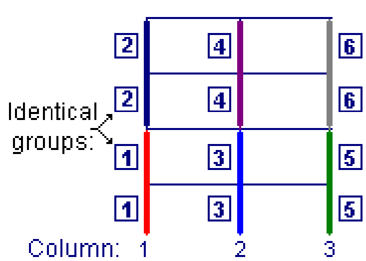
Copy

Copy the vertical division into identical groups from one column to another.

- select a column with existing identical groups
- select one or more columns that the vertical division will be copied to.

Note:

- identical groups are created in the other columns; only the vertical division into groups will be the same. In the following example, copying the groups in column 1 to columns 2 and 3 creates groups 3 to 6



- columns 1, 2 and 3 may have different cross-section dimensions as different groups are created. However, the first two segments in column 2 must have the same dimensions if group 3 is to be created.
- the program copies the division into groups even if there are additional nodes in one of the columns, the nodes are offset vertically in one of the columns, etc.

Delete

Delete the vertical division into identical groups on selected columns.
- select a column with existing identical groups

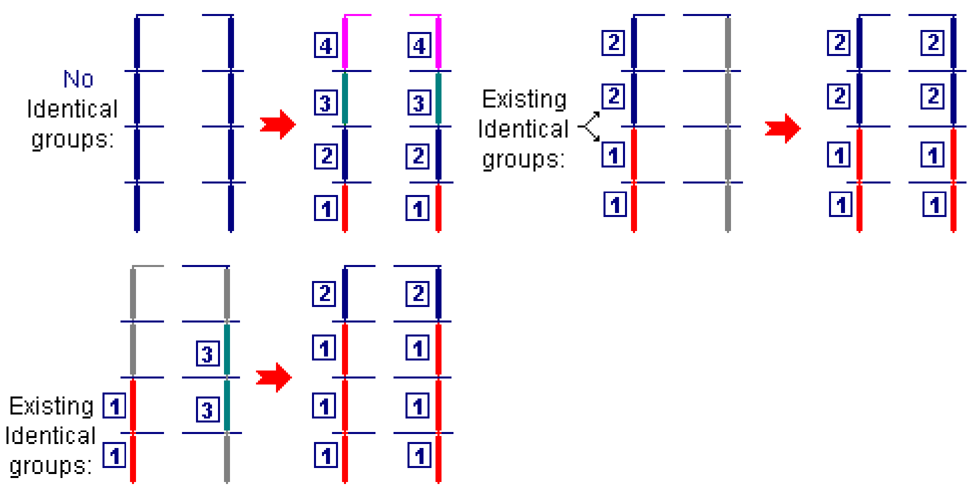
8.14.2.2 Identical columns

Define

Specify that two or more columns are "identical". The program combines the vertical group division of the columns.

Note that only columns with identical section dimensions may be "identical" (orientation is not important).

Both columns are selected in the following examples:



Note:
- the program creates the identical groups even if there are additional nodes in one of the columns, the nodes are offset vertically in one of the columns, etc.

Delete

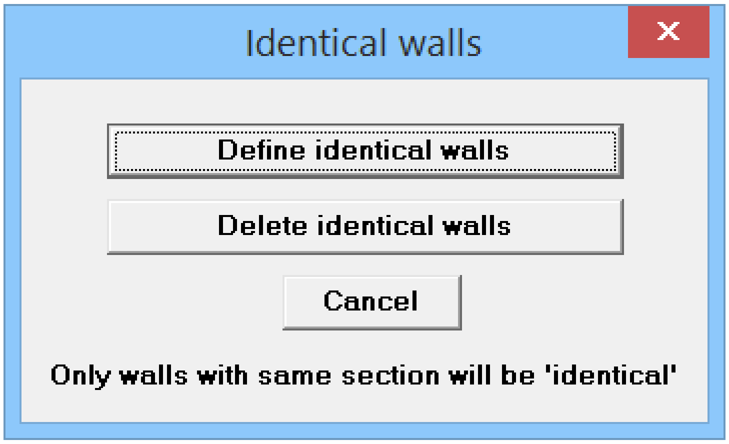
Remove a column from a group of identical columns.

Note that if a vertical division into groups was previously defined for this column using the Define identical levels on single column option, the vertical division will not be deleted.

8.14.3 Identical walls

Define identical walls; identical walls -
- must have identical wall sections
- must be stacked vertically

The program calculates the reinforcement required for each wall segment and assign the heaviest reinforcement to the corresponding segments in the "identical wall list".



For both options, select existing walls using the standard wall selection option.

8.15 Specify bars

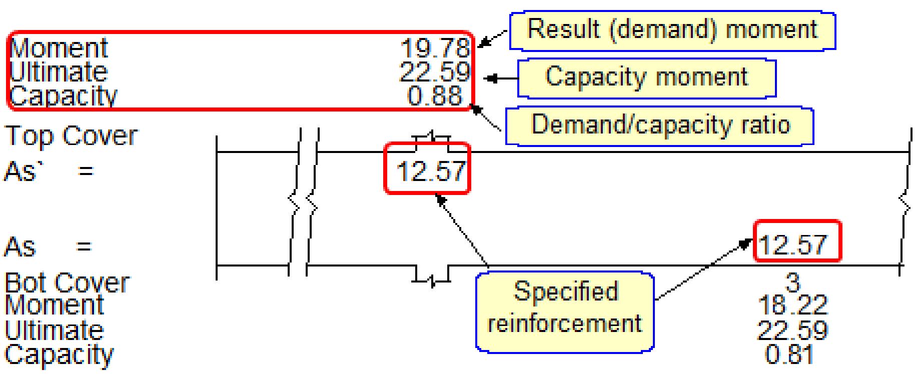
Specify the actual reinforcement - number of bars, diameter, spacing - for :
- beams 882
- columns 883
- walls 884

This option is used to check existing structures.

Parameters required by Codes for checking existing structures can be defined, e.g.
- material strength reduction factors
- shear capacity factors.

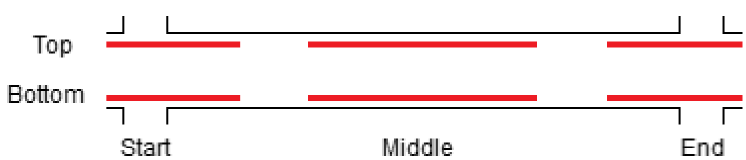
The specified reinforcement is used to calculate moment and shear capacities and the demand/capacity ratio. This reinforcement is displayed in -
- result summary tables
- detailed result tables
- drawings that include beam/column/wall reinforcement.

For example: beam detailed results:



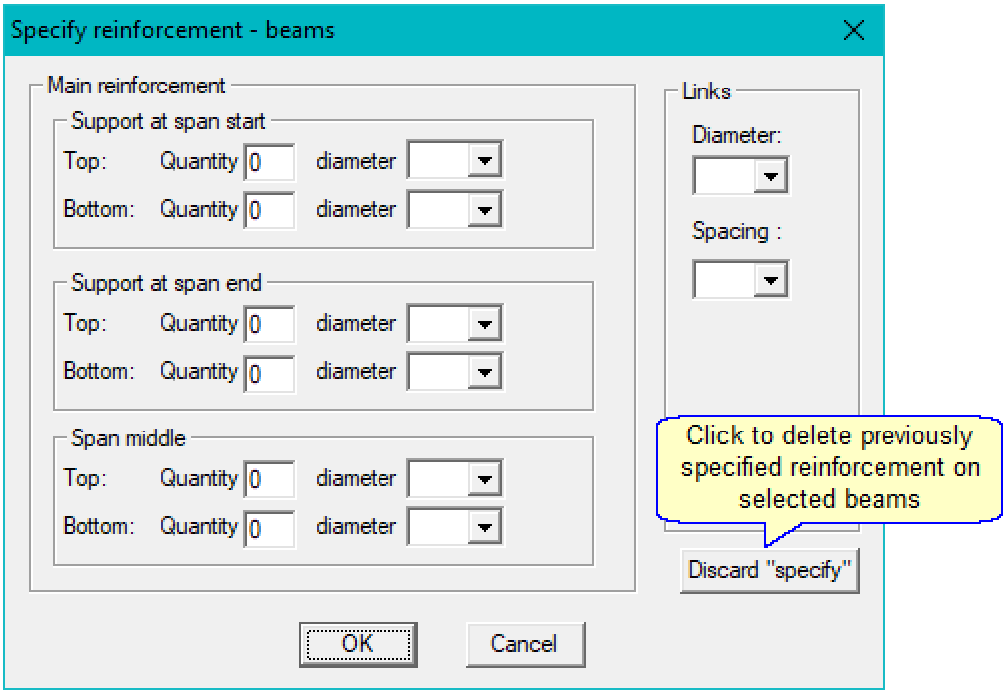
8.15.1 Beams

Specify the actual reinforcement in selected beams.
- the longitudinal reinforcement can be specified at six locations in every span:



Enter the diameter and number of bars.

- the shear reinforcement specified here is applied along the entire length of the span.
  Enter the diameter and spacing



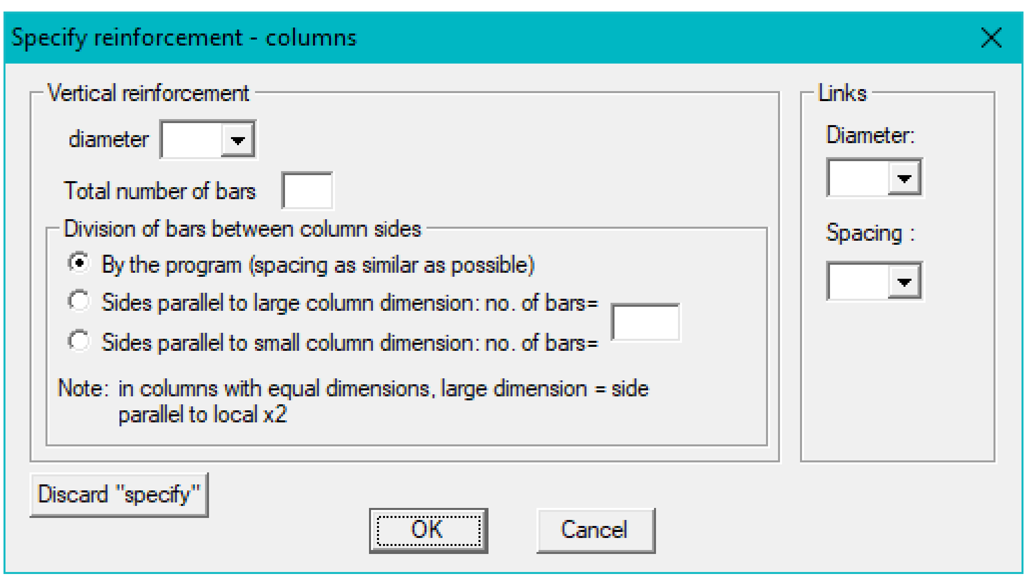
Note:

- the longitudinal reinforcement must be specified at all six locations. The program assumes no reinforcement is present at an unspecified location.
- shear reinforcement: the program assumes 2 legs.

Select beams using the standard Beam selection option

8.15.2 Columns

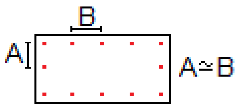
Specify the actual reinforcement in selected columns:



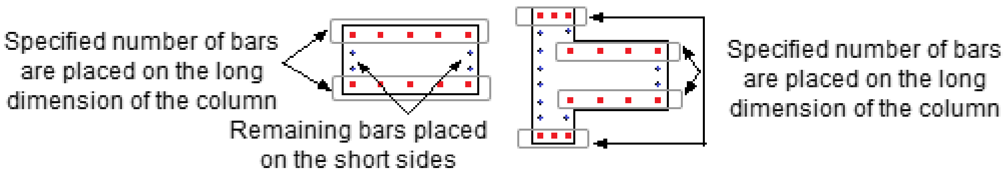
Specify the diameter and ***total*** number of vertical bars in the column.

There are three options to instruct the column how to distribute the bars to the column faces:

By the program
The program distributes the bars to all faces so the the spacing between bars is as close to identical as possible.



Sides parallel to the large column dimension =



Sides parallel to the small column dimension =
Similar to the previous option.

Note:
- An even number of bars should be specified. The program will round down an odd number of bars.
- For "solid section", the program always uses the first option.

Select columns using the standard Beam selection option

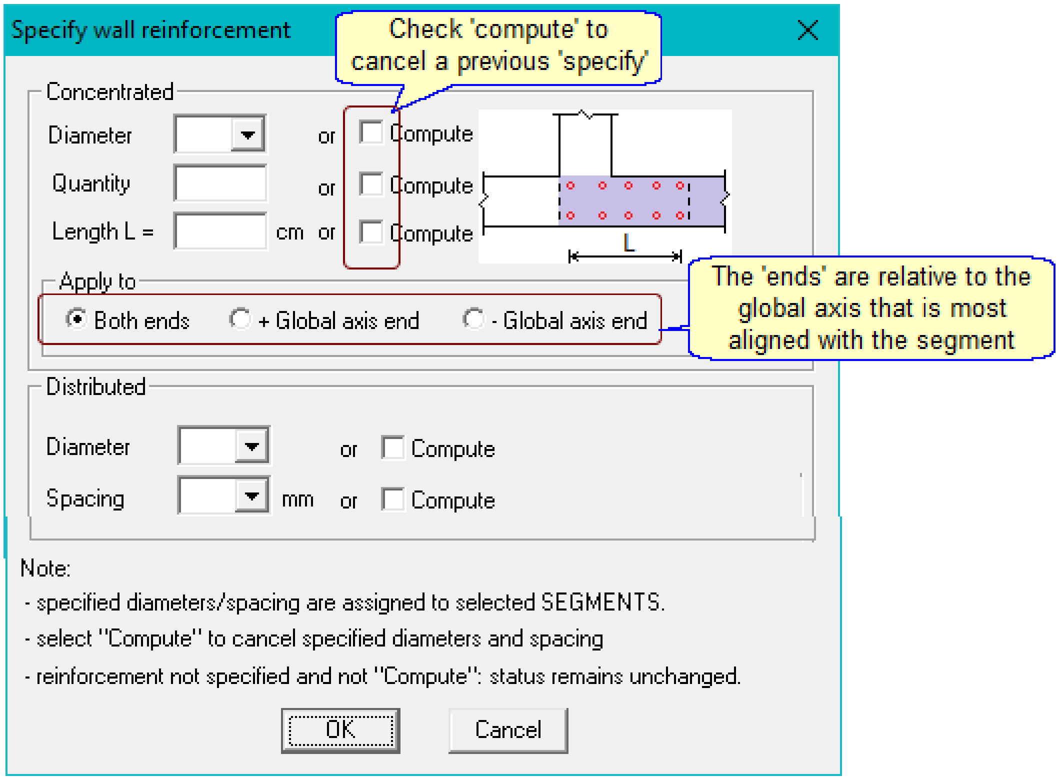
8.15.3 Walls

Revise the diameter or number of bars for the corner and/or distributed reinforcement in any segment; the program automatically recalculates the wall capacity for the new arrangement.

To specify different reinforcement:

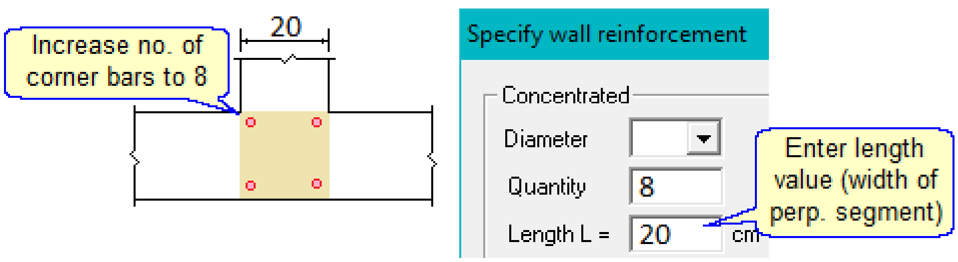
- select Specify bars in the side menu
- specify new values for no. of bars, diameter and/or length for corner and/or distributed bars
- select **Both ends** **+Global axis** end or **-Global axis** end for selected segments
- select one or more ***segments***
- click Compute in the side menu
- select **Result summary** or **Detailed results** to view the new arrangement and capacity factor.

Note:
- the diameters selected do not have to conform to the current min/max diameter parameters.
- the program does ***not*** check spacing for the number of bars selected; it is the user's responsibility to ensure that the spacing conforms to Code requirements.



Note:

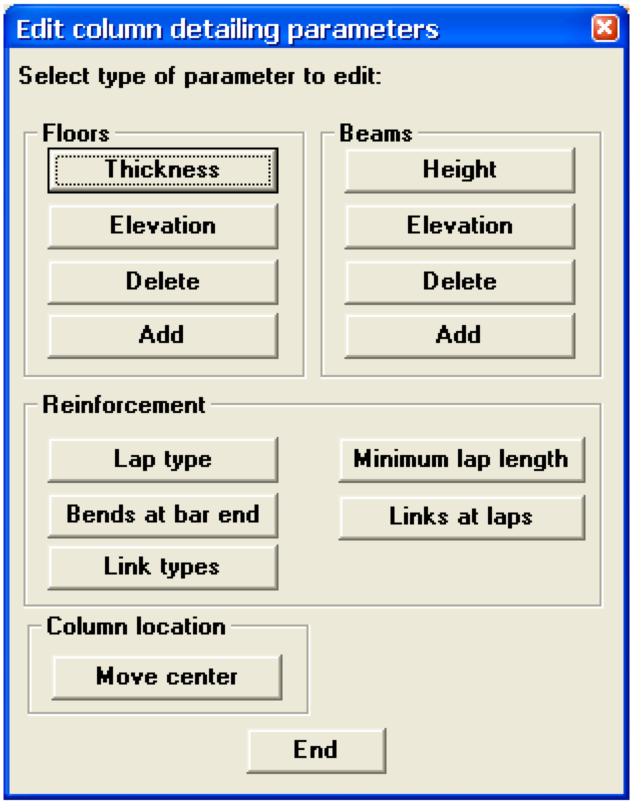
- if Lx,Ly lengths are not listed in the detailed results then a value for **Length L =** must be entered. For example:



- to check the "specify" data, select Data tables - wall-specify 983.

8.16 Column detailing parameters

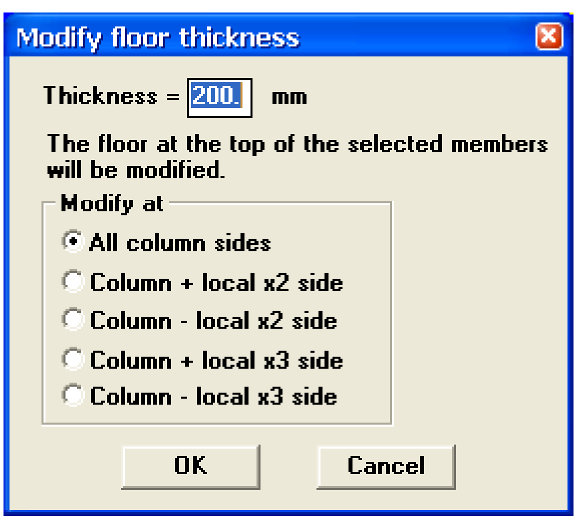
Modify the column drawing parameters at any specified column location. Note that the parameter may be modified at any of the four column sides at each column end.

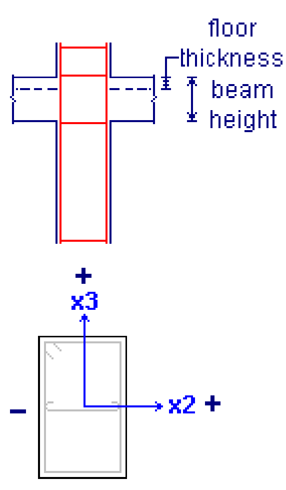


Floor thickness / beam height

The beam and slab dimensions may be revised at each of the four sides at the column top.

- Specify the new floor or beam height and the location relative to the column orientation:





- Select one or more columns using the standard beam selection option.

Add/delete

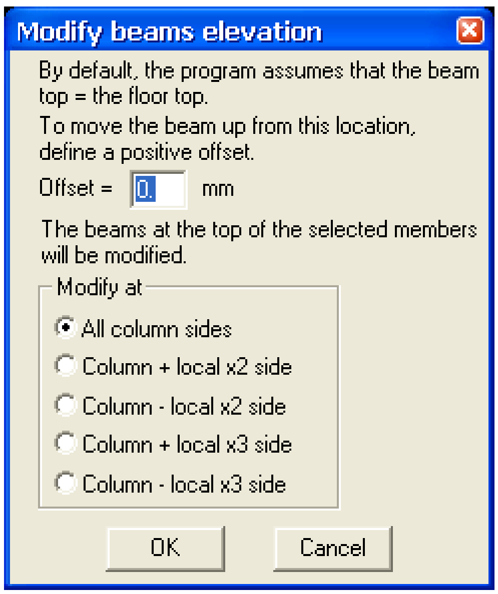
The beams and slabs may be deleted/added at each of the four sides at the column top.

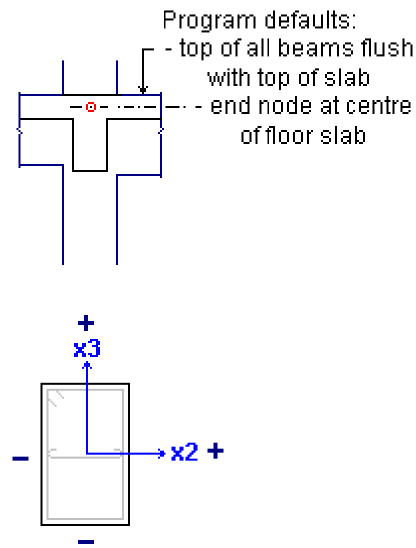
- Specify the location relative to the column orientation 910
- Select one or more columns using the standard beam selection option.

Floor / beam - elevation

The beam and slab elevations may be revised at each of the four sides at the column top.

- Specify the floor or beam offset from its current location and the location relative to the column orientation:





- Select one or more columns using the standard beam selection option.

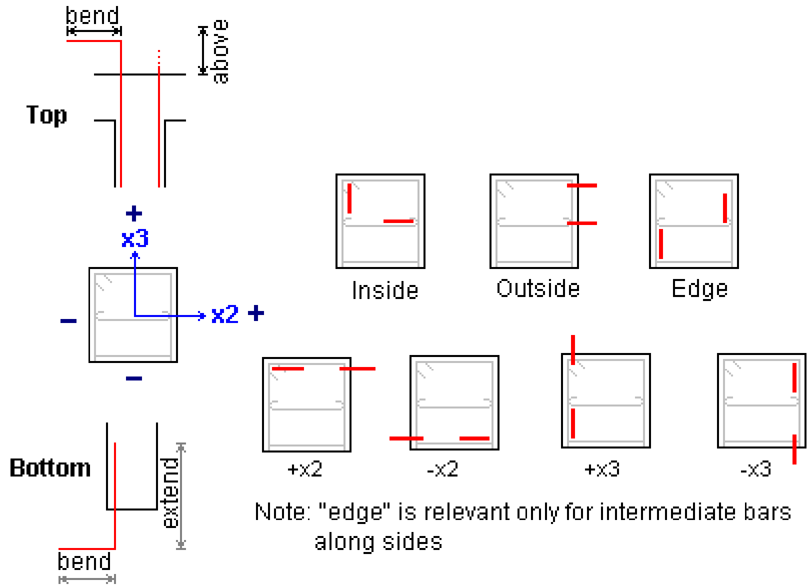
Lap type

Specify a different lap type for selected columns (the selection overrides the default lap type).

- Select:
  Crank, **Alongside** or **Inverted crank** to maintain the lap at the selected locations, or -
  Unify with bar above to delete the lap at the top of the selected columns
- Select columns using the standard beam selection option

Bends at bar end

Specify a different detail for all reinforcement at the ends of selected columns (the selection overrides the default option).

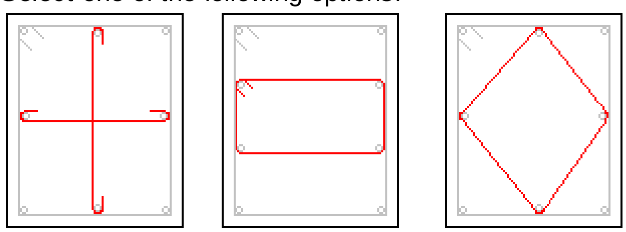


- Select columns using the standard beam selection option

Note:
- this option also applies at intermediate locations where a step in the column face does not allow the bars to be extended to the column above.

Link type

Specify a different type of additional links (used to tie bars placed between corner bars) for selected columns (the selection overrides the default type).
- Select one of the following options:



- Select columns using the standard beam selection option

Note:
- Rectangle and diamond links are used only when there are sufficient intermediate bars and internal link angles comply with the Code requirements. Otherwise one-legged ties are used.

Minimum lap length

Specify a different minimum lap length for selected columns (the selection overrides the default length).
- Enter the minimum lap length
- Select columns using the standard beam selection option

Note:
- the program calculates the lap length according to the Code requirements. The lap length used is the maximum of the required lap length and the minimum lap length defined here.

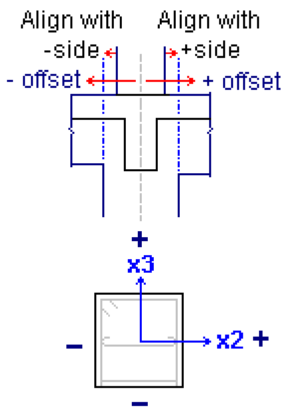
Links at lap

Specify a different maximum spacing for links within the lap length for selected columns (the selection overrides the default spacing).

- Define the maximum spacing; the spacing that will be detailed in the lap length is the minimum of that specified here and the spacing calculated according to the Code requirements.
- Select columns using the standard beam selection option

Move center

Any column may be offset relative to its centre-line or aligned with one of the faces of the column below.

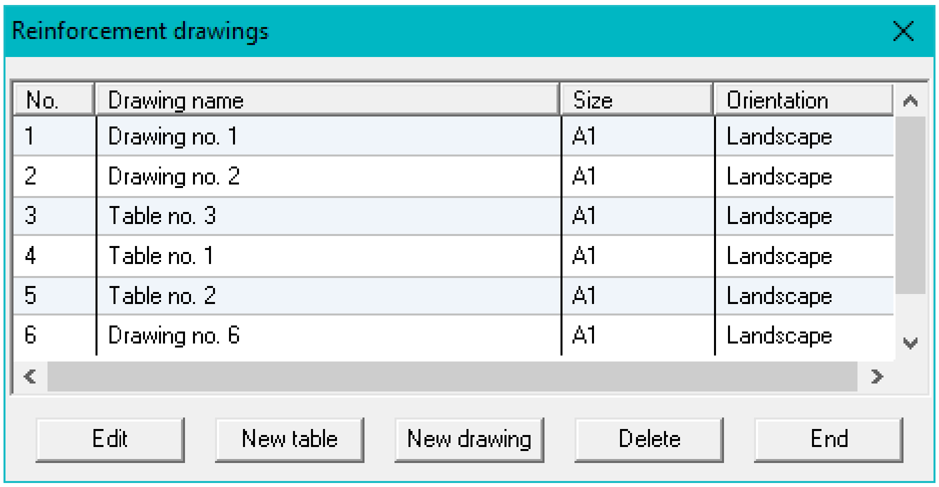


- Specify the column offset from its current location or the alignment face and the location relative to the column orientation 910
- Select one or more columns using the standard beam selection option.

8.17 Drawings

Use this option to:

- Create drawings with beam detailing, column elevations & sections, slab reinforcement, wall sections, foundation plans, etc
- Edit existing drawings
- Delete existing drawings



New drawing

To create a new drawing:

- click New drawing ; a new line is added to the table
- Specify the size and orientation of the drawing
- click Edit

New table

This option is available only for columns

Delete

- click and highlight a drawing in the list
- click Delete

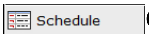
The drawing (existing or new) is then displayed on the screen. The following options are available in the side menu:

Add — Add new objects: beam detailing, column elevations & sections, slab reinforcement, wall sections foundation plans, etc.

Move — Move existing objects on the drawing.

Delete — Delete existing objects from the drawing

Edit — Revise the object, modify text, orientation, location, etc.



Create or update a bar schedule 917 file for ***all*** drawings in the model

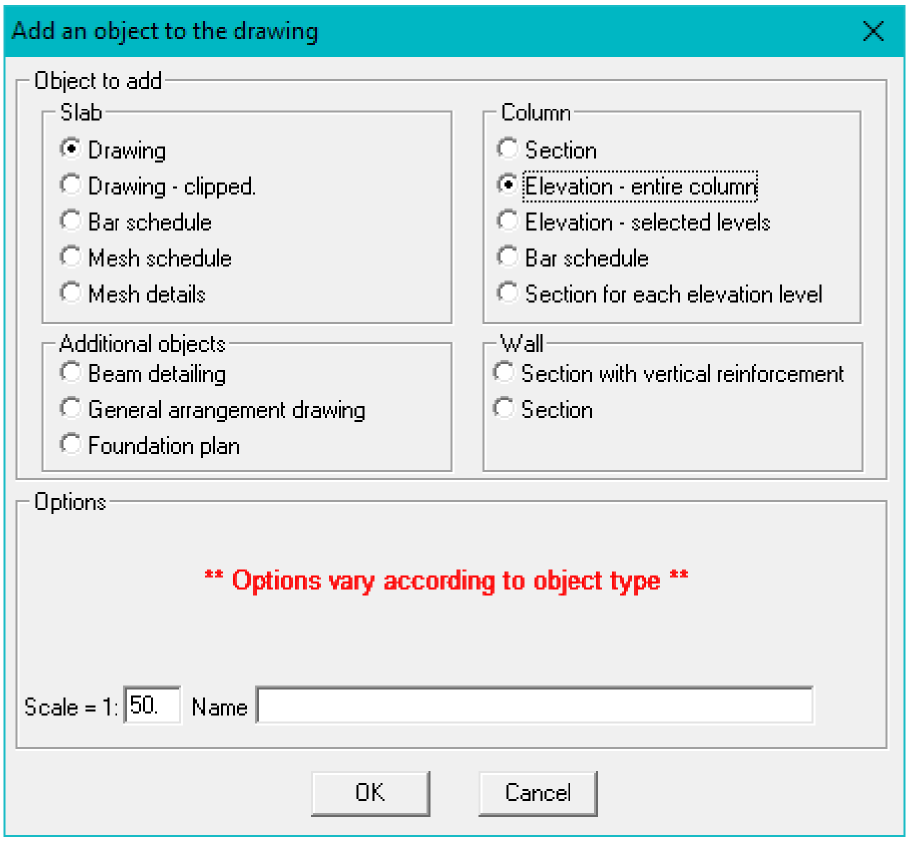
Print the drawing

Select and edit another column drawing/table or create new ones.

Return to the main column design menu.

Add

Add an object to the drawing:



For all object types:
- type in the **Name** (optional) and specify the **Scale**

Move

- Move the adjacent to the elevation/section/schedule so that it is highlighted with the ■ and click the mouse.
- Move the frame around the section to its new location and click the mouse.

Edit

Select one of the following for more information:
- edit a slab drawing 894
- edit a column drawing 912
- edit a column table 916
- edit beam detailing 920 drawing

- edit a general arrangement 921 drawing
- edit a wall section 922 drawing

Delete

Move the cursor adjacent to the elevation/section/schedule so that it is highlighted with the ■ and click the mouse; click Delete.

8.17.1 Slabs

Select one of the following topics:

- Add 892 a new slab drawing
- Edit 894 an existing slab drawing

Note:

- Refer to Draw slabs - general 794

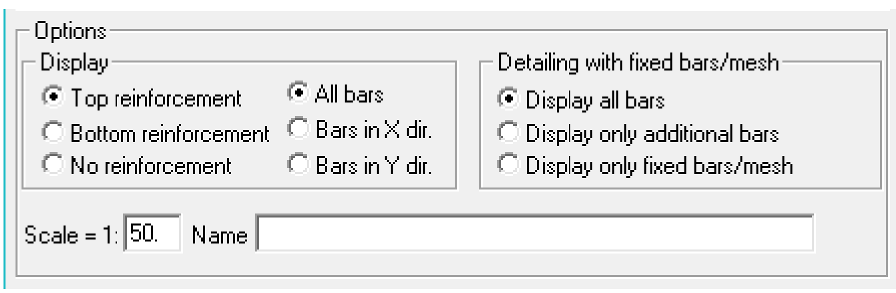
8.17.1.1 Add

Add a slab object to the drawing:

- the reinforcement in an entire or partial slab subspace; top or bottom reinforcement are displayed separately
- for 'bar' reinforcement: a bar schedule
- for 'mesh' reinforcement: a mesh schedule and/or mesh detail drawings

Add a slab drawing

The options are:



Options vary according to the slab detailing method selected in the Default - General 822 option:

- For all detailing methods (bars, meshes, fixed bars/meshes+additional), display:
  - **Top reinforcement** or **Bottom reinforcement**
  - **No reinforcement**:
    in this case the program draws the slab contour and beams and columns (if present); grid lines and dimension lines may be added to

  and -
  - **All bars**
    Display bars in both the X and Y directions
  - **Bars in X/Y dir.**
    Display bars in one direction only
- If one of the following detailing methods was used:
  - **User defined mesh + additional bars**
  - **User defined mesh + additional meshes**

User defined bars + additional bars

Also select:

All bars

display the user-defined reinforcement and the additional bars added by the program where required

Additional bars

Fixed bars/mesh

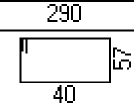
display only the constant user-defined reinforcement

To add a slab drawing:
- select the reinforcement to draw (see above)
- type in the **Name** (optional); the name is drawn either above or below the drawing according to the Slab - drawing parameter 831 specified
- specify the **Scale**.
- click OK
- Move the to any element in the slab so that it is highlighted with the ■; click the mouse
- For a clipped drawing, create a window defining the outline of the drawing (lower-left and upper-right corners).
- drag the rectangle that outlines the drawing to the correct location and click the mouse

Note:
- use the Edit option to:
  - revise the reinforcement selected by the program
  - add grid lines or dimension lines to the slab drawing
  - revise drawing parameters
  - clip the rectangle.

Add a bar schedule

Add the bar schedule for the bar reinforcement in the slab. For example:

Bar Mark	Type	Diam mm	Total no.	Length bar	Shape code	A mm.	B mm.	C mm.	D mm.	E mm.	Drawing
1	T	8	6	330	38	25.	290.	15.			
2	T	10	12	354	77	40.	57.				

The only option is **Text size.**

To add the bar schedule:
- specify the **Text size**.
- click OK
- Move the to any element in the slab so that it is highlighted with the ■; click the mouse
- drag the rectangle that outlines the schedule to the correct location and click the mouse

Add a mesh schedule

Add a schedule of the prefabricated meshes in the slab. For example:

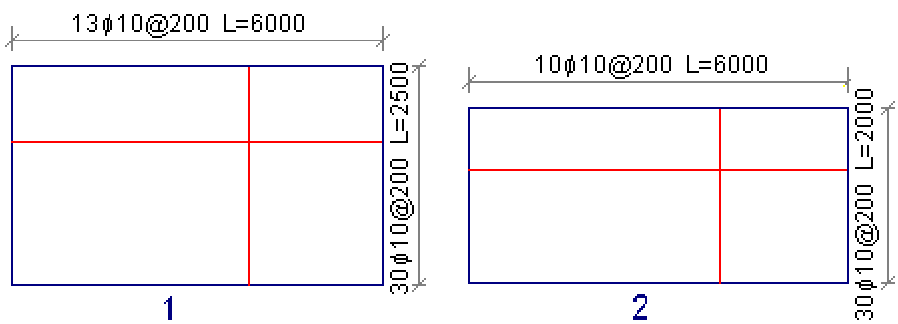
Bark Mark	Dist. A	Dist. B	Type A	Diam A	Dist a	Quant A	Dist a1	Type B	Diam B	Dist b	Quant B	Dist b1	Total
A9-3	650.	470.	TN	5.5	22.	22	4.	TN	7	22.	22	6.	5
A10-3	400.	530.	TN	6	22.	24	12.	TN	6	22.	24	6.	5

The only option is **Text size.**

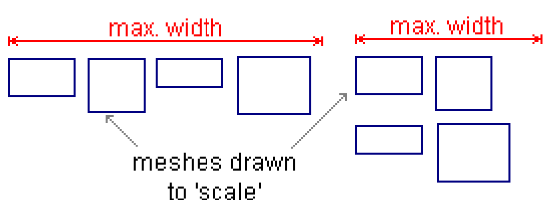
To add the bar schedule:
- specify the **Text size**.
- click OK
- Move the to any element in the slab so that it is highlighted with the ■; click the mouse
- drag the rectangle that outlines the schedule to the correct location and click the mouse

Add mesh details

Display a small drawing showing each of the meshes in the slab. For example:



The options are:
- **Scale**
- **Maximum width**; for example, the same meshes drawn with two different 'width values':



8.17.1.2 Edit

Parameters
- select the reinforcement to display (top/bottom), etc
- revise the scale, clip the drawing
- revise the drawing parameters for the space

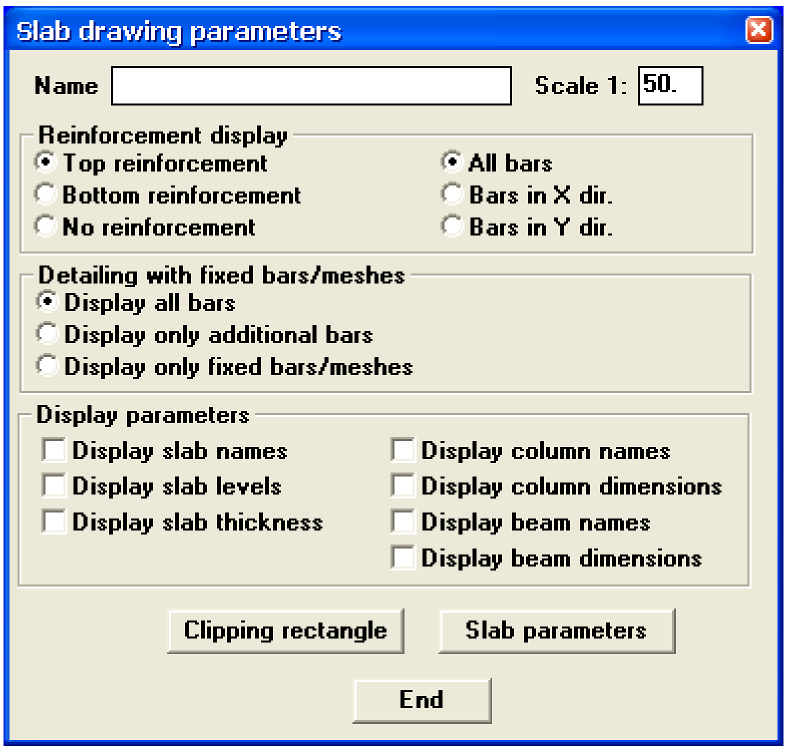
Dim. lines — define dimension lines and add them to the drawing.

Grid lines — add grid lines to the drawing

Bars — modify the reinforcement on the drawing:
- revise diameter/length/spacing of existing bars; add hooks
- add/delete/move bars

	modify the meshes on the drawing: • revise diameter/length/spacing of mesh bars • add/delete/move meshes
	Display graphically (color-coded) the ratio of the provided reinforcement area to the required area.
	Add text, beam sections and lines to the slab drawing.
	Return to the Edit menu 892

8.17.1.2.1 Parameters



Reinforcement display

Top reinforcement and bottom reinforcement are drawn separately. Select one of the following options:

Top reinforcement only

Bottom reinforcement only

No reinforcement

the program draws only the slab contour and beams and columns (if present); grid lines and dimension lines may be added to the drawing.

In addition, you may display:

All bars - bars in both directions

Bars in X direction only

Bars in Y direction only

Fixed bars/mesh

This option is relevant only when one of the following options was selected in the Default - General 822

option.
User defined mesh + additional bars
User defined mesh + additional meshes
User defined bars + additional bars
The user-defined reinforcement and the additional reinforcement may be displayed as separate objects on the drawing or may be drawn together.

Display parameters

Add any of the selected text to the current drawing:
Note:

- Elevations are added for each subspace.
- "Beam names" are the names defined in Beam - define names 839
- "Column names" are the names defined in Columns - define names 844
- "Slab names" are defined in Define - slabs 846.

All of these texts may be edited in the Edit - Additional 905 options menu

The default text size for these options are specified in Setup - Text 961.

Clipping rectangle

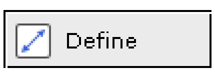
Select any rectangular area of the full drawing to display.

To restore the full drawing, zoom out and define the clipping area over the invisible drawing limits.

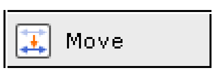
Slab parameters

Revise the slab parameters for the current space; refer to Parameters - Slabs 869.

8.17.1.2.2 Dimension lines



Define a new dimension line



Move an existing dimension line

Delete an existing dimension line

Specify dimension line text size, arrowhead style, etc.

Define

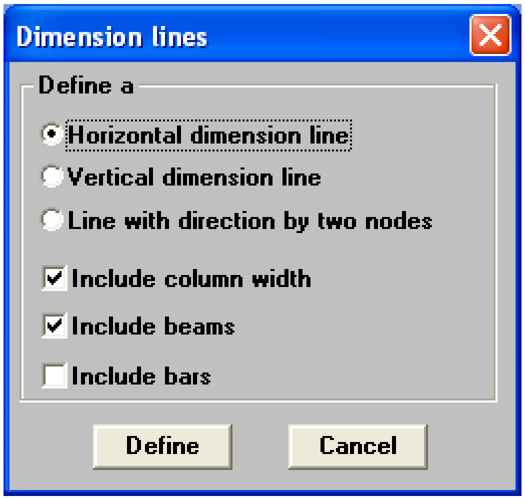
Add dimension lines to the drawing:

General:

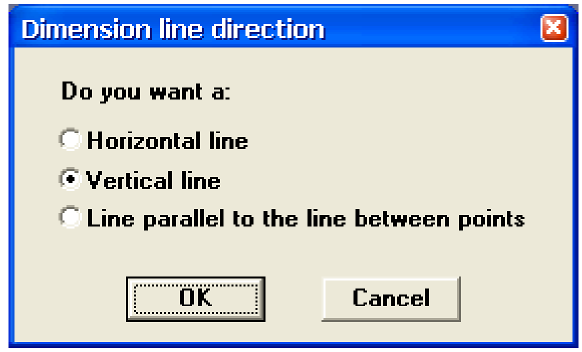
- specify the direction and the parameters:

Wall sections:

- select two wall corner points
- specify the direction:



- select the beams/bars/columns defining the dimension lines using the standard node selection option
- move the cursor to the line location and click the mouse

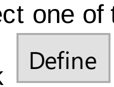


- select the wall corners defining the dimension lines using the standard node selection option
- move the cursor to the line location and click the mouse

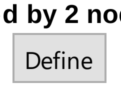
Direction:

Select one of the following options:

Horizontal/vertical

- Select one of these options to plot a vertical or horizontal dimension line.
- click 

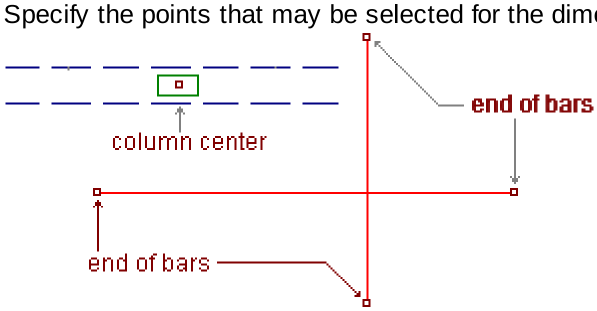
Defined by 2 nodes:

- click 

- Select two nodes defining the dimension line direction; the dimension line will be drawn parallel to a line connecting two selected nodes. The nodes that may be selected are column centers, beam ends or slab corners.

Include:

Specify the points that may be selected for the dimension line:



Note:

- the dimension lines are drawn with the column dimensions and the beam width even though only the center points are selected.

Move/delete

Move/delete a dimension line or elevation line from the model
- Highlight a dimension line or elevation line and click the mouse
- for "Move", move the ⮰ to the new location and click the mouse.

Parameters

Refer to Display - dimension line parameters 972.

8.17.1.2.3 Grid lines

Add existing grid lines to the drawings. Note that the grid lines must be defined in geometry

Remove an entire set of grid lines from the drawing:
- move the ⮰ to the centre of the set so that the ■ is displayed; click the mouse.

Add/restore all available grid lines to the drawing

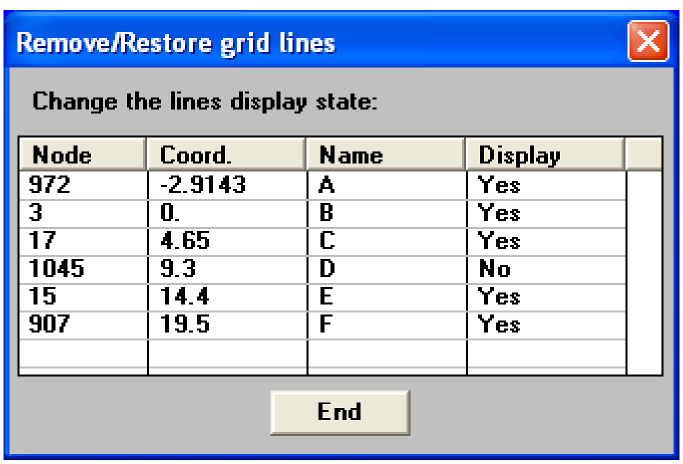
Delete an individual grid line from a set

Restore individual grid lines to a set

Move the location of the grid line text (not the grid line locations)

Restore / delete lines

Individual lines may be removed from the display and later restored:
- move the ⮰ to the grid line set and click the mouse; the program display a list of grid lines in the set:


Remove/Restore grid lines

Change the lines display state:

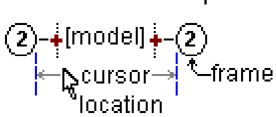
Node	Coord.	Name	Display
972	-2.9143	A	Yes
3	0.	B	Yes
17	4.65	C	Yes
1045	9.3	D	No
15	14.4	E	Yes
907	19.5	F	Yes

End

- move the ⮰ to the appropriate row and click the mouse to toggle the 'Display' to **Yes/No**.

Move grid names

- highlight the grid line (the ■ appears at the midpoint of the line) and click the mouse
- place the ⮰ at the new grid line location and click the mouse. Note that the ***side*** of the frame closest to the slab will be placed at the ⮰ location.

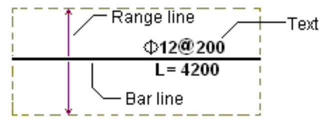


8.17.1.2.4 Bars

Unify	Combine two groups of bars to a single group.

The following options revise the bar details only and do not change the reinforcement:

Move text	Move the bar text - along the bar.
Move bar l...	Move the bar line within the frame (with the text).
Move range	Move the range line, within the frame.



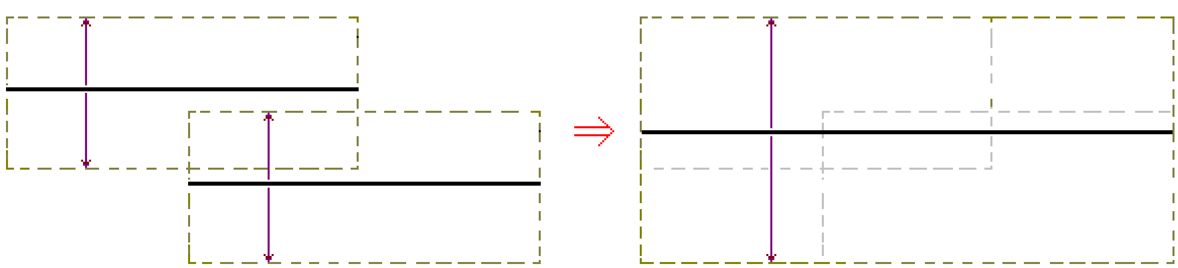
For all options:
- select a bar group
- drag the line/text to the new location

The following options revise the reinforcement:

Edit length	Revise the length of a bar.
Edit range	Revise the range of a bar
Diam./spa...	Revise the diameter and spacing of a selected bar.
Delete	Delete one or more bar groups.
Add	Add a new bar group
Bends	Add bends to bars in an existing group.

Unify

Combine two bar groups into a single group.
- the range rectangle of the new group is defined by the range rectangles of the two groups that are combined:



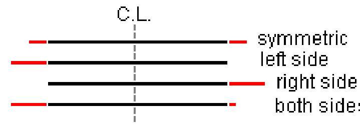
- the bar diameter and spacing of the new group is identical to that of the old group with the maximum value of Av/s (area per unit slab width).

Note:
- the program does not check whether the range rectangles overlap.

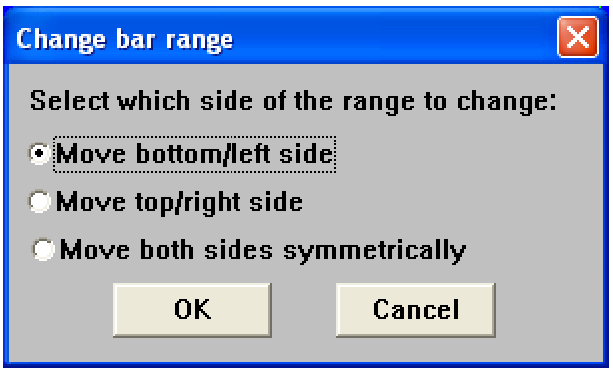
Edit length

Revise the length of a bar

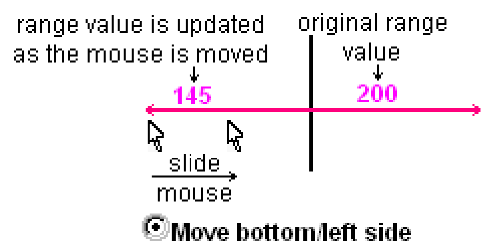
- type in the new length
- specify from which side of the bar to add/subtract the difference:



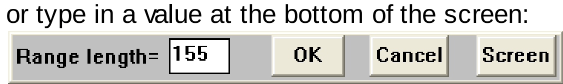
Edit range



- charge the range symmetrically or on one side only.
- slide the mouse to the new location:



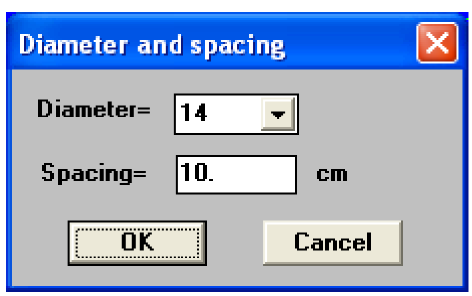
or type in a value at the bottom of the screen:



Diam/spacing

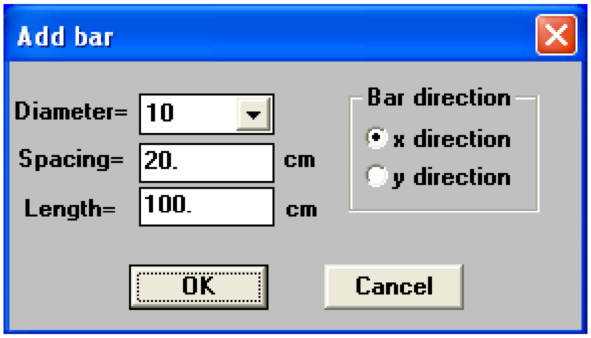
- Select a bar:

- enter new values for diameter and/or spacing:

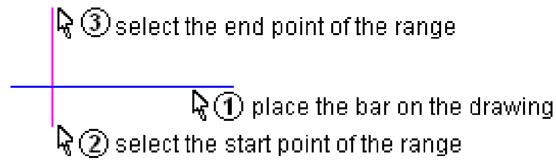


Add

Add a new bar anywhere on the drawing:

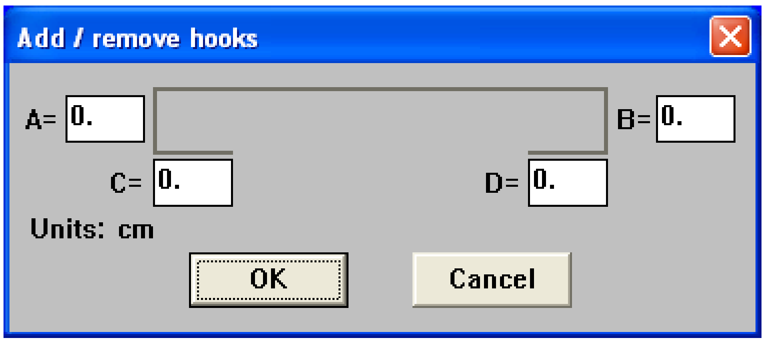


- specify the bar direction and details
- define the location on the drawing:

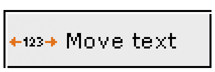


Bends

Add hooks and bends at either end according to the sketch in the dialog box:



8.17.1.2.5 Mesh



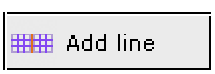
Move the mesh number along the diagonal.

Revise the diameter and spacing in one or more meshes.

Revise the dimensions of one or more meshes.

Delete one or more meshes.

Add a single mesh.

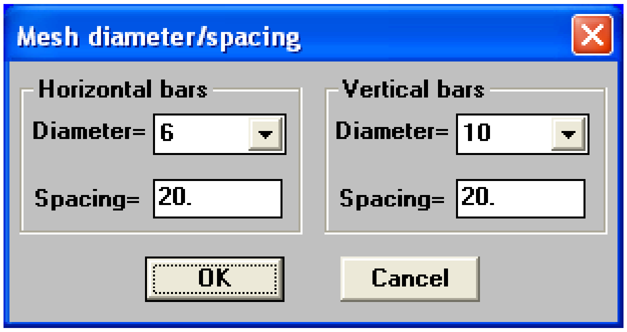


Add a line of meshes.

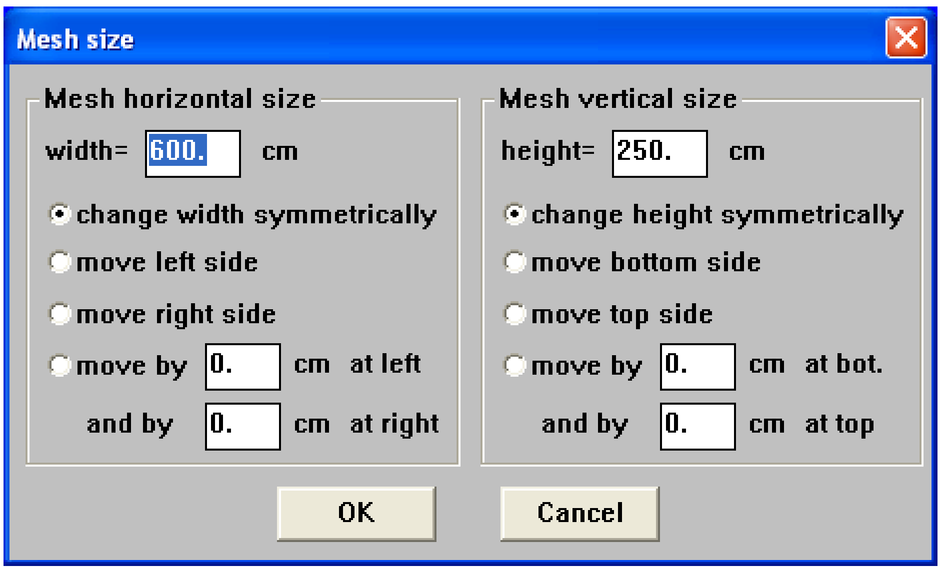
Add a grid of meshes.

Diameter/spacing

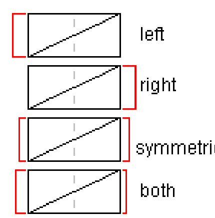
Enter new values for the diameter and spacing in both directions:



Size

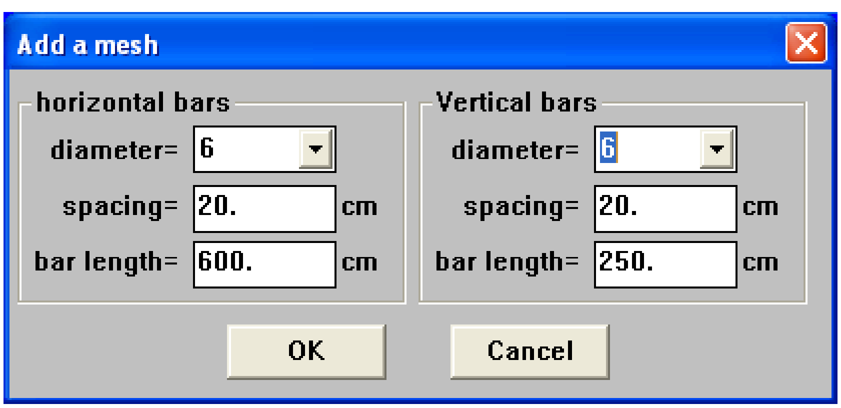


- enter new dimensions
- select the side of the mesh:

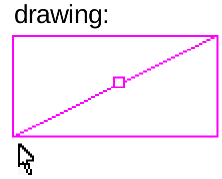


Add one

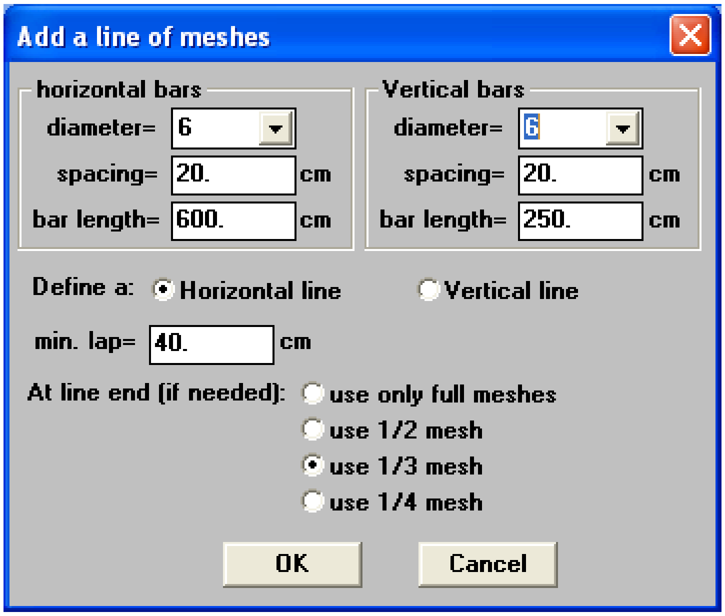
Add a new mesh anywhere on the drawing:



- enter the bar details in both directions
- locate the mesh on the drawing:

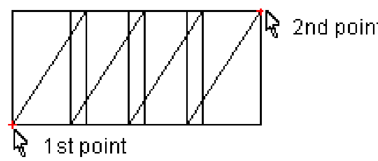


Add line

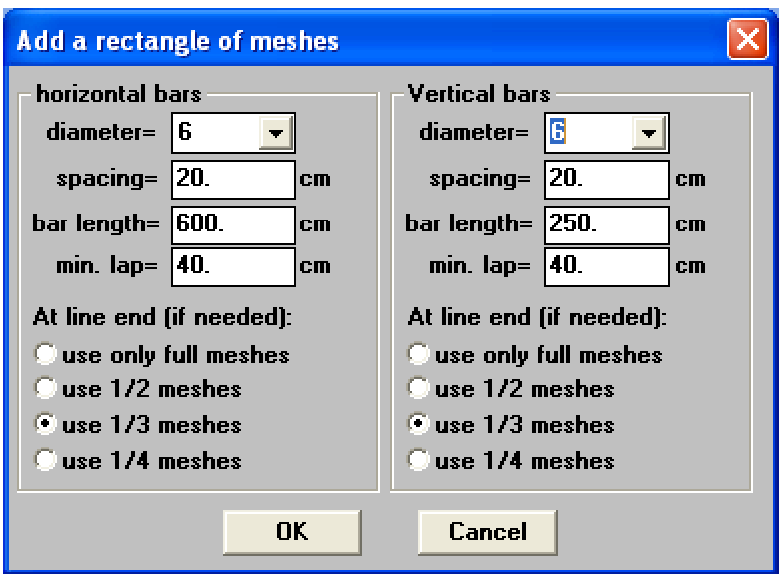


- enter the bar details in both directions
- specify the direction of the line (X,Y)
- Specify the minimum lap length; Specify a minimum lap length for meshes. the program uses the maximum of the value specified here and the Code value.
- indicate whether partial meshes may be used

- specify the mesh location by selecting two points:

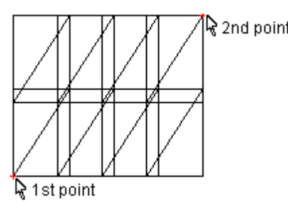


Add rectangle



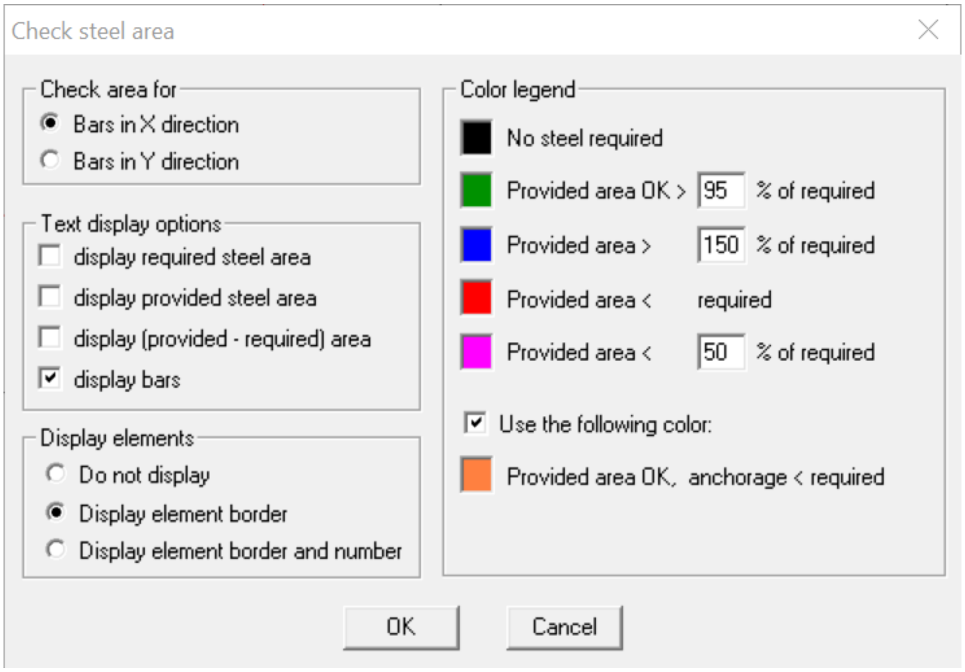
Create a grid of meshes; the option is similar to "Line of meshes".

- specify the mesh location by selecting two points:



8.17.1.2.6 Check

Display a color coded check of the reinforcement representing the ratio of (As,prov / As, req'd). Refer to Draw slabs - general 794.



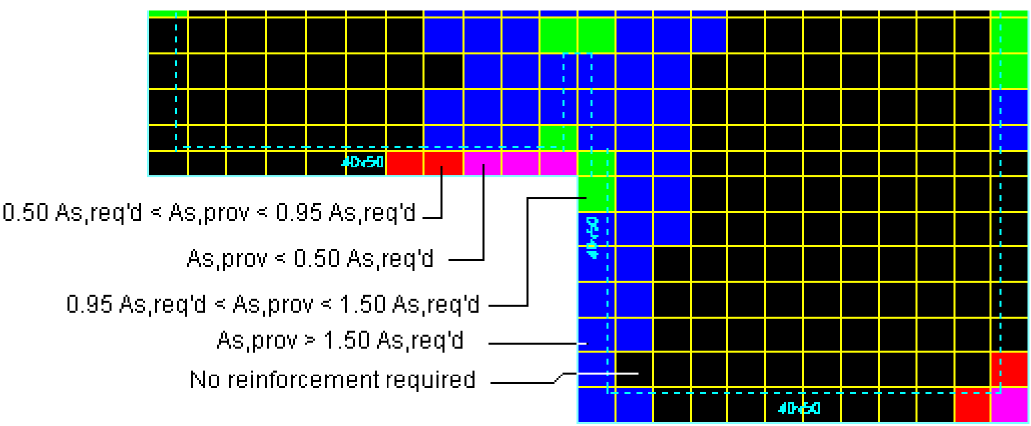
Check area for

Select the bars in one of the principal direction. The check is for the bars displayed on the drawing - top or bottom.

Color legend

Each color represents a different range of the ratio (As,prov / As, req'd); the ranges may be specified by entering different percentage values.

For example, for the percentages shown above:



The program also checks provided anchorage length for each bar group if ☑ **Use the following color** is checked; the program colors the element (orange) instead of (green) if the area is satisfactory but the anchorage length is insufficient.

Text display options

The following may be superimposed on the color-coded drawing:
- required steel area - Area/width. (the units are displayed on the title bar of the dialog box).
- provided steel area.

- (provided - required) area - a negative value indicates insufficient reinforcement.
- the reinforcement bars.
- *STRAP* element numbers.

Display elements

Select one of the elements display options:

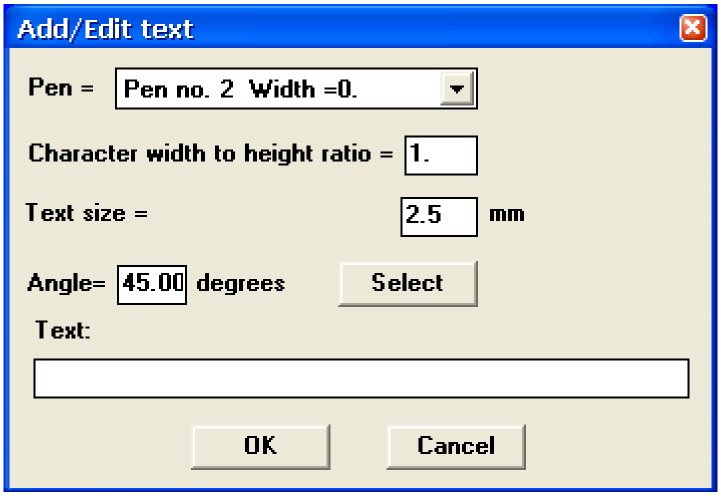
- Do not display: the elements contour lines will not be displayed.
- Display element border: the elements contour lines will be displayed.
- Display element border and number: the elements contour lines and numbers will not be displayed.

8.17.1.2.7 Additional

123 Add text	Add text 905 anywhere on the drawing.
←123→ Move text	Move an existing text. The text that may be selected is: • text defined with the previous **Add text** option. • slab name/level/thickness, beam/column names and dimensions - if added as specified in the Edit - Parameters 895 option.
Delete text	Select one or more texts. Refer to **Move text**.
Restore text	Restore slab name/level/thickness, beam/column names and dimensions that were delete using the previous **Delete text** option; text defined using the **Add text** option cannot be restored
Edit text	Select one of more texts and revise parameters (the text string may be edited for user-defined text if a single string is selected). Refer to **Move text**.
Add section	add a beam section 906 at at beam location on the drawing.
Del. section	Select and delete any of the defined sections
Add line	Add a line 906 or rectangle anywhere on the drawing.
Del. line	Select and delete any of the defined lines.
Back	Return to the Edit menu 894.

Add text

Define text and add it to the drawing:



Pen:

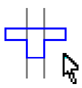
Pen widths (for User pens 1 to 6) are defined in Setup - Line width 960.

Angle:

Enter the angle (counter-clockwise = positive) or click Select and select two points (beam supports) on the drawing that define the angle.

Add section

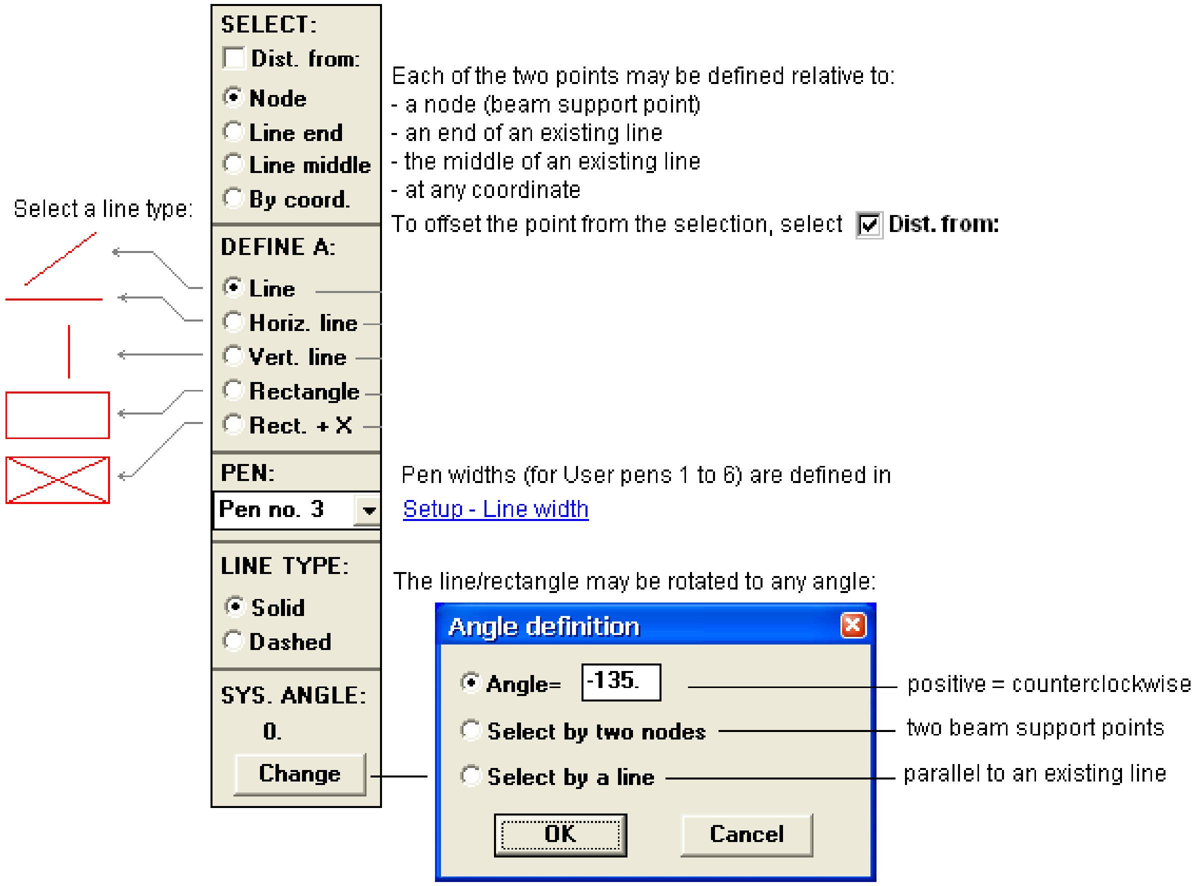
Move the along any beam and select the section location:



The section is drawn according to the parameters specified in Setup - slab sections 960.

Add line

Add a line or a rectangle anywhere on the drawing. The line/rectangle is defined by two points:



8.17.1.2.8 Undo

Undo changes made to the bars on slab drawing (similar options are available for meshes).

Select one of the following options:

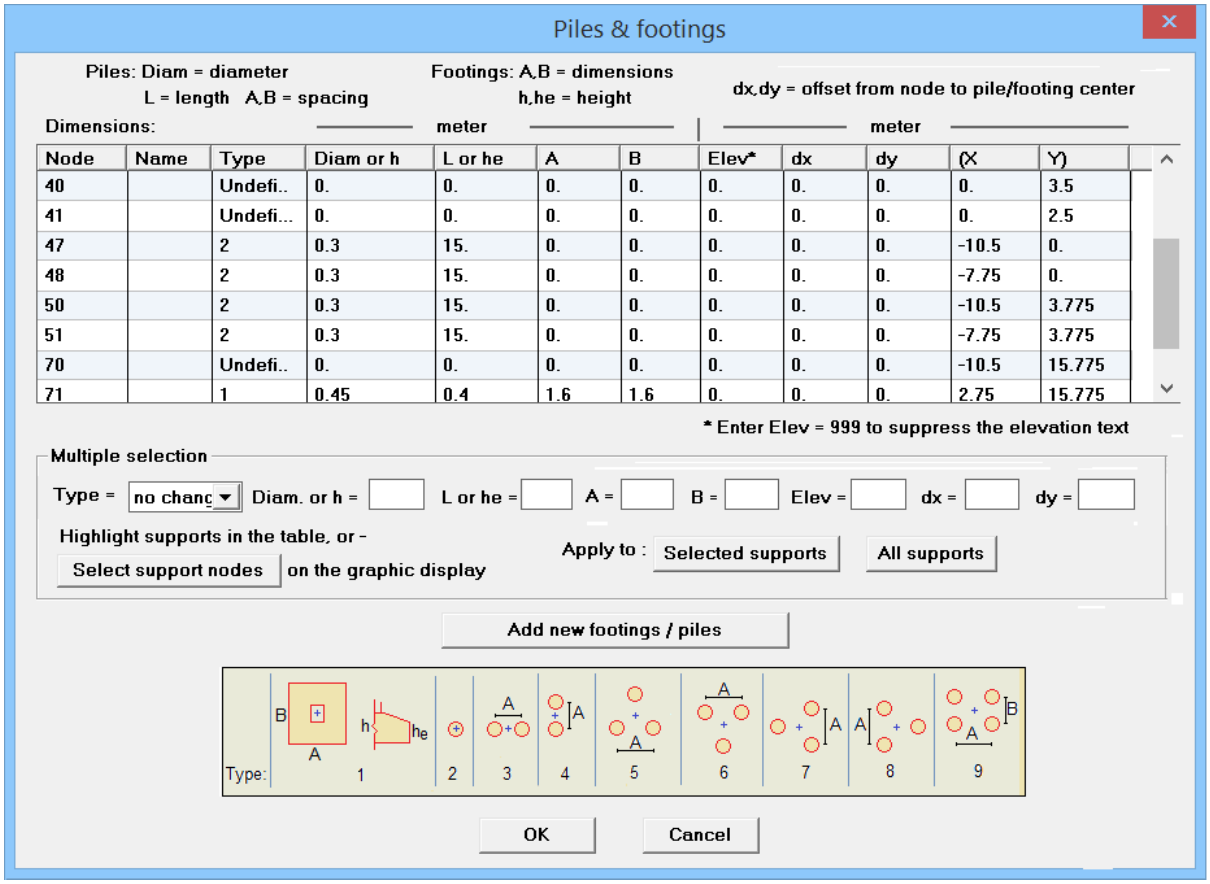
Add all deleted bars	-	restore all deleted bars to the drawing
Remove all added bars	-	erase all bars that were added manually to the drawing
Undo all changes for selected bars	-	select bars; the program restores the initial detailing calculated automatically by the program.
Undo selected changes for all bars	-	☑ select edit options; the program restores the initial program values for ***all*** of the bars in the drawing.

8.17.1.2.9 Add/edit piles ..

Use this option to add or edit piles and footings.
- the table is initially displayed with a list of all *STRAP* support nodes
- if footings were designed using the "Design footings" option in the *STRAP* results module, the program automatically retrieves the footing dimensions and displays them in the table.
- new locations may be added but they must be defined offset to one of the support nodes (the same node number may appear more than once in the table).

The table headings are:

Node	:	*STRAP* node number
Name	:	Default names are those from *STRAP* footing design or the column name. Only the names of footings/piles added here can be edited.
Type	:	Footing (type1) or piles (2 to 9, according to the sketches at the bottom of the dialog box
Diam or h	:	Type 1 : footing height Type 2-9: pile diameter
L or he	:	Type 1 : exterior footing height (= h if not defined) Type 2-9: pile length
A, B	:	Type 1 : footing dimensions Type 3-9: pile spacing
Elev	:	Footing/pile reference elevation. ***All*** values (including 0.00) are added to the drawing if ☑ **Draw foundation/pile elevation** is selected in the defaults 833. Enter 999. to suppress individual values.
dx,dy	:	Horizontal/vertical offsets from the node coordinates
(X,Y)	:	*STRAP* node coordinates (for information only)

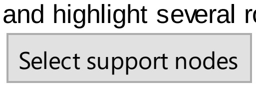


To add/edit footings or piles:

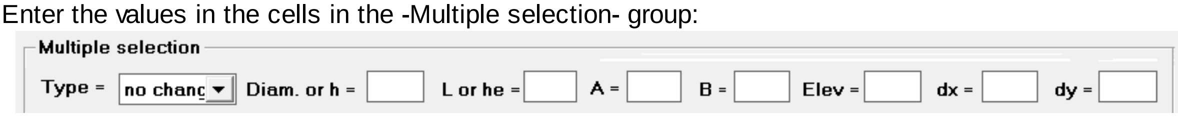
Individual footings/piles:

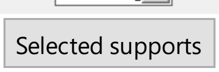
- click on the relevant line, select a type (1-9) and enter the values in the cells on the selected line..

Multiple footings/piles:

- Select the supports: there are two methods -
  a. click and highlight several rows in the table
  b. click 

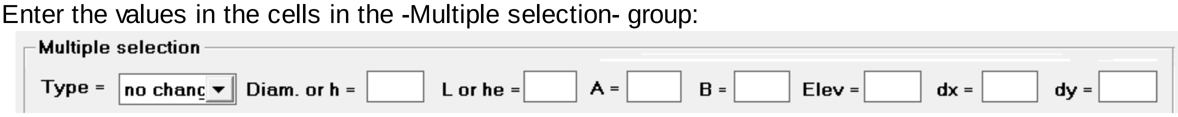
.
     - The program displays all nodes at the support levels; select several
     - the program returns to the table and highlights the selected nodes.
- Enter the values in the cells in the -Multiple selection- group:



- click 

All footings/piles:

- Enter the values in the cells in the -Multiple selection- group:



- click All supports

8.17.2 Column drawing

Select one of the following topics:
- Add 910 a new column drawing
- Edit 912 an existing column drawing

8.17.2.1 Add

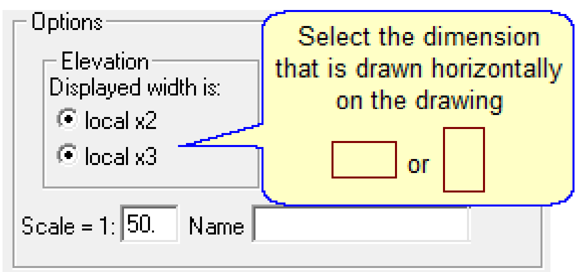
Add column elevations, column sections and bar schedules to the drawing.

Note:
- elevations should be added to the drawing first so that the cut marks of sections are drawn automatically on the elevation.

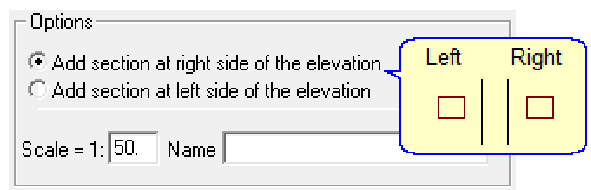
Column sections

The parameters are:

Add a column section



Create sections for all elevation levels

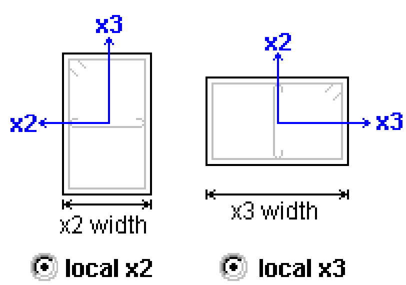


The section orientation is the same as that on the elevation

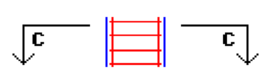
Note:

local x2 -
the section is displayed with +x2 pointing to the left

local x3 -
the section is displayed with +x3 pointing to the right



The second local axis is always drawn pointing to the top of the drawing, as shown above. For unsymmetric sections (e.g. L-shape), the flanges are drawn on the side specified in *STRAP* geometry.

To add a column section to the drawing:
- set the menu to **Section** or **Sections for each elevation level**
- specify the elevation scale and the orientation 910
- enter the column name: To draw the section mark on the elevation as , enter **c-c** (*no blank spaces !!!*). If **Create sections for all elevation levels** was selected, the first character is incremented

- for **Create sections for all elevation levels**, the program automatically places the sections on the drawing; select **Add sections at right side** or **Add sections at left side**
- Select the column: move the adjacent to the column/segment so that it is highlighted with the ■ and click the mouse.
- click OK
- for **Add a column section -** move the frame around the section to its correct location and click the mouse.

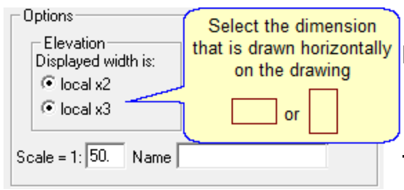
The section and the section mark on the elevation are added to the drawing.

To revise specific details on the section, return to the main menu and select the 

option.

Column elevations

The parameters are



Note:
. **local x2**
the elevation is displayed with +x2 pointing to the left
- **local x3 -**
the elevation is displayed with +x3 pointing to the right

To add a column elevation to the drawing:

- set the menu to **Elevation - entire column** or **Elevation - selected levels**
- specify the elevation scale and the orientation 910
- Select the column:

  Elevation - entire column

  move the adjacent to the column so that it is highlighted with the ■and click the mouse.

  Elevation - selected levels

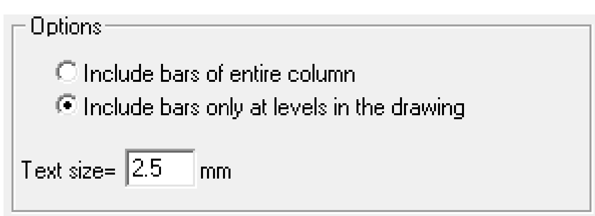
  move the adjacent to the bottom column segment so that it is highlighted with the ■ and click the mouse; repeat for the top column segment
- click OK
- Move the frame around the elevation drawing to its correct location and click the mouse.

To revise specific details on the elevation, return to the main menu and select the 

option.

Column bar schedule

The parameters are:



To add a bar schedule to the drawing:

- set the menu to **Bar schedule**
- specify the text size
- select **Include bars of entire column** or **Include bars only at levels in the drawing**
- click OK
- Move the frame around the schedule to its correct location and click the mouse.

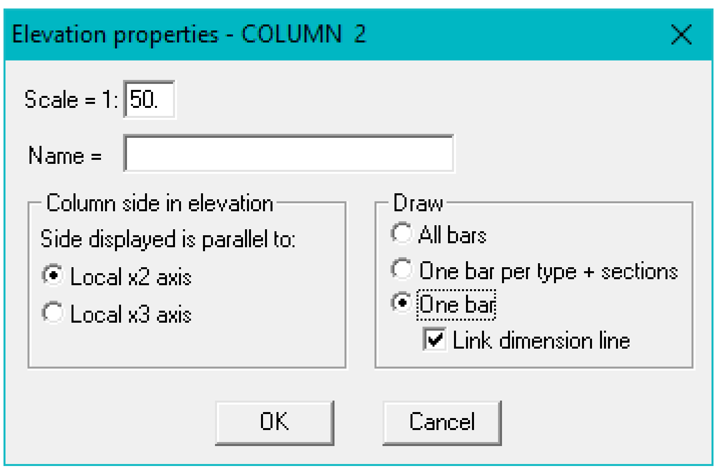
Note:

- A "**Column** (is) **on drawing**" if either its elevation *or* section is on the drawing.
- If a section is drawn at one level only and **Include bars of entire column**, then the bars in the entire column are added to the schedule.
- The bars in the schedule are numbered consecutively starting from **1**. Therefore, the same bar added to different schedules on different drawings will have different number in each schedule.

8.17.2.2 Edit

- Move the adjacent to the elevation/section/schedule so that it is highlighted with the ■ and click the mouse.
- Revise the relevant data and click OK.

Elevation



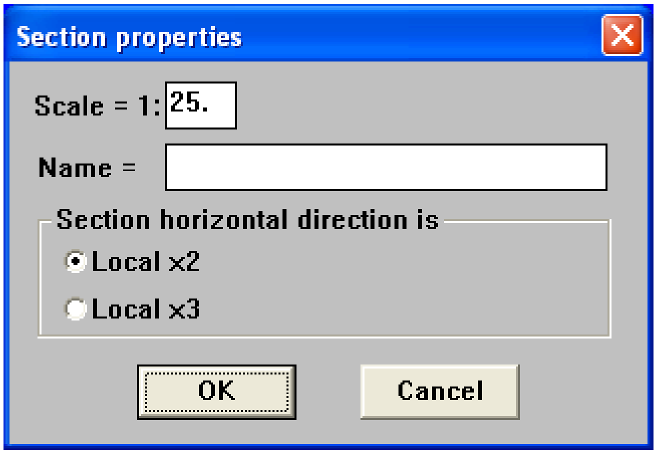
Column side in elevation:

Refer to Add - orientation 910

Draw:

Refer to Setup - column drawing - elevation 965

Section



Section horizontal direction:

Refer to Add - orientation 910

8.17.3 Column table

For a new table:

- select the columns using the standard beam selection option
- specify the table parameters 914

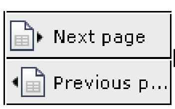
To edit a table: the following options are displayed in the side menu:

Revise the general table parameters, including the bar numbers.

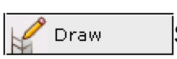
Add/delete elevations to the table, delete columns or rearrange the column order in the table.

Add new columns to the table.

Create a bar schedule for the reinforcement displayed in the table.



Displayed only when the table does not fit into a single page



Select and edit another column drawing/table or create new ones.

Return to the main column design menu.

8.17.3.1 Parameters

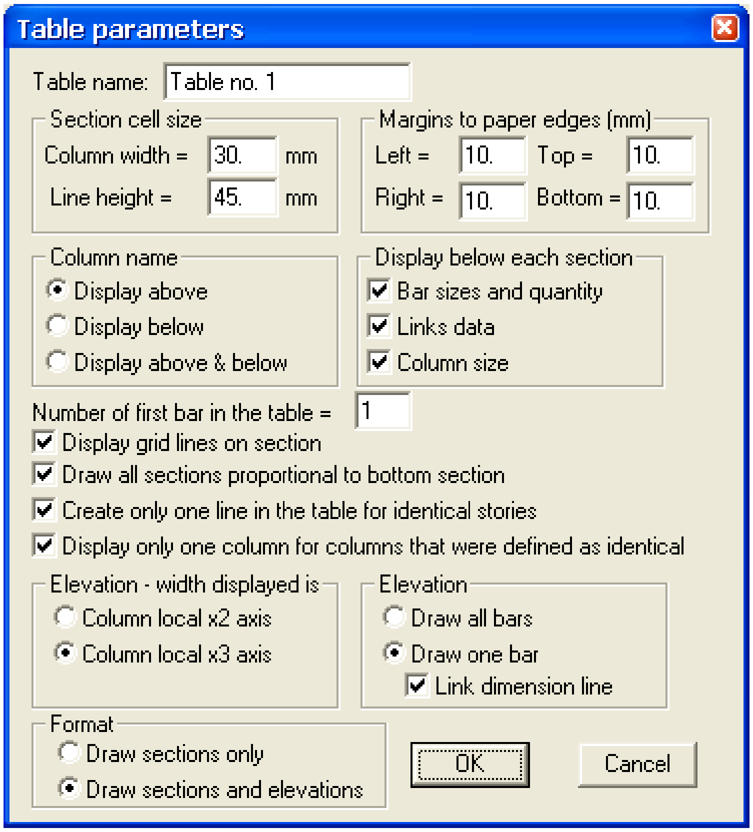
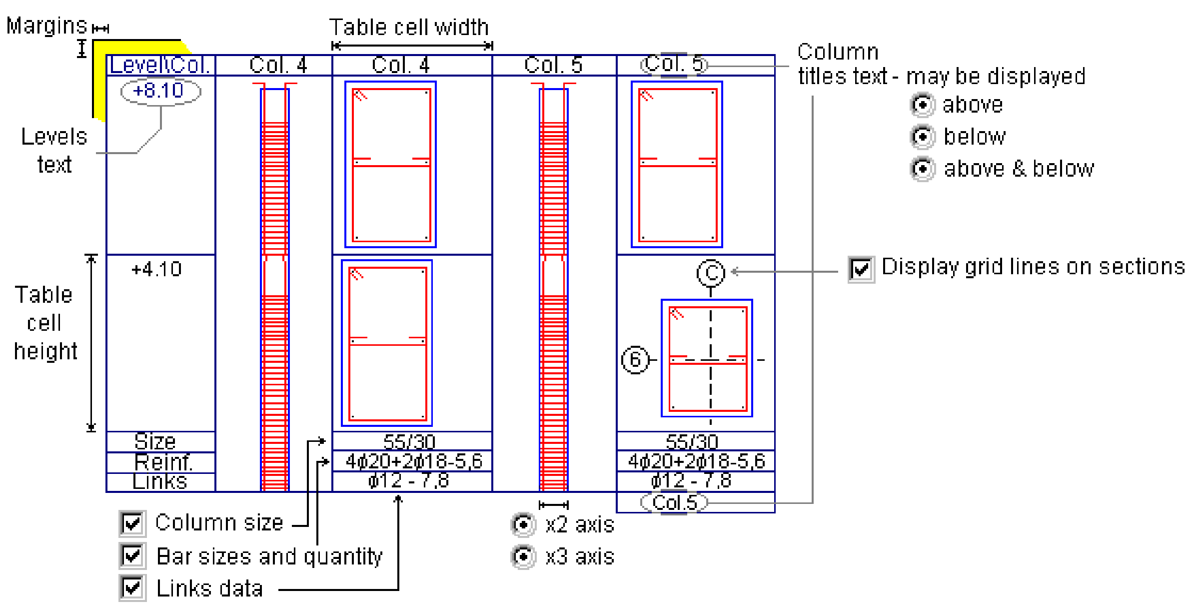


Table name

Define a title for the column table.

Parameters

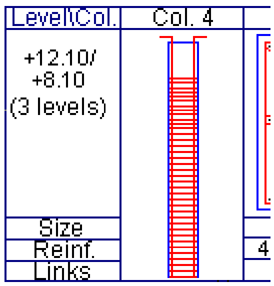


Number of first bar

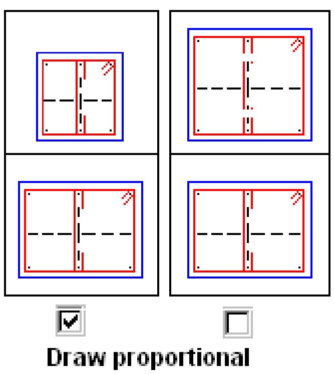
Specify the bar number of the first bar in the table; the bars are numbered consecutively.

Create only one line

Identical columns at adjacent levels may be combined to one row in the table. For example:



Draw all sections proportional to the bottom section

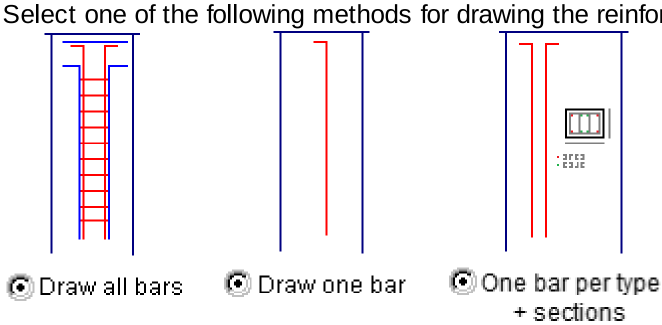


Display only one column if identical

Display only one column in an identical list.

Elevation

Select one of the following methods for drawing the reinforcement in the elevation:



Sections and elevations

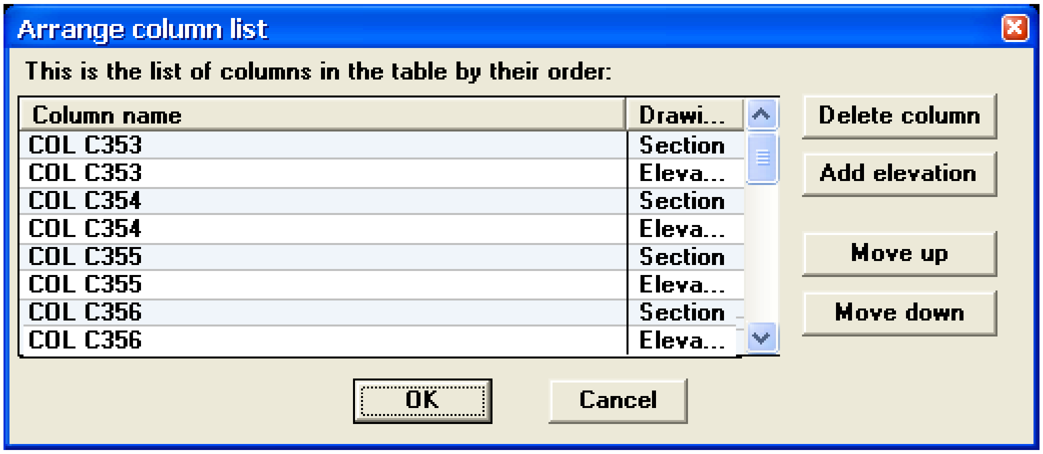
Create the table with sections only or create it with both sections and elevations.

Note that the elevations may be added later using the 

option

8.17.3.2 Edit

Add/delete elevations to the table, delete columns or rearrange the column order in the table.



Add/delete

The table is initially drawn with only the sections. To draw the column elevation alongside the section:
- click and highlight the column line in the list
- click the [Add elevation] button

To delete a section or elevation from the table:
- click and highlight the line in the list
- click the [Delete column] button

Move up/down

Revise the order of the columns in the table by rearranging the column lines in the menu:
- Click on a line in the list to highlight it
- Click the [Move Up] or [Move Down] to move the line to its new location

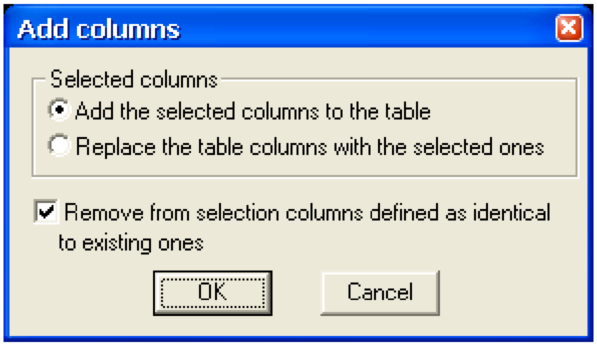
Note:
- For any particular column, the elevation is drawn to the right of the section if the **Elevation** line in the list is below the **Section** line for that column

---

Note:
- to add new columns to the table, select the [Add column] option.

8.17.3.3 Add column

Add new columns to the table or replace existing columns with others:

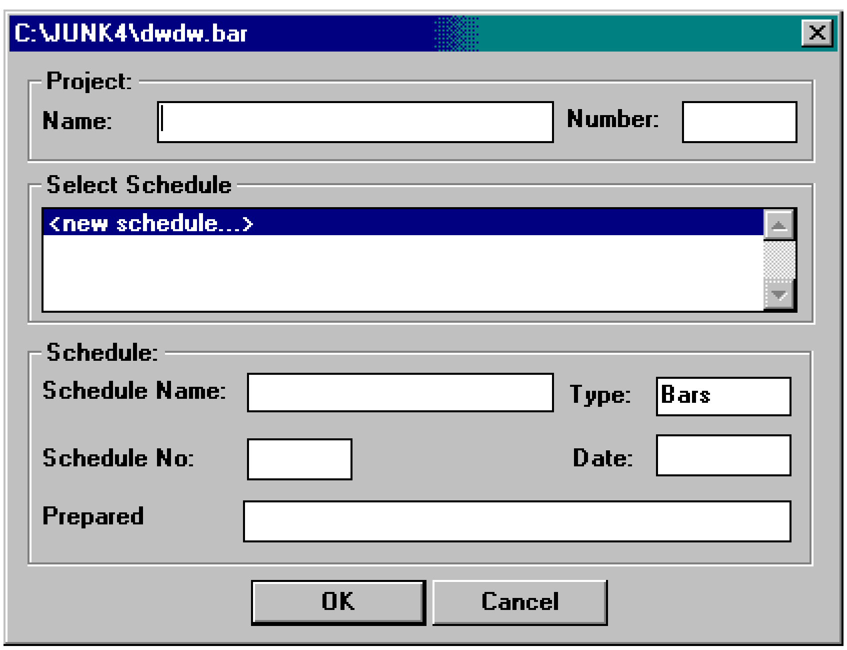


- Select **Add...** or **Replace...**
- set if you want "identical" columns to be displayed only once in the table
- Select columns using the standard beam selection option.

8.17.3.4 Bar schedule

Create a bar schedule file that can be edited and printed using the ***BARSW*** program (ask your *STRAP* dealer for more information). The file lists all reinforcement contained in the columns in the current table/ drawing.

To create the schedule:
- enter the schedule file name (new or existing)
- enter the general schedule data:

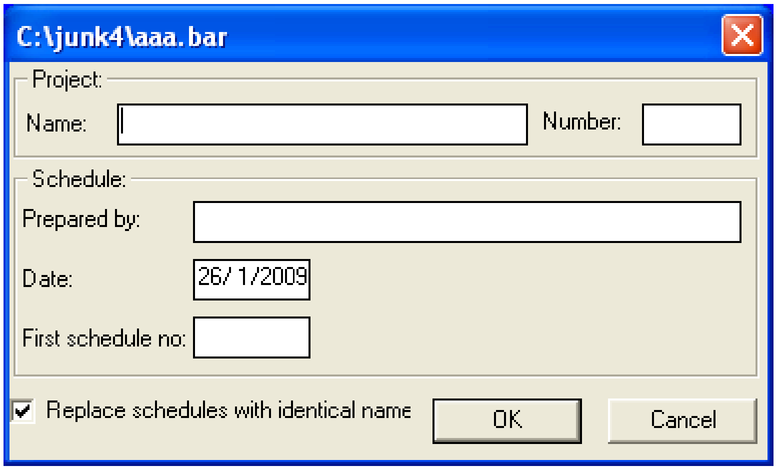


Note that the **Schedule name** is added to the **Select schedule** list.

8.17.4 Bar schedule

Create or update a bar schedule file for ***all*** drawings in the model:
- separate schedules (bars and/or mesh) are created for each drawing.
- the schedules can then be updated and printed (refer to the BARSW manual for more information)

To create/update the schedules:
- enter the name of a new file or select an existing file
- enter/update the project (model) details:



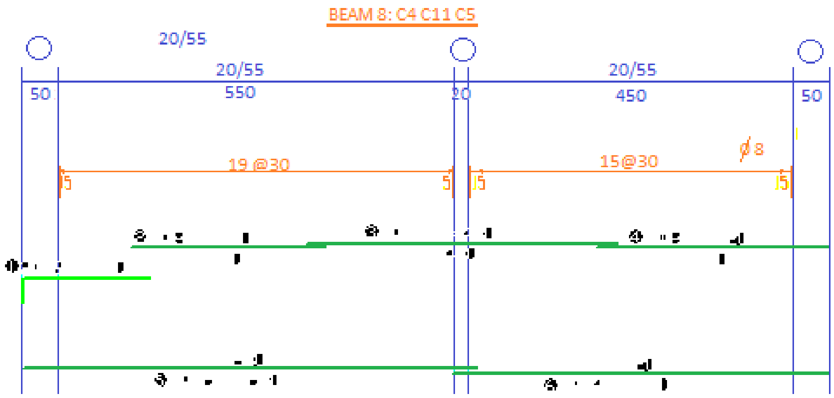
where:
- **First schedule no**:
  all schedules are numbered consecutively starting from this value (the number may be revised when editing the schedule)
- **Replace schedules with identical name**
  Note that the initial Schedule name is the same as the Drawing name.
  ☑ the program searches for schedules with the same Drawing names and replaces them.
  ☐ the schedules are appended to the end of the file (i.e. there may be two schedules for the same drawing in the file)

Note:
- if additional bars are added manually to the schedule, these bars are not erased if ☑ **Replace schedules with identical name** is selected.
- if either a Drawing name or the corresponding Schedule name is revised after the schedule is created and ☑ **Replace schedules with identical name** is selected, the program adds the schedule to the end of the file and the old version must be deleted manually.

8.17.5 Beam detailing

Add the beam reinforcement drawing for beams detailed in *BEAMD*.

For example:



Select one of the following topics:

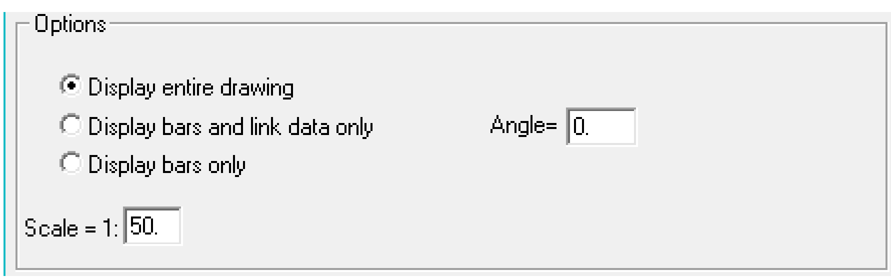
- [Add](910) a beam detailing drawing
- [Edit](912) an existing beam detailing drawing

To display a demo video that explains how to create, design & detail BEAMS and add the beam detailing to drawings:

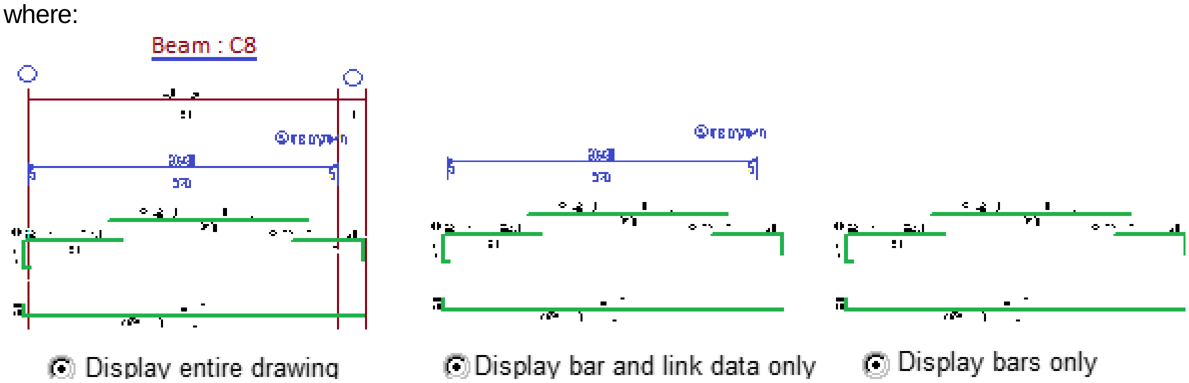
- *click on*  *to start the video*
- *then click on*  *to enlarge the display.*

8.17.5.1 Add

The options are:



where:



and the reinforcement drawing can be rotated to any angle.

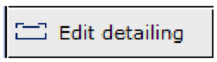
To add a beam reinforcement drawing:

- select Beam detailing
- specify the parameter and click OK
- select a beam in the model display.
- drag the rectangle that outlines the drawing to the correct location and click the mouse

Note:
- only beams that have been detailed in BEAMD can be added to the drawing

8.17.5.2 Edit

revise the display type, scale and angle. Refer to beam detailing - add 919



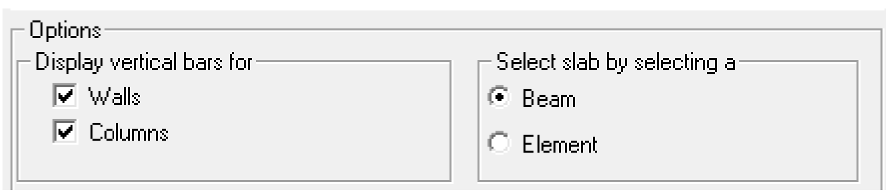
Modify the detailing for the selected beam in BEAMD. The program will automatically update the detailing displayed on the drawing.

Return to the Edit menu 892

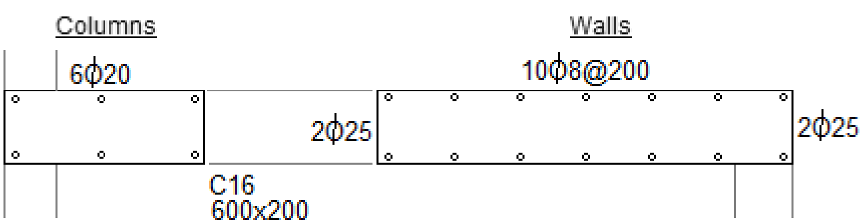
8.17.6 General Arrangement

Add a slab drawing without the slab reinforcement. The drawing displays beams, column & wall sections (with reinforcement), along with associated text.

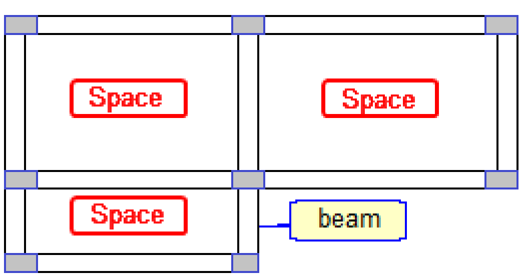
The parameters are:



Display the vertical reinforcement in columns and walls ***below*** the selected plane. For example:



Note:
- The program automatically identifies all spaces bounded by beams. For example:



- text for space names, elevation and thickness can be added to the drawing. Refer to Edit 921.

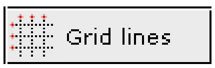
To add a general arrangement drawing:

- Click General arrangement drawing
- Specify the rotation **Title** and **Scale**
- Select a level by moving the to any beam in the floor so that it is highlighted with the ■; click the mouse
- drag the rectangle that outlines the general arrangement drawing to the correct location and click the mouse.

8.17.6.1 Edit

- revise the scale, clip the drawing
- add texts such as elevations, space names, thickness, column names, beam names.

define dimension lines and add them to the drawing.

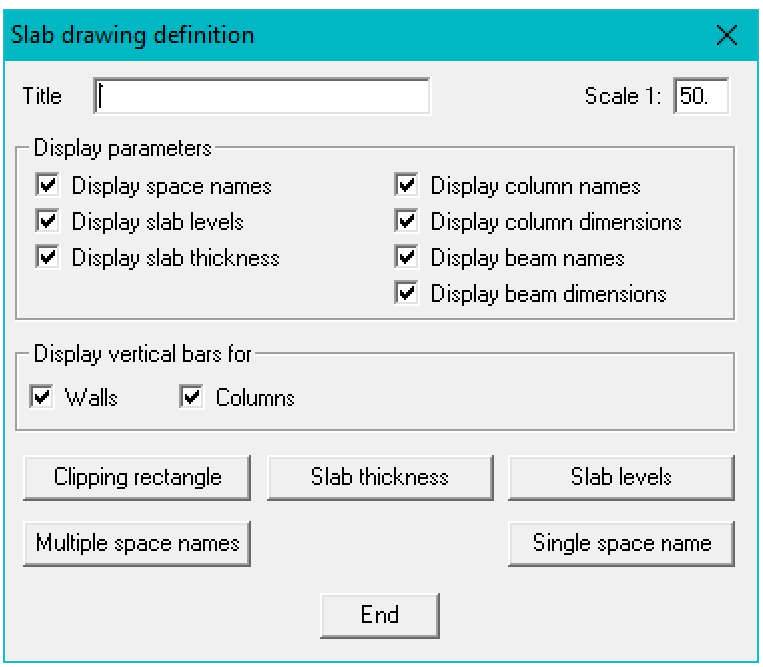


add grid lines to the drawing

Add text, beam sections and lines to the slab drawing.

Return to the Edit menu 892

Parameters

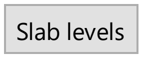


Texts:

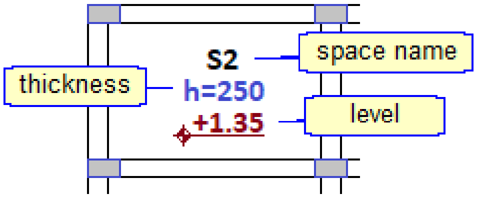
Define and display any of the selected texts on the drawing:
Note:

- "Space names", "slab thickness" and "slab levels" are added for each space.

  these names and values are defined by clicking on Single/multiple space names, Slab thickness and



. For example:



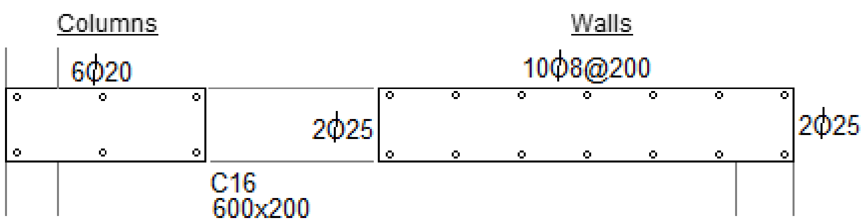
- "Beam names" are the names defined in Beam - define names 839
- "Column names" are the names defined in Columns - define names 844
- the program automatically retrieves the beam and column dimensions

All of these texts may be edited in the Edit - Additional 905 options menu (space names, thickness and level texts can be revised by redefining them).

The default text size for these options are specified in Setup - Text 961.

Vertical bars:

Display the vertical reinforcement in columns and walls ***below*** the selected plane. For example:



Clipping rectangle:

Select any rectangular area of the full drawing to display.

To restore the full drawing, zoom out and define the clipping area over the invisible drawing limits.

8.17.7 Wall sections

Select one of the following topics:

- Add 892 a new wall section drawing
- Edit 894 an existing wall section drawing

To display a demo video that explains how to create, design & detail WALLS and add the wall detailing to drawings:

- *click on*  *to start the video*
- *then click on*  *to enlarge the display.*

Note:

- *this video includes an explanation on "design units" and result interpretation*

8.17.7.1 Add

Add any *STRAP* wall section (defined in the geometry - walls option) to the drawing, with or without reinforcement:

Without reinforcement:

- Click **Section**
- Specify the rotation **Angle** and **Scale**
- select a section from the list
- locate the section on the drawing

Note that additional lines, text and dimension lines can be added in the [Slabs - edit](894) option.

With reinforcement:

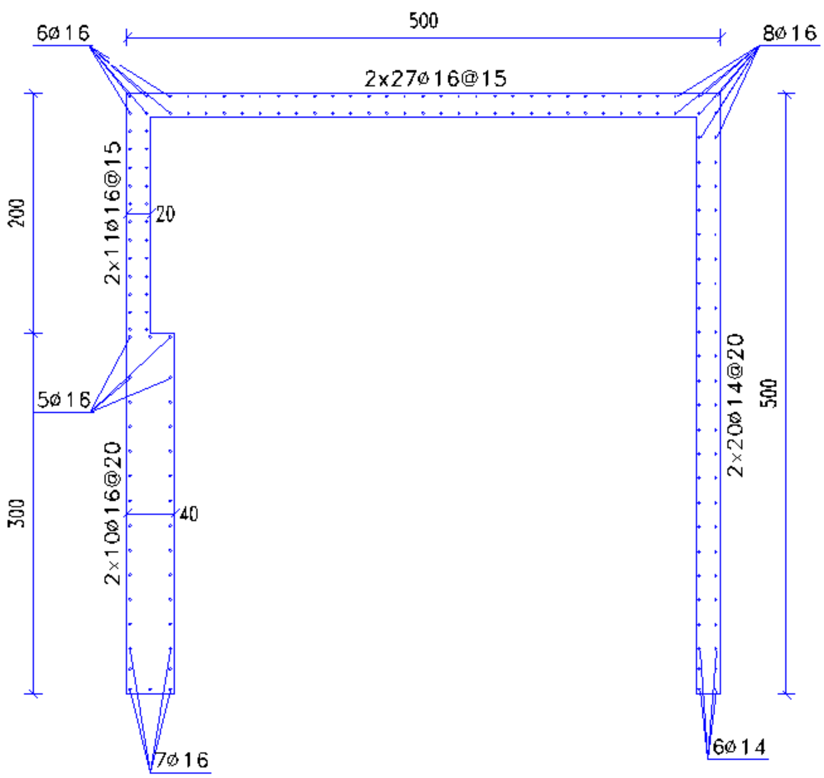
- Click **Section with vertical reinforcement**
- Specify the rotation **Angle** and **Scale**
- select a wall on the graphic display (highlight any of its segments and click)
- locate the section on the drawing

Note:

- the walls must be designed before they are placed on the drawing
- additional lines, text and dimension lines can be added in the [Slabs - edit](894) option.
- for information on the method used by the program to unify the reinforcement at the wall corners, refer to [Wall reinforcement - general](923).

8.17.7.2 Reinforcement-general

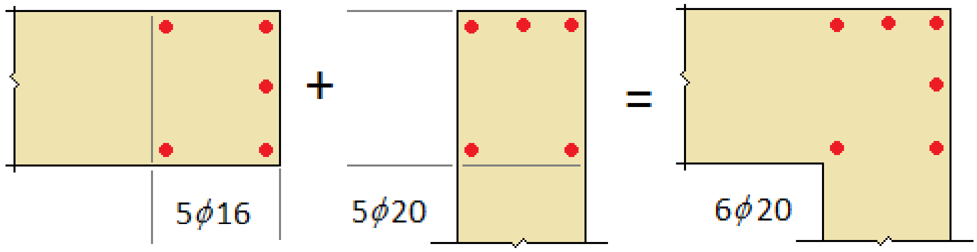
This option creates a wall cross section drawing with reinforcement. The program currently draws only the vertical reinforcement. For example:



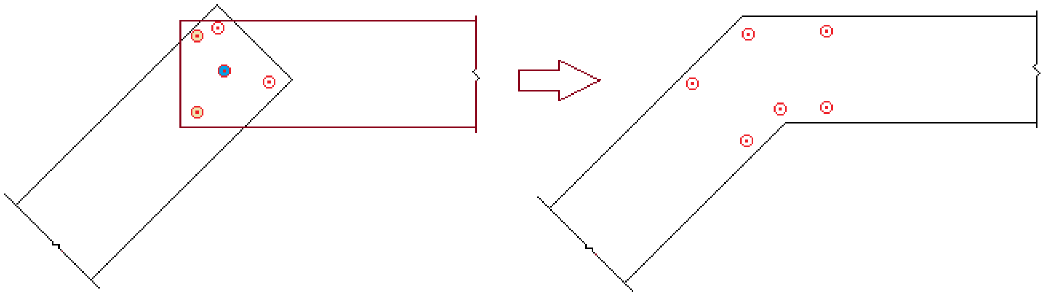
Creating "design units" will give a more economical reinforcement arrangement at the corners.

The program combines the reinforcement at the common corner of connected "design units":

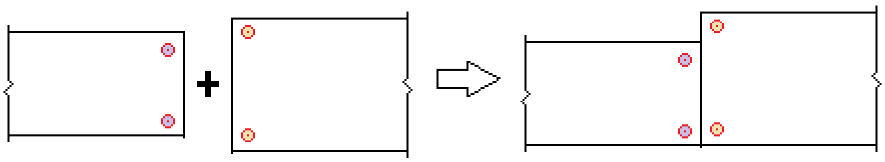
- the maximum diameter is used if the diameters in the attached segments are not equal
- the bars at the faces are combined. For example:



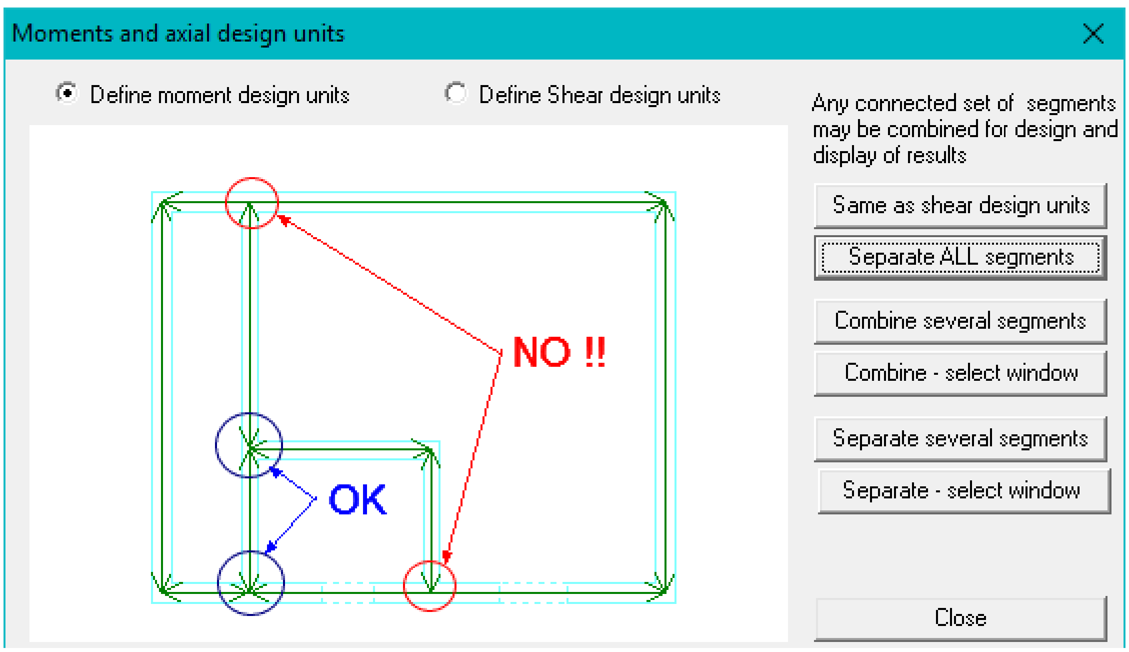
- attached segments that meet at angles > 0° and < 90° are combined as follows:



- attached segments that are a continuation (e.g. change of thickness, with or without offset) are combined as follows:



- "design units" should not be created so that other units/segments end along the length of the unit because there will be no end reinforcement to combine:



8.17.8 Foundation plan

The program draws foundation plan, including square footings and piles:
- the program automatically retrieves the dimension data for footings designed in the results module. These dimensions can be modified only in the footing design module.
- pile dimensions must be input manually by the user

The following elements can be placed on the drawing:
- footings and piles, including a name, dimensions and elevation.
- beams and walls at the foundation level
- grid lines, including labels and dimensions
- a coordinate grid

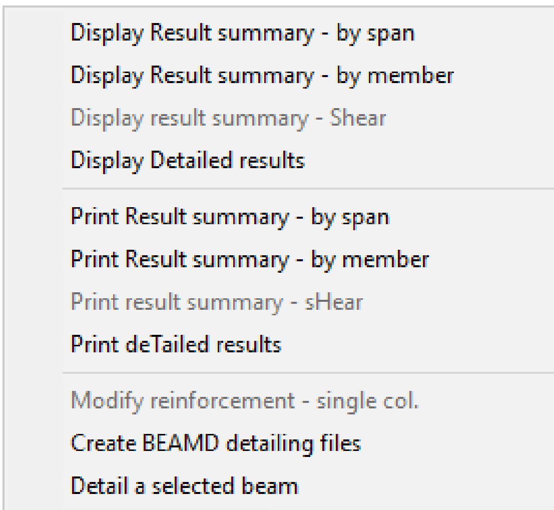
Add a foundation plan

- Click **Foundation plan**
- Specify the rotation **Angle** and **Scale**
- locate the plan on the drawing

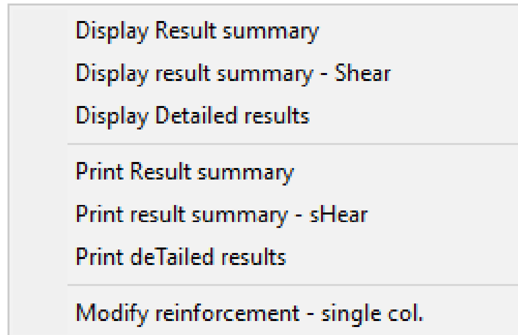
Note:
- additional lines, text and dimension lines can be added in the Edit 894 option.

8.18 Results

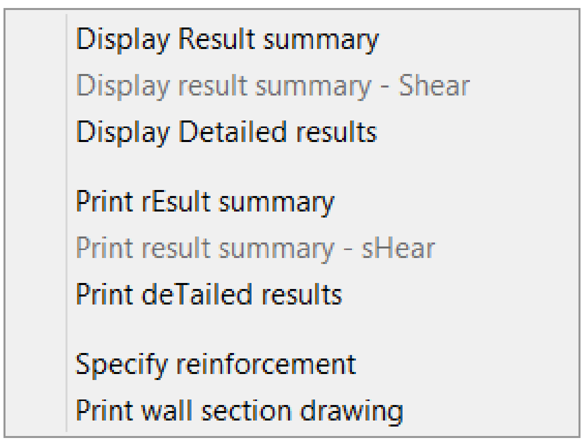
Beams



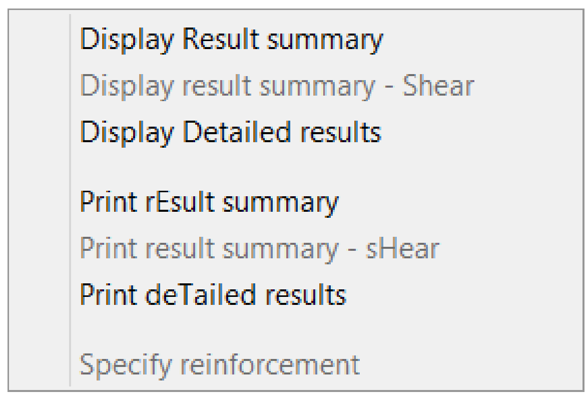
Columns



Walls



Slabs

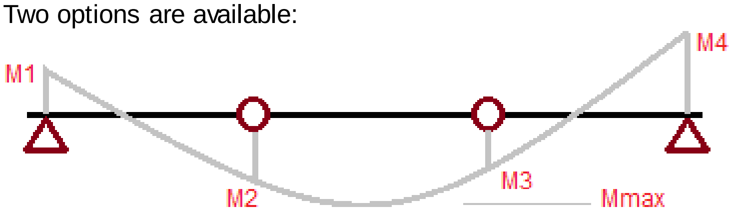


8.18.1 Result summary

Result summary - beams 926
Result summary - columns 928
Result summary - walls 930
Results summary - slabs 931

8.18.1.1 Beams

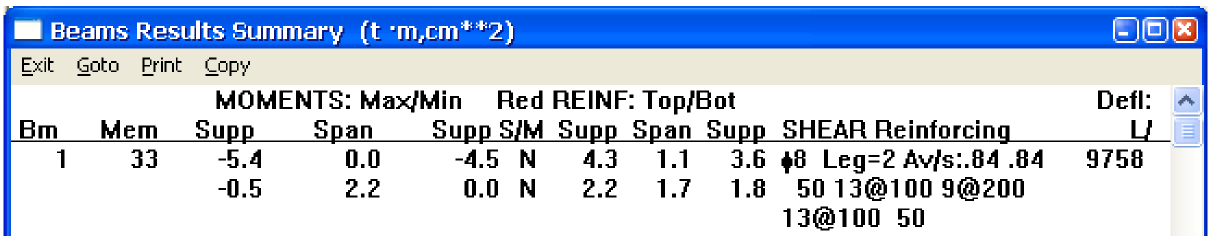
Two options are available:



Display result summary - by span
The program displays only one entry for the three members forming the span shown above; moment values M1, Mmax, M4 are displayed along with the corresponding reinforcement areas.

Display result summary - by member
The program displays three entries, one for each of the members forming the span; the first entry shows moments M1 and M2, etc

For beam result summary of walls coupling beams. Refer to Coupling beams-Result summary 927



Beam: Concrete module beam number.
Mem: *STRAP* member number
for wall beams: the start and end nodes of the segment.

Moments: 1st line: minimum moments at support / span / support. If the minimum span moment is the same sign as the maximum span moment, 0.0 is displayed.
2nd line: maximum moments at support / span / support. If the maximum support moment is the same sign as the minimum support moment, 0.0 is displayed.
Note that "minimum" refers to the smallest positive moment or the largest negative moment; "maximum" refers to the largest positive moment or the smallest negative moment.

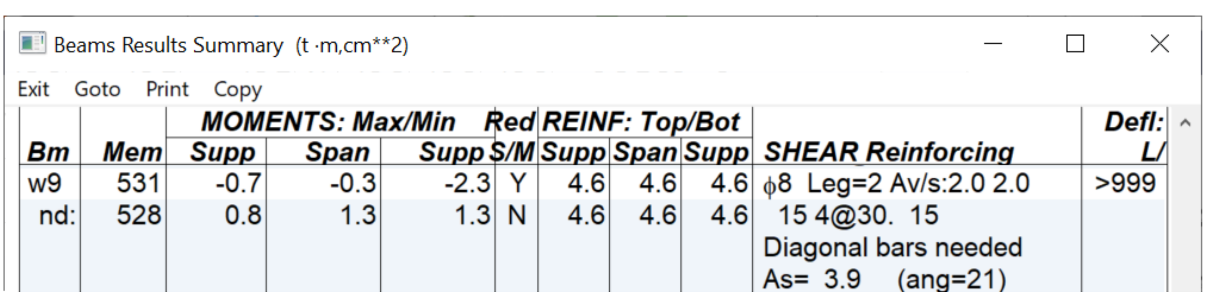
Red S/M: 1st line: shear reduction at supports: Y = Yes , N = No.
2nd line: moment redistribution : Y = Yes , N = No

Reinf: 1st line: reinforcement at top face of the beam at support / span / support (in²)
2nd line: reinforcement at bottom face of the beam at support / span / support (in²).

Shear: 1st line: Stirrup diameter, number of legs per stirrup and Av/s required at each end of the member.
2nd+ line: Gap from face of support to the first stirrup, stirrup details, gap from last stirrup to face of right support.

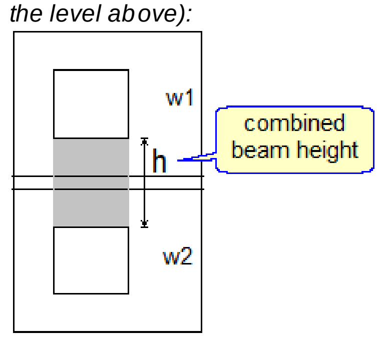
Defl: ACI,CSA.. the deflection expressed as L/**xxxx**
EC2, BS.. ratio = (Actual L/d)/(Allowable L/d): if > 1.0 a ***** is displayed adjacent to the ratio.

8.18.1.1.1 Coupling beams-Result summary



Beam: The wall number, e.g. **W51**.

note: If there are openings in the middle of both segments above and below a level, the program combines the two resulting beams to form a single beam for design (the results are shown only in the level above):



Mem: The start and end nodes of the segment.

Moments:
- 1st line: minimum moments at support / span / support. If the minimum span moment is the same sign as the maximum span moment, 0.0 is displayed.
- 2nd line: maximum moments at support / span / support. If the maximum support moment is the same sign as the minimum support moment, 0.0 is displayed.

Note that "minimum" refers to the smallest positive moment or the largest negative moment; "maximum" refers to the largest positive moment or the smallest negative moment.

Red S/M:
- 1st line: Shear reduction at supports: Y = Yes , N = No.
- 2nd line: Moment redistribution : Y = Yes , N = No

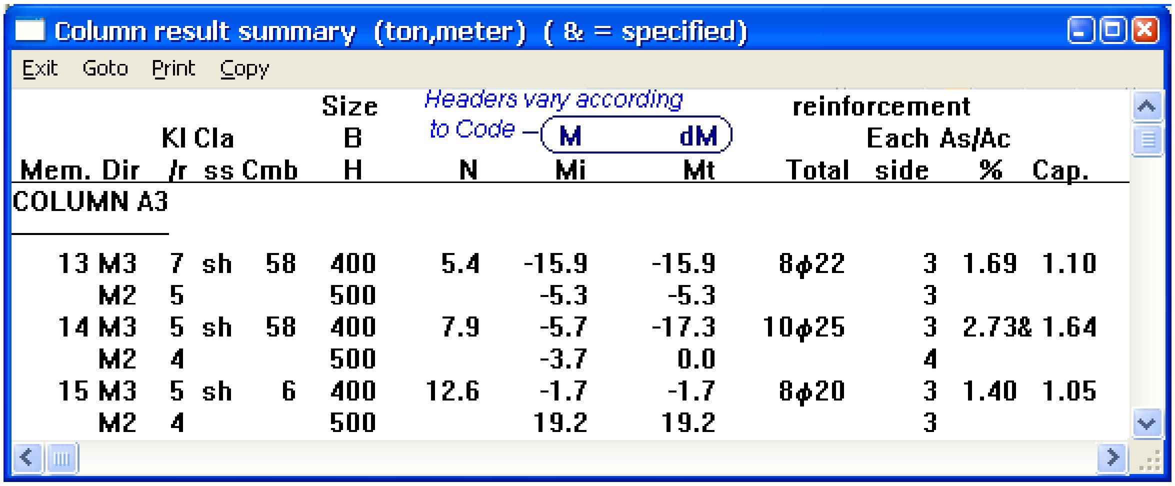
Reinf:
- 1st line: Reinforcement at top face of the beam at support / span / support.
- 2nd line: Reinforcement at bottom face of the beam at support / span / support.

Shear:
- 1st line: Stirrup diameter, number of legs per stirrup and Av/s required at each end of the member.
- 2nd line: Gap from face of support to the first stirrup, stirrup details, gap from last stirrup to face of right support.
- 3rd line: Under certain conditions some codes require diagonal diagonal bars. If the conditions are met the program displays "Diagonal bars needed".
- 4th line Reinforcement for diagonal bars at each direction, the reinforcement angle (ang=).

Defl:
- ACI,CSA.. The deflection expressed as L/**xxxx**
- EC2, BS.. Ratio = (Actual L/d)/(Allowable L/d): if > 1.0 a ***** is displayed adjacent to the ratio.

8.18.1.2 Columns

The program design all selected columns in sequence and automatically displays a design summary table:


Column result summary (ton,meter) (& = specified)

Exit Goto Print Copy

Headers vary according to Code – M dM

COLUMN A3

Mem.	Dir	Kl /r	Cla ss	Cmb	Size B H	N	Mi	Mt	reinforcement Total	Each side	As/Ac %	Cap.
13	M3	7	sh	58	400	5.4	-15.9	-15.9	8ϕ22	3	1.69	1.10
	M2	5			500		-5.3	-5.3		3		
14	M3	5	sh	58	400	7.9	-5.7	-17.3	10ϕ25	3	2.73&	1.64
	M2	4			500		-3.7	0.0		4		
15	M3	5	sh	6	400	12.6	-1.7	-1.7	8ϕ20	3	1.40	1.05
	M2	4			500		19.2	19.2		3		

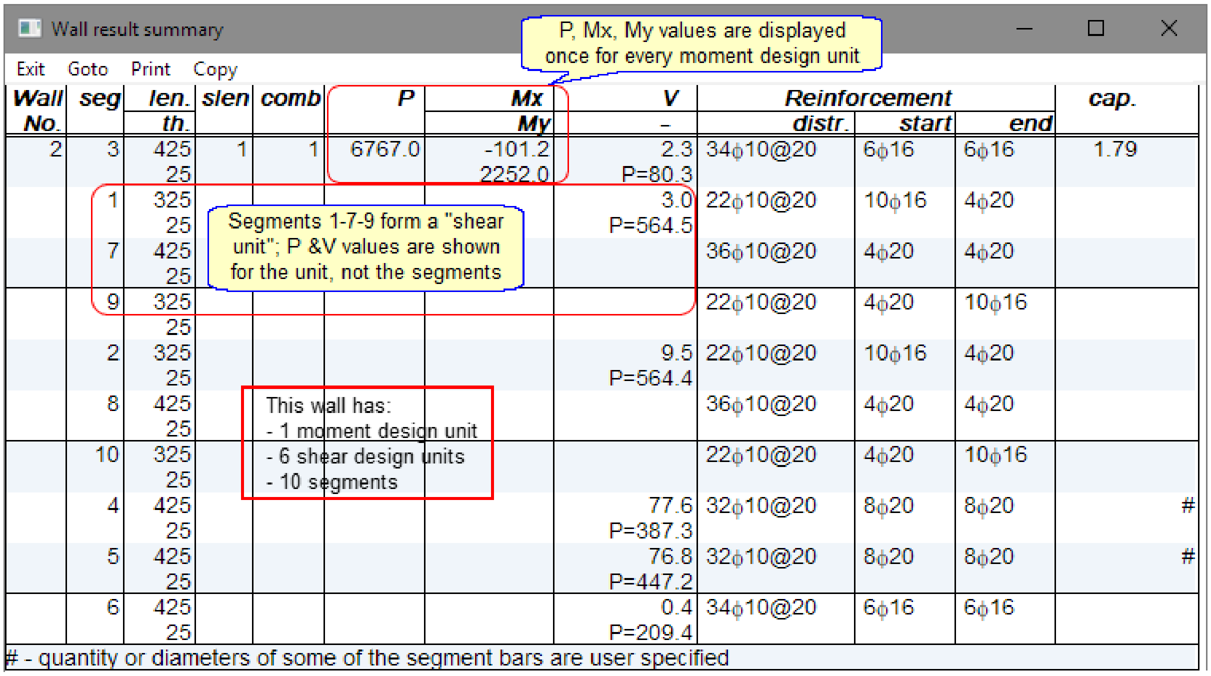
Column -concrete design module beam serial number.
Member -*STRAP* beam number
Dir -M3 or M2 ; local axes moments
Kl/r -column slenderness (Kl/h is certain codes)
Class -columns may be **SH**ort or **SL**ender
Comb -the critical design combination
Size -the external dimensions of the column
P/N -axial load in critical combination
Mi -bending moment in critical combination ***without*** additional moment.
dM/Mt -total design moment, i.e.

- dM = *M
- Mt = Mi + Madd

Total -the total number of bars in the column
Side -the total number of bars on the long face in each direction, including the corner bars.
% -the reinforcement percentage = As/Ac
& - indicates that the reinforcement is "specified"
Capacity -ratio of design capacity to the critical loads. Normally, the capacity is > 1.00. However if the maximum diameter and the minimum spacing does not provide an adequate solution, Capacity < 1.00

Note: **Te** - at the end of the line indicates that the axial load in the design case is **Tension**.

8.18.1.3 Walls



Wall No.	seg	len. th.	slen	comb	P	Mx My	V -	Reinforcement distr.	start	end	cap.
2	3	425 25	1	1	6767.0	-101.2 2252.0	2.3 P=80.3	34φ10@20	6φ16	6φ16	1.79
	1	325 25					3.0 P=564.5	22φ10@20	10φ16	4φ20	
	7	425 25						36φ10@20	4φ20	4φ20	
	9	325 25						22φ10@20	4φ20	10φ16	
	2	325 25					9.5 P=564.4	22φ10@20	10φ16	4φ20	
	8	425 25						36φ10@20	4φ20	4φ20	
	10	325 25						22φ10@20	4φ20	10φ16	
	4	425 25					77.6 P=387.3	32φ10@20	8φ20	8φ20	#
	5	425 25					76.8 P=447.2	32φ10@20	8φ20	8φ20	#
	6	425 25					0.4 P=209.4	34φ10@20	6φ16	6φ16	

- quantity or diameters of some of the segment bars are user specified

Wall no. - The *STRAP* geometry wall number.

Seg - The segment number; this is the segment number display in Geometry, not necessarily the segment number displayed in detailed results.

len/th - The wall segment dimensions - length and thickness

Slen - The minor axis slenderness value = (kl / thickness). **Slen** is displayed below the value if the wall segment is slender.

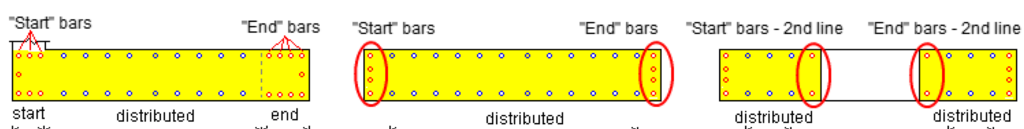
Comb - The design load combination for the moment design unit. Note that each design unit in the wall may be designed according to a different load combination

P/M/V - The design loads for the unit:

- Mx and My refer to the X,Y axes displayed by the Display - Local axes - Walls option
- the major axis shear and axial load that gives the maximum ***vertical*** reinforcement in the segment

Note that the loads are the design loads and include any additional/minimum moments as specified by the Code.

Reinf - The total number of bars - concentrated at the ends and distributed between:



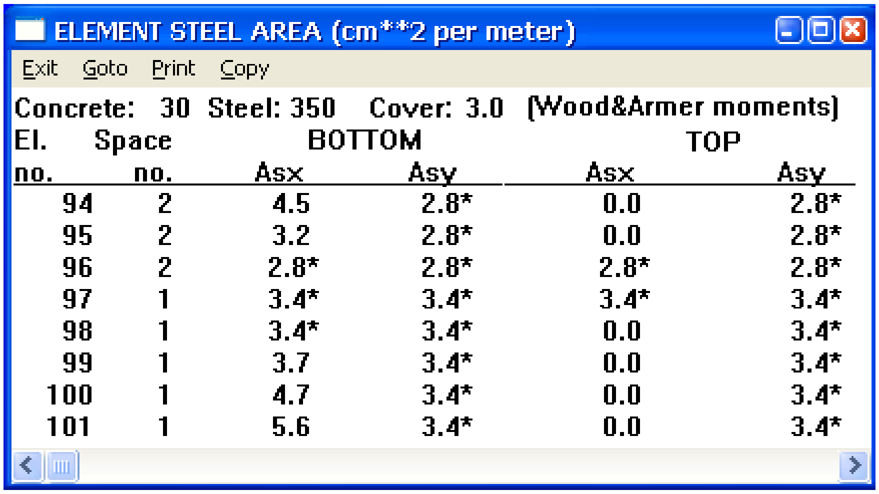
Note:

- the number of bars required (as displayed in Detailed results 948) is increased in the summary table in order to round off the spacing.
- "Start" and "End" area is the greater of width of the perpendicular wall or a seismic boundary (hinge) area. If there is no perp. wall or boundary area, the "Start" / "End" bars are those at the exterior face of the wall.

Capacity - Ratio of design capacity to the critical loads. Normally, the capacity is > 1.00. However if the maximum diameter and the minimum spacing does not provide an adequate solution, Capacity < 1.00

Note:
* - indicates that the section os inadequate (capacity < 1.00)
# - indicates that the reinforcement was defined by the user in "Specify bars"

8.18.1.4 Slabs



ELEMENT STEEL AREA (cm**2 per meter)

Exit Goto Print Copy

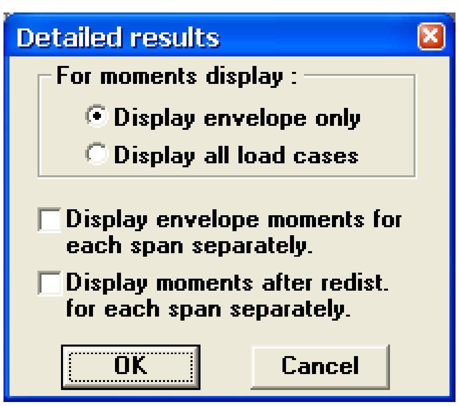
Concrete: 30 Steel: 350 Cover: 3.0 (Wood&Armer moments)

El. no.	Space no.	BOTTOM Asx	BOTTOM Asy	TOP Asx	TOP Asy
94	2	4.5	2.8*	0.0	2.8*
95	2	3.2	2.8*	0.0	2.8*
96	2	2.8*	2.8*	2.8*	2.8*
97	1	3.4*	3.4*	3.4*	3.4*
98	1	3.4*	3.4*	0.0	3.4*
99	1	3.7	3.4*	0.0	3.4*
100	1	4.7	3.4*	0.0	3.4*
101	1	5.6	3.4*	0.0	3.4*

where:
El no. = element number
Space no. = area in slab (bounded by 'dividing lines')
Asx, Asy = reinforcement area; ∗ indicates minimum reinforcement

8.18.2 Detailed results

8.18.2.1 Beam detailed results



Moments display

- **Display envelope only**
  To display only the moment/shear envelope created from all cases.
- **Display all load cases**
  To display the moment/shear diagrams for each of the individual load cases comprising the envelope.

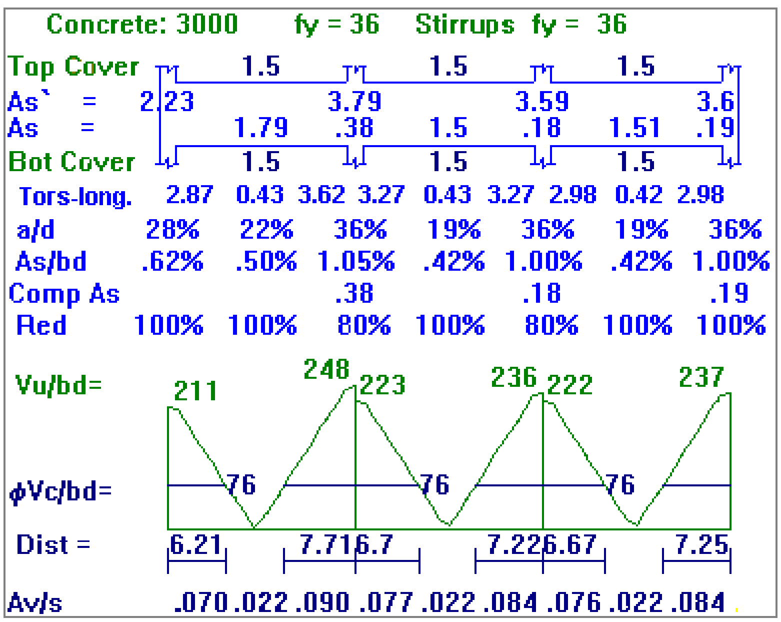
Each span

display the moment diagram for each span on a separate line.

The program displays the following screens (the display varies slightly for different Codes):
Reinforcement 932
Shear 933
Deflections 935

For beam detailed results of walls coupling beams. Refer to Coupling beams - detailed results 937

Reinforcement:



As`	=	Top reinforcement
As	=	Bottom reinforcement
Tors-long	=	The ***total*** amount of longitudinal torsion reinforcement required at the span ends and midspan (to be distributed around the entire perimeter of the web).
a/d	=	Height of the compression block, **a**, as a percentage of the depth of the beam, **d**.
As/bd	=	Reinforcement percentage used for calculating minimum reinforcement. The program uses **bw·d** for T and I sections with the flange in tension
Redist	=	the moment at a section after redistribution/the moment at the section before redistribution.
Vu/bd	=	Actual shear stress in the beam.
φVc/bd	=	Design shear stress in concrete.
Dist	=	The distance from the centre of the support to the point where **Vu = φVc**.

Note:
- the program checks minimum reinforcement areas. If the calculated area is less than the minimum

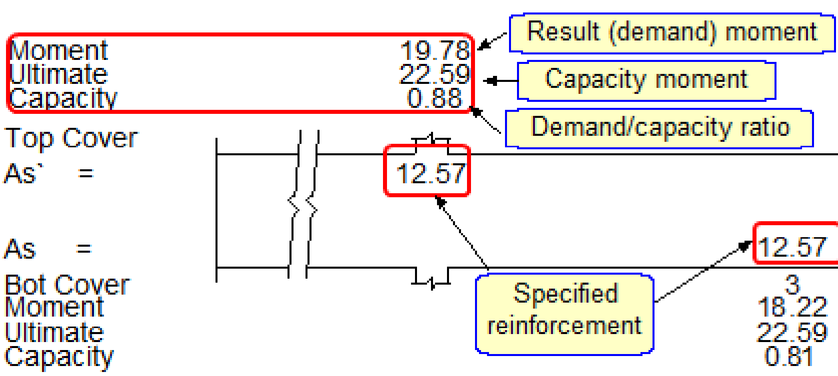
area, the minimum area is printed along with a "*".

- for seismic analysis, the program checks the maximum reinforcement percentage specified by the Code. If the percentage exceeds the limit, the calculated area is printed along with a "**&**".
- If the compression block exceeds the maximum allowed, the program calculates compression steel required. For example, where compression reinforcement is required for a negative moment:

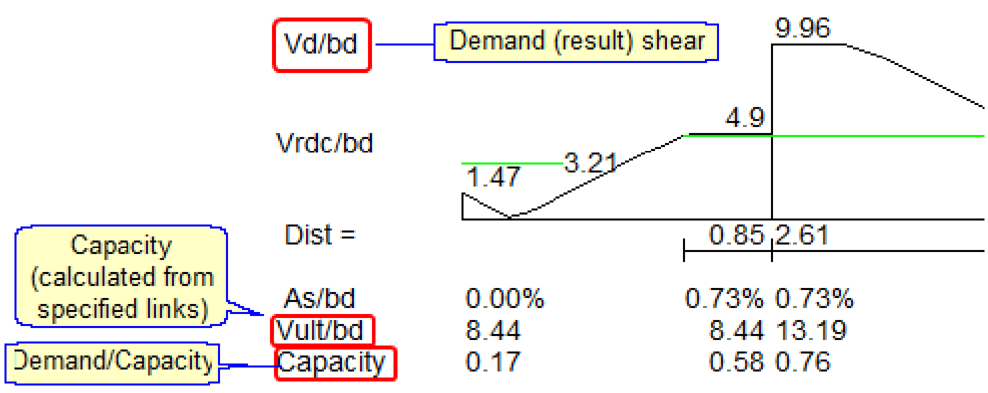
  As`   5.1
  As   +1.7
  Comp As   1.7

  1.7 in² is the compression steel required . Top reinforcement of 5.1 in² includes the 1.7 in² of compression reinforcement; Comp As = 1.7 in² is displayed for information only.
- An area may be displayed in parenthesis below/above the top/bottom steel; this area is the one calculated from the bending moment. The required area in such cases has been increased for either of two reasons:
  - the user specified a larger area using the "Specify reinforcement" option (Default or Parameters).
  - a larger area was automatically provided by the program according to seismic requirements, e.g. positive moment capacity at support must be greater than 50% of the negative moment capacity.

If the Specify reinforcement (capacity check) option was used for the beam, the program displays the capacity and the demand/capacity ratio. For example:



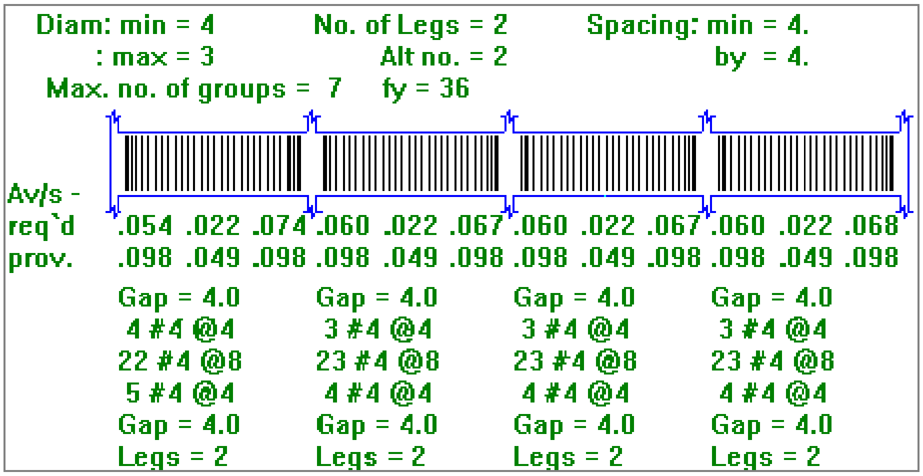
Similarly, if the links (stirrups) were specified:



A series of warnings is displayed if -

- The reinforcement percentage at a span or support exceeds 4%.
- The clear span is less than twice the effective depth.

Shear



where:

- ***Av/s - required***

Av/s = required transverse reinforcement area for shear and torsion, as specified by the code:
= **(Av/s)v + (Av/s)t**

For more information on the calculation, refer to:

BS8110	CSA A23.3
Eurocode 2	IS:456
ACI 318	NBr 6118

Note:

- The area will not be less than the minimum specified by the code for shear and torsion (derived from minimum area and maximum spacing requirements)
- The torsion area is always calculated for 2 legs only, even if more than 2 legs are provided. In such a case, the value of **Av/s** displayed in the table is:
  (Av/s)req'd = (Av/s)v + n (Av/s)t
  where n =(no. of legs/2)

- ***Av/s - provided***

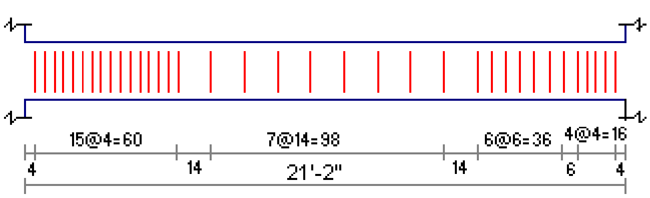
Av/s provided in the first and last link groups in the spans.The program selects links so that Av/s provided > Av/s required.

For example: the first link group is 8 mm bars (A = 50 mm²) at 125 mm spacing, 2 legs.
Av/s = 2 * 50 /125 = 0.8

- ***Link details***

"4#12..."	=	refers to the number of stirrups, not spaces.
"GAP"	=	distance from face of support to the first stirrup
• Space between groups	=	the larger spacing of the two groups

For example, check the following span:

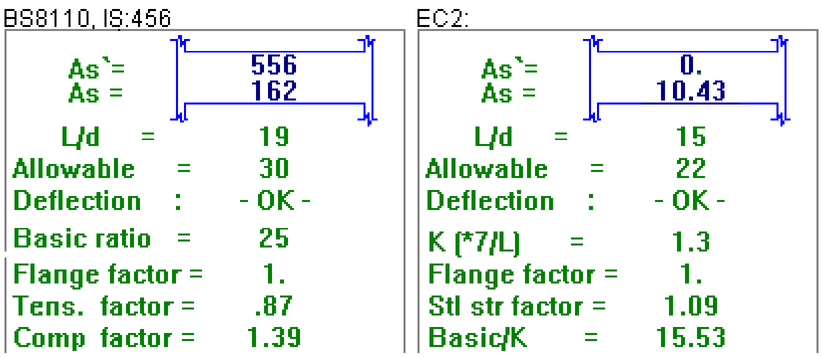


Gap =	=	4
1st group =	(16-1) * 4 =	60
space =	max(4,14) =	14

2nd group =	(8-1) * 14 =	98
space =	max(6,14) =	14
3rd group =	(7-1)*6 =	36
space =	max(6,8) =	8
4th group =	(5-1)*4 =	16
Gap =	=	4
		254

Note that in seismic design, many Codes specify the maximum distance to the first link (e.g. ACI318 = 2 in). It is generally not possible to adjust the number and/or spacing of the links so that the exact gap is achieved and in such cases the sum will exceed the net span length.

Deflections:



L/d	=	Actual span length-beam depth ratio.
Allowable	=	Allowable span-depth ratio as calculated according to the Code, based on the factors listed in this Table.
Deflection	=	**OK** - if **L/d** value is less than the **Allowable** value
Flange factor	=	Factor for flanged beams: BS8110: Code Table 3.10, EC 2: 7.4.2(2) IS:456: Fig.5

BS8110, IS:456

- Basic ratio	=	The basic span-depth ratio, based on end conditions: BS8110: Code table 3.10 IS:456: 22.2.1
- Tension factor	=	Modification factor for tension reinforcement: BS8110: Code Table 3.11, and calculated according to Equations 7 and 8, where βb is the redistribution ratio listed in the reinforcement results table. IS:456: Fig. 3
- Compr. factor	=	Modification factor for compression reinforcement: BS8110: Code table 3.12 IS:456: Fig. 4

EC2:

- K (* 7/L)	=	"K" value from Table 7.4N, reduced by (7/L) when L>7 m.
- Stl str factor	=	$500/f_{yk}$ $(1/\beta b)$ (Eq. 7.17) where βb = redistribution ratio = 'moment after' / 'moment before'
- Basic/K	=	$11 + 1.5\sqrt{f_{ck}}\,(\rho_o/\rho) + 3.2\sqrt{f_{ck}}\,(\rho_o/\rho - 1)^{3/2}$ if $\rho = \rho_o$ (7.16.a) $11 + 1.5\sqrt{f_{ck}}\,(\rho_o/\rho - \rho') + 1/12\sqrt{f_{ck}}\,(\rho'/\rho_o)^{1/2}$ if $\rho > \rho_o$ (7.16.b)

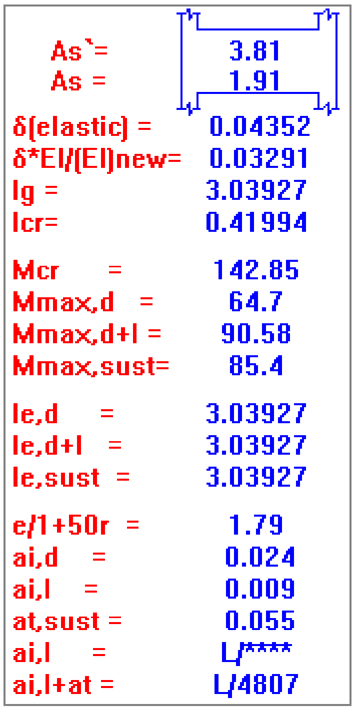
ρ_o = vf_{ck} /1000.

$\rho = A_s/b_w d$ for rectangular sections.

$\rho = A_s/[b_w d + (b_f - b_w)t_f]$ for T and L-sections.

The Code tables are based on limiting the total deflection to L/250 . This value cannot be revised. Refer to:

- BS8110: 3.4.6.3
- 456: 22.2a



A's = span top reinforcement, used to calculated the moments-of-inertia

As = maximum span bottom reinforcement, used to calculate the moments-of inertia.

δ(elastic) = elastic *STRAP* deflection, where:

- **Load cases -** maximum service deflection
- **Combinations -** maximum factored deflection

δ(new) = The program deflection calculations are based on the elastic deflections calculated by *STRAP*. These values are calculated from the section and material properties defined in *STRAP* geometry but which may have been modified in the concrete module. These values are the deflection modified according to the current section and material values

The program calculates the effective moment-of-inertia, **Ie**, according to the equation:

$$I_e = \left(\frac{M_{cr}}{M_{max}}\right)^3 I_g + \left[1 - \left(\frac{M_{cr}}{M_{max}}\right)^3\right] I_{cr} \le I_g$$

where:

Ig = moment-of-inertia of the gross concrete section, neglecting reinforcement

Icr = moment-of-inertia of the cracked transformed section

Mr = cracking moment of the section

Mmax = maximum service moment in member at stage deflection is calculated

The deflection is calculated for three load conditions, therefore:

Mmax,d maximum service moment due to dead loads

Mmax,d+l maximum service moment due to dead and live loads

Mmax,sust maximum service moment due to sustained loads

and the resulting effective moments-of-inertia are **Ie,d**, **Ie,d+l** and **Ie, sust**

e/(1+50r) = The additional long-term deflection factor:

ACI: $\lambda = \xi/(1+50\rho')$ (9-10)

CSA: $S/(1+50\rho')$ (9-5)

ai,d = immediate deflection due to dead loads

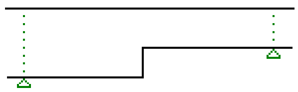
ai,l = immediate deflection due to live loads = **ai,d+l - ai,d**

at,sust = long-term deflection due to sustained loads

The latter two values must be checked according to the Code (ACI - Table 9.5, CSA- Table 9.2) and the values are also displayed in the **L/x** format.

Note:

- for span with variable sections (formed by two or more *STRAP* beam members):



The program calculates the deflections for ***each*** sub-span using the properties of the sub-span, but uses the ***total*** span length for 'L'. In the example above, the deflection calculated for the left sub-span is obviously too small while the deflection for the right sub-span is obviously too large. Since the empiric Code method does not deal with such cases, it is the user's responsibility to interpret the results and judge their relevance and applicability.

8.18.2.1.1 Coupling beams - detailed results

The program displays the following screens (the display varies slightly for different Codes):

Reinforcement 932
Shear 933
Deflections 935

Reinforcement:

Reinforcement: span 1

Concrete: B30 fsd= 435 fsd Links = 435 N/mm²
Seismic: High

Shear design:
Links only

Top Cover		4	
As` =	4.56	4.56	4.56
	(2.10)	(0.26)	(2.10)
	(2.10)	(2.10)	(2.10)
As =	4.56	4.56	4.56
Bot Cover		4	

* = min. reinforcement

x/d	10%	10%	10%
As/Ac	0.13%	0.13%	0.13%

As`	=	Top reinforcement
As	=	Bottom reinforcement
x/d	=	Height of the compression block, **x**, as a percentage of the depth of the beam, **d**.
As/Ac	=	Reinforcement percentage used for calculating minimum reinforcement.

Note:

- the program checks minimum reinforcement areas. If the calculated area is less than the minimum area, the minimum area is printed along with a "*****".
- for seismic analysis, the program checks the maximum reinforcement percentage specified by the Code. If the percentage exceeds the limit, the calculated area is printed along with a "**&**".
- If the compression block exceeds the maximum allowed, the program calculates compression steel required. For example, where compression reinforcement is required for a negative moment:

  As` 5.1
  As +1.7
  Comp As 1.7

  1.7 in² is the compression steel required . Top reinforcement of 5.1 in² includes the 1.7 in² of compression reinforcement; Comp As = 1.7 in² is displayed for information only.
- An area may be displayed in parenthesis below/above the top/bottom steel; this area is the one calculated from the bending moment. The required area in such cases has been increased for either of

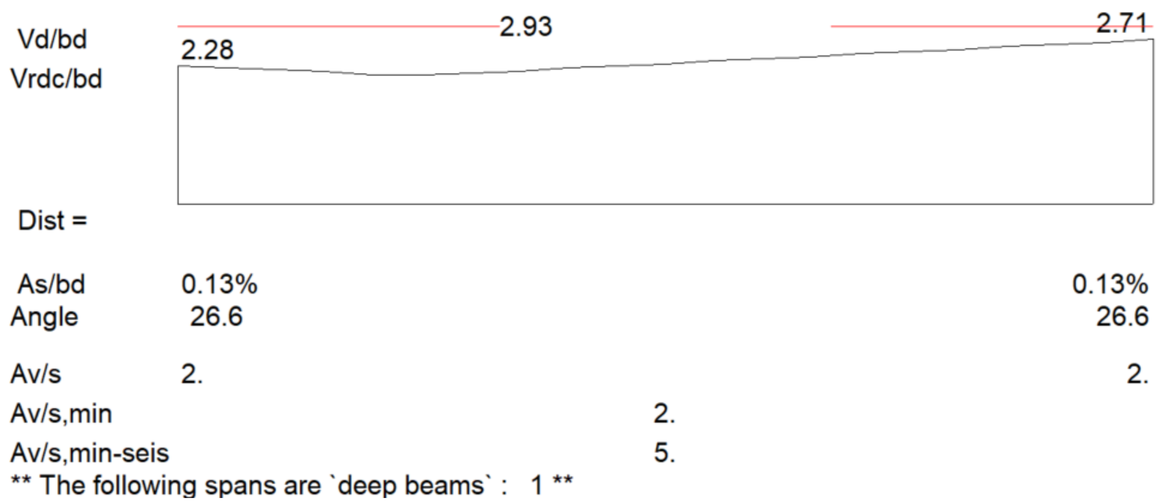
two reasons:

- a larger area was automatically provided by the program according to seismic requirements, e.g. positive moment capacity at support must be greater than 50% of the negative moment capacity.

A series of warnings is displayed if -

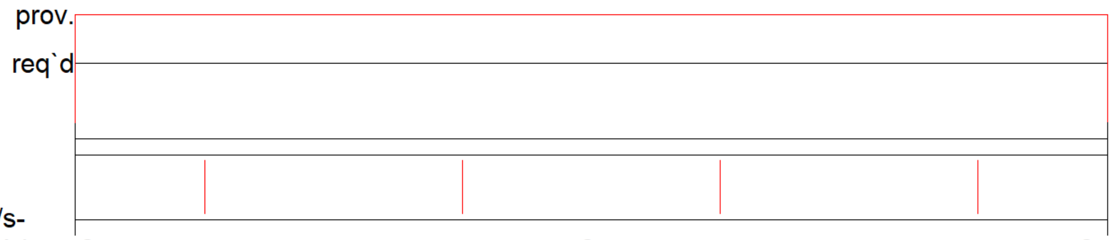
- The reinforcement percentage at a span or support exceeds 4%.
- The clear span is less than twice the effective depth.

Shear:



Vd/bd = Actual shear stress in the beam.
Vrdc/bd = Design shear stress in concrete.
Dist = The distance from the center of the support to the point where **Vd = Vrdc**.
As/bd = Shear reinforcement percentage.
Angle = The angle between the concrete compression strut and the beam axis perpendicular to the shear force.
Av/s = The cross-sectional area of the shear reinforcement **(Av)** divided by the spacing of the links.
Av/s, min = The minimum **Av/s** required by the code.
Av/s, min-seis = The minimum **Av/s** required by the code for seismic design.

Diam: min = 8 No. of Legs = code Spacing: min = 10.
: max = 16 increase if needed by = 5.
Max. no. of groups = 3 fsd = 435



Av/s- req`d	2.	2.	2.
prov.	3.4	3.4	3.4
angle	26.6		26.6

Gap = 15/15
Legs = 2
4ϕ8@30
Diagonal bars needed: As= 3.2 V=10.2 (angle=21)

where:

- ***Av/s - required***

  Av/s = required transverse reinforcement area for shear, as specified by the code: **(Av/s)v**

  For more information on the calculation, refer to:

  BS8110 CSA A23.3
  Eurocode 2 IS:456
  ACI 318 NBr 6118

  Note:

  - The area will not be less than the minimum specified by the code for shear (derived from minimum area and maximum spacing requirements).

- ***Av/s - provided***

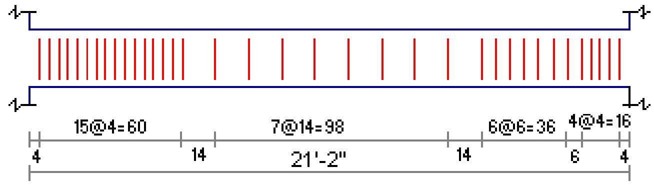
  Av/s provided in the first and last link groups in the spans.The program selects links so that Av/s provided > Av/s required.

  For example: the first link group is 8 mm bars (A = 50 mm²) at 125 mm spacing, 2 legs.
  Av/s = 2 * 50 /125 = 0.8

- ***Link details***

"GAP"	=	distance from face of support to the first stirrup
"Legs="	=	number of link legs
"4#12..."	=	refers to the number of stirrups, not spaces.
"Diagonals bars needed:	=	diagonal bars for seismic design. Based on code reuirements. "As=" is the diagonal reinforcement needed in each direction. "V=" is the shear force from seismic analysis for the calculation of the diagonal reinforcement. "(angle)=" is the diagonal reinforcement angle.

For example, check the following span:

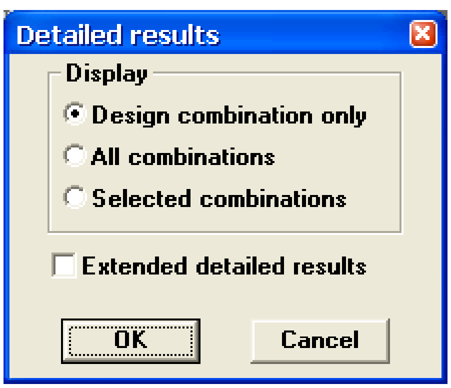


Gap =	=	4
1st group =	(16-1) * 4 =	60
space =	max(4,14) =	14
2nd group =	(8-1) * 14 =	98
space =	max(6,14) =	14
3rd group =	(7-1)*6 =	36
space =	max(6,8) =	8
4th group =	(5-1)*4 =	16
Gap =	=	4
		254

Note that in seismic design, many Codes specify the maximum distance to the first link (e.g. ACI318 = 2 in). It is generally not possible to adjust the number and/or spacing of the links so that the exact gap is achieved and in such cases the sum will exceed the net span length.

8.18.2.2 Column detailed results

Select the display options:



Design combination only
Display the results for the critical combination only.

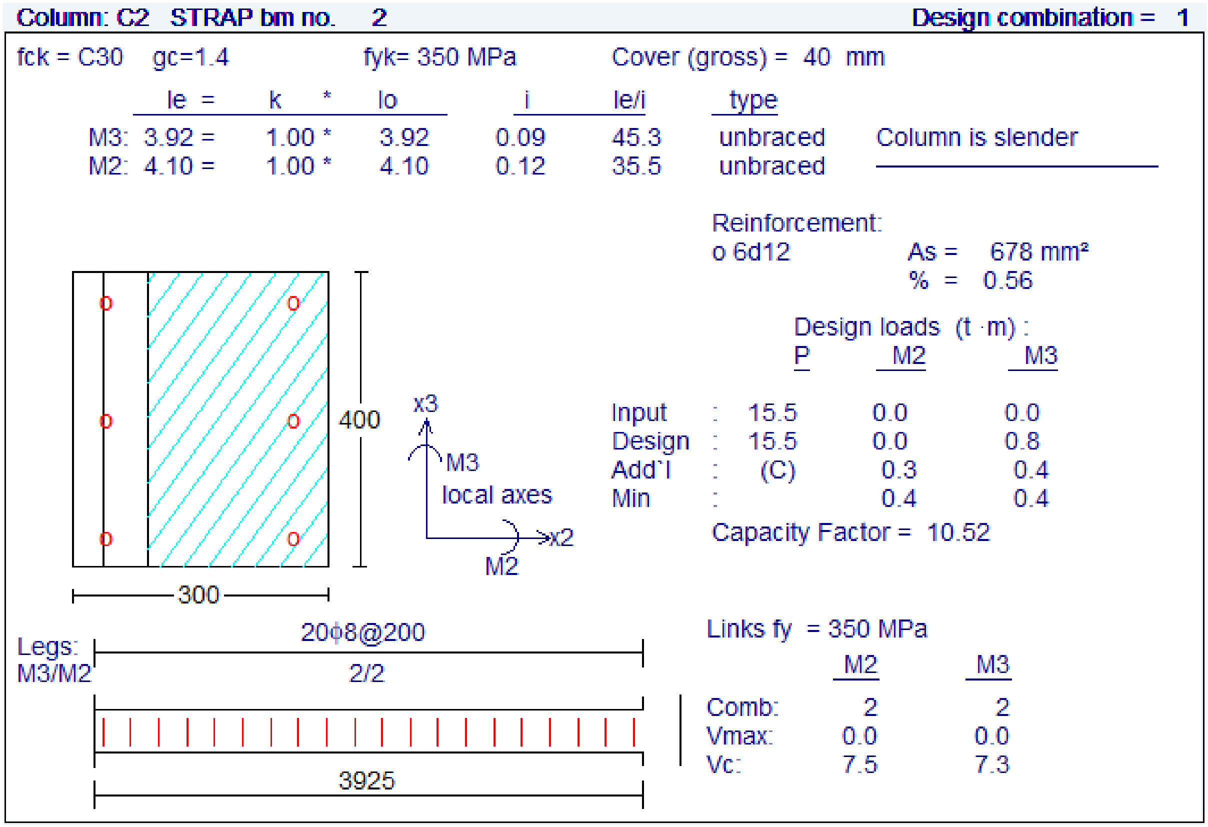
All combinations
Display the results for all combinations

Selected combinations
Select a combination from the list; only results for the selected combination are displayed.

Extended detailed results
Display additional results tables. Refer to:

- ACI318 942
- BS8110 / IS:456 944
- EC2 946

The following is an example of the detailed results summary (the display varies slightly for different Codes):



Design combination

The combination with the smallest capacity factor (or the combination where the calculation stopped, if reinforcement percentage exceeds the maximum allowed).

Length data

Note that the symbols vary according to the Code:

le	=	effective height of the column/wall in the plane of bending considered.
k, β	=	effective length factor.
lu, lo	=	clear height between end restraints.
r	=	radius-of-inertia.
h	=	column/wall dimension in relevant direction
le/r, le/h	=	column slenderness.

Reinforcement data

The number of bars and diameters, the total area and reinforcement percentage are displayed.

Warnings are displayed if:
- the percentage exceeds the allowable.
- the spacing is less than the optimal.
- a reduced area was used for lightly loaded columns.

Section

The section orientation is displayed relative to the local axis orientation displayed at the right of the

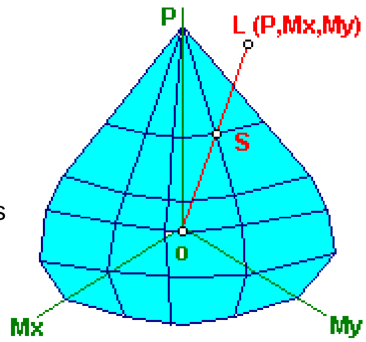
screen.
The concrete compression area is hatched. The neutral axis is drawn parallel to the compression area block.

Capacity factor

Load Capacity :
Ratio of the column capacity to the design loads
= OS/OL

Ratio > 1.00, the column is adequate. However, if the program is unable to place sufficient reinforcement in the section due to spacing or percentage limitation, then the capacity factor will be less than 1.00.

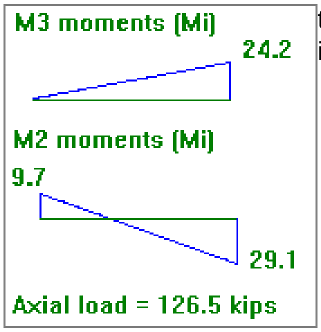
Note that the factor represents a vector ratio on the column capacity diagram.



Refer also to Extended results:
- ACI318 942
- BS8110 944
- EC2 946

8.18.2.2.1 Extended results - ACI318

- ***Moment diagrams:***



the moments displayed are the *STRAP* elastic analysis results and do ***not*** include the additional moments.

- ***Magnified Moments:*** (§ 10.11.5)

  The moment magnifier calculation is displayed for slender columns only. For example:

Magnified Moments: (§ 10.11.5)						
	M2	M3		Mc =	δ·M2	(10-6)
Pu =	342.2	342.2	kip		1	
Pc =	2909.0	3560.8	kip	δ =	――――	
klu =	8.0	8.0	ft		1-Pu/φPc	
Ig =	2304.0	4096.0	in		π²EI	
Ise =	36.5	58.8	in	Pc =	――――	
βd =	.000	.400			(klu)²	
δ =	1.202	1.159		EI =	(0.2EcIg+EsIse)/(1+βd)	

Note:

- the moment magnifiers are always calculated for both directions, unless the user specified a moment magnifier in the "Parameters" option.
- all symbols used are as defined in the Code.
- refer to Design assumptions for a detailed explanation of the assumptions made by the program when calculating the moment magnifier.

- ***Design Loads:***

Design loads:

Location	Pu	Mu2	Mu3	δMu2	δMu3	(*=design case)
Bottom	342.2	.0	.0	27.4	.0	
				-27.4	.0	
				.0	-30.8	
				.0	30.8	
#Middle	342.2	.0	-40.3	.0	-46.7	
Top	342.2	.0	-67.1	.0	-67.1	*

where:

Pu, Mu2, Mu3	=	*STRAP* result moments and axial force
dMu2, dMu3	=	Design moments (magnified/minimum moments)
Top/Middle/Bottom	=	Three design locations as required by the Code. For method used by the program to generate **dMu** at each location, refer to Design assumptions.

- ***Minimum Moments:*** (for slender columns only)

Minimum Moments = P·emin

P	emin2	M2min	emin3	M3min
342.2	.080	27.4	.090	30.8

where **emin2** and **emin3** are calculated from **(0.6 + 0.03h)** inch

- ***Equilibrium Check:***

The final table in the detailed results provides proof that the section is in equilibrium, i.e.

Σ**F = 0** and Σ**M = 0**

As the section usually is slightly over-designed, the equations can be written:

Σ**Pi >** Σ**Pe** and Σ**Mi >** Σ**Me**

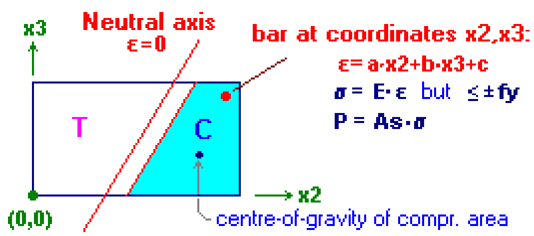
where the subscripts "i" and "e" refer to the internal and external forces and moments, respectively.

Moments are calculated separately about the x2 and x3 axes (x and y), about the lower-left corner of rectangular of L-sections, or the centre of circular sections.

Reinforcement:

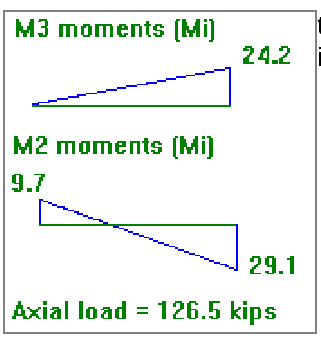
Bar	Group	ϕ	ε	σ(kg/cm²)	P (t)	x2(m)	P·x2(t·m)	x3(m)	P·x3(t·m)
1	1	12	-.00359	-4000.0	-4.5	.03	-.1	.03	-.1
2	1	12	.00253	4000.0	4.5	.22	1.0	.03	.1
3	1	12	.00253	4000.0	4.5	.22	1.0	.42	1.9
4	1	12	-.00359	-4000.0	-4.5	.03	-.1	.42	-1.9
5	2	0	.00253	4000.0	.0	.22	.0	.22	.0
6	3	12	-.00359	-4000.0	-4.5	.03	-.1	.22	-1.0
Concrete:					58.7	.20	11.8	.22	13.2
Compr. Area (90%) = 43991. mm²				Σ =	58.7		14.4		13.2
External Loads:				P :	46.6	.125	5.8	.225	10.5
				M3:			5.6		
				M2:					.0
				Σ =	46.6		11.4		10.5

Load Capacity Factor = 1.27



8.18.2.2.2 Extended results - BS8110/IS:456

- ***Moment diagrams:***



the moments displayed are the *STRAP* elastic analysis results and do ***not*** include the additional moments.

- ***Additional Moments:*** (BS8110-§ 3.8.3, IS:456-§ 38.7)

The additional moment calculation is displayed for slender columns only. For example:

Additional Moments: (§ 3.8.3.1)

		M2	M3	
h	=	.450	.250	m
N	=	62.8	62.8	ton
Nuz	=	229.7	229.7	ton
Nbal	=	89.1	75.6	ton
le	=	5.70	2.85	m
βa	=	.080	.065	
K	=	1.000	1.000	
Madd	=	2.27	1.02	kN·m

$$Madd = N \cdot au \qquad \text{(eq. 35)}$$

$$au = \beta a \cdot K \cdot h \qquad \text{(eq. 32)}$$

$$K = \frac{Nuz - N}{Nuz - Nbal} \qquad \text{(eq. 33)}$$

$$\beta a = \frac{1}{2000} \cdot \left(\frac{le}{h}\right)^2 \qquad \text{(eq. 34)}$$

}

Note:

- the additional moments are always calculated for both directions, unless the user specified a moment magnifier in the "Parameters" option.
- all symbols used are as defined in the Code.
- refer to Design assumptions for a detailed explanation of the assumptions made by the program when calculating the moment magnifier.

- ***Design Loads:***

Design loads:

Location	P	M2i	M3i	M2t	M3t	(*=design case)
Bottom	46.6	.0	-4.5	.0	-4.9	
				-.9	.0	
				.9	.0	
#Middle	46.6	.0	2.2	.0	2.9	
Top	46.6	.0	5.6	.0	5.6	*

P, M2i, M3i	=	*STRAP* result moments and axial force
M2t, M3t	=	Design moments (initial moments + additional moments)
Top/Middle/Bottom	=	Three design locations as required by the Code. For method used by the program to generate **Mt** at each location, refer to Design assumptions.

- ***Minimum Moments***: (for slender columns only)

Minimum Moments = P·emin

P	emin2	M2min	emin3	M3min
342.2	.080	27.4	.090	30.8

where **emin2** and **emin3** are calculated from: **0.05h** but not more than 20 mm.

- ***Equilibrium Check:***

The final table in the detailed results provides proof that the section is in equilibrium, i.e.

$\Sigma \mathbf{F} = \mathbf{0}$ and $\Sigma \mathbf{M} = \mathbf{0}$

As the section usually is slightly over-designed, the equations can be written:

$\Sigma \mathbf{Pi} > \Sigma \mathbf{Pe}$ and $\Sigma \mathbf{Mi} > \Sigma \mathbf{Me}$

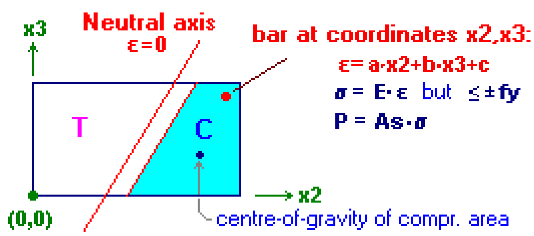
where the subscripts "i" and "e" refer to the internal and external forces and moments, respectively.

Moments are calculated separately about the x2 and x3 axes (x and y), about the lower-left corner of rectangular of L-sections, or the centre of circular sections.

Reinforcement:

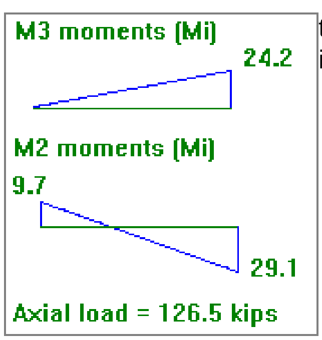
Bar	Group	φ	ε	σ(kg/cm²)	P (t)	x2(m)	P·x2(t·m)	x3(m)	P·x3(t·m)
1	1	12	-.00359	-4000.0	-4.5	.03	-.1	.03	-.1
2	1	12	.00253	4000.0	4.5	.22	1.0	.03	.1
3	1	12	.00253	4000.0	4.5	.22	1.0	.42	1.9
4	1	12	-.00359	-4000.0	-4.5	.03	-.1	.42	-1.9
5	2	0	.00253	4000.0	.0	.22	.0	.22	.0
6	3	12	-.00359	-4000.0	-4.5	.03	-.1	.22	-1.0
Concrete:					58.7	.20	11.8	.22	13.2
Compr. Area (90%) = 43991. mm²				Σ =	58.7		14.4		13.2
External Loads:				P :	46.6	.125	5.8	.225	10.5
				M3:			5.6		
				M2:					.0
				Σ =	46.6		11.4		10.5

Load Capacity Factor = 1.27



8.18.2.2.3 Extended results - EC2

- ***Moment diagrams:***



the moments displayed are the *STRAP* elastic analysis results and do ***not*** include the additional moments.

- **Additional Moments**: (§ 3.8.3)

  The additional moment calculation is displayed for slender columns only. For example:

Additional Moments:	M2	M3			
d =	.420	.220	m	Madd =	N·(e2+ea)
le/i =	46.188	41.569			
k1 =	1.000	1.000		e2 =	k1·le²/10·(1/r)
εyd =	.002	.002		k1 =	(le/i)/20 - 0.75
1/r =	.011	.020			
Nsd =	59.8	59.8	ton	1/r =	2·k2·εyd/0.9·d
Nud =	239.4	239.4	ton		
Nbal =	85.2	75.6	ton		Nud - Nsd
lo =	6.00	3.00	m	k2 =	————
l =	12.00	12.00	m		Nud - Nbal
ea =	.015	.004	m	ea =	v·lo/2
e2 =	.038	.018	m	v =	1/(100√l)
Madd =	3.2	1.3	kN·m		

Note:

- the additional moments are always calculated for both directions, unless the user specified a moment magnifier in the "Parameters" option.
- all symbols used are as defined in the Code.
- refer to Design assumptions for a detailed explanation of the assumptions made by the program when calculating the moment magnifier.

- ***Design Loads:***

Design loads:

Location	P	M2i	M3i	M2t	M3t	(*=design case)
Bottom	46.6	.0	-4.5	.0	-4.9	
				-.9	.0	
				.9	.0	
#Middle	46.6	.0	2.2	.0	2.9	
Top	46.6	.0	5.6	.0	5.6	*

where:

P, M2i, M3i	=	*STRAP* result moments and axial force
M2t, M3t	=	Design moments (initial moments + additional moments)
Top/Middle/Bottom	=	Three design locations as required by the Code. For method used by the program to generate **Mt** at each location, refer to Design assumptions.

- ***Minimum Moments:*** (for slender columns only)

Minimum Moments = P·emin

P	emin2	M2min	emin3	M3min
342.2	.080	27.4	.090	30.8

where **emin2** and **emin3** are calculated from: **0.05h** but not more than 20 mm.

- ***Equilibrium Check:***

The final table in the detailed results provides proof that the section is in equilibrium, i.e.

ΣF = 0 and **ΣM = 0**

As the section usually is slightly over-designed, the equations can be written:

$\Sigma Pi > \Sigma Pe$ and $\Sigma Mi > \Sigma Me$

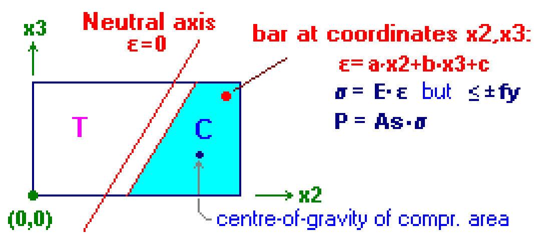
where the subscripts "i" and "e" refer to the internal and external forces and moments, respectively.

Moments are calculated separately about the x2 and x3 axes (x and y), about the lower-left corner of rectangular of L-sections, or the centre of circular sections.

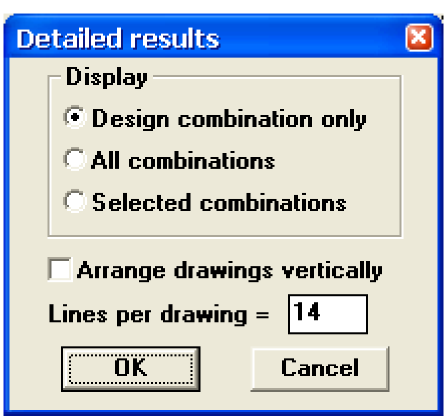
Reinforcement:

Bar	Group	ϕ	ε	σ(kg/cm²)	P (t)	x2(m)	P·x2(t·m)	x3(m)	P·x3(t·m)
1	1	12	-.00359	-4000.0	-4.5	.03	-.1	.03	-.1
2	1	12	.00253	4000.0	4.5	.22	1.0	.03	.1
3	1	12	.00253	4000.0	4.5	.22	1.0	.42	1.9
4	1	12	-.00359	-4000.0	-4.5	.03	-.1	.42	-1.9
5	2	0	.00253	4000.0	.0	.22	.0	.22	.0
6	3	12	-.00359	-4000.0	-4.5	.03	-.1	.22	-1.0
Concrete:					58.7	.20	11.8	.22	13.2
Compr. Area (90%) = 43991. mm²				Σ =	58.7		14.4		13.2
External Loads:				P :	46.6	.125	5.8	.225	10.5
				M3:			5.6		
				M2:					.0
				Σ =	46.6		11.4		10.5

Load Capacity Factor = 1.27



8.18.2.3 Wall detailed results

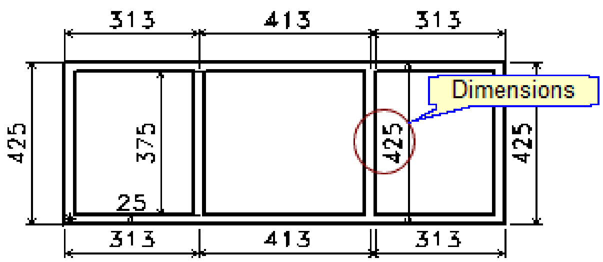


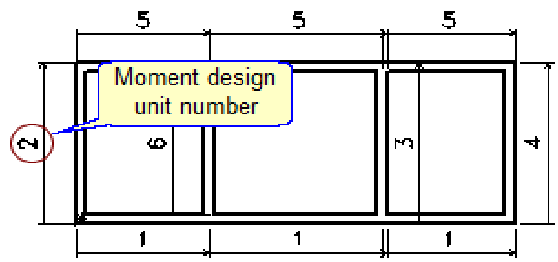
Design combination only
Display the results for the critical combination only.

All combinations
Display the results for all combinations

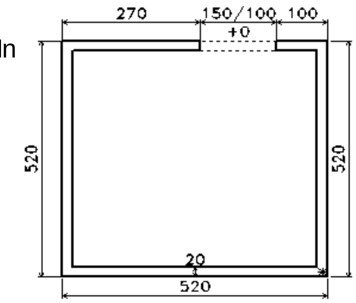
Selected combinations
Select a combination from the list; only results for the selected combination are displayed.

The program first displays the wall section twice - one drawing with the dimensions and the other with the moment design unit numbers   For example:





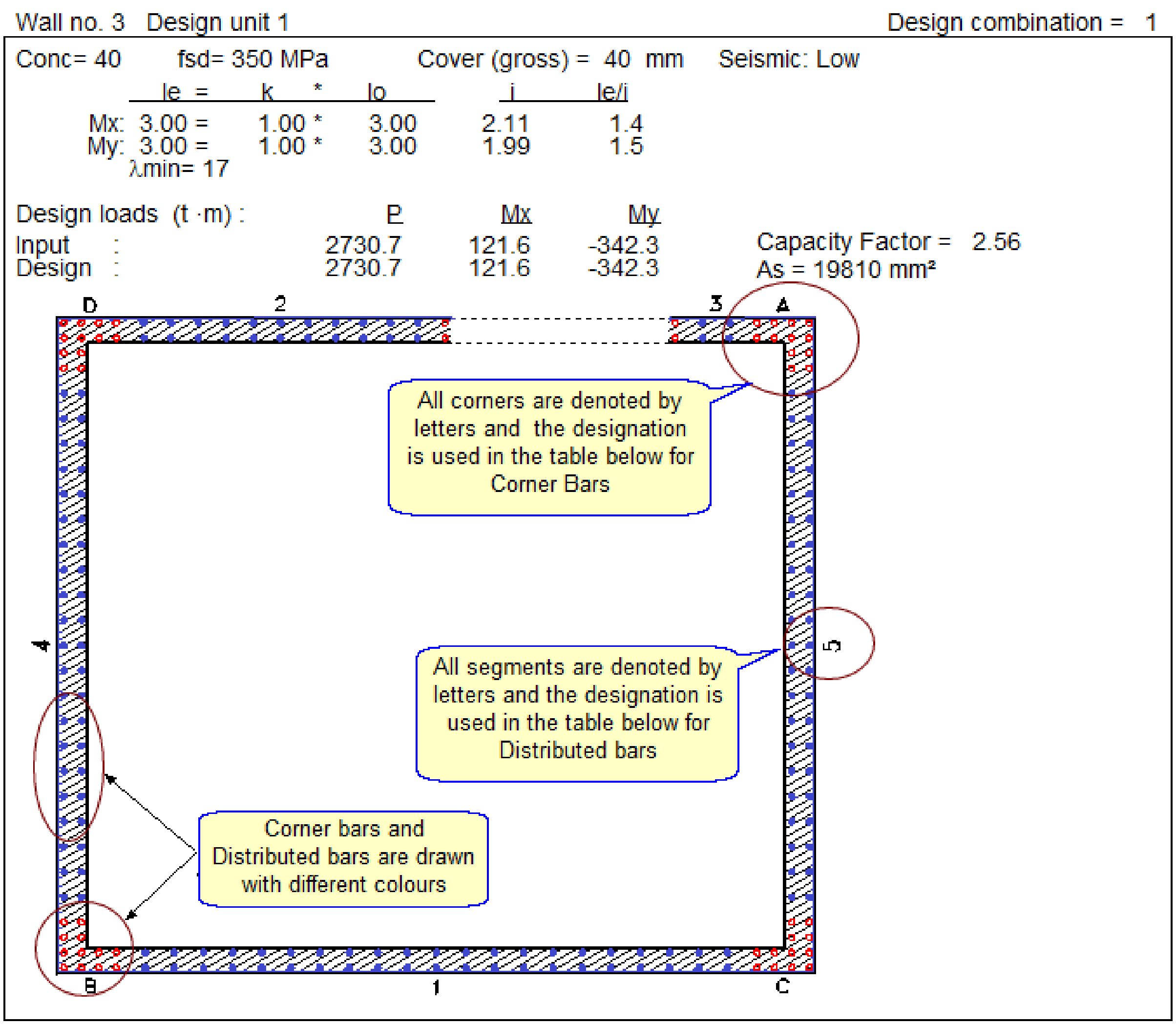
The second drawing is not displayed if the entire wall section is one moment design unit, the normal case for small walls. In this example, the pier is one moment design unit while each segment is a separate shear design unit:



Note:
- these drawings can be printed ***to any scale*** (on more than one page) using the Print wall section drawing 952 option.

The results are displayed separately for each design unit in the wall section.

The following is an example of the detailed results summary (the display varies slightly for different Codes):



Design combination

The combination with the smallest capacity factor (or the combination where the calculation stopped, if reinforcement percentage exceeds the maximum allowed).

Length data

Note that the symbols vary according to the Code:

le	=	effective height of the wall in the plane of bending considered.
k, β	=	effective length factor.
lu, lo	=	clear height between end restraints.
h , **r**	=	wall thickness / radius-of-inertia
le/r, le/h	=	wall slenderness.

Reinforcement data

The number of bars and diameters, the total area and reinforcement percentage are displayed.

Note that the quantity displayed is the quantity required; the number of bars in the result summary table 930 may be different as it is adjusted in order to round off the spacing.

Warnings are displayed if:
- the percentage exceeds the allowable.
- the spacing is less than the optimal.
- a reduced area was used for lightly loaded columns.

Refer also to Walls - specify.

Design loads

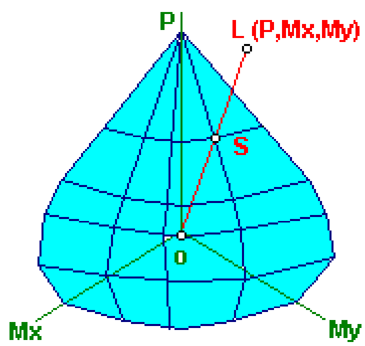
The input loads and the design loads for the design combination.
- input loads - from *STRAP* results
- design loads - includes additional moments for slender walls and minimum moments where required.

Capacity factor

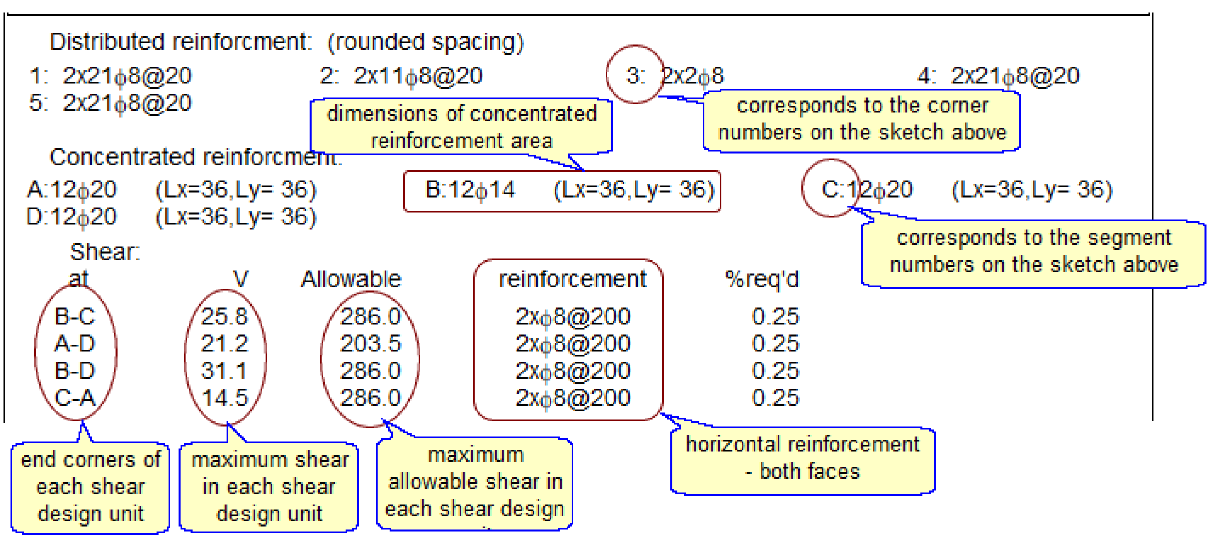
Load Capacity :
Ratio of the wall capacity to the design loads
= OS/OL

Ratio > 1.00, the wall is adequate.
The factor will be less than 1.00 if the program is unable to place sufficient reinforcement in the section due to spacing or percentage limitation.

Note that the factor represents a vector ratio on the wall capacity diagram.



The program then displays the reinforcement table. For the pier example above:



Note:
- the program can enlarge Lx,Ly beyond that required by the Code if a greater dimension gives a more economical result.
- the segment numbering refers only to the diagram above and may be different from the segment numbering in Geometry.

For the following tables, refer to Column detailed results 940

- Additional moments
- Design loads
- Equilibrium check

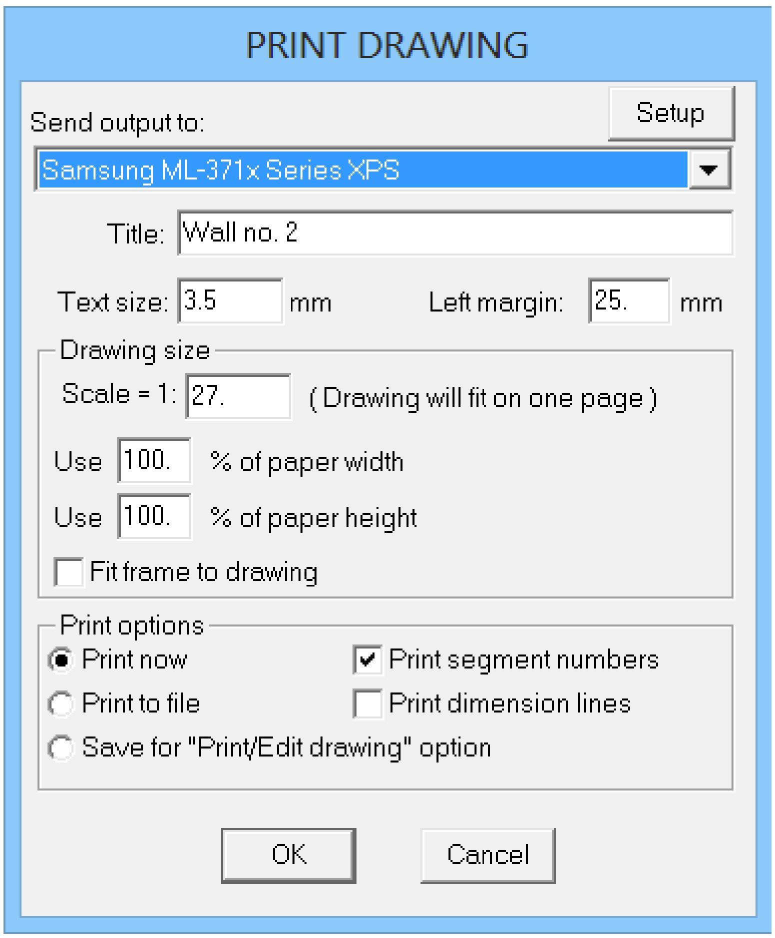
8.18.2.4 Slab detailed results

The program displays for each level and each space:
- Top/bottom/total steel weight
- Concrete area and volume
- Steel weight/area and weight/volume ratios

The totals for the model are also displayed.

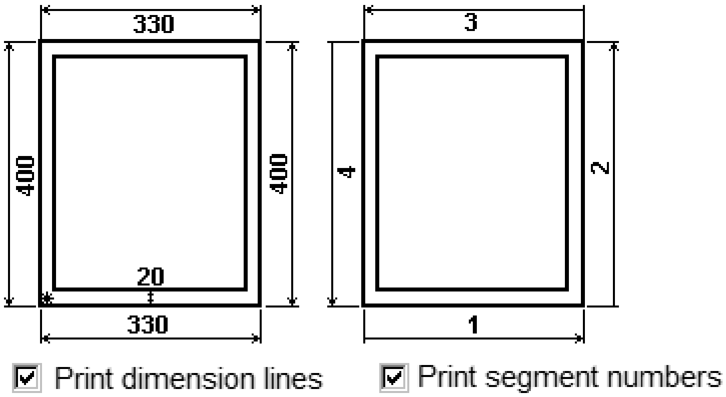
8.18.2.5 Print wall section drawing

Print the wall section drawings (shown at the top of the Detailed results 948) ***to any scale***.



Print segment numbers
Print dimension lines

Select the wall drawings to print:

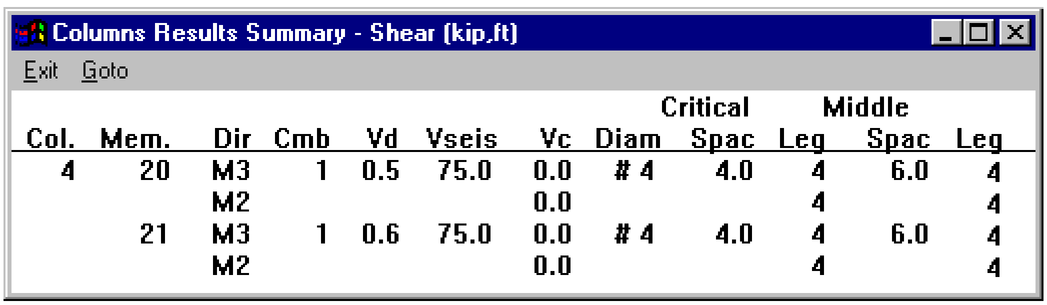


All other options

Refer to Print drawing.

8.18.3 Shear

Display the shear and tie (hoop/spiral) data for the computed columns: For example, a column with seismic design according to ACI 318 (the format varies for plane/space models, models without seismic design and for other codes):



Columns Results Summary - Shear (kip,ft)

Exit Goto

Col.	Mem.	Dir	Cmb	Vd	Vseis	Vc	Critical Diam	Critical Spac	Critical Leg	Middle Spac	Middle Leg
4	20	M3	1	0.5	75.0	0.0	# 4	4.0	4	6.0	4
		M2				0.0			4		4
	21	M3	1	0.6	75.0	0.0	# 4	4.0	4	6.0	4
		M2				0.0			4		4

where:

Comb	=	combination with maximum design shear, i.e. **(V - Vc)max**, where **V = max(Vd , Vseis).**
Vd	=	maximum factored shear in the column (combination "Cmb"), from *STRAP* results
Vseis	=	the design shear calculated from the moment capacities at the ends of the column, calculated from the probable moment strength (**Mpr**) of the beams framing into the column. Refer to Seismic - general 787.
Vc	=	nominal shear strength provided by the concrete considering the axial force in the column.
Critical	=	Link (stirrup) details in the critical (hinge) area adjacent to the column ends
Middle	=	Link (stirrup) details in the middle of the column

8.18.4 Specify reinforcement

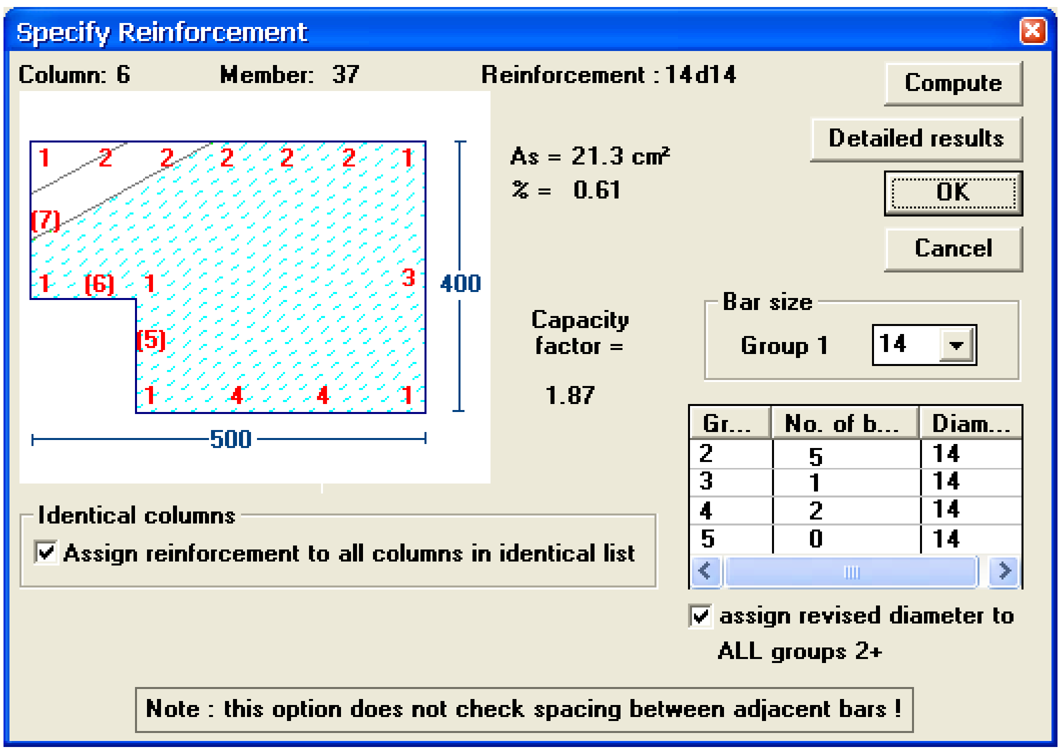
Revise the diameter or number of bars in any group in a column or wall; the program automatically recalculates the capacity for the new arrangement.

Refer to :
Specify reinforcement - columns 954
Specify reinforcement - walls

8.18.4.1 Specify - columns

Revise the diameter or number of bars in any group; the program automatically recalculates the column capacity for the new arrangement.

- the diameters selected do not have to conform to the current min/max diameter parameters.
- the program does ***not*** check spacing for the number of bars selected; it is the user's responsibility to ensure that the spacing conforms to Code requirements.
- the number and location of groups for the current section shape cannot be changed; to create a different group pattern:
  - return to *STRAP* geometry, define the identical section in the *CROSEC* section generator and assign the new property to the relevant members.
  - solve the model and return to the Concrete design module
  - select **Properties** and [Edit STRAP solid section]; define new corner bars and add new groups.
  - Compute the column; use this option to modify the reinforcement.



Note that groups without bars are represented by a single bar in parentheses (groups 5,6,7 in the above example).

Compute

Recalculate the section capacity with the new arrangement. Note that the program does not check minimum reinforcement requirements, but displays a warning if they are exceeded.

Detailed results

Display detailed results 940 for the current specified reinforcement

OK

Accept the new reinforcement

Bar size

Revise the bar sizes for Group 1.

Groups - Diameter & number of bars

Revise the diameter and/or number of bars for ***each line*** in the group. The number of bars in Group 1 (corner bars) cannot be revised (except in round sections).

Assign revised diameter to ALL groups 2+

- [x] the diameter for all groups in the table is automatically revised when the diameter of any one of the groups is revised.
- [ ] specify a different the diameter for each group in the table.

Identical columns

If the selected column is "identical" to other column segments:

Assign reinforcement to all other columns in identical list

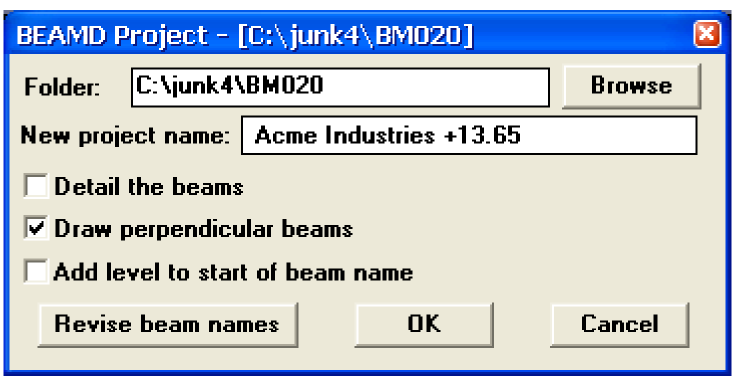
- [x] The reinforcement specified for this column are assigned to ***all*** other columns in the identical list
- [ ] Only the selected column segment will have the specified reinforcement. Note that the column will still be in the "identical list", even the reinforcement is no longer be identical.

Note:

- groups without bars are represented by a single bar in parentheses (group 5 in the above example).
- 'Zoom' and 'Pan' options are available.

8.18.5 BEAMD files

The *BEAMD* program is a software package for the design & detailing of reinforced concrete beams. This option creates for each selected beam a detailing file that can be read by *BEAMD*. Please contact your *STRAP* dealer for more information.



Folder

The detailing files are created by default in the sub-folder **\BMnnn** of the current working folder, where **nnn** = the *STRAP* model number. Type in a different folder name or click Browse.

Project

For convenience, the files may be grouped together in "projects". Up to 10 projects may be created in any folder. Beams may be added to an existing project or a new project may be created: select an existing project in the "Project" list box, or select "**Open a new project**" (in the list box) and type in the project name in the "**New project name**" edit box.

Detail

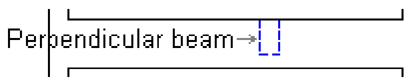
There are two options:

☐ **Detail the beams**
The program only creates the files; the actual detailing is done in the *BEAMD* program.

☑ **Detail the beams**
The program creates the files and carries out the detailing; the user specifies the detailing parameters for the first beam and the program uses these parameters for all of the beams.

Perpendicular beams

Perpendicular beams may be added to the beam drawing. For example:



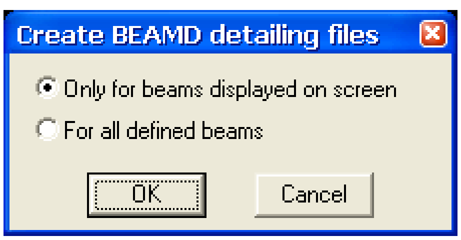
Add level

The level may be added to the start of the beam title (if not already included in the default title).

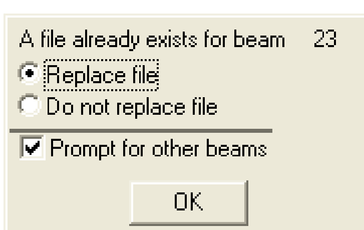
Revise beam names

Beam names are automatically created along with the beams. Refer to Define - beam names 839. Use this option to revise the current names.

Specify the model beams for which the detailing files are to be generated:



The program checks whether a file for the beam already exists in the directory (by comparing the member "lists"). If a file is found for the beam:



Select:

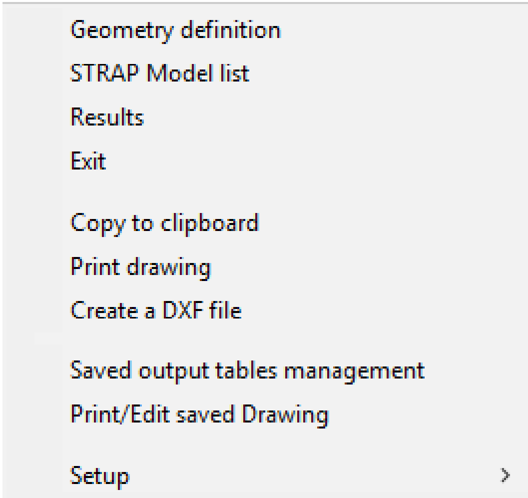
◉ **Replace file**
Overwrite the existing file

◉ **Do not replace file**
Do not create a new file

Note:

- Up to 3999 beams may be saved in each folder
- The files created by this option are:
  - **BMPROJ.ATR** - an index of the beams and projects in the directory
  - **Bnnn.BMD** - the file (for each beam) containing the detailing data, e.g. B001.BMD, B002.BMD, …., B999.BMD and C000.BMD to C999.BMD (for beams 1000-1999).

8.19 File options



Geometry

Exit the concrete design module and return to the Geometry definition for the same model.

STRAP model list

Exit the concrete processor and return to the ***STRAP*** main menu to select another model.

Results

Display the results for the current model.

Exit

Exit the concrete module and ***STRAP***. All data defined in the module is saved.

Copy to clipboard

Copy the current graphic display to the clipboard.

Print drawing

Print the current graphic display. Refer to Print drawing.

Create a DXF drawing

Refer to DXF drawing 959

Saved output tables management

Delete tables, rearrange the order of tables or display/print tables that were saved using the **Save output for report generation** option. Refer to Saved tables management,

Print/edit a saved drawing

Refer to Print/edit a saved drawing.

Setup

Refer to Concrete - setup 960.

8.19.1 DXF drawing

Create a DXF file from the current drawing or display:

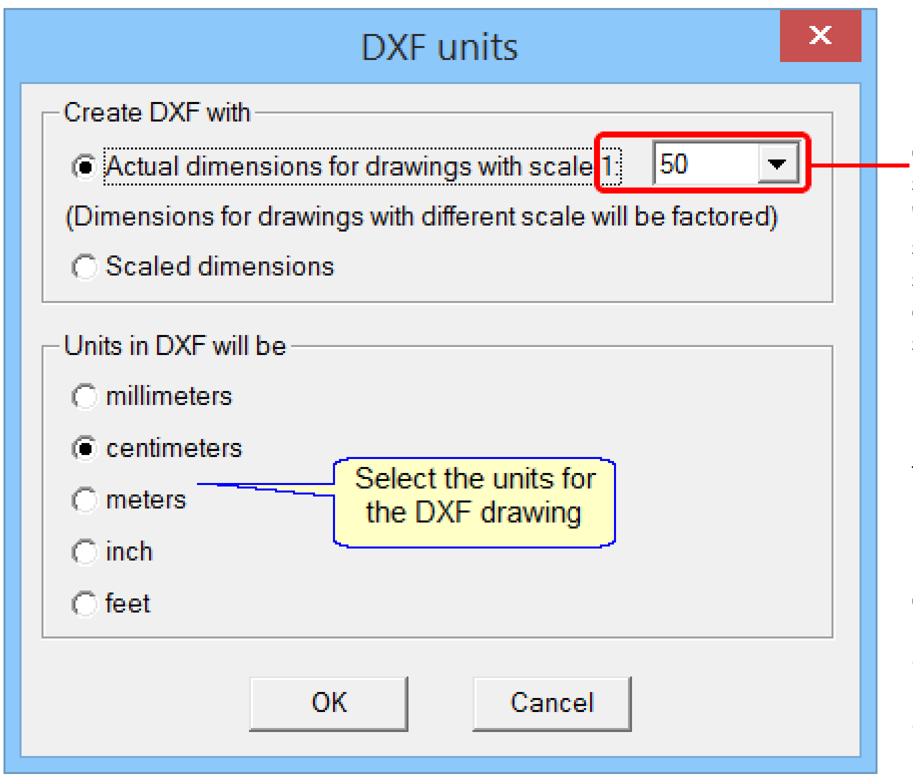
Display

Refer to Print - DXF 94

Drawing

The drawing can be transferred with "actual dimensions" or "scaled dimensions". For example, a line 1.0 meter long on a drawing with a scale of 1:50 and "Units in DXF" = mm will be transferred to the DXF drawing as:

- actual dimensions: 1.0 x 1000 = 1000 units
- scaled dimensions: 1.0 x 1000 / 50 = 20 units

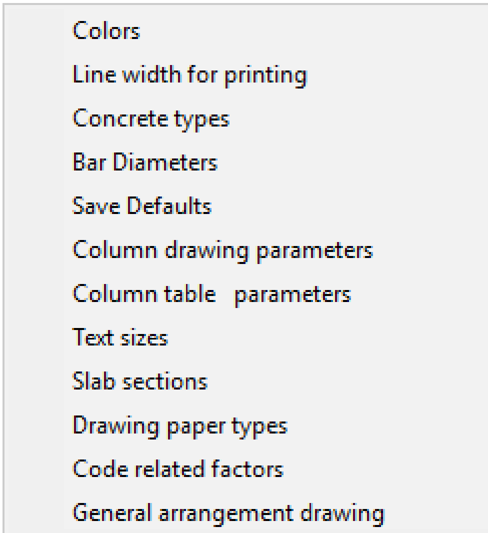


If there are objects with different scales on the same drawing and "actual dimensions" are selected, one of the scales will be used for ***all*** of the objects. Select the scale from the list.

For example, a 1:50 floor plan and a 1:20 are on the same drawing. "1:50" is selected from the list. Lines 1.0 meter long will be transferred to the DXF drawing as follows ("Units in DXF" = mm):

- 1:50 floor plan: 1.0 x 1000 = 1000 units
- 1:20 section: 1.0 x 1000 x 50/20 = 2500 units

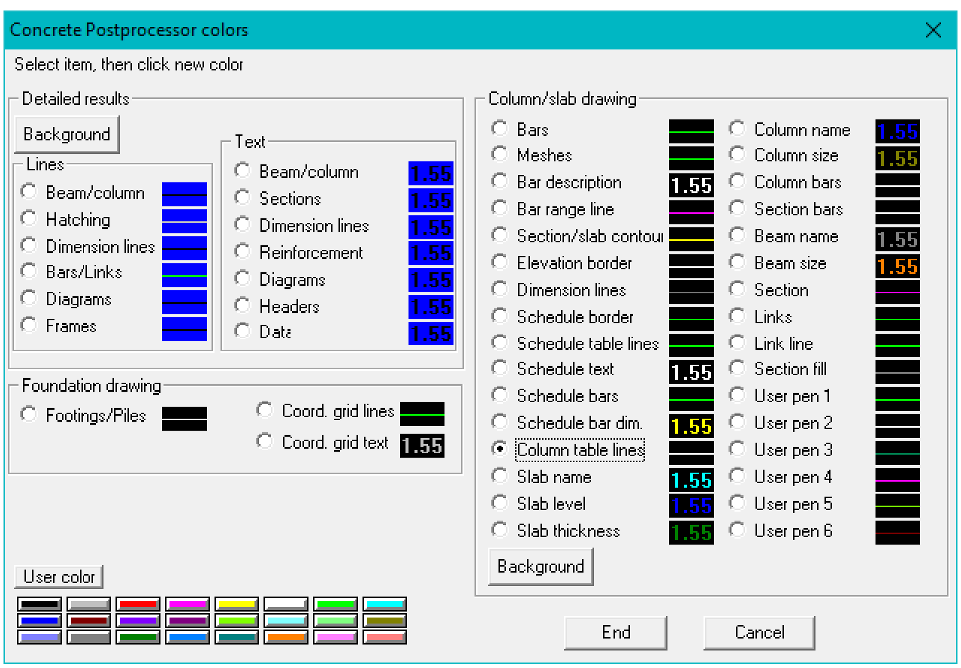
8.19.2 Setup



Colors

This option enables the selection of colors for the graphic screens displayed by the **Detailed results** option only.

All colors in the general model display may be changed using the Setup - colors option in the *STRAP* main menu.



Refer to Setup - colors for detailed instructions.

Line width for printing

Specify the line width when printing for each of the column elements in the drawing.

Note that a width of 0.0 indicates a width of 1 pixel.

Concrete types

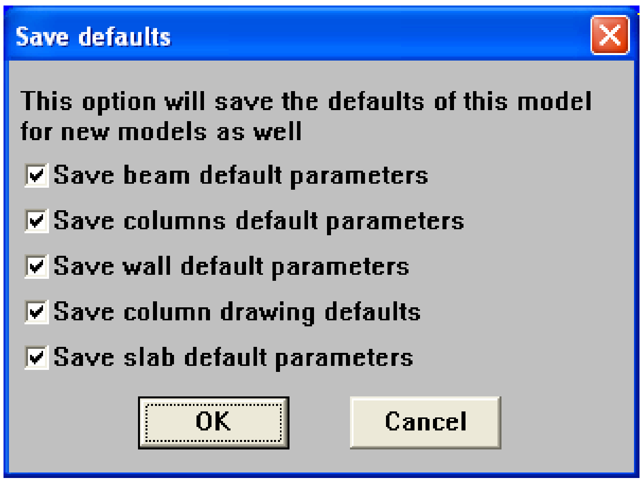
Enter a list of nominal concrete strengths. The program automatically calculates all allowable stress values from the nominal strength.

Bar diameters

Refer to Setup - bar diameters 963.

Save defaults

The default parameters specified for the current model may be saved as the default parameters for all new models. Select the parameters to be saved:



Column drawing parameters

Refer to Setup - column drawing parameters 965

Column table parameters

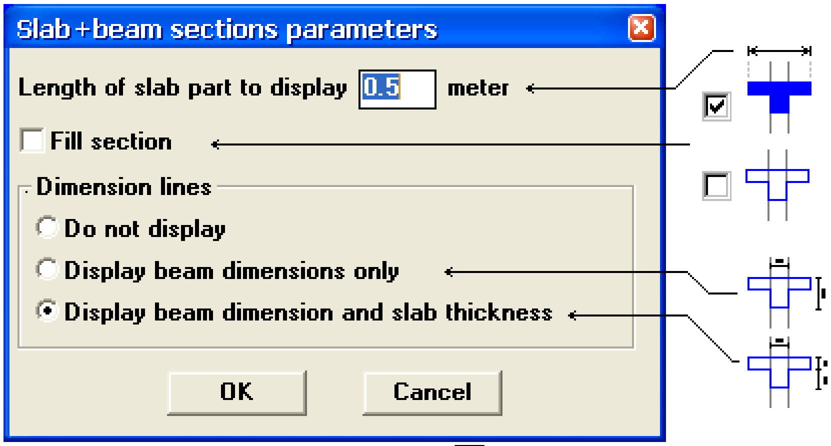
Refer to Setup - column table parameters 967

Text sizes

Specify the size of various slab drawing text. Refer to Slab drawing - parameters 895.

Slab sections

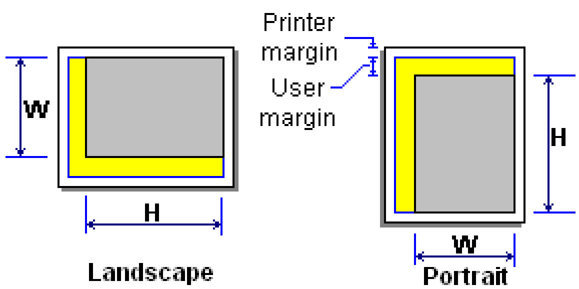
Specify the text parameters for the beam/slab sections added to the slab drawing.



Refer to Slab - Edit - Additional options 905.

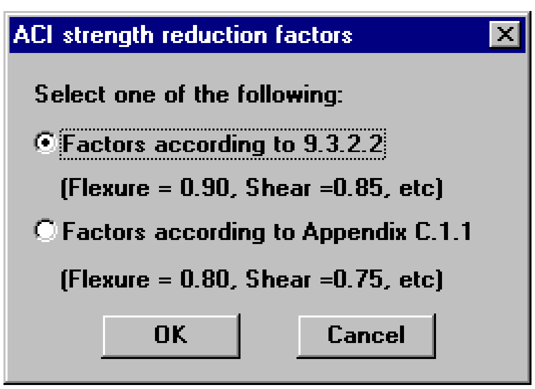
Drawing paper types

Specify the **net** paper size available for the drawing. The dimensions represent the actual paper size less the margins specified in the printer manual less any additional margin.



ACI reduction factors

By default, the program uses the strength reduction factors specified in ACI 9.3.2.2.



Select the second option to use the alternate factors specified in ACI Appendix C.

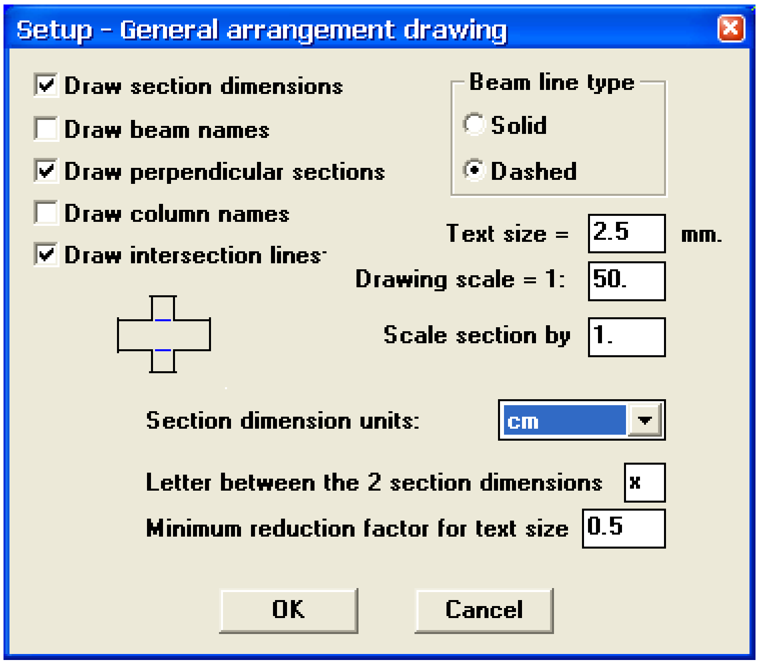
The factors are:

	9.3.2.2	App. C
Flexure, without axial load:	0.90	0.80
Axial tension:	0.90	0.80
Compression + flexure:	0.70	0.65
Compression + flexure, spiral:	0.75	0.70

IS456- Torsion

- Torsion - IS:456
  According to the Indian Code IS:456, all beams with a torsional moment must be designed and detailed accordingly. Specify a minimum torsional stress; torsion is ignored in all beams with a stress value less than that specified here.

General arrangement drawing

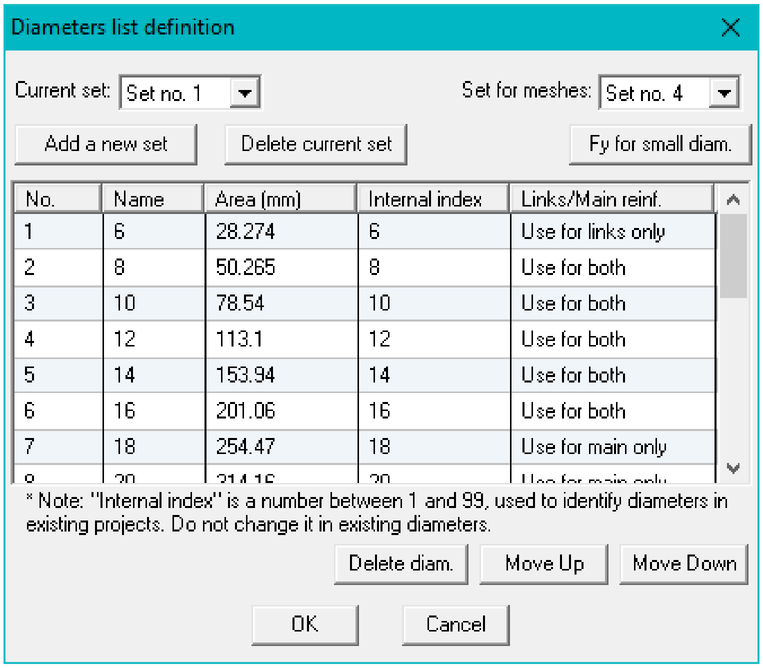


refer to General arrangement drawing - parameters 974

- units for section dimension
- delimiter character for section dimensions, e.g. 300/500, 300x500, etc.
- text size reduction factor: the program adjusts the size of the section dimension text so that it fits within the beam and does not clash with other text. Enter the smallest reduction factor permitted.

8.19.2.1 Bar diameters

Create and modify sets of bar diameters and select a set for the current model. Note that the sets are used by all models in all directories.



Enter /revise the data for each reinforcement bar:

- **Name**
  A text string that is displayed in all menus and result tables (maximum length = 4 characters)
- **Area**
  design bar area, according to the units displayed at the top of the column
- **Internal index**
  An index used by the program; any value between 1-99 is permissible when adding a new bar or creating a new table. **Do not modify the index for existing bars *!!!***
- **Links/main**
  Specify whether a diameter can be selected by the program for links only, main reinforcement only, or both. The selection applies to both beams and columns.

Current set

Select a new set of reinforcement bars for the current model only

Note:

- the current set for other existing models does not change
- the current set is the default set for new models

Add a new set

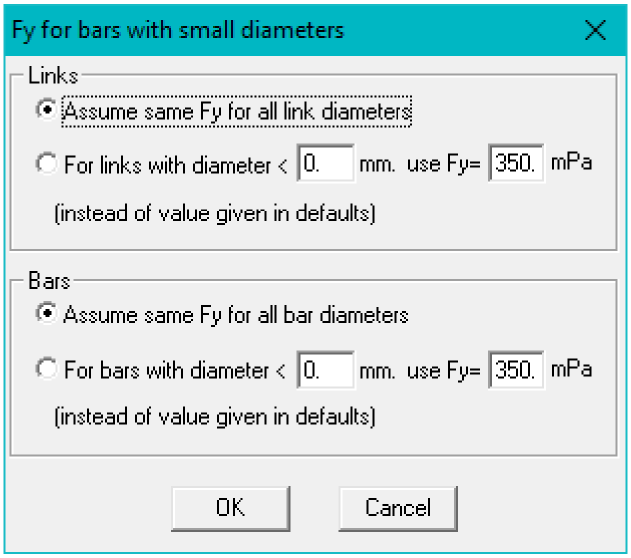
Add a new table of reinforcement bars. The program automatically copies the data of the current set to the new set and the data may then be updated.

Delete current set

- do not delete a set that is the current set of any model on the computer.
- do not delete a set if any of the following sets is used by a model (i.e. if Set no. 3 is deleted, Set no. 4 changes to Set no. 3, etc).

Fy for small diameters

Override the default f_y value for small diameter bars. Different diameter limits can be specified for main reinforcement and links/stirrups. :



Move up/down

Highlight a line in the table, then click Move up or Move down move the line to a different location.

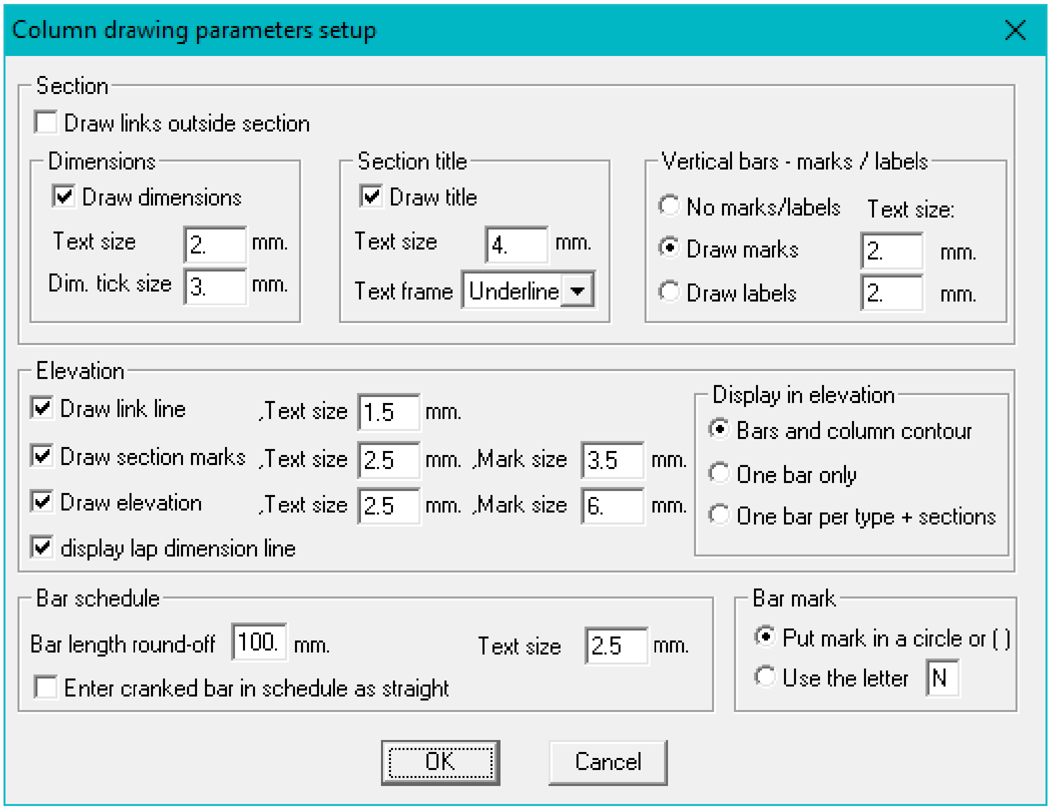
Note that this option is for convenience only as the program internally always sorts the list by area when selecting reinforcement.

Note:

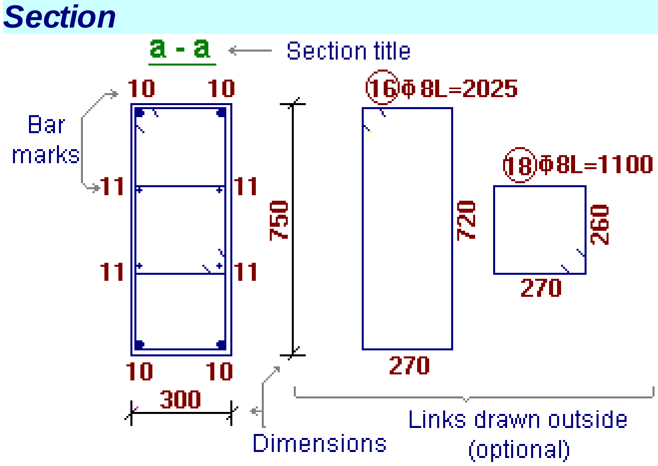
- all changes applies to all models that use the current set.
- to add a new diameter to the list, enter the data in a blank line at the end of the list, then click Move Up to move the line to its correct location in the table
- to delete a diameter from the list, highlight the line then click Delete diam.. Note that the diameter is deleted from all models that use the current set.

8.19.2.2 Column drawing parameters

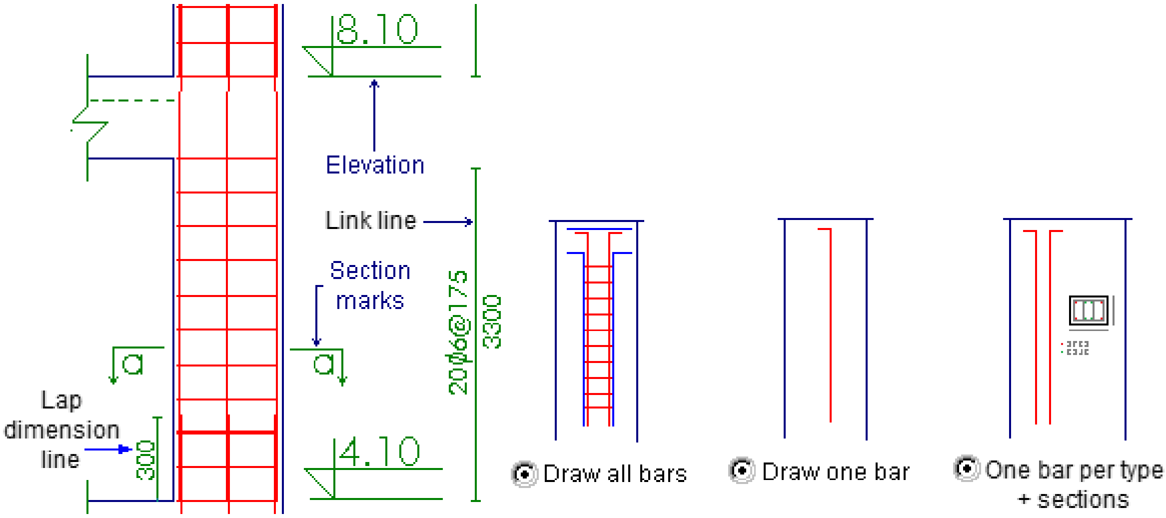
Set the default parameters for all details on the column drawing:



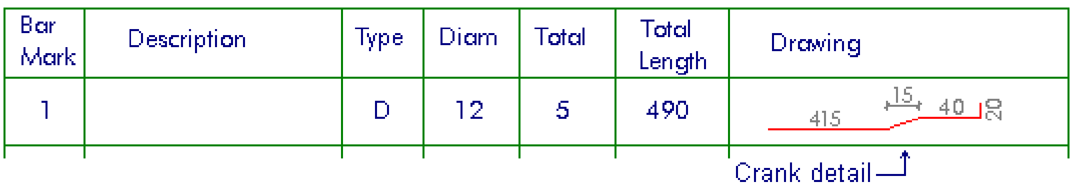
Section



Elevation



Bar schedule



Bar Mark	Description	Type	Diam	Total	Total Length	Drawing
1		D	12	5	490	415 15 40 20

Bar mark

Two formats are available for the mar mark:

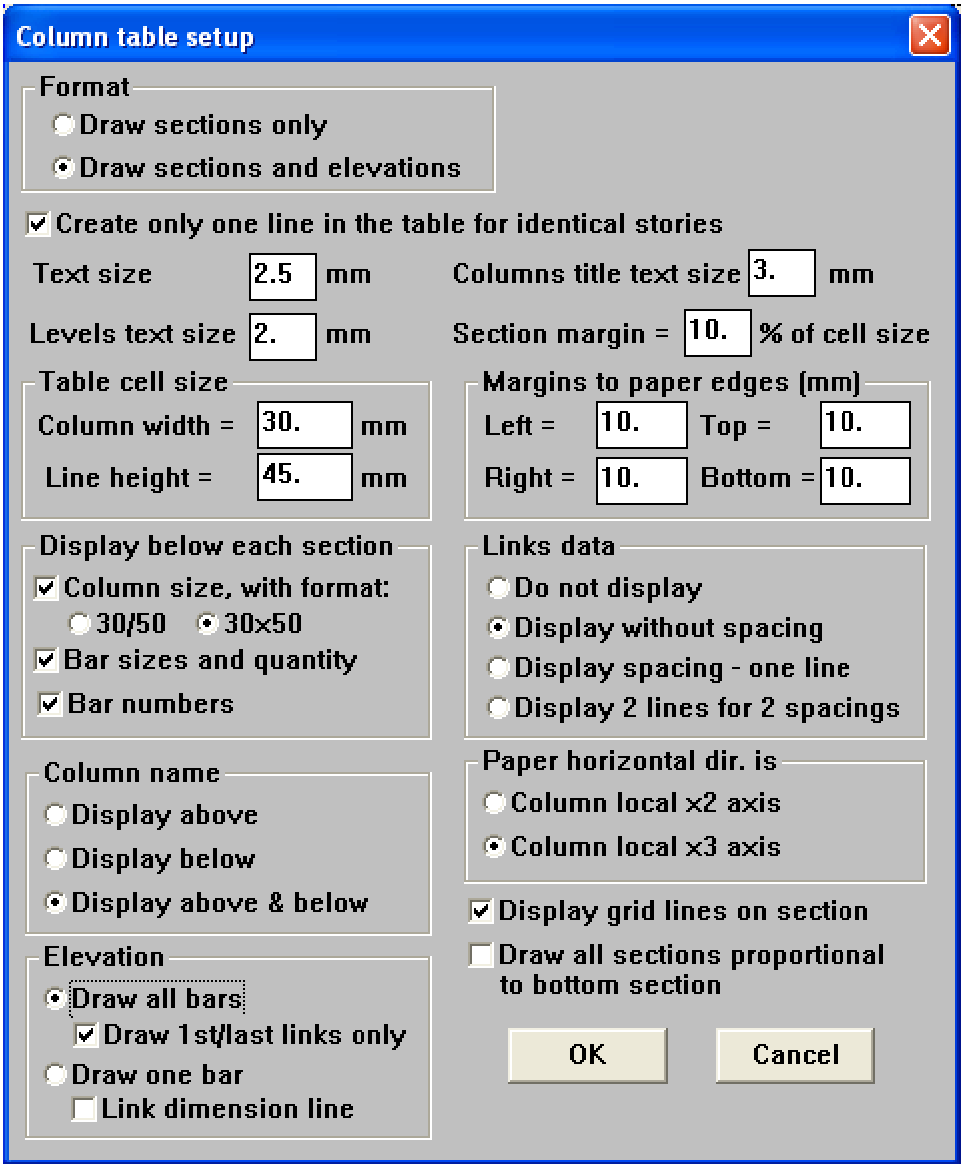
Example:

Put mark in a circle or () **[34]5ϕ12**

Use the letter [N] **5N34ϕ12**

8.19.2.3 Column table parameters

Specify the default parameters for the column drawing tables:



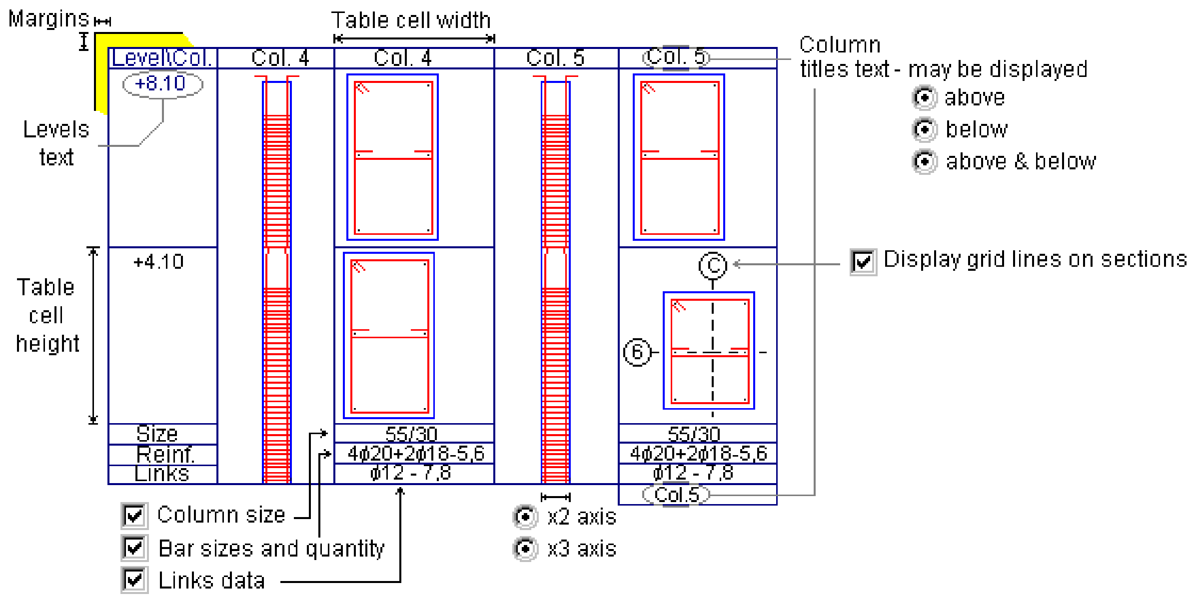
Format

Create the table with sections only or create it with both sections and elevations.

Note that the elevations may be added later using the 

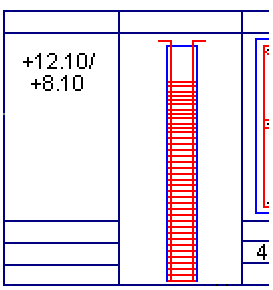
option

Parameters



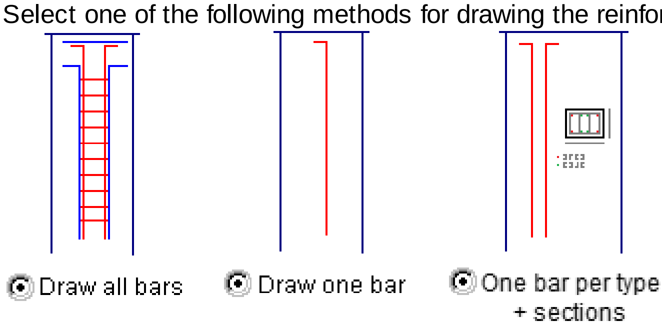
Identical stories

Identical columns at adjacent levels may be combined to one row in the table. For example:



Elevation

Select one of the following methods for drawing the reinforcement in the elevation:



Links data

Select one of the following options for the link data in the table:

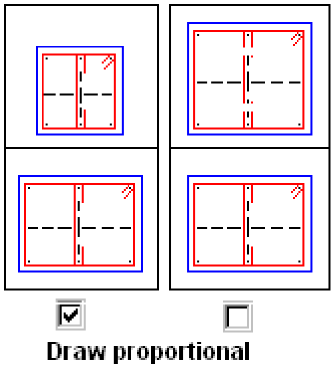
- do not display
- display without spacing, i.e. number and diameter

for columns with two different spacing groups (seismic):

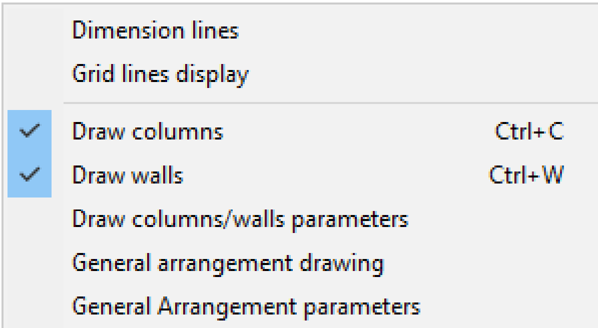
- display the groups on one line, e.g. ϕ8@50/150
- display the groups on two different lines, e.g:
  ϕ8@ 50

ϕ8@150

Draw all sections proportional to the bottom section



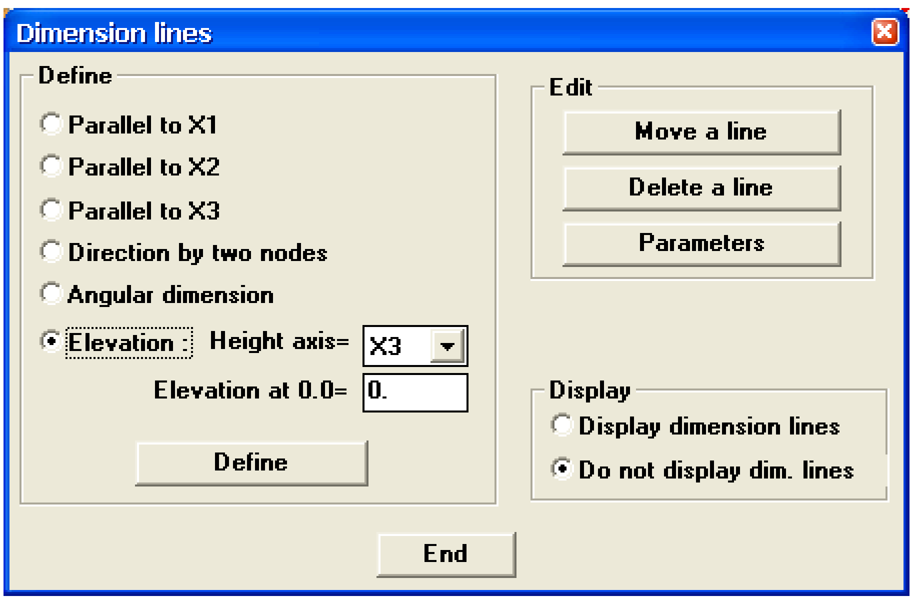
8.20 Draw options



8.20.1 Dimension lines

Add dimension lines or elevation marks (e.g. +24500) to the drawing:

- arrow style, text style and number of digits may be specified. Refer to [Parameters](972)972
- dimension lines defined with this option are saved in Views



Define

- Select on of the following options:

  Parallel to X1/X2/X3
  Select one of these options to plot the dimension line parallel to a global axis.

  Defined by 2 nodes:
  The dimension line is drawn parallel to a line connecting two selected nodes. Select the nodes to be included in the dimension line with one of the following options:

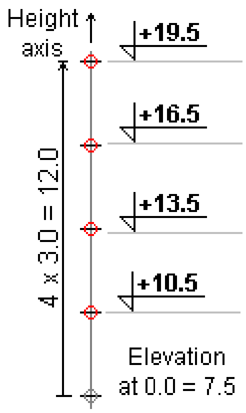
  click Define

- select the nodes defining the dimension lines using the standard node selection option and then move the cursor to the dimension line location:

The dimension line is drawn at the crosshair location parallel to the direction selected previously.

Elevation

Display elevation marks along any of the global axes at selected levels. For example:



- Select the model height axis and specify the elevation at the model 0.0 coordinate of that axis
- click Define
- Select the nodes at the levels that require a section mark using the standard node selection options
- move the cursor to the elevation mark location (perpendicular to the height axis) and click the mouse

Note that the arrow and text dimensions can be revised in the Parameters 972 option.

Move/delete

Move/delete a dimension line or elevation line from the model

- Highlight a dimension line or elevation line and click the mouse
- for "Move", move the to the new location and click the mouse.

Parameters

Refer to Dimension line - parameters 972

Display

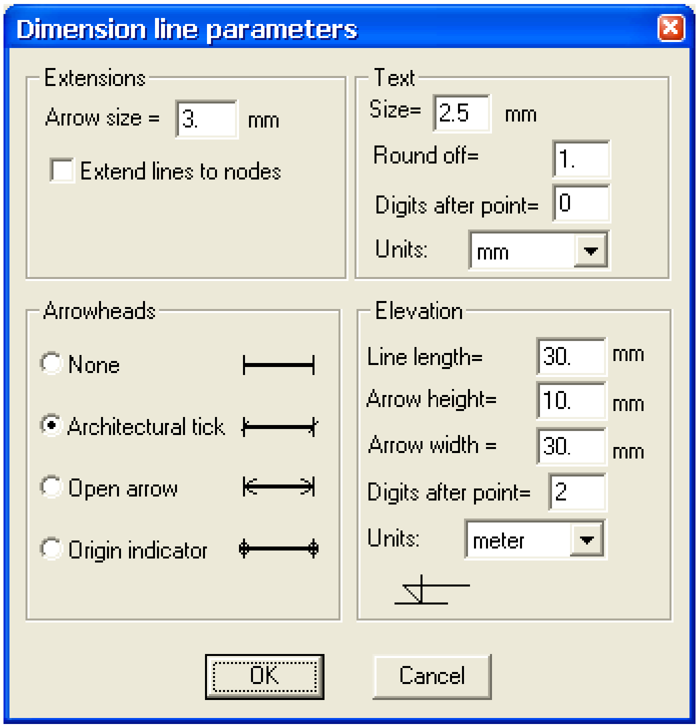
All dimension lines/elevations defined for this model may be temporarily deleted from the display.

Note:

- the dimension and elevation lines is erased if the display is rotated or a different plane is selected. To retain the lines, save the current View.

8.20.1.1 Parameters

Specify the parameters for dimension lines and elevation lines. Note that any changes to the parameters also revises existing dimension lines.



Extensions

Note:

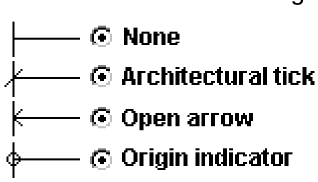
- "arrow size" is revised for the print options only and remains constant on the screen.
- "arrow size" dimension affects all arrowhead types.

Text

- **Round off**
  Round off all dimensions and elevations to the value specified here
- **Digits after point**
  Specify the number of digits to be displayed after the decimal point. Note that this number of digits will always be displayed, even if the **Round off** value requires more digits.

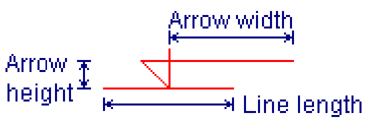
Arrowheads

Select one of the following arrowhead types:



Elevation

Specify the elevation mark parameters:

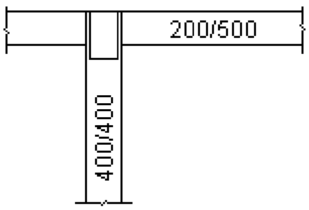


Note:

- all dimensioned items (text size, arrow size, etc) are printed with the specified size only if the scale selected when printing is the same as the scale selected in the General arrangement parameters option; otherwise sizes are modified according to the ratio of the scales. For example, a scale of 1:50 is specified here but a scale of 1:100 is specified when printing: the actual text/arrow sizes is one-half (50/100) of the sizes selected in this option.

8.20.2 General arrangement drawing

Create a "general arrangement" drawing for any plane in the model. For example:

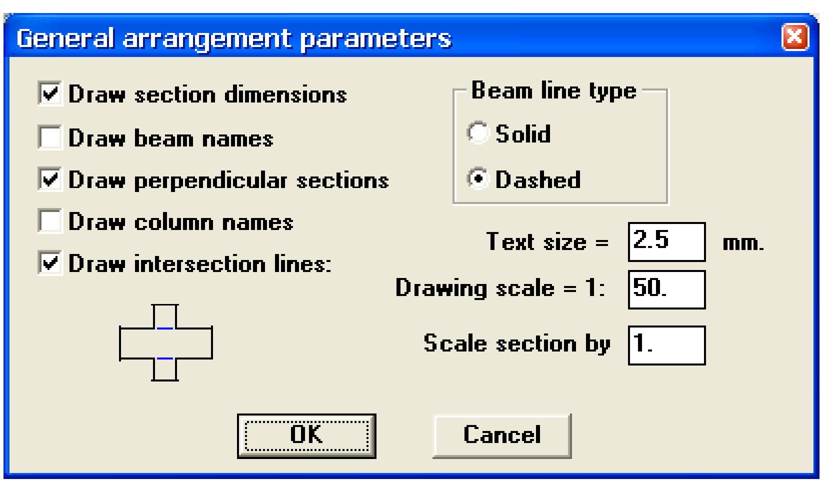


- the drawing may be generated on any plane, e.g. plans or elevations. If more than one plane is displayed on the screen when this option is selected, the program prompts the user to select a plane defined by three nodes.
- all sections are drawn as rectangular, e.g. for T-sections, only the web is drawn.
- the program writes the rectangular section dimensions adjacent to each member. The dimensions are written only once for a string of identical sections. The text size is specified by the user.

For more details, refer to General arrangement - parameters 974.

8.20.2.1 Parameters

Specify the parameters for General arrangement drawings.



Section parameters

Draw section dimensions
Draw the section dimensions are drawn at the center of the beams.

Draw beam names

Draw the current beam names (as defined in the Define beam names 839 option) at the center of the beams.

Draw perpendicular section
Members perpendicular to the displayed plane may be deleted from the drawing. The sections are drawn to scale but may be increased/ reduced by the scale factor.

Draw column names
Add the column names to the drawing (defined in the Define column names 844 option). Note that the names are displayed only for columns perpendicular to the display plane.

Draw intersection lines
Draw/do not draw lines where beams with different height intersect.

Beam line type
Draw all beam with solid or dashed lines.

Scale / text size

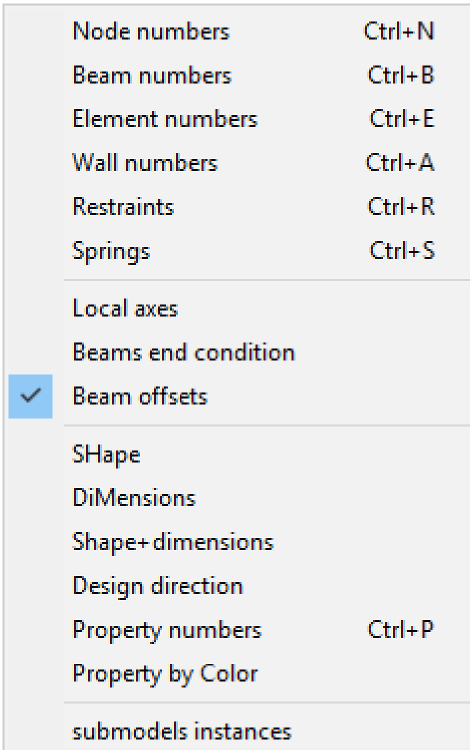
Specify the drawing scale and the text size. The drawing scale is required to determine the size of the text on the screen display.

Note that the text is printed with this size only if the scale specified when printing is the same as the scale specified in this option; otherwise the text size is modified according to the ratio of the scales. For example, a scale of 1:50 is specified here but a scale of 1:100 is specified when printing: the actual text size is one-half (50/100) of the size selected in this option.

Note:
- any changes to the parameters also revises existing arrangement drawings (saved as views).
- default General arrangement drawing parameters may be specified in the Setup option

8.21 Display

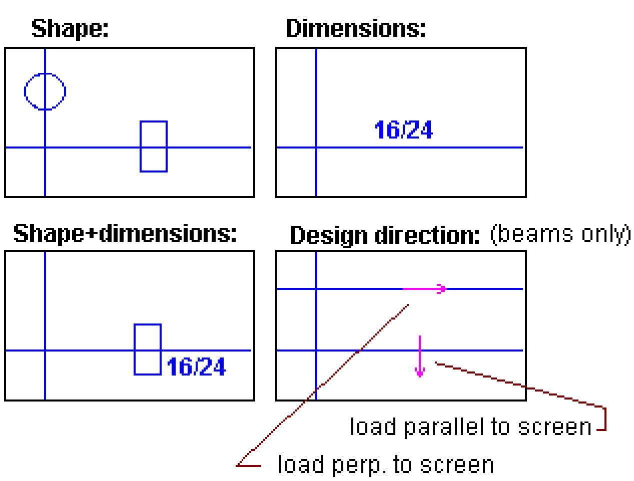
Use this option to ***graphically display*** the input data and the results alongside each member:



Shape / dimensions / direction

Display the section shape, dimension and/or design direction adjacent to each member in the model:

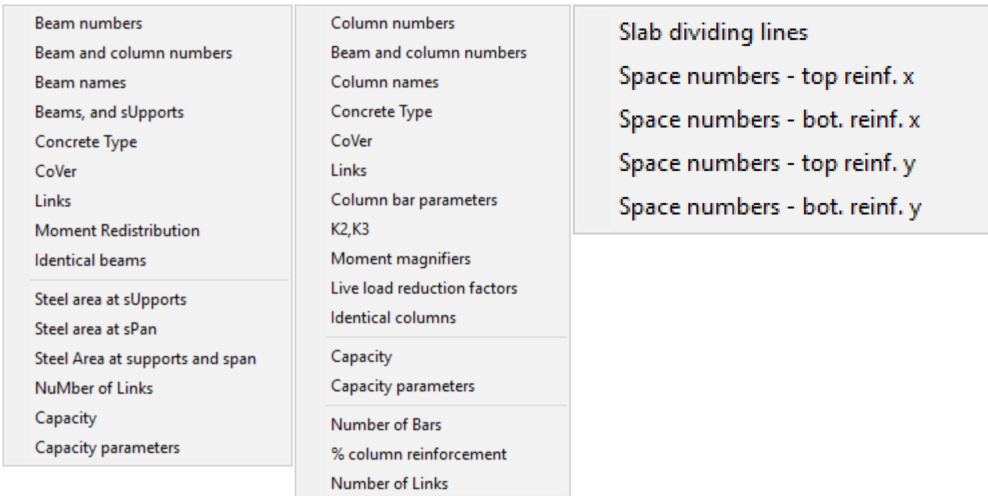
Examples:



8.22 Data display

Display graphically concrete design parameter and result data alongside each member:

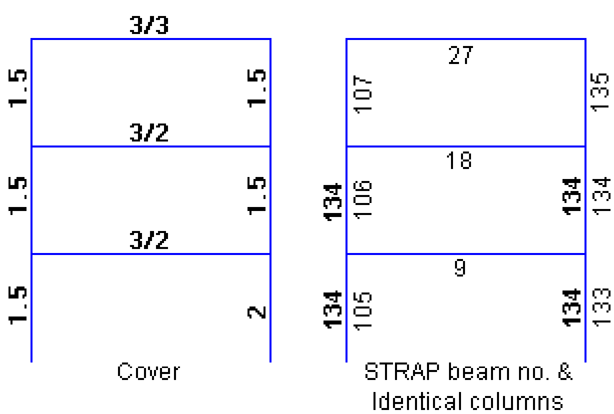
Beams: Columns, walls: Slabs:



Parameters / Results

The design parameters and results may be displayed alongside the beams/columns/walls in the graphic display. Select one of the design parameters or result types in the menu:

For example:

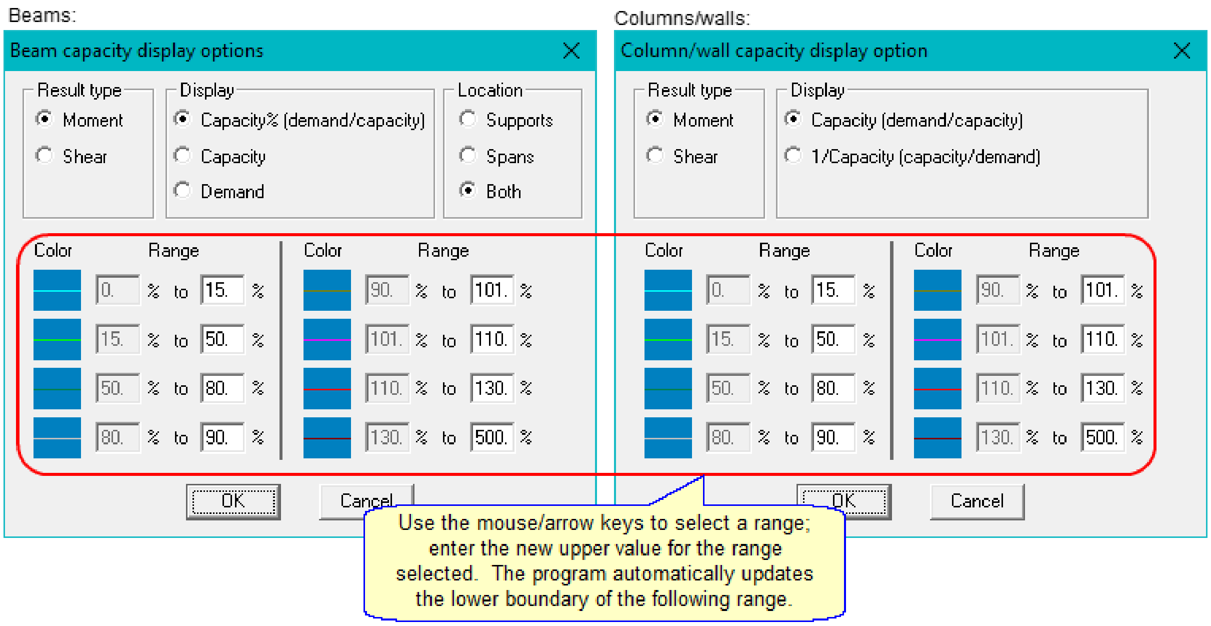


Note

- top/bottom cover values are displayed for beams.
- identical columns: the program displays the largest *STRAP* beam number of all columns in the group (in the above example, 105, 106, 133, 134 are identical)

Capacity parameters

The capacity/demand data may be displayed with a colour code, where each colour represents a range of the result. The range limits are defined in the following menu:



Beams:

Display the moment/shear capacity or demand or the capacity/demand ratio at supports span, spans or both
Note:
- only the demand/capacity can be displayed: ratio for an adequate column/wall < 1.00

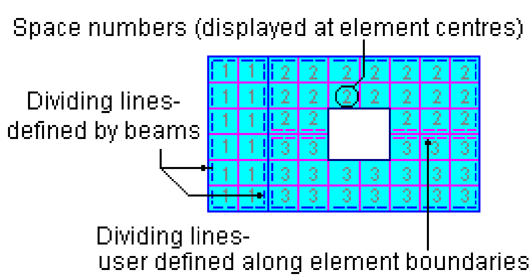
Columns/walls:

Display the moment/shear capacity or demand or the capacity/demand ratio
Note the ratio can be displayed in one of two ways\;
- demand/capacity: ratio for an adequate column/wall < 1.00
- capacity/demand: ratio for an adequate column/wall > 1.00

Colours are revised in the Setup option in the *STRAP* main menu.

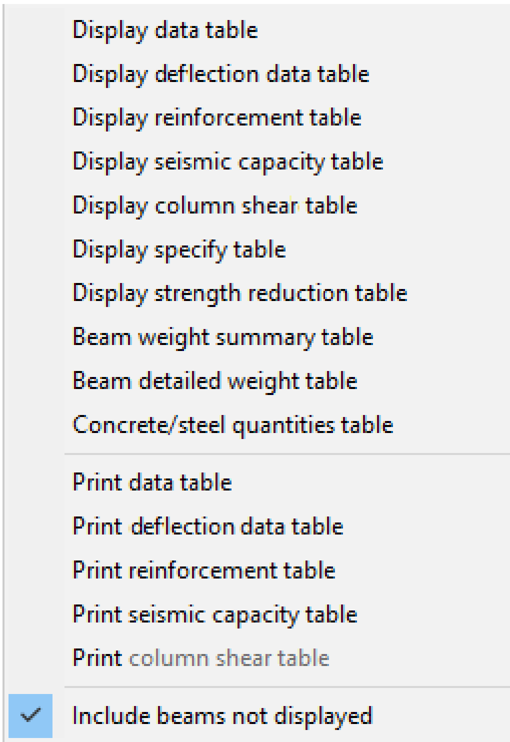
Slabs

The following data can be superimposed on the graphic display:



8.23 Data tables

To display the current design data - Default or Parameters - for each beam, column or wall element. Note that member data is displayed only for members that are part of a continuous beam or column.



8.23.1 Data table

Beam data table 980
Column data table 981
Wall data table 981
Slab data table 952

8.23.1.1 Beams

(Headers may vary slightly for different Codes)

Mem No.	=	*STRAP* member number
Beam no.	=	concrete module beam number.
Nextmem.	=	next *STRAP* member in continuous beam.
Prop	=	*STRAP* property group number
Dir	=	design direction - M2 or M3
f'c	=	concrete grade.
ds	=	cover to center-of-gravity of main reinforcement.
Mom Red	=	automatic moment redistribution: No or Max/Min percentages if Yes.
ShearRed	=	shear reduction to distance 'd' from face of supports: Yes or No.

Shear reinforcement:

#	=	range of allowable stirrup diameters.
Leg	=	default number of legs in stirrups and alternate (larger) number.

Spc = for "Stirrups Only" : minimum stirrups spacing
for "Stirrups + Bent-up bars": constant stirrup spacing along length of beam
By = for "Stirrups Only" : stirrup spacing increment.
Grp = maximum number of stirrup groups allowed in this span.

Supports:

JA,JB = denotes member start, end nodes.
T = support type: S = Support ; N = No support; C = Cantilever
Wid = support width.
Def/L = maximum allowable deflection (BS 8110, EC2, IS:456 only)

8.23.1.2 Columns

(Headers may vary slightly for different Codes)

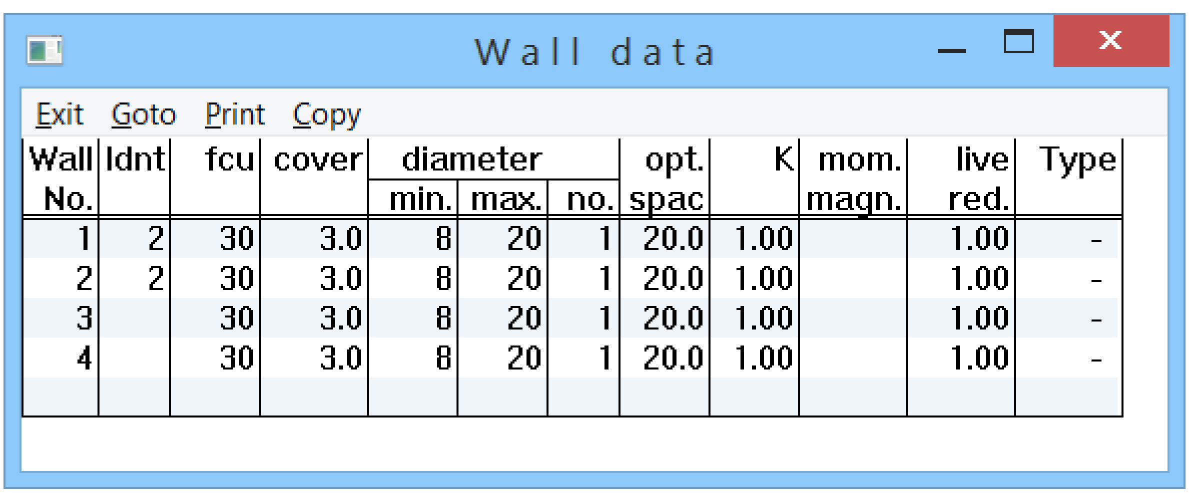
Mem No. = *STRAP* member number
Colno. = Concrete module column number.
Nextmem = next *STRAP* member in continuous column.
Prop = *STRAP* property group number.
f'c = concrete grade.
ds = cover to center-of-gravity of main reinforcement.

Diameter:

Min = minimum diameter allowed in this member.
Max = maximum diameter allowed in this member
No = number of different diameter allowed (1 or 2)

Opt.Spacing = optimal spacing between adjacent bars. If no solution is available with this spacing, the program adds more bars and reduce the spacing.
Dir = design moment direction - M2 or M3
Type = Braced or Unbraced
K = Effective length factor
Support = Height of the beam framing into the column member at its end.
Flange = for U,L and T-sections: the flange location relative to the local axes.
Magn = Moment magnifier for slender columns. If a value is not displayed, the program calculates the value automatically.
Live red. = Live load reduction factor

8.23.1.3 Walls



Wall No.	Idnt	fcu	cover	diameter min.	max.	no.	opt. spac	K	mom. magn.	live red.	Type
1	2	30	3.0	8	20	1	20.0	1.00		1.00	-
2	2	30	3.0	8	20	1	20.0	1.00		1.00	-
3		30	3.0	8	20	1	20.0	1.00		1.00	-
4		30	3.0	8	20	1	20.0	1.00		1.00	-

(Headers may vary slightly for different Codes)

WallNo. = *STRAP* member number
Idnt = the wall that this wall is identical to (highest wall number in the identical list)
fcu = concrete grade.
cover = cover to center-of-gravity of main reinforcement.

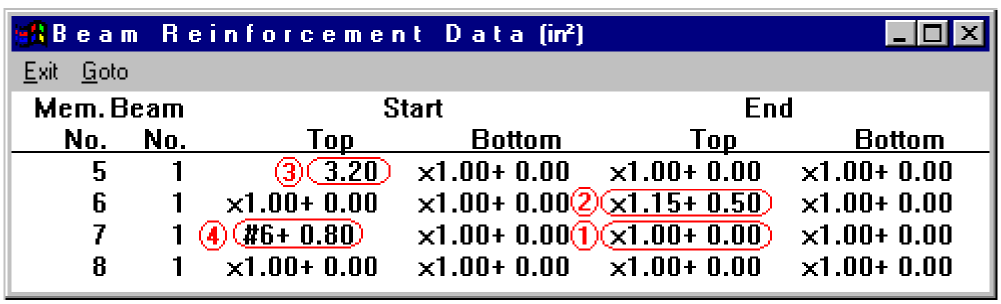
Diameter:
Min = minimum diameter allowed in this member.
Max = maximum diameter allowed in this member
No = number of different diameter allowed (1 or 2)

Opt.Spacing = minimum spacing between adjacent bars. If no solution is available with this spacing, the program adds more bars and reduce the spacing.
K = Effective length factor
Magn. = Moment magnifier for slender columns. If a value is not displayed, the program calculates the value automatically.
Live red. = Live load reduction factor
Type = [not used]

8.23.2 Deflections

8.23.3 Reinforcement

This table displays the user specified reinforcement ("Default" and "Parameters"). For example:



Mem. No.	Beam No.	Start Top	Start Bottom	End Top	End Bottom
5	1	(3) 3.20	x1.00+ 0.00	x1.00+ 0.00	x1.00+ 0.00
6	1	x1.00+ 0.00	x1.00+ 0.00	(2) x1.15+ 0.50	x1.00+ 0.00
7	1	(4) #6+ 0.80	x1.00+ 0.00	(1) x1.00+ 0.00	x1.00+ 0.00
8	1	x1.00+ 0.00	x1.00+ 0.00	x1.00+ 0.00	x1.00+ 0.00

The table shows the actual reinforcement at the start/end, top/bottom of every beam. Four examples are highlighted:

(1) The design area has not been modified by the user, i.e. the design area is increased by a factor **x1.00** and an additional area of **0.00** is added
(2) The design area is increased by a factor **x1.15** and an additional area of **0.50** is added.
(3) An area of **3.20** is specified (the design area will be used if greater than this value).
(4) The design area is rounded off to **#6** bars and an additional area of **0.80** is added

8.23.4 Seismic

The program displays the beam "Seismic capacity table". For example:

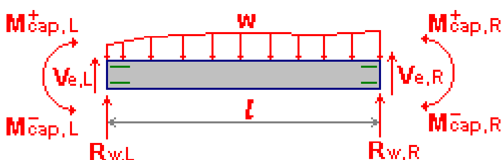
Beam - Seismic Capacity (ft-kp)

Exit Goto

Beam no.	Mem. no.		Reinf. A's	As	Moment-prob Mpos	Mneg	Moment-nom Mpos	Mneg	Shear Vpos	Vneg
1	5	Start	3.5	1.9	298.1	534.0	240.8	430.7	3.8	-68.6
		End	3.3	1.9	298.1	511.7	240.8	412.7	67.8	-4.6
1	6	Start	3.3	1.9	298.1	511.7	240.8	412.7	2.3	-67.8
		End	3.0	1.9	298.2	466.4	240.8	376.0	66.3	-3.8

At the start and end of each beam:

- **A's, As**:
  The actual reinforcement areas at the span supports, as modified by the user.
- Moment capacities, based on the actual reinforcement:
  nominal : capacity calculated using concrete and steel strengths not reduced by Code factors
  probable : capacity calculated using increased steel strength, i.e. actual conditions
- The seismic design **Shear** forces (Ve), calculated from the probable moment strength of the beam together with the factored beam loads:



$$V_e = \frac{M^{\pm}_{cap,L} + M^{\pm}_{cap,R}}{l} + R_w$$

8.23.5 Shear

Display the column shear parameters as specified in the Default and Parameter options. For example:

Column- Shear Data (in.)

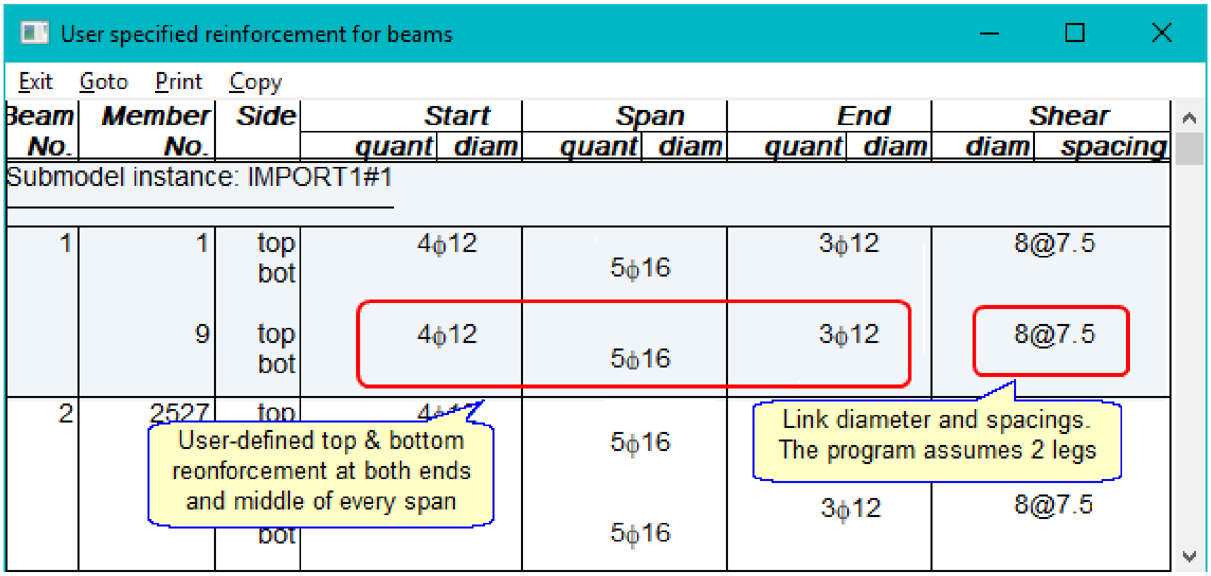
Exit Goto

Mem. no.	Col no.	Next Mem.	Diam Min	Middle Spacing Min	Inc	Legs M3	M2	Seismic Spacing Min	Inc	Legs M3	M2
17	4	18	4	2.	2.	4	4	2.	2.	4	4
18	4	19	4	2.	2.	4	4	2.	2.	4	4
19	4		4	2.	2.	4	4	2.	2.	4	4

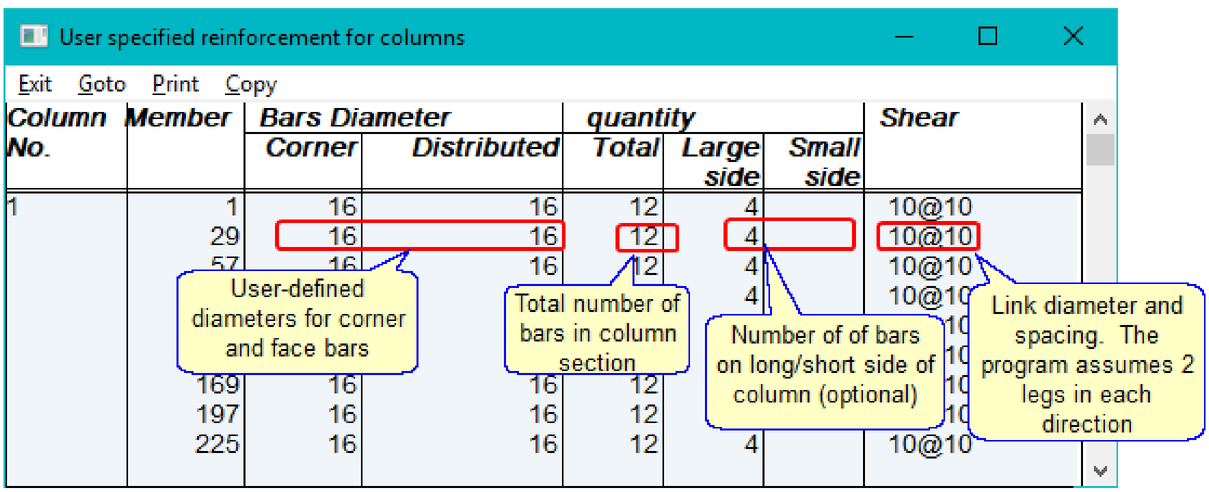
8.23.6 Specify table

Display the "specify bars 882" data. The reinforcement is used to calculate the moment and shear capacities for elements in existing structures

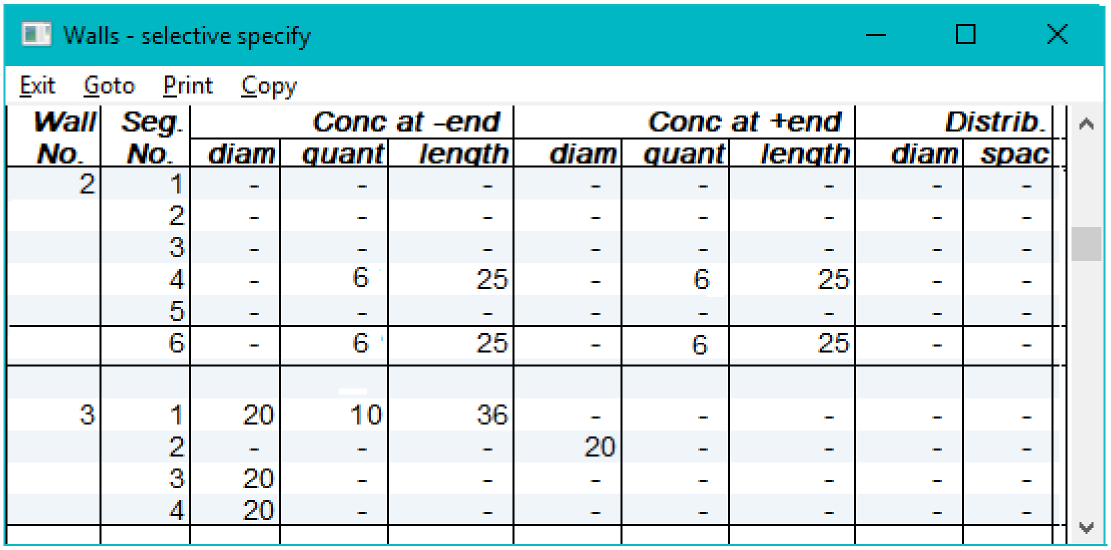
Beams



Columns



Walls



Wall No.	Seg. No.	Conc at -end diam	Conc at -end quant	Conc at -end length	Conc at +end diam	Conc at +end quant	Conc at +end length	Distrib. diam	Distrib. spac
2	1	-	-	-	-	-	-	-	-
	2	-	-	-	-	-	-	-	-
	3	-	-	-	-	-	-	-	-
	4	-	6	25	-	6	25	-	-
	5	-	-	-	-	-	-	-	-
	6	-	6	25	-	6	25	-	-
3	1	20	10	36	-	-	-	-	-
	2	-	-	-	20	-	-	-	-
	3	20	-	-	-	-	-	-	-
	4	20	-	-	-	-	-	-	-

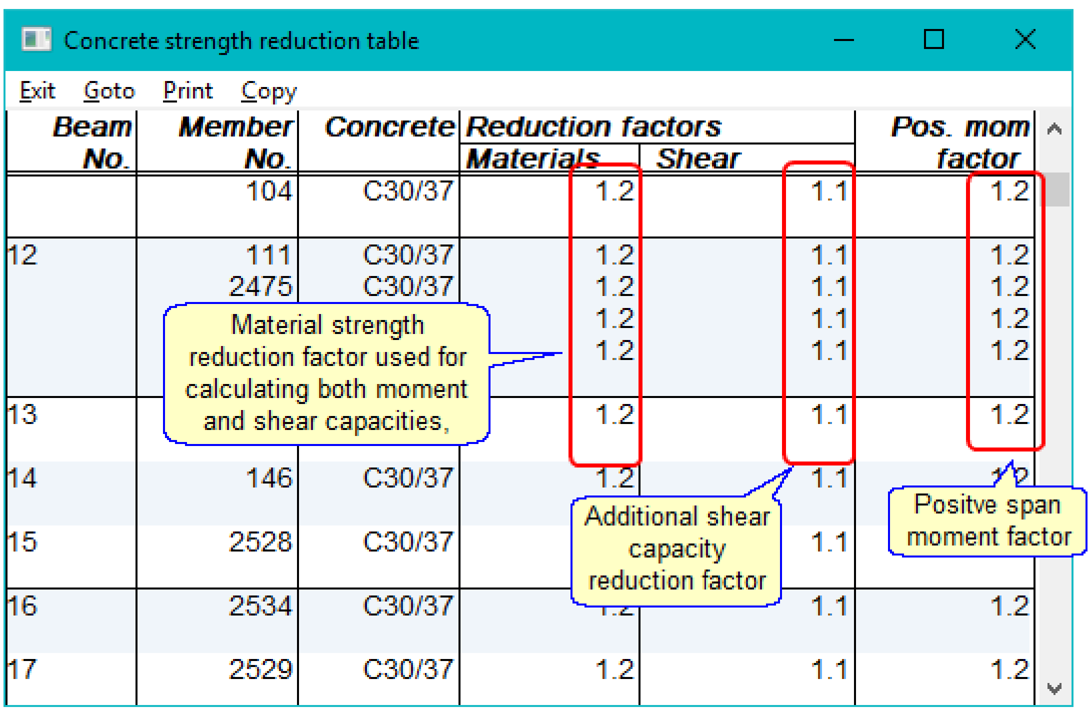
where:

Conc. = specified "concentrated" wall reinforcement quantity, diameter.and/or length

Distrib = specified "distributed" wall reinforcement diameter and/or spacing.

8.23.7 Strength reduction

Display the strength reduction factors defined in the Specify bars 882 option for beams, columns and/or walls. For example, for beams:

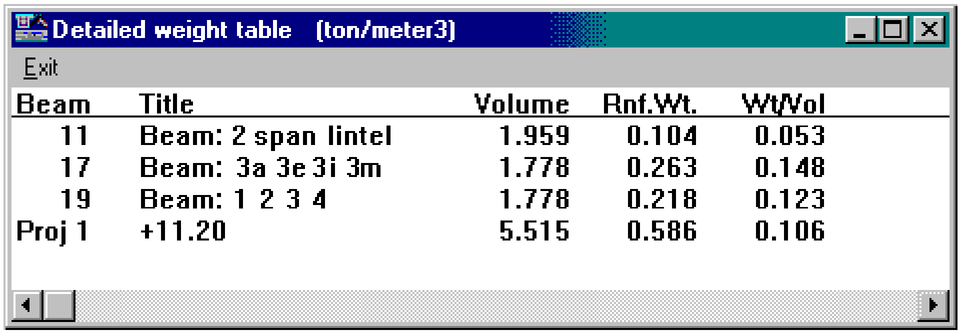


Beam No.	Member No.	Concrete	Reduction factors Materials	Reduction factors Shear	Pos. mom factor
	104	C30/37	1.2	1.1	1.2
12	111	C30/37	1.2	1.1	1.2
	2475	C30/37	1.2	1.1	1.2
			1.2	1.1	1.2
			1.2	1.1	1.2
13			1.2	1.1	1.2
14	146	C30/37	1.2	1.1	1.2
15	2528	C30/37		1.1	
16	2534	C30/37		1.1	1.2
17	2529	C30/37	1.2	1.1	1.2

8.23.8 Beam weight

Display the ratio of reinforcement weight to concrete volume for detailed beams. Select one of two options:

- **Display detailed weight table**
  the program displays the data for every detailed beam, sorted by projects as well as the totals for each project. For example:


Detailed weight table   (ton/meter3)
Exit

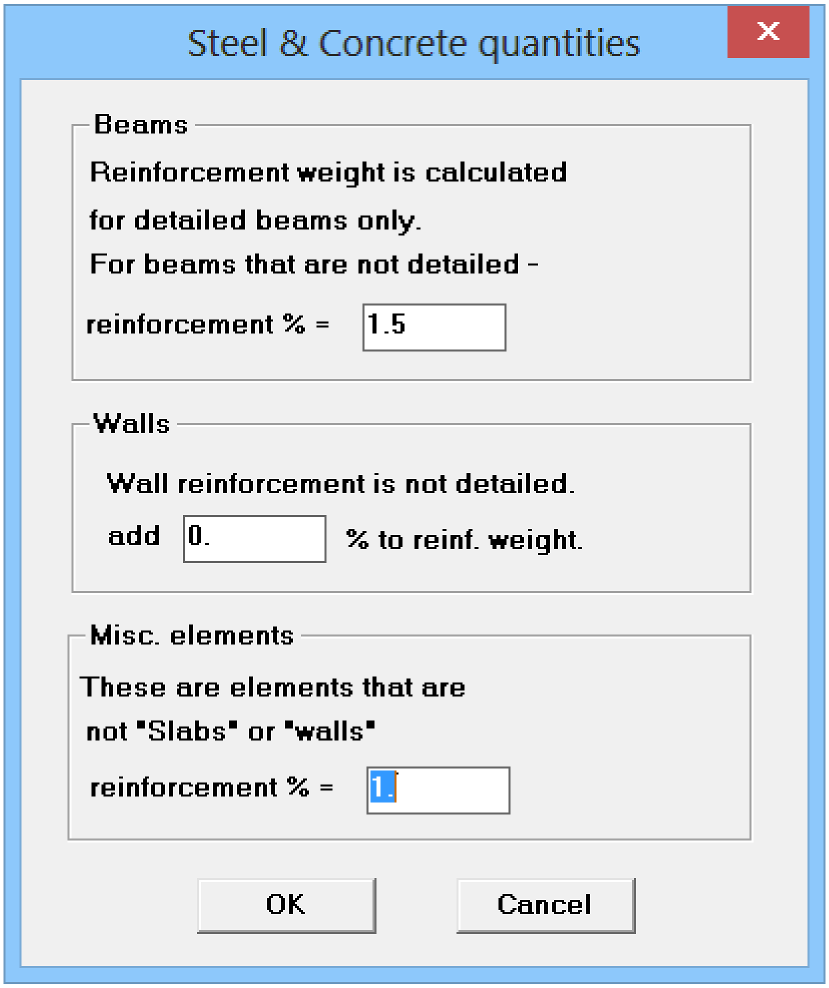
Beam	Title	Volume	Rnf.Wt.	Wt/Vol
11	Beam: 2 span lintel	1.959	0.104	0.053
17	Beam: 3a 3e 3i 3m	1.778	0.263	0.148
19	Beam: 1 2 3 4	1.778	0.218	0.123
Proj 1	+11.20	5.515	0.586	0.106

- **Display weight summary**
  the program displays only the totals for each project (the last line in the above example).

Note:
- Concrete volume is calculated from the web area only.
- Weight is not calculated for beams that have not been detailed.

8.23.9 Quantities

Calculate concrete and reinforcement quantities for the entire model:



- ***Beams***
  - Quantities are calculated for defined beams only.
  - Concrete volume is calculated from the web; projecting flanges are assumed to be part of the slabs.
  - The exact reinforcement weight is calculated only for beams that have been detailed in *BEAMD*; for the other defined beams the program uses the reinforcement percentage defined by the user in the dialog box above.

- ***Columns***
  - Quantities are calculated for defined columns only.
  - The program "computes" defined columns that have no results in order to calculate the quantities for them.
  - The program calculates the reinforcement quantities from the detailed reinforcement on the column drawings. For columns not placed on drawings - the program first "details" the columns according to the current column drawing default parameters; if not all columns are on drawings.it is important to revise the default parameters before selecting this option

- ***Walls***
  - Quantities are calculated for all wall elements.

- The program "computes" walls that have no results in order to calculate the quantities for them.
- The program calculates the reinforcement quantities from the detailed results but does not calculate lap lengths, etc. The program adds the percentage defined by the user in the dialog box to account for the additional amount.

- ***Slabs***
  - Quantities are calculated for all elements perpendicular to the height axis.
  - The program calculates the reinforcement quantities from the detailed reinforcement on the slab drawings. For slabs not placed on drawings - the program first "details" the slabs according to the default "spaces" and the current slab drawing default parameters; if not all slabs are on drawings it is important to define the spaces and revise the default parameters before selecting this option .

- ***Miscellaneous elements***
  - "Miscellaneous elements" are finite elements that are neither "walls" nor "slabs".
  - Reinforcement weight is calculated according to the percentage defined by the user in the dialog box

- ***Footings***
  - Quantities are calculated only for footings that were designed with the "Files-Footings" option in the *STRAP* results module.
  - The program calculates the quantities for all footings currently in the model footing file (including additional footings added manually).

Example:

Concrete and steel quantities (m3, ton)

Exit Goto Print Copy

	Conc. Vol.	Reinf. Wt.	Notes
Beams	4.664	0.549	Reinforcement % in beams not detailed = 1.5% (6 beams)
Columns	4.8	0.034	
Slabs	67.25	384.3	
Walls	7.62	7.058	Additional reinforcement = 0.%
Misc. elements	7.26	0.569	Reinforcement % = 1.%
Footings	-	-	
Total	91.59	392.5	

** Beams/columns: quantities are for DEFINED beams/columns only
(All STRAP members are in beams or columns)
** Misc. elements: Elements not in Slabs and are not Wall elements
** Footings: designed using the 'File-Footing' option in STRAP Results

Part IX

Dynamic analysis

9 Dynamic analysis

This dynamic analysis module analyzes the modal shape of the model:

- solves for the natural frequencies and the corresponding mode shapes
- calculates the earthquake response and the resulting moments and forces in the model based on the calculated mode shapes and Code related factors
- calculates the time history response for forced vibrations.

The following tabs for dynamic analysis are displayed in the tab bar:

:

Weights

- define the dynamic nodal weights.
- calculate the mode shapes and natural frequency (after nodal weights have been defined).

Dynamics

- display the results of the mode shape and natural frequency calculation.
- carry out a seismic analysis on the model (frame structures).

Time his.

- calculate the transient (history) response of a model subject to dynamic loads in which viscous damping is present. This option enables the dynamic analysis of models subject to impact, impulse or cyclic loads or any other type of load that varies with time.

Note:

- The static analysis, dynamic analysis and seismic analysis must be carried out in the correct sequence. Refer to Seismic analysis - Procedure for more details.

9.1 Nodal weights

This dynamic analysis module analyses the modal shape of the model. The program solves for the natural frequencies and the corresponding mode shapes.

- The mode shape of the lowest frequency (longest period) will be numbered 1 and all the others will be numbered respectively in ascending order.
- The program assumes a lumped-weight model, i.e. the weight of the model is concentrated at the nodes. The weights (***not mass***) applied to the model must be defined prior to the start of the solution. The weights are lumped at the nodes.
- The first time the dynamic analysis is run for a model the weights are equal to zero.

Add weights	Define weights 991 at the model nodes.
Revise	Revise 993 the nodal weights
Self-weights	Define the self-weight 993 of the model as nodal weights.
Delete	Delete 994 nodal weights
Static load	Add loads from a static load 994 case to the nodal weight table
Mode shap...	Define the parameters for the mode shape 995 analysis calculation.
Submodel	Select a submodel instance to display.
Solve	calculate 1000 the mode shapes and the natural frequency of the model.

From the menu bar:

File Zoom Rotate Display REmove Output Help

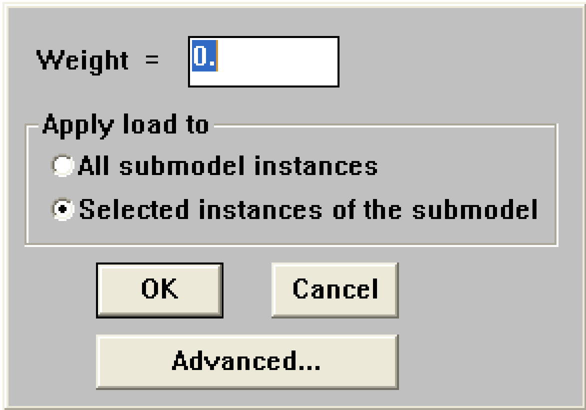
File 999
Start the calculation of the mode shapes and the natural frequency.

Output 1003
Display the nodal weights table.

Display 1002
Display the nodal weights graphically.

9.1.1 Add

Define a weight applied to any node in the model. Note that if a weight has already been defined at the selected node, the new weight will be ***added*** to the existing one.



Weight

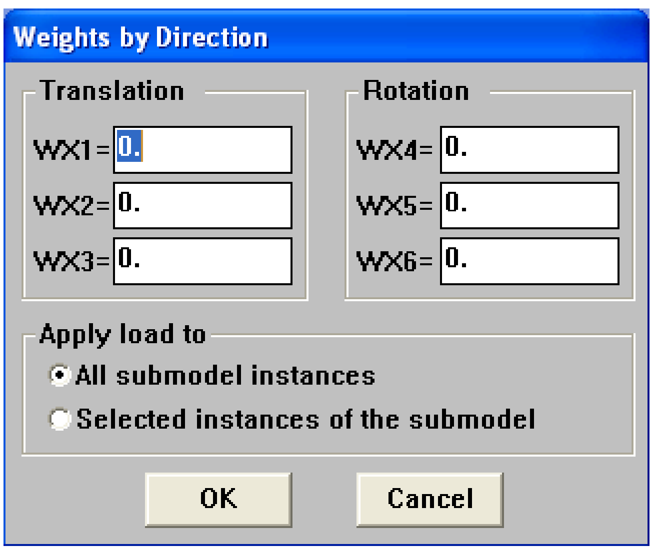
Define the weight in the current weight units. Select the nodes using the standard Node Selection option. Nodes in submodels can be selected.

Advanced

For most standard models the assumptions that the weight acts equally in all directions and that the weight is centered at the node give sufficiently accurate results.

In certain models a more refined definition of the weights may be required. The weight may not act equally in all directions (e.g. sliding supports) or the weight may be eccentric to the node (e.g. tributary area not centered at the node, machinery and bases, etc.)

Use this option to define different weights in different directions and/or rotation "mass moments of inertia":

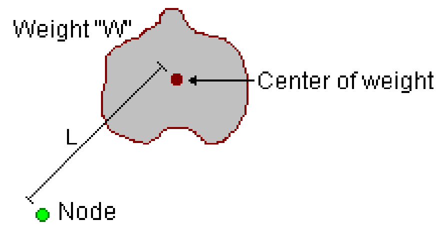


Mass moments of inertia are defined as follows:

- General mass transformed about a support node:

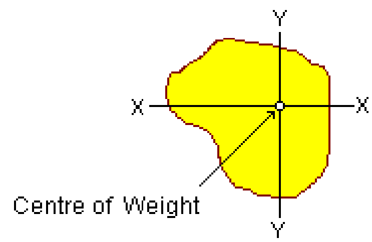
$(WX) = (WX)_o + WL^2$

where:
$(WX)_o$ = mass moment of inertia through centre of weight
L = projected distance in relevant global direction



The mass moment of inertia through centre of weight for various shapes:

- General thin plate:
  $(WX)_o = (W/A)(Ix + Iy)$
  where:
  Ix = moment-of-inertia about X
  Iy = moment-of-inertia about Y
  A = area



- Rectangular plate: dimensions **a,b**:
  $(WX)_o = (W/12)(a^2 + b^2)$

Submodels

This option is displayed when a submodel is currently displayed and there is more than one instance of it in the model. Select:

All submodel instances
the program applied loads now defined to all instances of the submodel currently displayed

Selected instances of the submodel
the loads are applied only to the instances selected from the list.

Note:
- nodal weights are applied ***only to the Main model nodes***. The weights can be defined on the submodel but the program calculates 'applied loads' at the connection points [387] and applies them to the main model.
  - For standard structures with submodels the dynamic results are very similar to those obtained from the same model without the submodels.
  - If the floor is linked with rigid links [204] the results are identical.

9.1.2 Revise

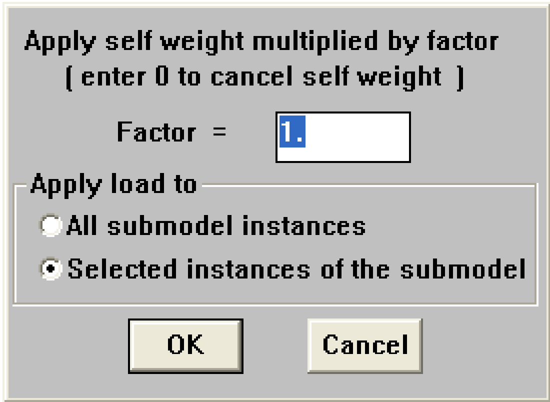
Revise a weight previously applied at at node.
- Select the nodes using the standard Node Selection option.
- Define a new value for the weight.

To revise self-weight, use the self-weight [993] option.

9.1.3 Self weight

Instruct the program to calculate the self-weight reactions of all beams and elements attached to specified nodes and to automatically apply them as nodal weights.

- Select the nodes using the standard Node Selection option.
- The calculated self-weight may be multiplied by a factor:



Entering a factor of zero deletes the self-weight.

Submodels

Refer to - Add - submodels 993

9.1.4 Delete

Delete nodal weights applied to selected nodes:

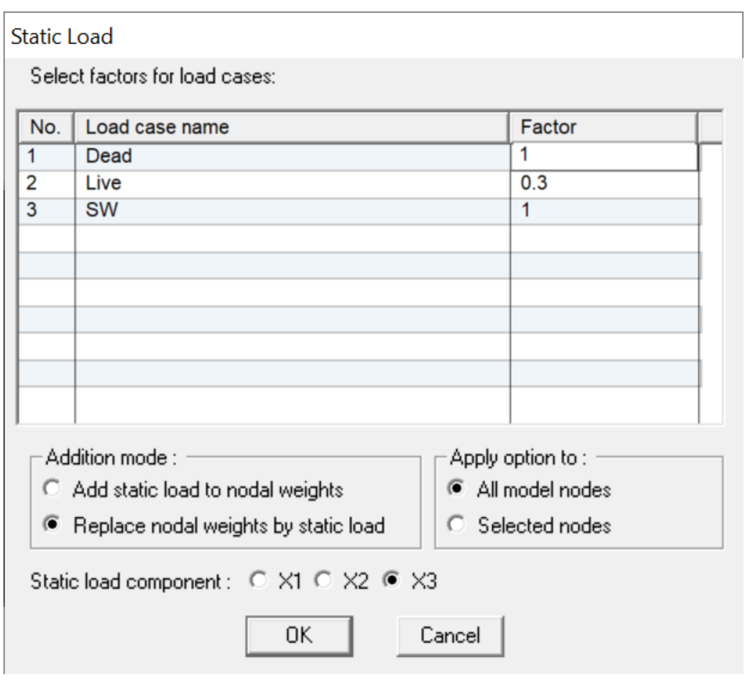
- select the nodes using the standard Node Selection option.

Note:

- use the self-weight 993 option to delete self-weight.

9.1.5 Static loads

Convert a static load case to nodal weights table.



Factor

Insert a factor for any load case from the table. The program will calculate the nodal weight which then multiplied by the factor.

Addition mode

Select one of the following:

Add static loads to nodal weights
The specified static load case will be added to the nodal weights already defined.

Replace nodal weights by static load
All defined nodal weights will be deleted and the specified staic load case will be added to the table.

Apply option to:

All model nodes
the selected static loads are applied to all relevant nodes in the model.

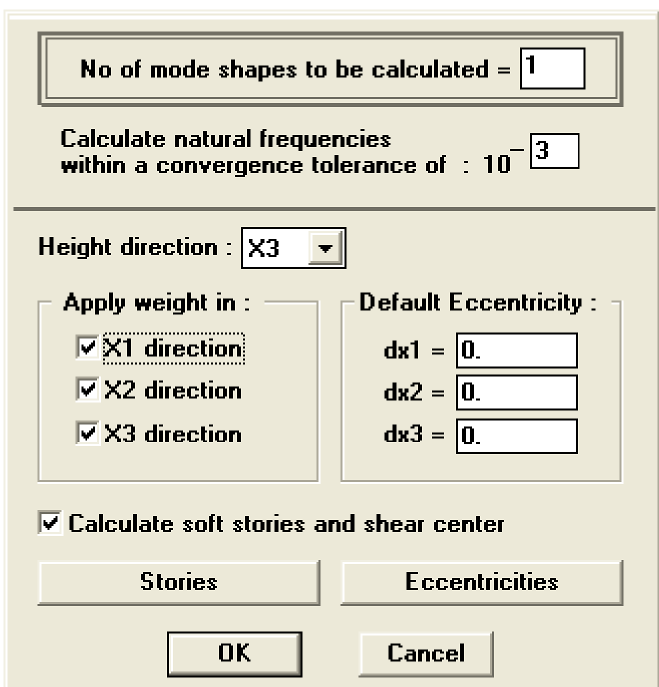
Selected nodes
Select nodes using the standard Node Selection option.

Static load component

The selected load case may contain loads acting in more than one global direction. Select the load direction that will be added to/replace the nodal weights.

9.1.6 Modes

Define the parameters for the mode shape and natural frequency calculation:



No. of mode shapes

The number of mode shapes equals the number of dynamic degrees-of-freedom with applied weights. For most structural models only the first few modes are important. Specify the number of mode shapes to be calculated.

The number of requested mode shapes should not exceed the following:
- 1000 mode shapes
- number of degrees-of-freedom with non-zero weight

For most structural models only the first few mode shapes are needed (up to five). Notice that the solution time increases with the number of mode shapes requested.

Tolerance

The program solves the eigenvalue equation by the "Subspace Iteration" method; the program bases the calculation of the eigenvalues in the current iteration on the eigenvalues of the previous iteration. When the previous and current values converge, the program terminates the calculation.

The measure of the difference between the values is called the "tolerance".

$$\text{Tolerance} = \frac{(\text{current - previous})\ \text{eigenvalue}}{\text{current eigenvalue}}$$

A stricter tolerance limit will increase accuracy but also increase the number of iterations required.

A tolerance of 1.E-3 (.001) is the program default value. A reduced tolerance exponent will significantly decrease the solution time. In larger models, the user may reduce the tolerance exponent according to his engineering judgment.

Apply weight in

Eliminate/add the weight effect in any of the global directions.

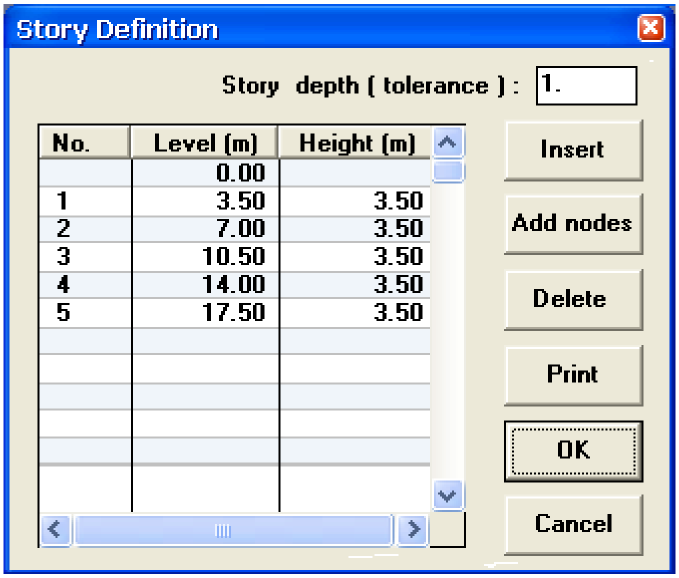
Default eccentricity

Many seismic design codes stipulate that the weights be applied offset from their center of mass, i.e. at a specified distance from the nodes. For example, UBC 1630.6: "...*the mass at each level shall be assumed to be displaced from the calculated center of mass in each direction a distance equal to 5 percent of the building dimension* ...".

Different "sets" of eccentricities may be defined and each "set" may be assigned to a different direction. ***This option defines the default eccentricities only for set #1***:
- define the default eccentricity (offset) of ***all*** nodal weights in the model from their nodes, where **dx1**, **dx2**, **dx3** refer to the X1, X2, X3 global axes. The default values are assigned to all levels
- select the Eccentricities option to define different eccentricities for specific levels, or to define different sets of eccentricities.

Stories

Eccentricities are defined at the story levels. Use this option to specify the levels where the eccentricities will be defined. The program initially lists all levels in the **Height direction**; use this option to add, remove or modify levels:



Select:

Insert	to add a new row to the table; enter the elevation value.
Add nodes	add new rows to the table by selecting nodes in the model
Delete	click and highlight a row in the table and click the **Delete** button; the row will be erased
Cancel	to cancel all changes to the table.

Eccentricities

Many seismic design codes stipulate that the weights be applied offset from their center of mass, i.e. at a specified distance from the nodes.

- define different "sets" of eccentricities and assign each set to an earthquake direction.

For each set:

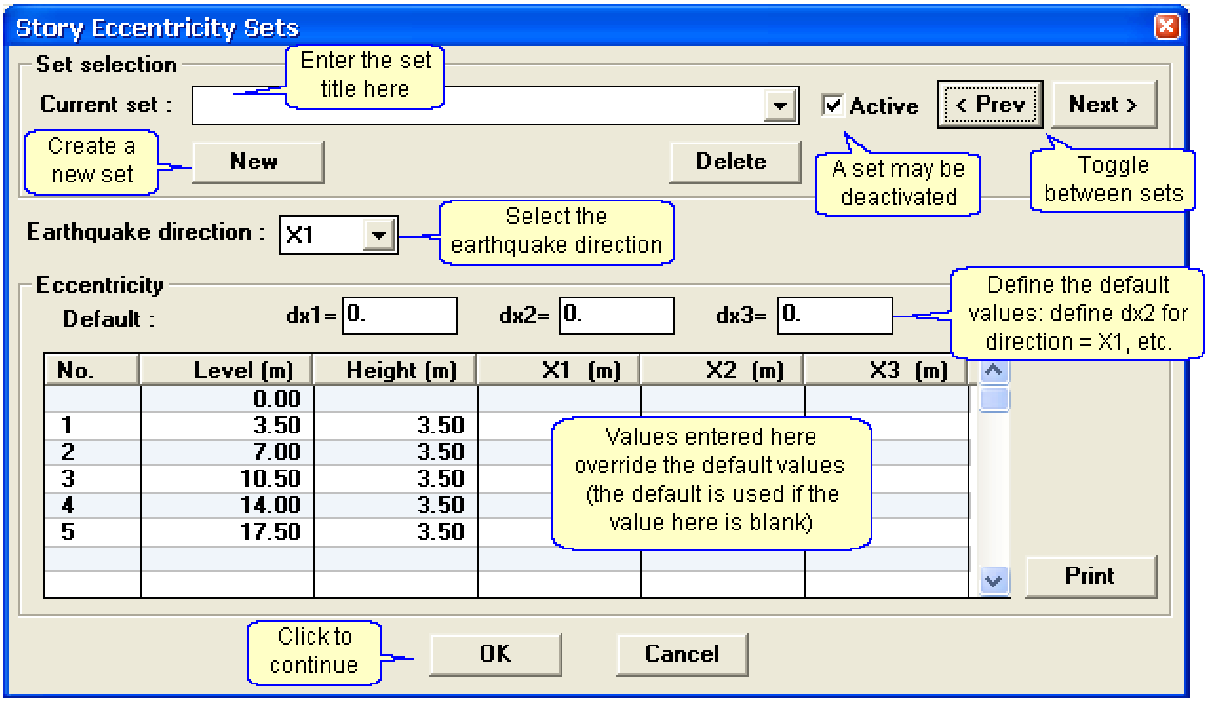
- define default values that are assigned to every defined set.
- define different eccentricities at specific levels in the model; values defined here override the default values 996.

Note:

- all sets are solved at the same time
- the levels are defined in the Stories 996 option.

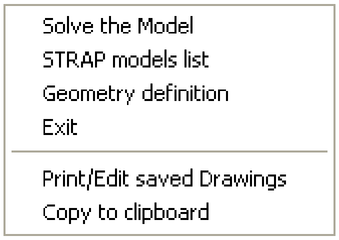
For example, the following sets are required for a typical model:

- +dx1 eccentricity, X2 earthquake direction
- -dx1 eccentricity, X2 earthquake direction
- +dx2 eccentricity, X1 earthquake direction
- -dx2 eccentricity, X1 earthquake direction



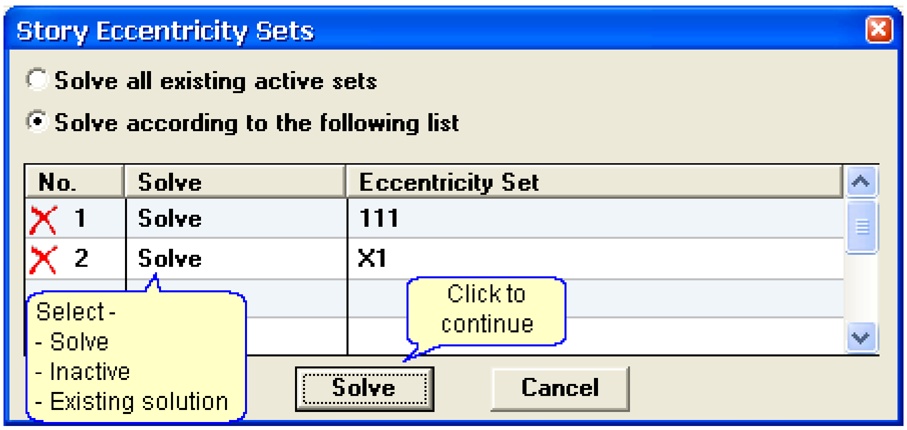
Enter values of eccentricity in any global direction at any level; the program will apply the eccentricity to the weights at all nodes at that level. Note that the default eccentricity will be used when a value is left blank.

9.2 File options



Solve the model

The program can solve all eccentricity "sets" simultaneously or only selected sets:



Note:

- "Inactive" means that the set will not be solved ***and*** existing results will not be displayed in the output tables.
- "Existing results" means that the set will not be solved but the existing results will be displayed in the output tables.
- Refer to Mode shape analysis 1000 for more details on the solution method.

STRAP models list

Return to the *STRAP* main menu

Geometry definition

Return to the *STRAP* geometry module

Exit

Exit STRAP and save weight data

Print/edit saved drawings

Refer to Print/edit drawings

Copy to clipboard

Copy the current graphic display to the clipboard.

9.2.1 Mode shape analysis

This dynamic analysis module analyses the modal shape of the model. The program solves for the natural frequencies and the corresponding mode shapes.

The program solves the problem of undamped free vibrations. This involves the solution of the generalized eigenvalue equation:

$$\mathbf{K}\,\phi = \mathbf{M}\,\phi\,\Omega^2$$

where:

K = stiffness matrix
M = mass matrix
Ω = eigenvalue matrix
ϕ = corresponding eigenvector matrix

The eigenvalues correspond to the natural frequencies by the following equations:

- eigenvalue = ϖ^2
- natural frequency = $\varpi/2\pi$

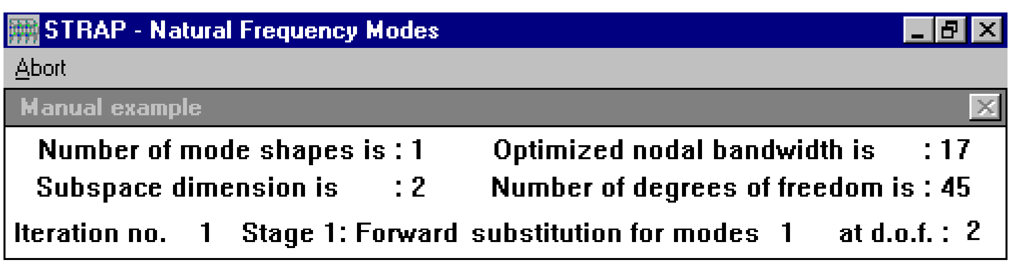
Each value of the eigenvector is the relative displacement of the corresponding degree-of-freedom.

- The mode shape of the lowest frequency (longest period) will be numbered 1 and all the others will be numbered respectively in ascending order.
- The program assumes a lumped-mass model, i.e. the mass of the model is concentrated at the nodes.
- The program solves the general eigenvalue problem using the Subspace Iteration method. Explained simply, the program bases the calculation of the eigenvalues in the current iteration on the eigenvalues of the previous iteration. When the difference between the previous and current values is very small, the program terminates the iteration process.

Each iteration contains three stages and the progress is displayed on the screen:

Stage 1:

For each d.o.f the program solves for all mode shapes requested. This stage takes up the most of the solution time.



Stage 2:

Subspace Iterations: The program solves the eigenvalue problem in a reduced subspace:

Stage 3:

The program calculates the tolerance and the eigenvectors for the next iteration (if required). The program displays the eigenvalues for the current iteration and the tolerances. The tolerance values reflect the rate of convergence and allow a rough estimates of the solution time.

Iteration no. 4	Stage 3: New eigenvectors approximation				
Current eigenvalues and relative tolerances:					
No.	Eigenvalue	Tol.	No.	Eigenvalue	Tol.
1	.1356772E+03	.827E-06			

If the specified tolerance is met, the program lists the eigenvalues:

Last iteration: 4	The calculated eigenvalues are:				
No.	Eigenvalue	Tol.	No.	Eigenvalue	Tol.
1	.1356772E+03	.827E-06			

The solution is automatically backed-up every iteration; select **Abort** in the menu bar to interrupt the calculation.

If you then select **Dynamics** in the ***STRAP*** main menu and **Solve the model** the program asks:

Solution was previously interrupted
after completion of iteration no. 1

You may either continue from next
iteration or start from the beginning.

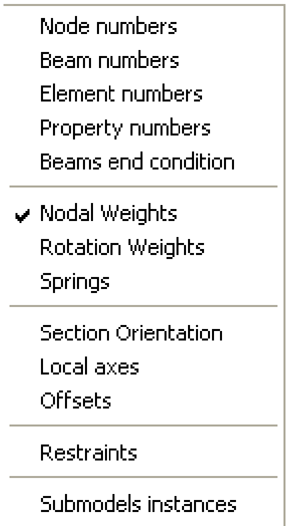
Continue   Restart

Select:

Continue - resume the solution from the point of interruption
Restart - restart the solution from the beginning.

This message also appears if the solution was interrupted by a power failure, computer malfunction, etc.

9.3 Display options



Nodal/rotation weights

Select:

- **Nodal weights** to graphically display the total weight (additional + self-weight) applied to the nodes.
- **Rotation weights** to graphically display the rotation weight) applied to the nodes.

9.4 Output options

Display Applied weights

Print applied Weights

Print Drawing

Applied weights

NODAL WEIGHTS (Units: ton)

Exit Goto Print Copy

Node	Total Weight	Additional Weight	Self Weight Factor
24	9.020	3.500	2.000
25	7.220	3.500	2.000
26	9.140	3.500	2.000
27	9.140	3.500	2.000
28	9.140	3.500	2.000
29	9.140	3.500	2.000
30	7.220	3.500	2.000

Total: 583.600 Eccentricity : (1.,2.,0.)

where:

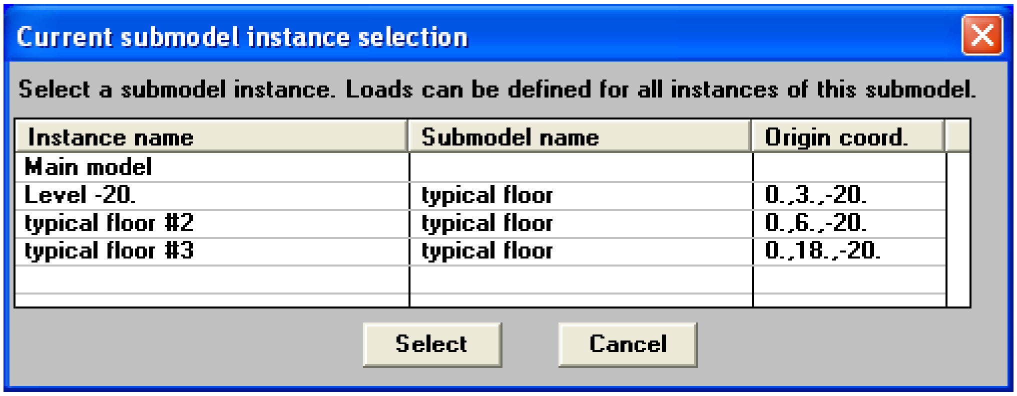
Total weight = sum of additional weights and self-weight applied at the node.
Additional weight = weights other than self-weight applied at the node.
Self weight factor = factor by which self-weight is multiplied
Eccentricity = eccentricity of ***all*** weights in the X1, X2, X3 global directions

Note:

- self-weight = (total weight - additional weight)/factor

9.5 Submodel

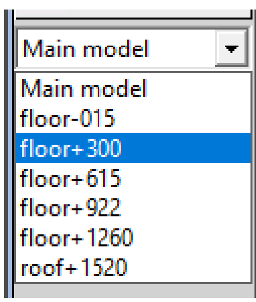
Display an existing submodel instance (or the main model):



Double-click the appropriate line or highlight the line and click Select.

Note:

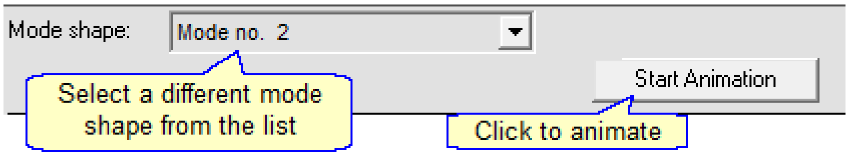
- loads applied to one instance of a submodel may be applied at the same time to all other instances of the same submodel.
- alternatively, select the submodel instance in the small list box:



9.6 Results/Seismic analysis

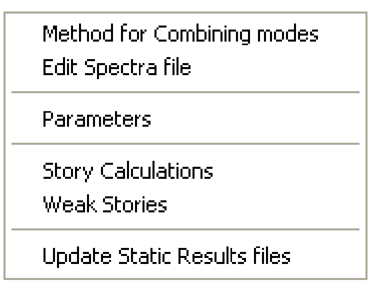
The results (deflections, moments, forces and stresses) for the Seismic Response Spectrum load case can be viewed in graphic or tabular form. These results may also be transferred to the ***STRAP*** results file in order to combine the dynamic results with static results for use by all design modules.

The model is initially drawn with "mode shape #1" superimposed on it. To select a different mode shape or to animate the current mode shape, select the options at the bottom of the screen:



Draw modes	Display mode shapes graphically (and animate them)
Print drawi...	Print the mode shapes graphically.
Display ta...	Display the tabular results for the mode shape analysis and seismic analysis.
Print tables	Print the tabular results for the mode shape analysis and seismic analysis.
Parameters	Specify the calculation parameters according to the relevant national seismic design code.
Story data	• check the drift (relative deflection between adjacent levels) according to the Code requirements. • display story shear forces, base shears and moments • calculate and display "stability coefficients" • identify and display "Weak stories" and "soft stories" as defined in the Code
Update re...	Add the dynamic results to the *STRAP* static results file.

These options are also available when Seismic analysis is selected in the toolbar:



For more information, refer to
Seismic analysis - General 1006.
Seismic analysis - Procedure 1006

9.6.1 General

This module calculates the earthquake response and the resulting moments and forces in the model based on the calculated mode shapes and Code related factors.

- The mode shape analysis calculates 'n' different mode shapes. The maximum response (deflection, base shear, etc.) for each shape is calculated from a "Response Spectrum". This spectrum is a graph which gives the acceleration as a function of the natural period, T, of the model.
- The spectrum may be an idealized one taken from a Code, e.g. Figure 1B in the SEAOC (California Blue Book) Code, or it may be based on ground motion histories at the specific site.
- When calculating the maximum response, the total response usually cannot be obtained simply by adding the maximum responses of the individual nodes because these maxima usually do not occur at the same time.

The user may select one of the following methods to estimate the maximum total response from the maximum calculated modal values;

- SRSS (Sum of Root of the Sum of the Squares) method.
- CQC (Complete Quadratic Combination)

In both methods the program calculates the response for each mode separately and then combines them according to a formula that accounts for the fact that when one mode achieves its maximum response, the responses of the other modes are less than their individual maxima.

Note :

- each maximum response is calculated separately, e.g. the maximum moments are calculated as the RSS/CQC of the moments from the individual mode shapes, and not as the moments resulting from the RSS/CQC deflections of the model.
- Refer to Method for combining modes 1007 for more information.
- For additional theoretical explanations and background, refer to any textbook covering dynamic response of multi degree-of-freedom systems.

9.6.2 Procedure

The general procedure is:

- Define the model geometry.
- If there are static loading cases, define and solve them ***before*** entering the dynamic Weights option**.**
- Select Weights and define the Nodal weights.
- Select 1+2= Solve
- Select Dynamics .
  - specify Parameters, then display the results by clicking the icons in the side menu.
  - select Update re... to append the dynamic load cases results to the ***STRAP*** static result file.

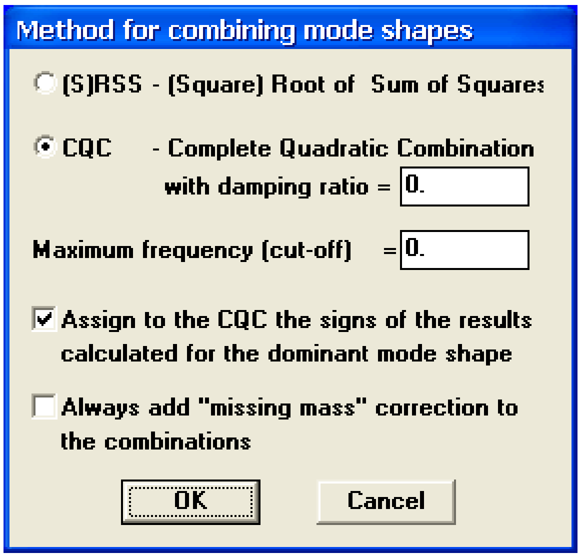
  Note: select each earthquake direction separately and transfer separately to the ***STRAP*** files.
- Select Results and then **Combinations** in the menu bar. Two combinations should be defined when combining dynamic and static results:
  a. (static results) * factor + (Seismic results) * factor
  b. (static results) * factor + (Seismic results) * factor * -1
- Select any of the options in the side menu for displaying or printing results.

Note:

- When the program adds the dynamic load case results to the static results file it assigns them the "**Use existing results**" parameter for the Loads - **Deactivate** option; the dynamic results will not erased if the static loads are solved again. If the dynamic load cases are "active" then solving the static loads again creates zero results for the dynamic cases.

9.6.3 Method for combining modes

Specify the method to estimate the total response from calculated modal values and the constant modal damping (ξ):



RSS/CQC

Specify the method to estimate the total response from calculated modal values and the constant modal damping (ξ):

- ***SRSS*** (square root of sum of squares)

  The estimated response R (force, displacement, etc) at a specified coordinate may be expressed as:

  $$R = \sqrt{\sum_{i=1}^{N} R_i^2}$$

  where R_i is the corresponding maximum response of the *i*th mode at the coordinate.

- ***CQC*** (complete quadratic combination)

  The estimated response may be expressed as:

  $$R = \sqrt{\sum_{i=1}^{N}\sum_{j=1}^{N} R_i \rho_{ij} R_j}$$

  where the cross-modal damping coefficient ρ may be approximated by:

  $$\rho_{ij} = \frac{8\xi^2(1+r)\, r^{3/2}}{(1-r^2)^2 + 4\xi^2 r(1+r)^2}$$

  where:
  $r = \omega_j/\omega_i$ = ratio of the natural frequencies of modes i and j
  ξ = the constant modal damping

Note:

- Application of the SRSS method generally provides an acceptable estimation of the total maximum response. However when some of the modes are closely spaced, the method may grossly underestimate or overestimate the maximum response. Large errors have been found in particular in space models in which the torsional effects are significant. The term "closely spaced" may be arbitrarily defined as the case where the difference between two natural frequencies is less than 10% of the smaller frequency.
- The CQC method is a more precise method of combining the maximum values of modal response.
- the two methods are identical for undamped models ($\xi = 0$).

Max. frequency cut-off

Instruct the program to ignore the mode shapes having a natural frequency greater than the specified value. The program will use all mode shapes if the value is set to 0.

Assign to CQC signs ...

The SRSS/CQC method calculates the maximum There are two options available:

☐ the moment diagrams are drawn entirely on one side of the member (i.e. single curvature, the critical case for column design. Therefore, *STRAP* transfers a negative moment at one end and a positive moment at the other (Refer to Sign conventions). All axial forces are positive.

☑ the default method: All results are transferred with the sign of the results calculated for the dominant mode shape, i.e. the mode shape with $(F_n)_{max}$

Missing mass

The "Missing mass correction" is a mathematical procedure to correct the results when not all mode shapes are used and hence **$\Sigma(W_n/W_{tot}) < 1.00$**.

9.6.4 Edit spectra file

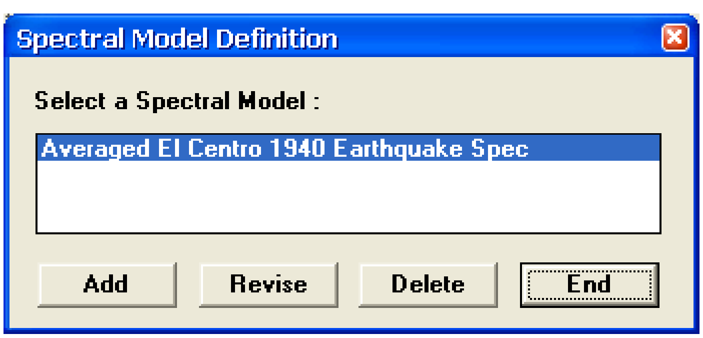
The program is supplied with the El Centro 1940 spectrum data. The user may add any spectrum other than El Centro by using this option. The spectra are stored in file UDAMPS.DAT 1010 (located in the program directory)

To edit/revise a spectrum:

- Select a spectrum 1008
- Select an accelerogram 1009
- Enter/revise the frequency and acceleration values 1009
- Add/revise the accelerogram 1010

Select spectrum

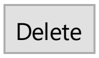
The program displays the following dialog box:



Select:

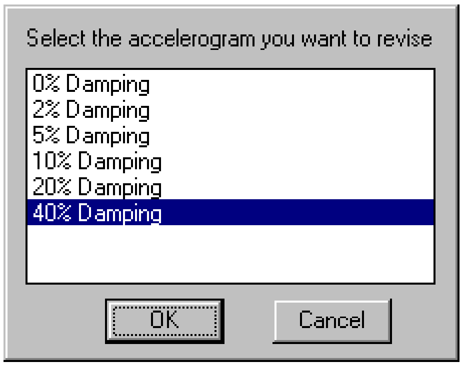
add a new spectrum.

revise one of the existing spectra. Click and highlight the line with the title.



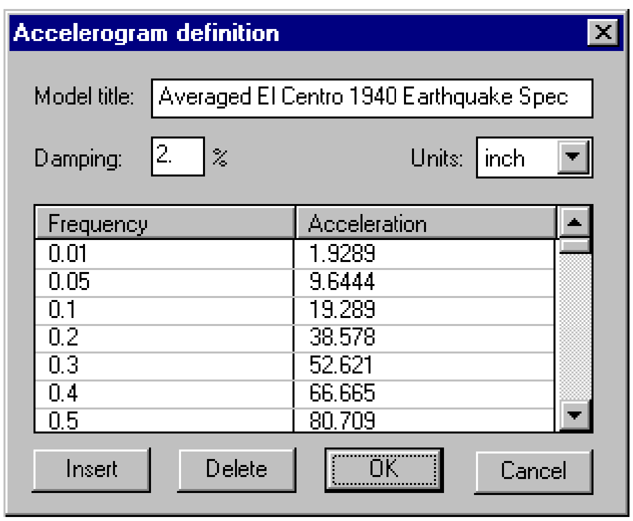
delete one of the existing spectra or to delete one of the accelerograms in the spectrum (corresponding to a specific damping). Click and highlight the line with the title.

Select accelerogram



To add a new accelerogram, select any existing value value and enter a new damping % in the next menu.

Enter/revise the frequency and acceleration values



- ***Damping***

  Enter a value for the damping (%). The value is used to create the title for the accelerogram.

- ***Units***

  Specify the length unit for the acceleration values (and the gravitational constant). For example, if you specify **inch,**
  - all acceleration values must be defined in in/sec²
  - the program will automatically use a gravitation constant g = 386.22 in/sec²

  If you select **None**, the program will use a value of 1.00 for 'g' (this value can be manually edited by the user - refer to Spectrum file1010).

- ***Frequency /acceleration***

Specify the frequency and the acceleration of each point on the accelerogram. The accelerations must be defined in the same units as the gravitational constant 'g'.

Press [Enter] to move from cell to cell, or point to a specific cell and click the left mouse button: Press -

Insert — insert a new line above the currently highlighted line. Note that the frequency values do not have to be entered in the correct order (the program will automatically rearrange the table after you click OK).

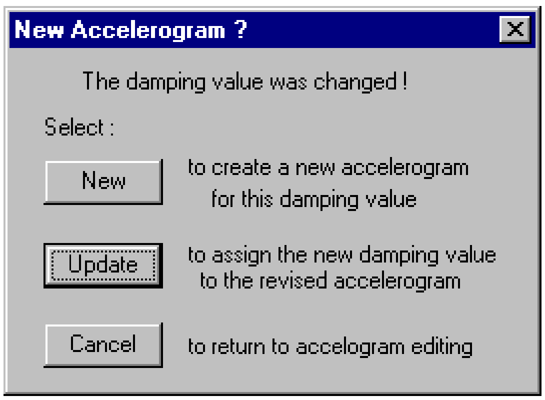
Delete — delete the currently highlighted line

OK — save the current changes and return to the previous menu.

Cancel — cancel all changes made in the current session and to return to the previous menu.

Add / revise accelerogram

Remember that an existing accelerogram is always selected:



Select:

New — The program adds a new accelerogram to the list (without modifying or erasing the existing ones).

Update — The program revises the selected accelerogram

Cancel — Cancel all changes and return to the previous menu

9.6.4.1 Spectrum file

The user may use any spectrum other than El Centro by appending the relevant data to the ASCII file already containing the El Centro information.

File name: **UDAMPS.DAT** (located in the program directory)

File format:

line 1:
Default gravitational constant 'g'.
Examples: for in/sec², enter 386.22047
for m/sec², enter 9.81
Note: the value is used only for accelerograms where a value for 'g' is not found in the title line (see 'ND lines' below).

line 2:

NS = the number of different spectra included in the file (maximum = 99 spectra)

NS title lines:
separate line for each spectrum containing the spectrum title (39 characters)

For each of the NS spectra:

line 1:
the spectrum number (1 to NS), preceded by a #, e.g. #1

line 2
ND = the number of different accelerograms in the spectrum, each for a different value of damping (maximum = 30).

ND title lines:
separate line for each accelerogram containing:

- a title (39 characters)
- gravitational constant, 'g' (starting in column 60). If not defined, the program uses the default value in line 1.

For each of the ND dampings:

line 1:
the spectrum number (1 to ND), preceded by a *, e.g. *1

line 2:
NP = the number of defined points on the accelerogram (maximum = 200 points).

NP lines:
The frequency and the acceleration of each point on the graph (free format). The accelerations must be defined in the same units as the gravitational constant 'g'.

The file as provided:

386.22047	-	gravitational constant
1	-	1 spectrum in file
Averaged El	-	El Centro Spectrum:
Centro 1940	-	title of one spectra
#1	-	6 damping accelerograms
6	-	titles of 6 damping accelerograms
0% Damping	-	"
2% Damping	-	"
5% Damping	-	"
10% Damping	-	"
20% Damping	-	"
40% Damping	-	1st accelerogram
***1**	-	contains 38 points
38	-	coordinate of 1st point (free format)
.010 2.02576		
etc.		

The program contains an option to revise/add the spectrum data, but the file can also be edited manually externally to *STRAP*.

Example:

- add a new spectrum titled "User Spectrum 1" to the file. Assume 2 damping factors are defined, each with 3 points per damping (note that the El Centro spectrum contains 38 points):

Revise the beginning of the file as follows:

386.22047
2
Averaged El Centro 1940
User Spectrum 1
#1
. . . . etc.

Append to the end of the file:

#2
2
5% Damping
10% Damping
***1**
3
.010 2.02576
.1 22.3
.8 10.2
***2**
3
.010 1.05
.1 18.3
.8 6.1

9.6.5 Parameters

Response spectrum 1012

SEAOC (1988) / UBC (1994)

UBC (1997)

ASCE 7

Eurocode 8

IS:1893 (2016)

IS:1893 (1984)

NBC of Canada

P100-1 (Romania) 1014

SNIP-II-7-81 1016

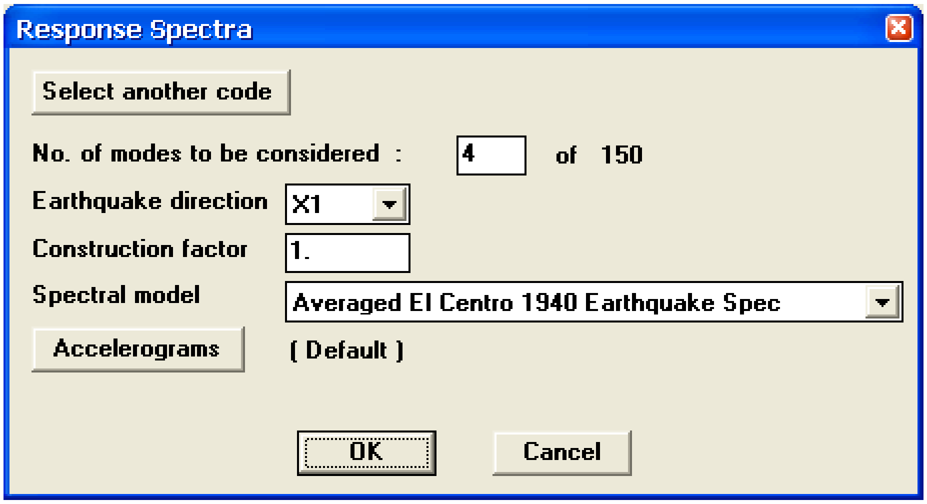
SNIP RK 2.03 1017

AS1170.4

Other codes

9.6.5.1 Response spectrum

The calculation is based on a specified response spectrum. The program contains one standard spectrum based on the El Centro 1940 earthquake. The user may define and specify any other spectrum; refer to Spectrum file 1010.



Code

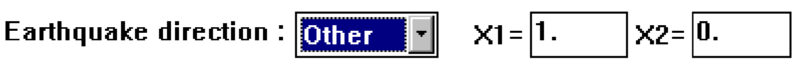
Select one of the design codes displayed in the pull-down menu.

No. of modes

Specify the number of mode shapes calculated in the Modal Shape Analysis program to be included in the Seismic analysis. (The higher mode shapes usually do not influence the results in standard models).

Earthquake direction

Specify the direction that the earthquake is applied. Select one of the global directions in the pull-down menu or select **Other** and define a vector as a combination of the three global directions:



The values serve only to define the direction of the vector and do not influence the intensity (the program will normalize the values so that the length of the resultant vector is unity). For example, **X1=1; X2=1** and **X1=2;X2=2** will give identical results.

Note that all mode shapes are used no matter in which direction the earthquake is applied. However the modes which have deflections in the direction that the earthquake is applied will dominate.

Construction factor

Specifies the intensity of the earthquake according to the code being used; the factor may be used to amplify the response spectrum or to include factors contained in local building codes reflecting structure type, soil factor, behaviour factor, etc.

Spectral model

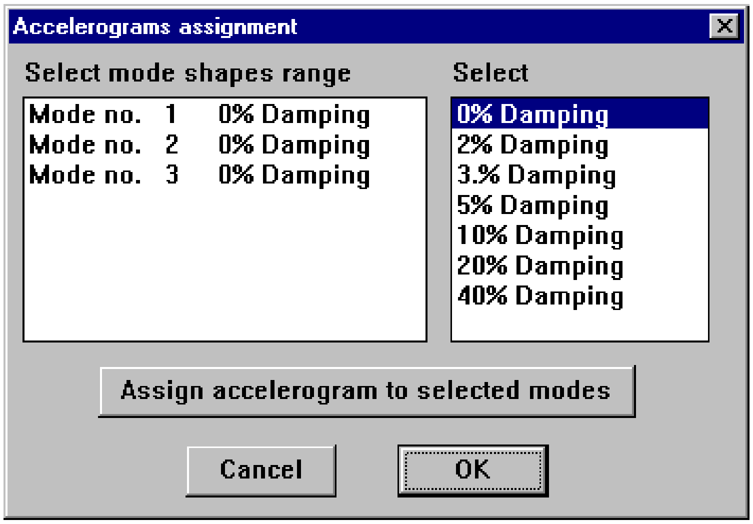
To select the response spectrum to be used for the calculation. The program contains the El Centro 1940 earthquake response spectrum and the user may define additional spectra. (refer to Spectrum file 1010). Select a spectrum from the pull-down list.

Accelerograms

Specify the damping percentage in the response spectrum to be used. Each mode shape may be assigned with a different damping percentage as damping is often greater in the second or higher modes.

By default, all mode shapes are assigned to the first damping value in the spectrum.

For the El Centro earthquake:



The different damping values contained in the spectrum file are listed in the right table; The damping values assigned to each mode shape are listed in the left table.

To modify the table:

- click the mode shape in the left table (you can drag the for multiple selections.
- click the mode shape in the right table.
- click the **Assign damping factor to selected modes** button to update the left table.

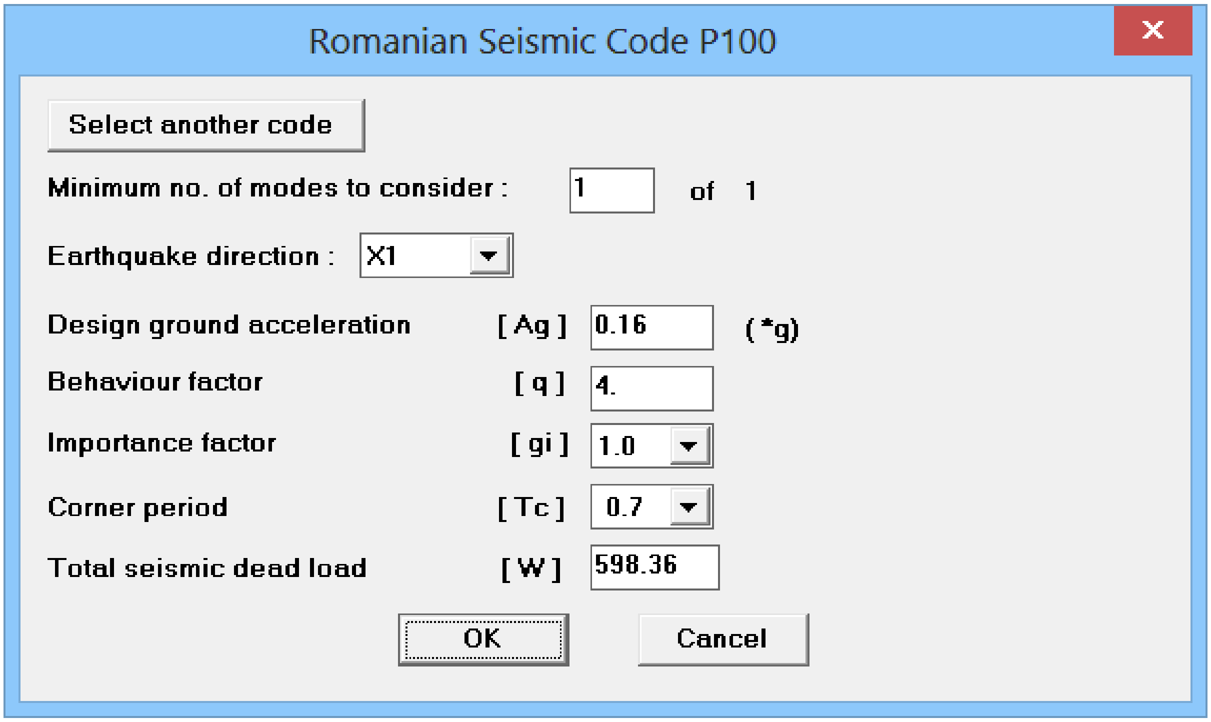
9.6.5.2 Miscellaneous codes

9.6.5.2.1 P100-1

This option calculates the seismic response according to P100 (Romania).

Two versions are available - 2006 and 2013. To change the version select Files - Code version selection 1034.

Define the required factors:



Code

Select one of the design codes displayed in the pull-down menu.

No. of modes

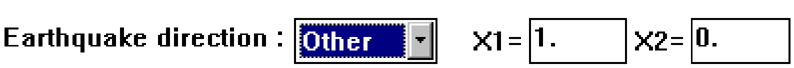
Specify the minimum number of mode shapes to be used in the seismic analysis. (Higher mode shapes usually influence the results only slightly in standard models).

Note that the program also calculates the significant number of modes, N, according to Section 4.5.3.3.1(8) in the 2013 Code [4.5.3.3 (8) in the 2006 Code] - "the effective modal masses for the modes considered amount to at least 90% of the total mass of the structure".

The program uses the ***maximum*** of the 'significant' no. of modes and the value entered here.

Earthquake direction

Specify the direction that the earthquake is applied. Select one of the global directions in the pull-down menu or select **Other** and define a vector as a combination of the three global directions:



The values serve only to define the direction of the vector and do not influence the intensity (the program will normalize the values so that the length of the resultant vector is unity). For example, **X1=1; X2=1** and **X1=2;X2=2** will give identical results.

Note that all mode shapes are used no matter in which direction the earthquake is applied. However the modes which have deflections in the direction that the earthquake is applied will dominate.

Design ground acceleration

Specify $\mathbf{a_g}$, the design ground acceleration, according to Figure 3.1 or Table A1 (2013 Code) / Table A.6 (2006 Code).

Behaviour factor

Specify the behaviour factor, **q** where:

- Concrete - refer to Section 5.2.2.2 - Table 5.1.
- Steel - refer to Section 6.3.2 (2013 Code) / Section 6.3 (2006 Code) - Table 6.3

Importance factor

Specify the Importance factor corresponding to the Importance classes listed in Section 4.4.5, Table 4.2 (2013 Code) / Section 4.4.4.5, Table 4.3. (2006 Code)

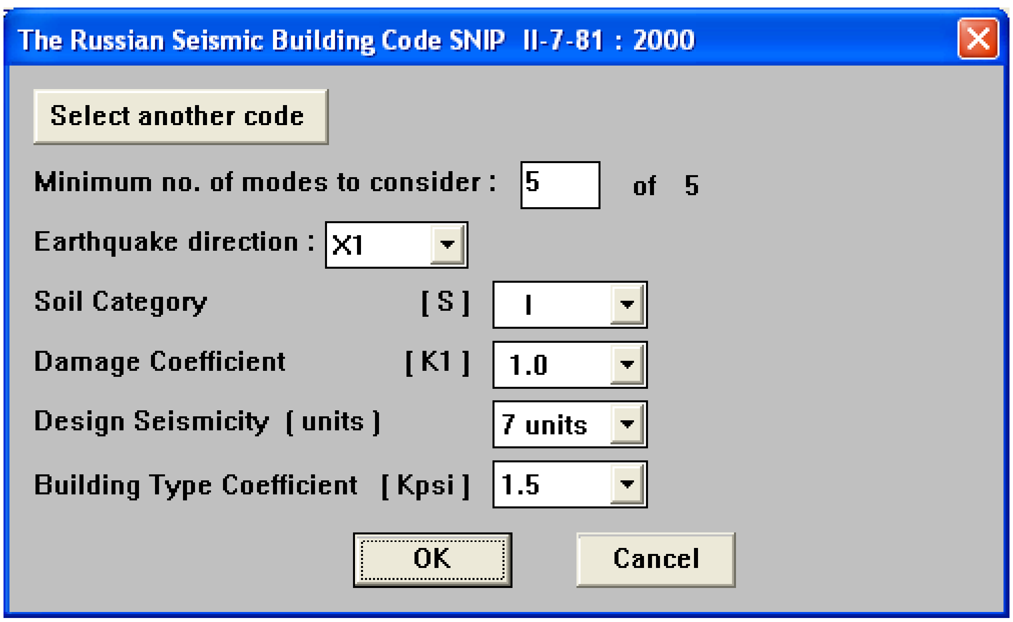
Corner period

Specify $\mathbf{T_c}$, the corner period at the upper limit of the constant acceleration region of the elastic spectrum. Refer to Figure 3.2 and Table A1 (2013 Code) / Table A.6 (2006 Code) .

9.6.5.2.2 SNIP-II-7-81

This option calculates the seismic response according to SNIP-II-7-81 (Russia).

Define the required factors:



Code

Select one of the design codes displayed in the pull-down menu.

No. of modes

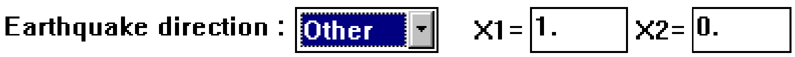
Specify the minimum number of mode shapes to be used in the seismic analysis. (Higher mode shapes usually influence the results only slightly in standard models).

Note that the program also calculates the 'significant' number of modes, N, required so that the effective modal masses for the modes considered amount to at least 90% of the total mass of the structure (as recommended by other Codes).

The program uses the ***maximum*** of the 'significant' no. of modes and the value entered here.

Earthquake direction

Specify the direction that the earthquake is applied. Select one of the global directions in the pull-down menu or select **Other** and define a vector as a combination of the three global directions:



The values serve only to define the direction of the vector and do not influence the intensity (the program will normalize the values so that the length of the resultant vector is unity). For example, **X1=1; X2=1** and **X1=2;X2=2** will give identical results.

Note that all mode shapes are used no matter in which direction the earthquake is applied. However the modes which have deflections in the direction that the earthquake is applied will dominate.

Soil category

Select the soil category, S, (I, II or III) according to Table 1*.

Damage coefficient - K1

A coefficient that reflects the allowable damage to a structure. Refer to Table 3 in the Code.

Design seismicity

Select the Seismicity, S_c, of the building area (7-9 units) according to the General Seismic Zoning table in the code..

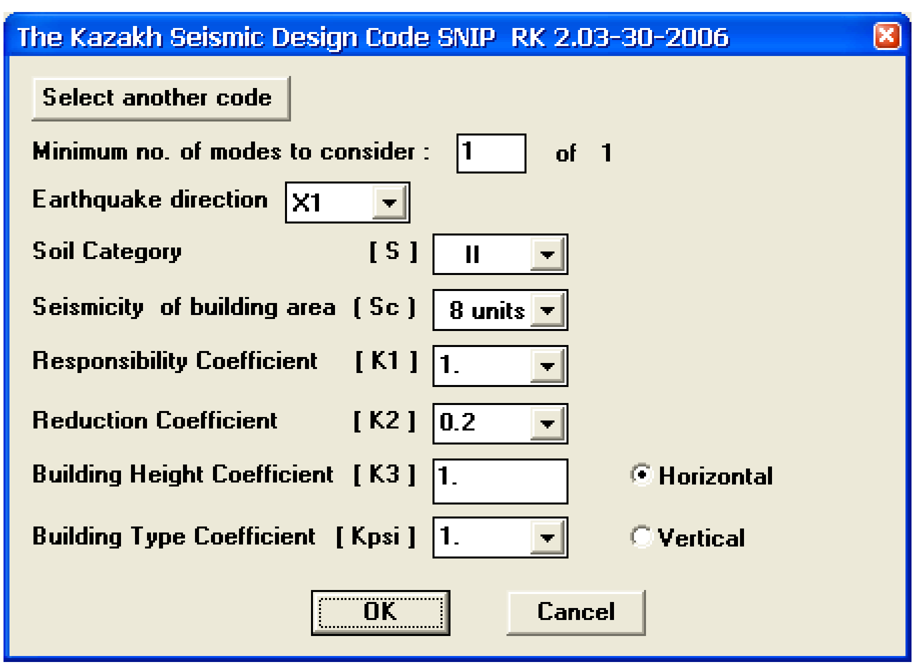
Building type coefficient

Select the Building type coefficient according to Table 6*.

9.6.5.2.3 SNIP RK 2.03

This option calculates the seismic response according to SNIP RK 2.03-30-2006 (Kazakhstan).

Define the required factors:



Code

Select one of the design codes displayed in the pull-down menu.

No. of modes

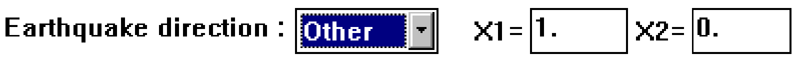
Specify the minimum number of mode shapes to be used in the seismic analysis. (Higher mode shapes usually influence the results only slightly in standard models).

Note that the program also calculates the significant number of modes, N, according to Section 5.17 in the Code - "the effective modal masses for the modes considered amount to at least 90% of the total mass of the structure".

The program uses the ***maximum*** of the 'significant' no. of modes and the value entered here.

Earthquake direction

Specify the direction that the earthquake is applied. Select one of the global directions in the pull-down menu or select **Other** and define a vector as a combination of the three global directions:



The values serve only to define the direction of the vector and do not influence the intensity (the program will normalize the values so that the length of the resultant vector is unity). For example, **X1=1; X2=1** and **X1=2;X2=2** will give identical results.

Note that all mode shapes are used no matter in which direction the earthquake is applied. However the modes which have deflections in the direction that the earthquake is applied will dominate.

Soil category

Select the soil category (I, II or III) according to Table 4.1.

Seismicity of building area

Select the Seismicity, S_c, of the building area (7-10 units) according to the map in Appendix 3.

Responsibility coefficient

Select the Responsibility coefficient, k_1, according to Table 5.2.

Reduction coefficient

Select the reduction coefficient, k_2, according to section 5.11, Table 5.3 and Table 5.4.

Building height coefficient

Enter the building height coefficient, k_3, calculated according to Equation (5.3).

Building type coefficient

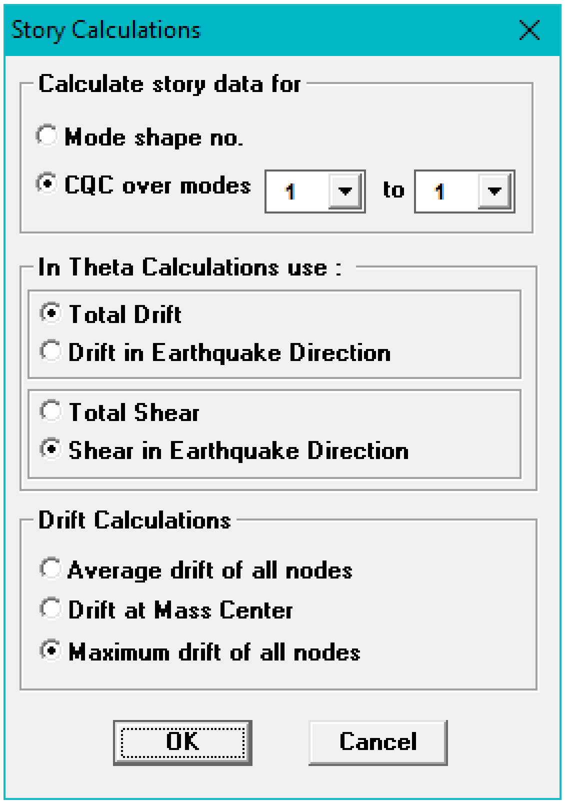
Select the Building type coefficient according to Table 5.7.

Horizontal / vertical

Specify the direction of the seismic action (refer to Table 5.5).

9.6.6 Story calculations

Check the drift (relative deflection between adjacent levels) according to the Code requirements or a user-defined limit.

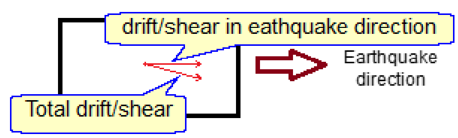


Calculate story data for

The drift may be calculated for RSS/CQC (range of mode shapes) or a specific mode shape:

In theta calculations use:

Calculate the story drift angle, $\boldsymbol{\theta = W \cdot \Delta_{el',i} \cdot K/(V_i \cdot h_{i})}$, using -



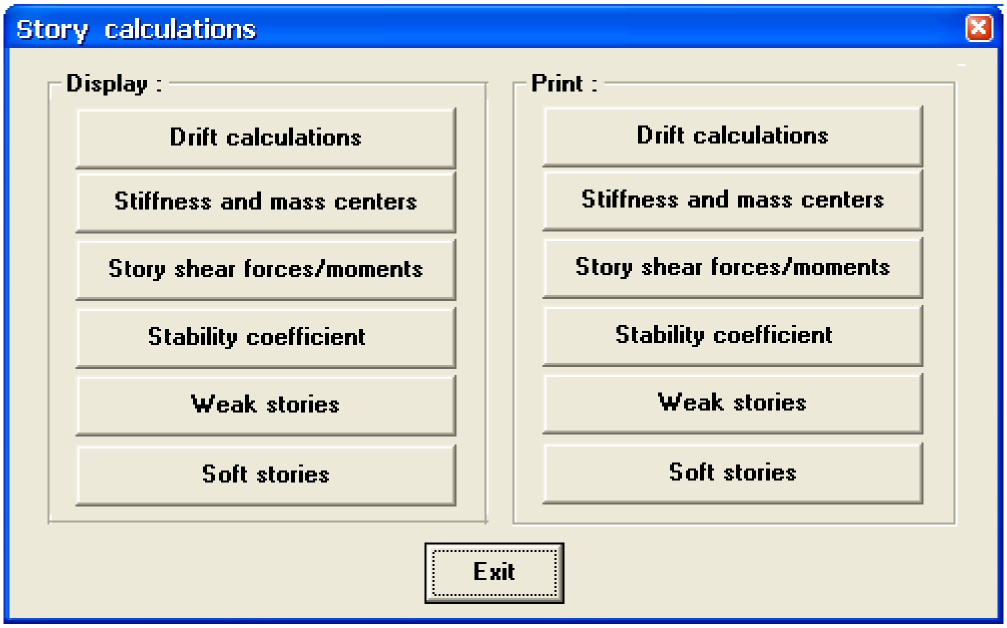
- total drift / total shear
- drift/shear component in earthquake direction

Drift calculations

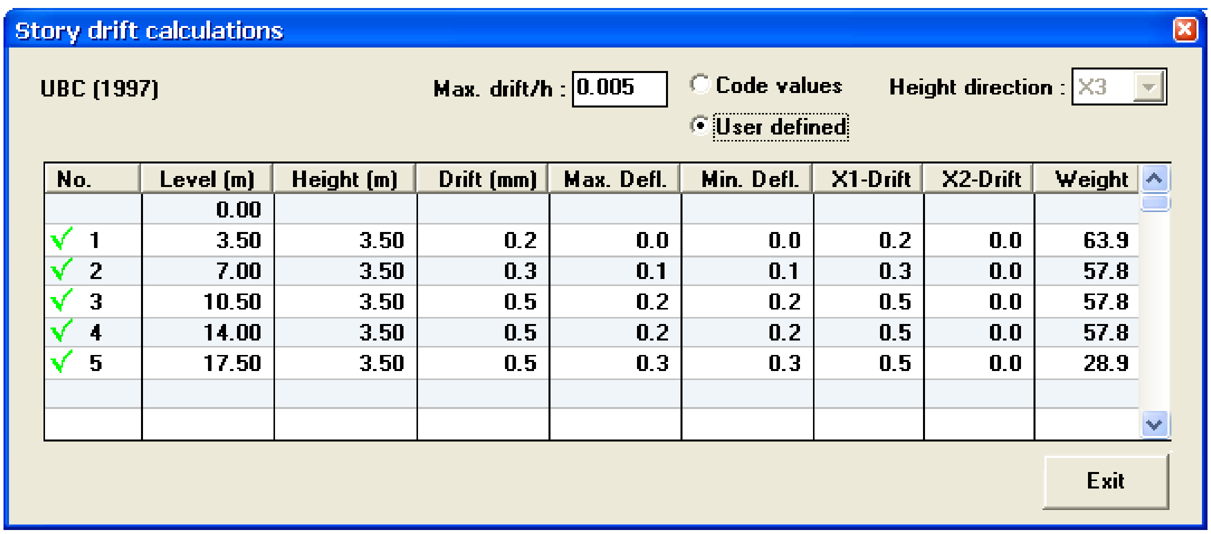
The story drift can be calculated as -
- the average drift of all nodes in the story
- the drift at the center-of-mass (all defined masses)
- the maximum drift among all nodes in the story

Note:
- EC8, ASCE/SEI7-10 and SP14.13333: these Codes specify "maximum drift" and the option is not displayed.

Calculate and display the following tables:



Drift calculations



The columns in the table are:

No. - Level number:
✓ indicates that the drift is less than the user-defined or code drift limits
X indicates that the drift exceeds the user-defined or Code drift limits

Level - coordinate along the height axis

Height - storey height; dimension between adjacent levels

Drift - drift value calculated according to Code equations, including all modification factors, etc.

Defl - min and max deflections at the level, calculated from deflection values at all nodes at that level (the values may not be equal because of the rotation of the model).

Weight - story weight

The program calculates the drift as follows:

Eurocode 8: - The interstorey drift, d_t, is calculated as the difference of the average lateral

Russia: displacements at the top and bottom of the storey. (4.4.2-6)

ASCE 7: - The design story drift is computed as the difference of the deflections at the centers-of-mass at the top and bottom of the story. (12.8.6)

All other codes: - The program calculates the drift for every pair of corresponding nodes at the top and bottom of the story and uses the ***maximum*** value.

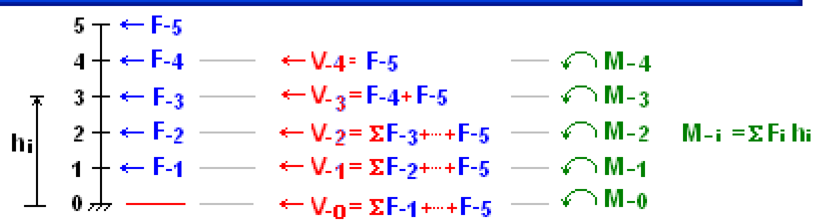
Story shear forces

Display the horizontal shear force applied at each level, the cumulative shear force and the cumulative moment. For example:

SHEAR FORCES/MOMENTS (Units: ton)

Exit Goto Print Copy

No.	Level	Story forces		Base shear		Story moments	
		F1	F2	V1	V2	M2	M1
0	0.00			6.32	0.04	80.41	0.52
1	3.50	0.27	0.00	6.05	0.04	58.29	0.39
2	7.00	0.77	0.00	5.28	0.04	37.11	0.25
3	10.50	1.48	0.01	3.80	0.03	18.64	0.13
4	14.00	2.27	0.02	1.53	0.01	5.35	0.04
5	17.50	1.53	0.01				



Stiffness and mass centers

The program calculates the centre of mass and the centre of rigidity for each floor (storey). The distance between the centres is also displayed.

Note:

- The centre of rigidity of a all stories are calculated, even if rigid links do not connect all of the nodes in its plane.
- The centre of mass is calculated from the weights defined in the program and ***not*** from the density of the elements

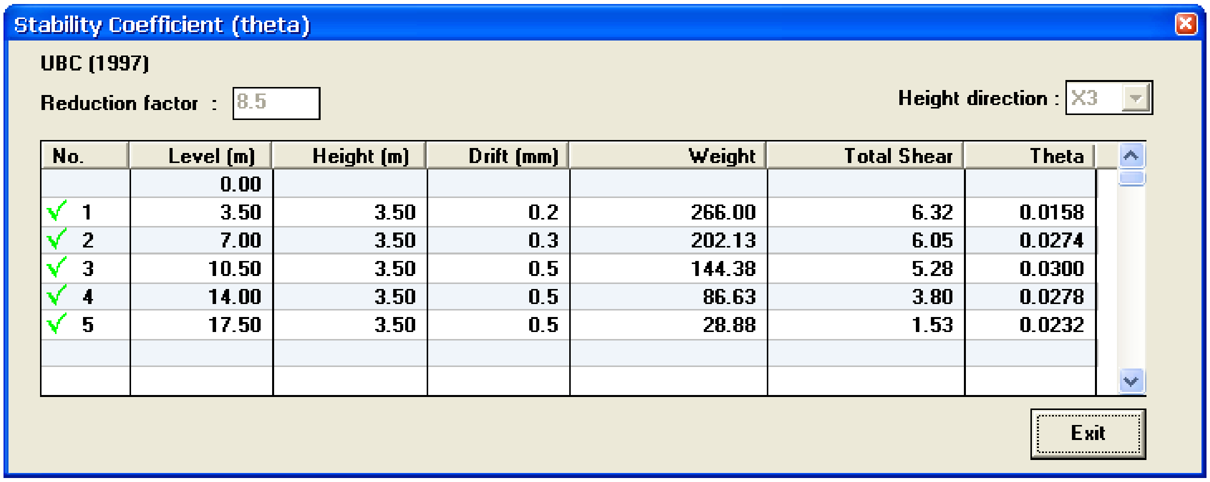
Stability coefficient

The program calculates and displays the "Stability coefficient *θ*". Most codes have a lower and upper limits for *θ:*

- A second order analysis is not required below the lower limit.
- A second order analysis is required between the lower and upper limits.
- Values greater than the upper limit are not allowed.

For example , ASCE 7-10, 12.8.7: "*P-Delta effects on story shears and moments .... are not required to be considered where the stability coefficient (θ) .. is equal to or less than 0.10*"

Example:



No.	Level (m)	Height (m)	Drift (mm)	Weight	Total Shear	Theta
	0.00					
✓ 1	3.50	3.50	0.2	266.00	6.32	0.0158
✓ 2	7.00	3.50	0.3	202.13	6.05	0.0274
✓ 3	10.50	3.50	0.5	144.38	5.28	0.0300
✓ 4	14.00	3.50	0.5	86.63	3.80	0.0278
✓ 5	17.50	3.50	0.5	28.88	1.53	0.0232

The program displays one of the following three symbols at the left side of the table:

✓ - θ less than lower limit (2nd order analysis not required).

⚠ - θ between lower and upper limits (2nd order analysis required)

X - θ above upper limit (not allowed by Code).

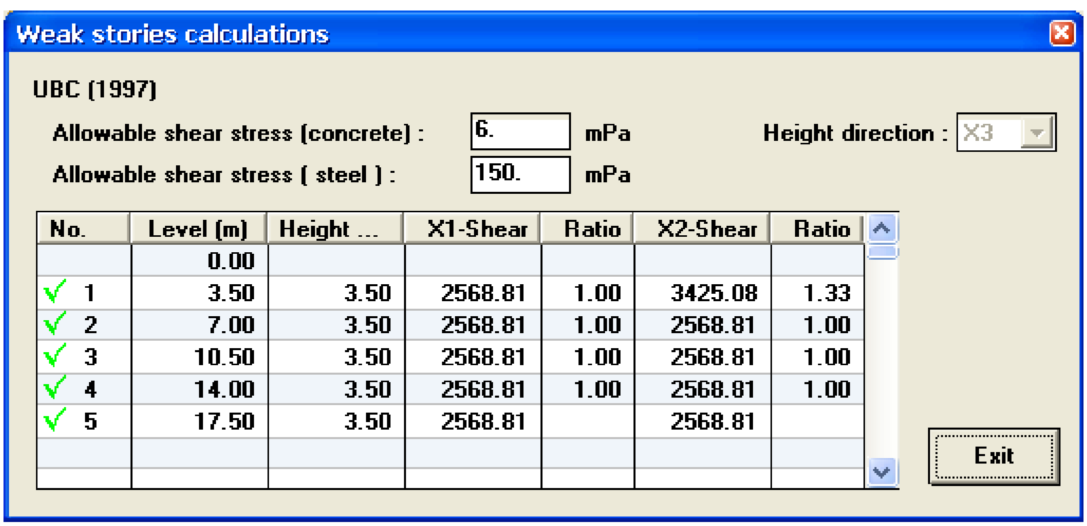
Weak stories

The program identifies "Weak stories" as defined in the Code. For example, UBC-1997, Table 16-L - "*A weak story is one in which the story strength is less than 80% of that in the story above*".

The shear strength is calculated as the sum of the shear capacity of all concrete walls and columns as well as structural steel sections in the direction being considered, i.e.

Shear = (Σconcrete area)* (allowable concrete shear stress) + (Σsteel area)* (allowable steel shear stress)

where the allowable shear stresses are specified in the menu.



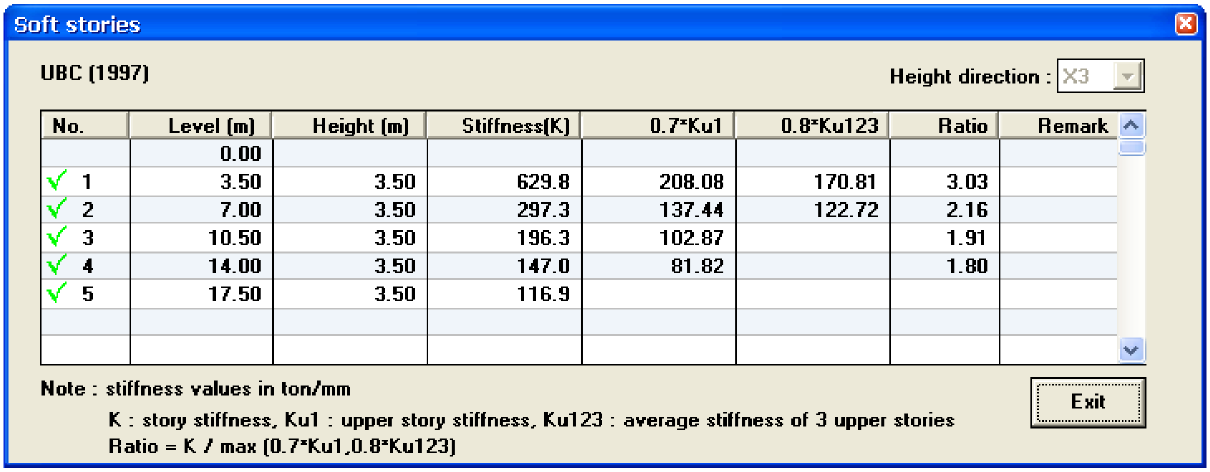
No.	Level (m)	Height ...	X1-Shear	Ratio	X2-Shear	Ratio
	0.00					
✓ 1	3.50	3.50	2568.81	1.00	3425.08	1.33
✓ 2	7.00	3.50	2568.81	1.00	2568.81	1.00
✓ 3	10.50	3.50	2568.81	1.00	2568.81	1.00
✓ 4	14.00	3.50	2568.81	1.00	2568.81	1.00
✓ 5	17.50	3.50	2568.81		2568.81	

An X is displayed at the left side of the table when the story is 'weak'.

Soft stories

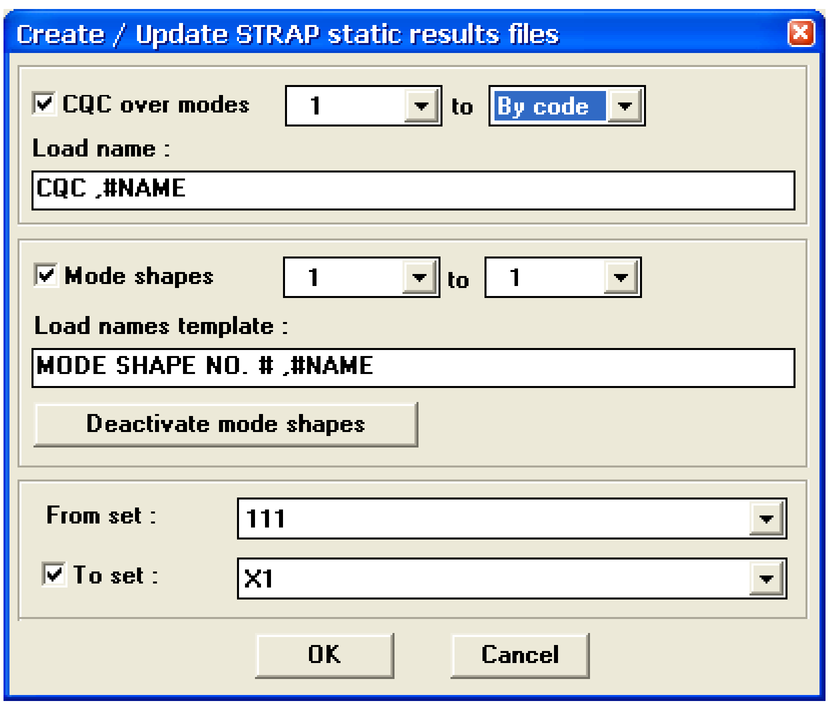
The program identifies "Soft stories" as defined in the Code. For example, ASCE 7-10, Table 12.3.2 - " *Stiffness soft story irregularity is defined to exist where there is a story in which the lateral stiffness is less than 70% of that in the story above or less than 80% of the average stiffness of the three stories* *above*".

For example:



9.6.7 Update result files

Add the dynamic results to the *STRAP* static results file. The number of *STRAP* load cases generated equals the number of mode shapes times the number of eccentricity "sets".



RSS over modes / Mode shapes

Select the method for combining the mode shapes into a structural design load case:

CQC / RSS over modes

a single load case containing the Seismic Response Spectrum analysis summation; select the range of modes to be included in the calculation using the **Deactivate** option.

Mode shapes

a separate load case for each mode; select the modes.

Both options may be selected.

The SRSS/CQC procedure is the method stipulated by all Codes for determining the maximum response. Models properly analyzed and designed by *STRAP* will comply with the Code requirements.

The structural response may be calculated separately for each mode. However, since the maximum response for each mode does not occur at the same instant of time, it would be over-conservative to simply add the separate maximum modal responses. Alternatively, carrying out a complete time-history analysis for the entire model is unfeasible. The SRSS/CQC procedures recommended by the Codes represents the most probable maximum response and takes in to account the fact that the peak modal responses occur randomly with respect to time.

Note:

- The SRSS/CQC method calculates the maximum ***ABSOLUTE (positive) value results*** as a weighted combination of the mode results. It is obvious that the method can generate only one load case to be transferred to *STRAP.*
- The program does the SRSS/CQC calculation separately for shear and moment. If there is more than one mode shape the shear values will not equal the sum of the moments divided by the span length.
- These positive results must be converted to design values when transferring the load case to *STRAP*. There are two options available:
  - the default method: the moment diagrams are drawn entirely on one side of the member (i.e. single curvature, the critical case for column design. Therefore, *STRAP* transfers a negative moment at one end and a positive moment at the other (Refer to Sign conventions). All axial forces are positive.
  - All results are transferred with the sign of the results calculated for the dominant mode shape, i.e. the mode shape with $(F_n/)_{max}$
- Two sets of combinations should then be generated in the *STRAP* results module, one with the transferred seismic load case multiplied by the positive load factor and the other multiplied by the NEGATIVE load factor. Remember that the structure vibrates in both directions so all results can have either sign.
- The program has an option to transfer the results of individual mode shapes to *STRAP* instead of the SRSS/CQC results. Equilibrium will be maintained in such load cases. However, this option is permissible only if the first mode shape is very dominant. The seismic Codes specify the minimum number of mode shapes that must be used in the calculation.

Load name template

- RSS results:
  Revise the default load case name.
- Mode shapes:
  Define a load case name "template". The program substitutes the load case number for the # in each load case.

Note:

- a warning is displayed if a case is created with the same name as an existing case.

Deactivate mode shapes

Mode shapes may be initially deactivated by selecting **Deactivate mode shapes,** i.e. they will not appear in the pull-down boxes displayed when selecting the above options. The program will automatically deactivate mode shapes not required by the selected design Code.

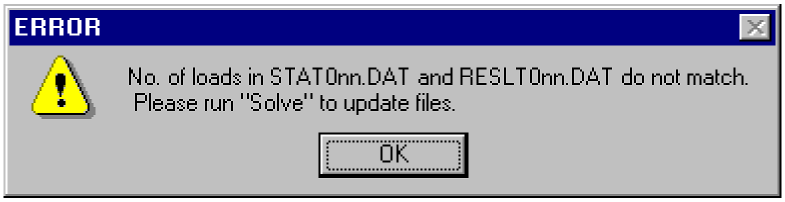
From set / to set

The the same *STRAP* load case(s) for each eccentricity "set".

Select a single set or a range of sets.

Note:

- If static load cases were defined or revised but not solved, the message -



will be displayed. ***All static load cases must be solved before entering this module***.

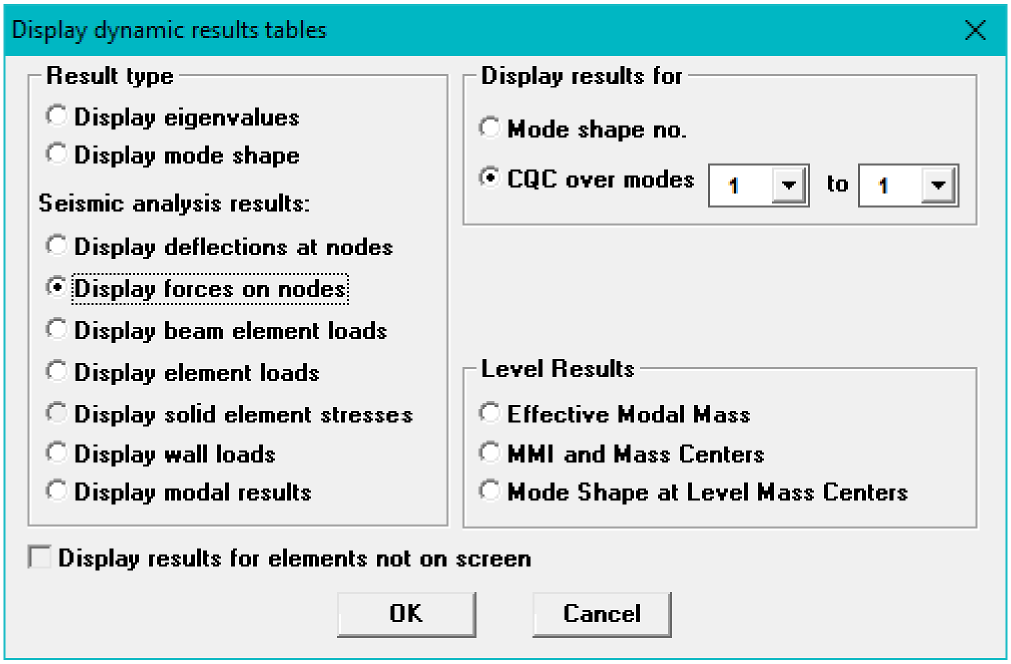
- The program updates two ***STRAP*** files:
  - Results file: each dynamic load case is appended to the file.
  - Applied forces file: in order that the number of load cases in this file will correspond to the number of load cases in the results file, "zero" load cases are appended to the end of the file.
- When writing the results to the static result file, the program also saves the following information:
  - direction earthquake applied
  - CQC/RSS results or individual mode results (and which mode number).

  When writing the results, the program checks if results were already written for the same situation. If yes, the program ***overwrites*** the existing load case in the results file. If not, the program ***adds*** a new load case.

9.7 Tabular results

Display/print the tabular results for the mode shape analysis and seismic analysis.

Display

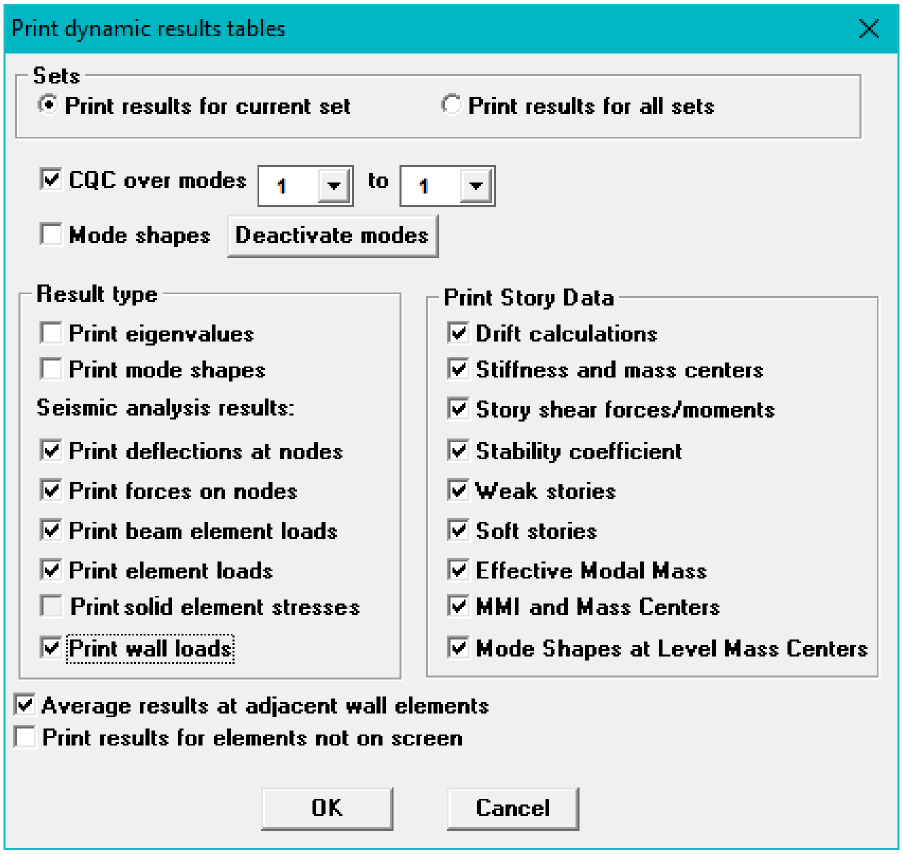


- ***Display results for***

Results may be displayed for individual modes or for a range of modes (CQC/RSS summation).

If you selected the "Missing mass correction" method, the line **MMC** will be displayed at the end of the individual mode shape list.

Print



- ***Display results for***

  Results may be printed for individual modes or for a range of modes (CQC/RSS summation).

  ☑ **CQC/RSS over modes** - all mode shapes in the range selected will be used.

  ☑ **Mode shapes** - results may be printed for selected mode shapes only. Click **Deactivate** to select the modes

- ***Deactivate mode shapes***

  Mode shapes may be initially deactivated by selecting **Deactivate mode shapes,** i.e. they will not appear in the pull-down boxes displayed when selecting the above options. The program will automatically deactivate mode shapes not required by the selected design Code.

9.7.1 Eigenvalues

For each mode shape requested the program displays:

- eigenvalue = ω^2
- natural frequency = $f = \omega/2\pi$
- period of vibration = 1/f (in seconds)

For example:

MODAL DATA : Eigenvalues

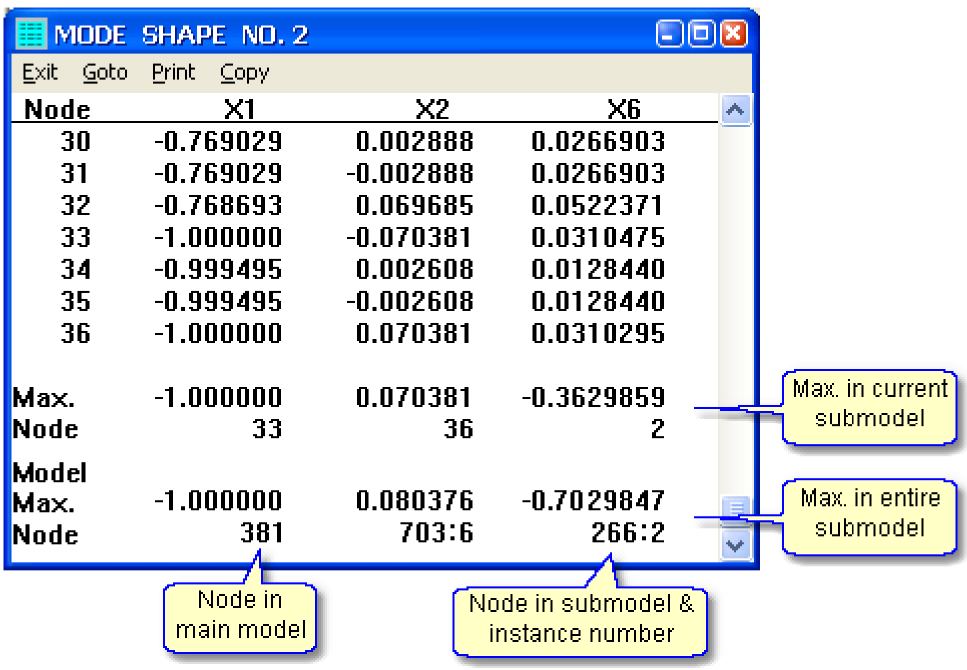
Exit Goto

Mode No.	Eigenvalue [Omega**2]	Natural Frequency	Period	Max translation Node-DOF
1	159.659	2.0110	.49726	30-1
2	2326.797	7.6771	.13026	25-1
3	10031.770	15.9408	.06273	10-1
4	25837.660	25.5827	.03909	15-1
5	41225.090	32.3148	.03095	18-1
6	47027.590	34.5141	.02897	19-1

9.7.2 Mode shapes

For each mode shape requested the program displays the mode shape displacements. The displacements are always relative values and dimensionless; the maximum deflection is set to 1.00 and all other deflections are proportional to it.

Note that the maximum relative deflection of 1.00 at node is the deflection in ***one*** of the global directions. The vector sum of the relative deflections at node 6 is greater than 1.000. For example (a model with submodels):


MODE SHAPE NO. 2

Exit Goto Print Copy

Node	X1	X2	X6
30	-0.769029	0.002888	0.0266903
31	-0.769029	-0.002888	0.0266903
32	-0.768693	0.069685	0.0522371
33	-1.000000	-0.070381	0.0310475
34	-0.999495	0.002608	0.0128440
35	-0.999495	-0.002608	0.0128440
36	-1.000000	0.070381	0.0310295
Max.	-1.000000	0.070381	-0.3629859
Node	33	36	2
Model Max.	-1.000000	0.080376	-0.7029847
Node	381	703:6	266:2

9.7.3 Seismic analysis

Display the deflections, beam forces and moments, element moments and stresses as generated by the CQC/RSS calculation.

- The results are in the same format as the ***STRAP*** static tabular results.
- Results may be displayed for individual modes or for a range of modes (CQC/RSS summation).

9.7.4 Modal results

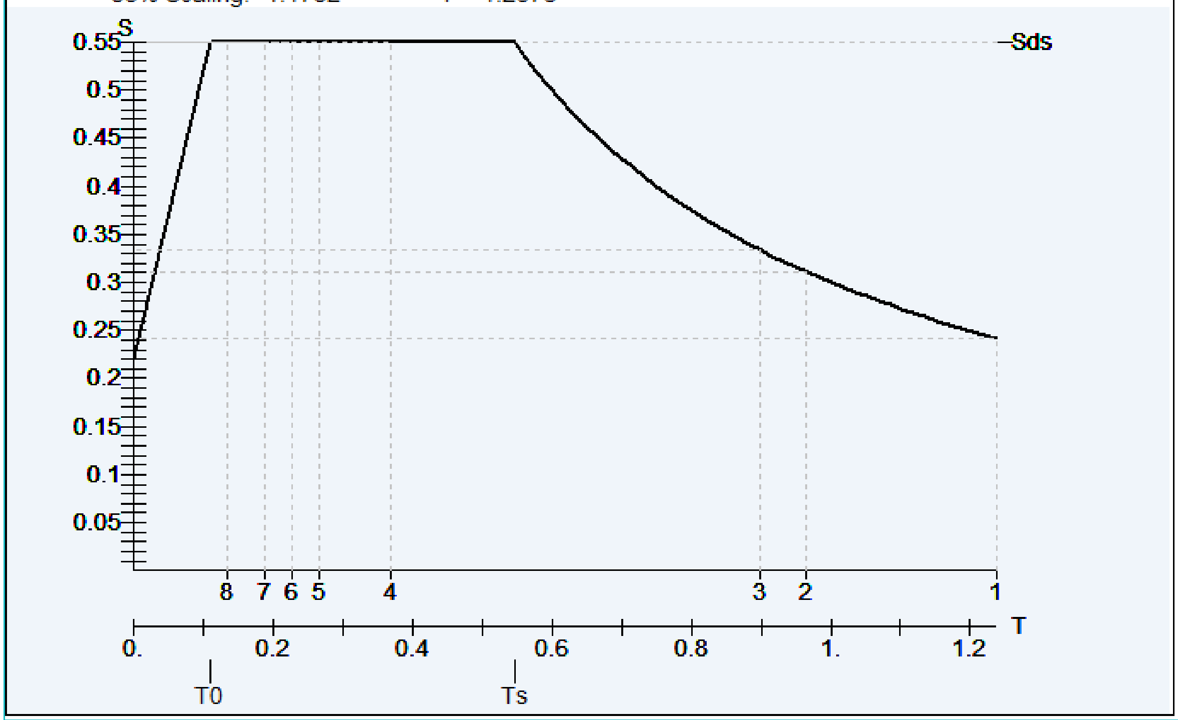
Display the seismic analysis summation over all calculated mode shapes and the response spectrum:

MODAL RESULTS

Exit Goto Print Copy

Mode	T	Wn/Wtot	Fn (ton)	Qn (m)	Vn (m/s)	An (m/s**2)	Fn'/Wn
1	1.2376	0.193	205.11	-0.0701	-0.1137	-0.1842	0.1616
2	0.9644	0.490	668.72	0.0804	0.1673	0.3481	0.2074
3	0.8991	0.053	77.34	0.0277	0.0618	0.1380	0.2224
4	0.3678	0.029	69.87	0.0053	0.0287	0.1567	0.3667
5	0.2659	0.116	280.36	0.0051	0.0383	0.2887	0.3667
	Total sum	0.933	1425.14				
	CQC results		766.63	0.1105	0.2173	0.5635	

The ASCE Standard (ASCE/SEI 7-10)
Direction:X1 Damping=0.000
S=C Ie=1.00 R=1.50 Ss=0.750S1= S1=0.300 Fa=1.100 Fv=1.500
85% Scaling: 1.1782 T= 1.2376



here:

T = period (seconds)

Wn/Wtot = a factor reflecting the relative influence of the mode shape.

Fn = sum of external forces in ***all*** global directions.

Qn = A factor which when multiplied by the mode shape 'deflections' calculated by the Mode Shape Analysis program (where the maximum deflection in each mode shape is always 1.0) gives the actual deflection at each node.

Vn = A similar factor for velocity.

An = A similar factor for acceleration

Fn/Wn = A factor that represents the fraction of the total applied weights that is applied as a horizontal force for each mode shape.
(This header varies according to the Code selected)

9.7.5 Level results

Effective modal mass

The participation factor for each solved mode in all directions.

Effective Modal Mass (Wn/Wtot)

Exit Goto Print Copy

Mode	X1	X2	X3
1	0.094	0.744	0.000
2	0.744	0.094	0.000
3	0.000	0.000	0.000
4	0.058	0.071	0.000
5	0.071	0.058	0.000
6	0.000	0.000	0.000
7	0.011	0.014	0.000
8	0.014	0.011	0.000
9	0.000	0.000	0.233
10	0.000	0.000	0.198
11	0.000	0.000	0.000
12	0.000	0.000	0.151
13	0.000	0.000	0.116
14	0.000	0.000	0.155
15	0.000	0.006	0.000
Total	0.992	0.999	0.852

MMI and mass centers

MMI = mass moment-of-inertia ($m*r^2$)

The program displays for every level:

- the mass and the mass center.
- the MMI, calculated as the sum of all masses multiplied by the square of the distance from the mass center.
- Mass radius of inertia, calculated as the square root of MMI divided by the mass.

For example:

MMI AND MASS CENTERS (Units: ton, meter)

Exit Goto Print Copy

No.	Level	Mass		Mass mom. of inertia	Mass rad. of inertia	Mass Center	
		X1	X2			X1	X2
0	0.00	No mass					
1	4.50	533.518	533.518	38989.61	8.549	10.000	10.000
2	7.50	499.033	499.033	36116.27	8.507	10.000	10.000
3	10.50	499.033	499.033	36116.27	8.507	10.000	10.000
4	13.50	499.033	499.033	36116.27	8.507	10.000	10.000
5	16.50	499.033	499.033	36116.27	8.507	10.000	10.000
6	19.50	434.874	434.874	30411.20	8.362	10.000	10.000

Mode shapes at level mass centers

The program displays for a selected mode:

- the mode shape at the center of mass at each level. If there is no node at the center of mass, the program calculates an average results from the surrounding nodes.

For example:

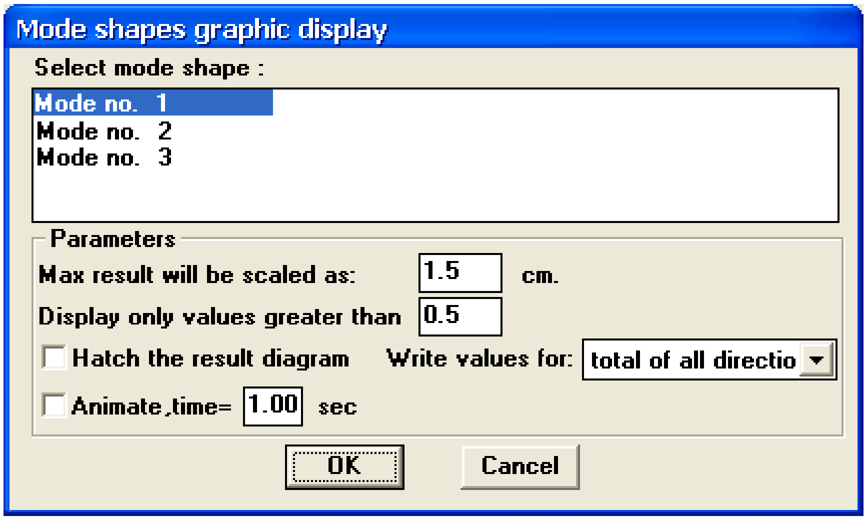
Mode shape 1 at level mass centers

Exit Goto Print Copy

No.	Level	X1	X2	X3	X4	X5	X6
0	0.00	0.000	0.000	0.000	0.000	0.000	0.000
1	3.00	-0.240	0.117	0.000	0.007	0.014	0.000
2	6.00	-0.422	0.205	0.000	0.000	0.000	0.000
3	9.00	-0.600	0.292	0.000	0.004	0.007	0.000
4	12.00	-0.801	0.390	0.000	-0.001	-0.001	0.000
5	15.00	-1.000	0.486	0.000	0.001	0.002	0.000

9.8 Graphic results

Display the mode shapes graphically:



Mode shape

Select the mode to be displayed from the list in the box.

Maximum result scale

The deflections are displayed relative to a scale chosen as follows:

The program searches for the maximum result in the plot area and plots it on the screen as the dimension listed above - the default value is 1.5 cm (0.6 in.). All other results are plotted in proportion to this value.

Move the into the text box and type a new dimension in cm.

Display values greater than

For clarity, part of the numerical values may be deleted from screen (the entire geometry and result diagram are plotted).

All values less than a given fraction (default = 0.5) of the maximum result are not displayed.

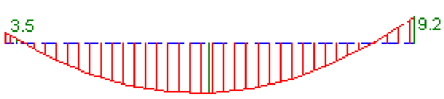
Move the into the text box and type a new percentage.

Example:

- Maximum deflection = 0.12m and fraction = 0.5 : Only numbers greater than 0.06 m are displayed.

Hatch diagram

Set the box to to hatch the results diagram. For example:



Write values for

- **Total of all directions**
  The program displays the vector sum of the deflections in the three global directions, i.e. v(dX1^2 + dX2^2

+ $dX3^2$).

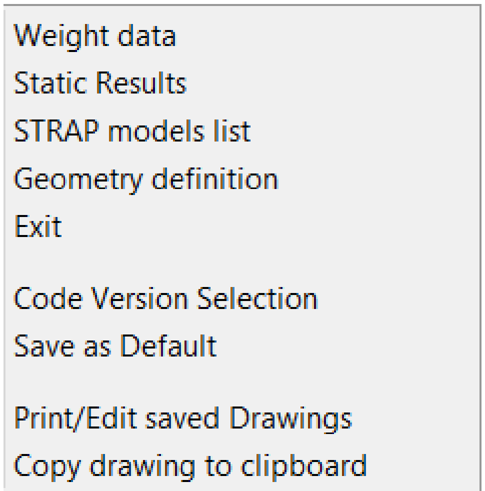
- **Global X1/X2/X3 direction**
  Display the deflection value for one of the global directions only.

Animate

Set this option to ☑ to animate the mode shapes deflections:

- The model deflects from its full positive deflection to its full negative deflection in seven equal steps during the **time =** interval specified.
- The animation continues until the **End animation** button at the bottom of the screen is clicked.

9.9 File options



Weight data

Revise the nodal weight data. All modifications and revisions are saved.

Static results

Display static results for the current model. All modifications and revisions are saved.

STRAP models list

Return to the ***STRAP*** model list (main menu). All modifications and revisions are saved.

Geometry definition

Return to the geometry definition for the current model. All modifications and revisions are saved.

Exit

Exit from the ***STRAP*** program. All modifications and revisions are saved.

Code version selection

For certain national seismic codes: either the current version of an older version may be used.

Save as default

Save the current parameter values as the default values for future models.

Print/edit saved drawings

Save the the current display, add/modify text and lines; print. Refer to Print/edit drawing

Copy drawing to clipboard

Create a drawing of the current graphic display to any scale and send it to the Windows clipboard.

9.10 Time history response

For background information, select one of the following topics:

General 1035 - a summary of the theory and program assumptions.
Procedure 1036 - an outline of the procedure for carrying out a forced vibration analysis of a model.

New load	Define a new load case (consisting of nodal forces) and the associated base acceleration and history function.
Revise load	Revise any of the existing load cases. Select a load case
Delete	Delete any of the existing load cases. Select a case from the list displayed.
Damping	select the number of mode shapes to be used for the calculation and the damping factor for each of the shapes.
Display ta...	Display tabular results 1044
Print tables	Print tabular results 1044
Display gr...	Display graphic results 1043
Print graphs	Print graphic results 1043
Display lo...	Display 1046 tables showing all load case data - history function, nodal forces and base acceleration.
Print loads	Print 1047 tables showing all load case data - history function, nodal forces and base acceleration.
Update re...	append results 1048 to the static result files.

From the menu bar:

File Zoom Rotate Display ReMove Combinations Time tables

Combinations 1049 - Create combinations of the existing dynamic load cases.
Time tables 1050 - Define the time intervals at which to calculate and display tabular results or append static result files (this option does not apply to graphic results).

9.10.1 General

This module calculates the transient (history) response of a model subject to dynamic loads in which viscous damping is present. It enables the dynamic analysis of models subject to impact, impulse or cyclic loads or any other type of load that varies with time.

The equations of motion are solved on the basis of the results from the Natural Frequency and Mode Shape analysis

$$[m]\{\ddot{x}\} + [c]\{\dot{x}\} + [k]\{x\} = \{P\}\,F(t)$$

where:
[m] = diagonal mass matrix
[c] = matrix of damping coefficients
[k] = stiffness matrix
{P} = joint load distribution

F(t) = time history of the applied forces

The program assumes:

- at each node the history behaviour of the load is represented by the input joint loads multiplied by the history function F(t).
- the history function F(t) is composed of either:
  - a series of straight line segments defined by a set of pairs of time and amplitude values:
    {t1, F(t1)}, {t2, F(t2)}, ... , {tn, F(tn)}
    where: tn > tn-1 > 0
  - a sine function curve

  A different history function may be defined for each load case.
- the damping matrix [c] is proportional to the mass matrix:
  [c] = 2 [ß] [m]
  where [ß] is a constant diagonal matrix.
  The damping is defined as a set of coefficients, one for each mode shape, where the coefficients represent a percentage of the critical damping ßcr = ω_n (ω_n = modal frequency).

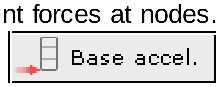
9.10.2 Procedure

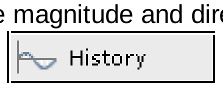
Solutions may be obtained for a single dynamic load or combinations (superposition) of up to 10 different dynamic loads.

- Each load case is defined as a series of joint loads, or as a base acceleration, or as a combination of both.
  - The joint loads act according to the history function.
  - The base acceleration is defined as a direction vector (X1, X2, X3) and an amplitude.
- the program solves the equations of motion at each point in time that is included in a user-defined table.

For each load case, define joint forces and/or a base acceleration:

- click the 

 icon
  define joint forces at nodes.
- click the 

 icon
- Enter the magnitude and direction of the base acceleration (factors for X1,X2,X3).
- click the 

 icon
- define the graph of the history function F(t).

Damping:

- click the 

 icon
- enter the damping coefficients (%) for each mode shape.

Compute times and Display times (for tabular results):

- select **Time tables** in the menu bar
- select **Define computed time table** in the pull-down menu.
- Define a set of intervals and steps that specify the times at which the results are computed and displayed.

Output:

- click the Display gr... icon to display graphic results or the Print graphs
- icon to display tabular results
- select the result type, display time, etc to display.
- click the Display ta... icon to print graphic results or the Print tables icon to print tabular results
- click Update re... to add load cases to the static results file. Note that this option uses the Compute times defined above.

9.10.3 Define/revise load case

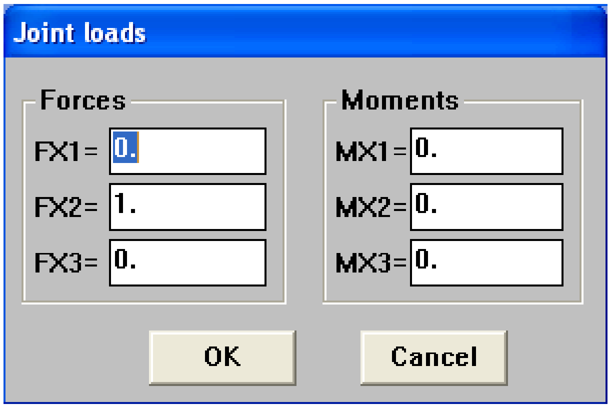
Define a load case as nodal forces and/or base acceleration acting according to a time history function.

Add	Define joint loads.
Revise	Revise joint loads
Delete	Delete joint loads
Base accel.	Define the acceleration vector (direction and magnitude) to be applied to all nodes in the model.
Static load	Add the joint loads in a static load case to the current dynamic load case.
Time load	Add the joint loads defined in any other dynamic load case defined in this section to the current dynamic load case.
History	Define the time behaviour of the nodal forces (or base acceleration) for the current load case. The history function may be defined as either:

- ***Linear*** 1040
  a series of pairs of time and amplitude values forming a series of straight line segments:
  $\{t_1, F(t_1)\}, \{t_2, F(t_2)\}, ... , \{t_n, F(t_n)\}$, where: $t_n > t_{n-1} > 0\}$
- ***Sine*** 1042
  a sine function curve

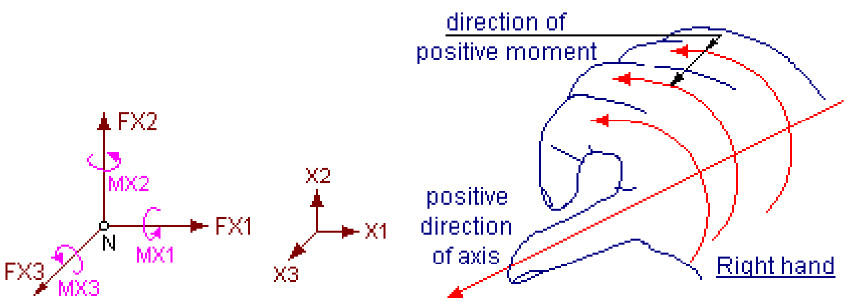
9.10.3.1 Joint loads - define

Joint loads are defined relative to the global coordinate system.



- Enter the load values; note that loads in more than one global direction can be defined at the same time.
- Select the nodes that the loads are to be applied to using the standard Node Selection option.

The positive force and moment sign conventions are:



9.10.3.2 Joint loads - revise

Select nodes with joint load to be revised/deleted using the standard Node selection option.

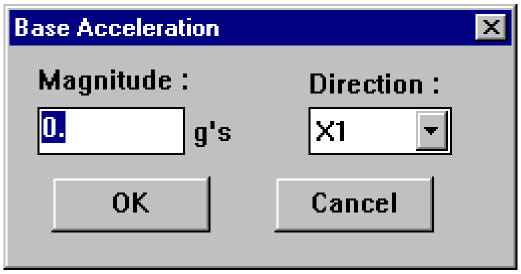
For Revise, enter new values for the load as explained in Joint loads - define 1038.

Note that unlike the static joint loads, the program does not remember that the same load was applied to more than one node at the same time. To revise a group of identical loads, ***all*** of the nodes must be selected.

The loads on the selected nodes are updated/deleted on the graphic display.

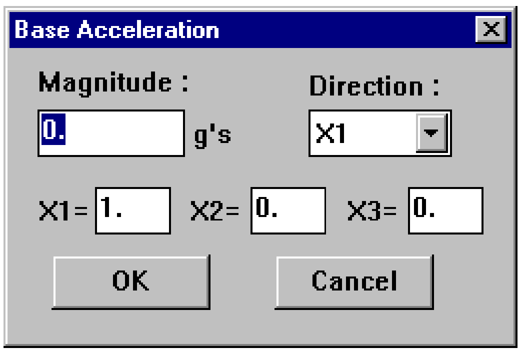
9.10.3.3 Base acceleration

Define the acceleration vector (direction and magnitude) to be applied to **all** nodes in the model:



- **Magnitude** - defined as a factor for 'g'
- **Direction** - select one of **X1, X2, X3** or **Other**

  If **Other** is selected, then define the vector components for X1, X2 and X3:

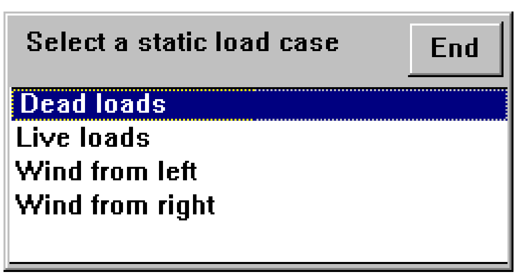


  Note that the values entered determine only the direction of the vector.

9.10.3.4 Add static load

Use this option to add the ***joint*** loads from an existing static load case to the current dynamic case.

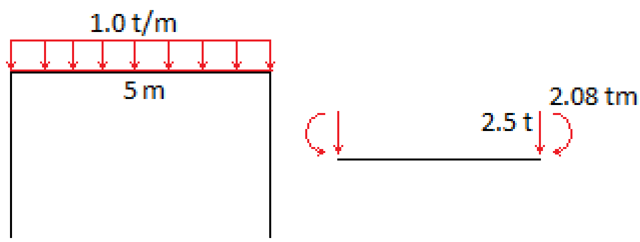
Select a load case from the list:



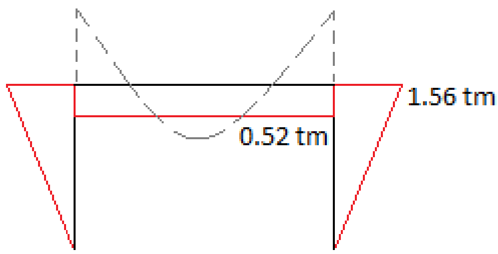
Note

- the program adds all beam loads in the static load case as "applied loads" only on the support nodes. Therefore the results ***along the beams*** will not be displayed properly in the results module.

  For example, a simple frame has a load applied to the top beam. The program adds the "Applied loads" as shown as shown.

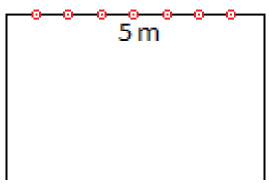


After solving the model, the results displayed are :



The moment distribution along the beam - shown hatched - is not displayed.

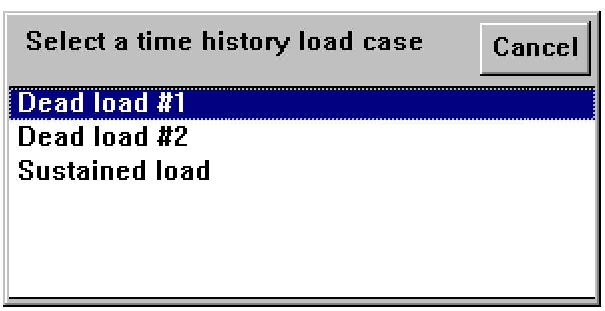
To display more accurate beam results, add dummy nodes along the beam:



9.10.3.5 Add time load

Use this option to add the joint loads from an existing dynamic load case.

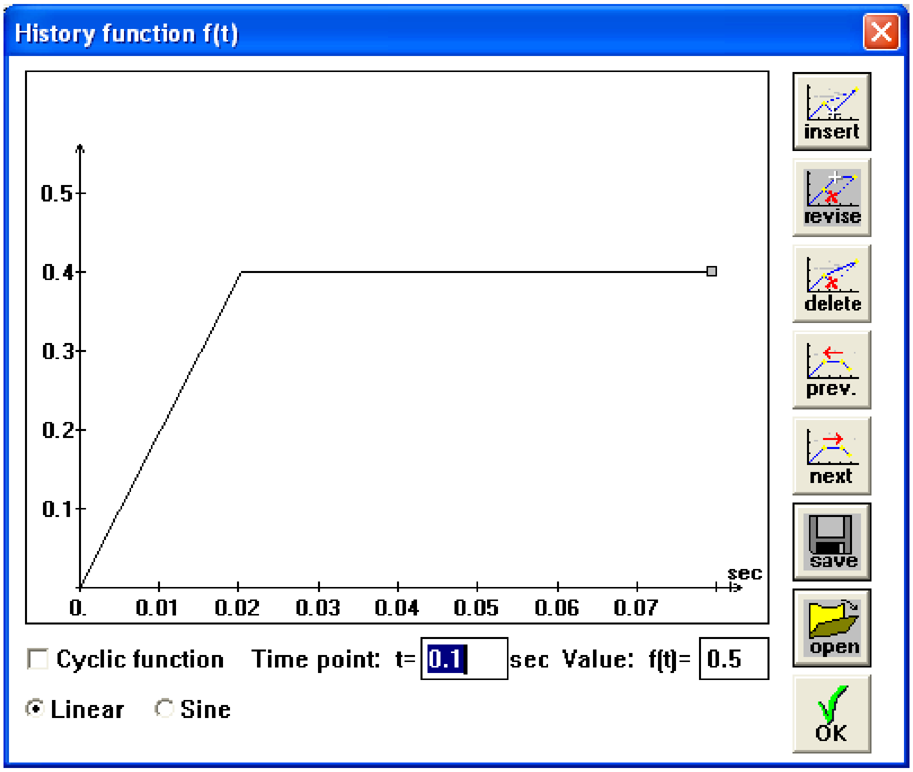
Select a case from the list displayed:

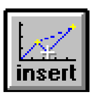


Note that the program does not copy the base acceleration from the selected case.

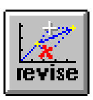
9.10.3.6 History function - Linear

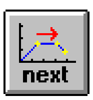
Create a time history function consisting of a series of straight line segment. The segments are created by defining their end points. Note that a function may be saved/retrieved to/from a file.

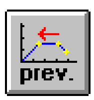




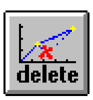
Add a new point to the curve:
define **Time point** and **Value** and click the button

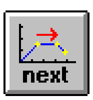


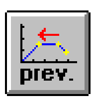
click 

or 

until the values of the point are displayed. Enter new values and click the button



click 

or 

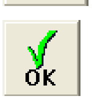
until the values of the point are displayed, then click the button .

The current linear function may be saved to a file and retrieved in a different load case or model. The default file extension is *.HIS

The function may also be written to an ASCII file using any standard editor program. The file format is:

Row 1:	**HIST**		where:
Row 2:	***t1***	***f(t1)***	• the lines are unformatted
Row 3:	***t2***	***f(t2)***	• the times must be in ascending order
	.		• the times are in milliseconds
	.		
Row n+1:	***tn***	***f (tn+1)***	

A saved linear function may be retrieved from a file. The retrieved function will erase any current function on the screen



click to exit.

Parameters:

Cyclic function

☑ The function is the first cycle of a periodic function that extends to infinity
☐ The amplitude of the function is assumed to be zero beyond the defined time domain.

Time point

Define the time at the point in seconds

Value

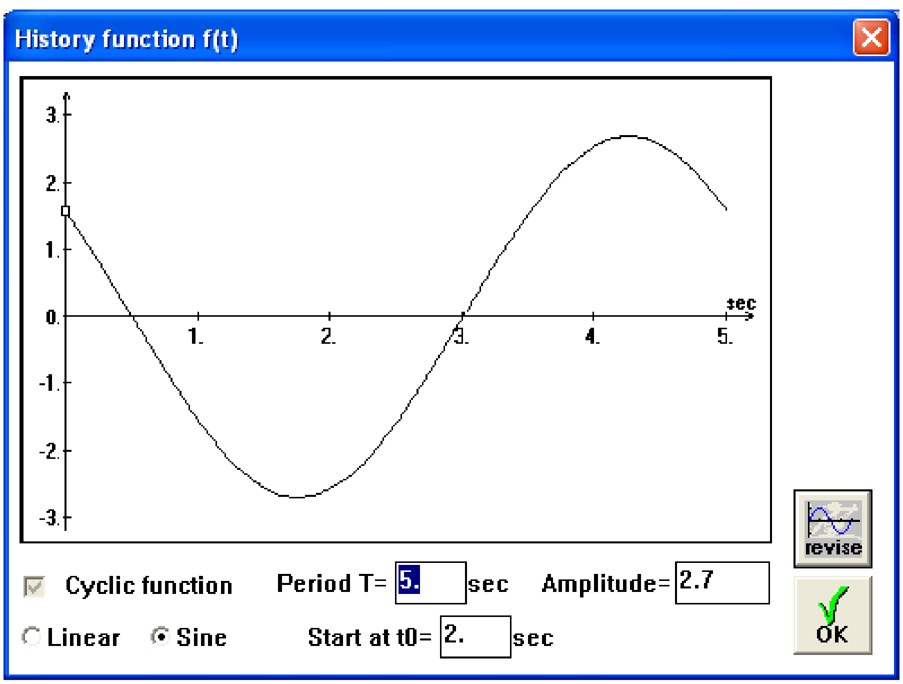
Define the amplitude of the function at the current point. The program will multiply the joint loads by the amplitude.

Linear / sine

Linear - define a series of linear function segments
Sine - define a sine curve

9.10.3.7 History function - Sine

Create a time history function consisting of a sine curve:



The curve above was defined with the values in the Text boxes.

- Enter the following parameters and click  to redraw the function curve
- click  to exit

Cyclic function

☑ The function is the first cycle of a periodic function that extends to infinity
☐ The amplitude of the function is assumed to be zero beyond the defined time domain.

Period

Define time in seconds of one cycle of the sine curve.

Amplitude

Define the amplitude of the sine curve.

Linear / sine

Linear - define a series of linear function segments
Sine - define a sine curve

Start at t0

Define the "phase shift" in seconds. Note that the initial sine curve displayed is the classic sine curve having an amplitude of zero at time t=0 and then increasing to the amplitude value. This curve can be "shifted" so that any other amplitude is present at t=0.

The shift is defined by entering the time value.

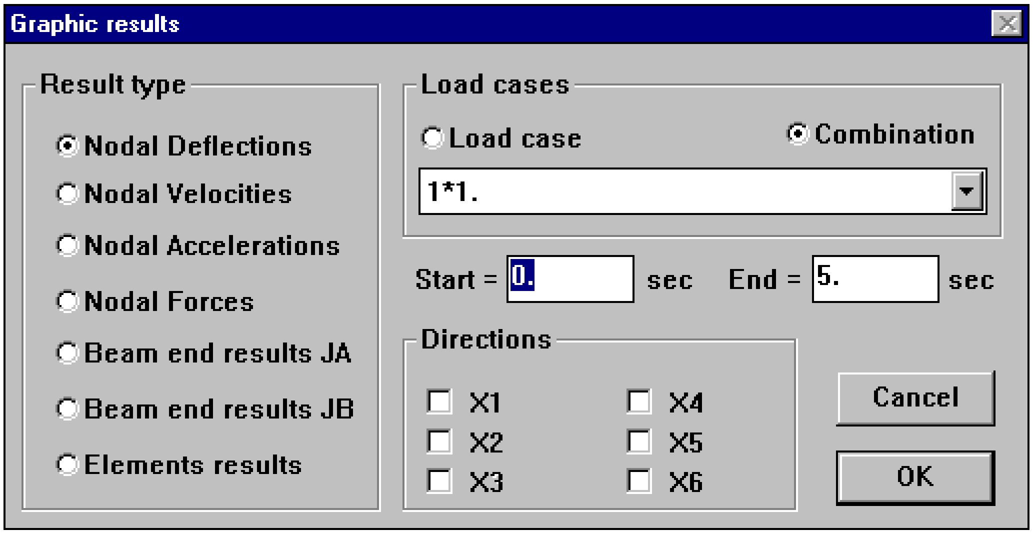
9.10.4 Output

Display and/or print the data and the results.

Display ta...	Display tabular results 1044
Print tables	Print tabular results 1044
Display gr...	Display graphic results 1043
Print graphs	Print graphic results 1043
Display lo...	Display 1046 tables showing all load case data - history function, nodal forces and base acceleration.
Print loads	Print 1047 tables showing all load case data - history function, nodal forces and base acceleration.

9.10.4.1 Graphic results

Select a result type, load case/combination, direction and the time domain. To display the graph, click OK and select a node/beam/element.



Result type / direction

Select one or more result types; note that the options for Directions varies according to the Result type:

Nodal results - select a global direction
Element results - select **Mx**, **My**, **Mxy**, **Fx**, **Fy** or **Fxy**
Beam results - select **axial** force, **V2/V3** shear or **M2/M3** moment

Load cases

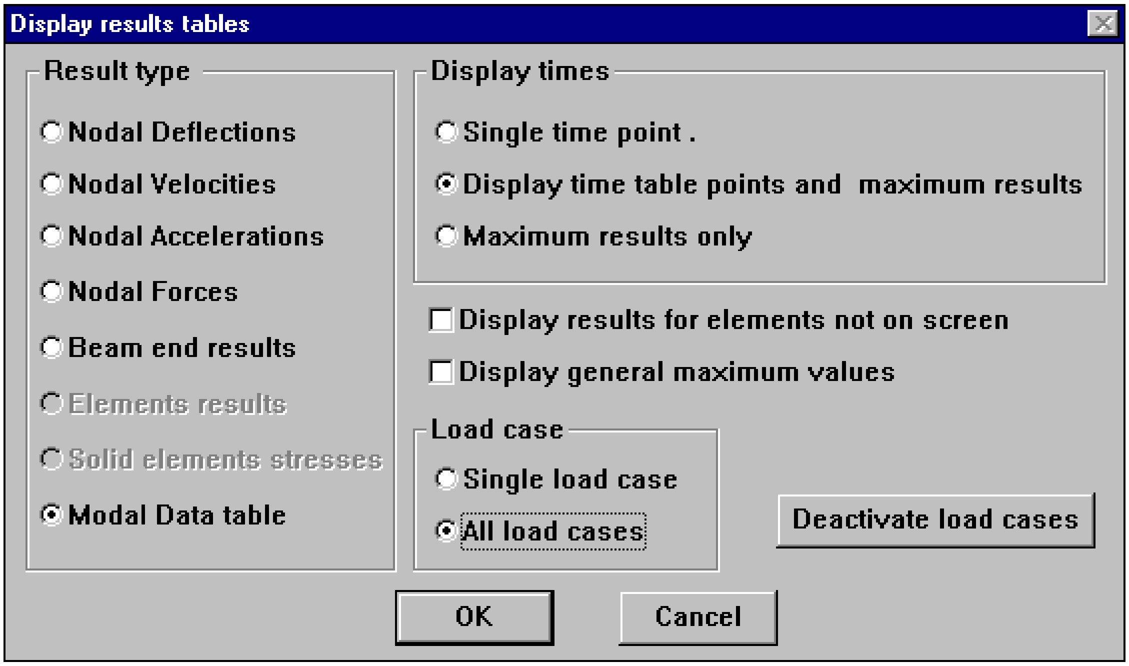
Load case - Select a load case from the list box
Load combination - Select a load combination from the list box

Start / end

Define a time range for the result display

9.10.4.2 Tabular results

Select a result type, the display times and the load cases, then click OK to display the tables.



Node results

Results are displayed at all nodes

Beam/element results

Moments and forces are displayed for all beams and elements.

Modal data table

Eigenvalues, frequency and period.

Display times

Select the times at which the results are to be displayed:

Single time point
Define any time value

Display time table points and maximum results
At all Display times defined by the Intervals, Steps and Sub-step values.

Maximum results only
The program will search for the maximum result within the min-max time range defined in the Time Table option Note that the program does not search for the maximum result only at the Display times specified but at all the Compute times.

Elements not on screen

display results for all elements/nodes even if not displayed because Zoom or Remove options were selected.

display results for displayed elements/nodes only.

General maximum values

Display the overall maximum values for each result:

- the values are displayed at the end of the table.
- the values are calculated from results for displayed elements/nodes only.
- the values are computed from all Compute times, not only Display times.

Load cases

Single load - search for the results in a single load case (combination)
All load cases - search for the results over all load cases (combinations)

Deactivate

This option is displayed only when all load cases (combinations) were selected in the "Load case" option. Load cases may be temporarily deactivated so that they do not appear in the list of cases (combinations). Use this option in conjunction with the "Load case" option above to select the load cases (combinations) to be used in the calculation.

9.10.4.3 Loads/time table

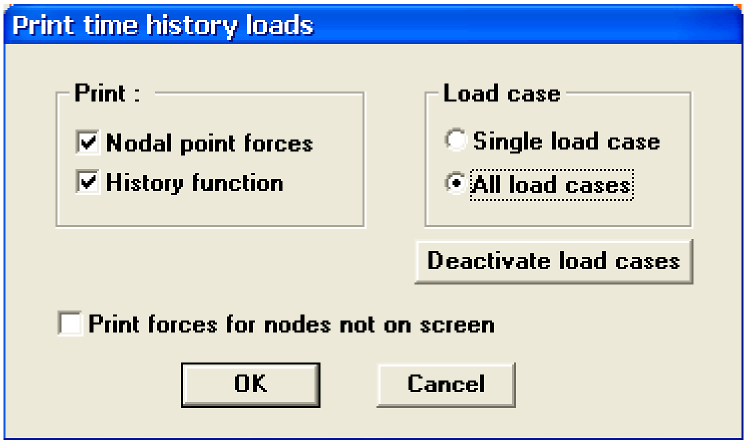
Display the joint loads and the history function in tabular form

where:

Nodal point forces - Display defined joint loads and/or the base acceleration
History function - Display the history function intervals (linear) or the sine curves parameters.

9.10.4.4 Print loads / Time table

Display the joint loads and the history function in tabular form

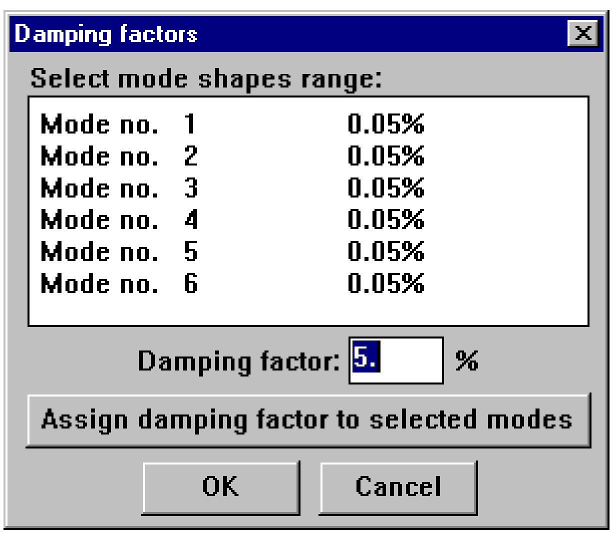


where:

Nodal point forces	-	Print defined joint loads and/or the base acceleration.
History function	-	Print the history function intervals (linear) or the sine curves parameters.
Single load case	-	Select a load case
All load cases	-	Print the data for all defined load cases or selected load cases (click the **Deactivate** button)

9.10.5 Damping

Define the modal damping factors as a percentage of the modal frequency. A different damping factor may be assigned to each mode shape.

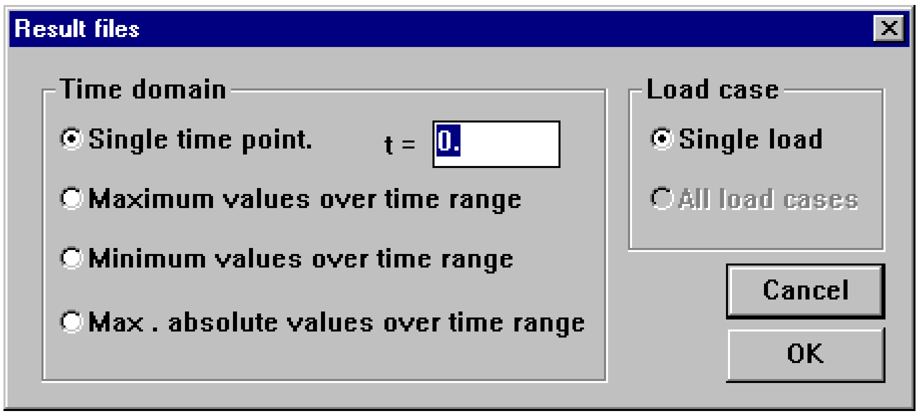


To define the damping factors:
- Click and highlight the mode in the box (or drag across a series of modes).
- Enter a damping value in the **Damping factor** text box.
- Click Assign damping factor to selected modes .

- Click OK when damping factors have been assigned to all mode shapes.

9.10.6 Result files

Update the ***STRAP*** result files according to the following options:



Single time point

Use the node/element/beam results at a single time only; enter the value.

Maximum values over range

Use the maximum values (maximum positive or minimum negative) over the time range specified in the "Time table" option and calculated at all Compute times.

Minimum values over range

Use the minimum values (maximum negative or minimum positive) over the time range specified in the "Time table" option and calculated at all Compute times.

Maximum absolute values

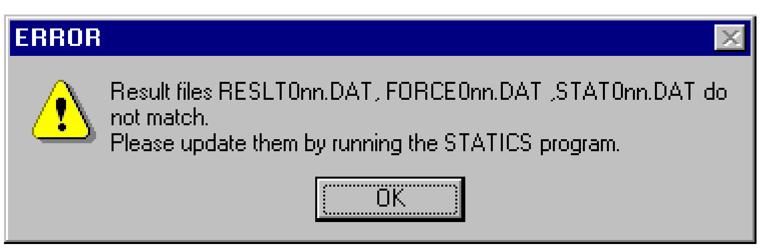
Use the maximum absolute value of "Maximum" and "Minimum"

Load case

Single load	-	search for the results in a single load case (combination)
All load cases	-	search for the results over all load cases (combinations)

Note:

- If static load cases were defined or revised but not solved, the message -

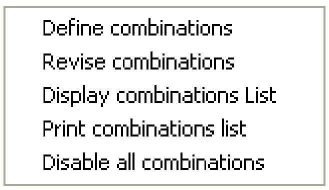


will be displayed. ***All static load cases must be solved before entering this module***.

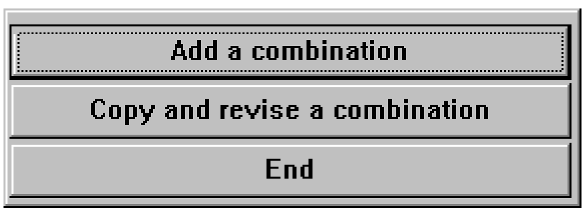
- The program updates several ***STRAP*** files:
  - Result files: each dynamic load case is appended to the file.
  - Force files: "zero" load cases are appended to the end of the file in order that the number of load cases in this file corresponds to the number of load cases in the results file.
  - Load file
- When writing the results to the static result file, the program creates a load case title that contains the following information:
  - time point/minimum/maximum/absolute maximum
  - load cases or combinations (but not "deactivated" cases).

  When writing the results, the program checks if a load case with the identical title exists. If yes, the program ***overwrites*** the existing load case in the results file. If no, the program enables the user to either add a new case to the existing ones or to replace all of the existing dynamic load cases with the new case.

9.10.7 Combinations

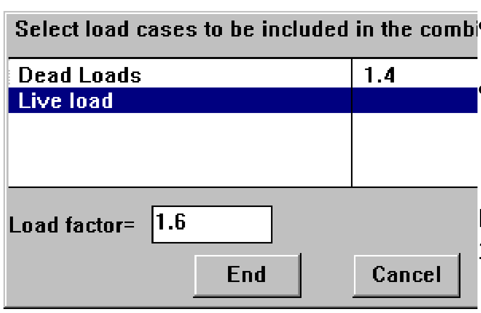


Define combinations



- ***Add a combination***

To define a new combination. The program displays a list of the load ***cases***; select load cases and enter the load factor. For example:



- Use the mouse/arrow keys to highlight a load case that you want to include in the combination; click the mouse.
- Enter the factor for this load case in the **Load factor=** text box. The factor is then written alongside the load case / group title

In the example, the combination will be 1.4 * dead load + 1.6 * live.

Repeat for additional load cases. Click End when the definition is completed.

The program then requests a title for the combination (the default title is the combination command).

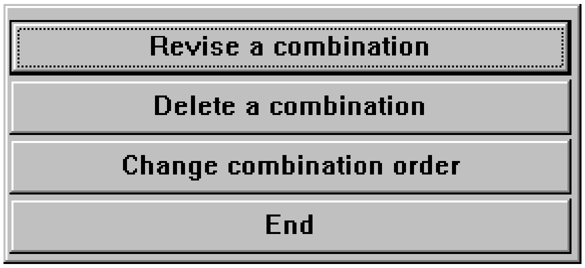
Enter a title or click OK to use the default title. The combination title is added to the combination list.

- ***Copy and revise a combination***

  To create a copy of an existing combination and revise the copy. The original combination will not be deleted or revised; refer to the previous option for an explanation.

Revise combinations

The options are:



- **Revise a combination:**
  To revise a combination definition. Select the combination and proceed as per Define a combination
- **Delete a combination:**
  To delete a combination. Use the mouse/arrow keys to highlight the combination that you want to delete; click the mouse.
- **Change combinations order**:
  To rearrange the order of the combinations. Use the mouse/arrow keys to highlight the combination that you want to move; click the mouse. Select the new location in the list and click the mouse - the selected combination will be placed before this location; the program will then display the combination list in the revised order.

Display/print combinations

Display the combinations defined for the current model.

Disable all combinations

The program calculates the results for the load ***cases*** only (the combinations are ***not*** erased)

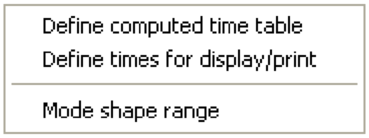
9.10.8 Time Table

Specify the times at which tabular results will be calculated and displayed (this option does not apply to graphic results). The values are also used when appending results to the static result files.

There are two options:

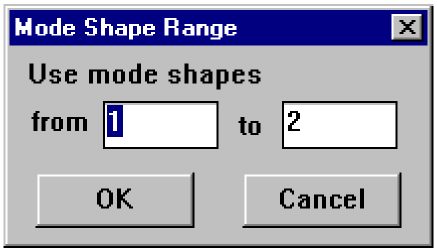
- ***Compute times***
  A series of time intervals can be specified. Each interval can then be divided into sub-intervals by defining a "Step". For example, define a time interval from 6 to 9 seconds, with a Step of 0.5; the program will calculate the results at 6.0, 6.5, ... , 8.5 and 9.0 seconds.

- ***Display times***
  The program can be instructed to display the results only at selected "Compute time" intervals , referred to as "Display times". For the example above, specify that the results will be displayed every third step in the interval, i.e. at 6.0, 7.5 and 9.0 seconds.



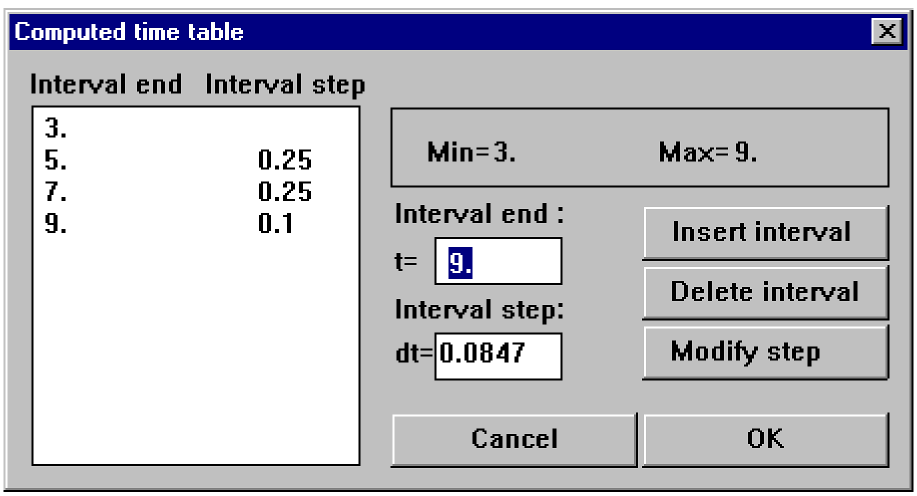
Mode shape range

By default, all results all calculated for the entire range of mode shapes. Use this option to exclude mode shapes from the calculation:



9.10.8.1 Compute Times

Specify the times at which tabular results are displayed.



The Time Table is defined as a series of "Intervals:

- Each interval is defined by a start time and the "Interval end" (the start time is the "Interval end" of the previous interval. In the example displayed there is an interval from to
- Each interval may be sub-divided into "Steps". The interval from 3 to 5 has a step of 0.25 seconds.

For the interval from 3 to 5 seconds, the program will calculate the tabular results at 3.0, 3.25, 3.50, …, 4.75, 5.00 seconds.

Min= / Max=

The upper and lower limits of the time table (for information only)

Interval end

Define the time at the end of the current interval (in seconds)

Interval step

Define the number of sub-intervals in the current "Interval.

For example, if the interval starts at 3 seconds and ends at 5 seconds and the Step is defined as 0.5 seconds, results will be computed at t = 3.0, 3.5, 4.0, 4.5, 5.0 seconds

Insert interval

To insert a new interval in the Time Table:

- enter the correct values of **Interval end** and **Interval step** in the Text boxes
- click [Insert interval]

Delete interval

To delete an interval:

- Click and highlight the interval in the table
- Click [Delete interval]

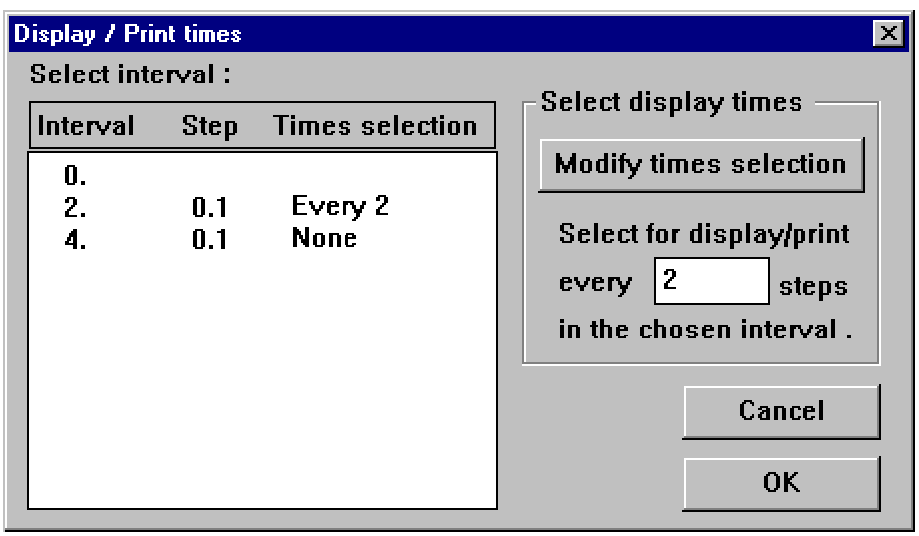
Modify step

To modify a "Step" value:

- Click and highlight the interval in the table
- Enter a new value in the **Interval step** text box

9.10.8.2 Display Times

Display results at only some of the "Compute times".



To define the "Display times":

- click an interval in the **Select interval** box
- define a value for Step in the **Select display times** box, where:

  0 = do not display results at any of the compute time step, i.e only at the interval ends

  1 - display results at all compute time steps

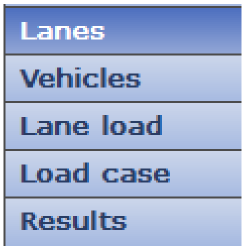
  n - any other number for a selected number of steps
- Click [Modify times selection]

- When the Display times have been defined for all intervals, click OK.

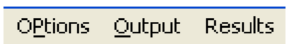
Part X

Bridge design

10 Bridge design



From the menu bar:



and the general display options:

Refer also to:
Bridge module - general 1056
How to use the bridge module 1060
Results - General 1086
Display - general

10.1 Bridge - general

Introduction

Most bridge design codes require that each point on the bridge be designed for the arrangement and combination of loads that produce the most adverse moments, shears, etc. at that point.

In order to comply with the requirements of the Code, the bridge designer has to calculate influence lines for each result type at every point along the bridge. Based on the results, he then decides where to load the bridge. It is obvious that as the bridge becomes more complicated, the amount of work required increases enormously.

The *STRAP* bridge module calculates the load patterns and the corresponding results, as follows:

- The user defines a regular *STRAP* model of the bridge consisting of beams and/or elements, all load cases (self-weight, wind, earthquake, etc) - other than highway (vehicle) loads, and solves the model for these loads.
- The user then selects the **Bridge design** option and defines the following:
  - lanes 1062
  - lane loads 1074
  - load cases 1078
- The program then calculates for every point in the model and for every result type (moment, shear, etc) the combination of uniform, vehicle and knife-edge loads that give the maximum and minimum result at the point. The calculation is carried out according to user specified parameters, such as the number of lanes that may be loaded simultaneously, the uniform load intensity for various lengths of load , etc.
- The user can then append load cases to the *STRAP* results file containing maximum/minimum results at each point.
- In addition, the user can select any point on the bridge and generate an influence line at that point for any result type. Both the influence lines and the corresponding load arrangement may be displayed graphically.

The basis of the calculation is the division of each lane into a series of strips. The width of each strip (perpendicular to the axis of the lane) is equal to the width of the lane and the length of the strip (parallel to the axis of the lane) is specified by the user. The program loads and solves each strip with a uniform unit load and calculates the influence lines and the critical load arrangements from the results. Similar to finite element analysis, the accuracy increases as the lanes are divided into smaller strips, but so does the solution time and the disk space required.

Refer also to:
How to use the Bridge Module 1060
Results - General 1086
Strips - General 1057

Solution Method and Program Assumptions

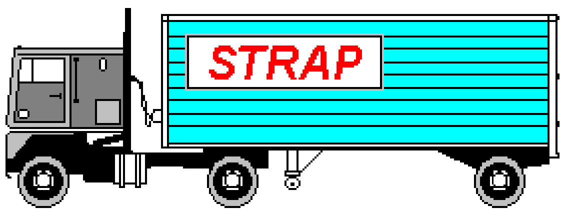
The program divides each lane into strips; the width of each strip is equal to the width of the lane at that point while the length of the strip (parallel to the axis of the lane) is defined by the user at the time the lane is defined.

- The program calculates the deflections in the entire model for a unit uniform load applied to each strip, e.g. for a model with 500 strips, the program solves 500 unit load cases.
- At every beam end (and at every 1/10 span) and for every result type (moment, shear, etc), the program searches for the strips in every loaded lane where the corresponding result for the unit load on that strip has the same sign as the result being calculated. The number of strips used corresponds to the load length defined. The program then multiplies the results from the unit loads by the load

intensity.

- A similar method is used for elements; the program calculates the results at the element centre and the corners. The maximum deflection is calculated at nodes, except at nodes that were defined as supports, where the max/min reactions are calculated.
- When calculating maximum/minimum results, the program applies the loads to the strips according to the influence line results; loads are applied only on the strips where the applied load gives a result with the proper sign. Note that the maximum load length (i.e. sum of loaded strip widths) may be limited by the user.

Related topics:
Bridge Module files 1059
Lane loads & Load cases - Example 1077

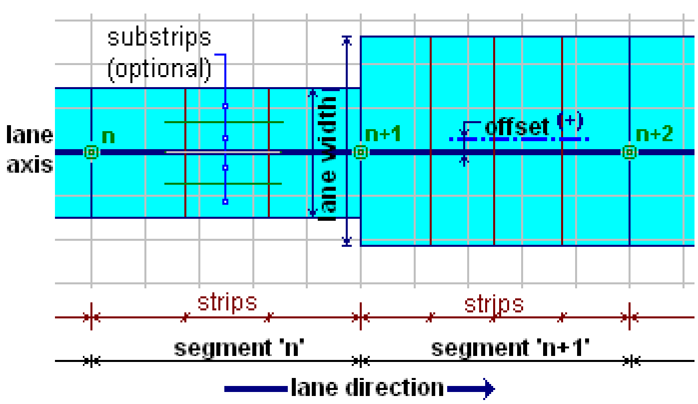


10.1.1 Strips - general

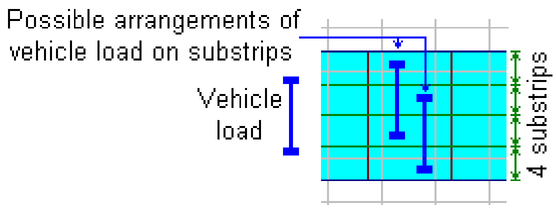
Lane strips are the basic geometry unit of the bridge module; lanes are divided into segments along their longitudinal axis and segments are divided into equal strips. The number of strips per segment is specified by the user.

When specifying the number of strips per segment, please note the following:

- When the **Solve** option is selected, the program creates a uniform unit load case for ***each*** strip in the model, i.e for a model with 500 strips the program must create and solve 500 load cases. The accuracy increases as the lanes are divided into smaller strips, but so does the solution time and disk space required.
- All loads, including vehicle and knife-edge loads are applied as uniform loads on strips; accuracy is reduced as the number of strips is decreased. For example,if the strip width in a lane is 1.0 m and the distance between axle 1 and axle 2 is 1.5 m, then the two axle loads are applied at 1.0 or 2.0 m spacing.



- if **substrips** are defined, the program checks are possible vehicle load locations across the width of the lane. For example:



10.1.2 Load factor tables

Load factor tables are stored in file **LOADFCT.DAT**. Several tables may be stored in this file.

File format:

1 table_1_title	• Title rows must start in column 1
dist1 fact1	• data rows must start in column 2+;
dist2 fact2	1+ space between "dist" and "fact"
| |	• "dist": read as "up to dist"
distn factn	• program interpolates values
2 table_2_title	• Load applied = W x fact
dist1 fact1	
dist2 fact2	
| |	
distn factn	
| |	
END	• File must terminate with "END"

The default units for **distn** are meters. To define a table with other units, add the following command before the table title:

UNITS n

where: **n** =

1 = mm **4** = inch
2 = cm **5** = ft
3 = meters

The units are in effect until another **UNIT** command is encountered in the file.

The program automatically converts the table to the current model default units.

Example for BS5400:

There are two alternatives:

a. define **W=1.0**; the table should be

```
1  BS5400 - Table 13
 30   30
 32...29.1
 34   28.3
  |
END
```

b. define **W=30.0**; the table should be

```
1  BS5400 - Table 13
 30   1.00
 32...0.97
 34   0.943
  |
END
```

- To designate a table as a South African Code 1095 (TMH7) table, add **(S)** to the end of the title (when a table with **(S)** at the end of the title is selected, the program automatically applies the lane loads according to the South African Code).

- To designate a table as a BD37/88 1095 table, add **(BD)** to the end of the title (when a table with **(BD)** at the end of the title is selected, the program automatically displays the dialog box that requests the data required to calculate the Table 14 factors).

10.1.3 Bridge module files

Module data files:

File	***Contains***
VEHICLE.DAT 1072	load and dimension data for standard vehicles. This file is copied to BRDGnnn.DAT (see below) when the bridge module is run for the first time for each model and is not modified by the program.
LOADFCT.DAT 1058	the uniform load vs load length factors

Model data files:

File	***Contains***
BRIDGnnn.DAT	Vehicles (copied from VEHICLES.DAT and those defined by user in the current model) and vehicle groups. "Revise/delete vehicles" options revise this file only (not VEHICLES.DAT) and so do not affect vehicle loads in other models.
NDLDnnn.DAT	For each strip: applied loads when loads distributed as joint loads
BMLDnnn.DAT	For each strip: applied loads when loads distributed as beam loads
RESDBnnn.DAT	Results for load applied to each strip separately

10.2 How to use the Bridge module

STRAP:

- Create a model of the bridge in the *STRAP* geometry module.
- Define all loads other than the bridge loads in the *STRAP* loading module.
- Solve the model (optional at this stage, but before *STRAP* results are displayed, design postprocessors are accessed, etc)
- Select Bridge in the tab bar.

Bridge Module:

- Define lanes:
  Select Lanes in side menu and click Define.
  Define the start and end nodes of each segment, the lane width and the number of strips per segment.
- Select the method of distributing the loads to the model:
  - select **Options** in the menu bar
  - select **Load direction** to specify the Global direction in which all loads are applied.
  - select **Load distribution** in the pulldown menu and specify the element types that the loads are applied to.(beams, specific beams, elements)
- Solve the model:
  Select **Files** in the menu bar and **Solve** in the pulldown menu. The program returns to the bridge module after completing the solution.
- Display influence lines:
  Select **Results** in the menu bar and
  - graphic display: select **Draw influence lines** in the pulldown menu
  - tabular display: select **Display influence lines table** in the pulldown menu
- Define lane loads:
  select Lane load in side menu and click Define.
  The following load types may be defined:
  - Uniform load:
    Specify the uniform lane load (units are t/m, kip/ft, etc - not load/area), the maximum length of the sum of the loaded strip widths and the load length reduction factor table.
  - Vehicle load:
    Select a vehicle load or vehicle group from the list box, define a factor to increase/decrease the load and the direction of travel (most vehicle loads are not symmetric). Note that the program can check both directions and use the worst.
  - Knife edge load (also used to define concentrated loads).
- Define load cases:
  Load cases are defined by assigning lane loads to specific lanes.
  Select Load case in side menu and click Define.
  - assign the defined lane load to the defined lanes
  - define permutations; the program creates additional load cases by interchanging the lanes the lane loads are assigned to.

  Note that load cases may be deactivated/activated.

  Refer also to Lane loads & Load cases - Example 1077
- Display a specific result for any node/beam/element or specific point. The results are the value of the

maximum/minimum result and the corresponding loaded strips, location of vehicle loads and location of knife edge loads.

- select **Results** in the menu bar
- select **Draw applied loads for selected result** in the pulldown menu
- specify max or min; select result type, specify location.

- Transfer results to *STRAP*:

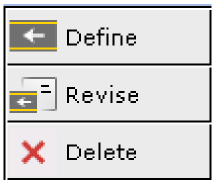
  The program appends ‘transfer’ load cases to the *STRAP* result files.

  - Maximum or minimum results may be selected.
  - For the same transfer load case, select either:
    - maximum/minimum value for each result type,
    - maximum/minimum for a specific result type and corresponding results from same calculated load case.

  Note that the transfer for very large models with many lanes, strips and calculated load cases may require several hours (the progress is displayed on the screen).

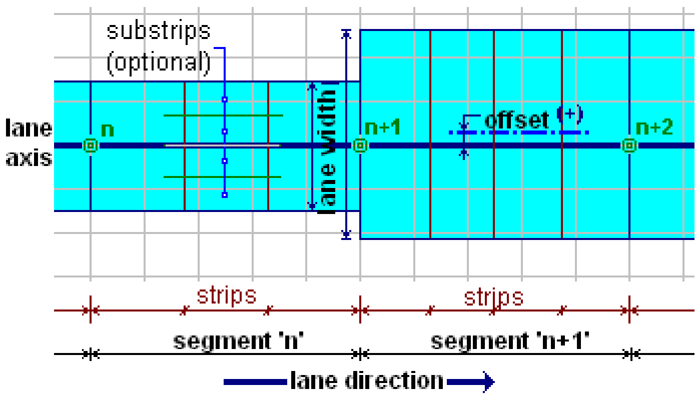
10.3 Lanes

A lane is defined as a line joining nodes. Each lane may consist of a number of segments, where the start node of any segment is the end node of the previous segment.



Lanes:

A lane is defined as a line joining nodes. Each lane may consist of a number of segments, where the start node of any segment is the end node of the previous segment.



For each segment, the user specifies the following properties:

- the lane **width**
- the **offset** of the lane centre line from the line connecting the start and end nodes. A positive offset is to the left of the line from node *n* to node *n+1*
- the division of the lane segment into **strips**. Uniform strips may be defined for all lane segments, or both the length and width of the strips may vary from segment to segment. For a 'polyline', the number of strips refers to the ***entire*** polyline
- the division of each lane into **substrips**.
- a **vertical tolerance**: loads will be applied to all nodes in the lane width that are within a specified vertical distance from the strip center line (perpendicular to the lane)

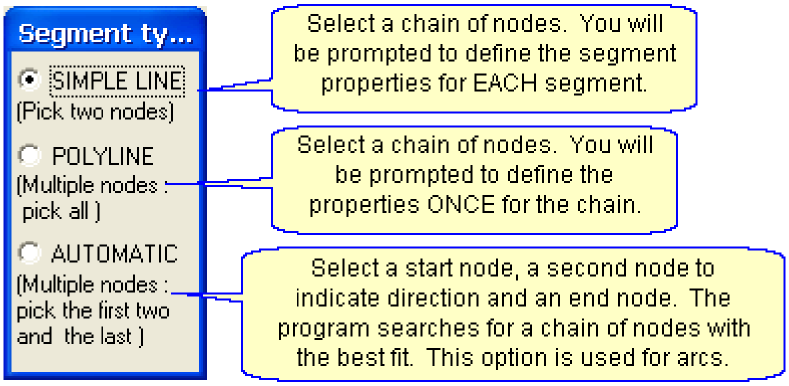
Refer to Strips - general 1057.

10.3.1 Define

A lane is defined as a line joining nodes:

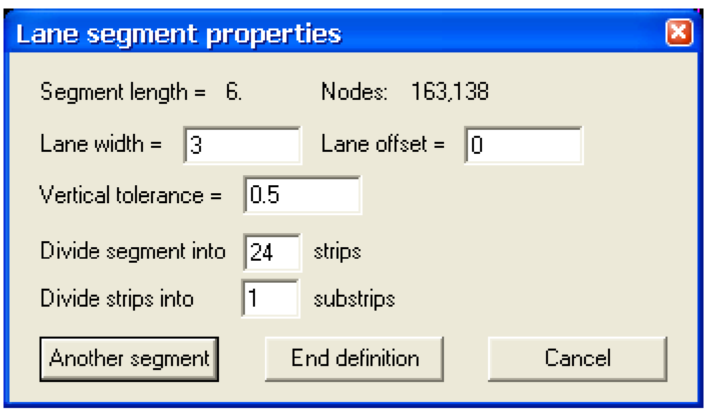
- Each lane may consist of a number of segments, where the start node of any segment is the end node of the previous segment.
- Each segment has properties (width, offset, number of strips)

Select one of the following options:



Simple line

To define a lane:
- select the lane **start** node using the standard node selection option.
- select the **end** node of the first segment
- define the segment properties 1065 (width, offset, number of strips, sub-strips):



- If there are no more segments in the lane, click the End definition button.

If there are more segments:

- click the Another segment button
- select the **end** node of the next segment
- define the segment properties 1065 (width, offset, number of strips). Note that all properties may be different for each segment.
- Repeat for additional segments.

To end, click the End definition button.

Polyline

To define a lane:
- select the lane **start** node using the standard node selection option.
- select the **end** node of the first segment
- select the end nodes of the next and all following segments

- If there are no more segments in the lane, click the last node again or click the End definition

button.
- define the segment properties 1065 (width, offset, number of strips, substrips). Refer to Strips - general 1057.

If there are more segments (with different properties):

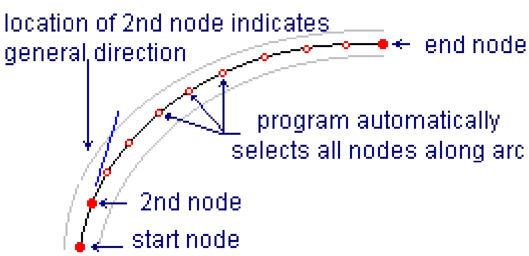
- click the Another segment button
- select the **end** node of the next and following segments
- continue as explained above.

Note:
- the "Divide segments in __ strips" option in the Lane - properties 1065 menu refers to the ***entire*** polyline.

Automatic

Define a lane along a curve formed by a chain of nodes by selecting:
- the start node
- the second node in the chain
- the end node.

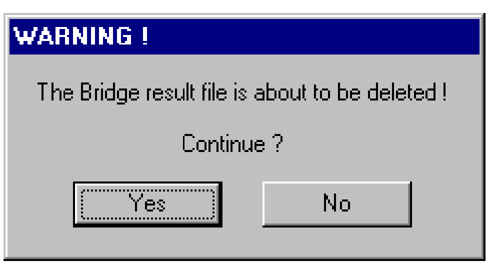
For example:



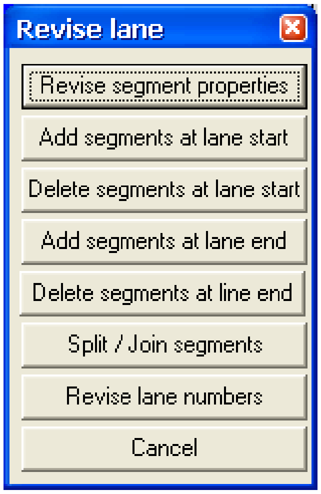
The program selects node 'n+1' that is closest to the extension of the line connected nodes 'n' and 'n-1'.

The lane is not created if the program is unable to connect the last node with the first one.

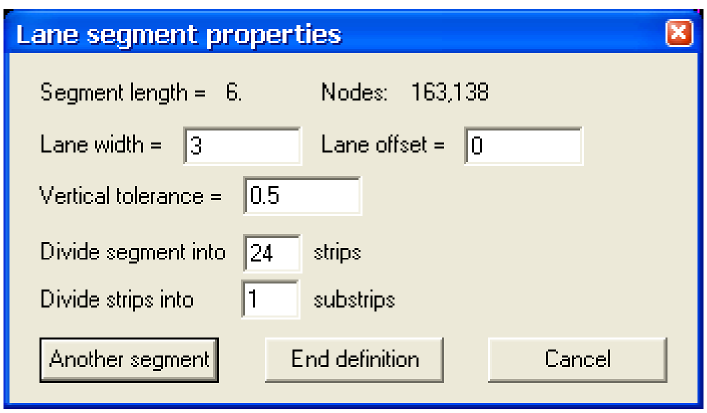
Note:
- Defining a new lane influences the distribution of the loads to the model and hence all results must be recalculated). The program displays a warning when this option is selected:



10.3.2 Revise



Revise segment properties



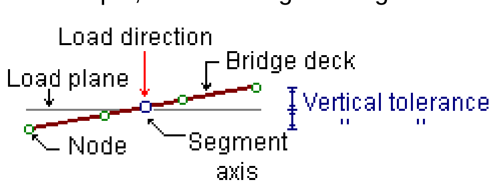
- ***Vertical tolerance:***

  The loads applied by the program are distributed to beams, elements or nodes. The distribution method and the load direction are specified by the user (refer to Options 1081).

  For each lane segment, the program draws an imaginary plane through the end nodes of the segment and perpendicular to the load direction. The program determines the distance from all nodes to this plane and applies loads to nodes/beam/elements only if they are located within a user defined distance from the plane. This distance is referred to as the ***vertical tolerance***.

  Limit the vertical tolerance to prevent the load from being applied to more than one level or enlarge the tolerance if the surface is not perpendicular to the load plane.

  For example, the following drawing shows a section through a banked bridge deck:



  Loads are not applied to the nodes if a zero vertical tolerance is specified. To apply loads to the entire deck, define the vertical tolerance as shown in the drawing.

Note that the tolerance dimension is applied both above and below the plane, i.e a defined tolerance of 1 unit creates a strip 2 units thick

Add segments at lane start

- Identify the lane by selecting any of its segment end nodes
- Specify the start node of the new beginning of the lane
- Specify the end node of the first new segment and define the segment properties
- Continue defining segments; the option terminates when the end node specified for new segment is the existing start node of the lane.

Delete segments at lane start

Specify the new start node of the lane (you may select only end nodes of existing segments); the program deletes all segments preceding this node.

Add segments at lane end

- Identify the lane by selecting any of its segment end nodes
- Specify the end node of the first new segment and define the segment properties
- Continue defining segments; the option terminates when you click the **End definition** button in the Lane properties menu.

Delete segments at lane end

Specify the new end node of the lane (you may select only end nodes of existing segments); the program deletes all segments following this node.

Split / join segments

Redefine the division of an existing lane into segments by:
- joining an interval of consecutive segments
- dividing an interval of consecutive segments

For example, an interval of four existing segments may "joined" to 2 new segments or "split" to 6 new segments.

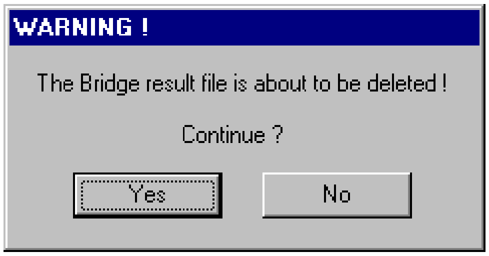
Procedure:
- Select the **start** node of the first segment in the interval to be redivided
- Select the **end** node of the last segment in the interval
- Specify the new end nodes of each of the new segments in the interval and define the properties of each of the segments

Revise lane numbers

Renumber the defined lanes. Note that the maximum lane number cannot be revised; the program can only interchange the numbers.

For example, a model with 7 lanes: Only 1 to 7 can be assigned to a lane; if lane 2 is renumbered to lane 6, then lane 6 is automatically renumbered to lane 2.

Note:
- Revising a lane (width, properties, strips) influences the distribution of the loads to the model and hence all results must be recalculated). The program displays a warning when this option is selected:



10.3.3 Delete

Delete a lane

To delete a lane completely, select any of the segment end nodes.

Delete segments from lane start

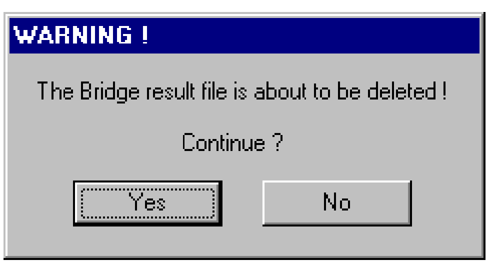
Specify the new start node of the lane (you may select only end nodes of existing segments); the program deletes all segments preceding this node.

Delete segments from lane end

Specify the new end node of the lane (you may select only end nodes of existing segments); the program deletes all segments following this node.

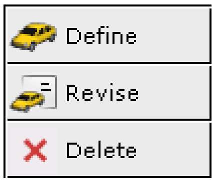
Note:

- Revising a lane (width, properties, strips) influences the distribution of the loads to the model and hence all results must be recalculated). The program displays a warning when this option is selected:



10.4 Vehicles

The program contains a file with standard vehicle loads and patterns. In addition, the program allows the user to create his own vehicle load by defining the number of wheels, the distance between them and the load applied by each wheel.



- ***Vehicles***

  The program contains a file 1072 with the standard vehicle loads and patterns specified by the various design Codes. In addition, the program allows the user to create his own vehicle load by defining the number of wheels, the distance between them and the concentrated load applied by each wheel.

  An optional associated uniform load may also be defined as part of the vehicle load.

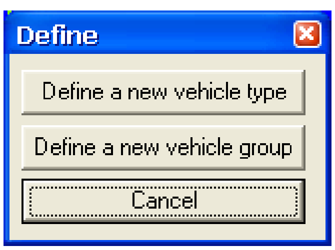
- ***Vehicle groups***

  Most Codes require that the several types of vehicles be checked, each having a different arrangement of wheels and/or a different distribution of the load. The user may define vehicle “groups” containing more than one vehicle type. The program then checks each vehicle in the group when calculating the max/min results at each point (note that different vehicles may be used for different points).

Refer also to Strips - general. 1057

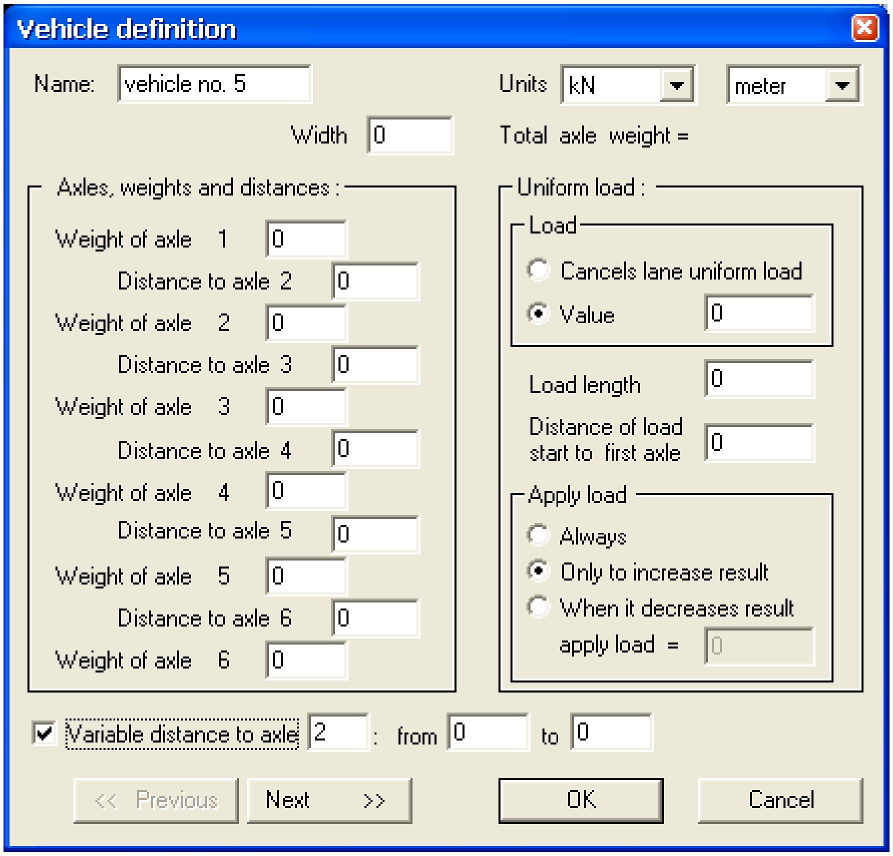
10.4.1 Define

Define a new vehicle type or vehicle group.



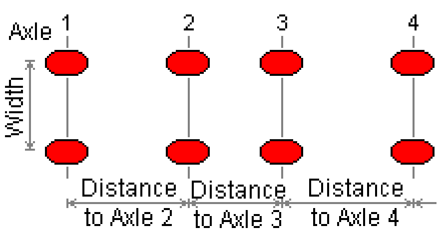
10.4.1.1 Define vehicle loads

A vehicle consists of a series of axles and an optional associated uniform load. Up to 100 axles may be defined for each vehicle.



Axles, weights and distances

The program assumes that each axle has two wheels separated by "width":



Weight:

Weight per axle = sum of the weights for the two wheels. The "**Total weight**" displayed at the top = sum of all axle weights.

For example, 2 axles:
- Weight of axle 1 = 15
- Weight of axle 2 = 20

Total weight = 15 + 20 = 35.0

Note:
- the load per axle is applied as a uniform load to the entire strip.

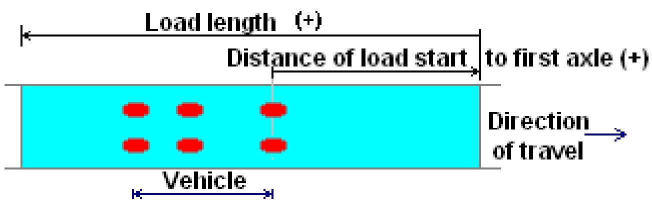
Distance:

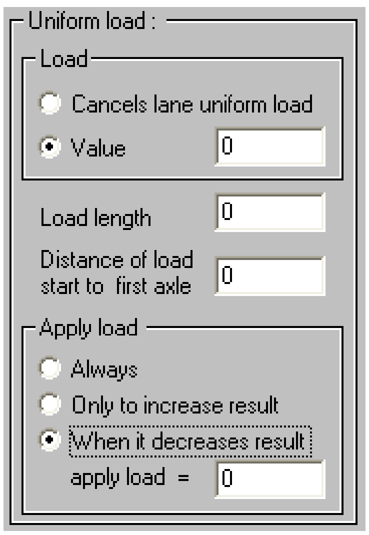
Note:

- "width" is not used by the program; ***the total load per axle is applied as a uniform load to the entire strip***.
- The distance between axles displayed in the results is the distance between the centres of the strips to which the axle is applied. For example, if the strip width in a lane is 1.0 m and the distance between axle 1 and axle 2 is 1.5 m, then the two axle loads are applied at 1.0 or 2.0 m spacing.

Uniform load

Define a uniform load associated with the vehicle load. The uniform load can represent:
- an additional uniform load that is always applied with the vehicle
- a length along the lane adjacent to the vehicle load where the uniform lane load is not applied.





Cancel / value:

Specify one of the following options:

Cancels lane uniform load
The uniform lane load is not applied on the length specified in this option.

Value
A uniform load - in addition to the uniform lane load - is applied on the length specified in this option. Specify the uniform load value; a value opposite to the sign of the uniform lane load reduces the uniform lane load in the specified length

Note:
- The program only checks the sign of the influence line in each strip and does not check whether each strip is actually loaded with a uniform lane load. Therefore, the vehicle uniform load may be subtracted from strips where the uniform lane load is not applied because of the maximum length limitation.

Load length:

Define the length of the uniform load:
- a positive length is ***opposite*** to the direction of travel of the vehicle

Distance from load start:

Specify the distance from the first axle to the start of the uniform load:
- a positive distance is in the direction of travel
- the first axle is the most forward axle in the direction of travel

Apply load:

Select one of the following options:

Always
The uniform vehicle load is always applied along with the vehicle, even if it reduces the value of the maximum result.

Only to increase result
The uniform vehicle load is applied only to those strips under the length that contribute to the maximum result.
Note:
- This is always the case when **Cancels lane uniform load** is selected or when **Value** is selected with a load of opposite sign to the uniform lane load.

When it decreases result apply load ...
- The uniform vehicle load is applied to those strips under the length that contribute to the maximum result ***and*** -
- The uniform load specified here is applied to those strips that decrease the maximum result (load = 0 gives the same result as the previous option).

Variable axle distance

Define a variable distance ***to*** axle *n* from axle (*n - 1*) . The program checks all distances within the specified range for the worst case.

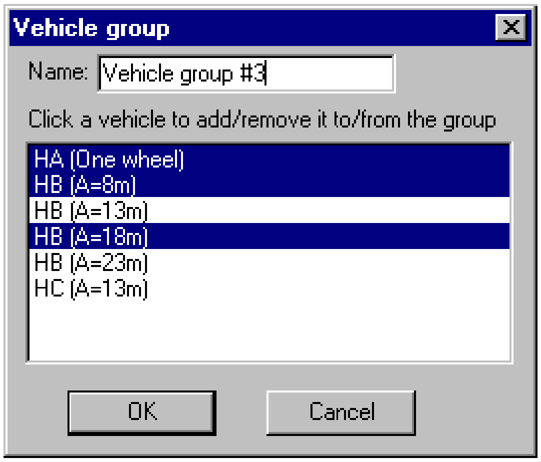
Previous / Next

Up to 100 axles may be defined per vehicle.
- click the **Next** button to display the next 5 axles
- click the **Previous** button to display the previous 5 axles

10.4.1.2 Define vehicle groups

A vehicle group consists of several defined vehicle loads. If a group is applied as a lane load, then the program checks ***each*** of the vehicle loads in the group and use the one that gives the maximum (minimum) result at the specified location.

The program displays a list of all defined vehicles. Click a vehicle name to highlight it or to remove the highlight. Only highlighted vehicles are included in the vehicle group. For example:



Three vehicles are included in Group #3: "HA (one wheel", "HB (A=8 m)" and "HB (A=18 m)"

10.4.2 Revise

To revise a vehicle or vehicle group:

- select a vehicle or group from the list box displayed on the screen.
- for a vehicle - refer to Vehicle loads - define 1068
  for a vehicle group - refer to Vehicle group - define 1071.

10.4.3 Delete

Delete any of the vehicles or vehicle groups displayed in the list box.

Note:

- deleted vehicles are automatically deleted from all groups
- deleting vehicles or vehicle groups does not affect other models.

10.4.4 Vehicles file

The file ***VEHICLES.DAT*** (in the program directory) contains the data for all standard vehicle loads and may be edited by the user to revise existing vehicles or to add new vehicles.

Note that vehicles defined by the **Define a new vehicle type** option are available only for the model for which they are defined and are not displayed in the vehicle loads list box for other models.

Each vehicle is defined by the following lines (must be in the following order):

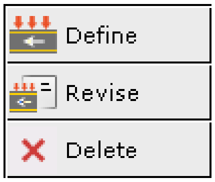
- ***Line 1***
  Title - vehicle name (displayed in the list box of vehicle loads)
- ***Line 2***
  Associated uniform load data (Optional)
  format: **UNIFORM** ***ic ia load dist length***
  where:

ic	=	1 : the uniform lane load is not applied on ***length***
	=	0 : ***load*** is added to uniform lane load
ia	=	1 : applied only when result is increased
	=	0 : always applied
load	=	pressure value (relevant when ***ic*** = 0)
dist	=	distance from 1st axle to start of load
length	=	length of load

- ***Line 3***
  Axle data
  format: ***na nw ulf uwf wid***
  where:
  na = no. of axles
  nw = no. of wheel per axle
  ulf = length factor (converts all length dimensions to meters)
  uwf = weight factor (converts all weight dimensions to tons)
  wid = width between wheels
- ***Line 4***
  list of distances; if more than one line is required then terminate lines with a ***/*** or **** (there must be a space between the text and the / or \)
- ***Line 5***
  list of weights; if more than one line is required then terminate lines with a ***/*** or **** (there must be a space between the text and the / or \)

Note:
- all data is entered in 'free-format' (the location in the line is not important)
- the following commands may be defined prior to line 1
  WEIGHT ***wt_unit***
  LENGTH ***len_unit***
  where:
  wt_unit = ton, kN, kip, kg, pound, gram
  l***en_unit*** = mm, cm, meter, inch, foot
  these commands override the ***ulf, uwf*** factors in line 3 and remain in effect for all following vehicles until a new command is encountered.

10.5 Lane loads



The following loads may be applied to a lane:

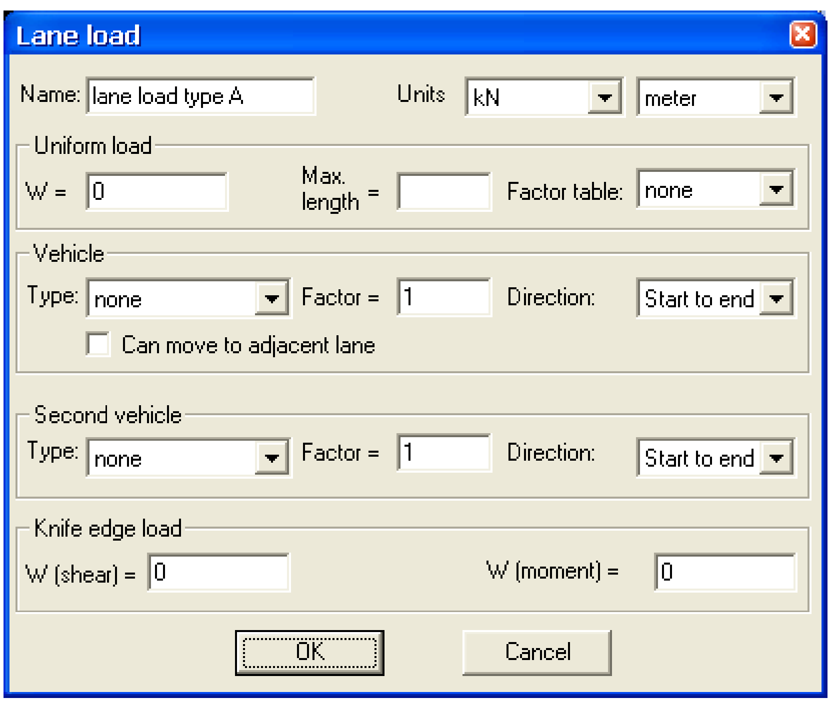
- **Uniform load**:
  Specify the load value, the maximum length of the sum of the loaded strip widths and the load length reduction factor table.
  Note:
  - units are t/m, kip/ft, etc - **not** load/area
  - loads are applied only to strips that contribute to the requested result
- **Vehicle load**:
  Specifies the vehicle or vehicle group, a factor for multiplying the loads in the file and the direction of travel along the lane. A single vehicle/group may be defined on each lane (a different vehicle/group may be applied to each lane).

  Note that a vehicle is applied to a lane only when it contributes to the requested result.
- **Knife-edge load**:
  The user specifies the load intensity. A single knife-edge load may be applied on each lane.

Refer also to Lane loads & Load cases - Example 1077

10.5.1 Define

Lane loads are a combination of uniform loads, vehicle loads and knife-edge loads.



Uniform load

- ***W***

  Specify the uniform load value per length of lane, i.e.

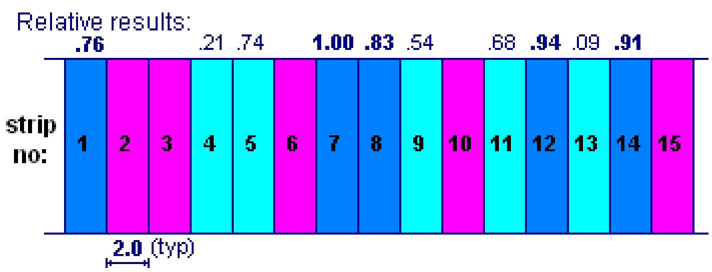
  Units are load/length (t/m, kip/ft, etc) - NOT load/area

- ***Max length***

  Specify the maximum total length of uniform load that may be applied in a lane:

  The program loads only those strips that contribute to the requested result. Use this option to limit the total width of the strips that are used. If the sum of the strip widths that contribute is greater than the length defined here, then the program uses the strips that contribute the most.

  For example: maximum length = 10 m, i.e. the program can use only 5 strips:



  The program uses strips 1,7,8,12 and 14.

  Note:
  - for South African Code TMH7, the "**Max. length**" specified refers to the sum of the lengths of the blocks that are used in all lanes. If different "Max. length" values are specified for different lane loads, the program uses the minimum length. Refer to South African Code 1095 more details.

- ***Uniform load - factor table***

  Many codes stipulate that the intensity of the uniform load applied to the lane is inversely proportional to the total length of the load (i.e. the sum of the loaded strips).

  For example: BS5400: Part 2: 1978: Table 13

Loaded length (m)	Load (kn/m)
Up to 30	30.0
32	29.1
34	28.3
etc.	....

  Select the appropriate load factor table from the list box or select "None" if this option is not applicable.

  Note that the load applied is "**W**" x factor (in table).

  Note:
  - The load factor tables are stored in file **LOADFCT.DAT**. For more information on this file, refer to Load factor tables 1058.
  - A "SA" load factor table must be specified in order to design according to the South African TMH7 Code; refer to Load factor tables 1058 for more details.

- A "BD" load factor table must be specified in order to design according to the BD37/88 Code; refer to Load factor tables 1058 for more details

Vehicle load

Note that two different vehicle loads can be applied to the same lane (as required by Eurocode). The distance between the loads can be variable and the minimum distance is specified by the Code.

- ***Type***

  Click this item to display a list box containing the titles of all defined vehicle loads and vehicle groups 1068; select one.

- ***Factor***

  The defined vehicle load may be increased or decreased by a factor.

- ***Direction***

  Vehicle loads are generally not symmetric about the axis perpendicular to the lane. Therefore, it is important to define the direction of travel of the vehicle.

  Select one of the following options:
  - **Start to end / End to start**
    Only one direction of travel is checked by the program. The direction refers to the order of the nodes selected to define the lane 1062.
  - **Both directions**
    The program assumes that the vehicle may travel in either direction and tests both possibilities.

- ***Adjacent lane***

  ☑ One set of wheels is places in the lane and the other set may be placed in the adjacent lane. If the vehicle move only slightly into the adjacent lane the program still applies an additional uniform or vehicle load in that lane.

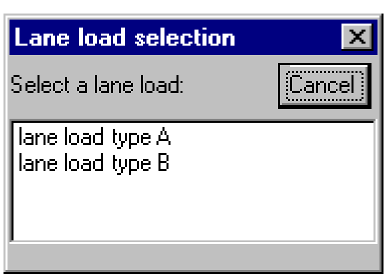
Knife-edge load

Specify the load value in weight units (total load)

Note:
- A single knife-edge load may be applied on each lane.
- Different load values may be defined for the moment calculation and the shear calculation (refer to AASHTO - Figure 3.7.6B).
- The program applies the knife-edge load as a uniform load to a strip.

10.5.2 Revise / Delete

Select an existing lane load from the list box:



For "Revise" - modify the existing lane load data 1074

10.5.3 Example - Lane loads & Load cases

The following example outlines the procedure for defining the lane loads and the load cases for a multispan, multilane bridge.

AASHTO.
Referring to AASHTO, Section 3.11:

- 3.11.4.2 - For continuous spans, the lane loading shall be continuous or discontinuous; only one standard H or HS truck per lane shall be considered on the structure
- 3.12.1 - Where maximum stresses are produced ... by loading a number of lanes simultaneously, the following percentages of live loads shall be used:
  - one or two lanes: 100%
  - three lanes: 90%
  - four lanes or more: 75%

For a four-lane bridge:

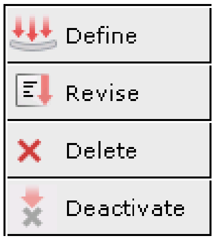
A. Define three lane loads, each with a uniform and concentrated load according to Figure 3.7.6B:
lane load 1 - full load
lane load 2 - 90% of full load
lane load 3 - 75% of full load

B. Define another three lane loads, each with vehicle loads according to Figure 3.7.7A:
lane load 4 - full load
lane load 5 - 90% of full load
lane load 6 - 75% of full load

C. Define six load cases, as follows:
1. lane load 1 on lanes 1,2 ; permutations on lanes 1 to 4
2. lane load 2 on lanes 1,2,3 ; permutations on lanes 1 to 4
3. lane load 3 on lanes 1,2,3,4 ; permutations on lanes 1 to 4
4. lane load 4 on lanes 1,2 ; permutations on lanes 1 to 4
5. lane load 5 on lanes 1,2,3 ; permutations on lanes 1 to 4
6. lane load 6 on lanes 1,2,3,4 ; permutations on lanes 1 to 4

10.6 Load cases

cases are defined by assigning lane loads to specific lanes.



A load case is defined by assigning lane loads to specific lanes.

The user may define a series of load cases. Note that the term "cases" refers only to a general framework for the actual load arrangement:

- the program applies the loads to each of the strips along the length of the lanes; only those loads that contribute to the requested maximum/minimum result are used
- the program can generate "permutations" of the defined load case by rotating the lane loads amongst the selected lanes. For example, the model contains four lanes and the user defines the loads on one lane only. The program then generates three additional load cases by applying the same load to each of the three other lanes.

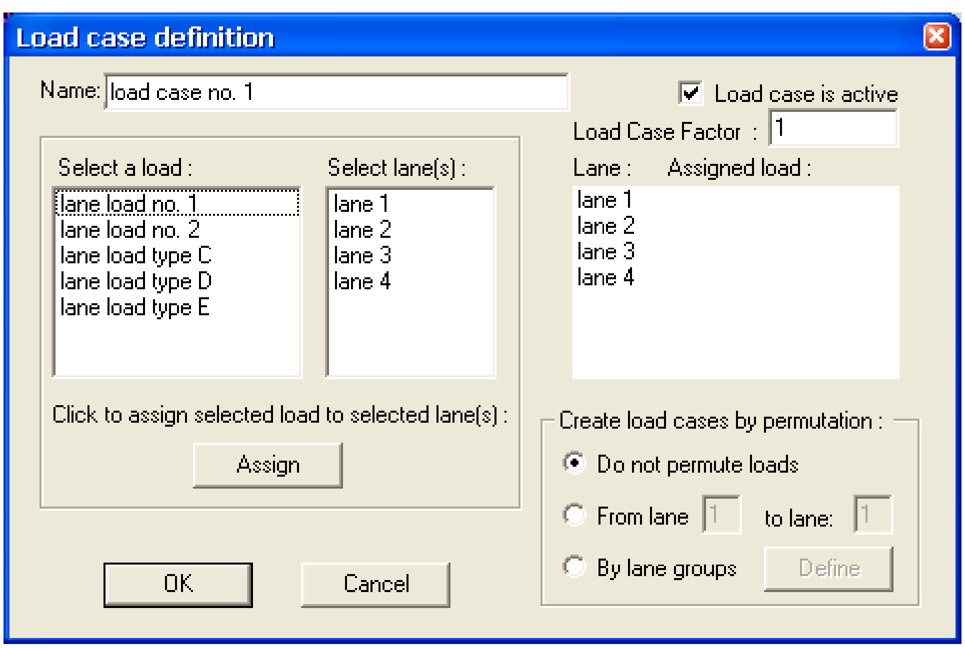
Some Codes allow load reduction, where the amount of reduction is a function of the number of loaded lanes. The user can define several load cases, where in each case the number of loaded lanes and hence the load intensity are different.

The user may deactivate load cases and reactivate them at any time.

Refer also to Lane loads & Load cases - Example 1077

10.6.1 Define

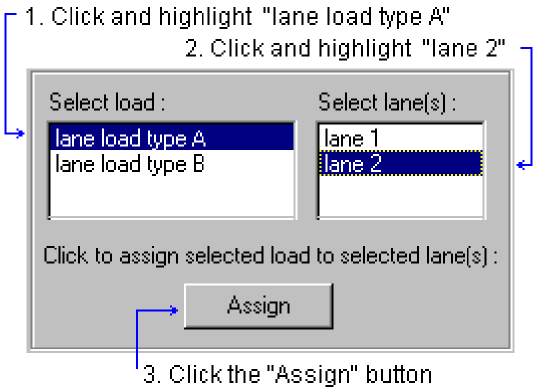
Define a load case by assigning a lane load to each lane:



Assign

Assign lane loads to lanes in order to define a load case:

For example, assign "lane load type A" to " lane 2":



The list box at the right is revised to:

Load case is active

Load cases may be activated/deactivated at any time; all permutations of the load case are activated/ deactivated.

Note that several cases may be activated/deactivated at the same time in the  option.

Load case factor

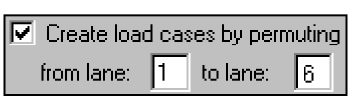
Apply a reduction factor to each load case (as required by the AASHTO code in certain cases).

Permutations

Load cases are defined by assigning lane loads to specific lanes. The user may instruct the program to generate all possible permutations of the lane loads on the selected lanes. The program creates load cases by interchanging the lane loads.

Two methods are available:

- **from lane *m* to lane *n***
  For example: a bridge with 6 lanes and the lane load must be applied simultaneously on any two lanes, i.e. 15 load cases must be created. To create them automatically, assign the lane load to lanes 1 and 2 (or any 2 of the 6 lanes), set this option to ⦿ and specify lanes 1 to 6:



- **By lane groups**
  For example: 'lane load 1" is to be applied to lanes 1 **or** 6 while 'lane load 2' is to be applied to **any two** of lanes 2 to 5:
  - assign 'lane load 1' to lane 1 (or 6)

- assign 'lane load 2' to any two lanes 2 to 5
- set this option to and click Define.
- set the list box to:

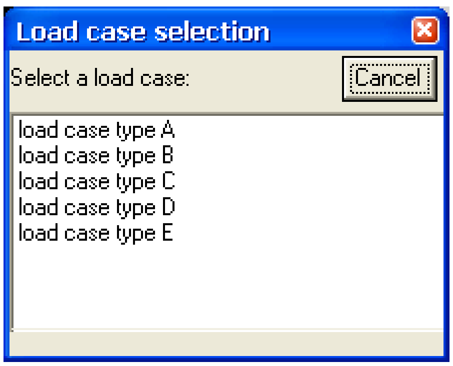
Lane	Groups
lane 1	1
lane 2	2
lane 3	2
lane 4	2
lane 5	2
lane 6	1

A total of 6x2 = 12 load cases are generated

Refer also to Lane loads & Load cases - Example 1077

10.6.2 Revise/delete

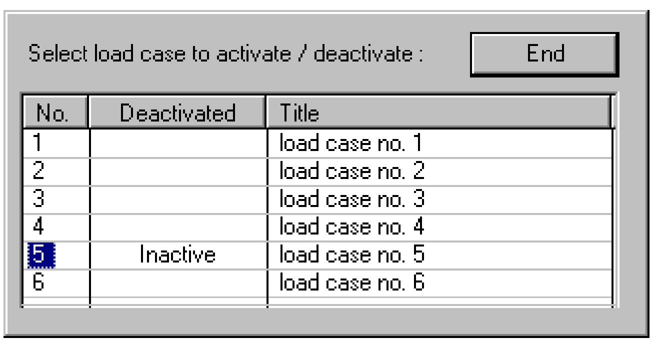
Select an existing load case from the list displayed:



When revising a load case, modify the defined data 1078.

10.6.3 Deactivate

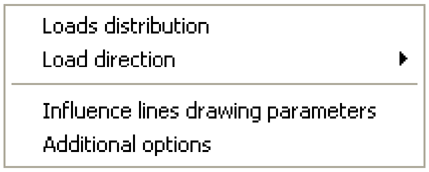
Load cases may be activated/deactivated at any time. Click on any line to toggle the status:



Note:

- all permutations of the load case are activated/deactivated.
- this option is equivalent to the **Load case is active** option in the Load case - Define 1078 option.

10.7 Options



Load distribution

Refer to Options - load distribution [1081].

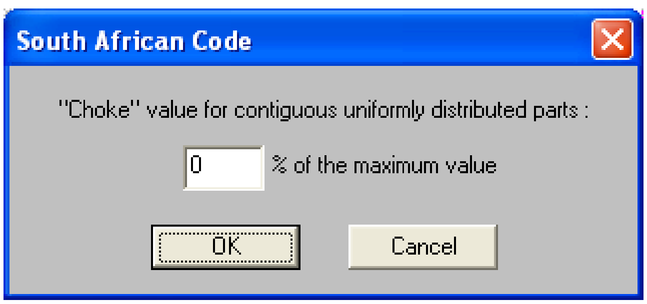
Load direction

For space models, select the global direction to apply the loads:

Influence line parameters

Refer to Options - influence line parameters [1082].

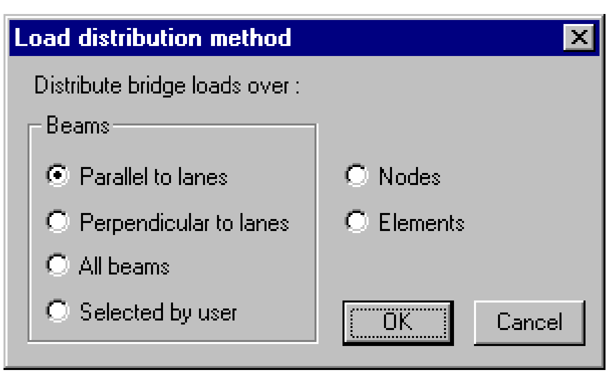
Additional options



For any influence line, the program does not apply the load on strips where the value is less than the specified percentage of the maximum value in the entire influence line diagram.

10.7.1 Load distribution

The uniform/vehicle/knife-edge loads may be applied to the model in one of the following ways

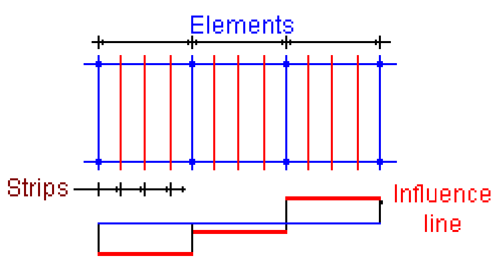


Select one of the following options:

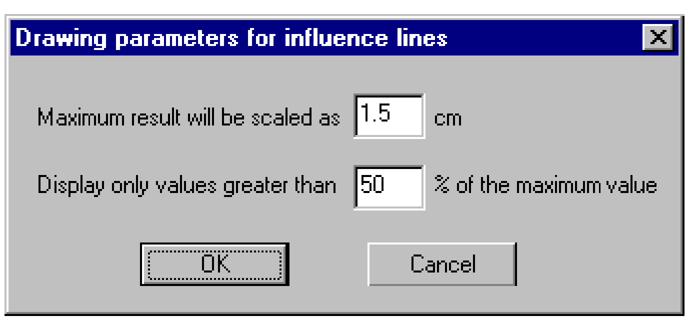
- **Beams "parallel" to lanes**
  Loads are applied as beam loads, but only to beams parallel to the lane axis (angle between the lane axis and the beam x1 local axis < 10°)
- **Beams "perpendicular" to lanes**
  Loads are applied as beam loads, but only to beams perpendicular to the lane axis (angle between the perpendicular to the lane axis and the beam x1 local axis < 10°)
- **All beams**
  Loads are applied as beam loads to all beams
- **Beams selected by the user**
  Loads are applied as beam loads only to beams specified by the user using the standard beam selection option
- **Nodes**
  Loads are applied as joint loads
- **Elements**
  Loads are applied to elements only

Note:

- Loads are applied to only those nodes/beams/elements lying within the "vertical tolerance [1065]" of the lane segment.
- Beam and element loads are applied as "Global" loads and not as "Global projected" loads.
- The loads are applied to the nodes/beams/elements according to the same methods used for applying Global loads. Refer to Global loads - Method of Application.
- When a strip width is less than the element dimension, the location of the strip on the element is ignored. The load on the strip is divided equally to the corner nodes; therefore the influence of all strips on the element is identical and the influence line is stepped.



10.7.2 Influence line parameters



Maximum result will be scaled as

The influence line diagrams are displayed relative to a scale chosen as follows:

The program searches for the maximum result in the plot area and plots it on the screen as the dimension listed above - the default value is 1.5 cm (0.6 in.). All other results are plotted in proportion to this value.

Move the into the text box, type a new dimension in cm. and press [Enter].

Example: bending moment diagram:
- Maximum moment = 12 kNm is drawn as 1.5 cm. on plotted diagram;
- Moment = 4 kNm will be drawn as 0.5 cm.

Display only values greater than

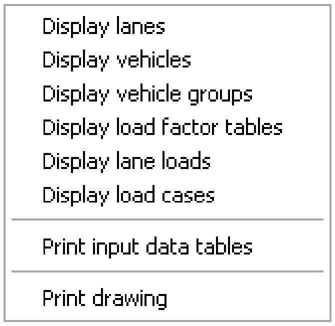
For clarity, part of the numerical values may be deleted from screen (the entire geometry and influence line are plotted).

All values less than a given percentage (default = 50%) of the maximum value on the line are not displayed.

Example:
- Maximum result on moment influence line = 12 kN m and fraction = 0.5 : Only numbers greater than 6 kN m are displayed on the screen.

10.8 Output

Display input data in tabular form:



Lanes

For each lane, display details of segments:
- segment no.
- start and end nodes
- width
- no. of rectangles
- rectangle size
- total length of segment
- vertical tolerance

Refer to: Lanes [1062], vertical tolerance [1065]

Vehicles

Display a list of the available vehicles (those copied from the general file and those defined in this model):
- axle loads
- distance between axles

Refer to vehicles [1068].

Vehicle groups

Display a list of the vehicles in each defined vehicle group.

Refer to vehicles [1068].

Load factor tables

Display the defined load vs. applied length tables.

Refer to:
load factors [1075]
bridge module files [1059]

Lane loads

For each lane load display:

- **Uniform load**:
  - force
  - maximum length

  - factor table
- **Knife-edge load**:
  - load for moment calculation
  - load for shear calculation
- **Vehicle load**:
  - vehicle or vehicle group name
  - load factor
  - direction of travel

Refer to Lane loads 1074

Load cases

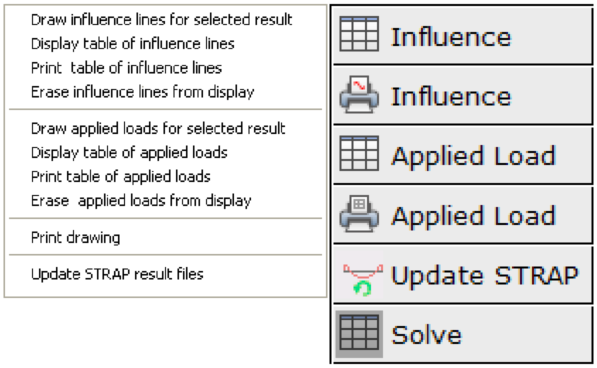
Display a table listing the lane loads applied in each load case.

Refer to:
- Load cases 1078
- Lane loads and load cases - example 1077
- Permutations 1079

Print

Print all tables

10.9 Results



Note:

- Select **Erase ... from display** to erase the current influence line diagram / applied loads diagram from the display.
- Refer also to [Results - General](1086)
- the model cannot be solved until the Load distribution method has been specified.

10.9.1 General

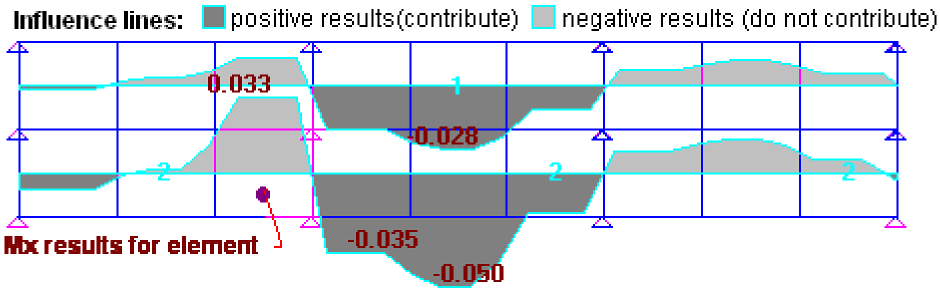
The user may select two different result types for any node/beam/element:

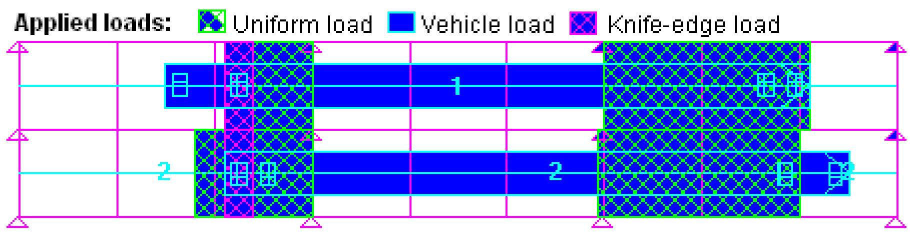
- influence line
- applied loads diagram

Influence line:
The program displays the influence line for unit loads applied to each strip in each lane, i.e. the contribution of a unit load on each strip to the specified result for the selected node/beam/element. A separate diagram is drawn for each lane in the model.

The most important information on the influence lines are the regions of positive and negative values. Strips with positive values contribute to the maximum result when loaded; Refer to the example below.

Applied loads:
The program displays the loads applied to the various strips that are required to generate the max/min result specified by the user. The display always corresponds to the influence line diagram for the same result and location, i.e. for maximum results, strips with positive values in the influence line diagrams will be loaded (unless the loaded length is limited by the user). This is illustrated in the following example.





Note that all loads, including vehicle and knife-edge loads, are applied as uniform loads on strips; accuracy is reduced as the number of strips is decreased because the program applies these loads to the closest strip and does not maintain the exact location.

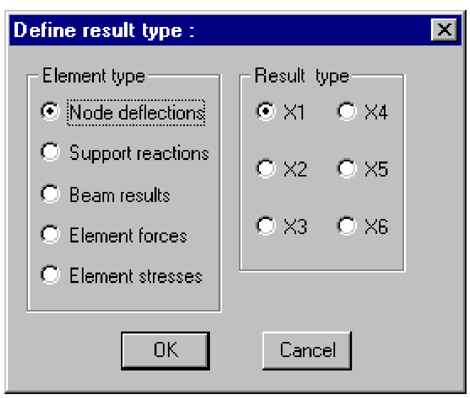
In the example above, the axle spacing for all vehicles is 1.5 m. The strip width in lane 1 is 1.0 m and 0.83 m in lane 2. The axle loads are applied at 1.0 and 2.0 m spacing in lane 1 and at 1.67 m spacing in lane 2.

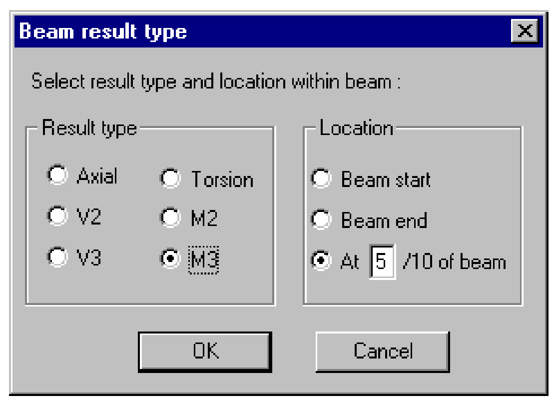
10.9.2 Influence lines

Display influence line values in graphic or tabular form.

- Select the result type and location

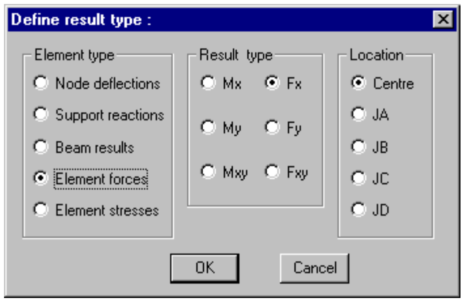
Node deflections / support reactions   Beam results





Results can be requested at any 1/10 point of the beam.

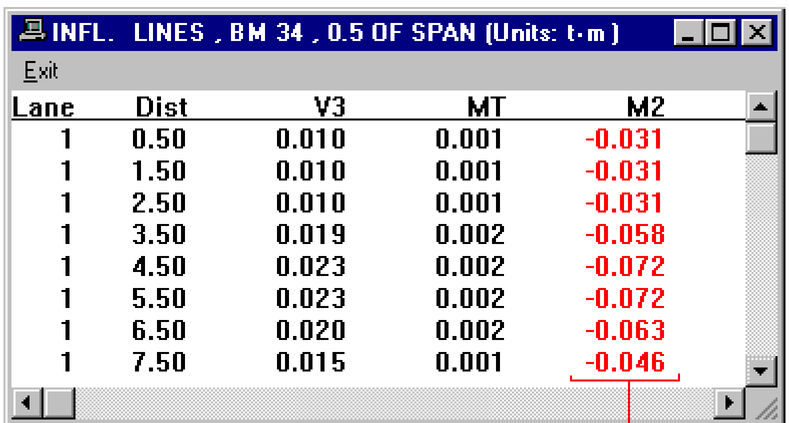
Element results



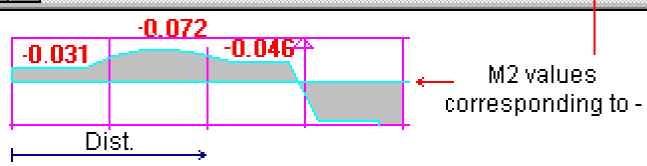
Results can be requested at any corner or the element centre.

- Select a node/beam/element.

Example (beam results):



Lane	Dist	V3	MT	M2
1	0.50	0.010	0.001	-0.031
1	1.50	0.010	0.001	-0.031
1	2.50	0.010	0.001	-0.031
1	3.50	0.019	0.002	-0.058
1	4.50	0.023	0.002	-0.072
1	5.50	0.023	0.002	-0.072
1	6.50	0.020	0.002	-0.063
1	7.50	0.015	0.001	-0.046

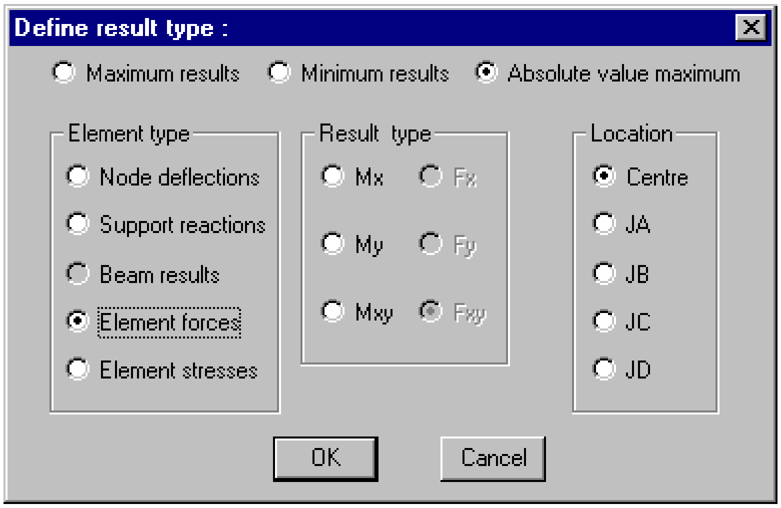


Refer also to [Results - General](1086)

10.9.3 Applied loads

Display the loads that must be applied to the model to generate the max/min result at the specified location. Refer to [Results - General](1086)

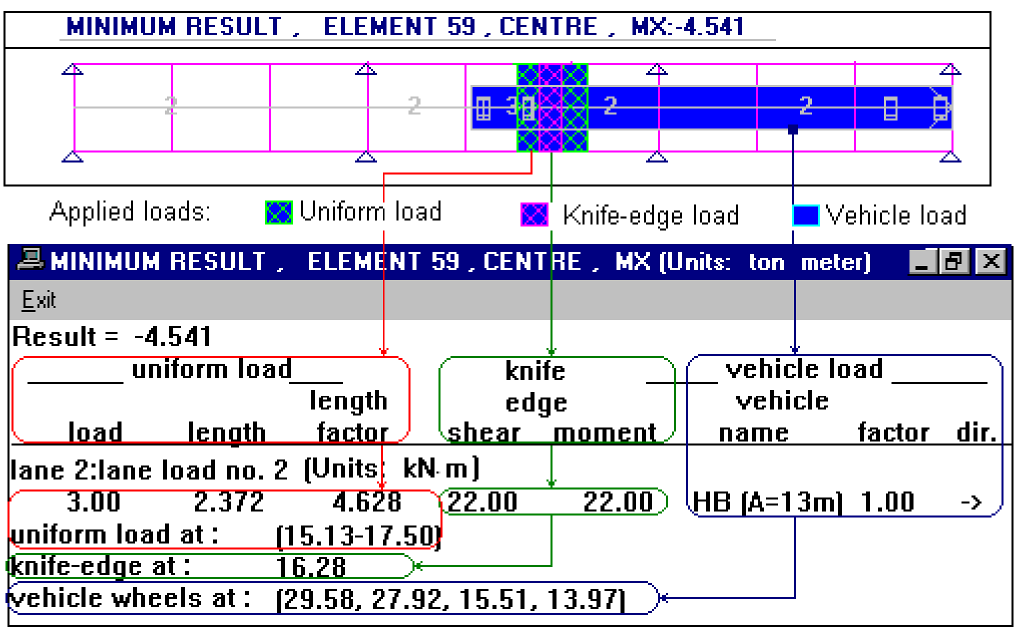
- Specify maximum or minimum results and the location:



For each node/beam/element, the program calculates separately one of the following envelope results:

- **Maximum**
  The largest positive result or the smallest negative result
- **Minimum**
  The largest negative result or the smallest positive result
- **Absolute value maximum**
  The largest absolute value, with the correct sign. For example, 3 load cases, results = 1.4, 5.5, and -7.7: The program writes -7.7 in the results file.

The program draws the applied loads on the graphic display. The following example shows the graphic and tabular displays for "Applied load results":



All loads are applied to strips:

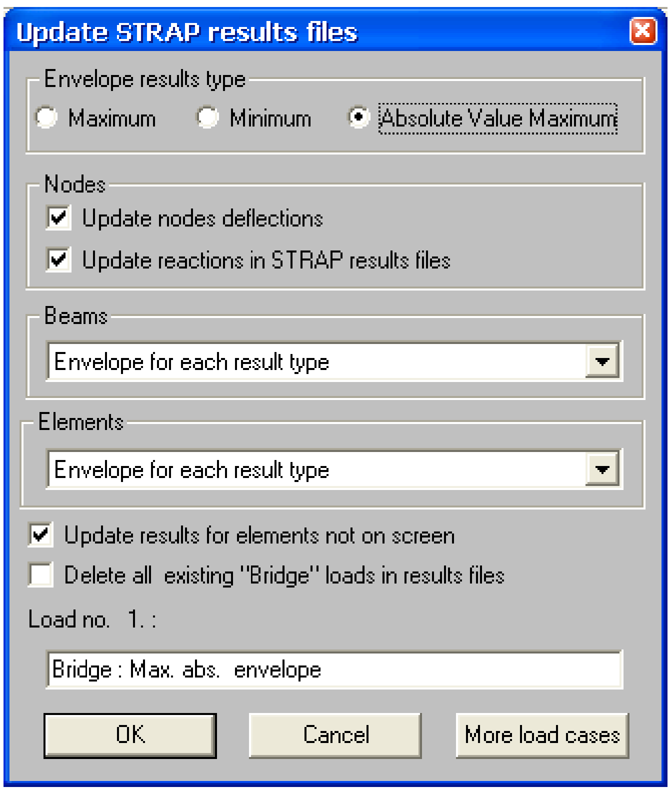
- Uniform loads (standard and uniform vehicle):
  The load always starts at the beginning of one strip and terminates at the end of another.

- Knife-edge loads:
  The load is applied as a uniform load to an entire strip; the dimension in the table refers to the centre coordinate of a strip.
- Vehicle loads:
  Vehicle loads are always applied as a uniform load to an entire strip. Therefore, the program cannot maintain the exact axle spacing specified in Vehicle loads. The coordinates in the table refer to centre coordinates of strips.

Note:
- The vehicle may be placed so that it is traveling in either direction on the lane. The direction is shown in both the table and the graph (by an arrow at one end of the box denoting the vehicle load). The direction in the table is relative to the lane definition (start node and end node).

Refer also to Results - general 1086.

10.9.4 Update STRAP results



Envelope results types

For each node/beam/element, the program calculates separately one of the following envelope results:

Maximum
The largest positive result or the smallest negative result

Minimum
The largest negative result or the smallest positive result

Absolute value maximum
The largest absolute value, with the correct sign. For example, 3 load cases, results = 1.4, 5.5, and -7.7: The program writes -7.7 in the results file.

Node deflections

☑ The program writes deflections in the generated load case.
☐ The program writes zero deflection values in the result file.

Note:

- the deflection values are either **Maximum,** **Minimum,** or **Absolute value maximum**
- the values are calculated separately for each global direction, i.e. the deflections for each direction may be from a different load case.

Update reactions

☑ The program writes reactions in the generated load case. Note that the calculation of the reactions requires a lot of time (relative to the calculation of node/beam/element results) and this may be significant for models with many supports.
☐ The program writes zero reaction values in the result file.

Note:

- the reaction values are either **Maximum,** **Minimum,** or **Absolute value maximum**
- the values are calculated separately for each global direction, i.e. the reactions for each direction may be from a different load case.

Beams

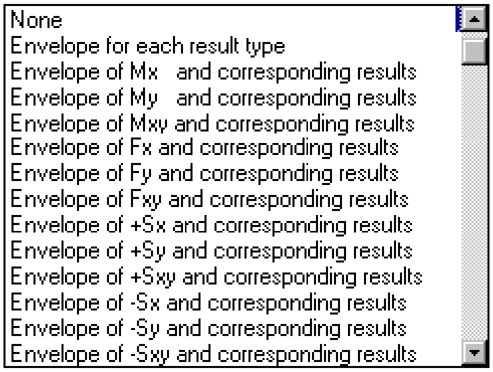
The program writes P, M2, M3, MT, V2, V3 values for each beam in the results files. Select one of the following options:

None
Envelope for each result type
Envelope of axial force and corresponding results
Envelope of V2 and corresponding results
Envelope of V3 and corresponding results
Envelope of torsion moment and corresponding results
Envelope of M2 and corresponding results
Envelope of M3 and corresponding results

- **None**
  Beam results are not written to the file; only element results, reactions and displacements are written (if selected)
- **Envelope for each result type**
  The program searches separately for the max/min/abs result for **each** result type, i.e. the results written for each type (e.g. M2 and V3) may be from different load cases.
- All other options:
  The program identifies the load case that gives the max/min/abs result for the specified result type and writes the results from that case for **all** result types.

Elements

The program writes Mx, My, Mxy, Fx, Fy, Fxy, ±Sx, ±Sy, ±Sxy values for each element in the results files. Select one of the following options:



- **None**
  Element results are not written to the file; only beam results, reactions and displacements are written (if selected)
- **Envelope for each result type**
  The program searches separately for the max/min/abs result for **each** result type, i.e. the results written for each type (e.g. Mx, My, etc.) may be from ***different*** load cases. Note that the Sx,Sy,Sxy may be calculated incorrectly if this option is selected as the values of moment and force used to calculate the stress may also be from different case. It is recommended that one of the following options be used to transfer the min/max stress results.
- **Envelope of ___ and corresponding results**
  The program identifies the load case that gives the max/min/abs result for the specified result type and writes the results from that case for **all** result types.

Elements not on screen

Select one of the following options:

☑ The results for the entire model are updated.

☐ Only the results for the elements displayed on the screen are updated, i.e, use the **Zoom** and **Remove** options to update selected elements only. ***Results for all other elements are written as zero***.

Delete existing result files

Select one of the following options:

☑ All load cases in the result file from the first bridge load to the end of the file are erased and the new load cases are written (if you solved static loads after generating bridge results, the static load case results are lost!)

☐ The program checks whether a load case with the same title was written previously to the results file for the model. If such a load case exists, the program automatically overwrites the case. If the load case does not exist, it is appended to the end of the result file.

Load case 'n'

Define a series of load cases to write simultaneously to the results file.

Note that **Load no. 1** appears at the bottom of the box:
- specify all of the options in the dialog box
- enter a title for the load case (the default titles correspond to the **Maximum/Minimum/Absolute value maximum** option selected)

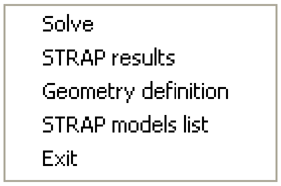
To define the next load case:

- click the Additional load button; **load no. 2** appears at the bottom of the box.
- specify the options for the new load case and enter a title.
- repeat for additional load cases.
- click OK to update the result files.

Note:

- Cancel decreases the load case counter by 1.

10.10 File menu



Solve

Solve the model for the defined bridge data (the program solves each of the strip unit load cases - refer to Bridge - general 1056).

Note:
- the model cannot be solved until the Load distribution method has been specified.

STRAP results

Display the results for the current model. Both the regular static results as well as the generated bridge load cases (refer to Update STRAP results 1090) will be available. All defined bridge data is saved.

Geometry

Return to the *STRAP* geometry module for the current model. All defined bridge data is saved.

STRAP models list

Return to the *STRAP* main menu. All bridge data is saved.

Exit

Quit the program and return to the Windows program manager. All data is saved.

10.11 Codes

Select one of the following Codes:

BD 37/88 1095
South Africa TMH7 1095

10.11.1 BD 37/88

Type HA uniformly distributed loads are stored in the BD 37/88 load factor table according to Code Table 13. These loads must be modified by the factors in Code Table 14 before being applied to the notional lanes.

The program automatically prompts for the parameters required to calculate these factors if a BD load factor table is specified.

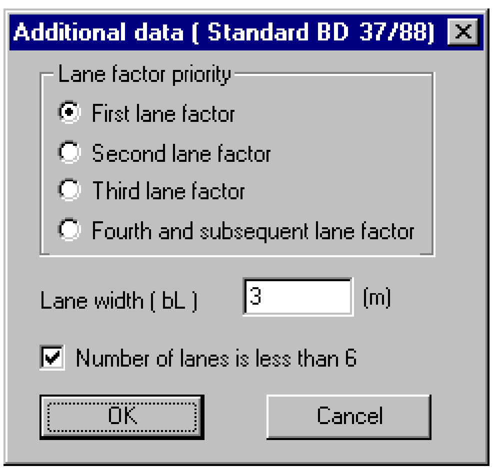
To design a bridge according to BD 37/88:

- define a different lane load for up to 4 lanes, where "1st lane factor" is specified for the first lane, "2nd lane factor" is specified for the second, etc.
- define load cases as permutations of the above lane loads; the program arranges the lane loads (reduced by the Table 14 factors) in all possible permutations across the width of the bridge.

BD 37/88 factors:

Type HA uniformly distributed loads are stored in the BD 37/88 load factor table according to Code Table 13. These loads must be modified by the factors in Code Table 14 before being applied to the notional lanes.

The following data must be specified to calculate the Table 14 factors:



- lane factor priority - to determine the column in Table 14
- lane width **bL** - to calculate factors **a1** and **a2**
- Number of lanes - less than 6 or greater/equal to 6: required when L > 50 for 2nd, 3rd and 4th lane factors.

10.11.2 South Africa TMH7

To design a bridge according to the South African TMH7 Code:

- ensure that an appropriate load factor table is included in the load factor file 1058; factor tables designated as South African must have **(S)** appended to the end of the table title.
- when defining lane loads, select a factor table that has **(S)** appended to the table title; when a table

with **(S)** at the end of the title is selected, the program automatically applies the lane loads according to the South African Code).

Design method:

The method used corresponds to TMH7 - Section 2.6.3 - Type NA loading.

- the program first calculates the contribution to the requested result of each of the strips in all the lanes .
- the program then identifies "blocks" of contiguous strips that have a positive contribution.
- the "blocks" (in all lanes) are ranked in the descending order of their contribution.
- the load on the first block is applied to the lane using the load factor corresponding to its length.

The following procedure is then carried out for each of the subsequent blocks:

- the program adds the length of the next block (block '*n*') to the sum of the lengths of the previous blocks, determines the load factor corresponding to this new length from the table and calculates the total load applied to blocks *1 to n*.
- the program subtracts from this value the load applied to blocks *1 to n-1* and applies the remainder uniformly to block *n*.
- if the remainder is less than zero, the program stops the calculation. If the remainder is positive, the program adds block '*n+1*' and repeats the calculation.

Example: Basic load = 36

Block	Length	ΣLen	Factor	Load	Remain	Applied
1	30	30	1.000	1080	1080	1080/30=36
2	30	60	0.812	1754	674	674/30=22
3	35	95	0.680	2324	570	570/35=16
4	40	135	0.600	2901	577	577/40=14
5	etc					

Note:

- If different factor tables are specified for different lane loads, the entire calculation will be according to the South African Code if at least one of the factors tables is designated as South African **(S)**.
- Because the above calculation method considers loads on all of the lanes, a single load factor table must be used; if different **(S)** tables are specified for different lane loads, the program uses the table specified for the lowest lane number in the current load case.
- The "**Max. length**" specified refers to the sum of the lengths of the blocks that are used in all lanes. If different "Max. length" values are specified for different lane loads, the program uses the minimum length.

Part XI

Prestress

11 Prestress

11.1 Main menu

PRESTRESS is a *STRAP* postprocessor program that designs pre/post-tensioned beams and post-tensioned slabs.
For general information refer to:

- General 1099
- How to use the program 1100
- Design assumptions

Beams / **Slab** — Note that the design procedure and parameters are different for beams and slabs; specify the element type before starting the design procedure.

Define	Create beams to be pre- or post-tensioned by selecting one or more continuous *STRAP* beam members. Add/delete supports at node locations and modify the top/ bottom beam levels in individual spans.
Design	Select cables and define their trajectory in the beam; specify loss parameters for the current beam; display/print all output tables (stresses, ultimate moment, shear, etc.)
Copy	Create a new beam and copy the cables and parameters from an existing beam to the new one.
Defaults	Define the default parameters for ***all*** beams in the model, including parameters for cables, reinforcement, losses, time steps, etc. Note that different loss parameters may be assigned to specific beams using the **Design - losses** option.
Restraints	Add/remove supports at different geometry stages for all beams. Supports for a specific beam may also be defined in the **Design** menu.
Section	Assign different *STRAP* sections to selected spans; add a topping to a section to create a composite beam and specify the casting stage.
Stages	Create a "stage table" when all cables are not prestressed at the same time or all loads are not applied at the same time. Each stage is defined by the number of days from the start of construction and may be linked to a different *STRAP* model.
Load table	Expand the combination table; specify the stage at which the load is applied or removed; designate the loads as service or factored
Solve	Solve the model for the cable force loads, including losses, at each of the stages; secondary moments are calculated.
Display ta...	Display all result or data tables on the screen.
Print table	Print the result or data tables.

File Zoom Rotate Display ReMove

11.2 General

PRESTRESS is a *STRAP* postprocessor program that designs pre/post-tensioned beams and post-tensioned slabs. The design may be carried out according to one of the following codes:

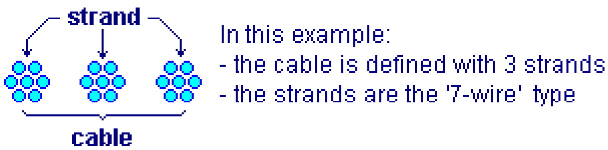
- AASHTO
- ACI318
- BS5400
- BS8110
- CSA A23,3
- Eurocode 2
- IRC
- Other codes

Both pretensioned and posttensioned cables can be defined in the same beam or slab.

The program uses the following terminology:

- **strand** - the basic prestressing unit, a single wire or a group of wires
- **cable** - a group of strands

For example:

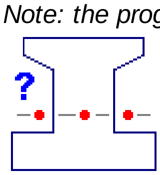


The *STRAP* model must either be a plane frame or a space frame; grids will not be accepted by the program (axial forces from prestressing cannot be added to load cases).

The following *STRAP* beam sections are accepted by the program:

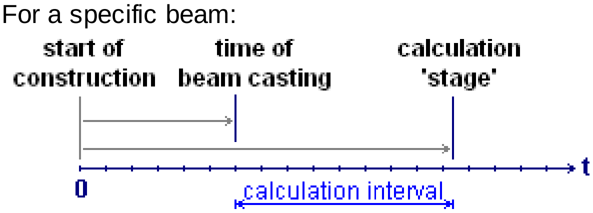
- all sections defined by "Dimensions", except round sections.
- all solid sections defined in *CROSEC* and imported into *STRAP* geometry.
- All sections defined by "Properties" (A,I, etc). The program creates an equivalent rectangular section, as follows:
  - H2 and H3 are defined: D2 = H2 and D2 = H3
  - I3 and A are defined: D2 = v(I3*12/A) and D3 = A/D2
  - I2 and A are defined: D3 = v(I2*12/A) and D2 = A/D3

Note: the program does not check whether the defined cables actually lie within the section.



Calculation of losses and deflections are calculated at various times stages for each beam measured from the day the beam is cast. All dates (casting and stages) are measured from an arbitrary zero date referred to as the 'start of construction'.

For a specific beam:



Refer also to:

- How to use the program 1100
- Design assumptions

11.2.1 How to use the program

PRESTRESS is a *STRAP* design module that designs pre- and post-tensioned beams and slabs in solved *STRAP* models.

STRAP model

The *STRAP* model must be defined as follows:

- Define the model geometry; if the geometry changes during construction (e.g. segmental construction, composite slab, etc), then each geometry construction stage must be defined in a different 'stage'. Refer to 'STRAP stages' 1101 below.
- Define load cases with all loads, but without prestressing loads.
- Solve the model.
- Define load combinations in the results module.
- Click the Prestress tab.

PRESTRESS

To design prestressed beams and slabs:

- Select the element to be designed at the bottom right corner, beams or slab.
- Specify the default parameters 1145 for all prestressed beams or slabs, e.g. concrete type, prestress loss parameters, etc.
- Define PRESTRESS stages 1167. PRESTRESS stages are defined by the number of days from the start of construction. PRESTRESS stages must be defined for any point in time where:
  - Change in geometry. (STRAP construction stage defined in geometry).
  - When a load is introduced. Prestress load or external load (load case or combination defined prior to PRESTRESS).
- Define all of the beams to be prestressed by selecting the start and end *STRAP* member.
- Define all of the slabs to be prestressed by specifying the start and end nodes of prestressing "lines" that pass through the relevant elements, and the perpendicular width (the lines do not have to be parallel to element boundaries). Each slab is designed with all cables are parallel to the line.
  Note that there are two options for defining slabs 1109:
  - Define a ***slab with a single cable*** (center line and "influence width"):
    The program calculates the result diagrams along the defined line similarly to the "Results along a line" option in the *STRAP* results module, i.e. the results are ***per unit width***, e.g. ton-meter/meter, ft-kip/ft, etc. Therefore, ***the influence width value does not affect the moments and shear in the slab*** and is used only to calculate the losses.
  - Define a ***slab area*** (length and width); a series of equally spaced cables are added to the area. Identical cables are uniformly spaced within the area. The program calculates the moments and forces for each cable according to its location in the slab area and designs each cable individually.
- Assign load cases/combinations to PRESTRESS stage in load table 1161:
  - Define the start and end time of any load case/combination by specifying the PRESTRESS stage at the start and end.
  - Specify the "Permanent" for load cases/combinations, for the calculation of deflections and time losses.
  - Specify "Factored or Service" for any load case/combination.
- For each beam/slab:

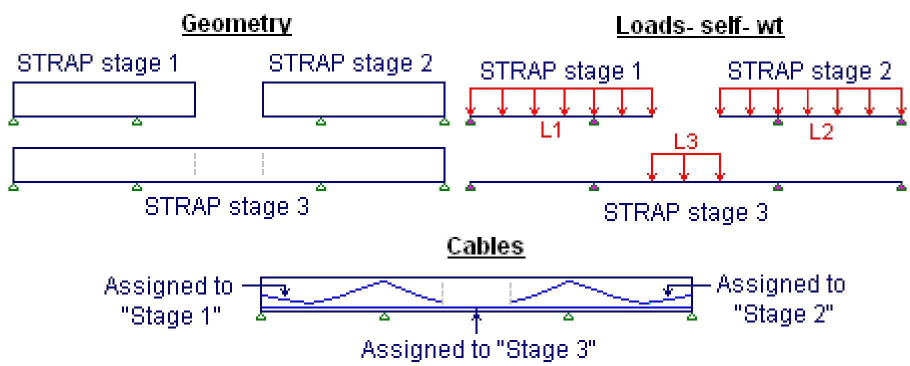
- Specify the type and number of cables and the prestressing force. The program displays a "Magnel diagram 1115" as a design aid for the selection of the prestressing. (refer to How to define cables 1102)
- Specify the trajectory of the cables. The program displays the upper and lower permissible cable boundary lines superimposed on the beam elevation.
- Assign selected cables to different stages according to the jacking sequence, defined in parameters jacking sequence 1136.

- select "Solve"; the program creates cable force load cases at each of the defined stages and time steps, including losses at each step, and solves the model for these cases. The new load cases and their results may be viewed in *STRAP's* results module. (For PRESTRESS, Do not create new combinations containing these cable loads - *STRAP* does it automatically).
- Display/print tables of stresses, deflections, shear and ultimate moments for each beam/slab.

STRAP stages:

If the prestressing is done on several distinct model stages that represent intermediate stages of construction, then the stages must be defined in *STRAP* geometry. These *'STRAP* stages' are then assigned to the relevant design stages in the *PRESTRESS* module.

Example: segmented construction



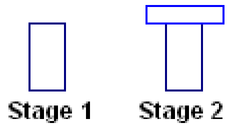
STRAP:

- Create the separate stages, then define the loads for each stage (without prestressing loads) in a separate load case and assign it to the relevant stage; solve the model. Note that ***each load is defined only once***, at the stage where it is ***first applied***. This is illustrated by the self-weight loads in the example above; the self-weight of the cantilevers are applied only in Stage 1 and 2, even though they continue to act in Stage 3.
- Define load combinations in the *STRAP* results module. These combinations should represent the loads acting at each stage. for example: C1=L1, C2=L2, C3=L1+L2+L3

PRESTRESS:

- Assign the relevant *'STRAP* stage' to each PRESTRESS stage in the **Stages - stages** option.
- Assign the combinations to the relevant stages in the **Stages - Load table** option.
- Define the cable; assign each cable to the relevant stage in the **Design - losses - jacking sequence** option, as shown above.

Example: composite beam



STRAP:

- Create two geometry stages, each with a different section. Define the loads for each stage (without prestressing loads) in a separate load case and assign it to the relevant stage; solve the model. Note that ***each load is defined only once***, at the stage where it is ***first applied***.
- Define combinations.

PRESTRESS:

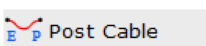
- **Stages - stages** option
  - Assign the relevant *'STRAP* stage' to each *PRESTRESS* stage, specify the time when the stage starts and specify the allowable stresses.
  - Set "**Add creep forces..**" to **Yes** for all Composite stages.
- **Defaults - Composite** tab: select ☑ to calculate differential creep and shrinkage.

Refer also to:

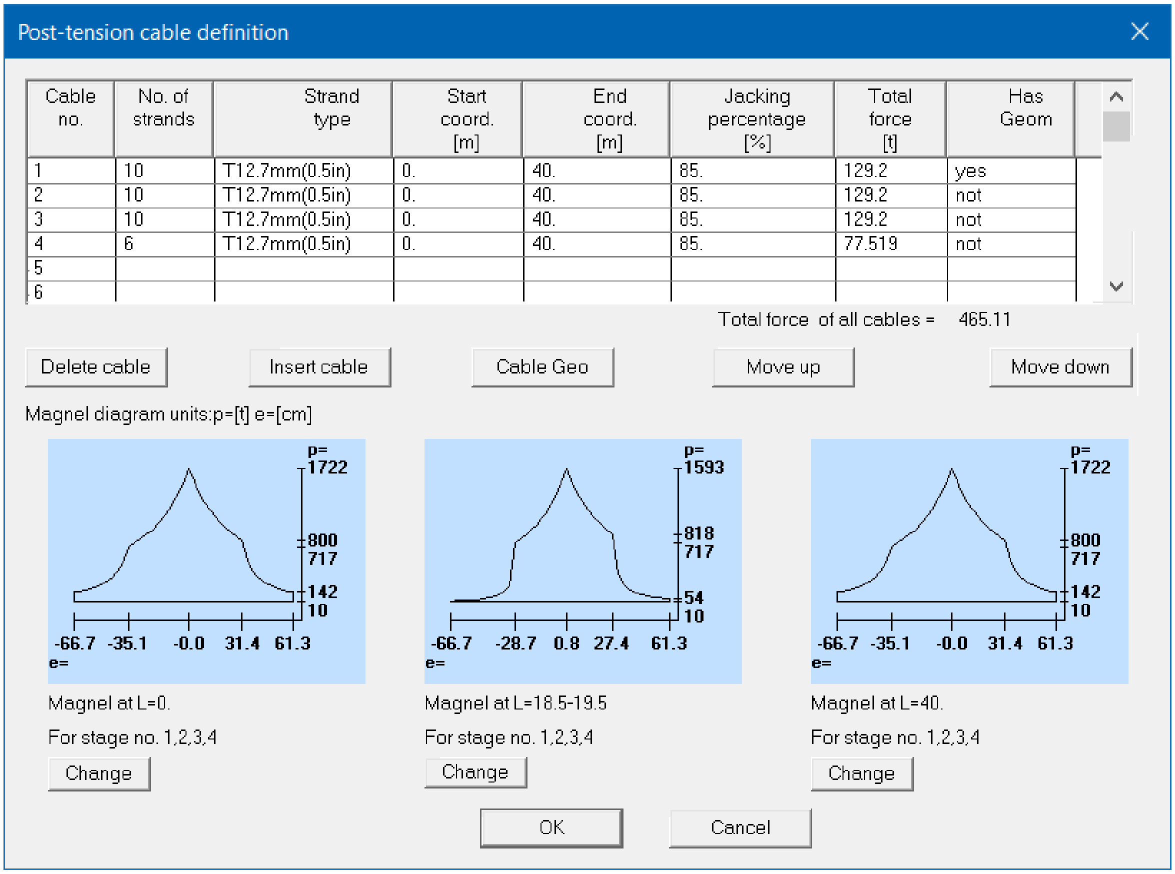
- General 1099
- Design assumptions

11.2.2 How to define Post-tensioned cables

- select 

- select the defined beam or slab for post-tension cable definition.
- select 

- Define the cable with the following table. For example:



Where:

- Cable no. - the index number of the cable.
- No. of strands - the number of strands the cable contains. (each cable may contain more than one strand).
- Strand type - assign a strand type 1154 to the cable.
- Start coord - specify the cable start coordinate along the beam or slab.
- End coord - specify the cable end coordinate along the beam or slab.
- Jacking percentage - specify the initial jacking percentage for the cable.
- Total force - the cable initial prestress force.
- Has Geom - The program displays "yes" if the cable geometry is defined. The program displays "not" if cable geometry in not defined.

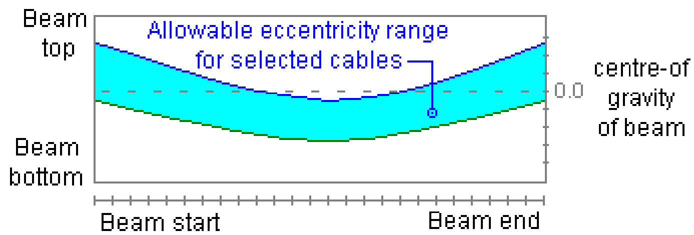
for "slab areas" the header is:

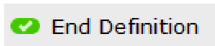
Cable type	Spacing of cables [cm]	Strand type	No. of strands	Dist. from beam start [m]	Dist. from beam end [m]	Jacking percentage [%]	Force per unit width [t]/[m]

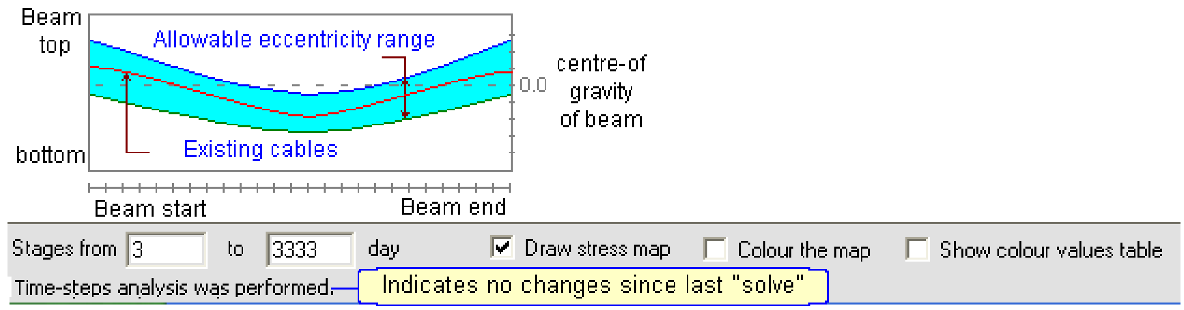
In addition to the above:

- Spacing of cables - specify the spacing between the cables for slab.

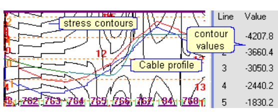
* Define cable geometry 1125. Select the cable from the table and click Cable Geo. The cables may be defined with a straight or parabolic trajectory; select one of the options and define the trajectory interactively on the screen.
* The program superimposes the minimum/maximum eccentricity range *for this cable*. For example:



- Click  to end cable geometry definition.
- Repeat the process to all defined cables.
- The program displays a beam profile that shows the allow trajectory for the cables and any cables that have already been defined:



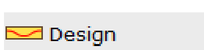
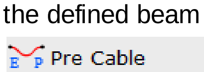
The eccentricity range is based on the allowable stresses and is calculated for the stages within the range specified at the bottom of the screen. A contour map of the stresses may also be superimposed on the beam profile. For example:

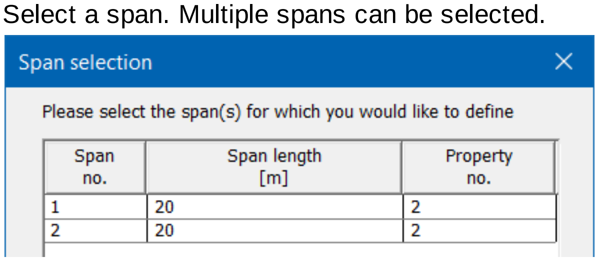


- The stresses are calculated at the top and bottom fibres along the beam and assumed linear between them.
- If there is more than one load combination, the program uses the one with highest stress/allowable ratio.

The allowable eccentricity range displayed will again be ***only for the selected cable***, i.e. it will reflect the prestressing and eccentricity of all other cables with defined geometry. Placing this cable within the range insures that the total force/eccentricity of all cables provides a satisfactory solution.

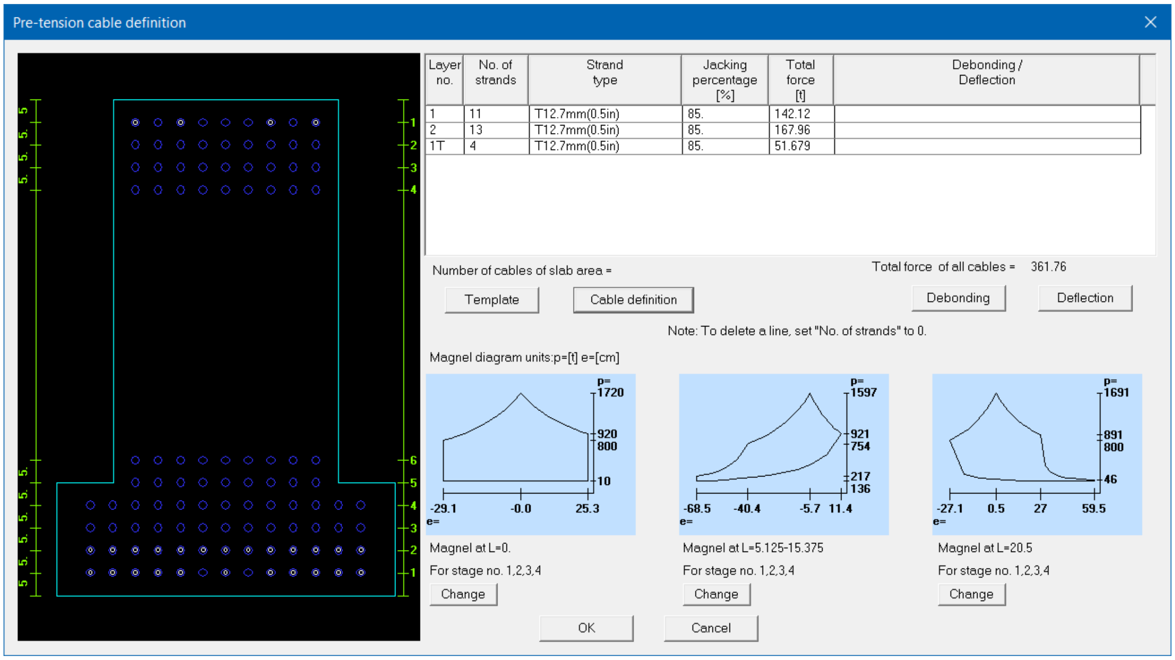
11.2.3 How to define Pre-tensioned cables

- Select  .
- Select the defined beam for pre-tension cable definition.
- Select  .
- Select a span. Multiple spans can be selected.



Span no.	Span length [m]	Property no.
1	20	2
2	20	2

- Define the cable with the following screen. For example:

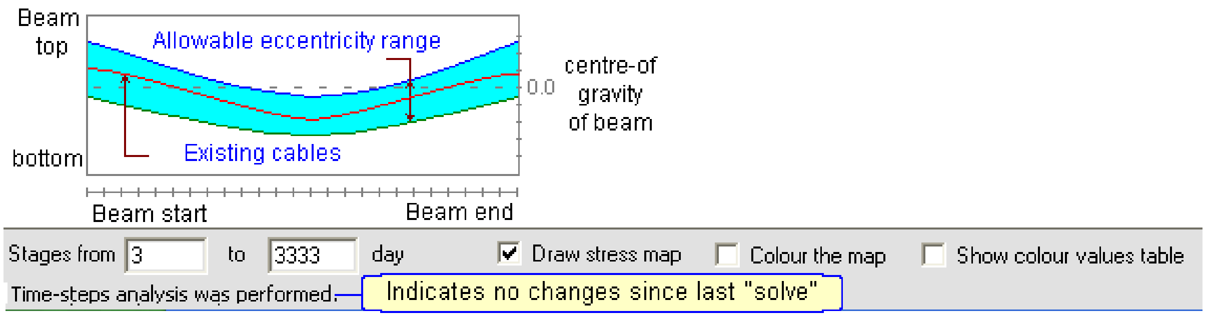


- Define a pretension cable template 1121 by clicking the Template Button.
- Click Cable definition and select a layer from the drawing at the left side of the screen.
- Specify the number of cables and the cables type.
- Specify debonding or deflection for cables within a layer by clicking Debonding or Deflection, selecting a layer from the drawing at the left side of the screen and specifying the debonding or deflection parameters.

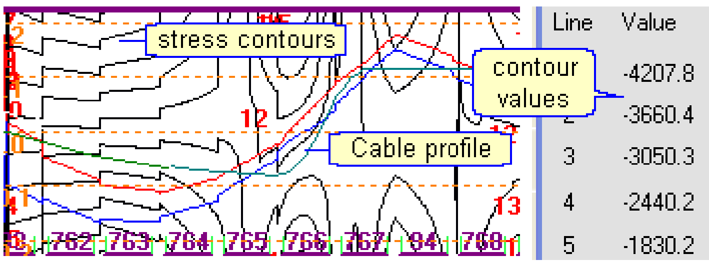
The data will than be presented in the table, where:

Layer no	=	the index number of the cables layer.
No. of strands	=	the number of strands in a layer. (Can be edited within the table).
Strand type	=	assign a strand type 1154 to the cable. (Can be edited within the table).
Jacking percentage	=	specify the initial jacking percentage for the cable. (Can be edited within the table).
Total force	=	the cable initial prestress force.
Debonding/Deflection	=	displays data about debonding and/or deflection.

During cable definition, the program displays a beam profile that shows the allow trajectory for the cables and any cables that have already been defined:



The eccentricity range is based on the allowable stresses and is calculated for the stages within the range specified at the bottom of the screen. A contour map of the stresses may also be superimposed on the beam profile. For example:



- The stresses are calculated at the top and bottom fibres along the beam and assumed linear between them.
- If there is more than one load combination, the program uses the one with highest stress/allowable ratio.

11.2.4 Configuration changes / Composite

Description

There are additional forces which are a result of loads and cables that start to act on the original configuration but their creep is restricted after the configuration changes.
Restraint of the creep generates new moments and forces that develop with time in the new configuration according to the creep coefficient φ.
The configuration change also relates specifically to beams when the section changes from non-composite to composite.

Two examples:

- A cable is jacked in a beam before the topping is poured. The initial stresses in the topping are zero. The beam wants to deform with creep as if the topping were not present, but it is restrained by the topping.
  The creep stresses are now divided between the beam and the topping. This restraint of the beam creep by the topping creates new moments acting on the beam.
- A load is applied to a cantilever beam. A support is then added to the free end, thereby changing the configuration to a simply-supported beam. The beam wants to deform with creep as if the support were not present, but it is restrained by it.

The change in configuration generates new moments that develop with time as the creep from the load progresses. The developing creep would not generate these moments if the configuration were not changed.

Calculation method

The new moment acting on the new configuration at time 't' including the influence of differential creep is:

$$\mathbf{M = M_1\, e^{-\varphi} + M_2\, (1 - e^{-\varphi})}$$

where:

M_1 = moment before the configuration change

M_2 = moment at the same point from the same loads after the configuration change

φ = creep coefficient from the configuration change to time 't'

The additional moment = **$M - M_1$**.

New forces and moments are calculated separately for each load using the creep coefficient φ corresponding to the concrete age at the time of load application.
Similarly, new forces and moments are calculated separately for each cable jacked prior to the configuration change using the creep coefficient φ corresponding to the concrete age at the time of jacking.
Similarly, the new axial force acting on the new configuration due to differential creep is:

$$P = P_1 e^{-\varphi} + P_2 (1 - e^{-\varphi})$$

Note:

- *the program automatically solves all loads applied prior to the configuration change again with the new configuration. This increases the solution time.*

There is another additional moment/force added to the composite section resulting from the difference in the beam/topping concrete ages and the resulting differential creep and shrinkage.
This differential creep and shrinkage are calculated as follows:

Shrinkage:

- Losses are computed for the section without consideration for differential shrinkage
- Shrinkage of the topping is computed assuming that it is not connected to the top of the beam, using the appropriate concrete age, area/perimeter ratio, etc.
- The shrinkage of the beam is computed assuming it is not connected to the bottom of the topping, using the appropriate concrete age, area/perimeter ratio, etc.
- The difference between the beam and topping shrinkages are converted to stresses relative to the center-of-gravity of the composite section.
- These stresses are then converted to axial forces and moments applied to the composite section.
- These axial forces and moments are then added (as equivalent joint loads) to the cable loads that are "Solved" by the program.

Creep:

- The differential creep results from the different beam/topping creep coefficients. The stress difference is calculated separately from the equation **$M = M_1 e^{-\varphi} + M_2 (1 - e^{-\varphi})$**, as explained above.
- the axial force and moment resulting from the differential creep is calculated similarly to differential shrinkage.

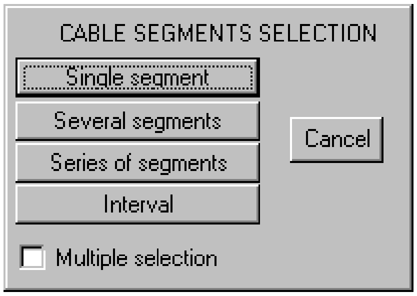
The forces calculated from differential creep and shrinkage are modified at every time step by **$(1 - e^{-\varphi})/\varphi$**, where φ **=** the creep coefficient at that time step, in order to account for the change in stresses during the step.

Calculation of stresses:

- Stresses are calculated in the beam just before the topping is cast using the properties of the non-composite section.
- Stresses are then calculated from the difference between loads acting at time 't' and the loads acting before the topping is cast, applied to the composite section.

- Stresses at time 't' are calculated from loads, solved at time 't', which take into account additional forces due to the configuration change.
- Stresses in the topping are calculated only for loads applied after casting.
- Stresses due to differential shrinkage and creep, calculated separately (as explained above) are then subtracted from these stresses.

11.2.5 Segment selection option



Select:

Single segment

Move the to any segment so that it is highlighted with the ■ and click the mouse

Several segments

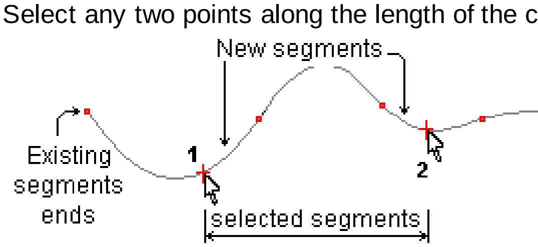
Move the to any segment so that it is highlighted with the ■ and click the mouse; repeat for other segments and click the last segment twice to end the selection

Series of segments

Select the first and last segments in a series as explained above; all intermediate segments will also be selected.

Interval

Select any two points along the length of the cable:



The program creates new segments starting and ending at the selected points and selects them as well as all intermediate segments.

Multiple selection

This option enables you to select multiple segments using two or more of the above methods; the program displays the above menu again after you complete the initial section - select a different option and add more segments to the selection.

11.3 Define

Beams

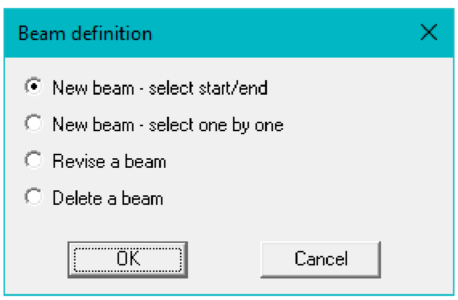
- Define beams to be prestressed by selecting one or more continuous *STRAP* beam members.
- Add/delete supports at node locations and modify the top/beam beam levels in individual spans.

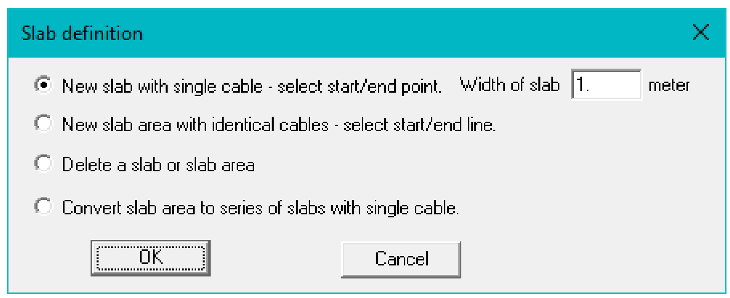
Slabs

There are two options for defining slabs fpr post-tension:

- Define a ***slab with a single cable*** (center line and "influence width"):
  The program calculates the result diagrams along the defined line similarly to the "Results along a line" option in the *STRAP* results module, i.e. the results are ***per unit width***, e.g. ton-meter/meter, ft-kip/ft, etc. Therefore, ***the influence width value does not affect the moments and shear in the slab*** and is used only to calculate the losses.
- Define a ***slab area*** (length and width); a series of equally spaced cables are added to the area. Identical cables are uniformly spaced within the area. The program calculates the moments and forces for each cable according to its location in the slab area and designs each cable individually.

Select one of the following options:





Note:

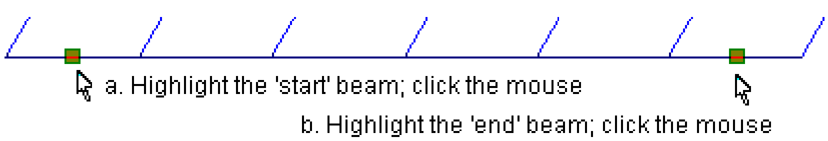
- *Specify the Parameters 1145 and Stages 1160 prior to defining the beams.*

11.3.1 New beam

Define a prestressed beam consisting of a chain of *STRAP* beam members. Select one of the following options:

New beam - select start/end

Select the 'start' and 'end' beams in a continuous line of *STRAP* members. For example:



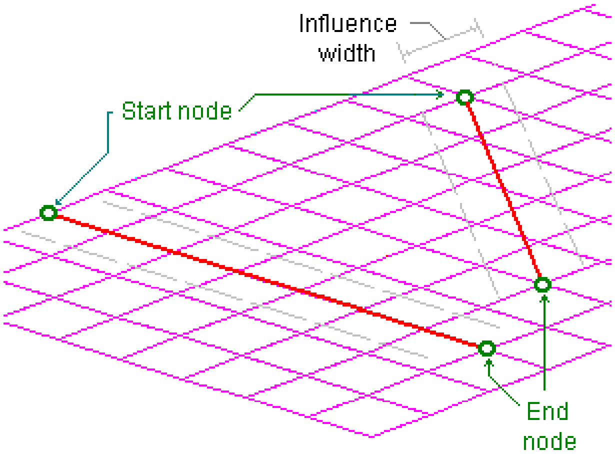
New beam - select one by one

Similar to the option above, except that all of the beams in the chain must be selected (in the correct order).

11.3.2 New slab

Slab - single cable

Define a slab strip by selecting start and end nodes on a plane of elements in the model. The resulting line defines the slab strip centre line. for example:



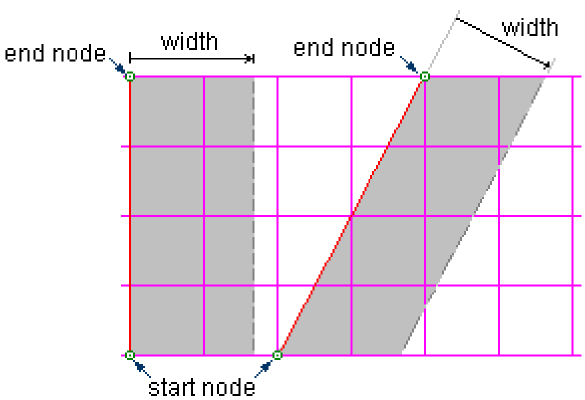
Note:

- *the center line does not have to coincide with a line of element edges.*
- *the slab 'influence width' is defined in* [Default](1145) *parameters or* [Design - cable geometry](1125) *(for specific slabs).*

The program calculates the result diagrams along the defined line similarly to the "Results along a line" option in the *STRAP* results module, i.e. the results are ***per unit width***, e.g. ton-meter/meter, ft-kip/ft, etc. Therefore, ***the influence width value does not affect the displayed results***. The program treats the influence width as the beam width and uses it to calculate the stresses, etc, resulting from the prestressing.

Slab area

Select a start node and an end node to define length of the area, then define the width:



Note: the width is always on one side of the line defined by the nodes.

Identical cables are uniformly spaced within the area. The program calculates the moments and forces for each cable according to its location in the slab area and designs each cable individually.

Convert slab area

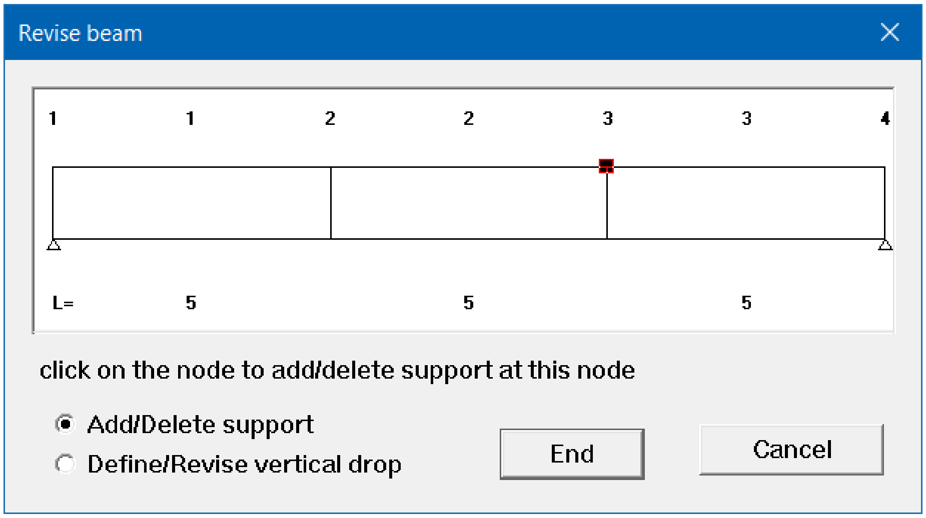
Convert a defined "Slab area" to a series of "Slab - single cable":

- the width assigned to each cable is the cable spacing defined for the "slab area". If no cables and spacing were defined, the program divides the area into equal strips with an default width.

11.3.3 Revise

Add/remove supports at the node locations or change the vertical location of individual spans.

- Select one of the beams by highlighting one of the component members; click the mouse.
- The program displays the beam elevation:



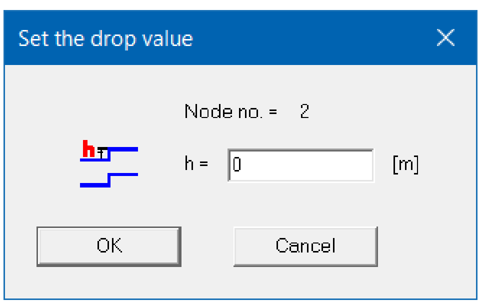
- Select one of the following options:

Add/Delete support

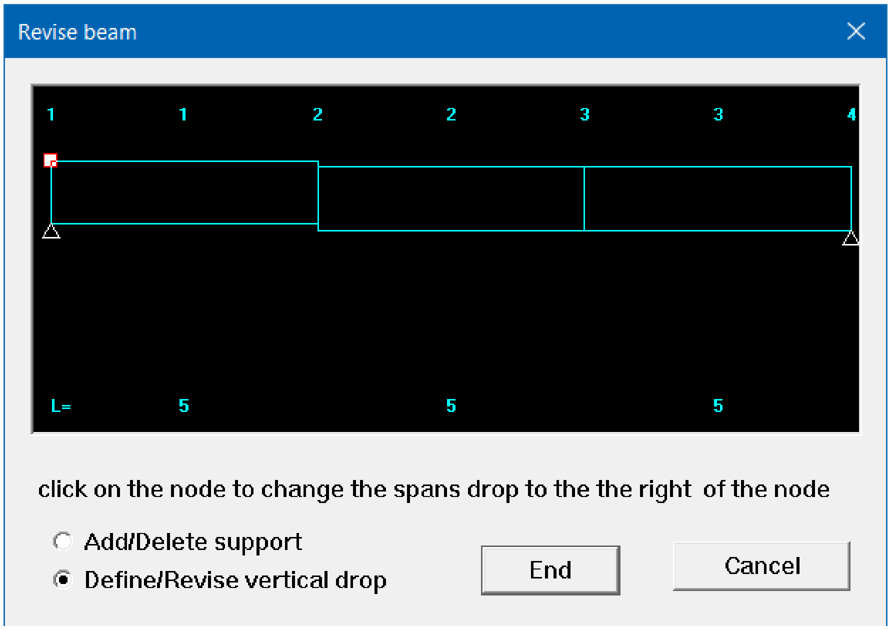
Move the cursor until the relevant node is highlighted with the ■; click the mouse. The support is deleted/added to the node.

Define/Revise vertical drop

Move the cursor until the node at the left of the drop is highlighted with the ■(all beams to the right of the node will be moved vertically); click the mouse:



Select the drop type and enter the drop dimension (in the same units as the span length)
The program displays the revised beam. For example:



11.3.4 Delete

Beams

Select existing *PRESTRESS* beams using the standard beam selection method. Note that it is sufficient to select only one of the *STRAP* members that comprise the beam.

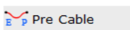
Slabs

Highlight and click on the slab center-lines using the standard beam selection method.

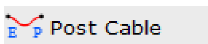
11.4 Design

- Define pre tension/post tension cables and define their trajectory in the beam.
- Specify parameters for prestress losses, creep/shrinkage of concrete and jacking sequence for the current beam.
- Specify reinforcement steel for the current beam.
- Display/print all output tables (stresses, ultimate moment, shear, deflections, etc.).

Note: All the parameters defined under "Parameters" here supersede the default parameters in the main menu. for more info refer to default parameters 1145.



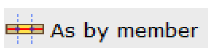
Beams only: specify the number, type and geometry of pre-tension cables in the beam



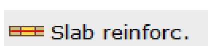
Specify the number, type and geometry of post-tension cables in the beam/slab

Define loss parameters for individual cables. Loss parameters defined here supersede the default parameters.

Remove/restore cables from the display (the cable are ***not*** deleted)



Beams only: specify the casting day for each span and the regular longitudinal reinforcement for the current beam. Area/cover defined here supersede the default values.



Slabs only: specify the regular reinforcement and cover for the current slab. Area/cover defined here supersede the default values.

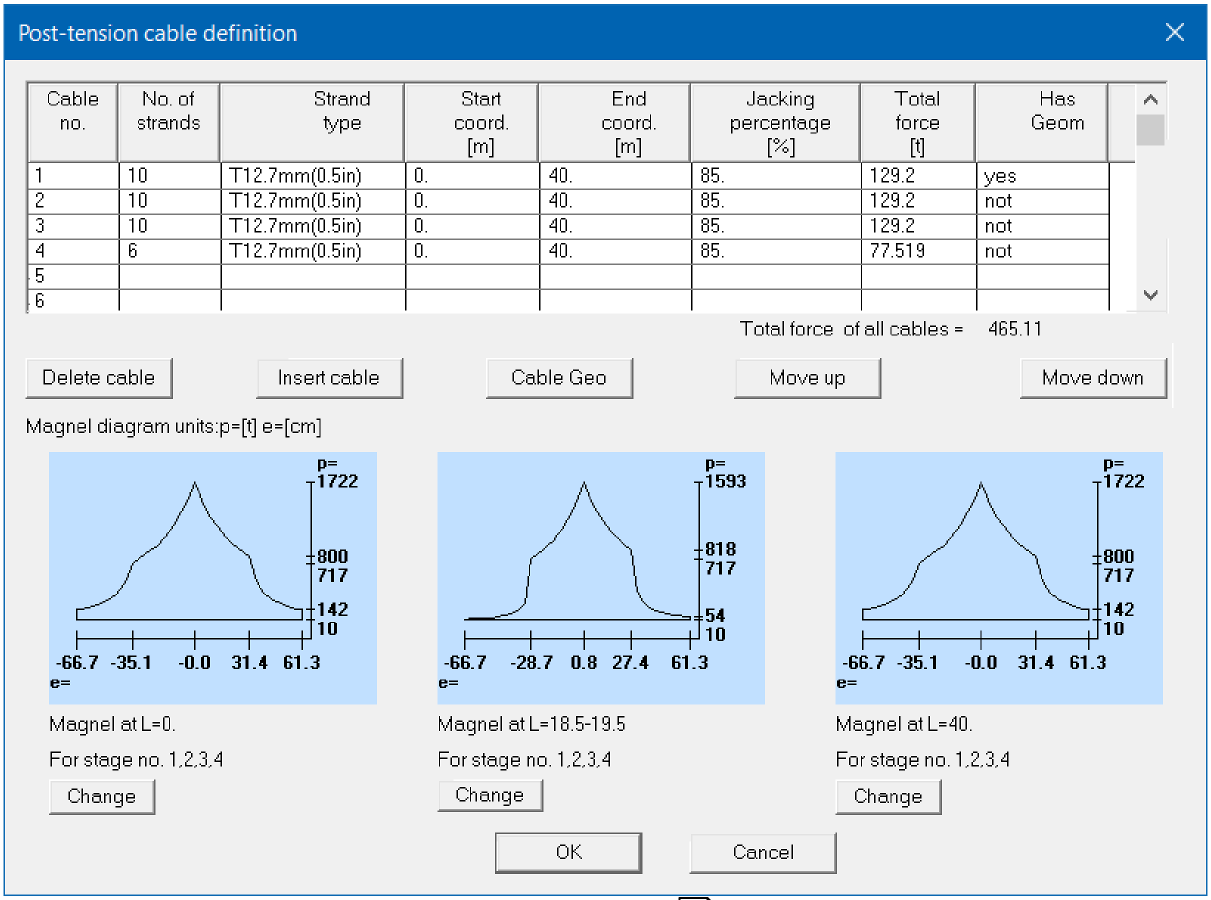
Select the design direction (M2 or M3) and/or invert the section.

Add/remove supports at different geometry stages for the current beam. Revisions specified here override those defined in the main menu.

Display all result tables on the screen.

Print the result tables.

11.4.1 Define post-tension cable



For more info, refer to How to define post-tensioned cables 1102

Cables

For each cable, specify:

Cable no. - the index number of the cable.

No. of strands - the number of strands the cable contains. (each cable may contain more than one strand).

Strand type - assign a strand type 1154 to the cable, select a type from the listbox displayed by clicking on the .

Start coord - the cable start coordinate along the beam or slab. (this may be revised when defining the cable geometry)

End coord - the cable end coordinate along the beam or slab. (this may be revised when defining the cable geometry)

Jacking percentage - the initial jacking percentage for the cable, i.e. jacking force calculated as a percentage of maximum allowable stress (defined in Defaults - strand type 1154)

Total force - the cable initial prestress force. The program calculates the jacking force for the cable and displays below the table the "total force for all cables".

Has Geom - The program displays "yes" if the cable geometry is defined. The program displays "not" if cable geometry in not defined.

for "slab areas" the header is:

Cable type	Spacing of cables [cm]	Strand type	No. of strands	Dist. from beam start [m]	Dist. from beam end [m]	Jacking percentage [%]	Force per unit width [t]/[m]

In addition to the above:
Spacing of cables - specify the spacing between the cables for slab.

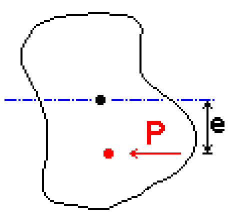
Note:

- the cable table may be rearranged by highlighting a cable and clicking Move up or Move down.
- to delete a cable, highlight a line and click Delete cable
- to add a new cable, highlight a line and click Insert cable

Cable Geo

To define post-tension cable geometry, refer to cable geometry - post-tension 1125

Magnel diagrams



The prestress force **P** and the eccentricity of the force **e** must be selected by the user; any number of combinations of **P** and **e** provide an acceptable solution.

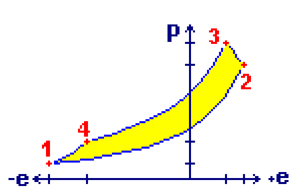
The stresses at the extreme fibres are limited to the Code values and are checked at every stage for maximum and minimum moments with the actual prestressing force (after losses) . This gives four limiting stress conditions:

- minimum moment - top fibre
- minimum moment - bottom fibre
- maximum moment - top fibre
- maximum moment - bottom fibre

The equations are in the form:

$$\sigma_e - \frac{P}{A} \pm \frac{P\,e\,y}{I} \leq \sigma_{all}$$

where σ_e is the relevant stress from the external loads.



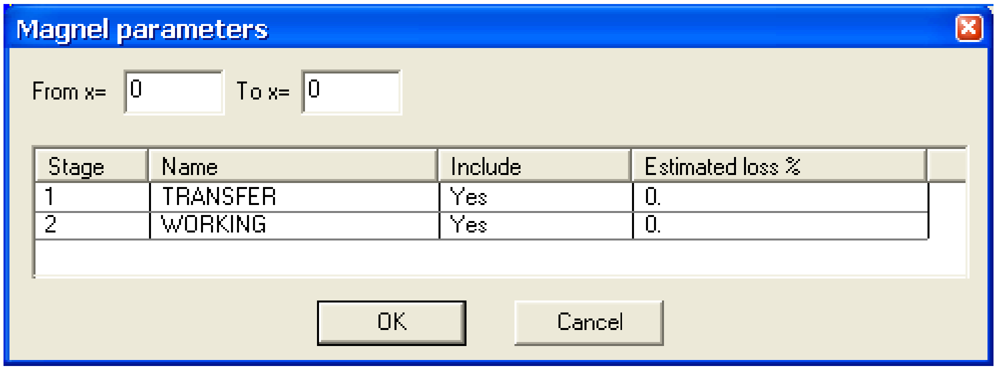
Plotting all of the possible solutions for each of the four conditions gives four intersecting lines: 1-2, 2-3, 3-4, 4-1 in the adjacent diagram; any combination of **P** and **e** within the enclosed area will result in acceptable stresses in the specific cross-section.

Refer also to Magnel diagrams - change 1115.

Magnel diagrams - Change

The Magnel diagrams 1115 are an envelope of the allowable cable force and locations calculated for various stages and selected locations. The program initially displays three Magnel diagrams - at the start, middle, and end of the beam - that include the stresses at all loading stages. Note that the "middle" diagram is a composite of the diagrams at all sections in the middle half of the beam.

Select the stages and the locations:



- ***From / to***
  Select the 'from' and 'to' coordinates from the beam start. Note that the Magnel diagrams are calculated at 1/20 intervals along the span.

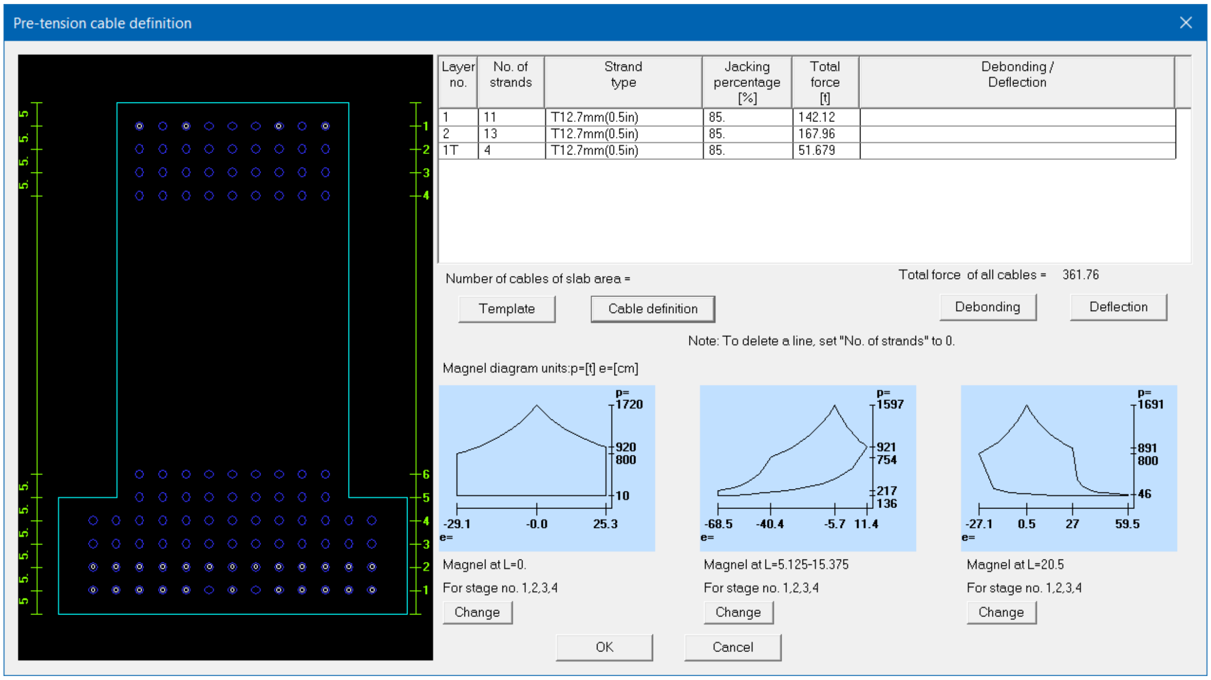
- ***Stage***
  - **Include*:***
    click on the cell - the program displays 

; click on the checkbox to include/exclude the stage from the diagram.
  - **Estimated loss**: Enter the estimated total loss for each stage(%).

The beam elevation is redrawn with the minimum and maximum allowable eccentricity at each section after the cables are selected.

11.4.2 Define pre-tension cable

For more information, refer to <u>How to define a pre-tensioned cable</u> 1104.

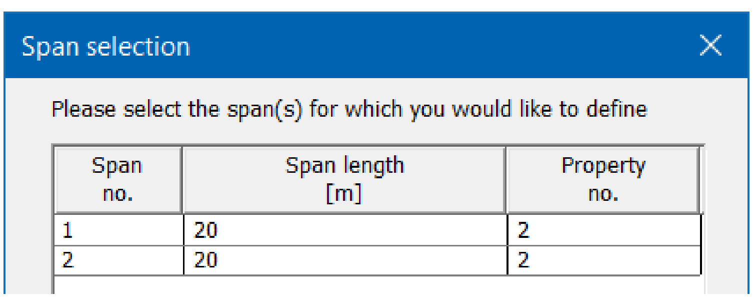


Defined pre-tensioned cables are displayed in the table, where:

Layer no.	-	the index number of the cables layer.
No. of strands	-	number of strands in a layer. (Can be edited within the table).
Strand type	-	assign a <u>strand type</u> 1154 to the cable. (Can be edited within the table).
Jacking percentage	-	specify the initial jacking percentage for the cable. (Can be edited within the table).
Total force	-	the cable initial prestress force.
Debonding/Deflection	-	displays data about debonding and/or deflection.

Span selection

When defining pre-tensioned cables for a multi-span beams, the user must select spans -



Note:

- for multiple spans, only spans with the same property can be selected.

Template

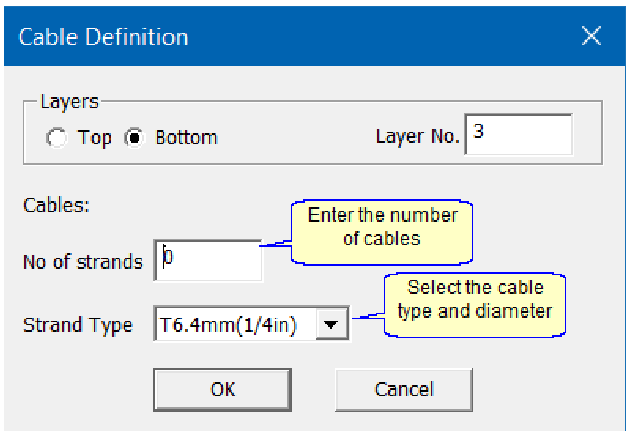
A template is a preset pattern for possible cable locations.

- every beam must have a template. The program will display a default template for each section.
- Cables can be defined only at template locations.
- The cable locations within a template are defined in layers. Layers may be revised, added or deleted at any time. Refer to pre-tension template 1121.

Cable definition

To start cable definition select Cable definition.

- Select a cable location from the template drawing; highlight a circle in the layer and click the mouse.
- define the cable details:

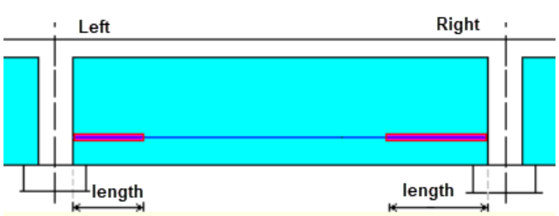


Note**:**

- the number of cables can be less than the number of template locations in the layer (not all locations must be used).
- The number of strands may be larger than the available locations. In this case, multiple cables will be placed at template locations. For example: a selected layer has 13 possible locations, The user entered "No of strands" = 15. Two cables will be placed at 2 locations of the 13 for a total of 15 cables. The program will display "2" in the circle at these locations.

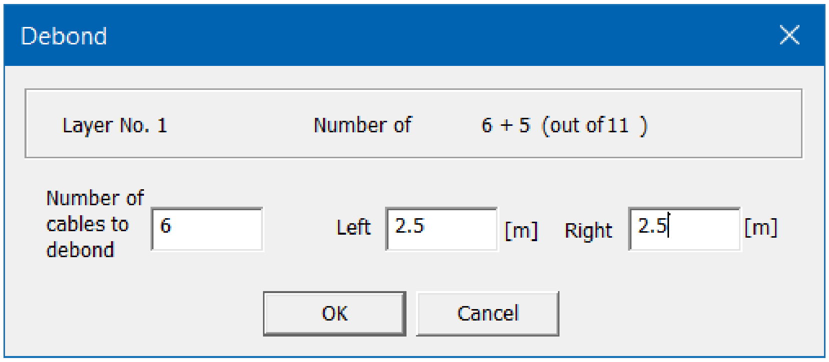
Debonding

Cables may be debonded at one or both ends. Each cable in a layer may be debonded for a different length.



To define debonding:

- select a layer from the ***table*** and click Debonding.
- define the debonding details:



where:

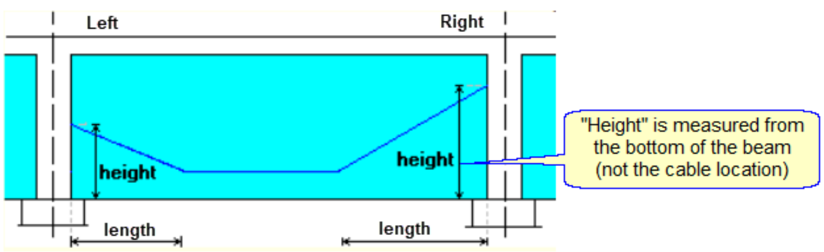
- Number of cables to debond - Specify the number of cables for debonding for the selected layer.
- Left/Right - Specify the length of debonding from the left/right end of the beam.

Note:

- the debonded cables are entered as a separate layer in the table. The remaining cables in the layer can be debonded to different lengths.

Deflection

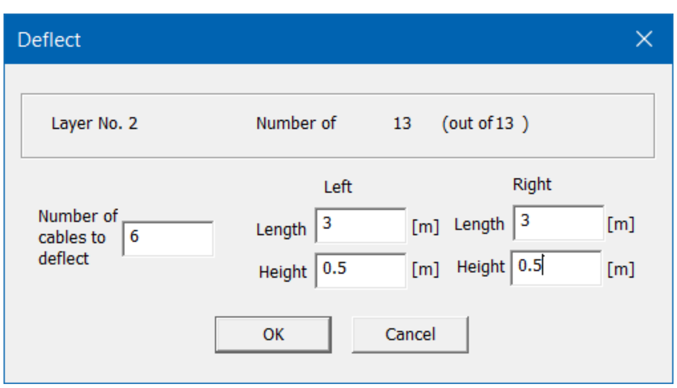
Cables may be deflected at one or both ends. Each cable in a layer may be deflected to a different height.



To define deflection:

- select a layer from the ***table*** and click  .
- define the deflection details:

For cable deflection definition select a layer from the table and click .



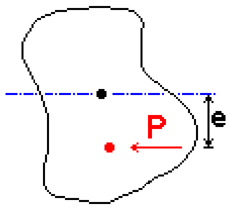
Where:

Number of cables to deflect - Specify the number of cables for deflection for the selected layer.

Length - Specify the length of deflection (horizontal dimension) from the left/right end of the beam.

Height - Specify the height of deflection (vertical dimension from the ***bottom of the beam***) at the left/right end of the beam.

Magnel diagrams



The prestress force **P** and the eccentricity of the force **e** must be selected by the user; any number of combinations of **P** and **e** provide an acceptable solution.

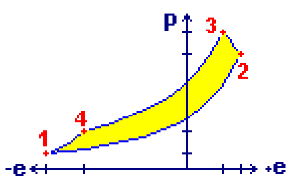
The stresses at the extreme fibres are limited to the Code values and are checked at every stage for maximum and minimum moments with the actual prestressing force (after losses) . This gives four limiting stress conditions:

- minimum moment - top fibre
- minimum moment - bottom fibre
- maximum moment - top fibre
- maximum moment - bottom fibre

The equations are in the form:

$$\sigma_e - \frac{P}{A} \pm \frac{P\,e\,y}{I} \leq \sigma_{all}$$

where σ_e is the relevant stress from the external loads.



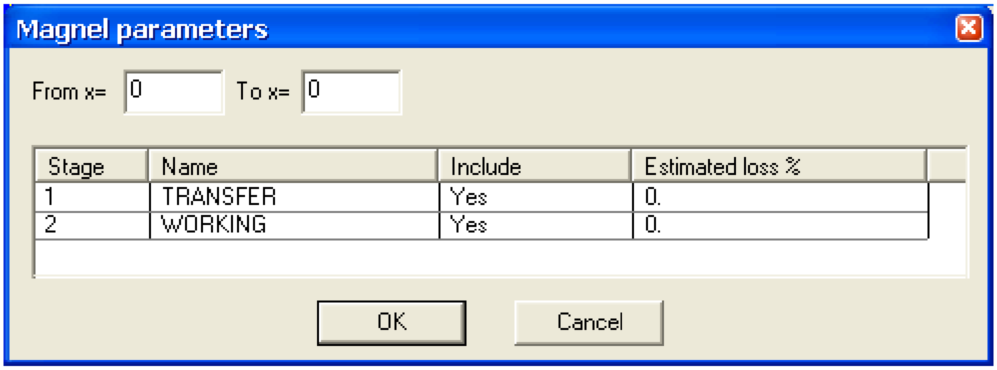
Plotting all of the possible solutions for each of the four conditions gives four intersecting lines: 1-2, 2-3, 3-4, 4-1 in the adjacent diagram; any combination of **P** and **e** within the enclosed area will result in acceptable stresses in the specific cross-section.

Refer also to Magnel diagrams - change 1115.

Magnel diagrams - Change

The Magnel diagrams 1115 are an envelope of the allowable cable force and locations calculated for various stages and selected locations. The program initially displays three Magnel diagrams - at the start, middle, and end of the beam - that include the stresses at all loading stages. Note that the "middle" diagram is a composite of the diagrams at all sections in the middle half of the beam.

Select the stages and the locations:



- ***From / to***

  Select the 'from' and 'to' coordinates from the beam start. Note that the Magnel diagrams are calculated at 1/20 intervals along the span.

- ***Stage***
  - **Include*:***

    click on the cell - the program displays ☑ Include; click on the checkbox to include/exclude the stage from the diagram.

  - **Estimated loss**: Enter the estimated total loss for each stage(%).

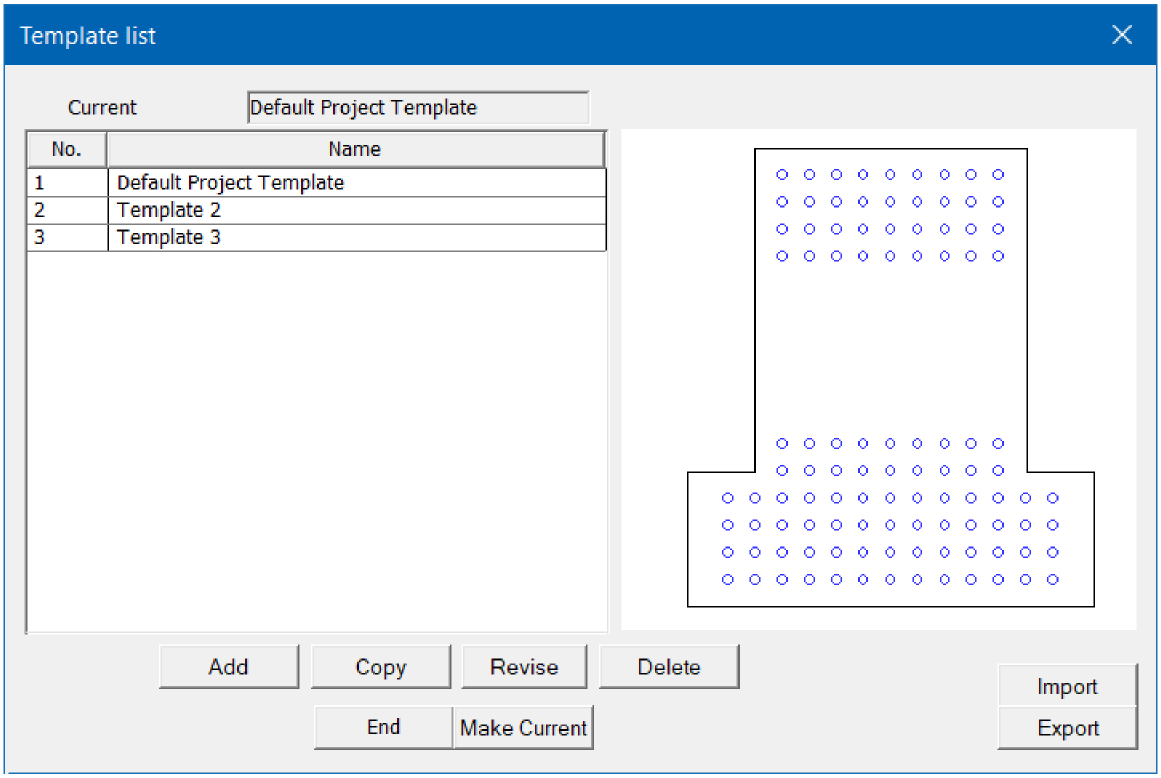
The beam elevation is redrawn with the minimum and maximum allowable eccentricity at each section after the cables are defined, debonding is specified, parameters are changed, etc..

11.4.2.1 Pre-tension Template

A template is a preset pattern for possible cable locations.

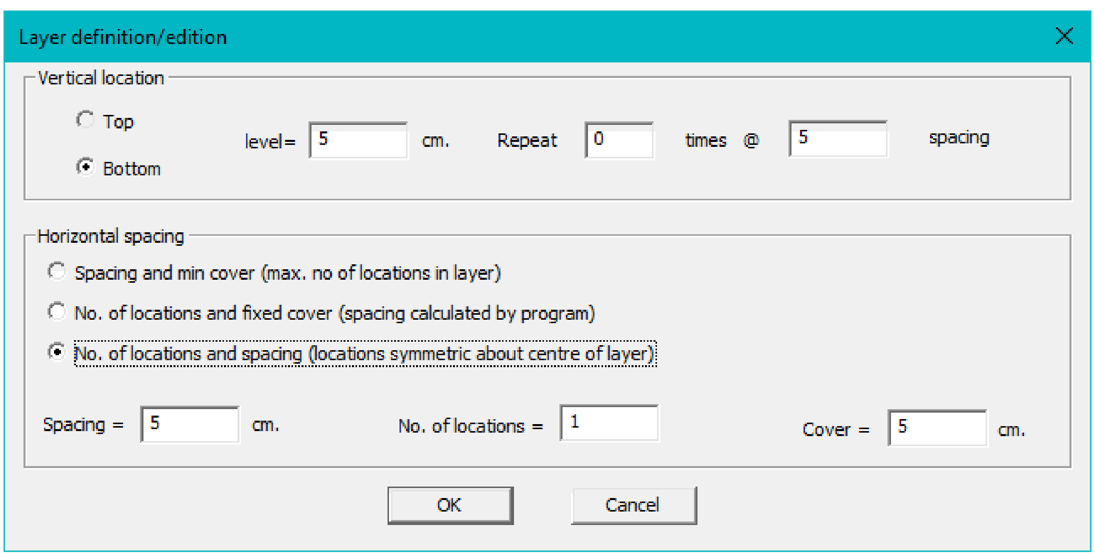
- Each beam must have a template. The program has a default template for every section type.
- Cables can be defined only at template locations.
- The cable locations within a template are defined in layers. Layers may be revised, added or deleted at any time.

The template list menu can be accessed by clicking Templete in the Pre-tension cable definition menu. For more information, refer to define pre-tension cable1117.



Add

Click Add to create a new template. Define the layers in the template as follows:



Vertical location:

Top/Bottom - The location of the layer can be measured from either to top or bottom face of the beam.

Level - Specify the vertical level of the first layer. (Vertical cover)

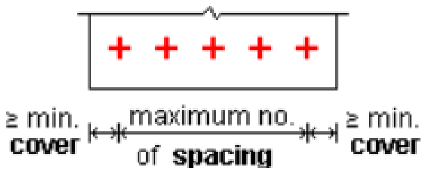
Repeat - The number of repetitions of the defined layer. 0 will result in the definition of 1 layer

Spacing - Specify the vertical spacing between multiple cable layers.

Horizontal spacing:

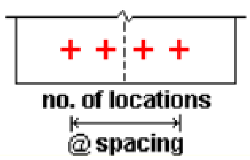
There are 3 options to define the horizontal spacing of cables in a layer:

- **Spacing and min cover.** (max.no of locations in a layer).
  The programs determines the maximum number of locations at equal spacing so that the horizontal cover at both sides is not less than the minimum. Minimum cover and spacing are specified by the user.



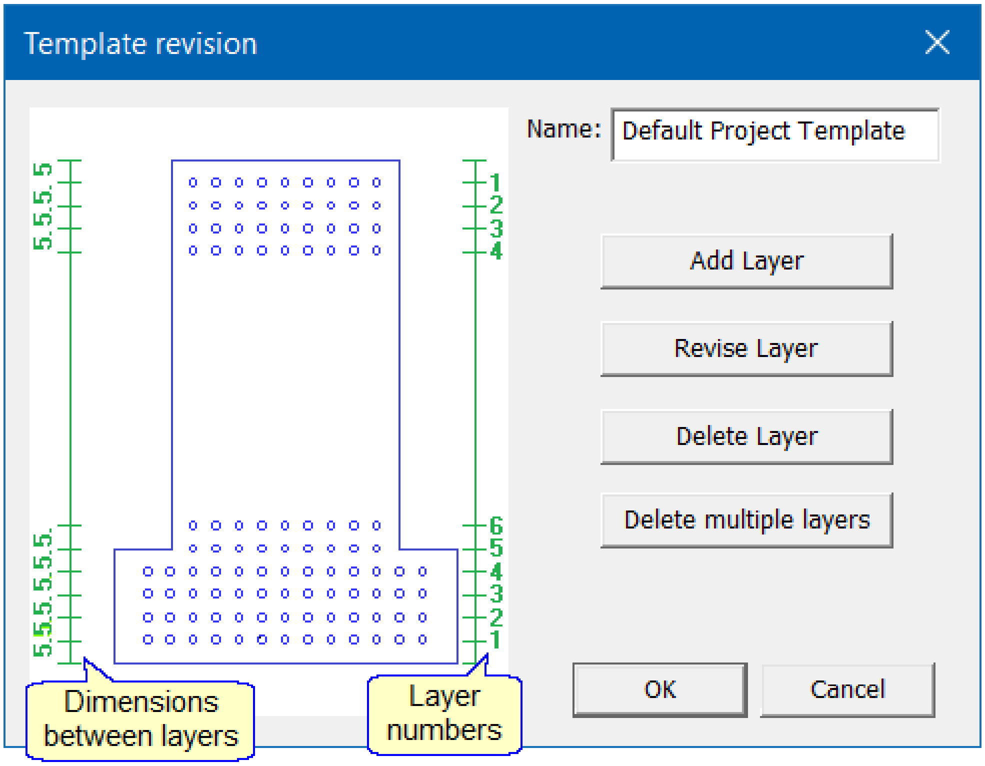
- **No. of locations and fixed cover**. (spacing calculated by the program).
  The two exterior locations are placed at a defined cover, the remaining optional locations are placed at equal spacing. Fixed cover and number of locations are specified by the user.

- **No. of locations and spacing**. (locations symmetric about the center of layer).
  The possible cable locations are arranged symmetrically about the center of a layer. Spacing and number of locations are defined by the user.



Revise

To revise a defined template click and highlight the template in the table, then click Revise.



The left of the screen will displays the beam section.

Name - Revise the template name.

Add layer - Add additional layer/layers. Refer to add template above.

Revise layer - Revise the horizontal spacing in an existing layer. Refer to add template above.

Delete layer - the program displays the beam section; highlight and click on a layer to delete..

Delete multiple layers - the program displays a table showing the layers; click and highlight two or more lines and click OK

Make current

At the top of the template list window the current template name is displayed.

To change the current template, select the template from the menu and click Make current.

Copy

To copy a template, select a template from the list and click Copy.

Delete

To delete a template, select a template from the list and click Delete.

Import/Export

Export:

Create a file **xxx.DAT** containing the details of ***all*** the templates in the list.

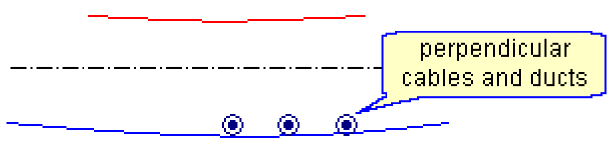
Import:
Select a file **xxx.DAT** that contains templates previously exported. If the program finds that a template in the file has the same name as a template in the current list, it asks whether to overwrite the current template or to ignore the template in the file.

11.4.3 Cable geometry - post-tension

Define the cable geometry; each cable may consist of a series of straight or parabolic segments:

- select the cables: highlight the relevant row in the table and click the mouse.
- for "slabs with single cable", specify the slab influence width1146, if different from the default value.

Note that the program displays perpendicular cables and ducts (from both beams and slabs) to avoid conflicts:



- define/edit the cable segments:

Prb. - 3 pts	Define a parabolic segment by specifying its start and end points and an additional point along its path.
Prb.- ang....	Define a parabolic segment by specifying its start and end points and the angle to the horizontal at the start point.
Prb.- seg....	Define a parabolic segment connecting the end of an existing segment and and a new point. The new segment will be tangent to the selected segment.
Prb - join	Define a parabolic segment connecting two existing segments. The new segment will be tangent to the selected segments.
Prbs - points	Define a series of points; the program will connect them with a series of parabolas.
Line - ends	Define a straight segment by specifying its start and end points
Line - join	Define a straight segment connecting two existing segments
Line- seg+pt	Define a straight segment connecting the end of an existing segment and and a new point.
By list	For slabs only: the program automatically defines a series of straight segments at the maximum positive and negative moments. Modify the segments if necessary; the program connects them with parabolas.
From DXF ...	Import a cable trajectory from a DXF file.
End Defini...	End the definition of the current cable.

Return to the main menu ***without saving*** all new segments and revisions to existing segments.

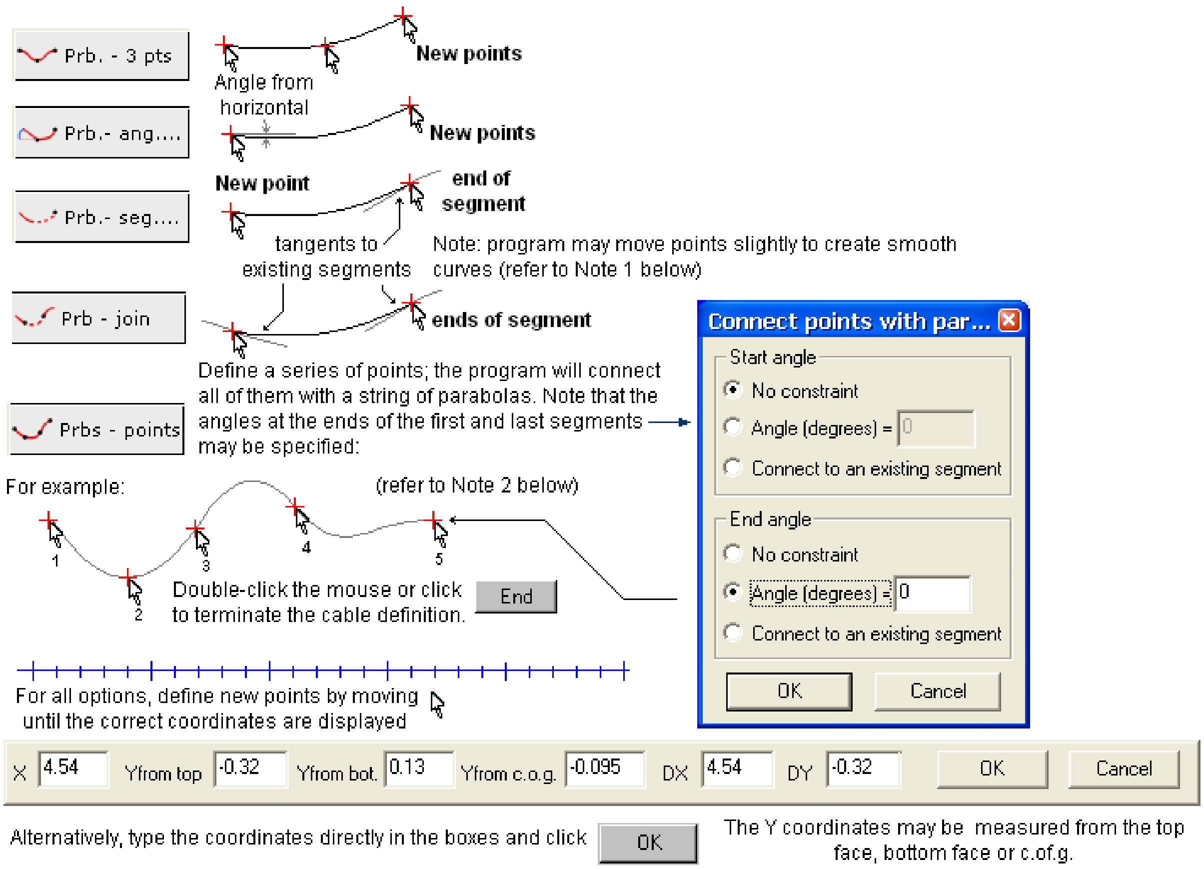
Edit existing segments; move end points or copy/delete/mirror segments

Note:

- *All zoom/pan options are available when using these options (horizontal only)*
- *refer to How to define cables* 1102 *for general information.*

11.4.3.1 Parabola

Select one of the following options:



Notes:

- The program tries to draw a parabola that connects smoothly with the existing segments. If it is not possible, it creates two different parabolas in the interval connected at the mid-point.
- The segments defined by the "points" option are created as follows:

Start segment:

- No constraint
  The segment is defined by points 1,2,3 (similar to "**3 pt**.")
- Angle =
  The segment is defined by points 1,2 and the defined angle, (similar to "α**+2 pt**.")
- Connect to an existing segment
  The first segment is defined using to "**seg.+ pt**" method.

Intermediate segment:

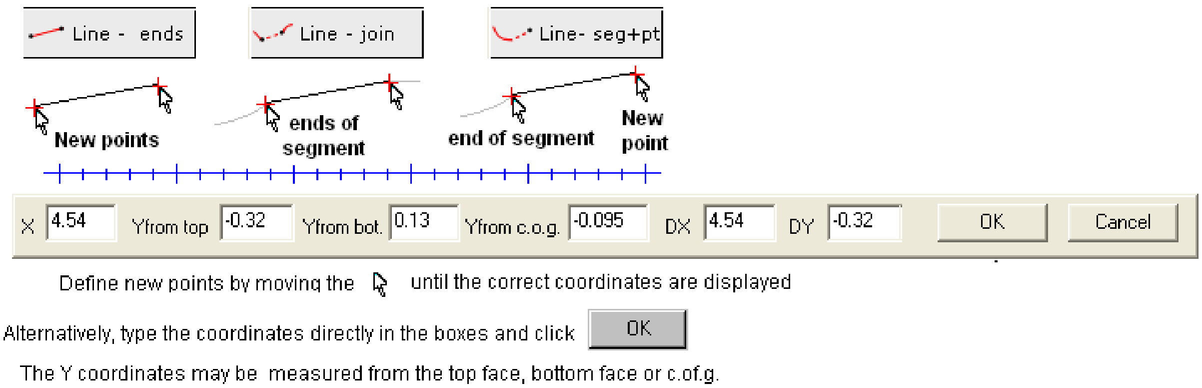
- Always created using the "**seg.+pt.**" method.

End segment:

- No constraint
  The segment is created as displayed on the screen.
- Angle =
  The displayed segment is replaced by 2 new segments, where the second one ends at the specified angle. The option is ignored if only a single segment is defined.
- Connect to an existing segment
  The option is ignored if only a single segment is defined.

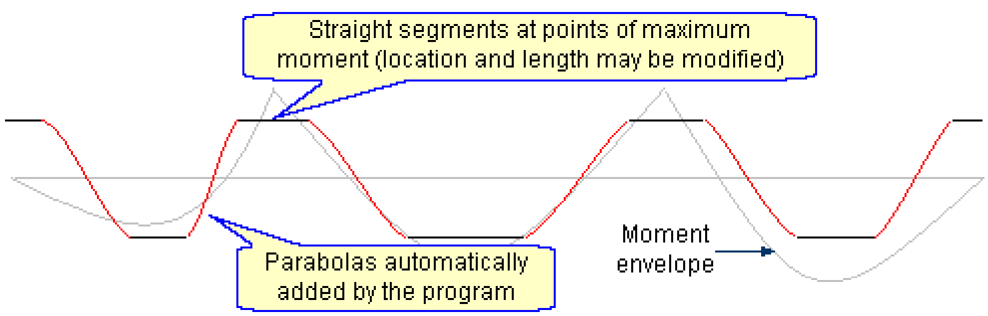
11.4.3.2 Straight

Select one of the following options:



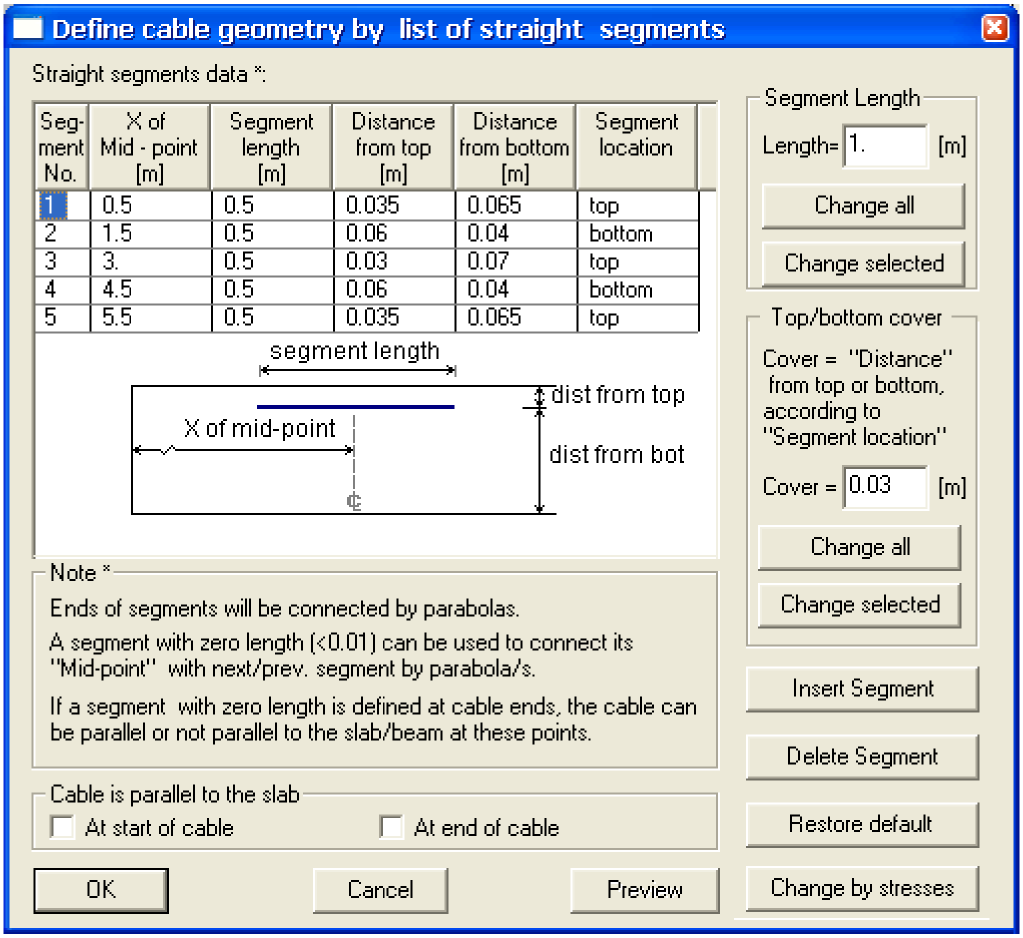
11.4.3.3 By list

The program identifies the maximum positive and negative moments in the slab and automatically creates a series of straight segments at these locations. Modify these segments; the program connects them with parabolas.



Alternatively, the program can automatically select a profile according to the stresses in the beam.

For example:



Revise segment length

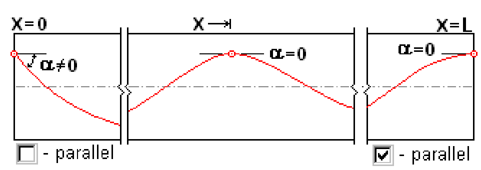
Individual segment:

- Type in a new segment length
- highlight a segment
- click Change selected

All segments:

- Type in a new segment length
- click Change all

Note:



- Cables with **zero length** are handled as follows:
  - at slabs ends:
    the parabola starts/terminates at the slab end but its tangent is not parallel to the slab axis unless specified by the user.
  - internal segments:
    the two parabolas join at the specified X coordinate.

Revise cover

Individual segment:

- Type in a cover value
- highlight a segment
- click Change selected

All segments:

- Type in a new cover value
- click Change all

Insert segment

- highlight the line before the new segment
- click Insert segment

Delete segment

- highlight the line
- click Delete segment

Change by stresses

The program attempt to find a trajectory that falls within the allowable eccentricity range.

For indeterminate beams or slabs -

- select the trajectory
- 1+2= Solve the model
- if the cable is outside the range return to this dialog and select this option again.

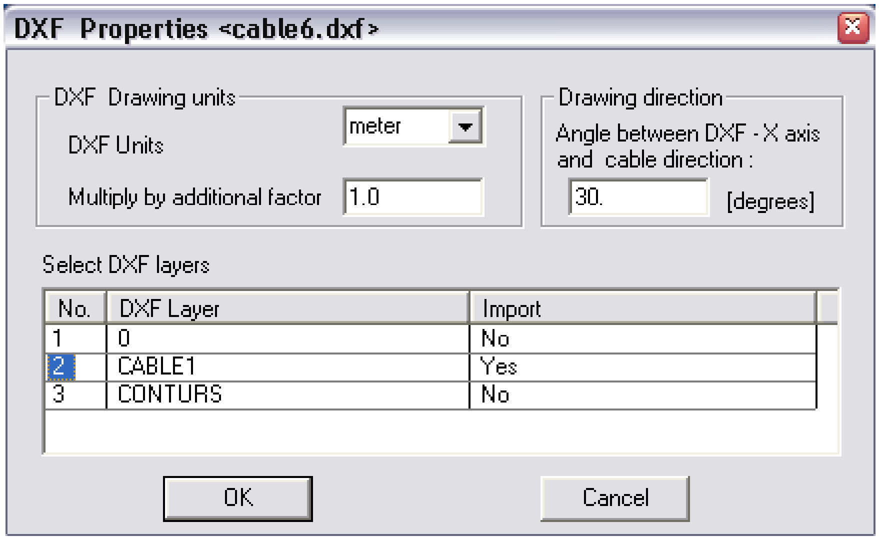
Select Restore default to restore the initial program selected trajectory.

Cable is parallel to the slab

This option is relevant only for end segments with zero length, where the cable may or may not be parallel to the slab axis. Refer to Revise segment length 1128 above.

11.4.3.4 DXF

Import a cable trajectory from a DXF file.



DXF Drawing unit

The DXF drawing will be scaled according to the program units:

- **DXF units:**
  Select a unit from the list box
- **Multiply by an additional factor**
  The DXF dimensions will be multiplied by the value entered here. For example, enter a factor of 2 if you want to double the size of the drawing in the background.

Drawing direction

The program assumes that the beam/slab axis is horizontal in the DXF drawing (angle = 0). Enter a different value if the cable trajectory is drawn at a different angle.

Select layers

Toggle the layer that contains the trajectory to YES.

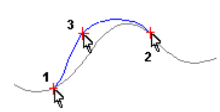
11.4.3.5 Edit

Select one of the following options:

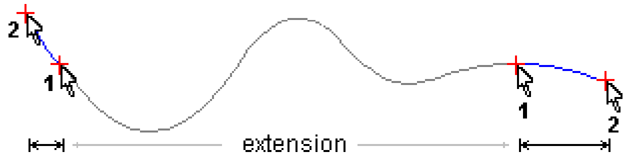
Modify any part of an existing cable:

- select the start and end points of the part to be modified (**1**),(**2**). Note that the points may be in two different segments.
- move the cursor to a new point (**3**) that defines the geometry of the part:

The program creates one or more new segments between the selected points.



Extend the end point of a cable:

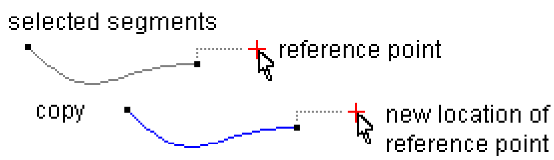


- select one of the end points of the cable (**1**)
- move the to the new end point (**2**); note that the program extends the existing end parabolic segment without modifying the existing part.

Delete selected segments or parts of segments from the cable. Segments are selected using the Segment selection 1108 options.

Copy cable segments to any location in the beam:

- select the cable segments using the Segment selection 1108 options
- select a reference point; this point is used to define the new location of the copied segments:



the reference point (both locations) can be either at a cable end point, either end of the beam or at any coordinate:

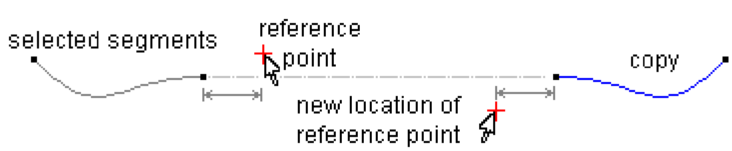
- ▫ cable end : move the to the end point so that it is highlighted with the ■; click the mouse.
- ▫ beam end : move the to adjacent to the beam end a ■ is displayed; click the mouse.
- ▫ coordinate : click  at the bottom of the screen and move the to the correct location; click the mouse.

- select the new location of the reference point using any of the above methods.

The program draws the copied segments in their new location.

Create a mirror image of selected cable segments at any location in the beam:

- select the cable segments using the Segment selection 1108 options
- select a reference point; the horizontal coordinate of this point is used to define the new location of the mirrored segments (the segments are mirrored about the vertical axis but their height coordinate is unchanged):



the reference point (both locations) can be either at a cable end point, either end of the beam or at any coordinate:

- ▫ cable end: move the to the end point so that it is highlighted with the ■; click the mouse.

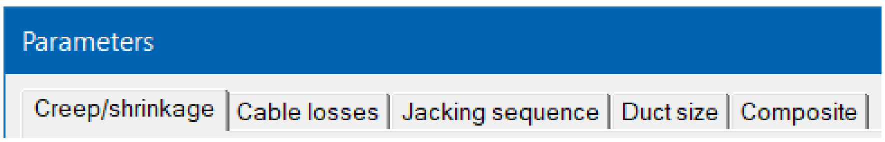
- beam end: move the to adjacent to the beam end a ■ is displayed; click the mouse.
- coordinate: click By coordinate at the bottom of the screen and move the to the correct location; click the mouse.

- select the new location of the reference point using any of the above methods.

The program draws the mirrored segments in their new location.

Define cable Return to the previous menu.

11.4.4 Parameters

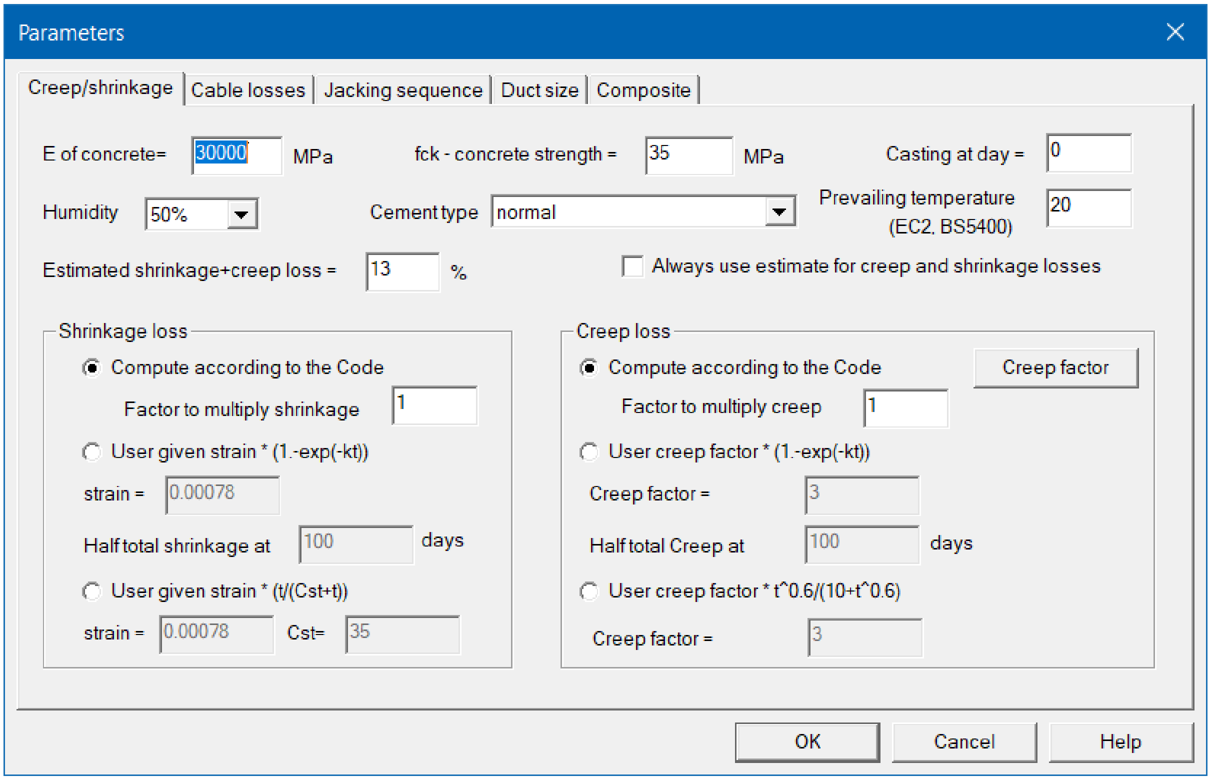


Creep/shrinkage 1132
Cable losses 1133
Jacking sequence 1136
Duct size 1138
Composite 1138

Note: these parameters override the values specified in the Default parameters 1145 option.

11.4.4.1 Creep/shrinkage

Specify creep/shrinkage parameters for the current beam.

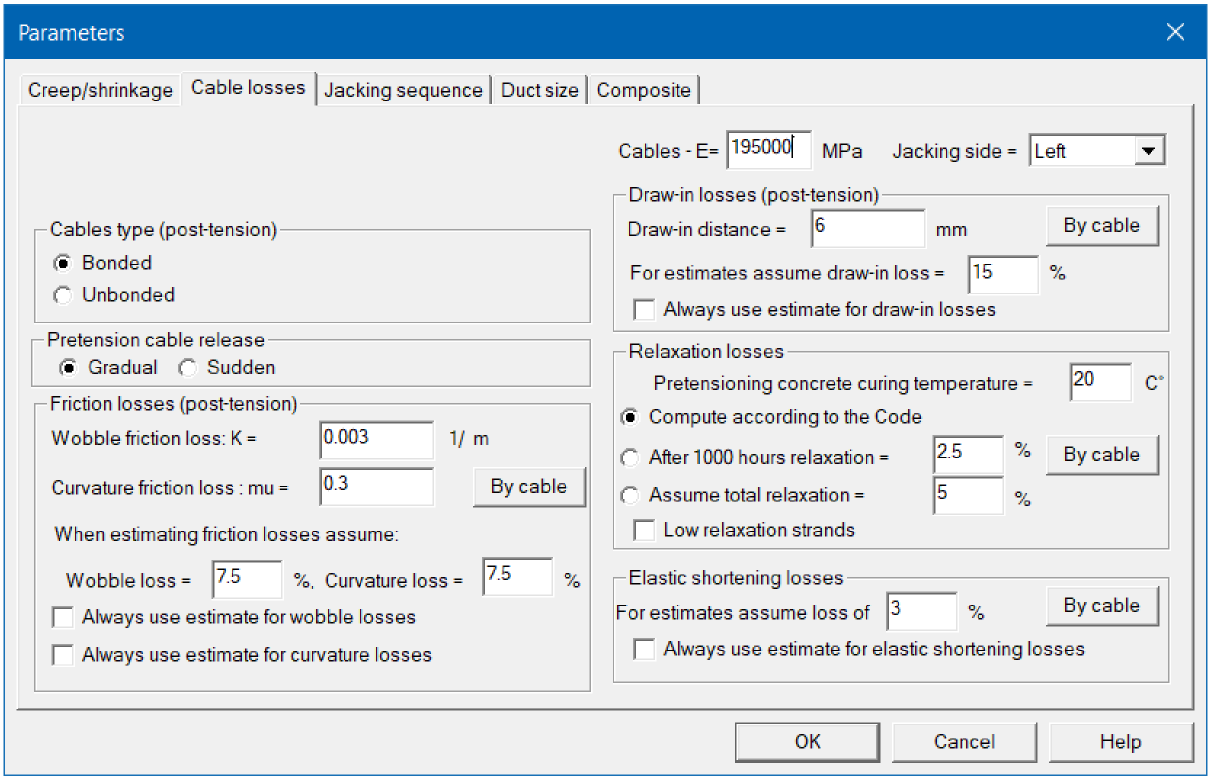


Note: These parameters override the values specified in the Default parameters 1148 option.

For all parameters refer to Default parameters - creep/shrinkage 1148.

11.4.4.2 Cable losses

Define the cable loss parameters for the current beam.
Note: parameters defined here override the values specified in the Defaults 1150 option in the main side menu.



Jacking side

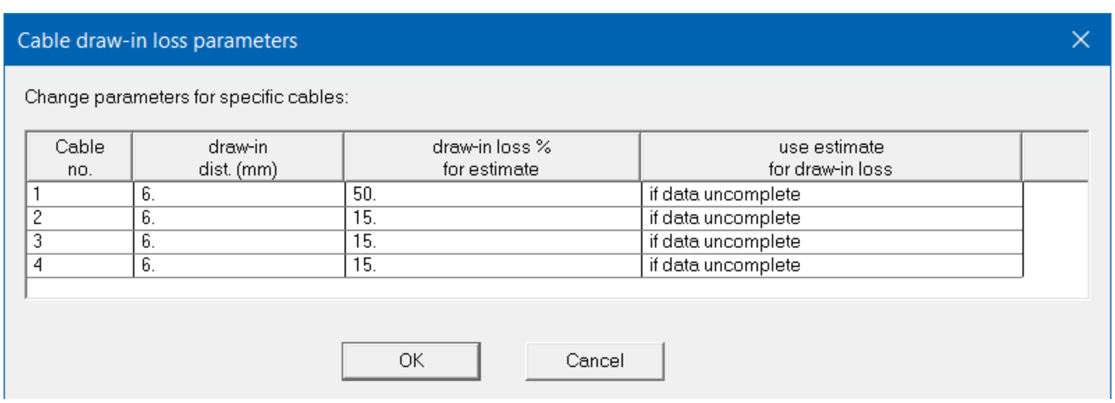
Specify the default jacking side for all cables in this beam. The jacking side for individual cables may be revised in the Jacking sequence option.

Draw-in losses

Refer to Defaults - Cable losses - Draw-in 1152 for an explanation on the option.

Different parameters may be defined for selected cables in the current beam:

- Click By cable
- Click and highlight a cable, then modify the parameters:



-

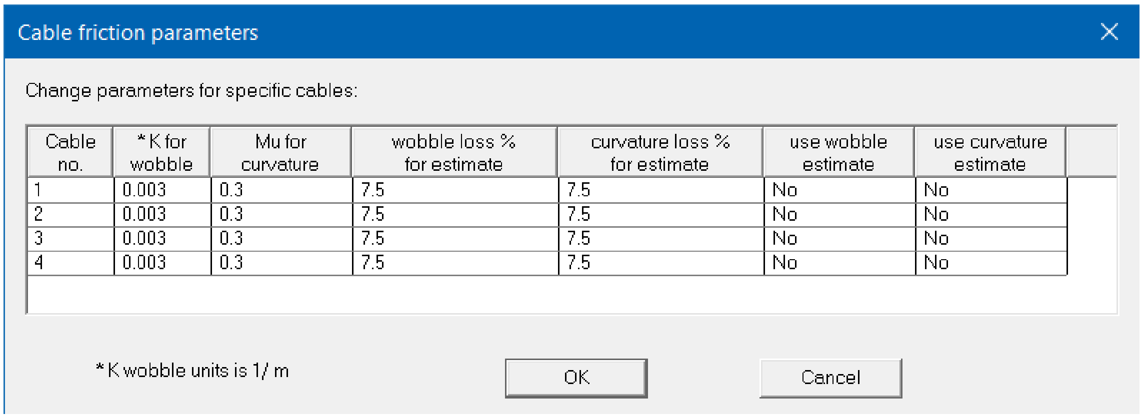
To use the draw-in loss estimate throughout the design and suppress the exact calculation, set the if data uncomplete / ☑ use estimate always check box to ☑; "**If data incomplete**" will be revised to "**Always**".

Friction losses

Refer to Defaults - Cable losses - Friction[1152] for an explanation on the option.

Different parameters may be defined for selected cables in the current beam:

- click By cable
- Click and highlight a cable, then modify the parameters:


Cable friction parameters

Change parameters for specific cables:

Cable no.	*K for wobble	Mu for curvature	wobble loss % for estimate	curvature loss % for estimate	use wobble estimate	use curvature estimate
1	0.003	0.3	7.5	7.5	No	No
2	0.003	0.3	7.5	7.5	No	No
3	0.003	0.3	7.5	7.5	No	No
4	0.003	0.3	7.5	7.5	No	No

*K wobble units is 1/ m

OK Cancel

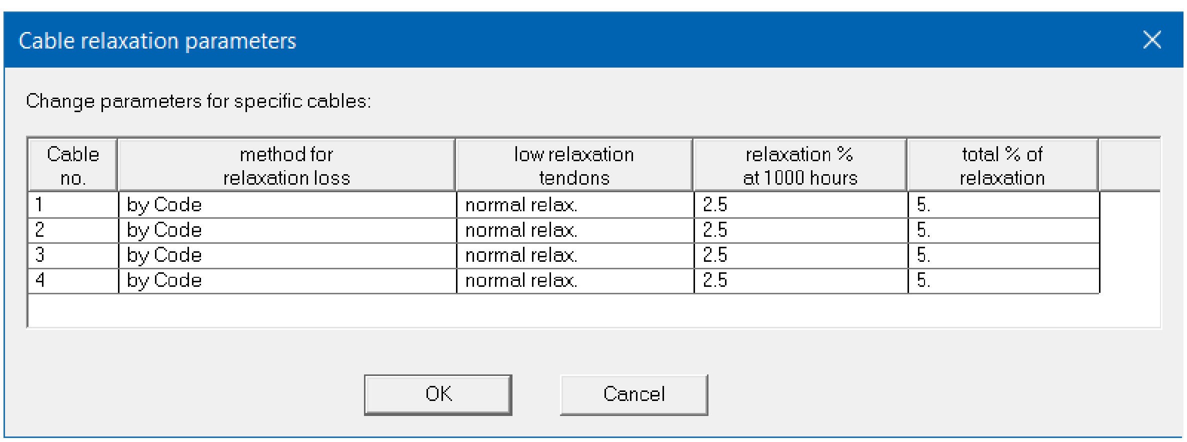
To use the wobble and/or curvature estimates throughout the design and suppress the exact calculation, set the No / ☑ use estimate, No / ☐ use estimate check boxes to ☑; "**No**" will be revised to "**Always**".

Relaxation losses

Refer to Defaults - Cable losses - Relaxation[1152] for an explanation on the option.

Different parameters may be defined for selected cables in the current beam:

- click By cable
- Click and highlight a cable, then modify the parameters:


Cable relaxation parameters

Change parameters for specific cables:

Cable no.	method for relaxation loss	low relaxation tendons	relaxation % at 1000 hours	total % of relaxation
1	by Code	normal relax.	2.5	5.
2	by Code	normal relax.	2.5	5.
3	by Code	normal relax.	2.5	5.
4	by Code	normal relax.	2.5	5.

OK Cancel

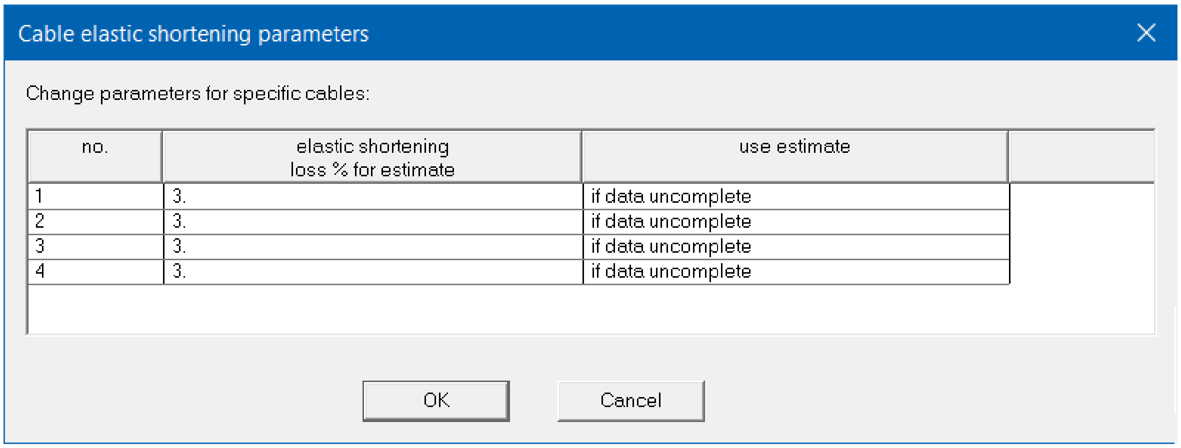
Elastic shortening losses

Refer to [Defaults - Cable losses - Elastic shortening]1153 for an explanation on the option.

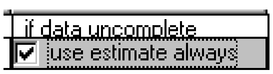
Different parameters may be defined for selected cables in the current beam:

- click 

- Click and highlight a cable, then modify the parameters:



To use the elastic shortening estimate throughout the design and suppress the exact calculation, set the



check box to ☑; "**If data incomplete**" will be revised to "**Always**".

All other parameters

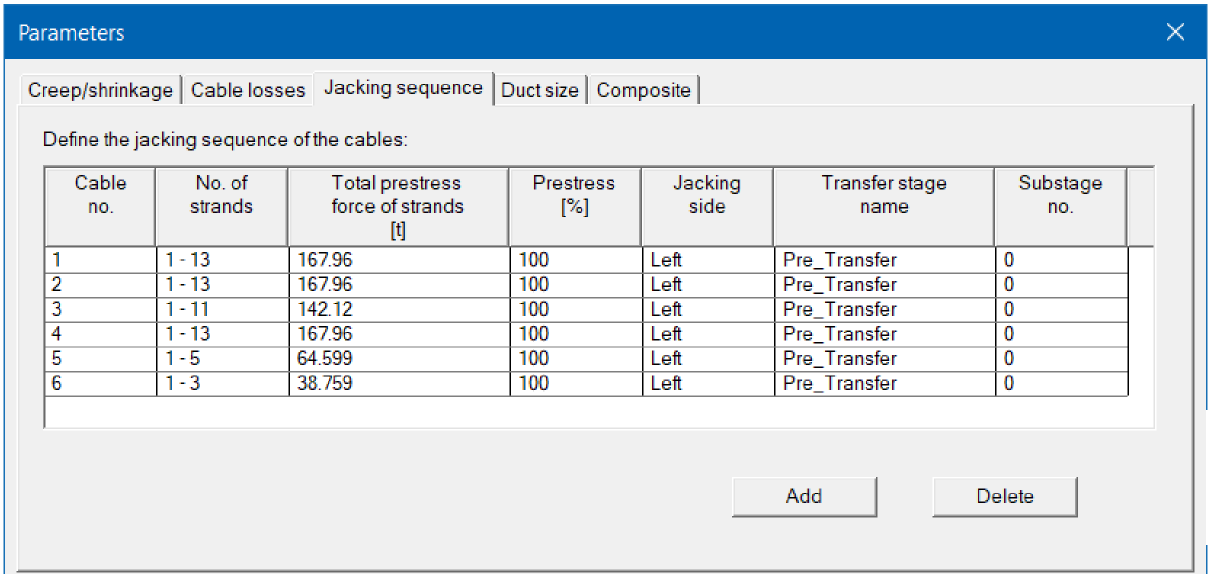
Refer to [Default parameters - cable losses]1150.

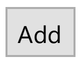
11.4.4.3 Jacking sequence

Specify the jacking sequence and side for each cable.
There are two options available:

- Add a jacking stage for all strands in a cable, i.e. jack all strands at two or more stages.
- Split the cables into two or more strand groups, each group jacked at a different stage or from a different end.

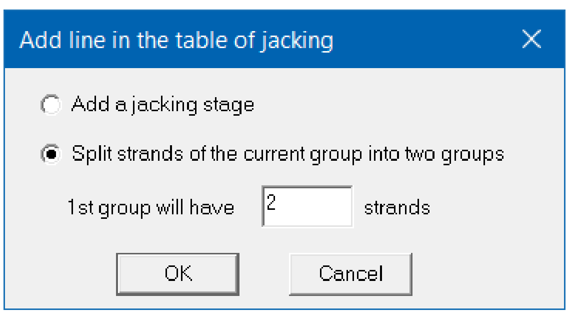
Select the cables:



- click and highlight a line, then click 

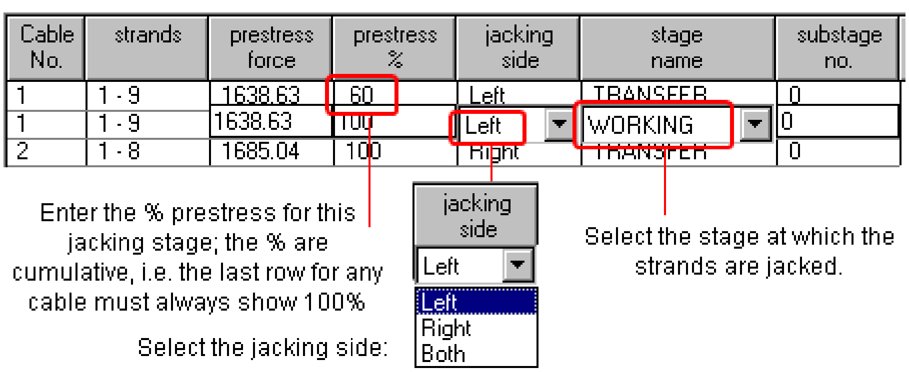
.

Specify whether to add a jacking stage or to split a cable:



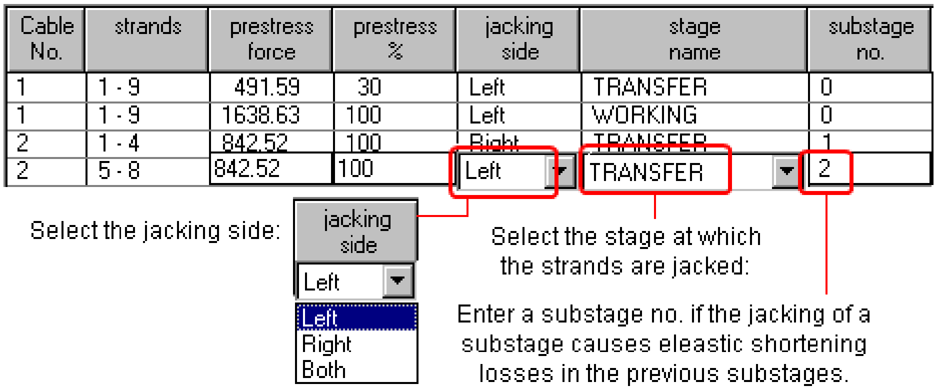
Add a stage

Add a jacking stage for all strands in a cable, i.e. jack all strands at two or more stages.



Split into strands

Split the cables into two or more groups of strands, each group jacked at a different stage or from a different end.

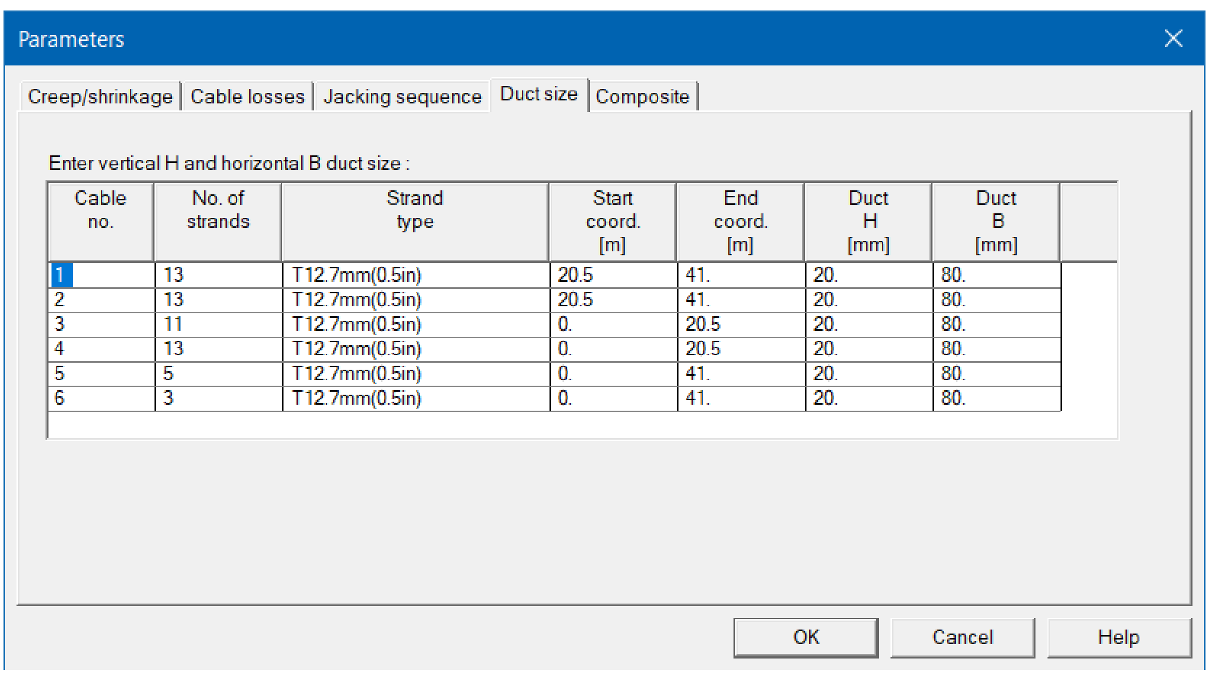


Note:

- *Cable no. 1 is jacked in 2 stages: to 30% and then to 100% of the prestress force (all of the strands are jacked together at the same side in both stages).*
- *Cable no. 2 is jacked in 2 substages of the same stage, i.e. both substages are at the same time (day), but the second substage causes elastic shortening of the strands jacked in the first substage.*

11.4.4.4 Duct size

Specify the width and height of the ducts for each cable:

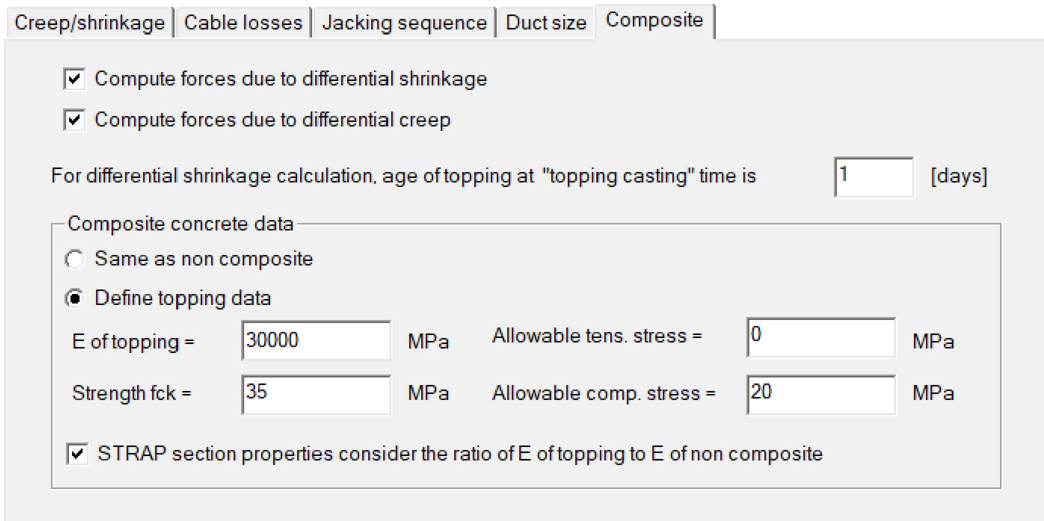


The program uses the duct sizes when drawing the perpendicular cables in the Define cable 1114 option.

11.4.4.5 Composite

Specify composite section properties:

Note: parameters defined here override the values specified in the Defaults 1150 option in the main side menu.



Compute forces due to differential shrinkage/creep

Check the check box for calculation differential shrinkage and creep. For more information refer to General - configuration/composite change 1106.

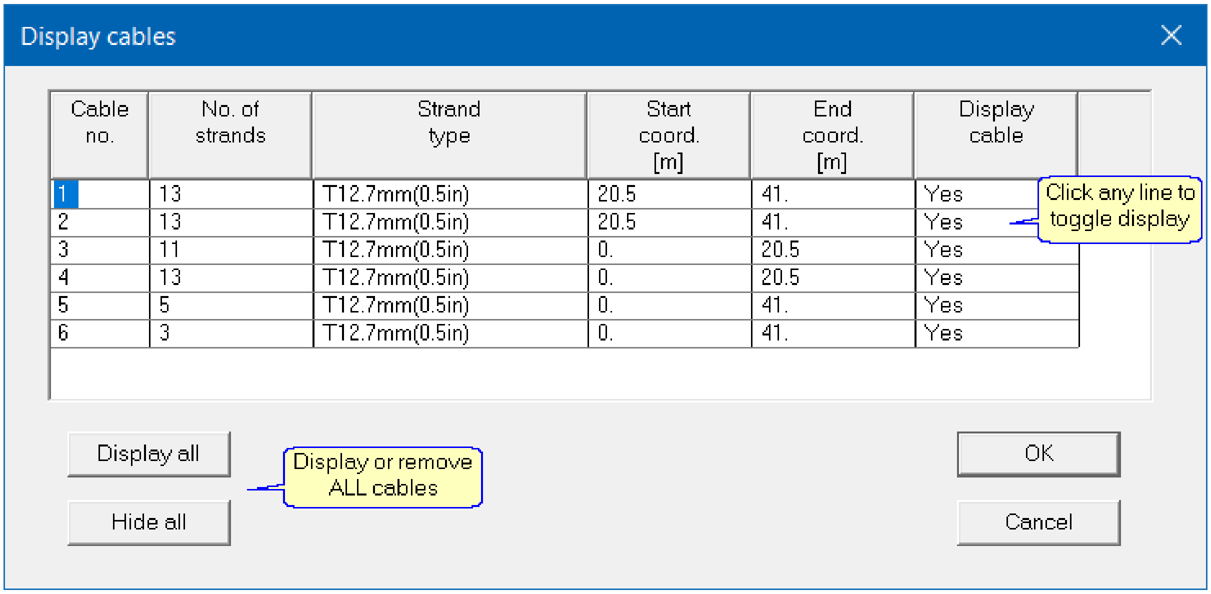
Composite concrete data

For the calculation of differential shrinkage/creep the user must specify the composite concrete properties:

- **Same as non composite**
  The program will assume that concrete properties of topping are identical to the non composite section.
- **Define topping data**
  Usually different type of concrete will be used for the cast in place topping. Elasticity module, nominal strength and allowable stresses must be specified.
- **STRAP section properties consider the ration of E topping to E non composite-**
  In case the composite section defined in STRAP geometry already includes the ration between E topping and E non composite, the user must check the check box.

11.4.5 Display cables

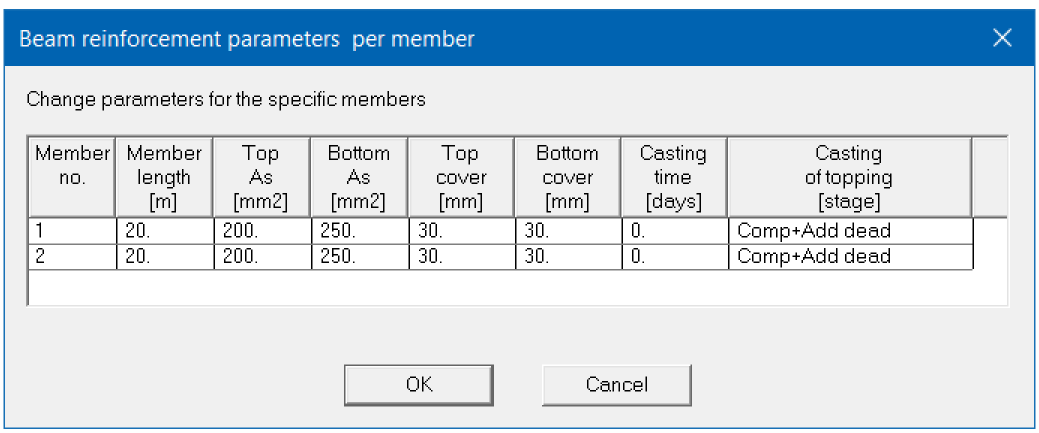
Cables may be removed from the display (they are ***not*** deleted):



Cable no.	No. of strands	Strand type	Start coord. [m]	End coord. [m]	Display cable
1	13	T12.7mm(0.5in)	20.5	41.	Yes
2	13	T12.7mm(0.5in)	20.5	41.	Yes
3	11	T12.7mm(0.5in)	0.	20.5	Yes
4	13	T12.7mm(0.5in)	0.	20.5	Yes
5	5	T12.7mm(0.5in)	0.	41.	Yes
6	3	T12.7mm(0.5in)	0.	41.	Yes

11.4.6 Beam As by member

Specify reinforcement parameters per *STRAP* member (not beam span):



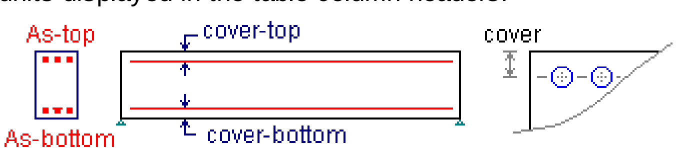
Member no.	Member length [m]	Top As [mm2]	Bottom As [mm2]	Top cover [mm]	Bottom cover [mm]	Casting time [days]	Casting of topping [stage]
1	20.	200.	250.	30.	30.	0.	Comp+Add dead
2	20.	200.	250.	30.	30.	0.	Comp+Add dead

Casting time

Enter the casting time for each span, measured from the 'start of construction' date (time=0 days).

Regular longitudinal reinforcement

Regular reinforcement may be added to the prestressed beam. Specify the area and cover according to the units displayed in the table column headers.



Note:

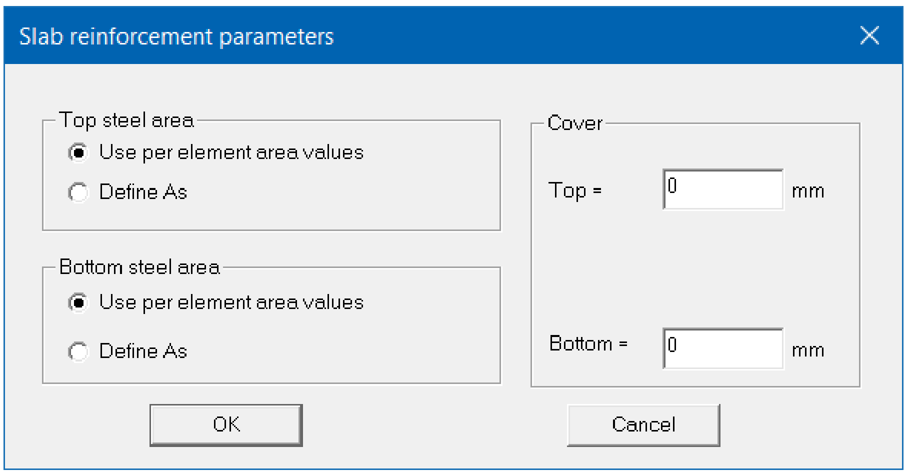
- the cover value indicates **gross** cover, from the surface to the center-of-gravity of the reinforcement
- the program assumes that the area is constant along the entire length of the member.
- the table initially displays the default values defined in the Defaults - Reinforcement parameters 1147 option.

Casting of topping

Specify the stage of slab casting for each span. The default is the construction stage where the section property values (area and moment-of-inertia) increase in *STRAP* geometry.

11.4.7 Slab reinforcement

Define regular slab reinforcement for the current slab that is different than the default values:



Note:

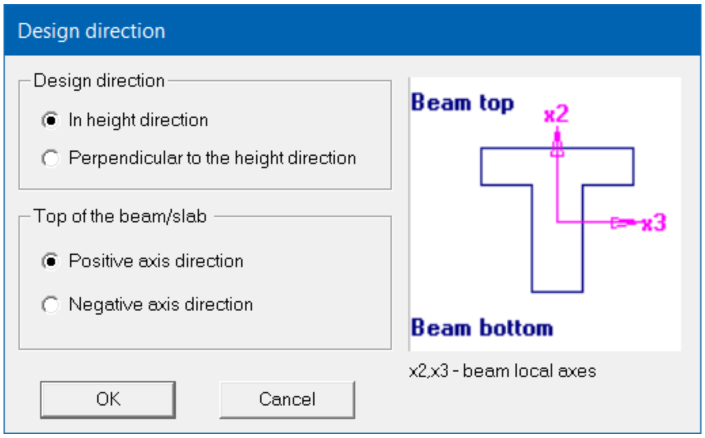
- the reinforcement area is per unit width of slab.
- these values supersede the default values, including area defined with the "By element" option.

11.4.8 Design direction

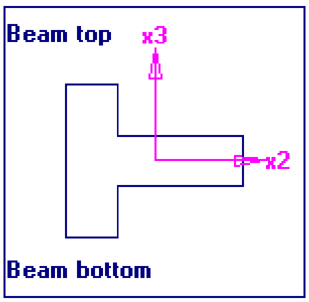
Specify the design direction, two options are available:

- For non-symmetric sections, the beam may be inverted without changing the design moments. The *STRAP* geometry section orientation is used by default.
- For space models only, the beams can be designed for either the M2 or M3 moments (but not biaxial). The default design direction is set in the Parameters - general option by specifying the 'height axis'; use this option to revise the design direction for a specific beam.

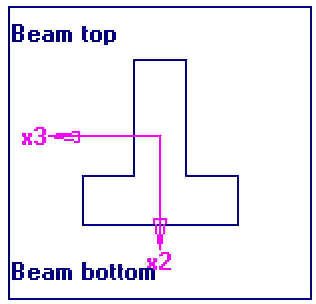
The following example shows the default design direction/top of beam and the effect of changing either of the options:



Perpendicular to the height direction

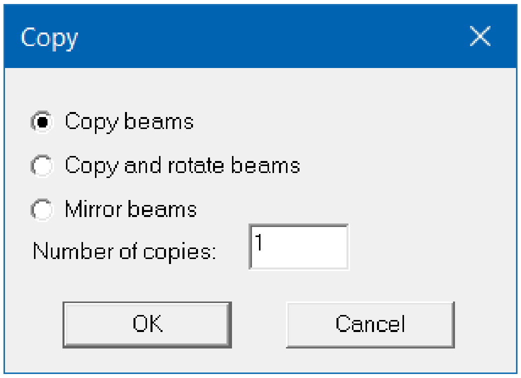


Negative axis direction



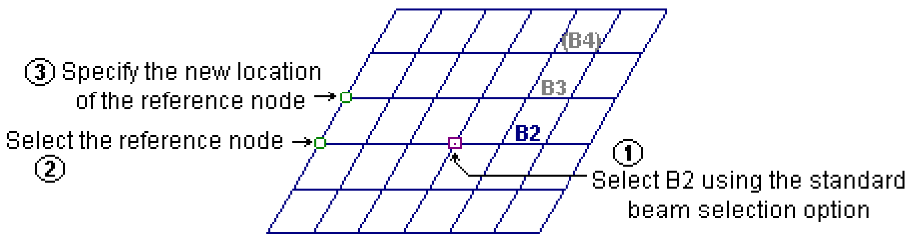
11.5 Copy

Create a new beam and copy the cables and parameters from an existing beam to the new one.



Copy beams

To create beams that are parallel to the original. For example, to copy beam B2 to create new beam B3:

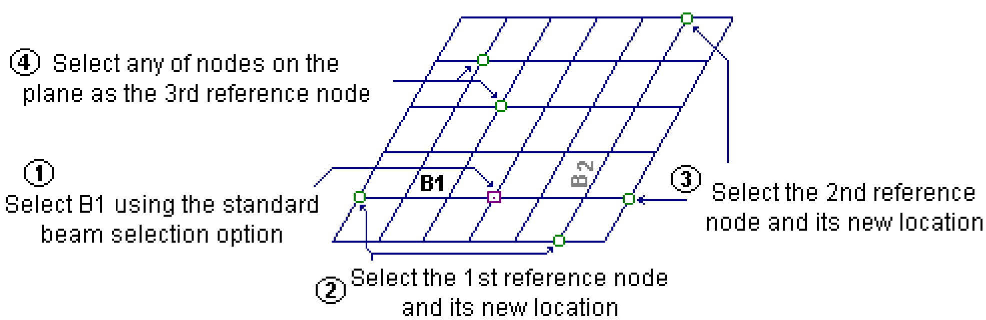


Note:

- *to create beam B4 at the same time, set **Number of copies** = 2*

Copy and rotate beams

To create beams that are not parallel to the original. For example, to copy beam B1 to create new beam B2:



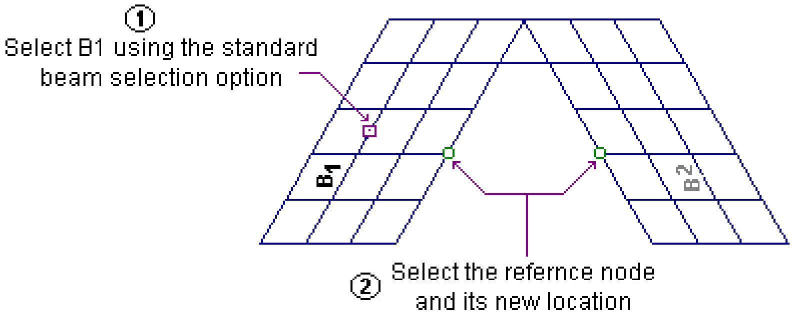
Note:

- the 3rd reference node is required when the two beams are not located on the same plane.

Mirror beams

To create beams that are a mirror of the original. For example, to copy beam B1 to create new beam

B2:



Note:

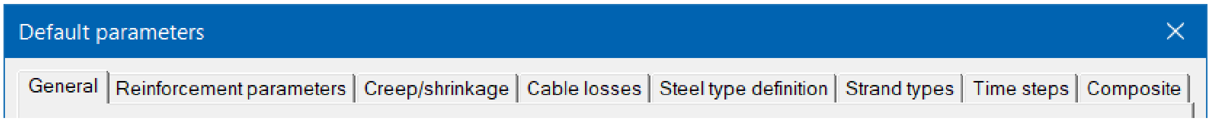
- the program mirrors the selected beam about the plane that bisects the line joining the reference node and its new location

---

Note:

- more than one copy can be created; the distance from the original to the 1st copy and the distance between any two copies are identical.

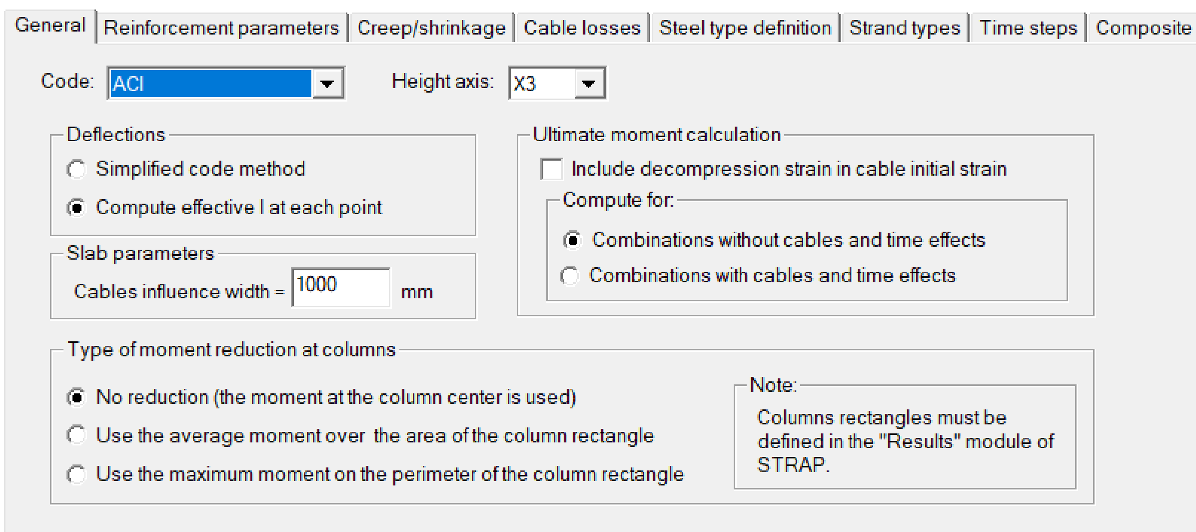
11.6 Default parameters



Refer to:
General 1145
Reinforcement 1147
Creep/shrinkage 1148
Cable losses 1150
Steel type definition 1153
Strand type 1154
Time steps 1154
Composite 1156

Note: All default parameters can be saved. From the top menu select File > Setup.

11.6.1 General

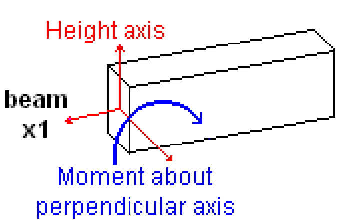


Code

Select a national design code from the list.

Height axis

For space models only:



The beams can be designed for either the M2 or M3 moments (but not biaxial). Select the moments by specifying a 'height axis' for the model; the program designs for the moments acting about the axis perpendicular to the plane formed by the height axis and the beam x1 axis.

The height axis may be revised for specific beams using the Design -

properties 1141 option.

Ultimate moment calculation

Include decompression strain in cable initial strain
The strain in the concrete becomes zero at the level of the prestressing cable at the decompression load. A corresponding decompression strain is developed in the cables. If the option is set to the program adds this decompression strain to the total strain when calculating the stress in the cables.

Compute for:

Combinations without cables and time effects
The prestress moment is the moment required to nullify the effect of the prestressing = **P(e)** and time effects (differntial creep and shrinkage). The value **is <u>not</u> added** to the ultimate moment capacity.

Combinations with cables and time effects
The prestress moment is the moment required to nullify the effect of the prestressing = **P(e)** and time effects (differntial creep and shrinkage). The value **is added** to the ultimate moment capacity.

For more info, refer also to Design assumptions

Deflections

Specify the method for calculating the deflections (for certain Codes only):

Simplified code method
The program calculates a single 'effective' moment-of-inertia for the entire span, based on the maximum service moment and the cracking moment at that point.

Compute effective I at each point
The program calculates the 'effective' moment-of-inertia at intervals along the span, the rotation at these intervals and the maximum deflection by integration of these results

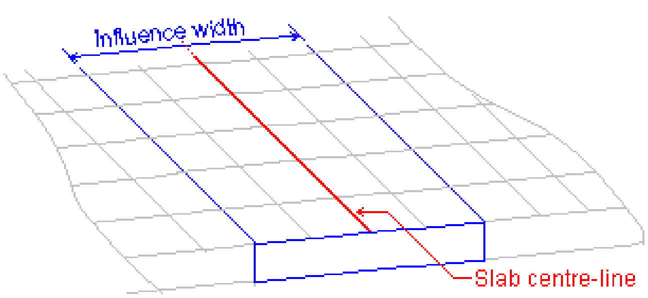
Note:

- *deflections cannot be calculated for slab "lines".*

For more info, refer also to Design assumptions for more details.

Cable influence width (for slabs only)

The slab center-line is defined in the Define option. This option defines the default associated slab width.



The program calculates the result diagrams along the defined line similarly to the "Results along a line" option in the *STRAP* results module, i.e. the results are per unit width, e.g. ton-meter/meter, ft-kip/ft, etc. Therefore, ***the influence width value does not affect the displayed results***. The program treats the influence width as the beam width and uses it to calculate the stresses, etc, resulting from the prestressing.

The influence width for specific slabs may be revised using the **Design** - Cable geom option.

Moment reduction at columns (for slabs only)

Finite element analysis usually gives exaggerated moment values at support nodes because the support is represented by a perpendicular line element with a zero dimension. In theory, a zero dimension support generates an infinite moment.

PRESTRESS has an option to 'reduce' the moments adjacent to the supports. You may calculate the reduced moments here if you defined the rectangles in Results.

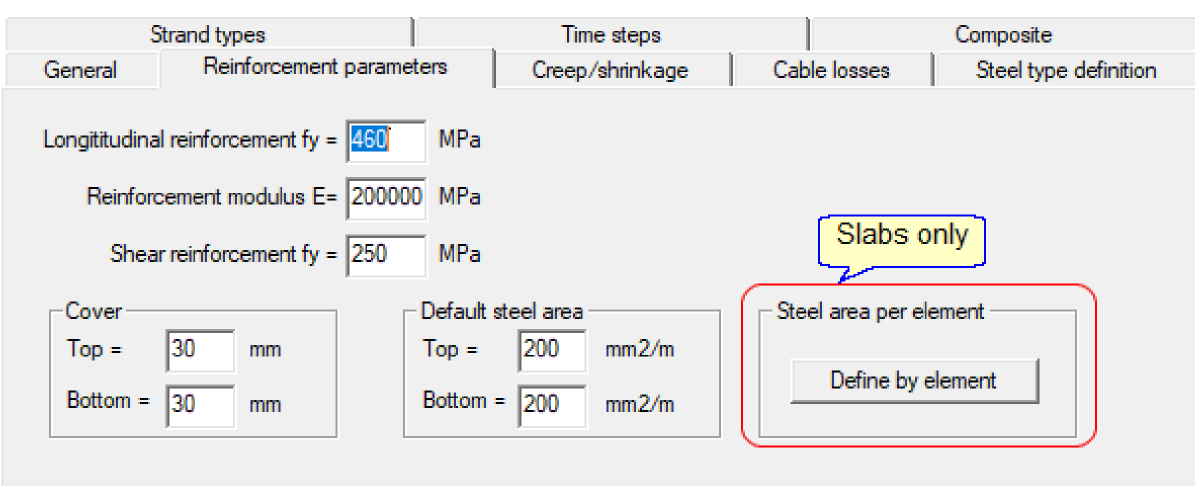
Select one of the following options:

- **No reduction**
  Use the moment at the column center as calculated, without reduction.
- **Use reduced moment ...**
  The program calculates and uses an average moment values over the defined rectangle area
- **Use maximum moment ...**
  The program uses the maximum moment value on the defined rectangle perimeter

For a detailed explanation and examples, refer to Results along a Line - General.

11.6.2 Reinforcement

Regular longitudinal reinforcement may be added to the prestressed beam or slab. Specify the default parameters for all beams or slabs in the model.
Area and cover values may be modified for individual beams using the **Design** - For beams As by member option and for slabs Slab reinforc..



fy

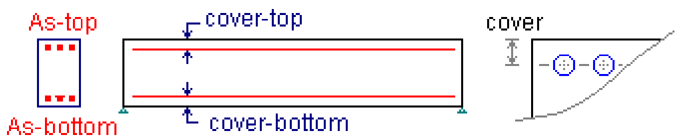
Specify the ***nominal*** steel grade for regular longitudinal and transverse reinforcement.

E modulus

Specify the modulus of elasticity (E) according to the stress units displayed adjacent to the option.

Steel area / cover

Specify the area and cover according to the units displayed in the dialog box.



The cover value indicates gross cover, from the surface to the centre-of-gravity of the reinforcement. Reinforcement areas defined here can be displayed graphically by selecting **Display - Element reinforcement** in the menu bar.

Beams:

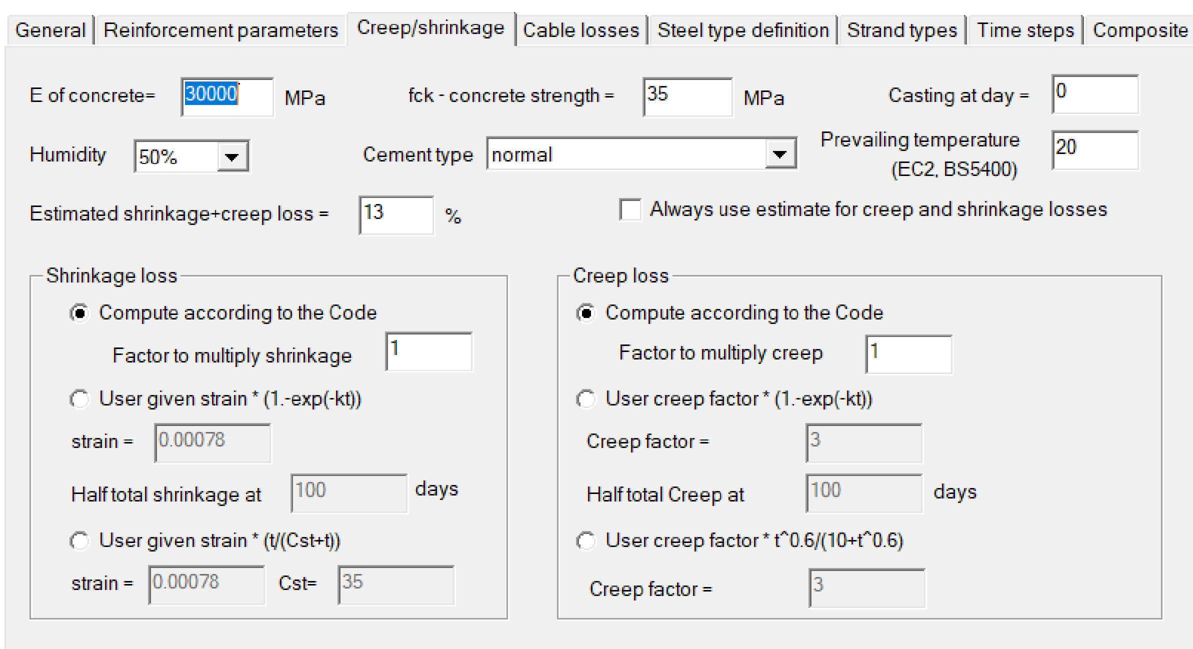
- the program assumes that the area is constant along the entire length of the beam.

Slabs:

- values are for both directions
- reinforcement area are per unit width.
- Slabs: different values of reinforcement area may be defined for selected elements:
  - click Define by element
  - select elements using the standard Element selection option.
  - enter top/bottom steel areas

11.6.3 Creep/shrinkage

Specify the default creep and shrinkage parameters for ***all*** beams/slabs in the model:



E of concrete

Specify the modulus of elasticity (E) according to the stress units displayed adjacent to the option.

Fck

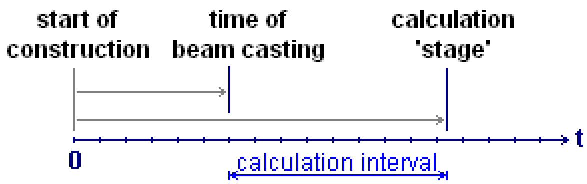
Specify the ***nominal*** concrete strength.

Casting at day

Specify the day that the beam is cast (from the start of construction).

Note:

- the beam calculation 'stages' are also defined from the start of construction; the creep and shrinkage equations require the 'Casting at day' value in order to determine the time from casting to the calculation date.



Humidity

Specify the average relative humidity (%)

Cement type

Select a cement type from the options displayed (required in some Codes only)

Temperature

Specify the average temperature value (ºC)

Estimated loss

Enter the estimated total creep + shrinkage loss; the program uses this value where required until the exact losses are calculated.

- To suppress the exact calculation and to use the estimate throughout the program, set ☑ **Always use estimate for creep and shrinkage losses**
- Note that in models with a 'configuration change' the program ***always*** uses the exact losses when calculating the stresses resulting from the change (e.g. differential creep between beam and topping); in such cases ☑ **Always use estimate ...** applies only to the calculation of the cable force. Therefore the following creep and shrinkage parameters must always be specified in models with a configuration change.

Shrinkage

Select one of the following methods for calculating the shrinkage loss:

Compute according to code
The program calculates the shrinkage loss according to the current Code. Refer to Design assumptions for more details. *Note: the shrinkage loss may be modified by a user-defined factor.*

User given strain * (1. - exp(-kt))

The shrinkage strain is a user-defined strain modified by a factor in the form $\mathbf{1\text{-}e^{-kt}}$. The program requires the date of 'half total shrinkage' in order to calculate the value of 'k'.

User given strain * (t/(Cst+t))

The shrinkage strain is a user-defined strain modified by a factor in the form $\mathbf{\frac{t}{Cst+t}}$, where **Cst** is also defined by the user.

Creep

Select one of the following methods for calculating the creep loss:

Compute according to code
The program calculates the creep loss according to the current Code. Refer to Design assumptions for more details. *Note: the creep loss may be modified by a user-defined factor.*

User creep factor * (1. - exp(-kt))
The creep factor is in the form $\mathbf{1\text{-}e^{-kt}}$ and may be modified by a user-defined factor. The program requires the date of 'half total creep' in order to calculate the value of 'k'.

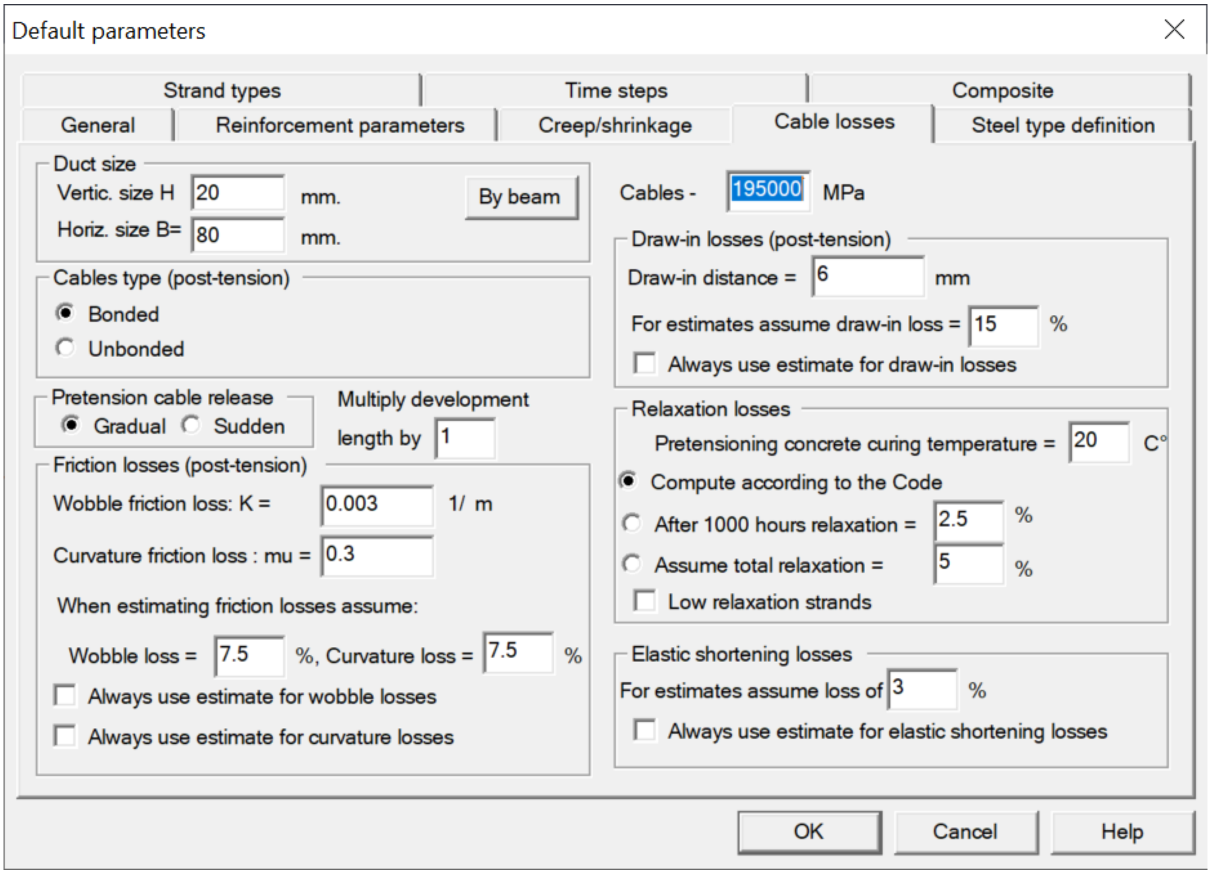
User creep factor * (t^0.6/10+t^0.6)

The creep factor is in the form $\mathbf{\frac{(t-t_i)^{0.6}}{10+(t-t_i)^{0.6}}}$ and may be modified by a user-defined factor.

In the "Design - Losses - creep/shrinkage 1132" option, click [Creep factor] to display the differential creep factor between any two dates (C_{t2}-C_{t1}). The result is for information only

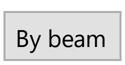
11.6.4 Cable losses

Specify the default parameters for all cable losses (except creep and friction) for ***all*** beams/slabs in the model.
Note: the symbols may vary according to the Code.



Duct size

Define the duct size. The perpendicular ducts are drawn on the design screen when designing beams and slabs in order to avoid conflicts.

Click 

to define different duct sizes for individual beams and slabs.

To display the current duct size values, select **Display - duct size** in the menu bar.

E modulus

Specify the value of the modulus of elasticity of the prestressing cables.

Cable type

For post-tension cables, specify the default model cable type as bonded or unbonded. The type for individual beams can be modified using the **Design - Losses - Cable losses** option
For unbonded cables, the program calculates the average stress in the cables as the effective stress (after losses) plus an additional stress. Specify the value of the additional stress.

Cable release

Specify the release for pre-tension cables. This setting effects the calculation of the transmission length, over which the prestressing force is fully transmitted to the concrete.

Multiple development length by

Insert a factor for the calculation of the development length. The program will calculate the development length according to the selected load and multiply it by the inserted factor.

Draw-in

Draw-in (or anchorage seating) losses occur in post-tensioned members due to the seating of wedges in the anchors when the jacking force is transferred to the anchors. Enter the draw-in (slip) distance according to the units displayed.

The program initially uses the estimated loss specified by the user and then calculates the exact loss based on the defined cable profile, if requested by the user.

- Estimated losses
  Enter the estimated % loss for draw-in. To use these estimates throughout the design and suppress the exact calculation, set the ☑ **Always use estimate ...** checkbox.

Friction

Loss of prestress occurs due to friction between the strands and the surrounding ducts. There are two components:

- curvature effect.
- wobble effect.

The program initially uses the estimated loss specified by the user and then calculates the exact loss based on the defined cable profile, if requested by the user.

Estimated losses:

Enter the estimated % loss for wobble and curvature. To use these estimates throughout the design and suppress the exact calculation, set the ☑ **Always use estimate ...** checkboxes

Exact losses:

- wobble effect

  The equation in all codes is in the form: $\Delta f = f_1(1-e^{-KL})$
  Define the value **K**.

- curvature effect

  The equation in all codes is in the form: $\Delta f = f_1(1-e^{-\mu\alpha})$
  Define the value μ.

Note:

- To define different values for selected cables, click [By cable]
- the symbols vary according to the Code.
- the maximum friction loss is at the far end if the jacking is from one end. The friction loss varies along the span and the program calculates the value at all locations along the beam.

Relaxation

Stress-relieved strands are subject to prestressing loss due to constant elongation with time. Select one of the following methods to calculate the relaxation loss:

◉ **Compute according to the Code**
The program calculates the relaxation loss according to the Code.

◉ **After 1000 hours relaxation =**

Enter the relaxation loss after 1000 hours; the program will calculate the loss at any stage from the Code equations based on this value.

Assume total relaxation =
Enter the total relaxation loss (500,000) hours; the program will calculate the loss at any stage from the Code equations based on this value.

For low-relaxation strands, check the **Low relaxation strands** parameter;

Refer to Design assumptions for more details

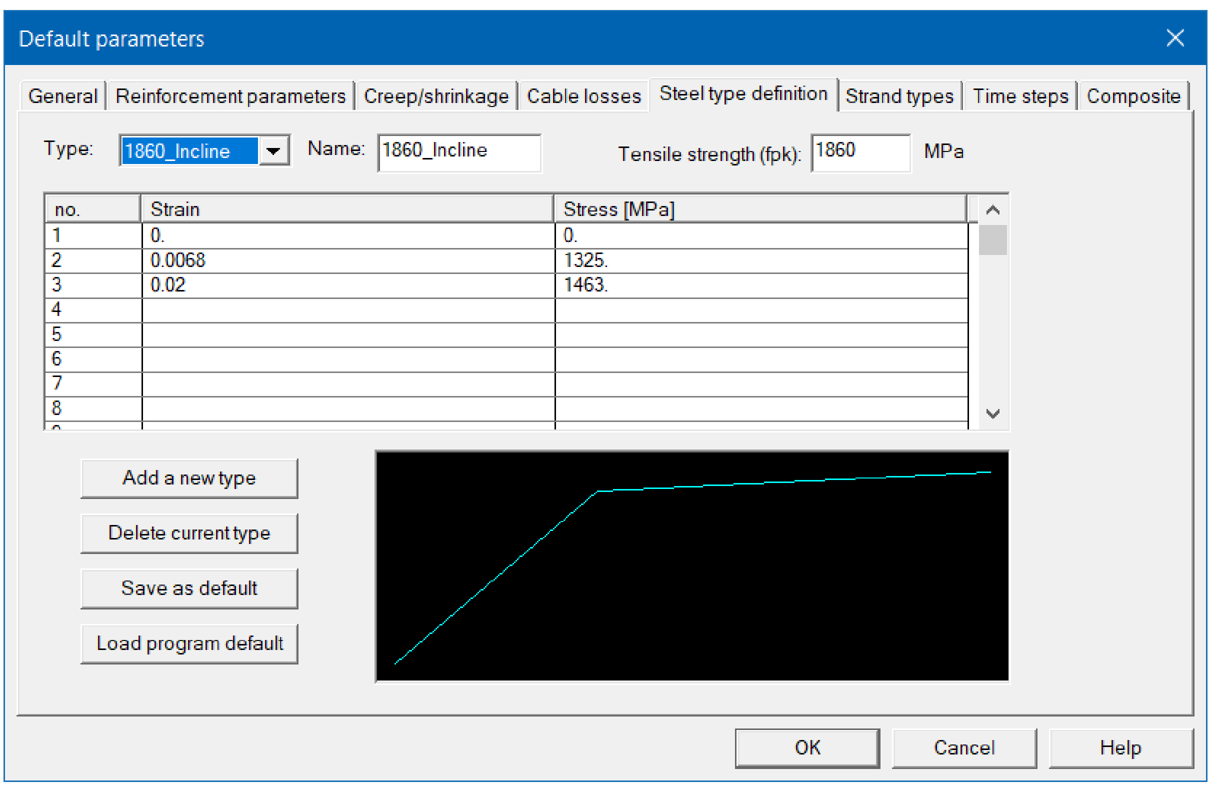
Elastic shortening

In post-tensioned beams, the elastic shortening loss varies from zero if all of the cables are jacked simultaneously to one-half the value for an equivalent prestressed beam if several sequential jacking steps are used.
The program initially uses the estimated loss specified by the user and then calculates the exact loss based on the defined cable profile, if requested by the user
Note that the jacking sequence is defined in the Jacking sequence 1136 tab in Design - Losses option.

- Estimated losses
  Enter the estimated % loss for elastic shortening. To use these estimates throughout the design and suppress the exact calculation, set the **Always use estimate ...** checkbox

11.6.5 Steel type

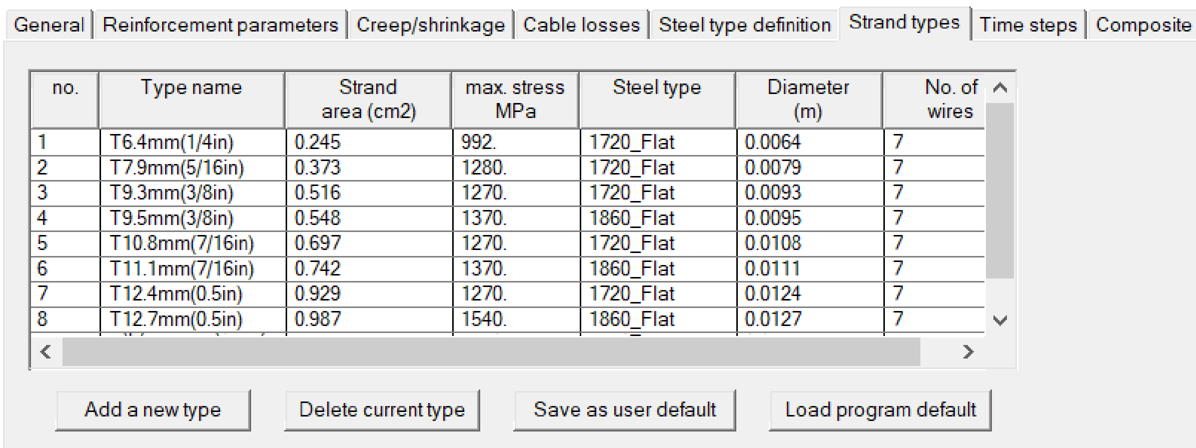
Define the stress-strain curves for all of the prestressing steel types. The curves are required when calculating the ultimate moment capacity of the beams. The Steel types are assigned to the strands in the Strand types option



- select an existing type from the list box or click Add a new type
- enter a name for the type and define the value of **fpk**
- enter the stress and strain values of points along the curve. The program assumes a linear distribution between the points.
- click Save as default to designate any type as the "program default". To recall the default values into the table (for the current type), click Load program default.

11.6.6 Strand types

Define/edit strand types; define/revise the data (area, max. stress, etc) and assign a steel type to the strand.



The defined strand types will be available during cable definition.

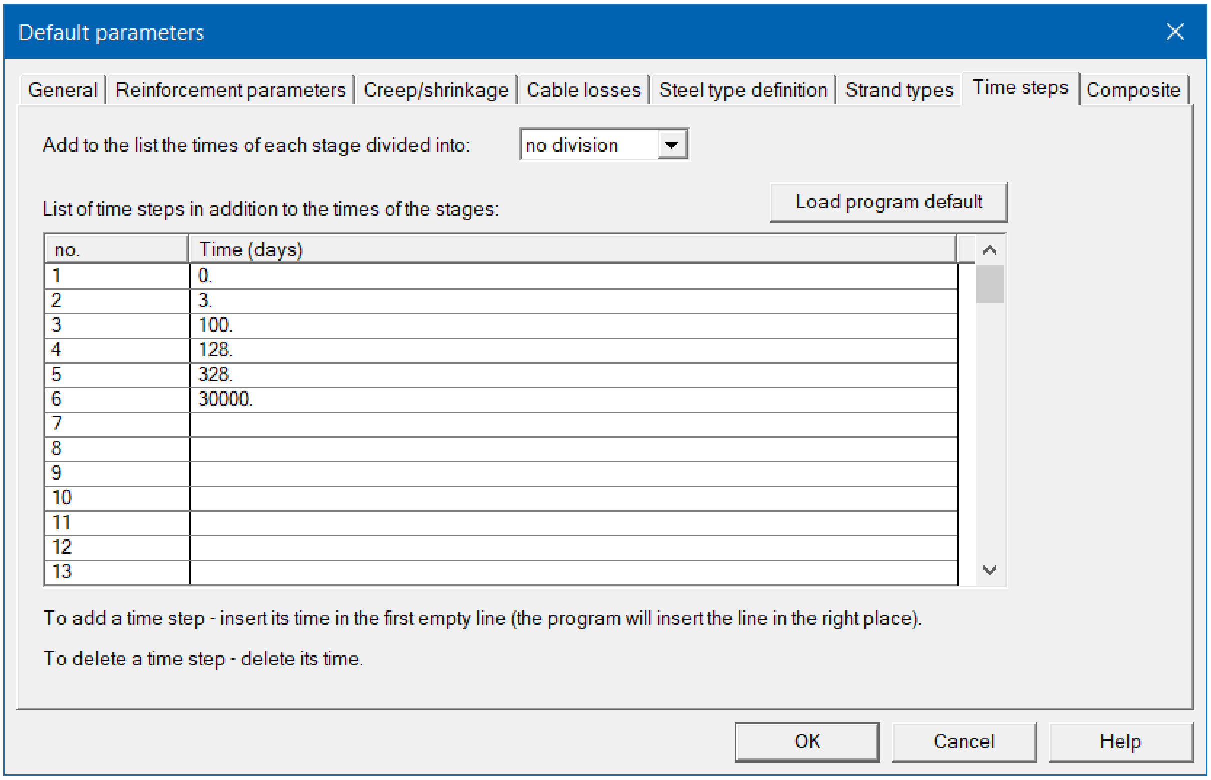
- click Save as default to save the current table as the "user default table".
- To restore the original program values into the table, click Load program default.

Note:
- the "User default table" will be the initial strand table for all new models.
- Load program default affects only the current model.

11.6.7 Time steps

The program calculates losses, results, etc. at various time intervals:

- stages defined by the user in the Stages option.
- When a pre-stress load is presented.
- at additional time steps specified in this option.

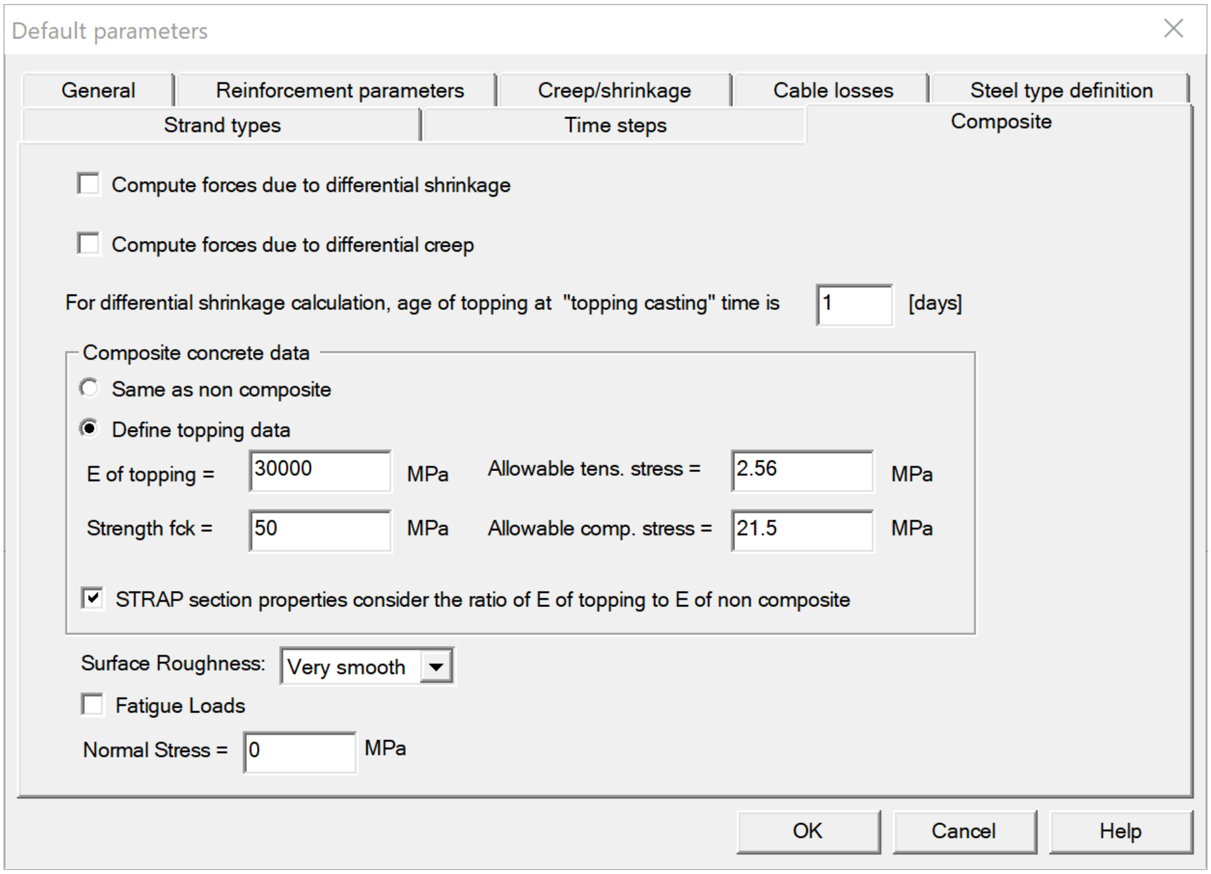


- Enter the step values (days) in any order
- Each step can be subdivided into additional intervals; select "**no division"** or 2 to 5 intervals from the list box.
- This table can be reset to the original settings by clicking Load program default.

Note:

- Note that a separate STRAP load case is created for each time step; solution time may be significantly increased in large models.

11.6.8 Composite



Compute forces

For composite beams, specify whether to calculate additional moments generated by differential creep and shrinkage.
For more information, refer to composite/configuration changes 1106.

Topping casting time

Differential shrinkage begins only when the section becomes fully composite. Specify the number of days after slab casting.

Composite concrete data

The program initially assumes that the topping concrete is the same as the beam/slab concrete.
To specify a different concrete for the topping, set **Define topping data** and enter the parameter values.

Note**:**
if the composite section **includes** consideration of the ration between E (elastic modulus) of the topping and the non-composite, check the box "STRAP section consider the ration of E topping to E of non composite".

Surface roughness/Fatigue loads/Normal stress

For horizontal shear calculations due to different casting times:

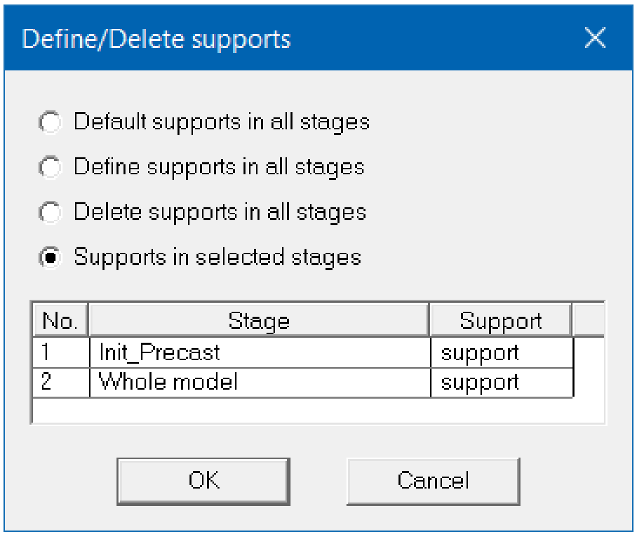
- Select the surface roughness.
- Specify if the loads are classified as fatigue loads. (according to the selected code).
- Insert the normal stress. The normal stress is perpendicular to the horizontal shear interface.

11.7 Restraints

Add/delete supports at *STRAP* nodes. Different supports may be defined for each construction stage.

Note:

- the supports defined in this option are used only for *PRESTRESS* deflection calculations; moment, stress or shear calculations are not affected.
- *STRAP* supports and deflections are not affected.



Default supports at all stages

The program uses the supports defined in *STRAP* geometry for each construction stage for the *PRESTRESS* calculations.

Define supports in all stages

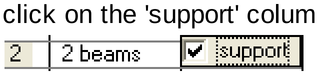
Select nodes using the standard node selection options; the nodes are assumed to be supports in all construction stages.

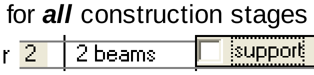
Delete supports in all stages

Select nodes using the standard node selection options; the supports at these nodes are removed in all construction stages.

Supports in selected stages

Define different supports for different stages:

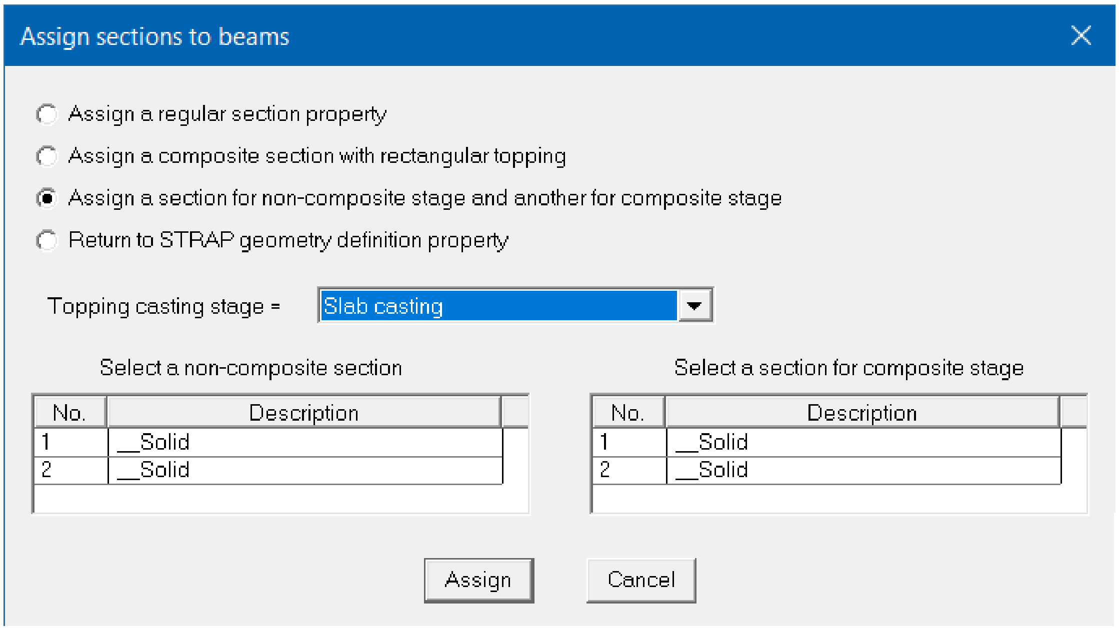
- click on the 'support' column for ***all*** construction stages and set 

or 

.
- Select nodes using the standard node selection options.

11.8 Sections

Assign a different section property to selected spans.
Note: the selection overrides the section specified in the STRAP geometry module.



Assign a regular section property

- Select a *STRAP* section from the list
- Select beams using the standard beam selection option.

Assign a composite section with rectangular property

- select a *STRAP* section from the list.
- define the rectangular topping dimensions.
- select a construction stage; the program assumes that the topping is cast at the start of this stage.
- select beams using the standard beam selection option.

Assign a section for non-composite stage and another for composite stage

- select the non-composite *STRAP* section from the list on the left.
- select the composite *STRAP* section from the list on the right.
- select a construction stage; the program assumes that the topping is cast at the start of this stage (i. e. the section changes).
- select beams using the standard beam selection option.

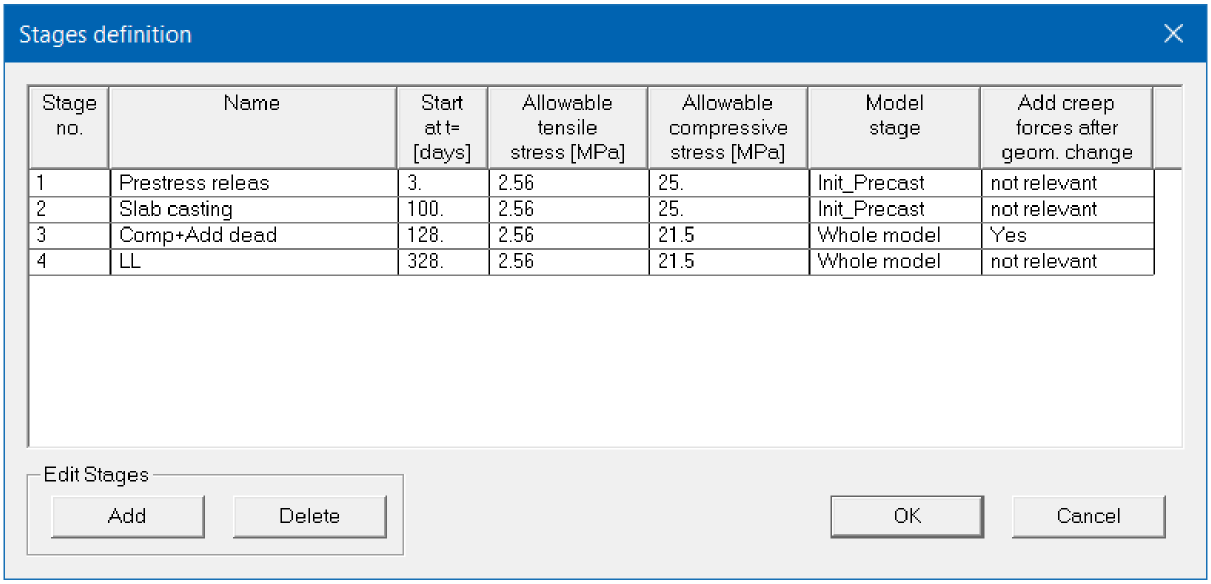
Return to STRAP geometry definition property

All revisions made in this menu are canceled.

- select beams using the standard beam selection option.

11.9 Stages

Define a "Stage table" for models where not all of the cables or beams are prestressed at the same time or where loads are applied at different times.
The stages are defined by the number of days from the start of construction, the allowable stresses and the relevant *STRAP* stage.



Stage no.	-	the stage number
Name	-	the stage name
Start at t=	-	specify the stage start day.
Allowable tensile stress	-	specify the allowable tension stress for a given stage.
Allowable compression stress	-	specify the allowable compression stress for a given stage.
Model stage	-	select the relevant construction stage (defined in geometry) for a given stage.
Add creep forces after geom. change	-	add the calculated forces due to change in schema. These forces are a result of different creep coefficients.

Note:

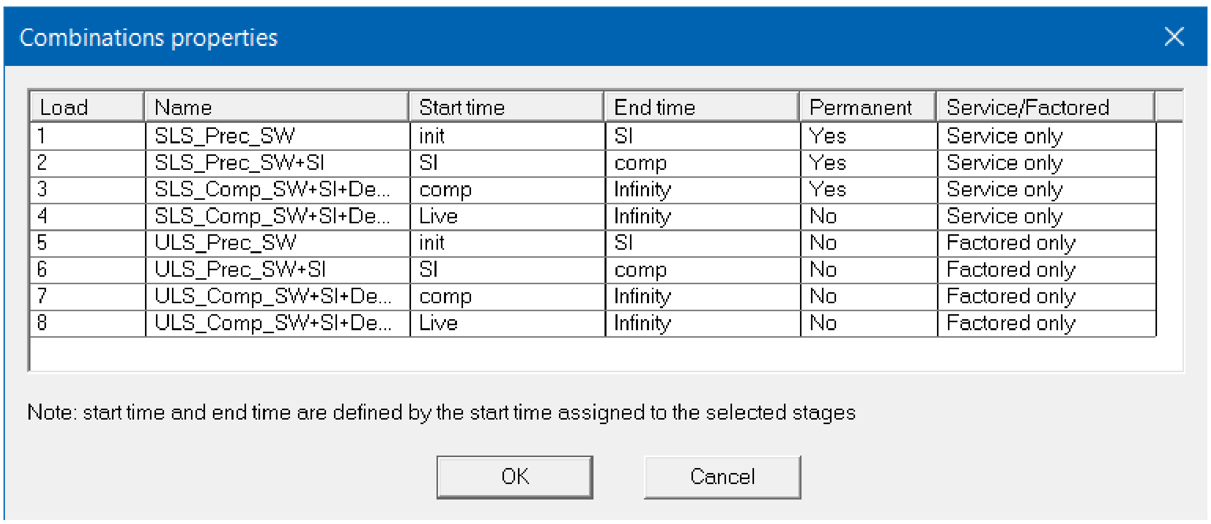
- If several stages were defined in STRAP geometry, then the additional forces from loads and cables that were applied prior to the configuration change may be calculated for relevant stages; specify **YES** in the "**Add creep forces ....**" column. Refer to General - configuration changes / composite 1106.
- click Add to add another stage to the list. The program adds a new line and adds the parameters from the previous line; enter a new title and edit the parameters. Note that each stage may be associated with a STRAP model 'stage'.
- click and highlight a stage, then click Delete to remove a stage from the list.

Refer also to Design assumptions.

11.10 Load table

Expand the load combination table defined in the *STRAP* results module.

- each combination is assigned a 'start' and 'end' time, defined as one of the stages or infinity. The program uses only the relevant combination results when calculating the stresses at a specific stage.
- each combination may be defined as 'permanent' (sustained) for calculation of deflections and losses.
- each combination may be used for only service load calculations (stresses, deflections), only for factored load calculations (ultimate moment, shear) or both.


Combinations properties

Load	Name	Start time	End time	Permanent	Service/Factored
1	SLS_Prec_SW	init	SI	Yes	Service only
2	SLS_Prec_SW+SI	SI	comp	Yes	Service only
3	SLS_Comp_SW+SI+De...	comp	Infinity	Yes	Service only
4	SLS_Comp_SW+SI+De...	Live	Infinity	No	Service only
5	ULS_Prec_SW	init	SI	No	Factored only
6	ULS_Prec_SW+SI	SI	comp	No	Factored only
7	ULS_Comp_SW+SI+De...	comp	Infinity	No	Factored only
8	ULS_Comp_SW+SI+De...	Live	Infinity	No	Factored only

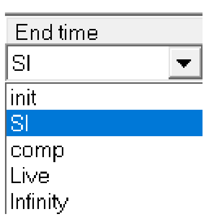
Note: start time and end time are defined by the start time assigned to the selected stages

OK Cancel

Start/end time

Specify the start and end times of each combination; the program uses only the relevant combination results when calculating the stresses at a specific stage.

- click on the combination line
- click on to display the list of stages; for example:



- click and highlight the relevant stage (or infinity)

Permanent

Each combination may be defined as 'permanent' (sustained) for calculation of deflections and losses.

- click on the combination line
- set the checkbox to if the load is permanent. For example:

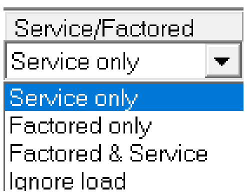
Service/factored

Each combination may be used for only service load calculations (stresses, deflections), only for factored load calculations (ultimate moment, shear) or for all calculations:

- Service only:
  Combinations must be defined with all factors ~1.00.
- Factored only:
  Combinations must be defined with the appropriate factors - 1.4, 1.6, etc.
- Service & factored:
  Combinations must be defined with the appropriate factors - 1.4, 1.6, etc. The program changes all factors to 1.00 when calculating stresses and deflections.
- Load combinations may be 'ignored' by all calculations.

To specify the combination type:

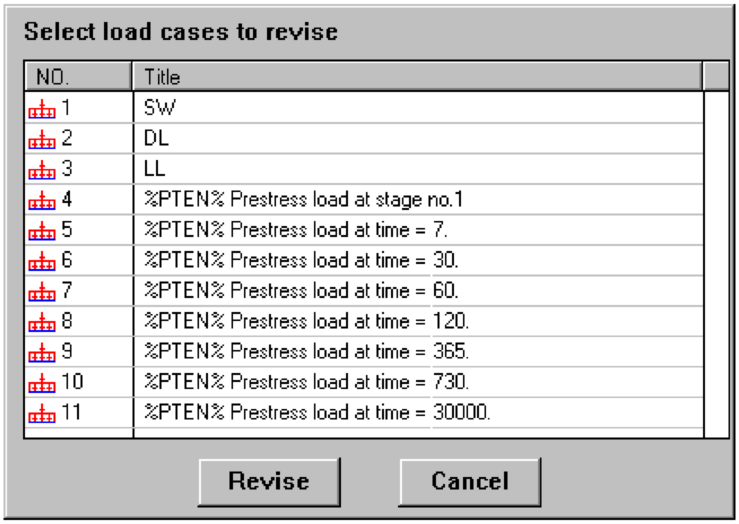
- click on the combination line
- click on to display the list of stages; for example:



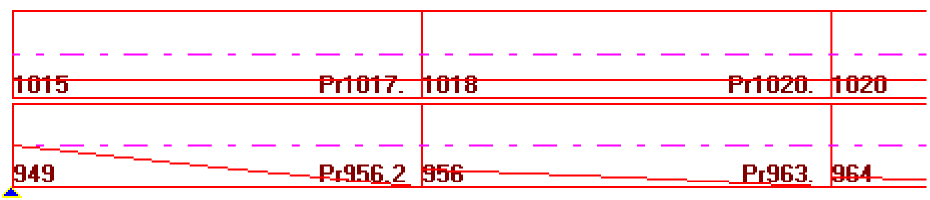
- click and highlight the relevant type.

11.11 Solve

The program creates cable force load cases at all stages where cables are prestressed and at all time steps, including calculated losses at each step.
For example:



The program applies prestressing loads similar to those that may be defined in *STRAP* beam loads. However, the program divides each member into 10 segments because the axial force is generally not equal along the length of the beam because of the losses.

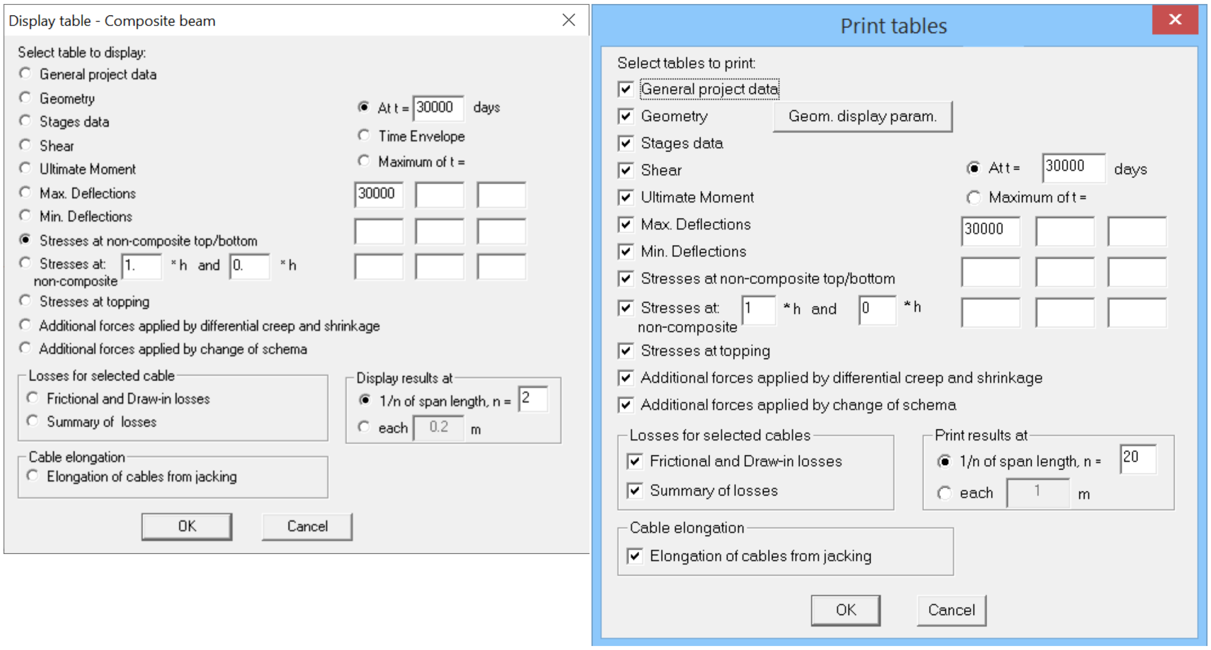


Note:

- Do not modify the %PTEN% load titles; if you select "Solve" again, the program identifies the previous pretension load cases according to the title and erases them before writing the new ones.

Refer also to Design assumptions

11.12 Screen / print - tables



11.12.1 General project data

General project data includes:

- ***Stages***. For example -

STAGES

Name	Start time (days)	Allow. tens. (MPa)	Allow. compr. (MPa)	model stage	add creep forces after schm. cng.
Prestress releas	3	2.56	25.	Init_Precast	not relevant
Slab casting	100	2.56	25.	Init_Precast	not relevant
Comp+Add dead	128	2.56	21.5	Whole model	Yes
LL	328	2.56	21.5	Whole model	not relevant

For more info refer to stages 1160.

- ***Time steps***. . For example -

TIME STEPS IN ADDITION TO THE TIME OF THE STAGES (DAYS)

time = 0

time = 30000

For more info refer to default parameters, time steps 1154

- ***Load combinations***. For example -

LOAD COMBINATIONS

Name	Start time (days)	End time (days)	perman.	type
SLS_Prec_SW	3	100		service
SLS_Prec_SW+SI	100	128		service
SLS_Comp_SW+SI+Dead	128	infinity		service
SLS_Comp_SW+SI+Dead+Live	328	infinity		service
ULS_Prec_SW	3	100		factored
ULS_Prec_SW+SI	100	128		factored

For more info refer to load table 1161

- ***Stand types***. For example -

STRAND TYPES

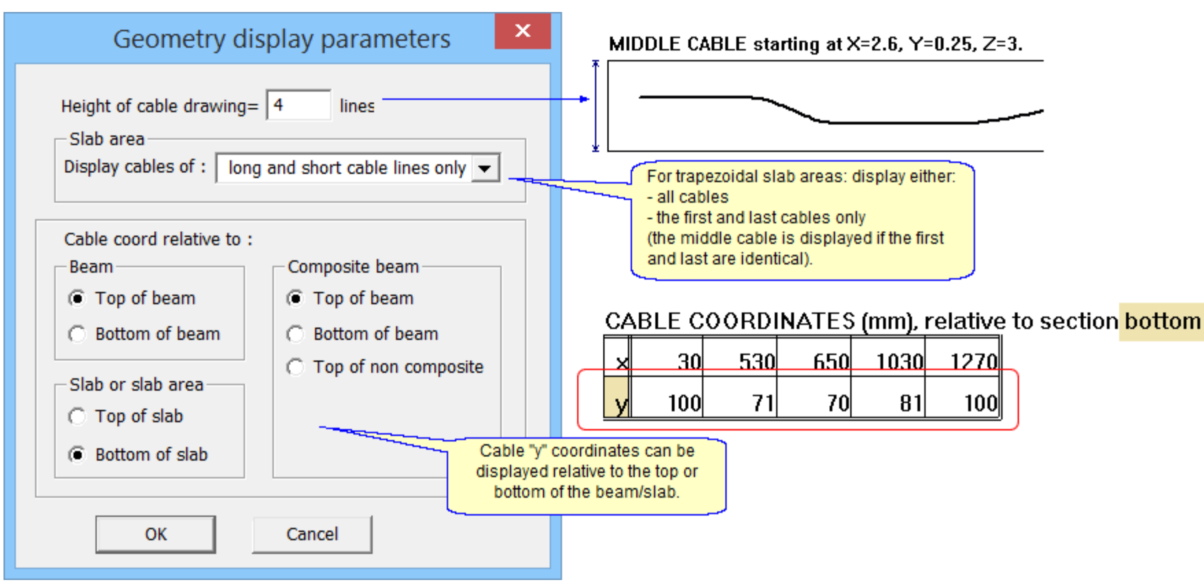
Name	Strand area (mm2)	Max. stress (MPa)	Steel type
T6.4mm(1/4in)	24.5	992.	1720_Flat
T7.9mm(5/16in)	37.3	1280.	1720_Flat
T9.3mm(3/8in)	51.6	1270.	1720_Flat
T9.5mm(3/8in)	54.8	1370.	1860_Flat
T10.8mm(7/16in)	69.7	1270.	1720_Flat
T11.1mm(7/16in)	74.2	1370.	1860_Flat

For more info refer to default parameters, strand types 1154.

11.12.2 Geometry

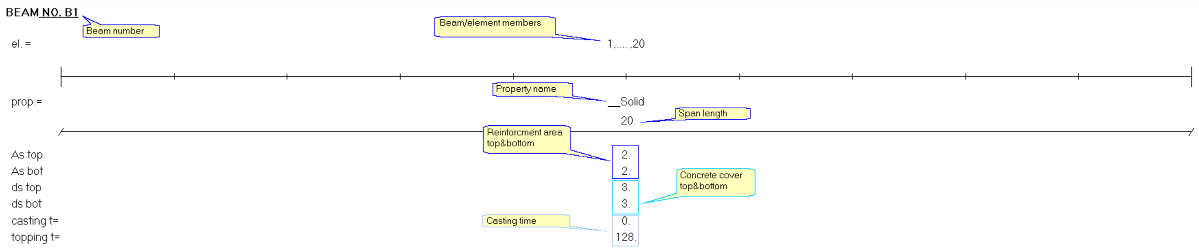
Display "Geometry" table -

Select the parameters for the geometry table display by clicking "Geom.display param.":

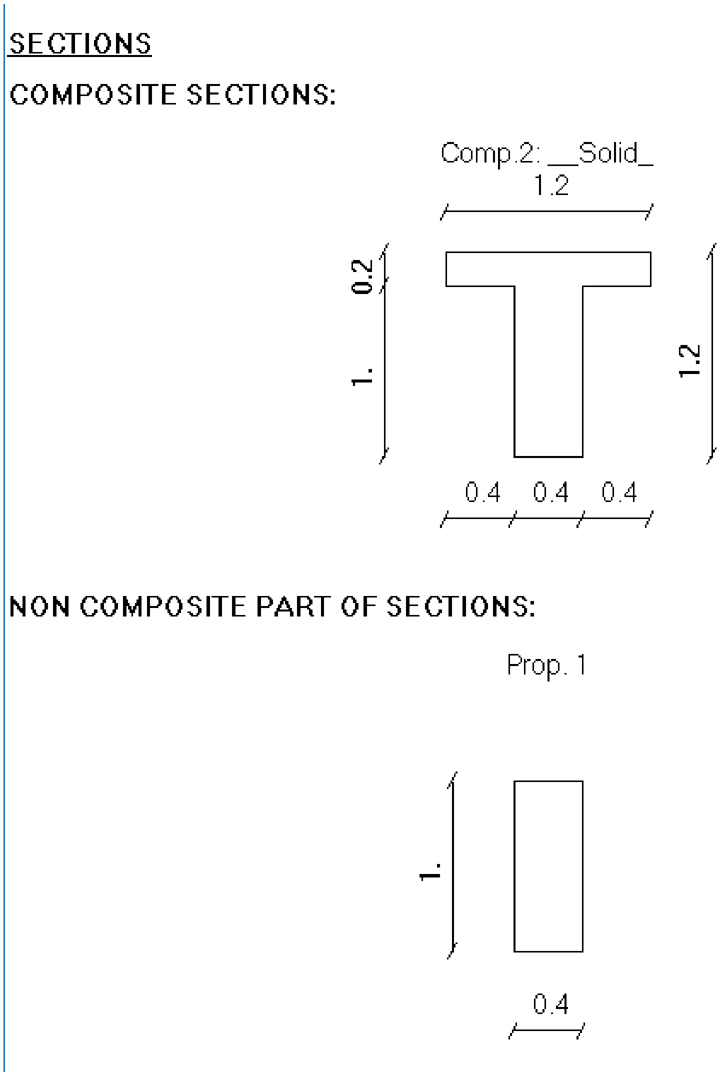


The geometry table includes:

- Beam data, for example -



- Section data, for example -



Parameters for prestress design, for example -

PARAMETERS

Design Code = EuroCode, conc. fc = 50., reinf. fy = 460., shear reinf. fy = 250.

Topping : conc. fc = 50. , allow. tens. stress =0., allow. comp. stress =20.

Reinf. modulus E = 200000., conc. modulus E = 30000., cables modulus E = 195000.

Topping : modulus elasticity of concrete E =30000.

Humidity = 50.%, cement type = normal, temperature = 20.

Calculation methods: Ultimate moment = do not include decompression strain,

Shear = inclined struts method, Deflections = use effective I at each point.

Cable data, for example -

CABLE NO. = 1

No. of strands = 1, Strand type : 7ws12.9mm, % of jacking = 10., Total force of all strands= 1.65[t]

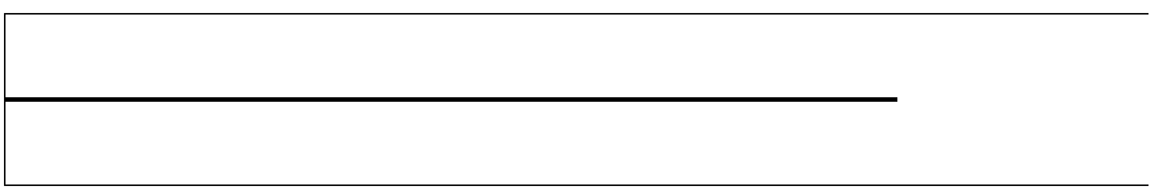
Cable is bonded

JACKING SEQUENCE :

Stage 2: Trans_1

Strands 1-1 jacked from left side to 100.%, Total force of jacked strands =1.65[t]

CABLE GEOMETRY :



CABLE COORDINATES (mm), relative to section top

x	0	500	1000	1500	2000	2500	3000	3500	4000	4500	5000
y	-500	-500	-500	-500	-500	-500	-500	-500	-500	-500	-500

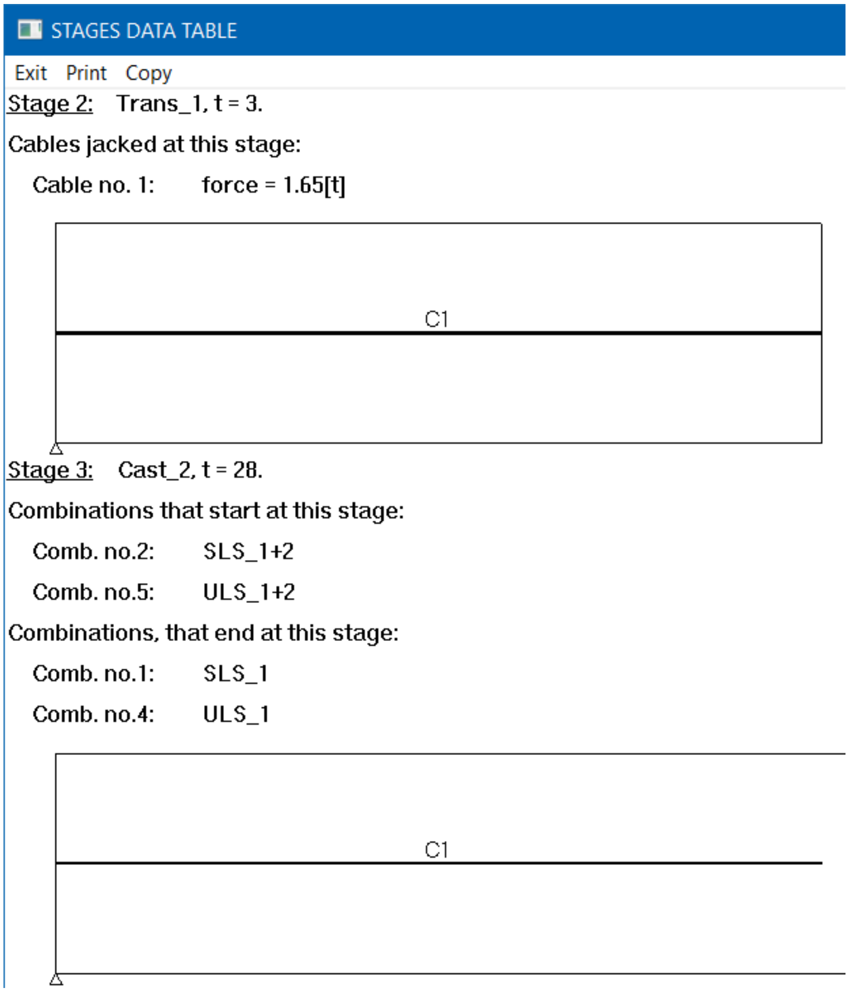
11.12.3 Stage data

Display "Stage data" table -

Display a table showing for each stage:

- the current beam geometry.
- the current cable layout.
- load combination applied at start of stage.
- load combinations that end at this stage.
- cables jacked at this stage.

For example:



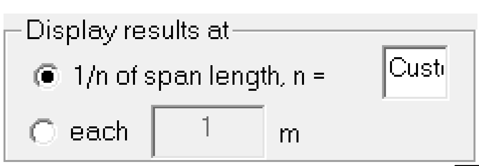
Note: the beam profile is shown only if there is a change in geometry or cables at that stage.

11.12.4 Shear

Display "Shear" table -

- Display the shear results for selected time [days], 

.
- The sections along the beam at which shear results are displayed:



For more info refer to display results at 1181.

The shear results table will displayed, for example -

Shear at t = 30000 (Units: ton meter), beam no. 1

Exit Print Copy

Member no.	Dist.	Comb	V	Vc	Vcmax	M	Mcr	Horiz. Av/s [cm*cm/m]	Av/s [cm*cm/m]
8	7.20	8	-11.05	12.63	148.43	77.0	16.2	5.1	6.9*
	7.40	8	-12.26	12.63	148.42	74.7	16.3	6.1	6.9*
	7.60	8	-13.46	12.62	102.36	72.1	16.3	7.0	6.9*
	7.80	8	-14.67	12.62	102.35	69.3	16.4	7.9	6.9*
	8.00	8	-15.88	12.62	102.35	66.2	16.5	8.9	6.9*
9	8.20	8	-17.09	12.61	102.34	62.9	16.6	9.8	6.9*
	8.40	8	-18.30	12.61	102.34	59.4	16.7	10.7	6.9*
	8.60	8	-19.50	12.60	102.33	55.6	16.9	11.7	6.9*
	8.80	8	-20.71	12.60	102.33	51.6	17.0	12.6	6.9*
	9.00	8	-21.92	12.59	102.32	47.3	17.1	13.5	6.9*
10	9.00	8	-44.32	12.59	102.32	47.3	17.1	30.8	7.9
	9.20	8	-45.53	12.58	102.31	38.4	17.3	31.8	8.1
	9.40	8	-46.74	12.39	102.09	29.1	17.4	32.7	8.3
	9.60	8	-47.94	11.58	101.17	19.7	17.6	33.6	8.5
	9.80	8	-49.15	43.40	145.35	10.0	17.7	34.6	8.7
	10.00	8	-50.36	41.44	143.98	0.0	17.9	35.5	9.0

Where:

Member = member number.

Dist = distance from beam start; the results are displayed at user-defined distances.

Comb = combination with maximum **As/v**.

V = analysis results for factored shear force result. *note: if V>Vcmax the program colors V result in red*.

Vc = shear capacity of the concrete.

Vcmax = maximum allowable shear force.

M = analysis results for factored moment.

Mcr = cracking moment.

Horiz, Av/s = total area of horizontal shear links (stirrups) required (all legs) per unit spacing. Only for composite beams.

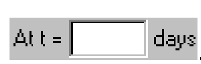
Av/s = total area of shear links (stirrups) required (all legs) per unit spacing. * *minimum reinforcement.*

Note:

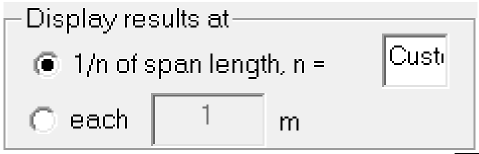
- Symbols may vary according to Code.
- For more info, refer to Design assumptions.

11.12.5 Ultimate / cracking moments

Display "Ultimate moment" table -

- Display the ultimate moment results for selected time [days], 
.

- The sections along the beam at which ultimate moment results are displayed:



For more info refer to display results at 1181.

The ultimate moment table will be displayed, for example -

Ultimate moments and factored moments at t = 30000 (Units: ton meter), ...

Exit Print Copy

Member no.	Dist.	Comb	M	Mcr	Ultimate Moment	x/d	Mult /Mcr	M/Mult
1	0.00	7	0.0	69.6	481.2	0.3447	6.91	0.00
	1.00	8	169.6	327.4	623.6	0.4150	1.90	0.27
	2.00	8	320.8	328.7	617.4	0.4180	1.88	0.52
12	3.00	8	453.7	327.3	621.6	0.4170	1.90	0.73
	4.00	8	568.2	326.5	625.4	0.4170	1.92	0.91
13	5.00	8	664.2	325.9	628.8	0.4170	1.93	1.06
	6.00	8	741.9	325.9	632.2	0.4160	1.94	1.17
14	7.00	8	801.3	326.0	634.8	0.4160	1.95	1.26
	8.00	8	842.2	326.5	637.1	0.4160	1.95	1.32
15	9.00	8	864.7	327.4	639.2	0.4150	1.95	1.35
	10.00	8	868.9	328.8	640.9	0.4150	1.95	1.36

Where:

Member = member number.
Dist = distance from beam start; the results are displayed at user-defined distances.
Comb = combination with maximum **M/Mult**.
M = analysis results for factored moment. *note: if M>Mult the program colors M result in red.*
Mcr = cracking moment.
Mult = ultimate moment capacity of the beam
x/d = ratio between the compression block (x) and the member effective depth (d).
Mult/Mcr = ratio of the Ultimate moment capacity to the Cracking moment.
M/Mult = ratio of the factored moment to the Ultimate moment capacity.

Note:
- Symbols may vary according to Code.
- For more info, refer to Design assumptions.

11.12.6 Stresses

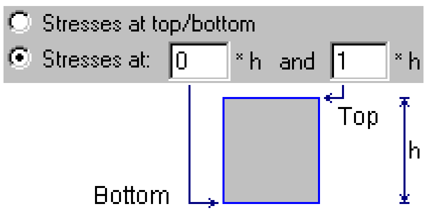
Display "stresses" table -

3 options are available:

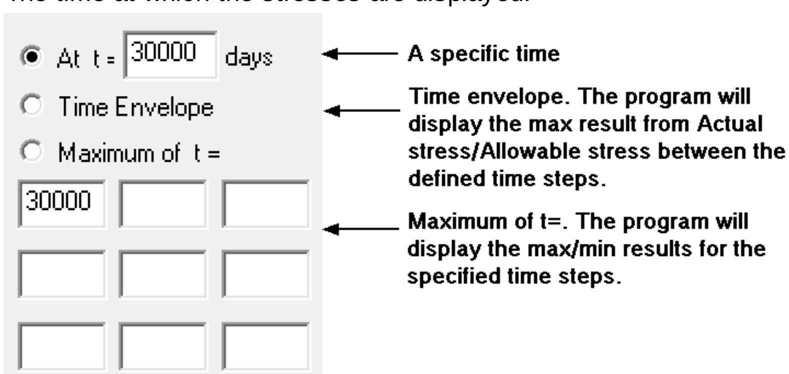
- Stresses at non-composite top/bottom. Displays the calculated stresses for non-composite section's top and bottom fibers.
- Stresses at: (non-composite). Displays the calculated stresses for non-composite section at a specified location along the section depth.
- Stresses at topping. Displays the calculated stresses at the topping.

Specify the following options:

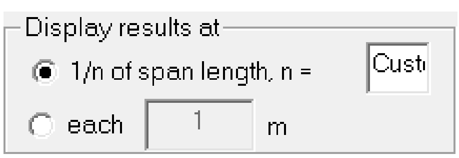
- The levels in the beam cross-section at which the stresses are displayed:



- The time at which the stresses are displayed:



- The sections along the beam at which stresses are displayed:



For more info refer to display results at 1181.

The calculated stresses table will be displayed, for example:

Maximum stresses [MPa] at t = 3 ,40 ,72 ,30000, beam no. 1

Exit Print Copy

Member	Dist.	Maximum				Minimum			
		Top		Bottom		Top		Bottom	
no.	[m]	Comb, t	Stress	Comb, t	Stress	Comb, t	Stress	Comb, t	Stress
1	0.00	1, 30k	-1.41	1, 72+	-3.28	1, 40+	-3.32	1, 30k	-4.23
	0.50	4, 30k	-1.29	1, 40+	1.43	1, 40+	-4.42	4, 30k	-0.85
	1.00	4, 30k	-1.15	1, 3+	6.32	1, 40+	-5.39	4, 40-	3.72
2	1.50	4, 30k	-0.68	1, 3+	6.17	1, 40+	-4.95	4, 40-	2.87
	2.00	4, 30k	-0.33	1, 3+	5.93	1, 40+	-4.72	4, 40-	2.01
3	2.50	4, 30k	-0.03	1, 3+	5.73	1, 40+	-4.51	4, 40-	1.27
	3.00	4, 30k	0.22	1, 3+	5.56	1, 40+	-4.35	4, 40-	0.63
4	3.50	4, 30k	0.43	1, 3+	5.44	1, 40+	-4.22	4, 30k	-0.19
	4.00	4, 72+	0.70	1, 3+	5.34	1, 40+	-4.13	4, 30k	-0.45
5	4.50	4, 72+	0.99	1, 3+	5.29	1, 40+	-4.08	4, 30k	-0.64
	5.00	4, 72+	1.16	1, 3+	5.27	1, 40+	-4.06	4, 30k	-0.76

The results are displayed for maximum and minimum stresses.
For the above example load combinations 3 & 4 are loaded at the same time interval. Stress calculation with combination 4 results in maximum stress at top fiber and minimum stress at bottom fiber.

Where:

Member = member number.
Dist = distance from beam start; the results are displayed at user-defined distances.
Comb,t = combination with maximum/minimum stress. t is the selected time (available for multiple time steps results).
Stress = calculated stress.

Note:
- Stresses are displayed at user-defined distances.
- Compression = positive ; Tension = negative.
- For each combination the actual stress is compared to the allowable stress specified in "Stages 1160"; stresses that exceed the allowable values are displayed in red.

11.12.7 Losses

Display "Losses" table -

- Display the losses results for selected time [days], At t = [ ] days.
- The sections along the beam at which losses results are displayed:

Display results at
1/n of span length, n = Cust
each 1 m

For more info refer to display results at 1181.

- For a specific cable selected by clicking Select cable .

Two tables are available:

- Friction and draw-in losses, for example -

Friction and Draw-in losses table for cable no. 1, at t = 30000, beam no. 1

Exit Print Copy

Parameters: curvature frict.: μ=0.300, wobble frict.: K=0.00300, draw-in: dist=0.006

Point	Coord. [m]		Cable	Accum.	Losses [t]			Resulting
no.	x	y	length	angle	curvature	wobble	draw-in	force [t]
1	0.00	-0.030	0.000	0.000	0.00	0.00	27.50	70.67
2	1.00	-0.354	1.051	0.065	1.90	0.29	23.13	72.86
3	2.00	-0.606	2.083	0.133	3.84	0.56	18.70	75.07
4	3.00	-0.786	3.099	0.203	5.79	0.83	14.27	77.29
5	4.00	-0.894	4.105	0.274	7.74	1.08	9.87	79.49
6	5.00	-0.930	5.106	0.346	9.67	1.32	5.53	81.66
7	6.00	-0.894	6.107	0.417	11.56	1.55	1.30	83.77
8	7.00	-0.786	7.113	0.489	13.38	1.76	0.00	83.03
9	8.00	-0.606	8.129	0.558	15.14	1.97	0.00	81.07
10	9.00	-0.354	9.161	0.626	16.81	2.17	0.00	79.20
11	10.00	-0.030	10.212	0.691	18.38	2.36	0.00	77.43

Where:

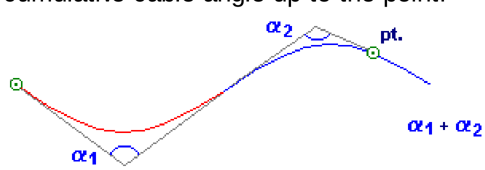
= distance from beam start; the results are displayed at user-defined distances

= vertical coordinate of the prestressing cable, measured from the beam top

= length of cable from the beam start to the point

=

cumulative cable angle up to the point.



= calculated curvature loss.

=

calculated wobble loss.

= for post-tensioned - calculated draw-in loss
for pre-tensioned - calculated transfer length loss.

= The remaining force in the cable after friction and draw-in losses.

Note: Friction and draw-in losses can be displayed only for individual cables.

- Summary of losses, for example -

Summary of tension force losses for all cables, at t = 30000, beam n...

Exit Print Copy

Parameters: curvature frict.: µ=0.300, wobble frict.: K=0.00500, draw-in: dist=0.006

Point no.	Coord. x[m]	Losses [t] friction + draw-in	elastic shorten.	shrinkage	creep	relaxation	Force after losses[t]
1	0.00	73.30	0.00	13.53	13.81	27.93	149.89
2	1.00	67.95	0.00	13.90	6.37	28.76	161.47
3	2.00	62.58	0.00	13.60	4.44	28.08	169.76
4	3.00	57.23	0.00	13.09	6.05	26.94	175.16
5	4.00	51.94	0.00	12.68	8.40	26.05	179.39
6	5.00	46.72	0.00	12.54	9.75	25.73	183.72
7	6.00	41.62	0.00	12.68	9.39	26.05	188.72
8	7.00	41.59	0.00	13.09	7.14	26.94	189.70
9	8.00	42.83	0.00	13.60	5.15	28.08	188.80
10	9.00	46.01	0.00	13.90	6.97	28.76	182.81
11	10.00	50.42	0.00	13.53	15.20	27.93	171.39

Where:

Corrd. x	=	distance from beam start; the results are displayed at user-defined distances
Friction+draw-in/ Dev'length	=	for post-tension - total calculated losses due to Friction and draw-in. for Pre-tensioned - calculated development length loss.
Elastic shorten.	=	calculated elastic shortening loss.
Shrinkage	=	calculated cable loss due to shrinkage.
Creep	=	calculated cable loss due to creep.
Relaxation	=	calculated cable loss due to relaxation of the prestressing steel.
Force after losses	=	the remaining force in the cable after all losses.

Note:

- symbols may vary according to Code.
- refer to Design assumptions.

11.12.8 Deflections

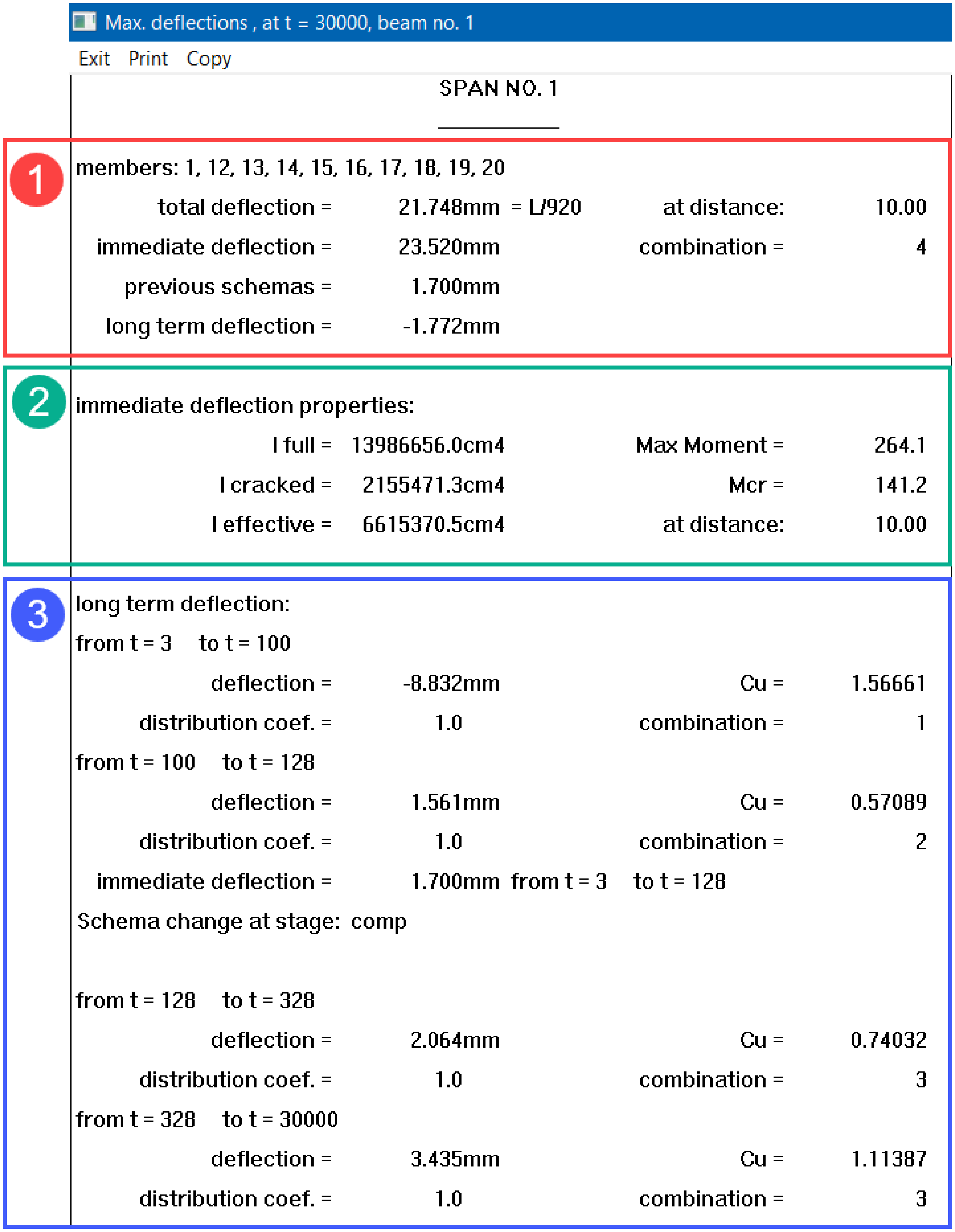
Display "Deflections" table -

2 options are available:

- Maximum deflections: maximum downwards deflection.
- Minimum deflections: maximum upwards deflection.

Specify the time At t = days for the deflections display.

The calculated deflections table will be displayed, for example:



The table subdivides into 3 section. Where:

1) Deflections results summary -

- Members - All the beam members numbers of a span.
- Total deflection - The total calculated deflection. The total deflection is equal to

immediate deflection + long term deflection

- Previous schemes - If a change in scheme is defined, The program displays the immediate deflection calculated for the previous scheme.
- Long term deflection - The calculated long term deflection. The total long term deflection is equal to the sum of of the calculated long term deflection of each time interval (Section 3).
- At distance - The distance from the beam start for which the deflection results are displayed.
- Combination - The load combination that is used for the deflection calculations

2) Immediate deflections properties -
- I full - Uncracked moment of inertia.
- I cracked - Cracked moment of inertia.
- I effective - The effective moment of inertia for deflection calculation.
- Max moment - Maximum moment along the span from analysis results.
- Mcr - Cracking moment
- At distance - The distance from the beam start for Max moment and Mcr.

3) Long term deflections
  - From t= to t= - Time interval period
  - Deflection - The calculated long term deflection for time interval.
  - Cu - The calculated creep coefficient for time interval.
- Combination - Load combination for every time interval.

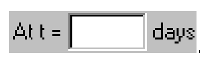
Note:
- symbols may vary according to Code.

Refer to Design assumptions.

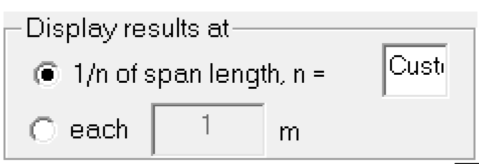
11.12.9 Additional forces/moments - Creep

Display the "Additional applied forces by differential creep and shrinkage" -

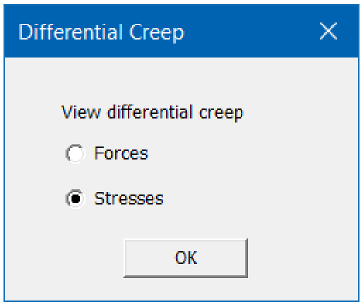
Note: This table is available only if differential creep and shrinkage calculation are being considered. For more info refer to composite 1156 in default parameters.

- Display the forces applied by differential creep and shrinkage 

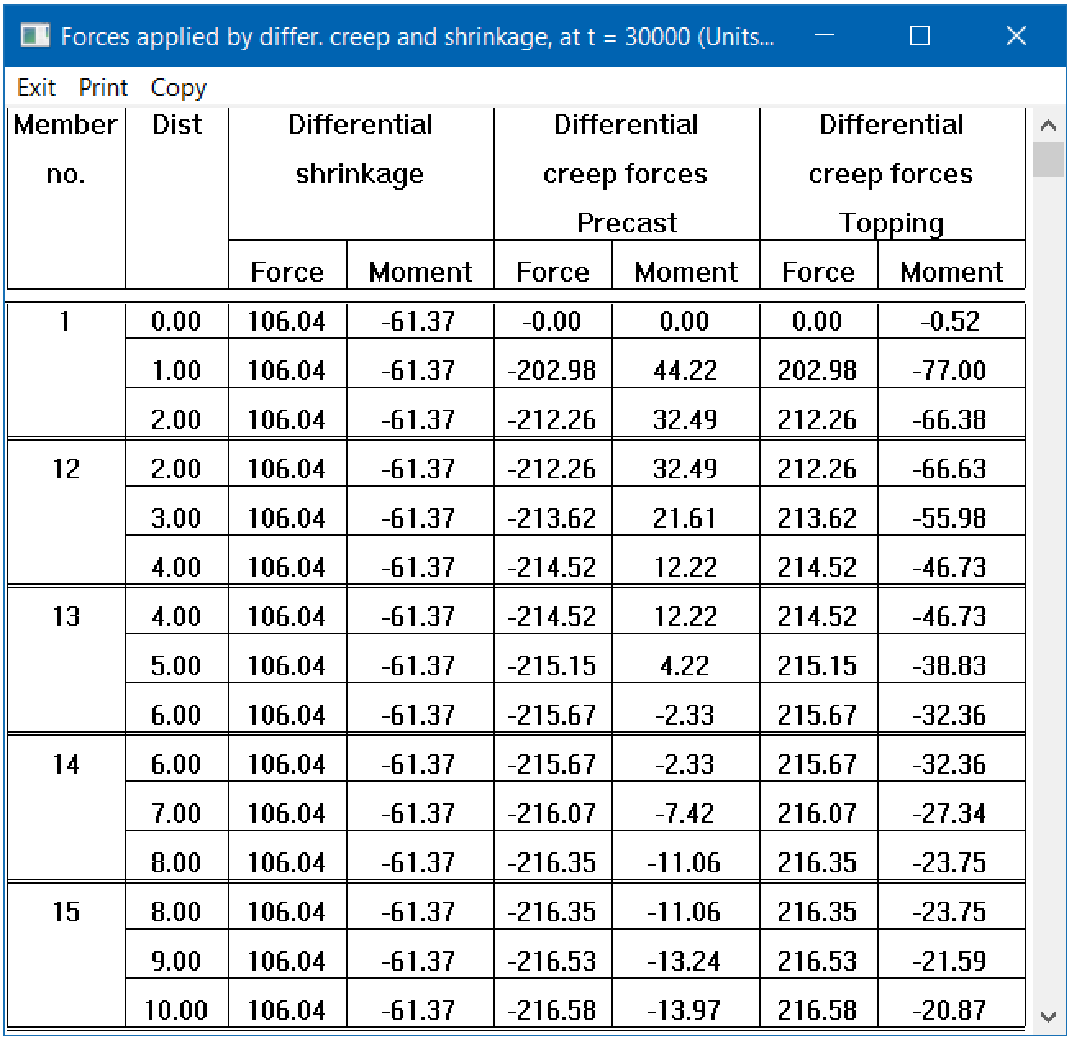
.
- The sections along the beam at which losses results are displayed:



For more info refer to display results at 1181.
- For differential creep, 2 option are available:
  Forces or stresses -



The calculated forces/stresses due to differential creep and shrinkage will be displayed, for example -



Member no.	Dist	Differential shrinkage		Differential creep forces Precast		Differential creep forces Topping	
		Force	Moment	Force	Moment	Force	Moment
1	0.00	106.04	-61.37	-0.00	0.00	0.00	-0.52
	1.00	106.04	-61.37	-202.98	44.22	202.98	-77.00
	2.00	106.04	-61.37	-212.26	32.49	212.26	-66.38
12	2.00	106.04	-61.37	-212.26	32.49	212.26	-66.63
	3.00	106.04	-61.37	-213.62	21.61	213.62	-55.98
	4.00	106.04	-61.37	-214.52	12.22	214.52	-46.73
13	4.00	106.04	-61.37	-214.52	12.22	214.52	-46.73
	5.00	106.04	-61.37	-215.15	4.22	215.15	-38.83
	6.00	106.04	-61.37	-215.67	-2.33	215.67	-32.36
14	6.00	106.04	-61.37	-215.67	-2.33	215.67	-32.36
	7.00	106.04	-61.37	-216.07	-7.42	216.07	-27.34
	8.00	106.04	-61.37	-216.35	-11.06	216.35	-23.75
15	8.00	106.04	-61.37	-216.35	-11.06	216.35	-23.75
	9.00	106.04	-61.37	-216.53	-13.24	216.53	-21.59
	10.00	106.04	-61.37	-216.58	-13.97	216.58	-20.87

where:

Member = member number.

Dist = distance from beam start; the results are displayed at user-defined distances.

Differential Shrinkage = Force - - The axial force due to differential shrinkage resulting from the different beam/topping concrete ages.

Moment - The moment due to differential shrinkage resulting from the different beam/topping concrete ages.

Differential creep forces Precast/Topping =	Force	-	The axial force due to differential creep resulting from the different beam/topping creep coefficients.
	Moment	-	the moment due to differential creep resulting from the different beam/topping creep coefficients.

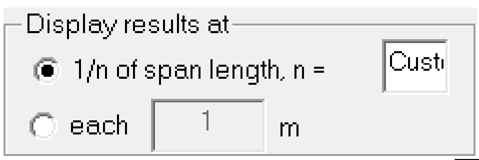
Note:

- Additional forces due to the difference in creep, caused by the difference in stresses in the beam and the topping, can be displayed/printed in the Additional moments - configuration change 1179 table.
- For more info refer to General - configuration change / composite 1106.

11.12.1(Additional forces/moments - Change of schema

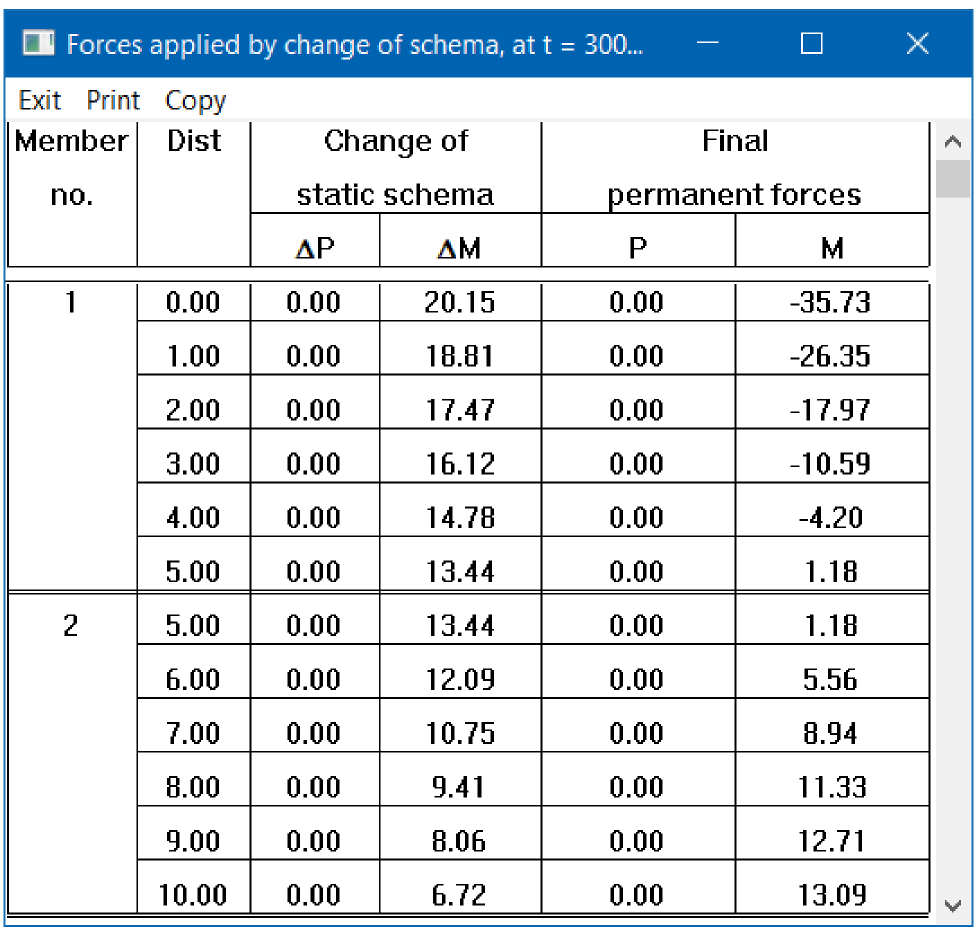
Display the "Additional forces due to change of schema" -

- Display the additional forces due to change of schema 
.
- The sections along the beam at which losses results are displayed:



For more info refer to display results at 1181.

The calculated additional forces due to change in schema will be displayed, for example -



Member no.	Dist	Change of static schema ΔP	Change of static schema ΔM	Final permanent forces P	Final permanent forces M
1	0.00	0.00	20.15	0.00	-35.73
	1.00	0.00	18.81	0.00	-26.35
	2.00	0.00	17.47	0.00	-17.97
	3.00	0.00	16.12	0.00	-10.59
	4.00	0.00	14.78	0.00	-4.20
	5.00	0.00	13.44	0.00	1.18
2	5.00	0.00	13.44	0.00	1.18
	6.00	0.00	12.09	0.00	5.56
	7.00	0.00	10.75	0.00	8.94
	8.00	0.00	9.41	0.00	11.33
	9.00	0.00	8.06	0.00	12.71
	10.00	0.00	6.72	0.00	13.09

Member = member number.

Dist = distance from beam start; the results are displayed at user-defined distances.

Change of static schema = ΔP,ΔM are the equivalent force and moment applied to beam after the configuration change. They are generated by the restraint creep from loads and cables forces applied prior to the configuration change.

Final permanent forces = P,M are the final force and moment, including ΔP and ΔM.

Note:

- If there are several configuration changes prior to time 't', the table displays the sum of the forces resulting from each change.
- refer to General - configuration change / composite 1106.

11.12.11 Cable elongation

Display the "Cable elongation" table -

- Display the cable elongation At t = [ ] days.

The cable elongation table will be displayed, for example -

Elongation of cables from jacking , beam no. 1

Exit Print Copy

Cable no.	Jacking time	Cab. len. before jacking	Jacking side	Jacking strands	Jacking force percent	Elongation at jacking time
	day	[cm]			%	[mm]
1	20.0	1022.1	Left	7	100	57
2	20.0	1005.9	Left	7	100	60

Where:

Cable no. = Cable number.

Jacking time = The cable jacking time. This value is set in jacking sequence under "transfer stage". For more info refer to jacking sequence 1136.

Cable length before jacking = The cable length prior to jacking. Calculated from cable geometry.

Jacking side = The jacking side left/right or both. This parameter is set in jacking sequence under "Jacking side". For more info refer to jacking sequence 1136.

Jacking strands = The number of strands jacked at the "jacking time".

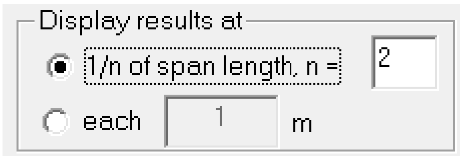
Jacking force percent = The jacking force percentage at the "jacking time".
This value is set in jacking sequence under "Prestress [%]". For more info refer

to [jacking sequence](1136).

Elongation at jacking time = The calculated cable elongation at "jacking time"

11.12.12 Display results at ...

Results along the span can be displayed at user defined intervals:



Select one of the following:

1/n of span: n=2 - results will be displayed at three points: start, middle and end.
n=3 - results will be displayed at four points: start, ($^1/_3$)L, ($^2/_3$)L and end.
etc.

each "n" m: results are displayed at the start, end and at every interval along the length.

11.13 Design assumptions

Select one of the following codes:

AASHTO
ACI318
BS5400
BS8110
CSA A23,3
Eurocode 2
IRC

Other codes

Part XII

Appendix

12 Appendix

12.1 General

Refer to:

List of Enhancements
Installation notes 1242
Print the manual 1243

12.1.1 Getting Started

This section is intended mainly for engineers who have no previous experience in the use of computers for structural analysis or who are unfamiliar with the finite element method.

The engineer must prepare a computer model of the structure. The model consists of a series of elements joined at points called nodes (or joints).

STRAP uses two types of elements:

Beam elements:
Beams are one dimensional elements that are used to model structural members that can be modelled by a line - beams, columns, bars, etc.

Beam elements always give accurate results, i.e., if a model naturally consisting of beam elements (e.g., a skeletal frame structure) is analysed by ***STRAP***, the results are identical to those calculated manually using any exact method. Similarly, the method of creating the ***STRAP*** model for such structures is usually obvious - each beam or column is represented by a single beam element.

Finite Elements:
Finite elements are two-dimensional elements that are used to model surfaces such as plates, shells and walls. They may be either triangular or quadrilateral in shape.

Finite elements, on the other hand, give inherently inaccurate results (the degree of inaccuracy is usually acceptable by most engineering standards), for the following reasons:

- The elements should be connected along their common boundaries, but in the finite element model they are connected only at their common nodes. Thus there is a relaxation of continuity along the boundary (although the mathematical development of the elements ensures satisfaction of some, but not all, of the boundary continuity requirements).
- The mathematical development of the elements assumes a linear stress distribution through the element. In reality, the distribution is usually more parabolic.

In a continuum structure such as a flat plate there is no natural subdivision of elements, so the structure has to be artificially divided. It is obvious that as the continuum is divided into a finer mesh (smaller elements), the degree of discontinuity is lessened, the stress distribution approaches linearity and the overall accuracy of the solution improves.

On the other hand, increasing the number of elements in the model increases the solution time and increases the size of the files required to store the input data and results.

Selection of the Computer Model

The preparation of the model for two-dimensional frames, grids or trusses is usually very simple, as each beam and column is represented by a single line element.

More complicated structures - space frames or structures with area elements - require more thought and good engineering judgment in the preparation of the model. Typically, structural members can be represented by either area elements or by line elements and the engineer must choose between them based on his experience.

Remember:

STRAP calculates numerically accurate results for the defined model.

It is the engineer's responsibility to:

- ***define a model that correctly represents the structure***
- ***thoroughly check for illogical or inconsistent results***

Example 1:

A square concrete plate of uniform thickness, simply supported on all edges. The plate should obviously be modelled by a regular pattern of quadrilateral elements. How many elements are required for a sufficiently accurate solution ?

The following table gives results for the centre deflection of the plate : (10x10, 0.2 thick, concrete):

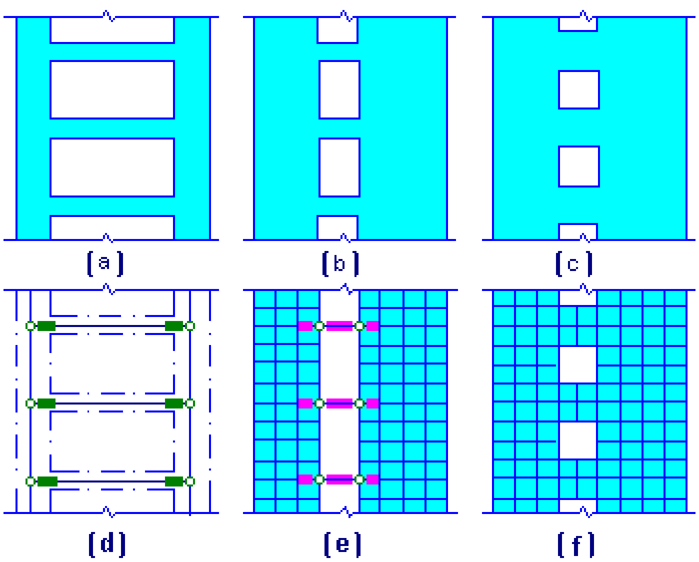
Vertical deflection:

No. of elements	***% error***
100 (10x10)	0.30
64 (8x8)	0.50
36 (6x6)	0.90
16 (4x4)	2.00
4 (2x2)	6.20

There is no obvious advantage gained from defining the model with more than 36 elements.

Example 2:

Consider the three shear wall structures in Figures (a) to (c). The walls are identical except for the size of the openings.



Figures (d) to (f) show three <u>possible</u> computer models for the corresponding shear walls (a rough mesh has been used for clarity):

- model (a): has ***relatively*** narrow walls and beams and so can be modeled entirely by beam elements as shown in (d). Note the 'rigid offsets' in the lintel beams.
- model (b) : has wide walls and shallow beams and so can be modeled by the combination of finite elements and beam elements as shown in (e). The lintel beams are extended into the wall for continuity.
- model (c): has wide walls and deep beams and should be modeled entirely by finite elements as shown in (f).

It is important to stress the following points:
- **all** of the models are inaccurate to some degree : Model (a) uses beam elements to model the wall. Model (b) and (c) use finite elements and so the degree of accuracy is dependant on the number of elements used (see below).
- there is no sharp boundary between beams that are considered 'deep' or 'shallow', i.e., it will not always be obvious which model is the most appropriate.
- As opposed to example (a), there are no exact solutions available that the computer results can be compared to . Therefore, good engineering judgment is required to select the correct model.

In summary:
- the model selected must represent the actual structural behaviour of the structure.
- accuracy increases as the number of elements increases but both solution time and file size also increase.

Thus, the selection of the model and the division of the model into elements is often a tradeoff between:
- more exact modeling of the structure vs. general simplicity of the model
- increased accuracy vs. reduced solution time and file size.

The following are guidelines for constructing an accurate finite element model:

- ***Use quadrilateral elements:***

  In general, try to use quadrilateral elements instead of triangular elements as they give more accurate results.

  Remember that the four corners of a quadrilateral element should all lie on the same plane. If this is not possible, use two triangular elements in place of each quadrilateral.

- ***Element shape:***

  Quadrilateral Elements:
  The greatest accuracy is achieved with a square - 1:1 - element. Elements with a base/height ratio up to 1:2 give good results, but elements with a ratio of 1:5 are unreliable.

  Try to use rectangular shaped elements whenever possible, If not, the internal angles should not vary greatly from 90°. Angles of 30° or 150° greatly reduce accuracy. Elements with convex angles should never be used.

  Triangular Elements:
  Equilateral triangles produce the most accurate results.

- ***Mesh Density:***

  The mesh density need not be constant throughout the model. The program assumes a linear result distribution through the element. If the actual result through the elements is not linear but parabolic, for example, it is obvious that there will be a decrease in the accuracy. In a fine mesh, the result diagram through any one element will always be approximately linear.

  Increase the number of elements where there is a greater rate of change in the internal forces. For example, around supports (where bending moments increase sharply), openings and large concentrated loads.

To decrease the number of elements:
Use a rough mesh in areas where relatively low results are expected. Remember that the connection to adjacent elements is through the element end nodes only and so nodes located along an edge of an element between end nodes are ineffective. Use triangular or trapezoidal shaped elements to step between rough and fine quadrilateral meshes.

If you have doubts as to the accuracy of the results in a particular area of the model, rerun the problem with a finer mesh in that area and compare results. The results converge to the exact solution as the mesh becomes more refined.

- ***Models with axes of symmetry:***
  Large symmetric structures can be modeled by defining only one half (or a quarter) of the model. Note that the symmetry must be present in **both** the geometry and loading diagrams.

  Care must be taken to define the correct boundary restraints along the line of symmetry. An error in defining the proper restraints will lead to incorrect results.

Inputting the Model

The program continuously displays the model graphically and updates the display automatically after every input command, thus enabling the user to instantaneously check the accuracy of the input.

STRAP allows you to use one of three input modes:

Graphic Mode:

The model is defined by moving the crosshair using the mouse/arrow keys to identify node locations, define elements, assign properties, loads, etc. All parameters such as current crosshair coordinate are displayed at the bottom of the screen and are updated continuously. Only a limited number of parameters need be typed in.

Nodes and elements are numbered automatically by the program when generated. Numbering is always consecutive, unless specified otherwise. The numbering may be revised by the user. The final numbering pattern should be one that will cause the results to be printed in a logical sequence.

Command Mode:

The model is defined by typing commands in standard format. The geometry is displayed simultaneously on the screen and is updated after every command.

The numbers of all nodes and elements are specified by the user. The system of numbering should be such that a minimum number of input commands are required. This varies from structure to structure, and is learned mainly from experience. In general, an ordered numbering pattern is recommended.

When numbering the model, note the following points:
- The program contains powerful commands for generating the definition of large groups of nodes and elements; these commands require an orderly numbering pattern.
- Numbering does not have to be consecutive.
- Nodes which are not connected to the model are ignored.
- Nodes may be located anywhere along a beam element.
- Node numbering and element numbering are independent. The structure may contain nodes 1,2,3,4.... as well as elements 1,2,3,4....
- If a structure consists of both beam and area elements, the same element number may not be given to an element of each type.
- Solution time is not dependant on the numbering.

In all cases, it is recommended that the user prepare a sketch showing all numbering before defining the model.

Batch Mode:

Similar to Command Mode, except that the data is entered in a data file external to ***STRAP*** using any editor program.

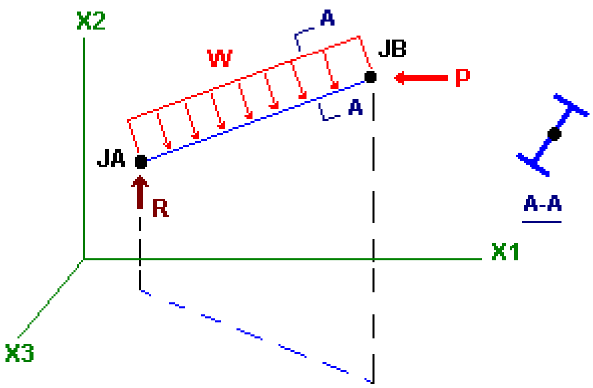
12.1.2 Coordinate systems

Coordinate reference frames are required to uniquely describe the position of a structure in space, the direction of applied loads and the direction of computed reactions, displacements, forces etc. In addition, the coordinate systems are required to reference other structural information such as element properties.

STRAP uses Cartesian reference frames. An auxiliary cylindrical system may be used for node coordinate definition.

Two types of reference systems are used in ***STRAP***. They are:
- the global coordinate system1189, denoted by **X1, X2, X3** (uppercase)
- the local element coordinate system1189, denoted by **x1, x2, x3** (lowercase).

In the following figure a beam element is located in space between nodes JA and JB. The location of the nodes are defined according to the **global** coordinates, i.e. their coordinates relative to the global axes X1,X2,X3.



The horizontal load **P** at node JB and the support reaction **R** at node JA should be referenced to the global coordinate system. But it is difficult to define the distributed load **w** on the beam relative to the global axes and obviously a method is required to define the section orientation. It is also apparent that results such as bending moments must be referenced to different directions for each beam.

Consequently, the beam is provided with a ***local*** axis system x1,x2,x3; section orientation, direction of loads and results are always relative to this local system.

Note that ***each*** element has its own unique local system independent of the local systems of the other elements in the model.

The X1, X2 and X3 axes (or x1, x2 and x3 axes) are always perpendicular to each other and the positive direction of the axes are specified by the right-hand rule1195. It is obvious that if the directions of any two of the axes are known, then the direction of the third axis is easily determined.

The directions of loads, forces, moments and stresses are referenced to the global or local axes by stansard sign conventions.

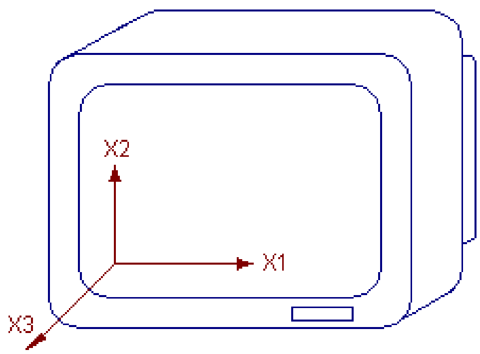
12.1.2.1 Global coordinate system

The geometry of the structure, joint loads and displacements, reactions and certain loads are referenced to the global coordinate system. The orientation of the structure with respect to the system is arbitrary and is implied by the engineer by node coordinate input. Generally, one or more global axes are selected to be parallel to one or more characteristic directions of the structure.

The default convention is:

X1 = horizontal axis on screen
X2 = vertical axis on screen
X3 = axis perpendicular to the screen and pointing towards the user

Plane structures are always defined on the X1-X2 plane.



12.1.2.2 Local coordinate systems

Each element of a structure modeled by ***STRAP*** has a local reference frame associated with it. Element properties, certain loads, member end and internal forces are referred to this local coordinate system.

The local axes directions are automatically specified by the program according to default conventions when the elements are defined, but may be revised by the user.

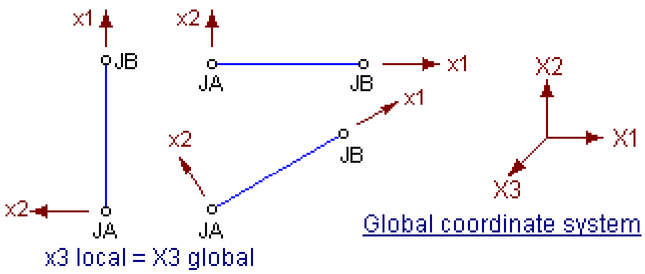
12.1.2.2.1 Beams

The directions of the local axis determine:
- the orientation of the section major and minor axes, which are ***always*** aligned with the x2, x3 axes
- the direction of the beam loads, when defined parallel to the x2, x3 axes

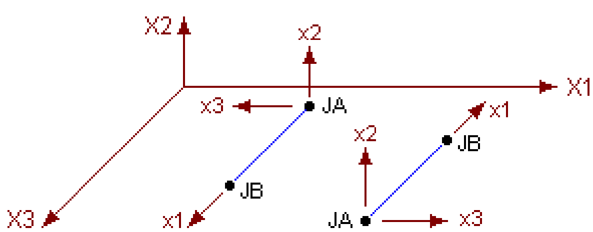
Default Conventions:

Plane models	
Axis	Direction
x1	**Always** coincides with the axis of the beam; the positive direction is from the start node (JA in the geometry tables) to the end node (JB).

x2	As x1 and x3 are known, x2 is determined by the right-hand rule 1195.
x3	Always parallel to the global X3 axis.
Note: For plane models the default convention is always satisfactory and there is no reason to modify the axis directions.	



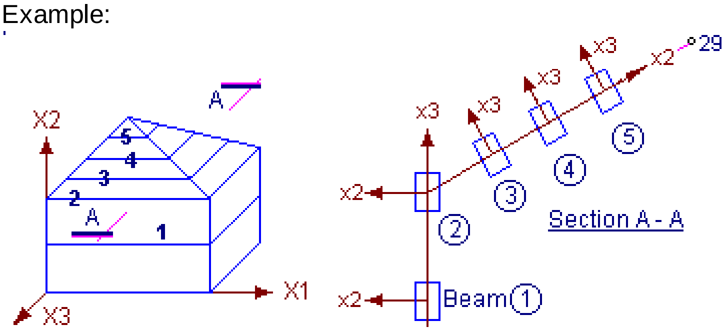
Space models		
Axis	Direction	
	General case: x1 ***not*** parallel to X3	Special case: x1 parallel to X3
x1	Always coincides with the axis of the beam; the positive direction is from the start node (JA) to the end node (JB).	As in 'General case'
x2	As x1 and x3 are known, x2 is determined by the right-hand rule 1195.	***Always*** parallel to the global X2 axis
x3	Perpendicular to x1 and lies on the plane formed by the beam and the global X3 axis. Of the two possible directions, the one with the smaller angle between x3 and X3 is chosen.	As x1 and x2 are known, x3 is determined by the right-hand rule 1195.
If the angle between the local x1 axis and the global X3 axis is greater than 0.006°, the axes are assumed not parallel. Angles of this magnitude can results from computer inaccuracy so the local x3 axis of all beams parallel to X3 should be specifically defined.		



User-defined local axis directions:

Options are available for aligning the local x2 or x3 axes with an existing node or any user-defined plane. Refer to Beams - local axes

Example:



define the local axes of beams 1 to 5 as shown.

- beams 1 and 2 : specify that the local x2 axis is parallel to the global X1-X3 plane.
- beams 3,4 and 5 : specify the local x2 axes as pointing in the direction of node 29.

The x3 axes are determined by the program according to the right-hand rule 1195.

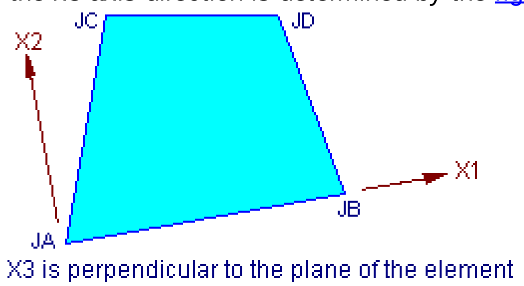
12.1.2.2.2 Quad elements

Each two-dimensional finite element has a local coordinate system associated with it.

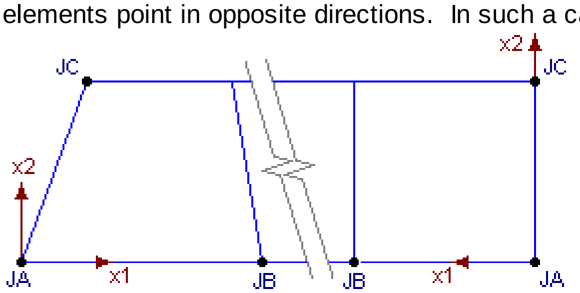
The local x1 and x2 axes ***always*** lie in the plane of the element and x3 is ***always*** perpendicular to the element.

The directions of the local axes are determined by the location of the element corner nodes.

- the local x1 axis lies along the edge of the element formed by nodes JA and JB and is positive in the direction of JB, where JA and JB are the first two corner nodes defined by the user.
- x2 is perpendicular to x1 and points in the general direction of JC, the third node defined.
- the x3 axis direction is determined by the right-hand rule 1195



The following figure shows a situation that can easily occur; the x1 axes of the adjacent elements point in opposite directions while the x2 axes point in the same direction; therefore the x3 axes of these elements point in opposite directions. In such a case, the sign of the results is opposite.



When elements are defined in the Graphic Mode, the program automatically ensures uniformity in the local x3 axis direction for adjacent elements in order to prevent confusion in the results. ***The +x3 direction always points in the general positive direction of the global +X3 axis*** (except for the special cases listed below). The program reverses the x1 direction if necessary by interchanging the order of first two nodes.

To summarize the local axis selection in the Graphic Mode:

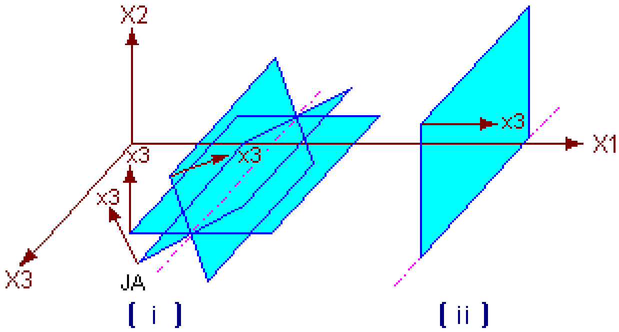
- the local x1 axis lies along the edge of the element formed by nodes JA and JB, where:
  - if a single element is defined, JA and JB are the first two nodes selected by the user.
  - if a surface of elements is defined, JA-JB are the nodes on the edge most parallel and closest to the base line.
  - if a mesh of elements is defined, JA-JB are the nodes on the edge most parallel to global X1. If X1 is perpendicular to the element, then JA-JB are the nodes on the edge most parallel to global X2.
- the +x2 axis lies in the element plane perpendicular to x1 and points in the direction of the other

nodes.

- the **direction** of x1 is from JA to JB. The program determines the direction of the +x3 axis using the right-hand rule 1195. If +x3 does not point in the direction closest to the global +X3 axis, the program interchanges the JA and JB nodes.

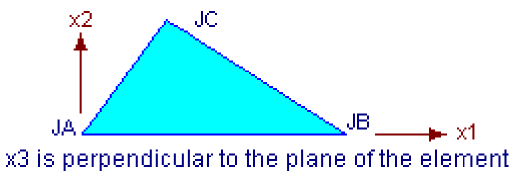
Special cases:

i. the element plane is parallel to the X3 axis: +x3 points in the direction closest to the global +X2 axis.
ii. the element lies parallel to the X2-X3 plane: +x3 points in the direction closest to the global +X1 axis.



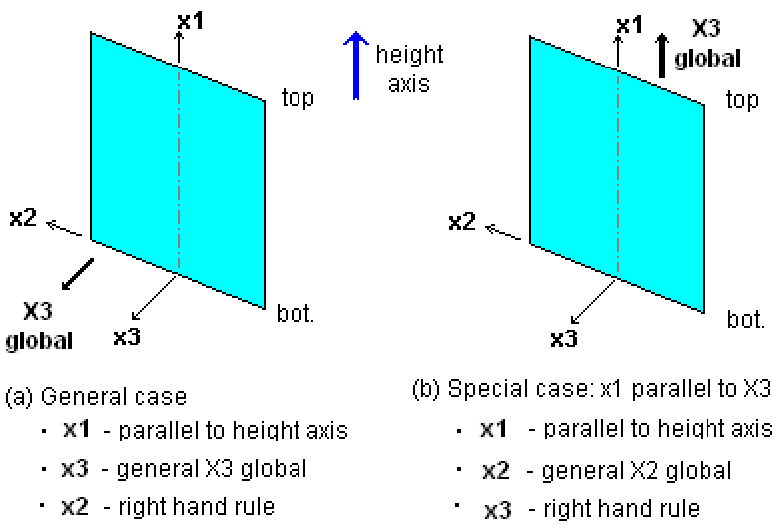
12.1.2.2.3 Triangular elements

The definition of local axes is similar to that for quadrilateral elements 1191.



12.1.2.2.4 Wall elements

The default local coordinate system for wall element segments is identical to the default system for beams 1189. The program assumes that the wall local x1 axis is parallel to the "height axis" specified when defining the wall and points in the positive direction of the height axis:

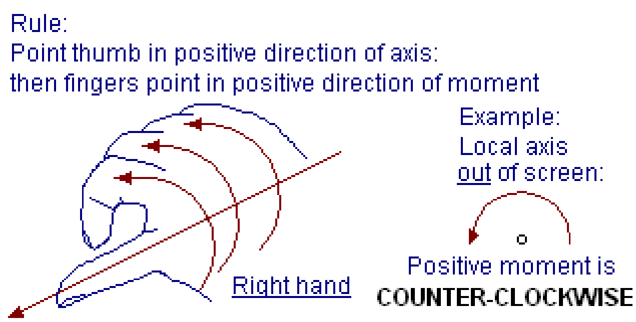


The default local axes cannot be revised.

12.1.2.3 Sign conventions

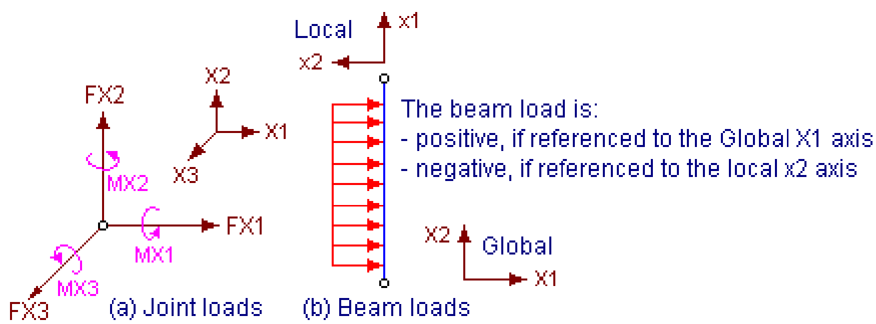
Forces and moments are referenced to a local or global coordinate axis:

- Forces:
  positive forces point in the positive direction of the relevant coordinate axis.
- Moments:
  Moments act ***about*** a local or global coordinate axis. The sign of the moment is determined by the following right-hand rule:



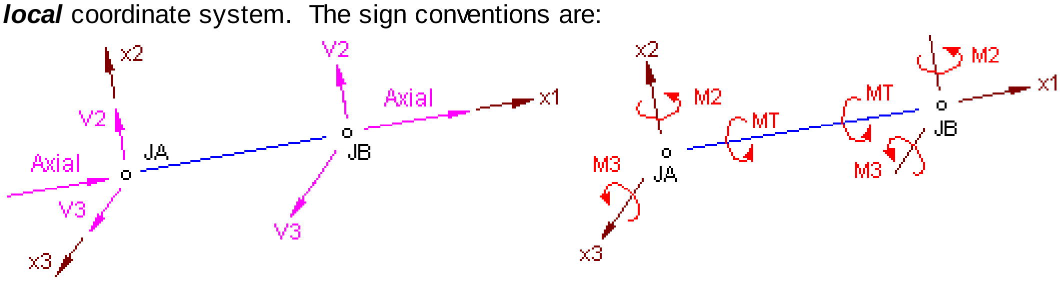
Examples:

- **Loads**
  Joint loads are **always** defined relative to the global coordinate system. Beam loads may be defined relative to either the global coordinate system or to the beam local coordinate system.



  For more information, refer to Loads

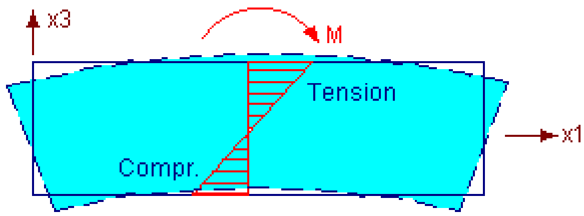
- **Beam Results (Tabular)**
  Moment, shear and axial force results for beam elements are always displayed relative to the beam ***local*** coordinate system. The sign conventions are:



  For more information, refer to Beam results sign conventions

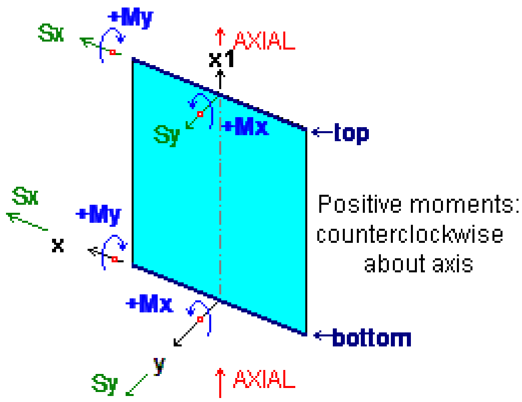
- **Finite Element Results (Tabular)**
  Moments, forces and stresses for finite elements are displayed relative to the element ***local*** coordinate system. Forces and stresses are positive if acting in the positive direction of the parallel local axis.

Referring to the equations in Element results sign conventions, a positive moment creates tension on the +x3 surface of the element.



Note that ***STRAP*** 's graphic postprocessors often unify and reverse the signs to display the results according to accepted engineering sign conventions. Refer to Element result types for more details.

- **Wall elements**
  Moment, shear and axial force results for wall elements are always displayed relative to the wall coordinate system. The sign conventions are:



12.1.3 Exponential format

Decimal values may also be entered exponentially. For example:

- **510** may be entered as **5.1E2** or **5.1E+2**
- **0.0037** may be entered as **.37E-2** or **3.7E-3**

Do not leave any blank spaces between the numbers and the letter E.

12.1.4 List format

A series of node or element numbers may be entered in "list" format, where the keywords **TO** and **BY** may be used to simplify the list.

list examples:

1 9 17 20
1 3 TO 6 12 15 18 TO 30
3 TO 11 BY 2 20 TO 24 34

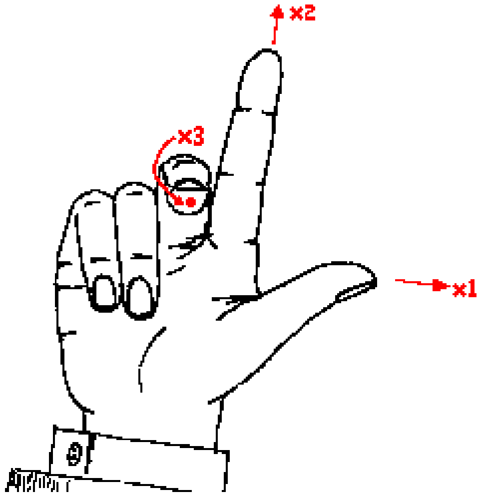
The last example is equivalent to entering:

3 5 7 9 11 20 21 22 23 24 34

A list can consist of up to 50 items, where " **1 TO 6** " is one item.

12.1.5 Right-hand rule

STRAP uses right-handed, Cartesian reference frames.



global axes: **X1, X2, X3** (uppercase)
local axes: **x1, x2, x3** (lowercase).

The X1, X2 and X3 axes (or x1, x2 and x3 axes) are perpendicular to each other and the positive direction of the axes are specified by the 'right-hand rule'. It is obvious that if the directions of any two of the axes are known, then the direction of the third axis is easily determined.

12.1.6 Batch mode

STRAP has facilities for processing geometry, loads and load combinations, solving the model and generating input/output data files without using the interactive graphic interface.

- geometry, loads and/or combinations may be defined by the user by typing commands in an ASCII file using any editor program
- models created in batch mode can be added to the model list using the Add a file to model list option in the **File** menu, and can then be solved, etc.
- alternatively, the model can be solved using the *STBatch* utility.
- for all models, input/output ASCII data files in user specified format can be generated by the *STBatch* utility

Geometry 1195
Loads 1197
Combinations 1198
STBatch

12.1.6.1 Geometry

There are two methods for defining ***STRAP*** geometry via external files:

- Using the Clipboard 1201
- Creating and importing an ASCII file

The file format is as follows:

1st line:
| REPLACE |
| ADD |

where:

REPLACE = the program will use this file instead of the existing geometry file.
ADD = the program adds the commands in this file to the data in the existing binary geometry

file (GEOMnnn.DAT).

(Do not write IGNORE in this line)

Command Lines:

All commands are in the regular format. Before the first line of each command type, a header must be entered on a separate line. The headers are:

/ JOINT COORDINATES
/ RESTRAINTS
/ PROPERTY NUMBER
/ BEAM END RELEASES
/ MEMBER INCIDENCES
/ MATERIAL
/ PROPERTY DEFINITION
/ SPRINGS
/ DUPLICATE A BLOCK
/ UNITS force length

Notes:

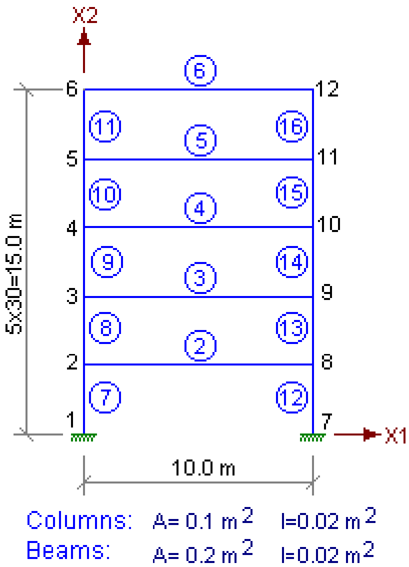
- There must be a space after the "/".
- Lines beginning with " ; " are comment lines and are ignored by the program.
- No blank lines are allowed.
- The order is not important; the commands of any type may appear in scattered groups as long as each group begins with the header.

If REPLACE is used, the program does not read the binary geometry file and hence does not know what the model is 'plane' or 'space'. It assumes that the model is 'space' and expects three coordinates in every node definition command. Type the command COORD 2 on a separate line (after / JOINT COORDINATES) if the following commands contain only two coordinates.

If the program discovers a format error in a command when reading the geometry file, it exits and displays a message. The error/warning messages are written to a file "ERR1.LST" that may be displayed or printed.

Refer also to GEOINnnn.DAT 1202.

Example:



The ASCII file is:

```
REPLACE
/ JOINT COORDINATES
COORD 2
1 0 0 TO 6 0 15 EQ
7 10 0 TO 12 10 15 EQ
/ RESTRAINTS
X1 X2 X6 1 7
/ PROPERTY NUMBERS
1 1 TO 10
2 11 TO 15
/ MEMBER INCIDENCES
1 TO 5 1 2
6 TO 10 7 8
11 TO 15 7 8
/ MATERIALS
CONC
/ PROPERTY DEFINITION
1 A 0.1 I 0.002
2 A 0.2 I 0.02
```

12.1.6.2 Loads

There are two methods for defining *STRAP* loads via external files:

- Use the Clipboard 1201
- Create a STATnnn.DAT file

The file name must be: ***STATnnn.DAT***

where "nnn" can be verified by selecting the **Display all model files** option in the **Files** pull-down menu on the main menu bar.

The file format is as follows:

1st line: ASCII

for ***each*** load case - 1st line: load case title

All loads must be in the command format as explained in detail in the Command Mode manual. Before the first line of each load type, a header must be entered on a separate line. The headers are:

```
/ BEAM LOADS
/ JOINT LOADS
/ DISPLACEMENTS
/ PRESSURE
/ LOAD COMBINATIONS
/ GLOBAL LOADS
```

end of load case: / END

end of file: / END STATIC (instead of / END)

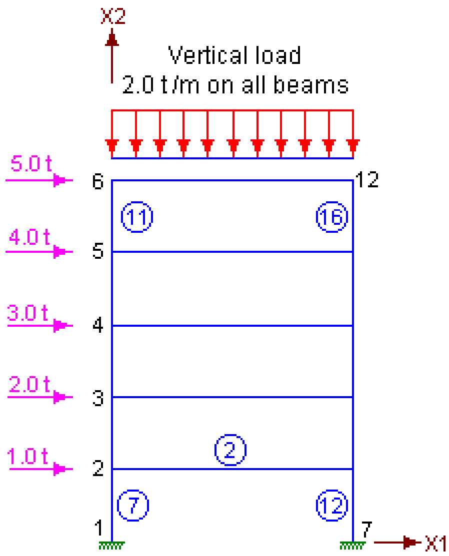
Notes:

- In all command lines, there must be a space after the "/".
- No blank lines are allowed.
- The order is not important; the commands of any type may appear in scattered groups as long as each group begins with the header.

- The last loading case should not have a "/ END" command prior to the "/ END STATIC" command.

Example:

The vertical and horizontal loads are in separate load cases for the following space frame example.



The STATnnn.DAT file is:

```
ASCII
 SELF-WEIGHT AND ROOF LOADS
 / BEAM LOADS
 SELF  X3  B  2  TO  16
 U  GLOB  FX3  -2.0  B  2  TO  6
 / END
 WIND LOADS
 / JOINT LOADS
 FX1  1.0  N  2
 FX1  2.0  N  3
 FX1  3.0  N  4
 FX1  4.0  N  5
 FX1  5.0  N  6
 / END  STATIC
```

12.1.6.3 Combinations

A file containing load combination commands can be imported into ***STRAP*** by "cutting and pasting" the commands in the Results "Define/revise combinations" option:

- type the combination commands in a Windows editor program such as "Wordpad" in the format:

  TITLE tit   (optional)
  lc1 f1 lc2 f2 ....lcn fn..G1 fg1 ...... Gn fgn

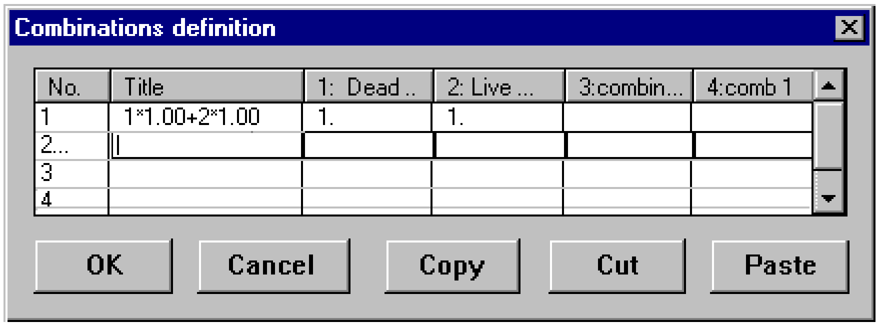
  where:

tit	=	combination title string. The program creates a default title if this line is omitted
lcn	=	load case number
fn	=	factor for load case 'n'
fgn	=	factor for group 'n'

  Example:
  for a combination 1.4*load case 1 + 1.6 * load case 3 + 1.2 * group 2, titled "Dead + Live + Group 2", type:

  TITLE Dead + Live & Group 2
  1 1.4 3 1.6 G2 1.2

- Highlight the commands (click and hold the mouse, drag the cursor), then select "**Edit**", "**Copy**" in the menu bar
- Press [Alt][Tab] to return to ***STRAP***



- place the cursor anywhere on the line where the command is to be written and click the mouse (if you select a line with an existing combination, then the command will be inserted above the line).
- click the Paste button

Note that multiple commands may be "cut and pasted" at the same time.

12.1.7 Command mode

The following section explains in general how to define the model using the Command Mode. For a complete explanation, refer to the Command Mode Manual.

The commands are typed in by the user in the Command Box at the bottom of the screen; the program automatically updates the graphic display of the model.

It is important to note the following:
- the Graphic Mode and the Command Mode may be used concurrently.
- every time a part of the model is defined in the Graphic Mode, the program automatically writes the equivalent command in the Command Box. Therefore, the Command Box contains a complete record of all that was defined ***in the current session***.

Enter a command 1200
Revise a command 1200
Retrieve command (from Clipboard) 1201
General format 1199

Refer also to:
Batch mode 1195

12.1.7.1 Command format

The commands must be in the format specified by the Command Mode Manual. For example, to define the self-weight of beams as a load on the structure, the manual specifies the command:

```
Self |X1| (f) BEAM list
     |X2|
     |X3|
```

- All data is input in free format. There must be at least one blank space between one data value and another (including words, numerical values, and special symbols - without exception).
- The words in capital letters are keywords which must be entered exactly as they appear in the format statement.
- The program normally requires only the first one or two letters of a keyword in order to identify it.

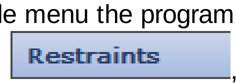
Underlined letters indicate the letters that the program reads. Simplify the input by typing these letters only. For example:
S instead of SELF
B instead of BEAM

- Lower-case letters indicate numerical data. In general, parameters beginning with i,j,k,l,m,n indicate integer values, and all other letters indicate decimal values. For example:
  n, n1, ... are symbols for node numbers (integers)
  p, p1, ... are symbols for section dimensions (decimals)
- Parameters in brackets ( ) indicate optional input. In the self-weight example above, f may be omitted.
- Keywords in brackets | | indicate a choice of one of the keywords listed. In the above example, type one of X1,X2,X3 to specify the direction the loads act.
- "list" indicates a list of nodes or elements in the List Format. For example:
  1 9 17 20
  1 3 TO 6 12 15 18 TO 30
  3 TO 11 BY 2 20 TO 24 34

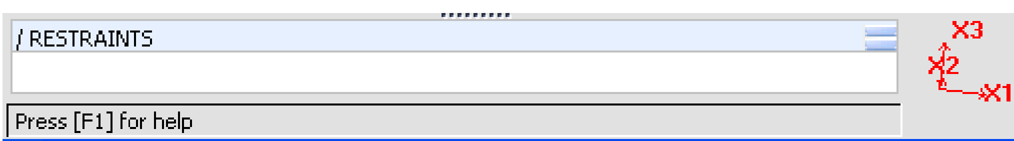
  The last example is equivalent to entering: 3 5 7 9 11 20 21 22 23 24 34
  A list can consist of up to 50 items, where " 1 TO 6 " is one item

Typical input for the self-weight example above:
SELF X1 B 10 11 12
S X3 -1.4 B 1 TO 90

12.1.7.2 Enter a command

When you click on an icon in the side menu the program automatically writes a header in the Command Box. For example, when you select 

, the Command Mode Box appears as:



To enter a command:

- move the cursor into the last line in the box and click the mouse; the **I** cursor is displayed.
- type in the command in the correct format and press [Enter]; the display is updated.

12.1.7.3 Revise a command

All commands are entered in the Command box as the model is defined. The box displays only two rows and so the commands are scrolled up and disappear (the box can be enlarged as any "Windows" box). To recall commands, scrolled using the up/down arrow buttons at the right side of the box.

Example: revise the definition for beam 16754.

- scroll up until the command is displayed.
- click on the line; the command is rewritten at the bottom of the Command Box and the **I** cursor is displayed at the beginning of the line.
- Correct the command and press [Enter]; the display is updated.

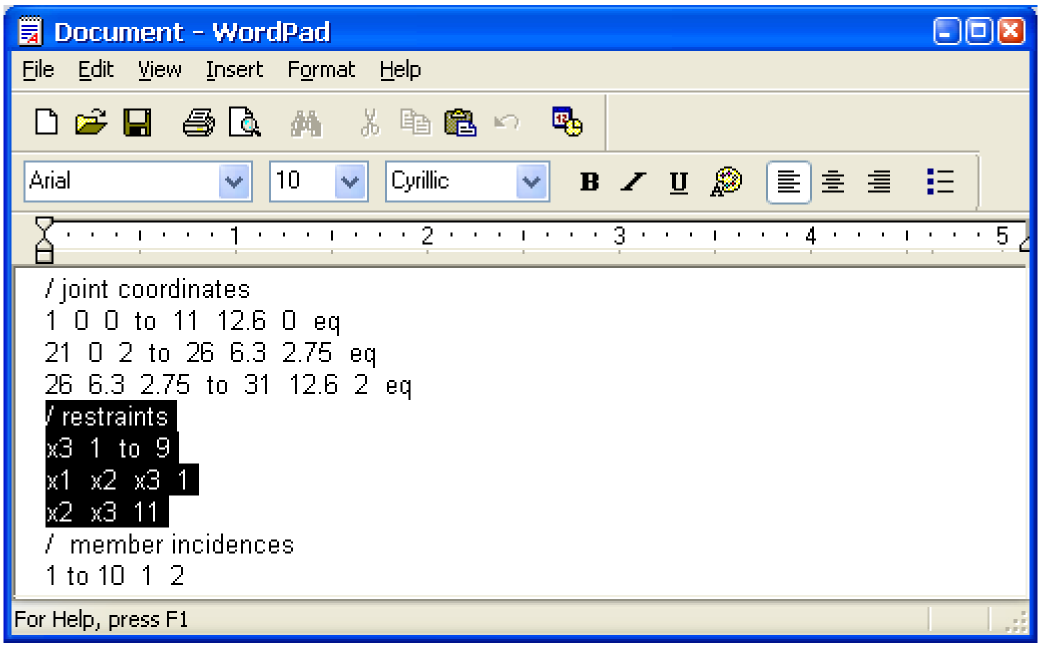
12.1.7.4 Paste a command

Commands located in any program or file may be copied and pasted into the STRAP command box.

This option is similar to the Batch Mode feature, but allows selected commands to be retrieved rather than the entire file.

To write the commands to the clipboard:

- run "Wordpad", "Notepad" or any ‘editor’ program..
- either type in the commands or retrieve an existing file using the **File** option.
- move the to the start of the block of commands; click the mouse.
- without releasing the mouse key drag the to the end of the last command in the block; release the mouse key (the block should be highlighted).



- Select the **Edit** option from the top menu bar.
- Select the **Copy** option from the pulldown menu.

To paste the commands into ***STRAP*** :

- Select the **Edit** option from the ***STRAP*** top menu bar.
- Select **Paste commands** from the pulldown menu.
- The program reads the commands, adds them to the Command Box and updates the graphic display.

If the program encounters commands with format errors or commands that generate warnings (i.e. redefinition of existing nodes), it pauses and displays the following menu:

Edit

The command is displayed in the Command Box at the bottom of the screen; edit the command as explained in Revise a command1200. The program then continues to the next command in the clipboard.

Ignore

The program ignores the current command and continues to the next command in the clipboard.

Quit

The program ignores the current command and all following commands in the clipboard.

12.1.8 GEOINnnn.DAT

The current geometry for each model is stored in a binary file named ***GEOMnnn.DAT***, where "***nnn***" can be verified by selecting the **Display all model files** option in the **Files** pull-down menu on the main menu toolbar.

However, ***STRAP*** simultaneously creates for each model an ASCII data file that contains all of the geometry data that was defined interactively in the form of commands. The command formats are explained in detail in the Command Mode manual.

The file name is: ***GEOINnnn.DAT***

This file may be edited or updated external to the program using a screen editor; ***STRAP*** can then be instructed to use this file instead of the existing geometry file (GEOMnnn.DAT) as the source for the current model geometry.

When you choose the **Geometry** option in the Main Menu bar, the program reads the first line of the GEOINnnn.DAT file. If the first line is:

IGNORE	=	the program ignores this file and uses the GEOMnnn.DAT file only. This is the normal case because when the model is entered interactively, the program automatically writes IGNORE in the first line of the GEOINnnn.DAT file
REPLACE	=	the program uses this file instead of the existing geometry file, i.e it ignores the GEOMnnn.DAT file
ADD	=	the program adds the commands in this file to existing geometry in the GEOMnnn.DAT file. If data is defined in both files the GEOINnnn.DAT data overwrites the GEOMnnn.DAT data.

If a format error is discovered in a command, the program exits with a warning. All warning and error messages are entered in a file "ERRnnn.LST" which may be displayed or printed.

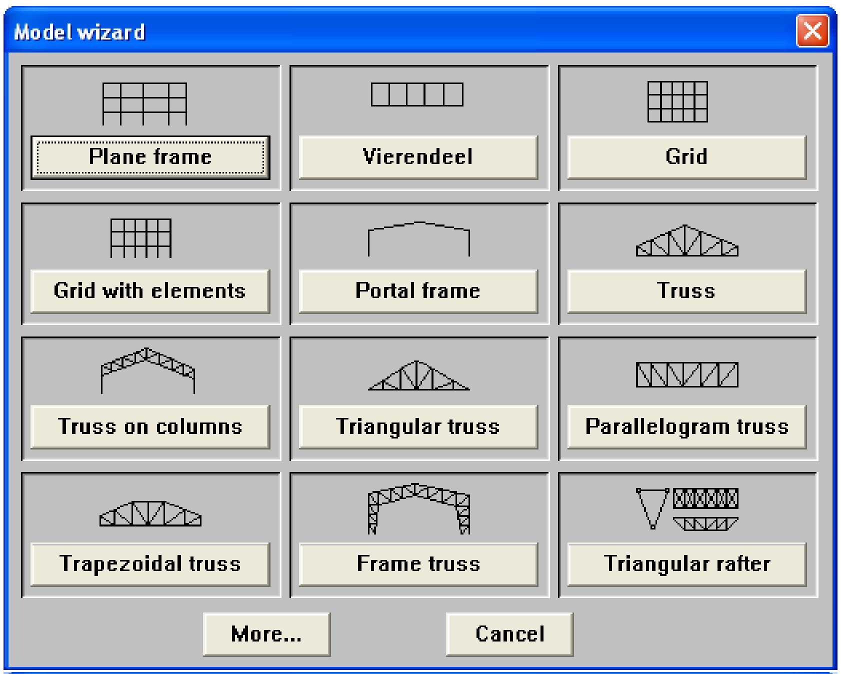
After reading all of the command lines in the file, the program changes the first line back to IGNORE.

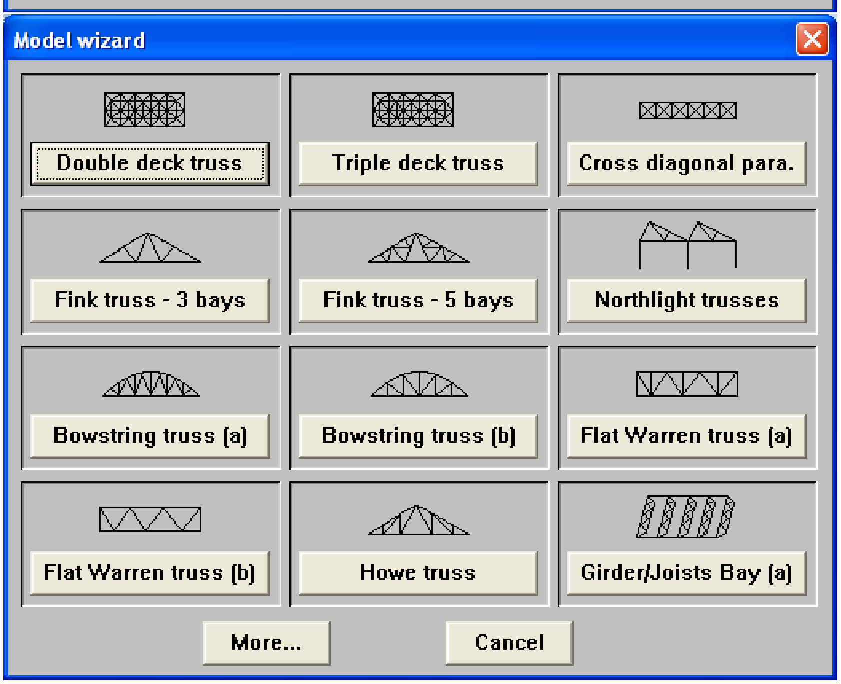
12.2 Geometry

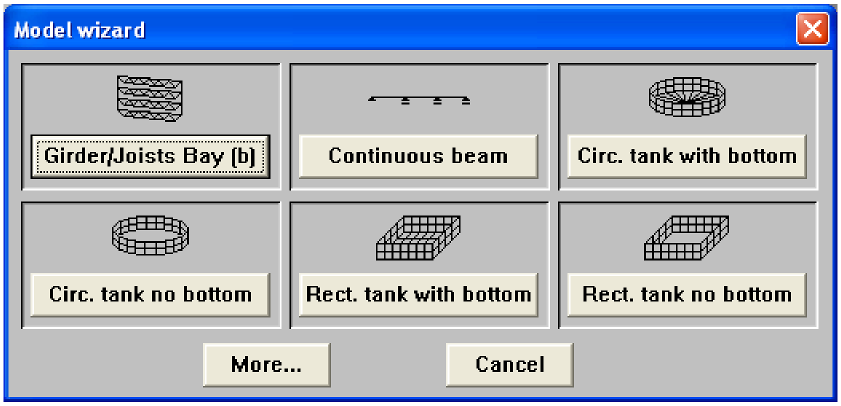
Select:
Wizard models 1203
Wizard models - add 1220
Nodes - equations 1230

12.2.1 Wizard models

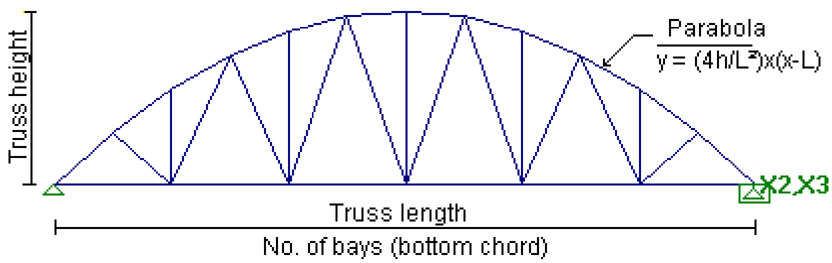
Space frames







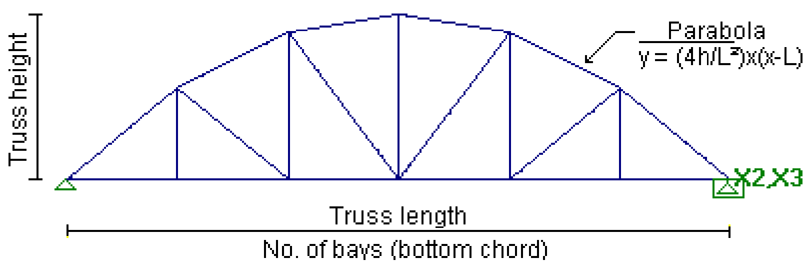
12.2.1.1 Bowstring truss (a)



Property Groups:

1 = Top chord
2 = Bottom Chord
3 = Diagonals and verticals

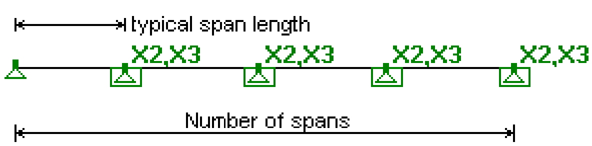
12.2.1.2 Bowstring truss (b)



Property Groups:

1 = Top chord
2 = Bottom Chord
3 = Diagonals and verticals

12.2.1.3 Continuous beam



Parameters:

- No. of spans
- Typical span length

Property groups:
1 = all spans

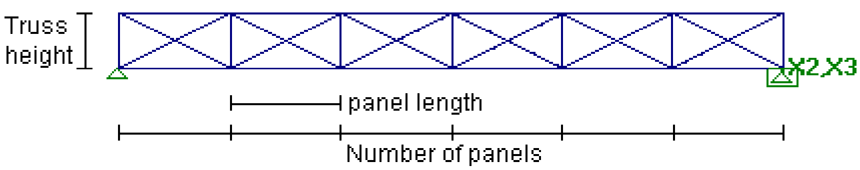
Loads:

Dialog box title	Data requested
Uniform vertical loads:	Dead/live load
	Self-weight factor
	Dead/live combination factors

The program generates the following 5 load cases:

	Load case	Description
1 -	Dead load	Dead load X 1.0
2 -	Live load	Live load X 1.0
3 -	Dead + live	Dead* max. factor + Live* factor
4 -	Alternate dead + live - 1	Staggered loads on alternate spans
		- Dead*max. factor + Live *factor
		- Dead * min. factor
5 -	Alternate dead + live - 2	Similar to 4

12.2.1.4 Cross diagonal truss



Parameters:
- No. of panels
- Panel length
- Truss height

Property Groups:
1 = Bottom chord
2 = Top Chord
3 = Verticals
4 = Diagonals

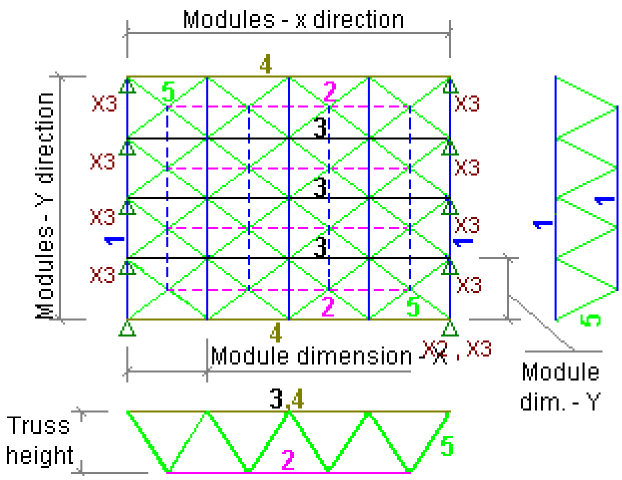
Loads:

Dialog box title	Data requested
Uniform vertical loads:	Dead/live load on top/bottom chords
	Self-weight factor
Combination factors:	Factors for:
	- Dead+Live

The program generates the following 3 load cases:

	Load case	Description
1 -	Dead load	Dead load X 1.0
2 -	Live load	Live load X 1.0
3 -	Dead + live	Dead* max. factor + Live* factor

12.2.1.5 Double deck truss



Parameters:

- No. of Modules - X direction
- No. of Modules - Y direction
- Module Dimension - X direction
- Module Dimension - Y direction
- Height of Truss

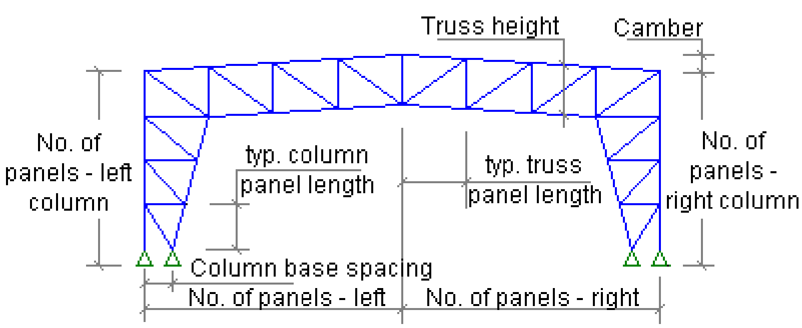
Property Groups:

1 = top and bottom chords - Y direction
2 = bottom chords - X direction
3 = top chords - X direction
4 = exterior top chords - X direction
5 = all diagonals

Loads:

- Top deck area load (global area load)
- Bottom deck area load (global area load)
- Self weight factor

12.2.1.6 Frame truss



Parameters:

- Total No. of Panels in Left Column **
- No. of Panels in Left Side of Truss **
- Typical Column Panel Length
- Typical Truss Panel Length
- Spacing between Column Chords at Base

- Total Height of Truss
- Camber of Truss
- Total No. of Panels in Right Column *
- No. of Panels in Right Side of Truss *

* - may be defined only after the structure has been created.
** one panel is located in both the truss and the column

Property Groups:

1 = Top chord - truss
2 = Bottom Chord - Truss
3 = Verticals in truss; Horizontals in columns
4 = Diagonals - Truss and Column
5 = Exterior column verticals
6 = Interior column verticals

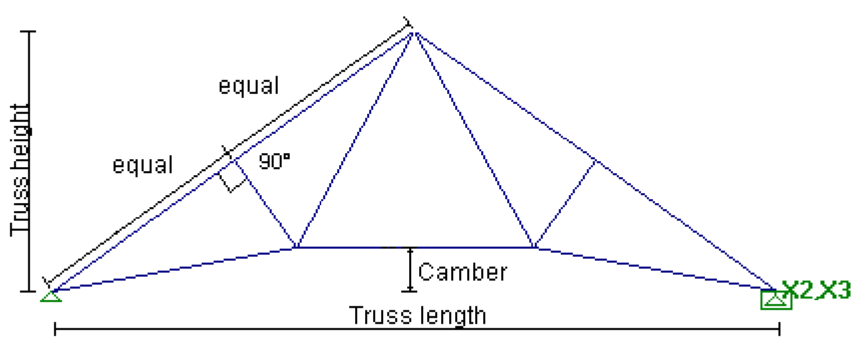
Loads:

Dialog box title	Data requested
Uniform vertical loads:	Dead/live load on top/bottom chords Self-weight factor
Wind loads:	Wind parallel to frame: wind on left/right roof/column Wind parallel to ridge: wind on roof/column
Combination factors:	Factors for: - Dead+Live - Dead+Live+Wind - Dead + Wind

The program generates the following 9 load cases:

		Load case	Description
1	-	Dead load	Dead load X 1.0
2	-	Live load	Live load X 1.0
3	-	Wind load (parallel to frame)	Wind load 1 X 1.0
4	-	Wind load (parallel to ridge)	Wind load 2 X 1.0
5	-	Dead + live	Dead* max. factor + Live* factor
6	-	Dead + live + wind 1	Loads x 2nd set of factors (above)
7	-	Dead + live + wind 2	Loads x 2nd set of factors (above)
8	-	Dead + wind 1	1.0*Dead + Wind * 3rd set factor
9	-	Dead + wind 2	1.0*Dead + wind * 3rd set factor

12.2.1.7 Fink truss - 3 bays

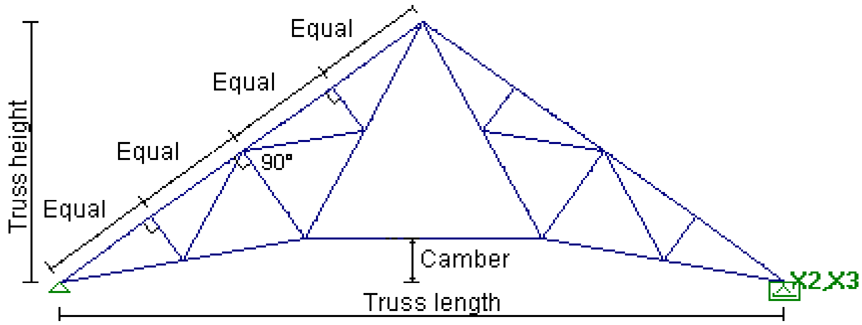


Property Groups:

1 = Bottom chord

2 = Top Chord
3 = Diagonals

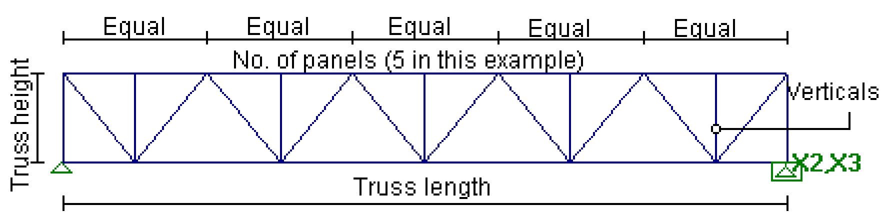
12.2.1.8 Fink truss - 5 bays



Property Groups:

1 = Bottom chord
2 = Top Chord
3 = Diagonals

12.2.1.9 Flat Warren truss



Parameters

- No. of panels
- Truss length
- Truss height

Property Groups:

1 = Bottom chord
2 = Top Chord
3 = Verticals and Diagonals

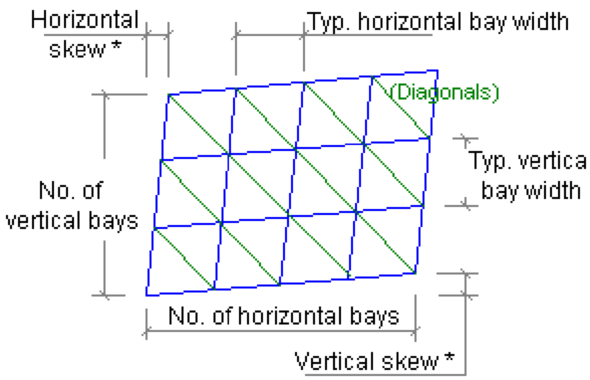
Loads:

Dialog box title	Data requested
Uniform vertical loads:	Dead/live load on top/bottom chords Self-weight factor
Combination factors:	Factors for: - Dead+Live

The program generates the following 3 load cases:

Load case	Description
1 - Dead load	Dead load X 1.0
2 - Live load	Live load X 1.0
3 - Dead + live	Dead* max. factor + Live* factor

12.2.1.10 Grid - beams/elements



Parameters:

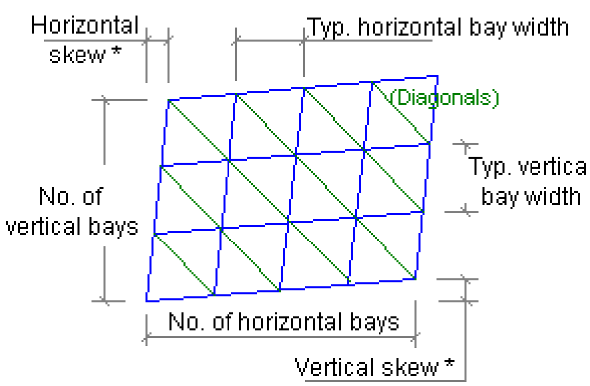
- No. of Horizontal Bays
- No. of Vertical Bays
- Typical Horizontal Bay Width
- Typical Vertical Bay Width
- Horizontal Skew *
- Vertical Skew **

* - may be defined only after the structure has been created.
** - "Grid", "Grid with elements" only: may be defined only after the structure has been created.

Property Groups:

1 = All beams/elements

12.2.1.11 Grid - beam with diagonals



Parameters:

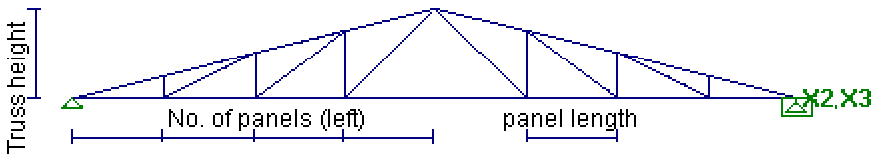
- No. of Horizontal Bays
- No. of Vertical Bays
- Typical Horizontal Bay Width
- Typical Vertical Bay Width
- Horizontal Skew *
- Vertical Skew *

* may be defined only after the structure has been created.

Property Groups:

1 = All beams

12.2.1.12 Howe truss



Parameters

- No. of panels, left side
- Panel length
- Truss height at centre

Property Groups:

1 = Bottom chord
2 = Top Chord
3 = Verticals
4 = Diagonals

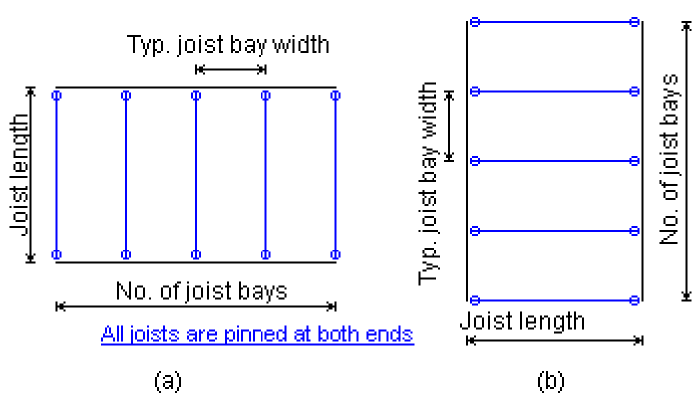
Loads:

Dialog box title	Data requested
Uniform vertical loads:	Dead/live load on top/bottom chords
	Self-weight factor
Wind loads:	Wind parallel to frame: wind on left/right roof
	Wind parallel to ridge: wind on roof
Combination factors:	Factors for:
	- Dead+Live
	- Dead+Live+Wind
	- Dead + Wind

The program generates the following 9 load cases:

Load case	Description
1 - Dead load	Dead load X 1.0
2 - Live load	Live load X 1.0
3 - Wind load (parallel to frame)	Wind load 1 X 1.0
4 - Wind load (parallel to ridge)	Wind load 2 X 1.0
5 - Dead + live	Dead* max. factor + Live* factor
6 - Dead + live + wind 1	Loads x 2nd set of factors (above)
7 - Dead + live + wind 2	Loads x 2nd set of factors (above)
8 - Dead + wind 1	1.0*Dead + Wind * 3rd set factor
9 - Dead + wind 2	1.0*Dead + wind * 3rd set factor

12.2.1.13 Joists



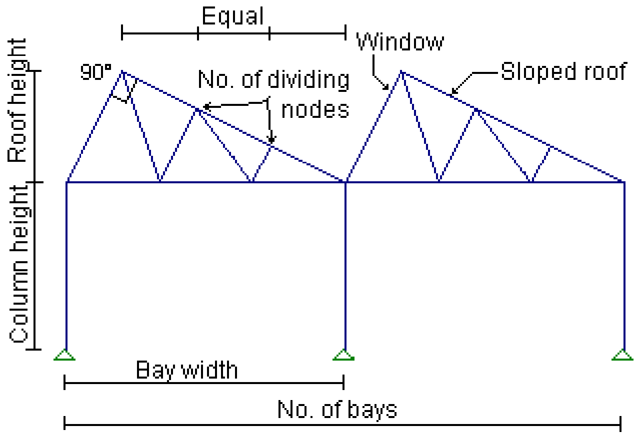
Parameters:

- No. of joist bays
- Joist bay width
- Joist length

Property groups:

1 = girders
2 = joists

12.2.1.14 North light truss



Property Groups:

1 = Bottom chord
2 = Top Chord
3 = Diagonals
4 =Columns

Loads:

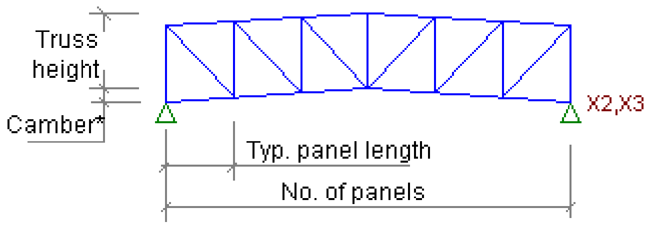
Dialog box title	**Data requested**
Uniform vertical loads:	Dead/live load on sloped roof/window
	Self-weight factor
Wind loads:	Wind from left: wind on sloped roof/window/leftcolumn/right column
	Wind from right:wind on sloped roof/window
	Wind parallel to ridge: wind on roof/columns
Combination factors:	Factors for:
	- Dead+Live
	- Dead+Live+Wind
	- Dead + Wind

The program generates the following 12 load cases:

	Load case	**Description**
1 -	Dead load	Dead load X 1.0
2 -	Live load	Live load X 1.0
3 -	Wind from left	Wind1 load X 1.0
4 -	Wind from right	Wind2 load X 1.0
5 -	Wind along ridge	Wind3 load X 1.0
6 -	Dead+live	Dead*max. factor + Live* factor
7 -	Dead+live+wind1	Loads * 2nd set of factors above
8 -	Dead+live+wind2	Loads * 2nd set of factors above
9 -	Dead+live+wind3	Loads * 2nd set of factors above
10 -	Dead+wind1	1.0*Dead + Wind1*3rd set of factors

11 -	Dead+wind1	1.0*Dead + Wind2*3rd set of factors
12 -	Dead+wind1	1.0*Dead + Wind3*3rd set of factors

12.2.1.15 Parallelogram truss

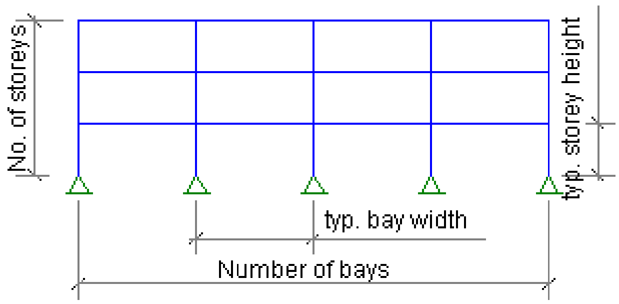


Parameters:
- No. of Panels
- Typical Panel Length
- Truss height
- Camber *

* - may be defined only after the structure has been created.

Property Groups:
1 = Bottom Chord
2 = Top Chord
3 = Verticals
4 = Diagonals

12.2.1.16 Plane frame



Parameters:
- No. of Bays
- No. of Storeys
- Typical Bay Width
- Typical Storey Height

Property Groups:
1 = Beams
2 = Columns

Loads:

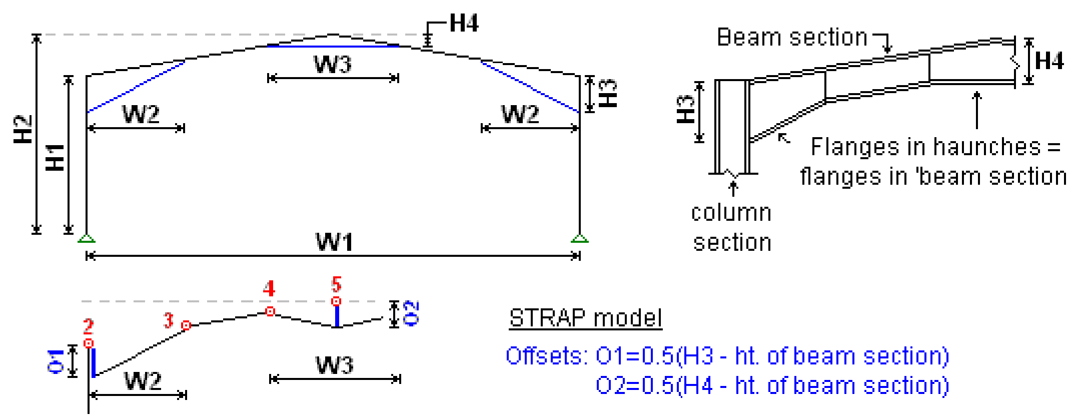
Dialog box title	Data requested
Uniform vertical loads:	Dead load
	Live load
	Self-weight factor
Uniform wind loads:	Wind on left column
	Wind on right column
Combination factors:	Factors for:

- Dead+Live
- Dead+Live+Wind
- Dead + Wind

The program generates the following 9 load cases:

	Load case	Description
1	- Dead load	Dead load X 1.0
2	- Live load	Live load X 1.0
3	- Wind load	Wind load X 1.0
4	- Alternate dead and live - 1	Staggered loads on alternate spans: - Dead*max. factor + Live *factor - Dead * min. factor
5	- Alternate dead and live - 2	Similar to 4
6	- Dead + live	Dead* max. factor + Live* factor - all spans
7	- Dead + live + wind	Loads x 2nd set of factors (above)
8	- Alternate dead + wind - 1	1.0*Dead + Wind * 3rd set factor
9	- Alternate dead + wind - 2	Dead * max. factor + wind * 3rd set factor

12.2.1.17 Portal frame



Parameters:

- H1 to H4 - shown on drawing above
- W1 to W3 - " " "

Property Groups:

1 = Beams
2 = Columns

Loads:

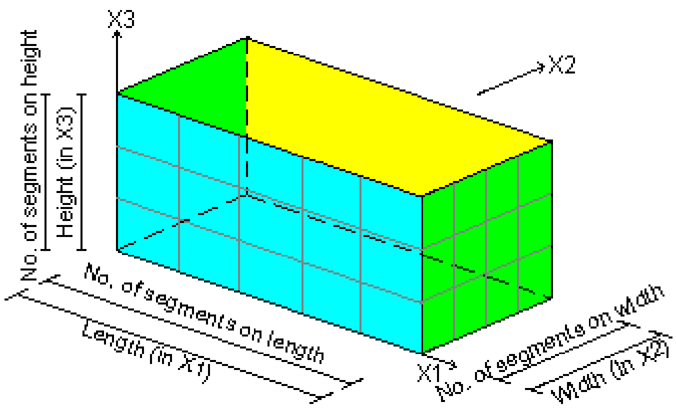
Dialog box title	Data requested
Uniform vertical loads:	Dead load
	Live load
	Self-weight factor
Side wind loads:	Wind on left column / roof
	Wind on right column /roof
Front wind loads:	Wind on roof
	Wind on columns

The program generates the following 4 load cases:

	Load case	Description
1 -	Dead load	(Dead load + Self-weight) X 1.0

2 -	Live load	Live load X 1.0
3 -	Wind load (side)	Wind load (side) X 1.0
4 -	Wind load (front)	Wind load (front) X 1.0

12.2.1.18 Tank - rectangular



Property groups:

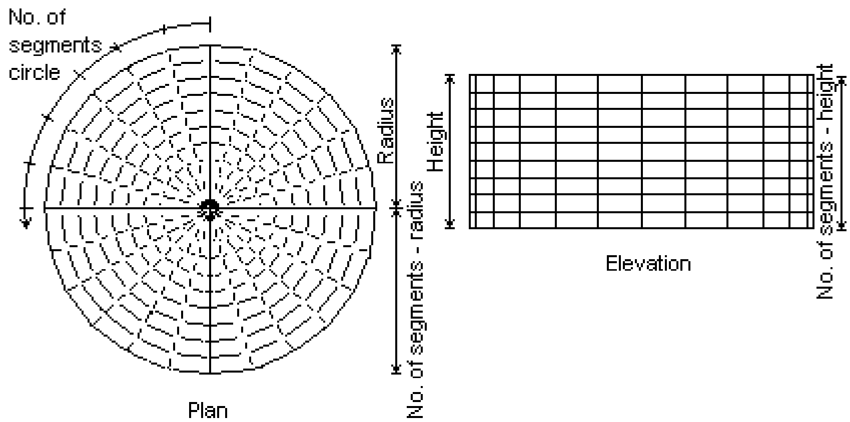
1 = bottom

2 = walls

Loads:

- (horizontal) pressure at tank bottom
- (horizontal) pressure at top of load
- Height at top of load
- Vertical pressure at bottom
- Self weight factor

12.2.1.19 Tank - round



Property groups:

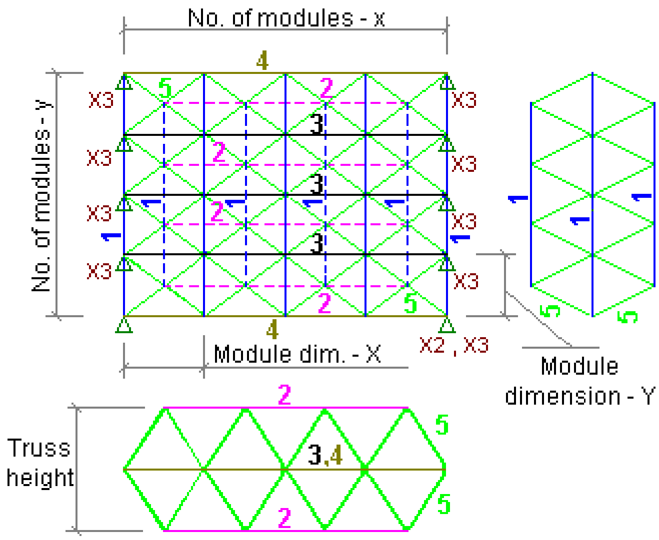
1 = bottom

2 = walls

Loads:

- (horizontal) pressure at tank bottom
- (horizontal) pressure at top of load
- Height at top of load
- Vertical pressure at bottom
- Self weight factor

12.2.1.20 Triple deck truss



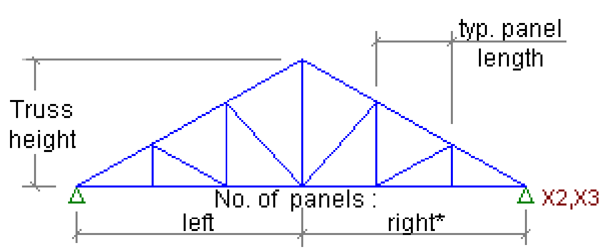
Parameters:
- No. of Modules - X direction
- No. of Modules - Y direction
- Module Dimension - X direction
- Module Dimension - Y direction
- Height of Truss

Property Groups:

1 = top, bottom and middle chords - Y direction
2 = top and bottom chords - X direction
3 = middle chord - X direction
4 = exterior middle chords - X direction
5 = all diagonals

Loads:
- Top deck area load (global area load)
- Bottom deck area load (global area load)
- Self weight factor

12.2.1.21 Triangular truss



Parameters:
- No. of Panels - Left
- Typical Panel Length
- Truss Height at Centre
- No. of Panels - Right *

* - may be defined only after the structure has been created.

Property Groups:

1 = Bottom chord
2 = Top chord
3 = Verticals
4 = Diagonals

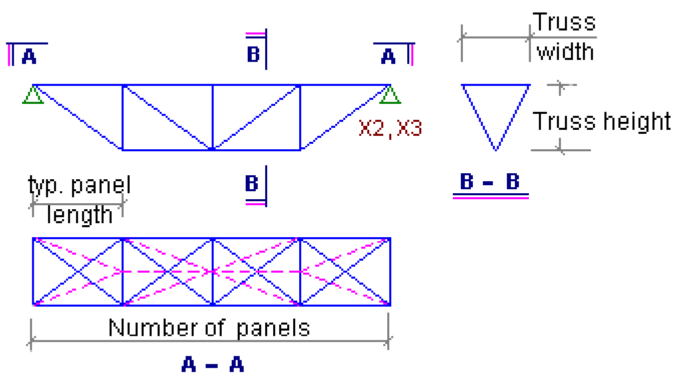
Loads:

Dialog box title	Data requested
Uniform vertical loads:	Dead/live load on top/bottom chords Self-weight factor
Wind loads:	Wind parallel to frame: Wind on left/right roof Wind parallel to ridge: Wind on roof
Combination factors:	Factors for: • Dead+Live • Dead+Live+Wind • Dead + Wind

The program generates the following 9 load cases:

	Load case	Description
1 -	Dead load	Dead load X 1.0
2 -	Live load	Live load X 1.0
3 -	Wind load (parallel to frame)	Wind load 1 X 1.0
4 -	Wind load (parallel to ridge)	Wind load 2 X 1.0
5 -	Dead + live	Dead* max. factor + Live* factor
6 -	Dead + live + wind 1	Loads x 2nd set of factors (above)
7 -	Dead + live + wind 2	Loads x 2nd set of factors (above)
8 -	Dead + wind 1	1.0*Dead + Wind * 3rd set factor
9 -	Dead + wind 2	1.0*Dead + wind * 3rd set factor

12.2.1.22 Triangular rafter



Parameters:

- No. of Panels
- Typical Panel Length
- Truss Width
- Truss Height

Property Groups:

1 = Top Chords
2 = Bottom Chord
3 = \/_ bracing

4 = X - bracing in top panel
5 = > , < diagonals from top to bottom chord

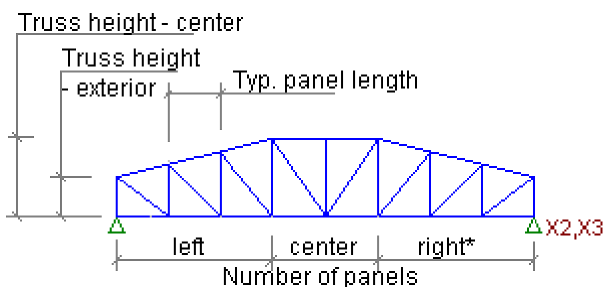
Loads:

Dialog box title	Data requested
Uniform vertical loads:	Dead/live load on top/bottom chords Self-weight factor
Wind loads:	Wind on top chords
Combination factors:	Factors for: - Dead+Live - Dead+Live+Wind - Dead + Wind

The program generates the following 6 load cases:

	Load case	Description
1 -	Dead load	Dead load X 1.0
2 -	Live load	Live load X 1.0
3 -	Wind load	Wind load X 1.0
4 -	Dead + live	Dead* max. factor + Live* factor
5 -	Dead + live + wind	Loads x 2nd set of factors (above)
6 -	Dead + wind	1.0*Dead + Wind * 3rd set factor

12.2.1.23 Trapezoidal truss



Parameters:

- No. of Panels - Left Sloped Section
- No. of Panels - Centre Flat Section
- Typical Panel Length
- Height of Truss - Exterior
- Height of Truss - Centre
- No. of Panels - Right Sloped Section *

* - may be defined only after the structure has been created.

Property Groups:

1 = Bottom Chord
2 = Top Chord
3 = Verticals
4 = Diagonals

Loads:

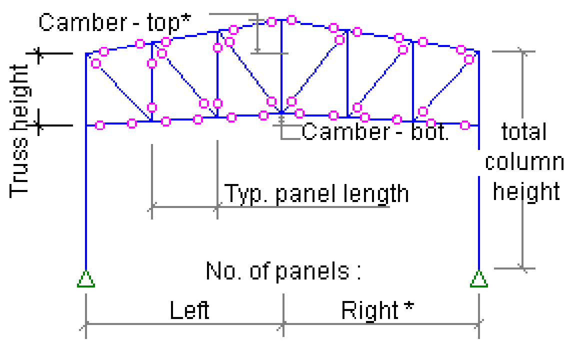
Dialog box title	Data requested
Uniform vertical loads:	Dead/live load on top/bottom chords Self-weight factor
Wind loads:	Wind parallel to frame: wind on left/right Wind parallel to ridge: wind on roof

Combination factors:	Factors for: - Dead+Live - Dead+Live+Wind - Dead + Wind

The program generates the following 9 load cases:

		Load case	Description
1	-	Dead load	Dead load X 1.0
2	-	Live load	Live load X 1.0
3	-	Wind load (parallel to frame)	Wind load 1 X 1.0
4	-	Wind load (parallel to ridge)	Wind load 2 X 1.0
5	-	Dead + live	Dead* max. factor + Live* factor
6	-	Dead + live + wind 1	Loads x 2nd set of factors (above)
7	-	Dead + live + wind 2	Loads x 2nd set of factors (above)
8	-	Dead + wind 1	1.0*Dead + Wind * 3rd set factor
9	-	Dead + wind 2	1.0*Dead + wind * 3rd set factor

12.2.1.24 Truss on columns



Parameters:

- Total Column Height
- Truss Height
- No. of Panels at Left
- Typical Panel Length
- Camber at Centre (initial top and bottom)
- Top Camber *
- No. of Panels at Right *

* - may be defined only after the structure has been created.

Property Groups:

1 = Bottom Chord
2 = Top Chord
3 = Verticals in Truss
4 = Diagonals
5 = Columns

Loads:

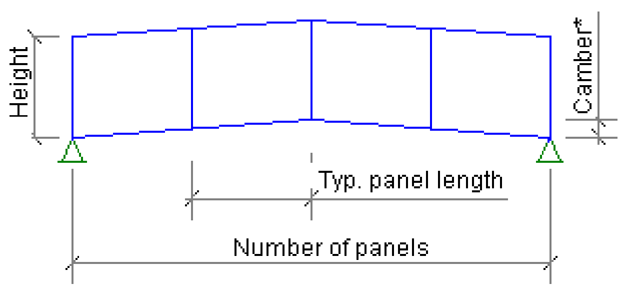
Dialog box title	Data requested
Uniform vertical loads:	Dead/live load on top/bottom chords Self-weight factor Dead/live combination factors
Wind loads:	Wind parallel to frame: Wind on left/right ridge/column Wind parallel to ridge: Wind on ridge/column

Combination factors:	Factors for: - Dead+Live - Dead+Live+Wind - Dead + Wind

The program generates the following 9 load cases:

	Load case	Description
1	- Dead load	Dead load X 1.0
2	- Live load	Live load X 1.0
3	- Wind load (parallel to frame)	Wind load 1 X 1.0
4	- Wind load (parallel to ridge)	Wind load 2 X 1.0
5	- Dead + live	Dead* max. factor + Live* factor
6	- Dead + live + wind 1	Loads x 2nd set of factors (above)
7	- Dead + live + wind 2	Loads x 2nd set of factors (above)
8	- Dead + wind 1	1.0*Dead + Wind * 3rd set factor
9	- Dead + wind 2	1.0*Dead + wind * 3rd set factor

12.2.1.25 Vierendael



Parameters:

- No. of Panels
- Typical Panel Length
- Height of Vierendeel
- Camber   (may be defined after the model has been created)

Property Groups:

1 = Chords
2 = Verticals

Loads:

Dialog box title	Data requested
Uniform vertical loads:	Dead/live load on top/bottom chords Self-weight factor Dead/live combination factors

The program generates the following 3 load cases:

	Load case	Description
1	- Dead load	Dead load X 1.0
2	- Live load	Live load X 1.0
3	- Dead + LIve	Dead*factor + Live*factor

12.2.2 Wizard - Add new models

The model wizard allows you to define the geometry and loads for standard structures by inputting a limited number of parameters.

For example, the four parameters required to define the nodes and beams for a simple plane frame are: number of bays, number of storeys, typical bay width and typical storey height.

Users familiar with *STRAP* are aware that the model geometry may be defined using the "command mode" where commands are typed in the box at the bottom of the screen. The model wizard file contains these commands but with parameters instead of numbers.

For example, a plane grid of nodes is defined by specifying three node points using the following format:

```
n1 x1 x2 TO n2 x1 x2 d1 TO n3 x1 x2 d2
```

where:

n1	=	node number of the first node on the base line n1-n2
n2	=	node number of the last node on the base line n1-n2
n3	=	node number of the last node on the height line n2-n3
x1,x2	=	node coordinates
d1	=	distances between adjacent nodes along n1-n2
d2	=	distances between adjacent nodes along n2-n3

The corresponding node generation command for the plane frame model in the wizard file is:

```
1 0 0 TO #end1 @total_w 0 DIST @distx TO #end2 @total_w @total_h DIST @disty
```

Obviously, "#end1" represents a parameter indicating the number of the node at the end of the base line and must be related to the "Number of bays" parameter entered by the user, i.e. End node number = number of bays + 1.

Therefore, the wizard file contains a series of "wizard commands" that define the variables required for the *STRAP* node, beam/element, property, property group, support and load commands.

The following sections describe how to build the wizard commands, create prompts for the parameters, write equations to calculate the variables in the *STRAP* commands, check for errors in the input and display warning and error messages.

The wizard file must be in the following format:

Model title list 1221
/ END

For each model:

Model title
This text in this line **must be identical** to the text in the corresponding title line at the start of the file

/ INIT
"INIT" block commands 1221
The commands in this block assign initial default values to the input parameters and define the variables required for the model definition.

/ MENU
"MENU" block commands 1222
The commands in this block define the variables that are displayed in the dialog box at the bottom of

the screen after the model is created

- / **DIMENSIONS**
  "DIMENSION" block commands 1223
  The commands in this block define the dimension lines that are automatically drawn by the program when the model is displayed.
- / **CHECK**
  "CHECK" block commands 1224
  The commands in this block define checks carried out automatically by the program to test the validity of the parameter values, either when defined or after they are revised in the bottom dialog box.
- / **PROP**
  "PROP" block commands 1224 (optional)
  The commands in this block specify the number of property groups that the wizard must prompt for.
- / **LOADS MENU**
  "LOADS MENU" block commands 1225 (optional)
  The commands in this block provide the program with instructions for prompting for loading data..
- / **COMMANDS**
  "COMMAND" block commands 1225
  This block contains the geometry definition commands in the "Command mode" format, but with variables (defined in the INIT block) in place of numerical values.
- / **LOAD COMMANDS**
  "LOAD COMMAND" block commands 1226 (optional)
  This block contains the load case definition commands in the "Command mode" format, but with variables (defined in the LOADS MENU and INIT block) in place of numerical values.
- / **END**

Refer to the Plane frame model 1228 for an example.

12.2.2.1 Model title lines

The wizard file must begin with a list of the models included in the file.

Each line contains data for one model. The format is as follows:
- column 1-30: model title
- column 61-68: name of bitmap file with small picture of typical model. The bitmap file cannot be defined for new models added by the user.

The list of models must terminate with a / END line.

For example, the plane frame file 1228 (PREFR1.DAT) begins with:

```
Plane frame            FRAMEA
Vierendeel             FRAMEB
Truss on columns       FRAMEC
/ END
```

Refer also to General syntax rules 1227.

12.2.2.2 "INIT" block commands

The INIT block must be the first block in the file and always starts with a "/ INIT" .command.

The commands in the block assign initial default values to the input parameters and define the variables required for the model definition.

There are four command formats available:

- *variable = expression*
  This command assigns default values to the model parameters or defines new model variables. For example, in the plane frame model:
  bays = 5
  num_dist = bays - 1
  Note that all length values should be multiplied by "UNITSFCT 1227" to convert the values to the default length units. In such cases the program assumes that the **default length values were defined in meters** (the program rounds off values after converting units).
- *variable = I "string"*
  This command adds a prompt with "*string*" to the dialog box that requests the model parameters. The parameter is defined as an integer value. For example, in the plane frame model:
  bays I "Number of bays ="
- *variable* = R *"string"*
  This command adds a prompt with *"string*" to the dialog box that requests the model parameters. The parameter is defined as a real value. For example, in the plane frame model:
  width R "Typical bay width ="
- LIST *variable(size_variable)*
  This command defines an array "variable" containing "*size_variable*" numbers. This command is used to define the list of distances for node generation commands. For example, if there are "*n*" bays in the plane frame model 1228, "*n-1*" distances are required for the node GRID command. The variable "*num_dist*" was defined to represent the "*n-1*" distances. The distance values are assigned to the array "*distx*" with the following command:
  LIST distx(num_dist)

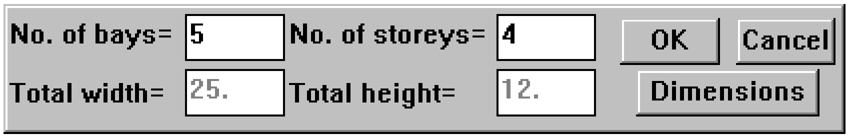
  Note that ***all*** terms in the array will be assigned with the same value; there is no method in the INIT block to assign different values to different terms in the array. Individual distances may be revised by clicking the Dimension button in the bottom dialog box after the initial model is displayed.

Refer also to General syntax rules 1227.

12.2.2.3 "MENU" block commands

The MENU block must be the second block in the file and always starts with a "/ MENU" .command.

The commands in the block define the variables that are displayed in the dialog box at the bottom of the screen after the model is created (i.e. after the user types in values for the model parameters). For example, the following dialog box is displayed for the plane frame model:



The command format is:

variable type "string"

where:

- "*variable*" must be a variable defined in the "INIT" block.
- "*type*" must be one of the following (according to the variable):

I - integer value
R - real value
IW - integer value, but do not allow value to be revised
RW - real value, but do not allow value to be revised

- "*string*" = the prompt for the variable in the dialog box

In the plane frame model 1228, the commands for "Number of bays" and "Total width" are:
bays I "No. of bays ="
total_w..RW "Total width ="

Note:
- The dialog box can contain 1 or two lines. The command
  "LINE 2"
  must be inserted in the file before the commands defining the variables in the second line.

Refer also to General syntax rules 1227.

12.2.2.4 "DIMENSION" block commands

The DIMENSION block must be the third block in the file and always starts with a "/ DIMENSION" . command.

The commands in the block define the dimension lines that are automatically drawn by the program when the model is displayed.

The command format is:

side variable list

where:
- *"side"* indicates the side of the model where the dimension line is drawn:
  D - below the model
  U - above the model
  L - to the left of the model
  R - to the right of the model
- "*variable_list*" includes a list of defined variables (may include LIST variables) that are to be included in the dimension line.

For example, the horizontal dimension line in the plane frame model 1228 is defined by the command:
D distx last_distx
where
- "distx" is the LIST variable with "n-1" bay dimensions
- "last_distx" is the "nth" bay dimension

Note:
- a maximum of 3 dimension lines may be defined on each *"side"*
- all dimension lines start at the (0,0) coordinate
- a maximum of 10 variable names may be included in "*variable_list*"
- all of the variables must be defined in the "INIT" block.

Refer also to General syntax rules 1227.

12.2.2.5 "CHECK" block commands

The CHECK block must be the fourth block in the file and always starts with a "/ CHECK" .command.

The commands in the block define checks carried out automatically by the program to test the validity of the parameter values, either when defined or after they are revised in the bottom dialog box.

The commands are defined in pairs:

1st line: *variable* $\left|\begin{smallmatrix}<\\>\end{smallmatrix}\right|$ *expression*

2nd line: "*string*"

where:

- "*variable*" is the variable (defined in the INIT block) to be checked
- "<" ">" defines the mathematical relationship
- "*expression*" is a numerical value or a program constant 1227
- "*string*" is the error message that is displayed if the check fails.

Note that if "variable" was not defined in the INIT block, it may be defined before "1st line" with the command:

variable = expression

For example, in the plane frame model 1228 the program must check that the total number of nodes does not exceed the program maximum (the total number of nodes must first be calculated). The commands are:

```
totn = ((bays +1 ) * (storeys + 1))
totn < MAXNODES
No. of nodes exceeds maximum
```

Refer also to General syntax rules 1227.

12.2.2.6 "PROP" block commands

The PROP block, if defined, must follow the CHECK block in the file and always starts with a "/ PROP " .command.

The commands in the block instruct the program to display the standard property dialog boxes for the property groups in the model.

The command format is:

n (E) "prompt"

where:

- *n* is the property group number
- *E* indicates an element property (default = beams)
- "*prompt*" = the dialog box header for the property group

Example:

Property group 7 is assigned to the corbel beams. Enter:

7 "Define the section for the corbel beams"

Note:

- this option always prompts for beam properties and cannot be used for elements.
- the user may "skip" over the property menus, however the "undefined" property groups will still be assigned to the beams as instructed in the / PROPERTY NUMBERS commands.

12.2.2.7 "LOADS MENU" block commands

The LOADS MENU block, if defined, must follow the last geometry block - PROP or CHECK and always starts with a "/ LOADS MENU" .command.

The commands in the block instruct the program how to prompt for load case information and define other variables required for load case generation.

Any number of menus may be created for convenience, i.e, the number of menus does not have to correspond to the number of load cases. There are three menus in the plane frame model: the first prompts for all vertical load data, the second prompts for all horizontal (wind) load data and the third prompts for combination data; eight load cases are created in the / LOAD COMMANDS section.

Each menu must start with the following header line:

** MENU*

There are two command formats available:

- *variable = expression*
  This command assigns default values to the load parameters or defines new load variables. For example, in the plane frame model:
  selfl = 1
  bay_min = bays - 1

- *variable* = R *"string"*
  This command adds a prompt with *"string*" to the dialog box that requests the load parameters. The parameter is defined as a real value. For example, in the plane frame model:
  deadl R "Dead load ="

Bitmaps cannot be displayed in user-define models.

12.2.2.8 "COMMAND" block commands

The COMMAND block must be the last block in the file (or prior to LOAD COMMANDS, if defined) and always starts with a "/ COMMAND" .command.

The block contains the geometry definition commands in the "Command mode" format, but with variables (defined in the INIT block) in place of numerical values.

The program calculates values for the variables, inserts them into the command and writes the command in the GEOINnnn.DAT file.

The variables in the command must be written with one of the following prefixes:
- indicates that the variable is an integer value
@ - indicates that the variable is a real value

Expressions must be enclosed in parentheses.

For example, the beam GRID command in the plane frame model 1228 is written as:

GRID #bays #storeys B 1 N 1 BY #(bays + 1) DEL 1 TO #bays

The command may be conditional on a variable being greater than zero; add to the start of the command:

? var

For example, the above command should be written only if there are more than 2 storeys. The commands should read:

stry = storeys - 2
? stry GRID #bays #storeys B 1 N 1 BY #(bays + 1) DEL 1 TO #bays

Refer also to the General syntax rules 1227.

Note:

The program writes the commands to the file when the user clicks OK in the bottom dialog box. However, the user may have changed the value of the some of the variables by editing the values in the dialog box !

Therefore, **all INIT commands defining variables that may be revised in the dialog box must be written again in the COMMAND block**; the program will recalculate the variable values before writing the command.

For example, in the plane frame model 1228 the user can revise the "Number of bays", represented by the "bays" variable. Therefore, the following commands are located at the start of the COMMAND block:

num_distx = bays - 1
num_disty = storeys - 1
total_w = distx + last_distx
total_h = disty + last_disty

12.2.2.9 "LOAD COMMANDS" block commands

The LOAD COMMAND block, if defined, must be the last block in the file and always starts with a "/ LOAD COMMAND" command.

The block contains the load definition commands in the "Command mode" format, but with variables (defined in the LOADS MENU block) in place of numerical values.

The program calculates values for the variables, inserts them into the command and writes the command in the STATnnn.DAT file.

Each load case must start with the following commands:

** LOAD (nm) (var_1 ... var_n)*
load_case_title

where:

nm : do not generate the load case if the user "skipped" over load menu no. "*nm*" (only 1 menu may be specified)
var_n : do not generate the load case if any of *var_1, .., var_n* is less than or equal to zero.

The load commands then follow:

The variables in the command must be written with one of the following prefixes:

\# - indicates that the variable is an integer value
@ - indicates that the variable is a real value

Expressions must be enclosed in parentheses.

For example, the dead load is applied to all beams in the plane frame model 1228 with the command:

DIST FX2 @deadlm B #(bays+1) TO #beams

The command may be conditional on a variable being greater than zero; add to the start of the command:

? var

For example, the above command should be written only if there are more than 2 bays. The commands should read:

bay = bays - 2
? bay DIST FX2 @deadlm B #(bays+1) TO #beams

Refer also to the General syntax rules 1227.

12.2.2.10 General syntax rules

Variables, operators, function, etc.

- lines starting with a "**;**" are comment lines and are ignored by the program
- All header lines must have at least one space between the '**/**' and the following text, e.g. '**/ INIT**'
- text must not be written after column 80. In cases where a command exceeds 80 characters, write a "**&**" at the end of the first line (leave at least one space before the '&') and continue the command on the following line.
- Variable names may have any length and must not include any of the following symbols:
  ( ) + - * / ; @ # > < & ,<space>
- Expressions that include the following mathematical operators may be used:
  ^ (exponent) **/** ***** **-** **+**
  (the above list also indicates the order of calculation)
  At least one blank must be entered on each side of the above operators.
- Parentheses **( )** may also be used
- The following functions may be used:
  SIN( ) COS( ) TAN( ) LN( ) ASIN( ) ACOS( ) ATAN( ) INT( )
- Loops may be created and must be in the format:
  ! DO
  ... commands ...
  ! UNTIL condition
  The loop is continued until "**condition**" is true, where "**condition**" must be in the form:
  variable **| = |** ***expression***
  | > |
  | < |

Note:

- the program does not differentiate between uppercase and lowercase letters in the variable names, e.g. **NODE** and **Node** refer to the same variable.

Program constants

Program constants are variables defined by the program; they may not be revised by the user:

The program constants are:

- MAXNODES - the maximum node number allowed
- UNITSFCT - a factor that identifies program default length unit. **All default length values defined in the INIT block must be multiplied by this program variable in order to be converted to the default length unit**

12.2.2.11 Plane frame example

Explanation:	**File data:**
Model titles: Model #1:	Plane frame FRAMEA
Model #2:	Vierendeel FRAMEB
Model #3:	Truss on columns FRAMEC
End of model list:	/ END
Model #1 title:	Plane frame
comment line:	;---
INIT block:	/ INIT
- model parameters:	bays = 5
default values	storeys = 4
	width = 5 * UNITSFCT
	height = 3 * UNITSFCT
- model parameters:	bays I "Number of bays="
menu prompts:	storeys I "Number of storeys="
	width R "Typical bay width="
	height R "Typical storey height="
- size of node DIST arrays:	num_distx = bays - 1
" " "	num_disty = storeys - 1
- define DIST arrays:	LIST distx(num_distx)
" "	LIST disty(num_disty)
- assign input values to ALL	distx = width
array terms:	disty = height
- define variables for last	last_distx = width
spacing (for Dim lines):	last_disty = height
- define variables for Dialog	total_w = distx + last_distx
Box:	total_h = disty + last_disty
MENU block:	/ MENU
1st line box:	bays I "No. of bays="
" "	storeys I "No. of storeys="
	LINE 2
2nd line box:	total_w RW "Total width="
" "	total_h RW "Total height="
DIMENSION block:	/ DIMENSIONS
Horizontal dimension line:	D distx last_distx
Vertical dimension line:	L disty last_disty
CHECK block:	/ CHECK
- max no, of bays:	bays < 100
warning:	Number of bays should not exceed 100
- max. no. of storeys:	storeys < 100
warning:	Number of storeys should not exceed 100
- Calc no. of nodes in model:	totn = ((bays + 1) * (storeys + 1))
Check if < program variable:	totn < MAXNODES
warning (if not):	no. of nodes exceeds maximum
PROP block:	/ PROP
Prompt for properties of 2 groups -	1 "Define the beam section"

beams and columns (/ PROP GRP commands below)	2 "Define the column section"
LOADS MENU block: First menu: all vertical loads * No bitmaps in user models * - Prompt for dead load - Prompt for live load - Default self-weight factor - Prompt for self-weight - Initialize menu 2,3 variables in 1st menu because other menus may be "skipped"	/ LOADS MENU * MENU BITMAP PLD11 deadl R "Dead load =" livel R "Live load =" selfl = 1 selfl R "self weight factor =" wind_t = 0 bay_min = bays - 1
Second menu: ...	* MENU .....

COMMAND block:
- Recalculate these variables as user may have revised values in bottom Dialog box:
NODE commands:
- plane model (2 coord only):
- define variables: node nos at end of base line & ht line:
- #= integer var; @= real var:
SUPPORT commands:
- pinned supports:
PROP GROUP commands:
- calc no. of beams in model:
- Prop. group 1 = beams
- calc total no. of members:
- Prop. group 2 = columns:
BEAM commands:
- define grid
	/ COMMANDS
	num_distx = bays - 1
	num_disty = storeys - 1
	total_w = distx + last_distx
	total_h = disty + last_disty
	/ JOINT COORDINATES
	COORD 2
	end1 = bays + 1
	end2 = end1 * (storeys + 1)
	1 0 0 TO #end1 @total_w 0 DIST @distx TO #end2 @total_w @total_h DIST @disty
	/ RESTRAINTS
	X1 X2 1 TO #end1
	/ PROPERTY NUMBERS
	beams = bays * (storeys + 1)
	1 1 TO #beams
	col_end = beams + storeys * (bays + 1)
	2 #(beams + 1) TO #col_end
	/ MEMBER INCIDENCES
	GRID #bays #storeys B 1 N 1 BY 1 BY #(bays + 1) DEL 1 TO #bays

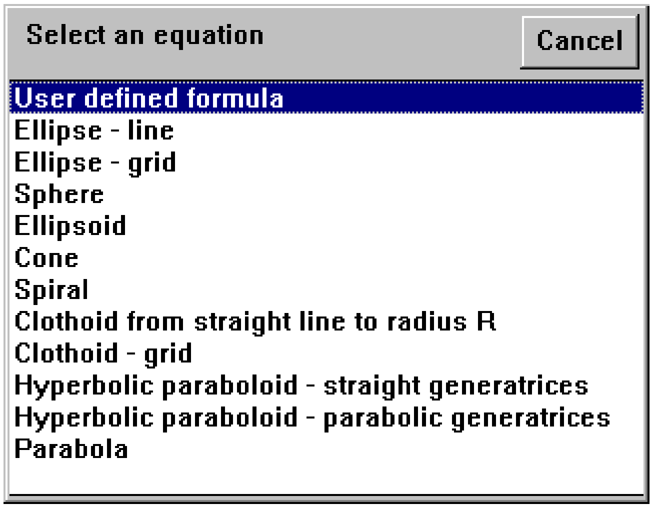
LOAD COMMANDS block
- 1st load case:
- load case title
- define beam loads
- reverse sign of dead load
- uniform bm load command
- 'self2' always => 0.
- apply s-w only if factor defined
	/ LOAD COMMANDS
	* LOAD
	DEAD LOAD
	/BEAM LOAD
	deadlm = 0 - deadl
	DIST FX2 @deadlm B #(bays + 1) TO #beams
	self2 = selfl * selfl
	? self2 SELF X2 @selfl B #(bays + 1) TO #col_end
	.....

3rd case: create only if menu 2 not skipped **and** wind_t>0	* LOAD 2 wind_t .....
End of model:	/ END
Model #2	Virendeel ;--

/ INIT
.
etc ...

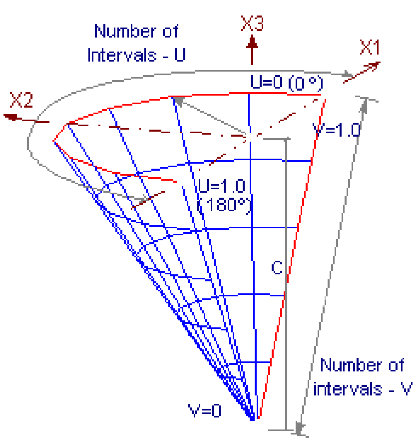
12.2.3 Nodes - equations

Select any of the geometric shapes in the menu for an explanation of the parameters and variables.



Note:
- only Ellipse - line contains an explanation on the program menu

12.2.3.1 Cone



Cone equation: $\mathbf{X1^2 + X2^2 = (R\ X3/C)^2}$

Program formulae:

$\mathbf{X1 = R\ V \cos(\pi\ U)}$
$\mathbf{X2 = R\ V \sin(\pi\ U)}$
$\mathbf{X3 = C\ V}$

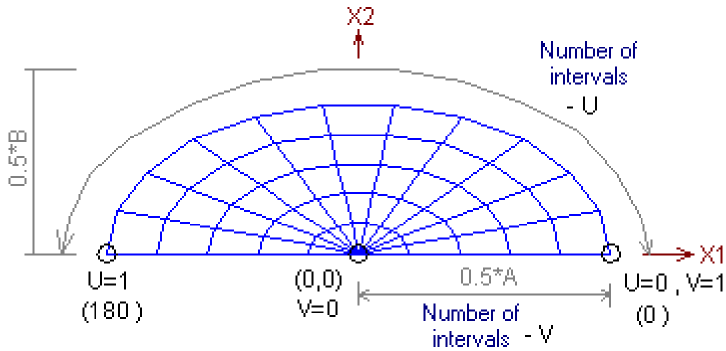
Refer to Ellipse - line for a detailed example of node definition by equations.

12.2.3.2 Ellipse - grid

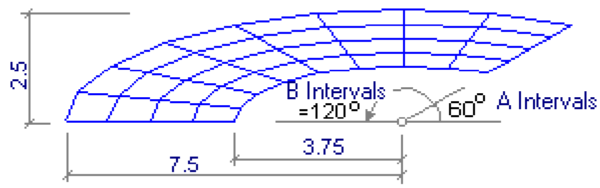
Ellipse grid equation: $\mathbf{X1^2/(A/2)^2 + X2^2/(B/2)^2 = V^2}$

Program formulae:

$\mathbf{X1 = 0.5\ A\ V \cos(\pi U)}$
$\mathbf{X2 = 0.5\ B\ V \sin(\pi U)}$



Example:



A = 15.0 B = 5.0
U: Start = 0.333 End = 1.0 Intervals = 8
V: Start = 0.5 End = 1.0 Intervals = 4

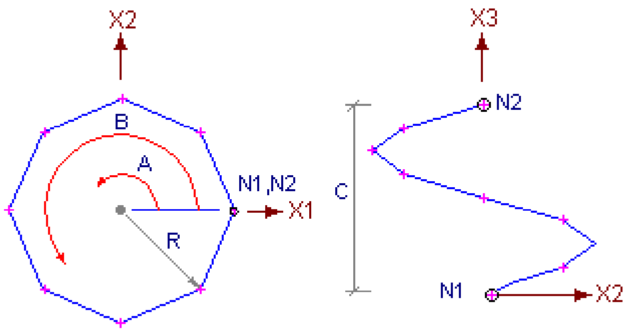
Refer to Ellipse - line for a detailed example of node definition by equations.

12.2.3.3 Spiral

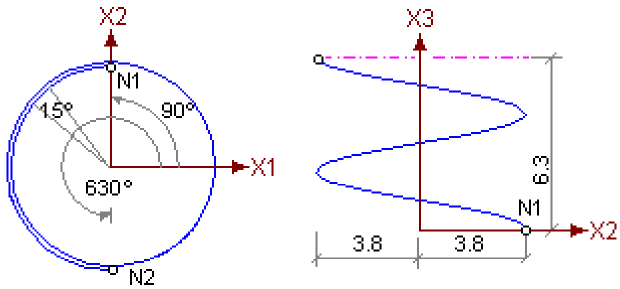
Program equations:

X1 = R cos[π/180 {A+U (B-A)}]
X2 = R sin[π/180 {A+U (B-A)}]
X3 = U C

- Each complete turn of the spiral is equivalent to 360°, i.e. for 2 complete turns, A = 0°, B = 720°.
- Always define the range of U from 0. to 1.
- C is the **total height** of the spiral, not the height of a turn.



Example:



Angle at N2 = 360+270 = 630
No. of intervals = (630-90)/15 = 42

Enter:
A = 90 B = 630 C = 6.3 R = 3.8
U: Start = 0. End = 1. Intervals = 42

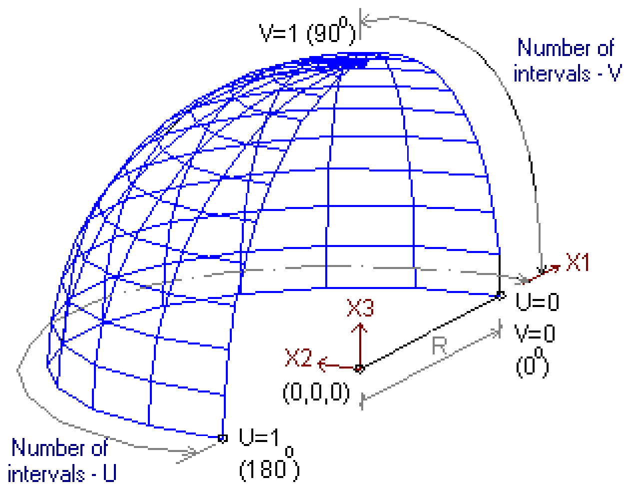
Refer to Ellipse - line for a detailed example of node definition by equations.

12.2.3.4 Sphere

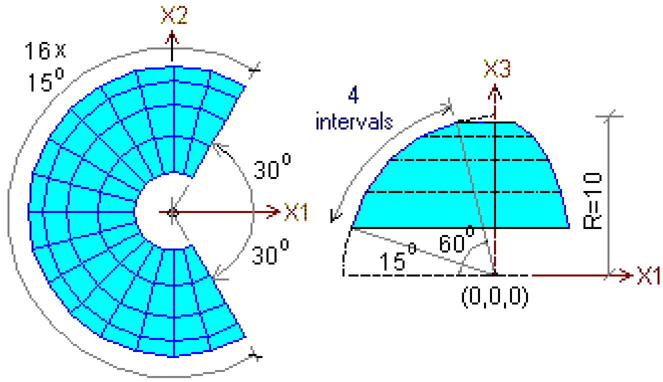
Sphere equation: $X1^2 + X2^2 + X3^2 = R^2$

Program formulae:

$$X1 = R \cos(\pi/2\ V) \cos(\pi\ U)$$
$$X2 = R \cos(\pi/2\ V) \sin(\pi\ U)$$
$$X3 = R \sin(\pi/2\ V)$$



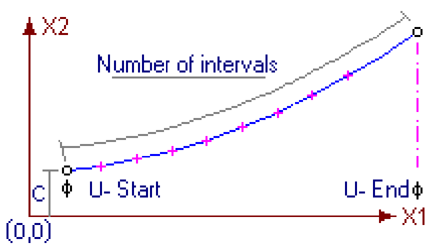
Example:



R = 10.
U: Start = 0.1667 End = 1.8333 Intervals = 16
V: Start = 0.1667 End = 0.8333 Intervals = 4

Refer to Ellipse - line for a detailed example of node definition by equations.

12.2.3.5 Parabola



The program formulae are:

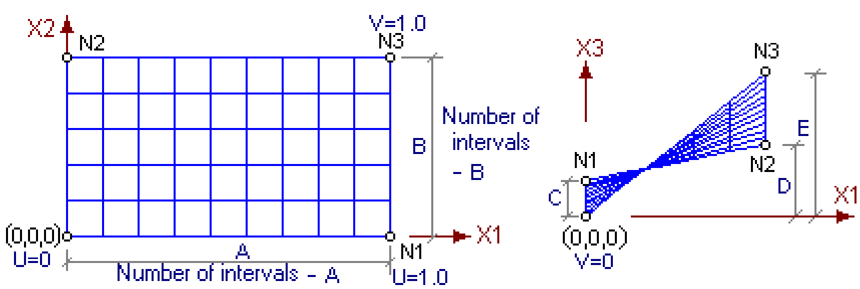
X1 = U
X2 = A U² + B U + C

Refer to Ellipse - line for a detailed example of node definition by equations.

12.2.3.6 Hypar - straight

The program formulae are:

X1 = A U
X2 = B V
X3 = V D + [C+V (E-C-D)]



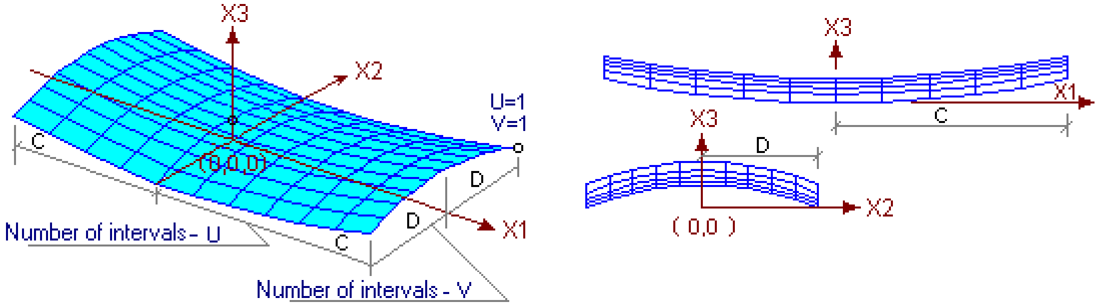
Refer to Ellipse - line for a detailed example of node definition by equations.

12.2.3.7 Hypar - parabolic

The program formulae are:

X1 = C U
X2 = D V
X3 = U² C²/A² - V² D²/A²

Note that the model is drawn from U = -1 to 1; V = -1 to 1 for clarity.



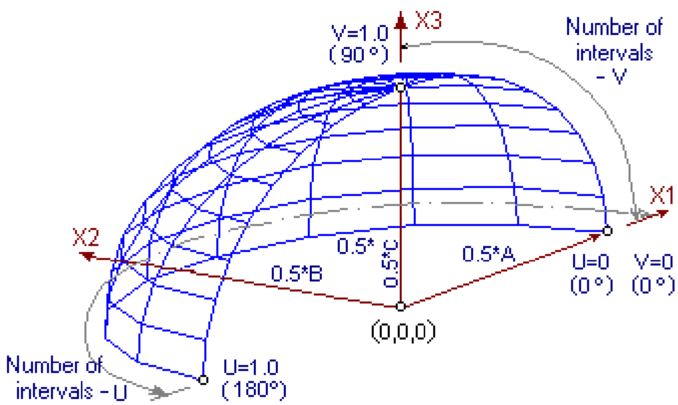
Refer to Ellipse - line for a detailed example of node definition by equations.

12.2.3.8 Ellipsoid

Equation: **X1²/A² + X2²/B² + X3²/C² = 1**

Program formulae:

X1 = A cos(π/2 V) cos(π U)
X2 = B cos(π/2 V) sin(π U)
X3 = C sin(π/2 V)



Refer to Ellipse - line for a detailed example of node definition by equations.

12.2.3.9 Clothoid

The clothoid represents a gradual transition from a curve with a specified radius to a straight line (radius = infinity). The program formulae are:

Clothoid:

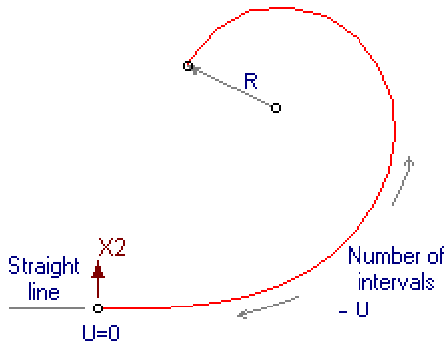
$$X1=\sqrt{\pi}A\int_0^{UA/\sqrt{\pi}R}\cos\frac{\pi}{2}x^2dx$$

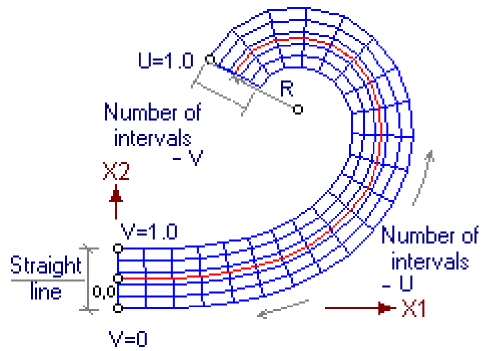
$$X2=\sqrt{\pi}A\int_0^{UA/\sqrt{\pi}R}\sin\frac{\pi}{2}x^2dx$$

Clothoid - grid:

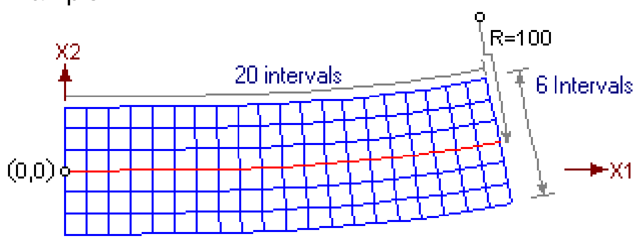
$$X1=\sqrt{\pi}A\int_0^{UA/\sqrt{\pi}R}\cos\frac{\pi}{2}x^2dx+(0.5-V)B\sin\left(\frac{(AU/R)^2}{2}\right)$$

$$X2=\sqrt{\pi}A\int_0^{UA/\sqrt{\pi}R}\sin\frac{\pi}{2}x^2dx+(0.5-V)B\cos\left(\frac{(AU/R)^2}{2}\right)$$





Example:



Radius = 100.   Assumed clothoid constant = 64.

Enter:

A = 64. B = 6.0 R = 100.

U:	Start = 0.0	End = 1.0	Intervals = 20
V:	Start = 0.0	End = 1.0	Intervals = 6

Refer to Ellipse 1230 for a detailed example of node definition by equations.

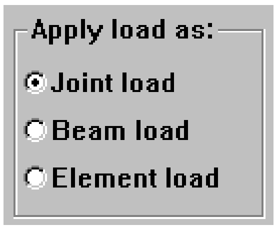
12.3 Loads

Select one of the following:

Global loads - method of application 1236

12.3.1 Global loads - Method of Application

Global loads may be applied to the model as joint loads, beam loads or element pressure loads.

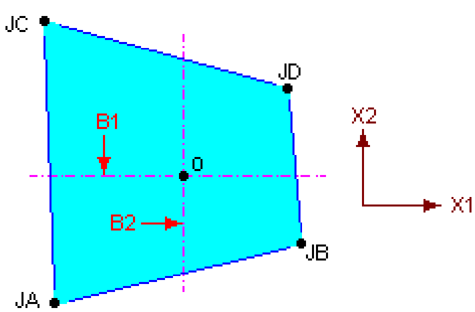


12.3.1.1 Applied as joint loads

Point loads applied as joint loads

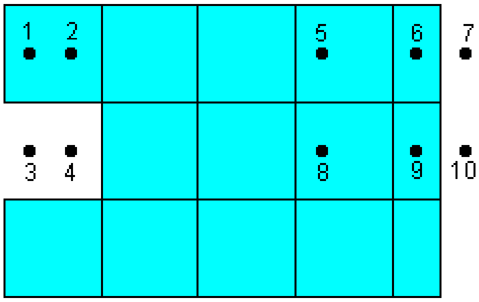
The figure below shows a point load applied at O located on the surface of the model.

- the program creates two imaginary beams (B1,B2) running through O and lying parallel to the global X1 and X2 axes.
- the program then searches for the node closest to O in each of the four quadrants created by B1,B2 and creates four imaginary support beams (JA-JB,JB-JD,JA-JC,JC-JD) lying between these nodes.
- the program calculates the reactions of beams B1,B2 from the point load at O and applies them to the four support beams. The reactions of these beams at the nodes JA,JB,JC and JD are calculated and applied as joint loads to the model. If ☑ **Apply moments due to load distance** is selected, then ***all six beams are assumed to have fixed ends, i.e moments are applied to the support nodes in addition to forces***.



The program ignores loads which lie outside the boundaries of the model, i.e. which are not surrounded by at least three nodes (nodes not connected to the model are ignored).

The figure below shows a pattern of wheel loads recalled from a file and applied as global loads. Loads at points 1,2,5,6,8,9 are within the model boundaries and will be applied to the model. The loads at points 7 and 10 are not applied. The loads at points 3 and 4 are outside the boundaries, but are surrounded by nodes, and so are applied to the model as explained above.



Area loads applied as joint loads

The program divides the load area into small quads and calculates an equivalent point load applied at the center of each quad. These point loads are then applied to the structure as explained above in **Point loads applied as joint loads**. The density of the load quads may be defined by the user in the Command Mode only.

Note:

- If the load area boundary does not coincide exactly with the edges of the model, small discrepancies will result in the equivalent point load calculations.

12.3.1.2 Applied as beam loads

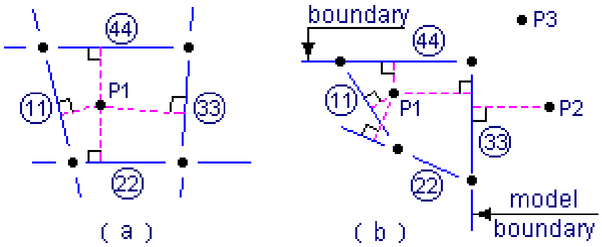
Point load applied as beam load

Referring to figure (a) below, a point global load P1 is applied to the surrounding beams inversely proportional to the lengths of the perpendiculars drawn from the load to these beams.

In the example shown in figure (b) below, a perpendicular cannot be drawn to beam 22 and so none of load P1 is applied to this beam, even if beam 22 is included in the "Apply load to selected beam/elem" option.

Loads falling outside the model boundary are treated in a similar fashion. For example, in figure (b), all of load P2 is applied to beam 33, while load P3 is ignored by the program.

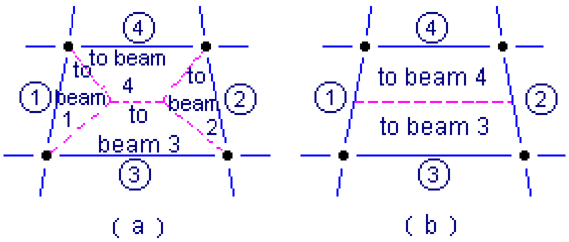
In the example of figure (a), the entire load P1 is applied to beam 44 if only beam 44 is seelcted in the "Apply load to selected beam/elem" option.



Area load applied as beam load

The program translates the global area load to linear beam loads as follows:

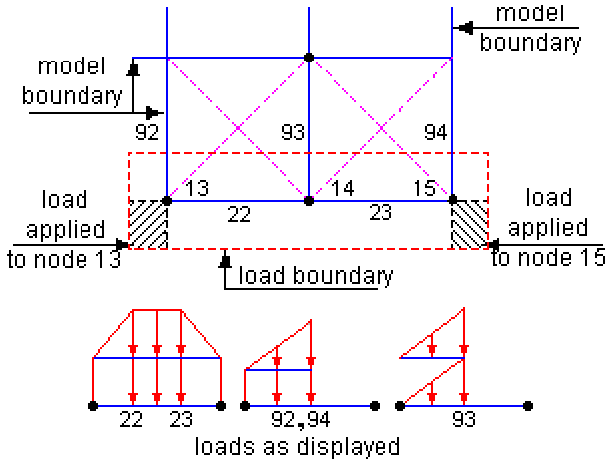
- Referring to figure (a) below, the program bisects the angle between each beam and the surrounding beams to create areas to be allocated to each beam. If a beam is not selected in the "Apply load to selected beam/elem" option, then the program ignores that beam, as in the example of figure (b) below, where beams 1 and 2 are not in the list.



Note:

The entire global area load is always applied to the model:

- Loads not completely covering a space and loads lying outside the model boundary are treated according to the rules explained above.
- Loads on areas that are closer to a node than to any of the adjacent beams are applied as joint loads. For example:



12.3.1.3 Applied as element loads

Point loads applied as element loads

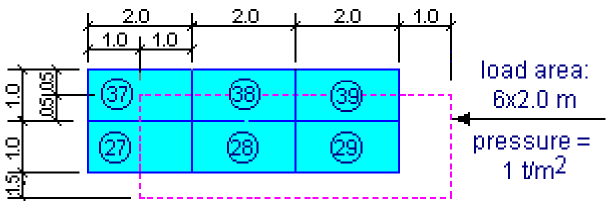
Point global loads applied as element loads are actually applied as Joint Loads as described in [Global point loads applied as joint loads](1236), but ***without*** the additional applied moments at the nodes.

Only point loads falling within an element boundary are applied.

Area loads applied as element loads

The program applies the area load as element pressure on all elements lying under the area load boundary. Loads on elements not selected in the "Apply load to selected beam/elem" option are ignored. See the figure below for a typical example.

If the area load only covers part of a specific element, the total load on the element is applied uniformly to the ***entire*** element surface, thereby reducing the element pressure. Loads lying outside the model boundary are ignored. For example:



The total load of 12 tons is allocated as follows:

• elements 28,29	:	1.00 t/m²	= 4.00 t
• elements 27,38,39	:	0.50 t/m²	= 3.00 t
• element 37	:	0.25 t/m²	= 0.50 t
• ignored	:	(2.0 + 3.0 - 0.5) = 4.50 m²	= 4.50 t

12.4 Results

Select one of the following:
BCF.DAT 1240

12.4.1 BCF.DAT

The allowable compression stress factors for all effective buckling lengths in axially loaded beams are contained in file BCF.DAT.

The file contains the data for 13 different types of structural steel. The user may update the file or may add additional steel types using the computer's editor.

The file format is:

```
  13
ST37      1           16000.0      -14000.0
ST52      1           16000.0      -14000.0
PIPE37    1           16000.0      -14000.0
PIPE52    1           16000.0      -14000.0
G43       2           15500.0      -15500.0
G50       2           21500.0      -21500.0
G55       2           26500.0      -26500.0
MAIN36    2           15200.2      -15200.2
BRAC36    2           15200.2      -15200.2
MAIN42    2           17733.6      -17733.6
BRAC42    2           17733.6      -17733.6
MAIN50    2           21111.5      -21111.5
BRAC50    2           21111.5      -21111.5
ST37
  100  100  101  101  101  101  102  102  102 102
  103  103  103  104  104  104  104  105  105 105
   106 106  106  106  107  107  107  108  108 108
   .
   .
   863 870  878  885  892  900  907  915  922 930
   937 945  953  960  968  976  984  991  999 1007
   1015
PIPE37
  .
  .
```

where:

line 1:
The number of steel type tables contained in the file. Format: I4

lines 2 - 14:
For each steel type:
- The steel type name.
- Type of code calculation:
  1: indicates that the code increases the actual axial stress by the factor in the table to take into

account the buckling effect. (German code)

2: indicates that the code decreases the allowable stress. (American, British codes)

- The allowable tension stress (units = ton/m²)
- The allowable compression stress when kl/r = 0.

Format: A6,I2,F9.1,1X,F9.1

line 15:
Steel type name of first table. Format: A6

lines 16 - 39:
Stress factors (*100) for all kl/r from 0 to 250.
Format (for each row): 10I5

Lines 15 to 39 are repeated for each steel type.

Examples:

- **Code Type 1 (German code)**:
  The program calculates the effective stress for any kl/r as follows:

  $F_{eff} = F_{actual} * K / 100$

  where K = number in table corresponding to kl/r of member.

  Example:
  kl/r = 26 ; corresponding no. in table = 107
  Steel type ST37
  Actual compression stress = 10750 t/m²
  F_{eff} = 10750 * 107 /100 = 11503 t/m²

- **Code Type 2 (British and American Codes)**:
  The program calculates the allowable compression stress for any kl/r as follows:

  $F_{all} = F'_{all} / (K / 100)$

  where:
  F'_{all} = F_{all} when kl/r = 0
  K = number in table corresponding to kl/r of member.

  Example:
  kl/r = 26 ; corresponding no. in table = 107
  F'_{all} = 15500
  F_{all} = 15500.0 / (107/100) = 14486 t/m²

12.5 Miscellaneous

12.5.1 Installation notes

Permanent files with user defined data:

The following data files are supplied with the program but data may be revised or added by the user during the course of his work:

PREFR1.DAT, PREFR2.DAT, PREFR3.DAT, PRETR1.DAT: data for "Model wizard" models

PATTERN.DAT	-	global load load patterns
UDAMPS.DAT	-	spectra for dynamic seismic analysis
CHESS.DAT	-	staggered load patterns
FORM.DAT	-	ode "equation" data
BCF.DAT	-	buckling data for "Beam axial stresses" in Results
USERSECT.DAT	-	cold-formed section type data
VEHICLES.DAT	-	vehicle loads for bridge module
LOADFCT.DAT	-	length vs. load factor table for bridge module

- These files are copied to the program directory from the installation set when the program is reinstalled, i.e. the revised/new data is lost.
- The files may be copied manually by the user from one computer to another.
- The files should be backed up periodically.

The file PROPTABS.DAT containing the user defined steel table is not found on the installation set; it should backed up periodically.

STRAP.INI

This file contains data on tabular print styles; screen colours and plot pen widths.

- This file will not be copied to the program directory from the installation set when the program is reinstalled, i.e. the revised/new data will not be lost.
- The file may be copied manually by the user from one computer to another.
- The file should be backed up periodically.

Date format:

STRAP uses the date format set in the Windows "Control panel" - "International - Short date format" option.

WIN.INI

The installation program adds the following lines to **WIN.INI** in directory \Windows or \Win95:

[STRAP]
PATH=*programdir*
LAST=*lastuserdir*

where:

programdir	=	the program folder, normally \STRAP1. If you install *STRAP* again in a different program folder, the installation program will revise *programdir*
lastuserdir	=	the current working folder. This line is updated by the program every time a different folder is specified in "Change current folder")

12.5.2 Print the manual

The entire *STRAP* manual is supplied with the program in Adobe 'PDF' format files. All text in the "Help" is included in the printed manual.

To print the manual, use Adobe 'Acrobat Reader' program (copy from the *STRAP* Installation CD or download from the Adobe internet site).

The manual is contained in a series of files:

File:	***Contents***:
Main manual:	
STRAP.PDF	*STRAP* manual, except for the following:
CODE_UK.PDF	BS5950, BS8110
COD_US.PDF	AISC (LRFD, ASD), AASHTO (LRFD, ASD), AISI, ACI318
CODE_CDN.PDF	CSA S16.1, A23.3, C136
CODE_EUR.PDF	Eurocode2, Eurocode 3, Eurocode 4
CODE_IND.PDF	IS:800, IS:456
CODE_CHN.PDF	GBJ17
CODE_BRZ.PDF	NBr6118, NBr8800
FOUND.PDF	Footing postprocessor
CROSEC.PDF	Compute section properties program
Miscellaneous:	
CMD_MODE.PDF	Command mode manual
DEMO_USA.PDF	Demo and Tutorial manual - American units (ft-kip)
DEMO_MET.PDF	Demo and Tutorial manual - Metric units (kN-meter)
VERIF.PDF	Verification manual
SHORT_MAN.PDF	The "short" printed manual supplied with the program
SMAN_EX.PDF	The detailed examples supplied with the program

12.5.3 Disclaimer

The *STRAP* programs have been written by a team of highly qualified engineers and programmers and have been extensively tested. Nevertheless, the authors of the software do not assume responsibility for the validity of the results obtained from the programs or for the accuracy of this documentation.

The user must verify his own results

The authors remind the user that the programs are to be used as a tool for structural analysis and design, and that the engineering judgment of the user is the final arbiter in the development of a suitable model and the interpretation of the results.

Windows is a registered trademark of Microsoft Corp.
Autocad is a registered trademark of Autodesk Inc.

Part XIII

Utilities

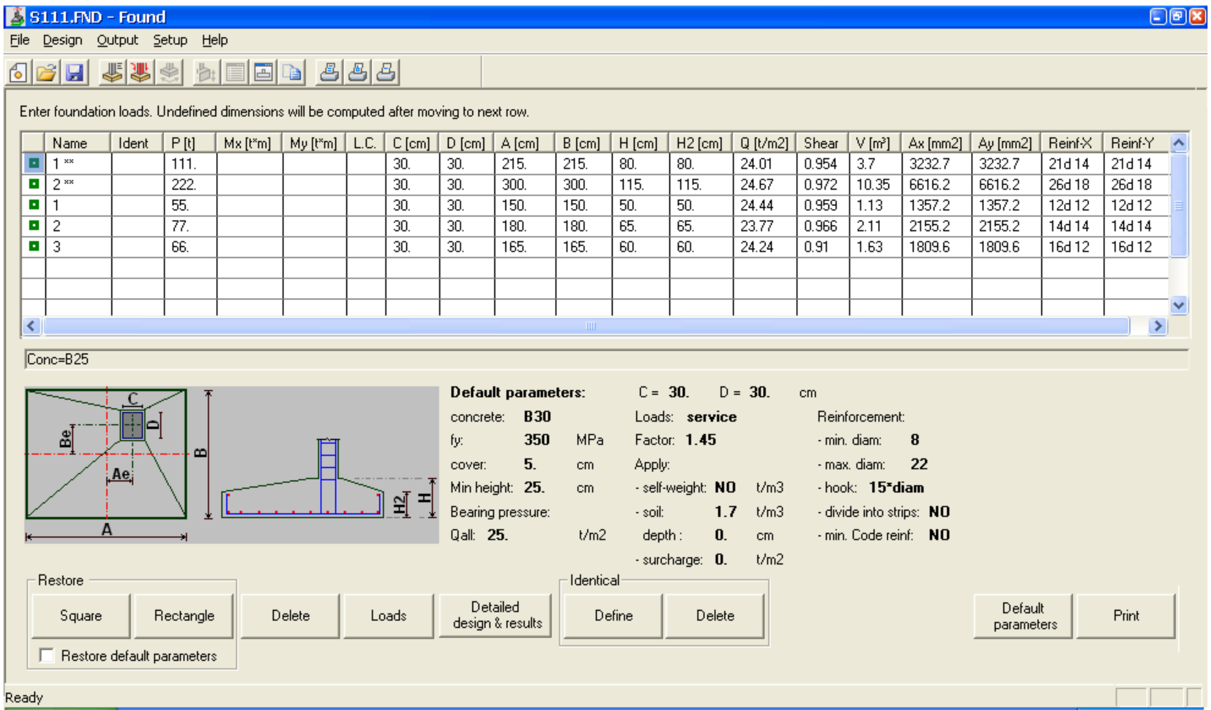
13 Utilities

13.1 Footing Design

13.1.1 Main menu

The main menu displays the "Project Table", a summary list of all foundations in the current project.

Refer also to:
General 1248
How to use FOUND 1249



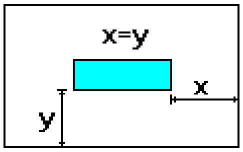
Restore

This option has two functions:

- convert a square footing to a rectangular footing (when the column is rectangular) or back to a square footing, using the current parameters for the selected footing.
  - Highlight the line
  - click Square or Rectangle.
- restore the default parameters for the footing.
  - Highlight the line
  - Restore default parameters
  - click Square or Rectangle.

Note:

- rectangular footings: the distance from the edge of the column to the edge of the footing in both directions is identical.



Delete

Delete one or more footings:
- click and highlight the lines in the table.
- click Delete.

Loads

Define additional load cases 1259 for the selected footing.

Detailed design and results

- revise individual parameters for the selected footing
- design an eccentric footing
- design a sloped footing
- display and print a drawing of the footing

Refer to Detailed design 1251.

Identical

Select a group of foundations and specify them as "identical":
- All foundations in the group will have the same dimensions and height.
- The foundations may have different column dimensions.
- the parameters from the first footing in the list will automatically be assigned to all other footings in the list.

Refer to Identical 1252.

Data table

Enter loads, footing dimensions and column dimensions may be specified:
- Loads: service or factored according to the option specified in Parameters 1256
- Footing dimensions (A,B): optional:
  - the program calculates a square footing if the dimensions are not defined
  - the program calculates the minimum second dimension required if only A or B are defined
  - the program calculates the remaining values (height, Q, etc) if both A and B are defined
- Column dimensions: the program uses the values specified in Parameters 1261 if values are not defined here.

Parameters

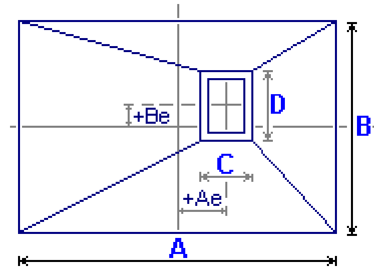
Specify default parameters 1254 for the footings in the project.

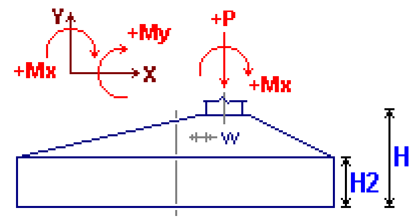
Note:
- the default parameters are automatically assigned to new footings in the Project list.
- to define different parameters for a specific footing, highlight the line in the table and click
  Detailed design & results

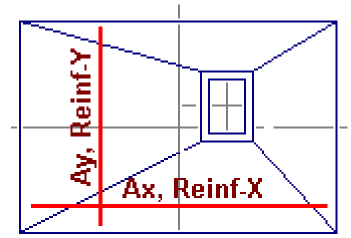
13.1.1.1 Project table

The following items are displayed in the Project table:

Name = Footing name: numbered automatically in ascending order but may be edited.
Ident = if present - name of the first footing in the identical group
P = Axial load in design load case
Mx = Moment about X in design load case
My = Moment about Y in design load case
L.C. = Design load case/total number of load cases
C = Column dimension parallel to A
D = Column dimension parallel to B
A = Footing length
B = Footing width
H = Height of footing adjacent to column
H2 = Height of footing at edge
Q = Max soil pressure (service)
Shear = max(shear,punching)/allowable shear stress
V = Volume
Ax = Reinforcement area - X
Ay = Reinforcement area - Y
Reinf-X= Reinforcement detail - X
Reinf-Y= Reinforcement detail - Y





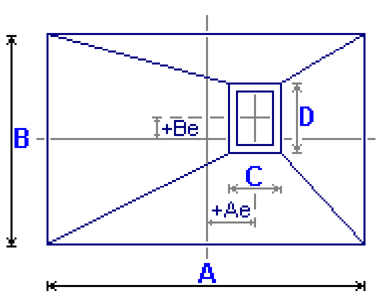


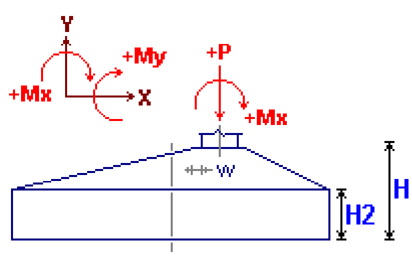
13.1.2 General

Program ***FOUND*** designs reinforced concrete spread footings. The program features are:

- design for axial force and biaxial moments (in same load case)
- eccentric column allowed in both directions
- multiple load cases (up to 25)
- automatic design of base dimensions and reinforcement according to Code requirements
- automatic design of height according to punching and shear, including sloped upper surface
- automatic design and detailing of reinforcement according to Code requirements
- user defined dimensions may be checked
- graphic output of footing - plan and section - to scale and in colour
- groups of identical footings 1252 may be designed
- DXF file of graphic output may be created

The following symbols are used by the program:





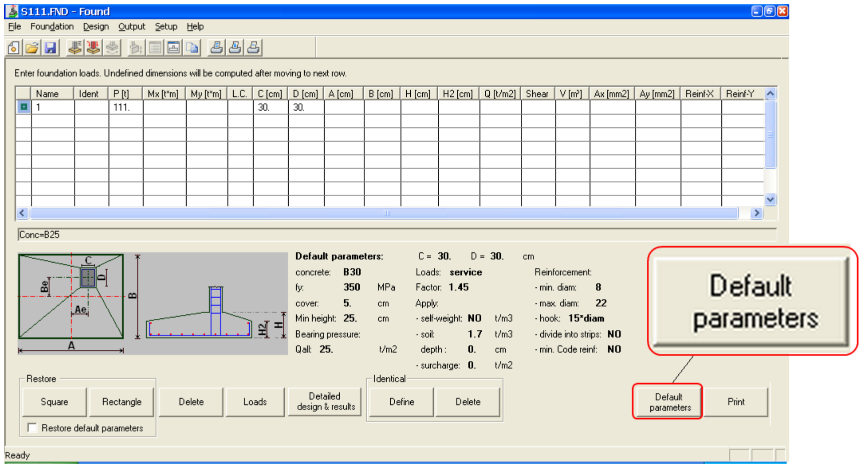
Note the sign conventions for:

- positive eccentricity of the column about the center of the base
- positive applied moments and axial load.

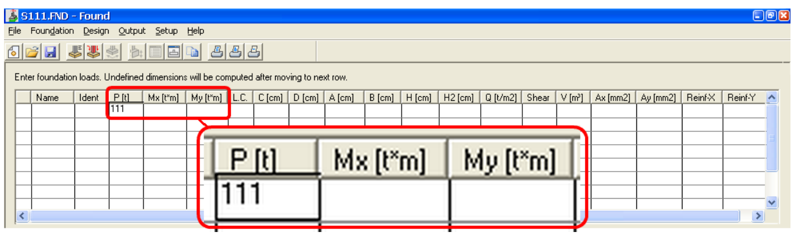
13.1.2.1 How to use FOUND

The "Project table" a list of all the foundations in the current project, is displayed on the screen. The following explains the basic steps for designing a footing:

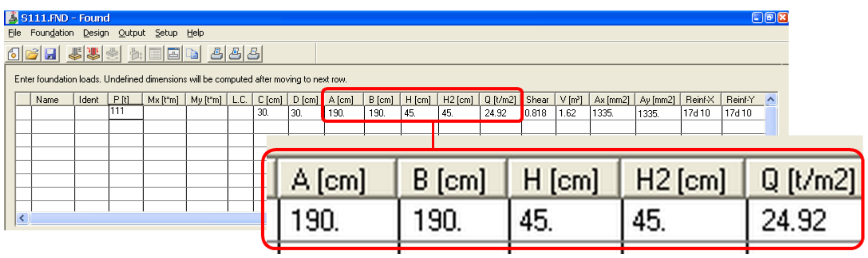
- set the default parameters [1254] for the entire project:



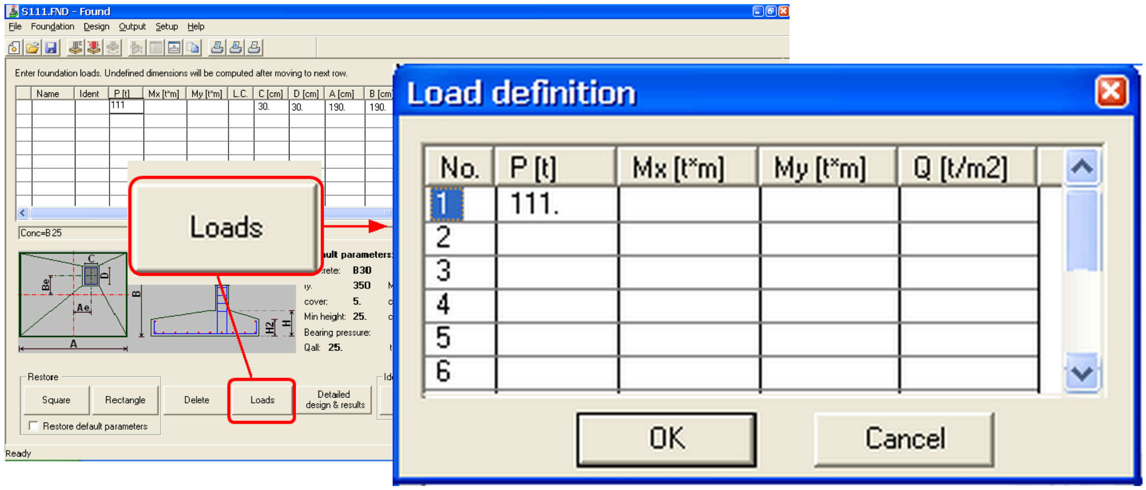
- click the 1st line in the table and enter a load case:



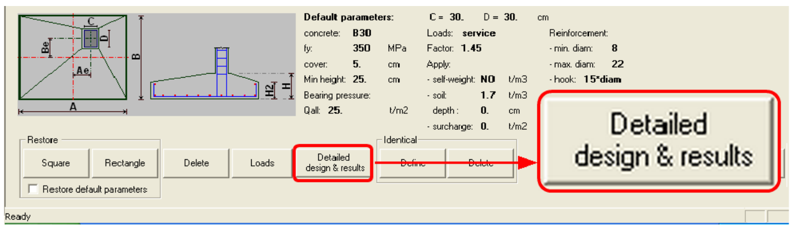
- Move the cursor down to the next line: the program automatically calculates a square footing:



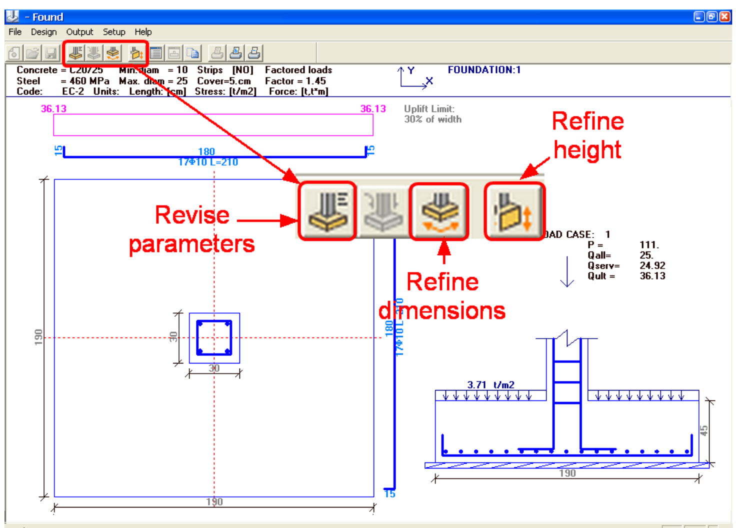
- Add additional load [1256] cases:



- Define individual parameters 1254 for this footing, display and print a drawing of the footing:



The program displays the following screen:



- return to the "Project table":

- Save the results:

  Select **File** in the Menu bar and the **Save** option or click the  icon.

Footings transferred from STRAP:

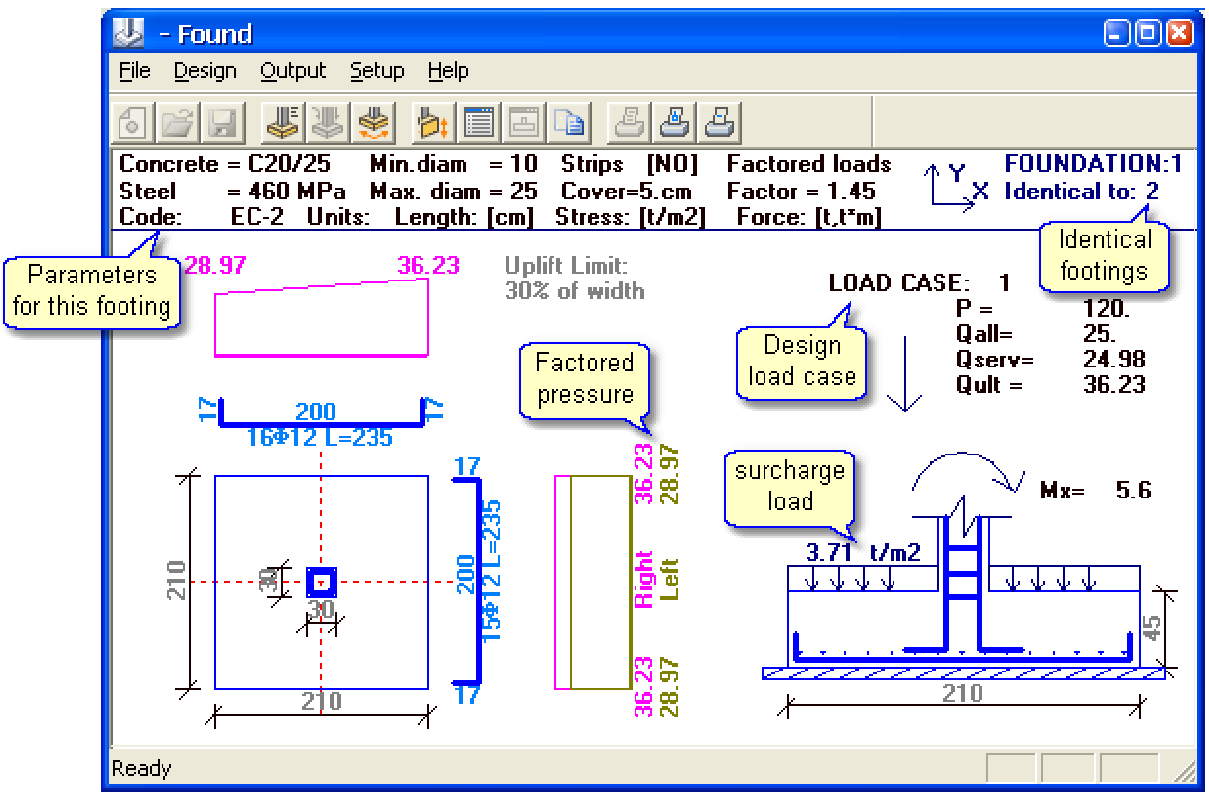
The *STRAP* footing postprocessor designs rectangular spread footings at all nodes assigned with restraints (and springs, if specified by the user).

The postprocessor automatically retrieves from *STRAP*:
- the reactions for all *STRAP* load combinations at these nodes (force and moments)
- the column dimensions (if possible) from the section of the member attached to the node.

Note:
- the program always transfers the column dimensions as a rectangle that bounds the actual section dimensions. The loads are assumed to act in the center of this rectangle.
- for columns defined by properties (A,I), the program transfers zero dimensions and the postprocessor begins the design with a default dimension that may be revised by the user.
- either Service or Factored reactions may be transferred from *STRAP*. Set the load type parameter 1256 accordingly.

13.1.3 Detailed design

The program displays a drawing of the selected footing and the current parameters:



The options available are:

- revise parameters 1254 for this footing only: click 
- revise base dimensions 1261 (A,B) or define an eccentric footing: click 
- revise the footing height 1261 or define a sloped footing: click 
- print 1265 the display: click 

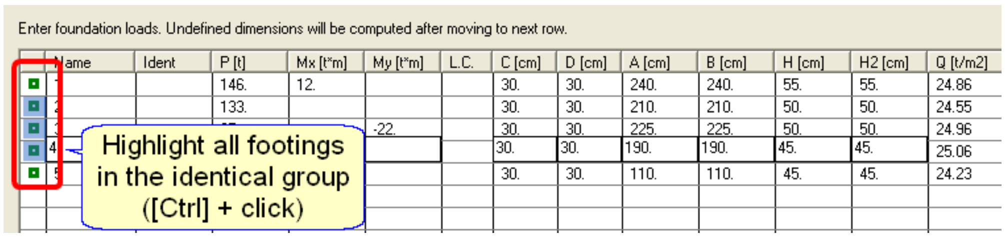
13.1.4 Identical

Select a group of foundations and specify them as "identical":

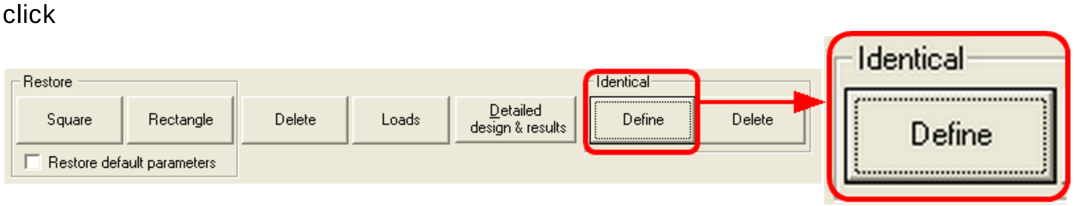
- All foundations in the group will have the same dimensions and height.
- The foundations may have different column dimensions.
- the parameters from the first footing in the list will automatically be assigned to all other footings in the list.

To create an identical group:

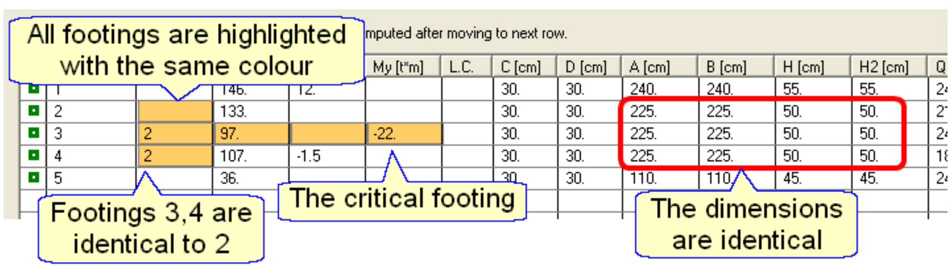
- click and highlight all foundations in the group:



- click



- The program redesigns all footings in the group:



Note:

- to add a footing to an existing group, highlight that footing and one of the existing footings in the group; click Define.
- to combine two groups, highlight one footing from each of the groups and click Define.

To remove footings from an identical group or to delete a group entirely:

- click and highlight the relevant footing (highlight ***all*** footings to delete a group entirely)
- click 

13.1.5 File options

New project

Open a new footing file. If you have not saved footings defined in the current session, the program asks whether to save them in the current file before opening the new one (the footings are erased if you do not save them).

The program prompts for the file name only when you select **Save**, **Save as** or **Exit**.

Open project

Open an existing file. The program displays the list of footing files in the current folder:
- double-click on one of the file names to open it, or -
- type in the file name in the **File name** edit box and click Open
- use the << >> options at the top of the dialog box to select a different folder or open a new folder.

Save project

Save all footings created in the current session in the current file (the file name is displayed at the top of the screen).

Save as

Save all footings in a **New** file.

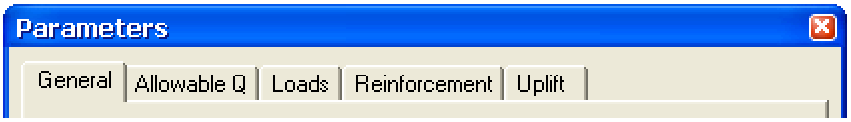
- type the file name in the **File name** edit box and click Save.

Exit

Exit from the program.

13.1.6 Design options

13.1.6.1 Parameters



Refer to:
- General 1254
- Allowable Q 1255
- Loads 1256
- Reinforcement 1257
- Uplift 1258

13.1.6.1.1 General

Click options where the ☝ *is displayed for more detailed Help:*

Project title

Enter a project title, i.e. a title describing the current project.

Concrete

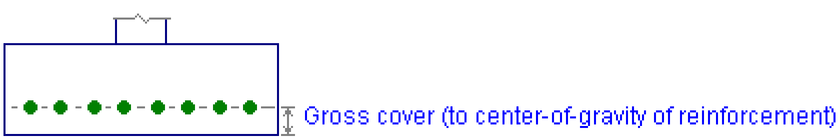
Select a concrete type from the list box. Concrete types and parameters may be revised in the Setup 1267 option.

Steel grade

Define the grade of the reinforcement steel according to the units displayed adjacent to the text box.

Cover

Define the gross concrete cover (to the center of gravity of the reinforcement)



Minimum height

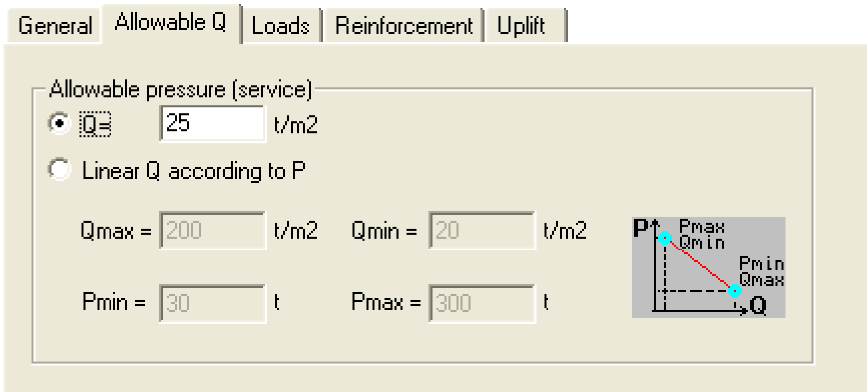
Specify a minimum footing height.

Column dimensions

Enter the column dimensions according to the units displayed adjacent to the text boxes (the units may be revised in the Setup 1267 option).

Note:
- column dimension **C** is parallel to footing dimension **A**
- column dimension **D** is parallel to footing dimension **B**.

13.1.6.1.2 Allowable Q

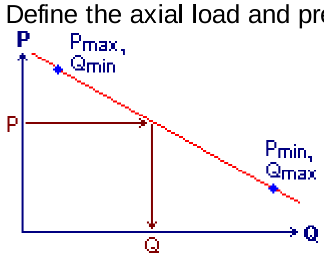


Specify the allowable bearing pressure value. The pressure is always a ***service*** value (i.e. not increased by the load factor).

Two options are available:

Q=
Enter a value for bearing pressure according to the units displayed adjacent to the text box.

Qmax, Pmin, ...
Define the axial load and pressure values at the two points on the following graph:

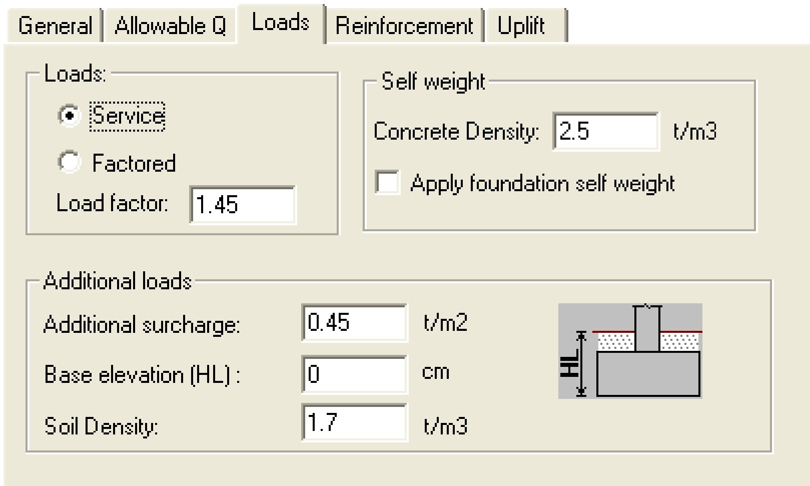


The program then uses the axial load value in each load case to calculate the value of **Q** for that case.

Note:

- the program does not accept a load outside of the range Pmax-Pmin.
- P values must be the same type as the loads - service or factored values

13.1.6.1.3 Loads



Loads

Service or factored loads may be defined:

Service
Define loads not increased by the Code load factors for Dead and Live loads (e.g. 1.4, 1.6, etc). The program multiplies all loads entered by the **Load factor** specified in this option.

Factored
Define loads already increased by the Code load factors for Dead and Live loads. Note that a value for Load factor must be specified even if this option is selected because the program must convert the loads to service values in order to calculate bearing pressures

If reactions were transferred from *STRAP*, the selection here must conform to the load type in the transferred combinations.

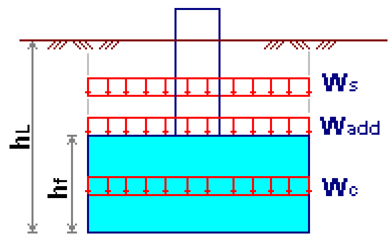
Load Factor:
Define the average load factor (normally between the Dead and Live load factors). The load factor must be defined for both service loads and factored loads:

- Service loads: all loads are increased by the factor
- Factored loads: loads are reduced by the factor to calculate the bearing pressure

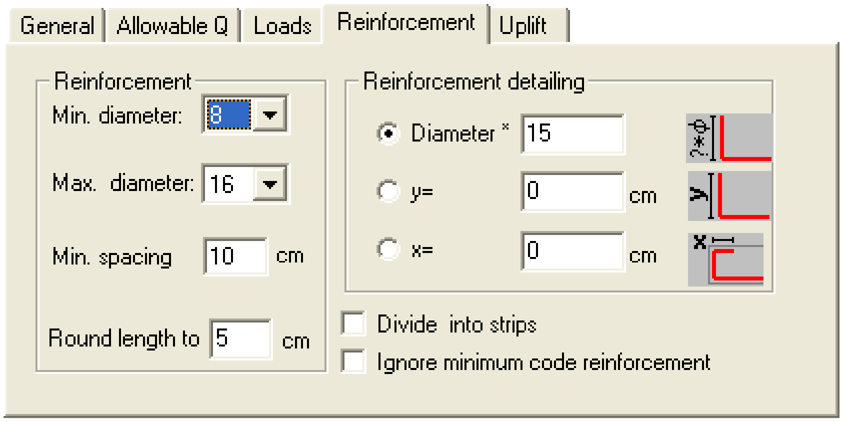
Additional loads / Self-weight

The additional loads used for calculating the stability of the footing may be defined:

- **Wc** = Self weight of the footing
  = **hf** x concrete density
  This load is applied only when ☑ **Apply foundation self-weight** is specified.
- **Ws** = Soil above the footing
  = (**hL** - **hf**) x soil density
  Enter **soil density =0** to cancel the application of this load
- **Wadd** = Additional uniform load acting on the top face of the footing. Enter a value according to the units displayed.



13.1.6.1.4 Reinforcement



Min/max diameter

Select the minimum and maximum reinforcement diameters

- The program selects the smallest diameter that provides a spacing greater than the Code minimum in both directions.
- Reinforcement is not detailed (only area required is displayed) if the spacing for the largest diameter is not adequate.

Minimum spacing

Specify the minimum spacing between parallel reinforcement bars.

- If a zero value is entered, the program uses the minimum bar spacing value specified in the Code (refer to Design assumptions 1268).

Round length

Round-off the length of all reinforcement bars to the value specified here.

Hooks

Specify the hook details for the reinforcement; select one of the following options:

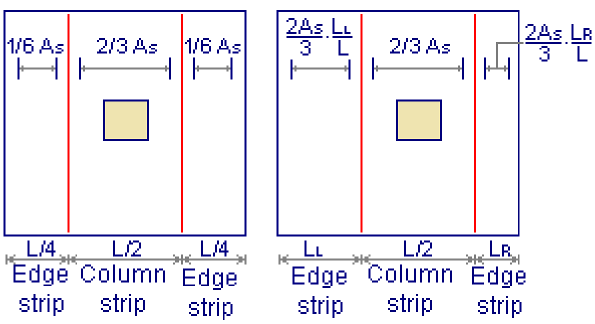
Diameter x
The total length of the hook is always the factor entered multiplied by the reinforcement diameter. A horizontal segment at the top of the footing is provided if the footing height (less the cover) is insufficient for the total hook length.

y =
Specify the length of the vertical hook:
- enter y = 0 if you do not want a hook
- enter a very large value to terminate the hook to the top of the footing (less cover); the program always terminates the bar inside the footing.

x =
Specify the length of the horizontal portion of the hook at the top of the footing; a full vertical segment (height - 2xcover) is provided

Divide into strips

Many codes specify that the reinforcement required should be concentrated adjacent to the column. For example:



☑ Divide As req'd into strips (spacing within each strip is uniform)
☐ Do not divide As req'd into strips; spacing is uniform over the footing width

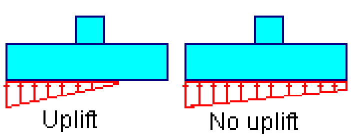
Refer to Design assumptions 1268 for the method used for each design Code.

Ignore code minimum

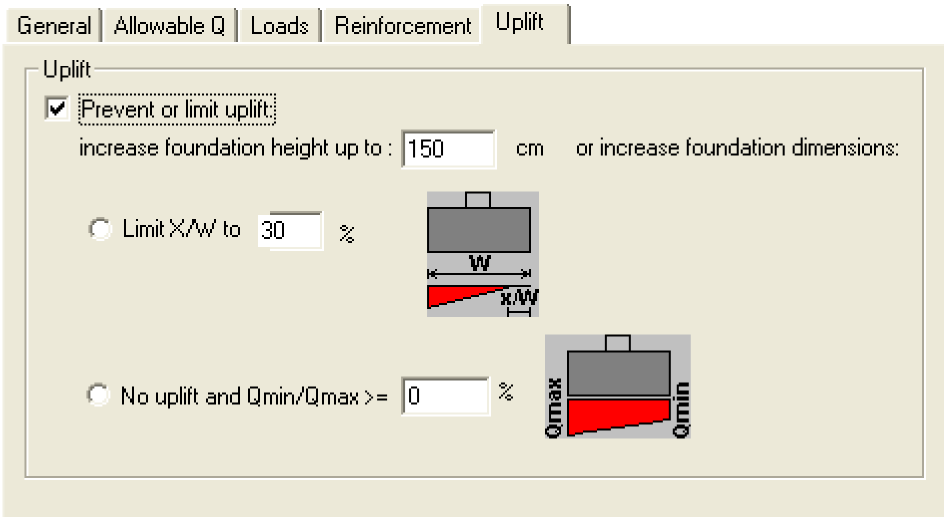
☑ The program ignores the minimum reinforcement percentage specified by the Code and displays the area required by the calculation only.

13.1.6.1.5 Uplift

"Uplift"occurs when the soil pressure diagram does not extend along the entire footing dimension:



The program can prevent or limit uplift by increasing the footing dimensions:



- The program first selects dimensions without consideration of the uplift options specified in this dialog box.
- If the uplift width exceeds the **x/W** limit or **Qmin/Qmax** exceeds the percentage specified, the program first tries to increase the footing height to solve the problem.
- If increasing the footing height up to the maximum specified does not solve the problem, the program increases the footing dimensions, resets the height to the value required by shear punching and returns to the previous stage.

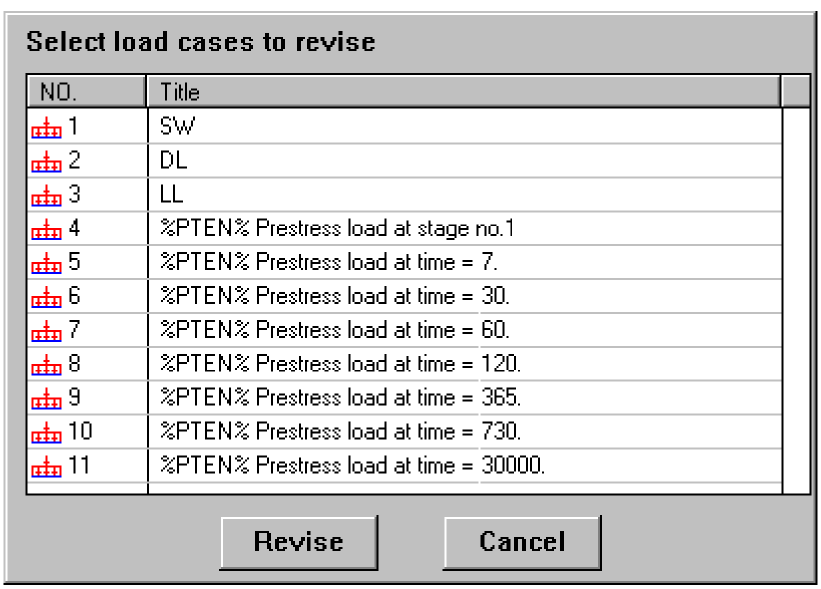
The calculation is repeated until the uplift requirements are satisfied.

13.1.6.2 Loads

Define the load cases:

- up to 25 different load cases may be defined, (in additional to any load cases transferred from a *STRAP* model).
- each load case may contain an axial load and moments about both axes (uniaxial or biaxial moments)
- Only the allowable bearing pressure (Q) may be revised for the *STRAP* load cases

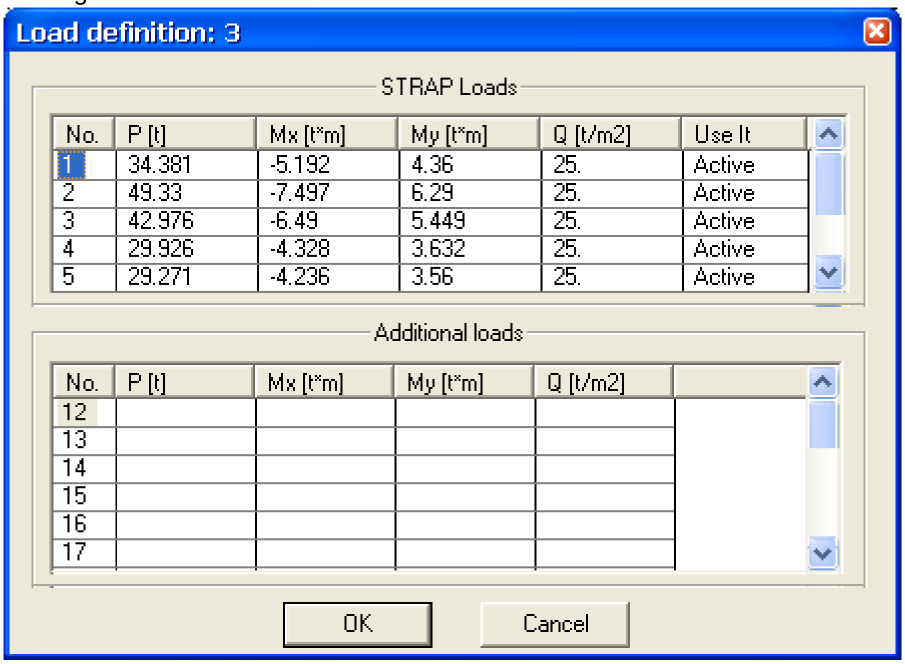
User-defined footings:



Select load cases to revise

NO.	Title
1	SW
2	DL
3	LL
4	%PTEN% Prestress load at stage no.1
5	%PTEN% Prestress load at time = 7.
6	%PTEN% Prestress load at time = 30.
7	%PTEN% Prestress load at time = 60.
8	%PTEN% Prestress load at time = 120.
9	%PTEN% Prestress load at time = 365.
10	%PTEN% Prestress load at time = 730.
11	%PTEN% Prestress load at time = 30000.

Footing from *STRAP* model:



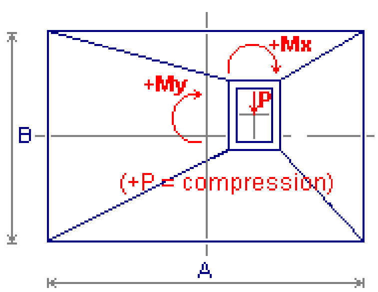
No.	P [t]	Mx [t*m]	My [t*m]	Q [t/m2]	Use It
1	34.381	-5.192	4.36	25.	Active
2	49.33	-7.497	6.29	25.	Active
3	42.976	-6.49	5.449	25.	Active
4	29.926	-4.328	3.632	25.	Active
5	29.271	-4.236	3.56	25.	Active

No.	P [t]	Mx [t*m]	My [t*m]	Q [t/m2]
12				
13				
14				
15				
16				
17				

P, Mx, My

Enter the axial load (P) and moments (Mx and My) according to the units displayed in the header bar. (units may be revise in the Setup 1265 option)

- Up to 25 different load cases may be defined; the program checks all of them when designing the footing and uses the worst condition.
- Sign conventions:



Q

A different value of **Q** (allowable bearing pressure) may be defined for each load case. The default value (defined in the Parameters 1255 option) is used if a value is not defined.

Use it (STRAP loads only)

STRAP load cases may be disabled:

- click on the cell; the following check box is displayed:

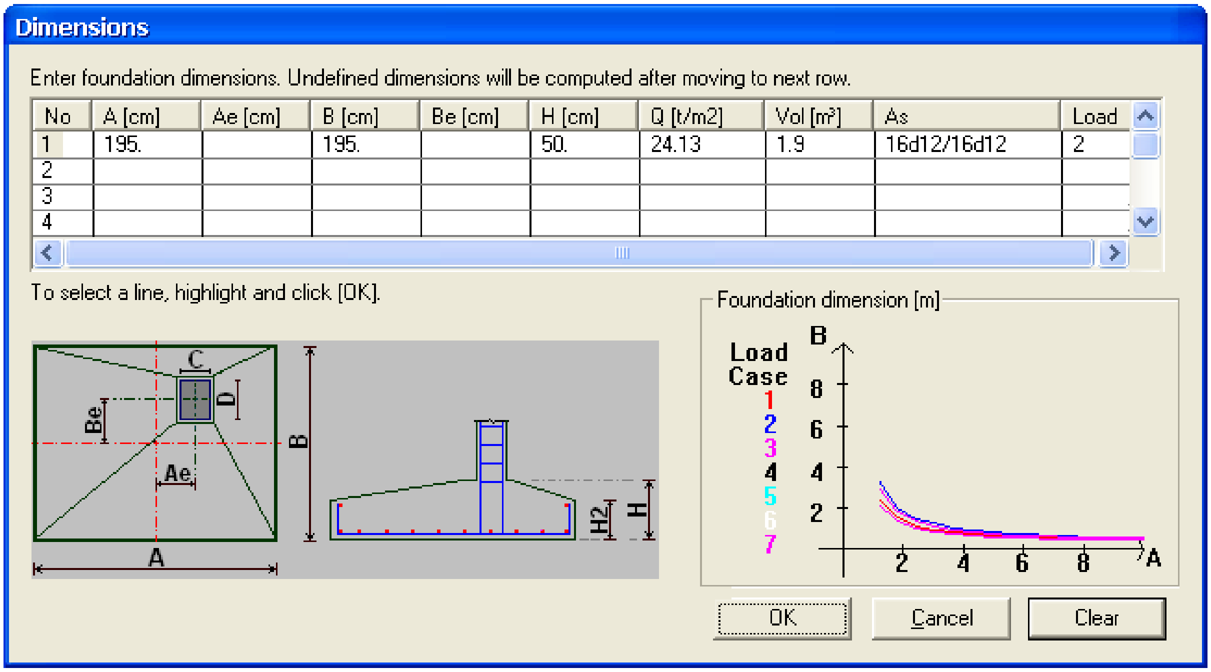
- set the checkbox to:
  - ☑ to use the load case
  - ☐ to disable the load case

Note:

- *STRAP* load cases with uplift or no axial load are automatically disabled and their status cannot be revised by the user.

13.1.6.3 Dimensions

Check alternate solutions by typing in the dimensions. You may check up to 40 different solutions. To select one of the solutions, click and highlight the line, then click the OK button; the selected footing is then displayed on the main screen.



The graph at the lower-right shows all possible combinations of **A** and **B** for each load case:

- type in values for **A** and **B**; the program calculates the height and reinforcement required and displays them in the continuation of the line
- type in a value for **A** or **B**; the program calculates the length required for the other dimension
- to design an eccentric footing, type in values for **Ae** and/or **Be**; note the positive direction of the eccentricity in the sketch displayed in the dialog box.

Note:

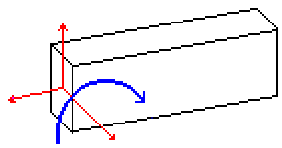
- the program displays two initial solutions:
  - square footing
  - A-C = B-D, i.e. footing dimensions proportional to column dimensions
- the footing height may be revised in the Sloped footing 1261 option
- the table is recalculated if parameters are revised.

13.1.6.4 Sloped footing

The program automatically calculates a footing with a uniform depth. Use this option to:

- specify a different uniform depth
- specify a sloped footing

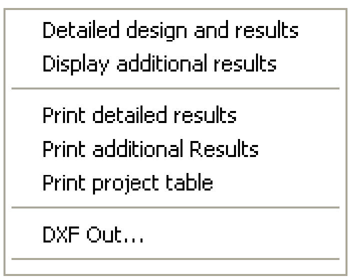
For example:



- up to 20 different combinations of **h** and **h2** may be tested
- define both **h** and **h2**: the program calculates both the punching and the shear stresses and displays the maximum value in the fifth column. The uplift status is shown in the last column.
- define either **h or hi** only: the program calculates the minimum value of the other dimension required so that max(punching, shear) displayed in the last column is less than the allowable shear stress.

To select design height values, highlight the line and click the OK button.

13.1.7 Output



13.1.7.1 Additional results

Display tabular results for each load case. For example:

Result Summary [t,m]

Exit

Case	P	Mx	My	Qall	Qmax	Md,x	Md,y	Punch Stress	Shear Stress	Max
1	20.89	1.38	4.55	25.0	15.9	5.18	7.16	10.4	22.4	
2	30.12	2.05	6.66	25.0	23.6	7.77	10.64	15.7	33.4	**
3	26.11	1.73	5.69	25.0	20.1	6.63	9.10	13.4	28.5	
4	18.52	0.89	3.46	25.0	12.6	4.32	5.91	9.1	18.5	
5	16.30	0.97	3.39	25.0	11.8	3.85	5.35	7.8	16.8	

Allowed punching & shear stress = 41.7 41.8

As,x:	Mdx=14.35	Required=1280	Provided=1357
As,y:	Mdy=13.91	Required=1280	Provided=1357

Area : mm²

Concrete volume: 1.15 m³

where:

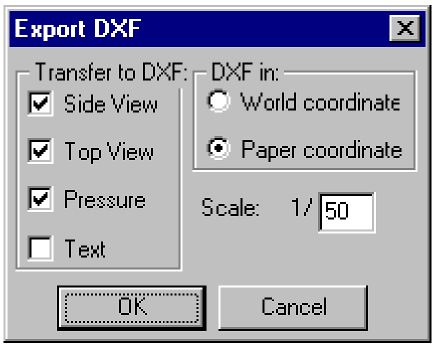
P, Mx, My	=	input loads
Qall	=	allowable bearing pressure
Qmax	=	maximum bearing pressure for load case
Punch	=	punching shear stress
Shear st	=	shear stress

Max	=	** indicates governing design case (pressure diagram, moments, etc. for this case are displayed on the screen)
Mdx,Mdy	=	design moments used for calculating the reinforcement

13.1.7.2 DXF out

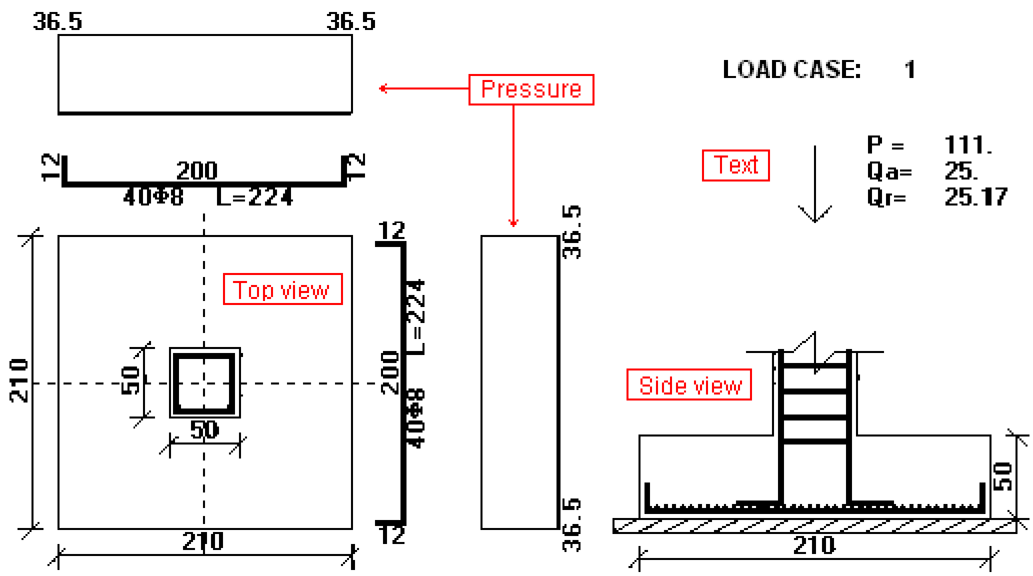
Create a DXF file of the footing drawing.

Specify the options:



Transfer to

any of the items in the group box to transfer them to the DXF file:



Coordinates

Select one of the following options:

World coordinates

The footing dimension transferred to the DXF file is the actual footing dimension. The "scale" option in this menu is required to determine the size of the text in the DXF file.

- metric units: the coordinates are written in millimeters
- American units: the coordinates are written in inches

Paper coordinates

The footing size transferred to the DXF file is the scaled footing dimension, i.e. dimension/scale. For example, a 2000 mm footing with a scale of 1:50 is written as 2000/50 = 40 drawing units in the file.

- metric units: the coordinates are written in millimeters/scale
- American units: the coordinates are written in inches/scale

Scale

Specify the scale of the DXF drawing.

13.1.7.3 Print drawing/results

Print the detailed results 1251, the additional results table 1262 or the Project list table 1246.

[Print]

Send output to

Select the output unit, e.g. printer, plotter, PDF, etc. The devices must be installed by the "Printers" option in the Windows "Control panel".

Setup

Specify general information for the output device selected:
- paper size
- graphic resolution
- etc.

Prepared by

Define information that is printed in the header at the top of every printed page:
- **Date:** the date format is specified in the Windows "Control panel"
- **Prepared by:** enter **a name**

Output to file

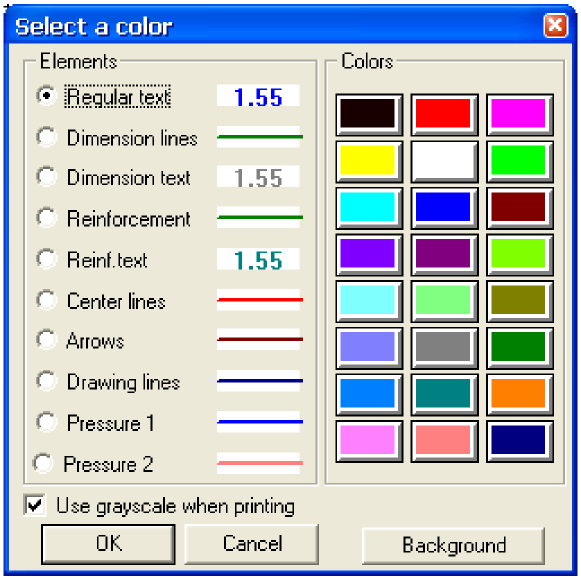
☑ to send the table/drawing to a file; the program prompts for the file name
☐ to send the table/drawing to the printer

13.1.8 Setup

Colors
DXF files
Steel & concrete
COdes
Units
Parameters

Colors

Specify the screen color for any of the elements displayed on the screen:



Elements and colors:

To change the color of any of the display elements on the screen:

- click the of the element in the left group box (e.g. Regular text)
- click on one of the colors in the right group box; the small display example is revised immediately

Background:

Revise the screen background color:

- click on one of the color boxes
- click the OK button

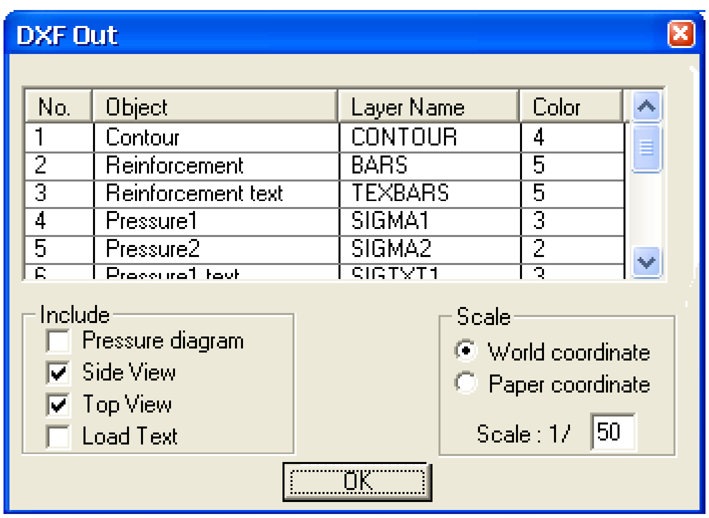
Black and white:

☑ Print the drawing using black for all colors, even if a color printer is used.

☐ Print the drawing with the screen colors (color printer) or gray shades (black & white printer)

DXF file

Specify default parameters for the DXF out 1264 option:



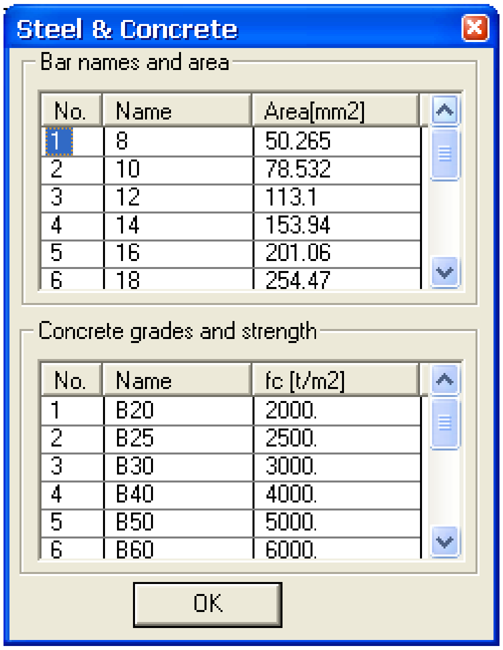
Specify the layer name and associated colour for each DXF object.

- No blank spaces are allowed in the layer name.

Steel & concrete

Specify:

- a list of reinforcing bars and their diameters
- a list of concrete grades and corresponding bending compression strengths.

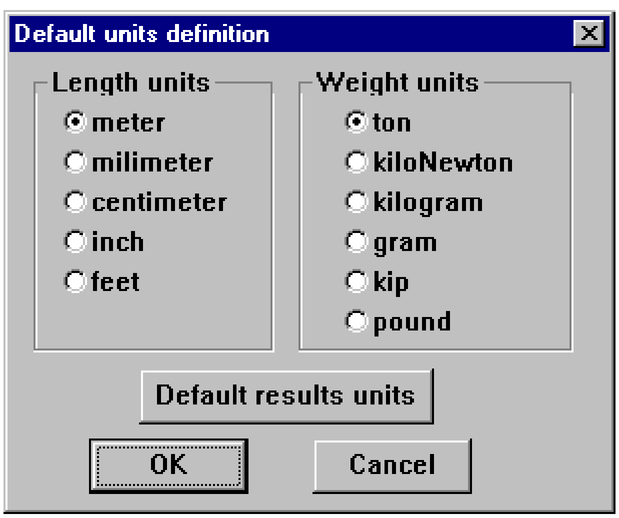


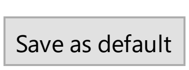
Codes

Select a national design Code from the list displayed. Refer also to Design assumptions 1268

Units

Select one of the following default units:





: the parameters specified here will be the default parameters every time a new footing is defined.

13.1.9 Design assumptions

13.2 Create/edit a Steel sections table

13.2.1 User steel section table

To display a demo video that explains how to create a "User" steel section table:

- *click on to start the video*
- *then click on to enlarge the display.*

This utility module allows you to build a customized "user" steel section table for the *STRAP* geometry and the Steel design module. Hot-rolled or cold-formed sections may be added:

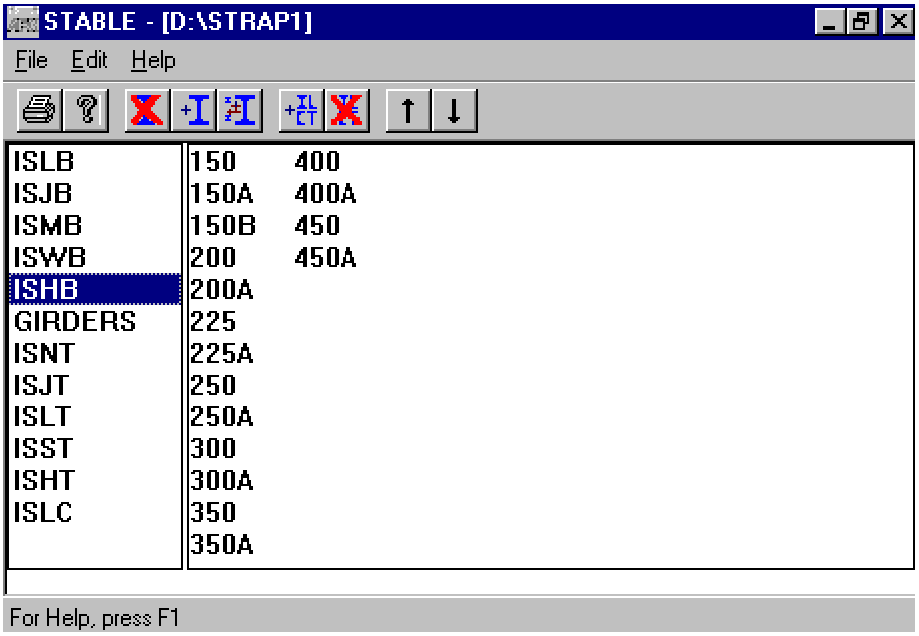
- **Hot-rolled**:
  - Sections may be copied from any of the standard steel tables (British, European or American) supplied with the program
  - New sections may be added to existing types by entering their dimensions; the program will calculate all the section constants.
  - New section types may also be defined for convenience; however new types must have the same shape as one of the standard types recognized by the program (I, [, RHS, etc). New shapes cannot be created.
- **Cold-formed**:
  - New sections may be added to existing types by entering their dimensions; the program will calculate all the section constants.
  - New section types may also be defined for convenience; however new types must have the same shape as one of the types recognized by the program (Z, angle with lips, etc).
  - New shapes may be created 1277.

Note:

- the "user" steel table is always stored in the file PROPTABS.DAT; this file must always be in the program directory.
- if a user steel table has been created, the option **The user steel table** is displayed in the Section table menu in the Steel design module.

To run the module:

- Select **Files** in the ***STRAP*** initial screen.
- Select **Utilities** in the pull-down menu.
- Select **Create/edit a steel sections table.**



Select from the menu bar at the top of the screen:

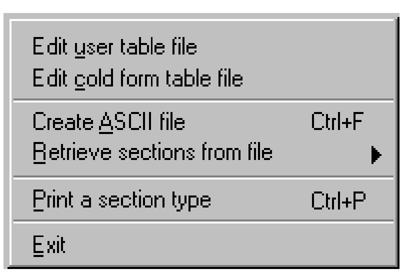
Files:

- specify the table 1270 to which to add the sections - "User table" or "Cold-formed".
- Create 1271 a file in ASCII format containing all of the section data in the table.
- Retrieve 1271 section date from an ASCII format file
- print 1272 the section dimensions and constants for the highlighted type.
- exit from the module and return to the ***STRAP*** main screen.

Edit:

- add a new section 1272 to the table.
- add a new section type 1273 to the table.
- revise the dimensions 1273 or the constants of a section.
- delete 1274 sections from the table.
- delete 1274 a section type from the table.

13.2.1.1 "File" options



Edit file

✓ **Edit user table file**
Add new sections - hot-rolled, cold-formed, or built-up - to the user steel table (PROPTABS.DAT).

✓ **Edit cold form table file**
Add new cold-formed sections to the standard steel tables (British, American, European, etc).

Create ASCII file

This option allows you to create an ASCII file containing all of the section data that can be edited using any standard editor program.

This option allows you to edit the section dimensions and data externally to this program, as follows:
- select **Create ASCII file**
- specify the name and directory of the ASCII file:
- use any standard text editor to revise the dimensions and data.
- select **Retrieve sections from file** and select **ASCII file** to transform the file back to internal format.

Note that section data can be revised interactively in this program by selecting **Revise section** in the **Edit** menu.

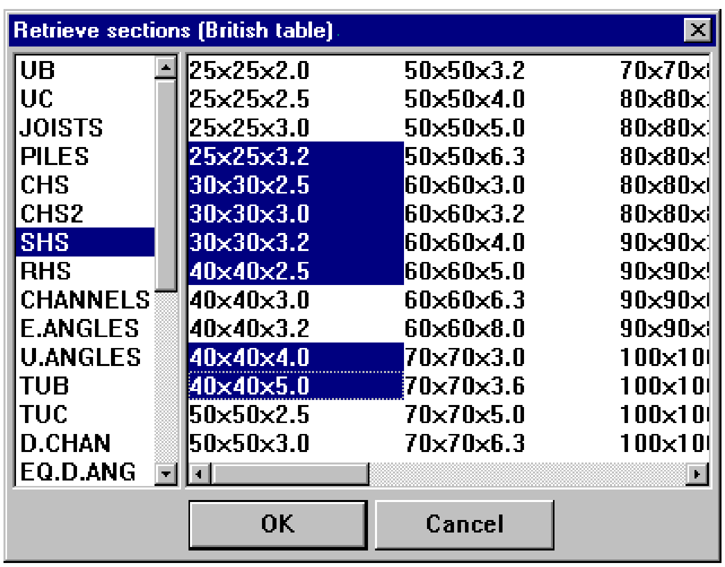
The ASCII file the data for each section consists of the section title (16 characters) and 36 numbers:
- the data item corresponding to each of the 36 numbers is listed in the header before each section type.
- each data item MUST contain a decimal point.
- you may change the number of digits after the decimal point, but the number must always be in the 10 columns allotted to it.

For more details on the file format, refer to ASCII file format 1274.

Retrieve sections

Add sections to the user steel table (PROPTABS.DAT). The program will create a new file if one does not exist in the program directory. Copy sections from one of the existing master tables or an ASCII file. Select:

- **British / European / American/... table**
  A list of the section types in the existing file is displayed. First select a section type and then select individual sections to be copied to the new table.



  Note that a section type with the same name is automatically added to the user table.

- **ASCII file**
  This option converts an ASCII file containing all of the section data to the internal format used by the steel postprocessor.
  The ASCII file must be created using the option **Create ASCII file.**

These two options allow you to edit the section dimensions and data externally to this program, as follows:

- select **Create ASCII file**
- use any standard text editor to revise the dimensions and data.
- select **Retrieve sections from file** and select **ASCII file** to transform the file back to internal format.
- specify the name and directory of the ASCII file.

Note:

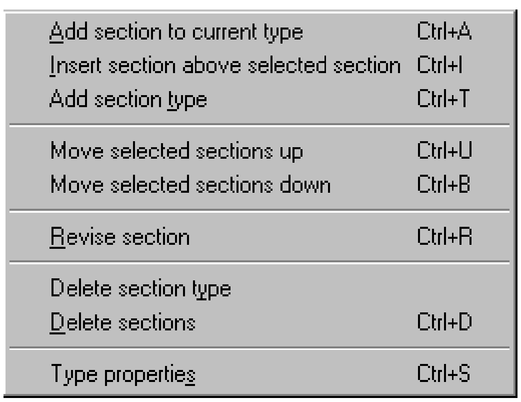
- section data can be revised interactively by using the Revise section[1273] option.
- when **Edit user table file** has been selected, both hot rolled and cold formed sections found in the ASCII file are added to the use table.
- when **Edit cold form table file** has been selected, the cold formed sections in the ASCII file are added to the standard tables (not the User table); hot rolled sections in the ASCII file are ignored.
- I+[sections cannot be added to the file.

Print a section type

To print the section type currently highlighted.

Continue for more details on the standard "Print tables" menu and Print "Setup".

13.2.1.2 "Edit" options



Add section to current type / Insert section above ...

Create new sections by entering dimensions. A user defined section can be added to any existing section type, including user-defined types. Note that a section type may contain a maximum of 640 different sections.

- the program displays a list of existing sections types in the left list box ; click and highlight one.
- the program will prompt for all section dimensions required according to the section shape of the type selected.

Units: if *STRAP* default units =

- feet or inch - enter values in inch, $inch^2$, etc
- metric units - enter values cm, cm^2, etc, except for "H", which is displayed in (decimeters)[6]

Note that the section will be added to the **end** of the list. To insert the section anywhere in the list, select **Insert section above selected section** in the **Edit** menu.

Note:
- **Hot rolled sections**:
  - all hot-rolled user-defined sections are assumed to be "rolled", not "welded".
  - the program does not prompt for bend radius and ignores rounded corners or fillets when calculating the section properties. The user should revise the calculated data accordingly.
- **Cold formed sections**:
  - the program prompts for the internal bend radius and calculates the section constants accordingly.
  - all dimensions are external

For more information on each of the constants, refer to ASCII file format 1274.

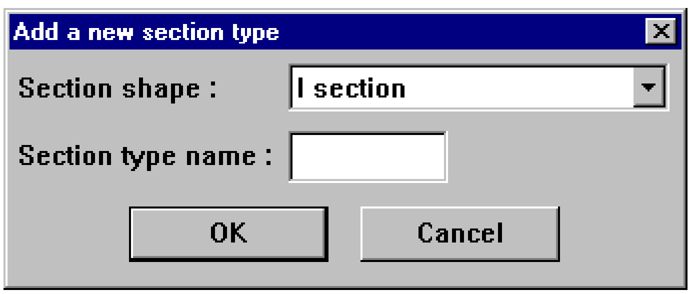
Insert sections above

Use this option to insert sections in the middle of the list (the **Add sections** option always adds the new sections at the end of the list).
- Highlight a section in the section list
- select this option
- proceed as explained in Add sections 1272

The new section is inserted above the highlighted section.

Add section type

Define a new section type which can include only user defined sections.



- ***Section shape:***
  Select a shape from the list box. The new type must be classified as one of the standard section shapes; all sections included in this type are assumed by the Steel Postprocessor to have that shape. Note that sections such as double angles, etc. are not allowed.
- ***Section type name:***
  Enter a name for the new section type - 16 characters maximum.

Note:
- Add user-defined sections to the new type as explained in Add sections 1272.
- To define sections for an existing type, use the Add Sections 1272 option.

Move sections up/down

Rearrange the section list by moving sections up or down:
- click and highlight a section in the list
- click the ↑ or ↓ buttons in the toolbar (or select the options in the **Edit** menu)

Revise section

Revise the dimensions of any section:
- first select the section type, then the section to be revised.

- the section dimensions and calculated data for the section are displayed on the screen. Edit the data as explained in [Add sections](1272).

Delete section type

- Highlight a type in the left-hand list box.
- Select the **Delete section type** option.

Delete sections

- Highlight the type in the left-hand list box.
- Highlight the sections to be deleted in the right-hand list box
- Select the **Delete sections** option.

Type properties

Display the type name and the number of sections in the list for any section type:
- click and highlight a section type name
- select this option

13.2.1.3 ASCII file format

The following is an example of the format of the user table ASCII file:

```
                   1         2         3         4         5         6         7         8
Column:   12345678901234567890123456789012345678901234567890123456789012345678901234567890

Type:     #UB          1
Section:   203x133x25         250.000   203.200   133.400     5.800     7.800   172.300
                             2360.000   310.000   260.000    71.400      .876    25.400
                                 .029     6.120    32.300   447.000      .338    19.800
                               15.600    12.900      .367   159.000      .325      .000
                                 .000      .000      .000      .000      .000      .000
                                 .000      .000      .000      .000      .000      .000
           203x133x30         300.000   206.800   133.800     6.300     9.600   172.300
                             2890.000   384.000   313.000    88.100      .882    21.500
               .  .  .  .  .
               .  .  .  .  .
Type:     END
```

For each section type:

- Header row:

  column 1: #
  column 2-9: type name (8 characters)
  column 14: section type:
  Hot-rolled: 1=I 2=U 3=RHS 4=L 5=T 6=Pipe 7=2L 8=2[9=I+[
  Cold-formed: 30=User-defined cold-formed section
  (files from older versions may contain the following data:
  21=C 22=C+lips 23=hat 24=Z 25=Z+lips 26=L)

- Footer row: column 1-3: END

For each section: six rows containing the tile and 36 data items:

- 1st row:
  column 2-17: title
  columns 18-27 , 28-37 , 38-47 , 48-57 , 58-67 , 68-77: data items 1-6

- rows 2-6:
  columns 18-27 , 28-37 , 38-47 , 48-57 , 58-67 , 68-77: data items 7-36 (6 per row)

The thirty-six data items are explained in the following table. Note that the units are "cm" or "in", depending on the default *STRAP* units.

Hot-rolled sections + Cold-formed (type 21-26)				
Item	**Description**		**for types**	**Units**
1	Weight per meter * 10.			kg/m*10
2	Section height			cm.
3	Section width Total width		All types, except - 2L, 2[	cm.
4	Web thickness			cm.
5	Flange thickness Internal bend radius		All types, except - Cold-formed	cm.
6	Net section height between fillets Width of single section Lip length		- I,[,T - 2L, 2[- Cold-formed	cm.
7	Major axis moment-of-inertia (I_x)			cm^4
8	Minor axis moment-of-inertia (I_y)			cm^4
9	Major axis plastic modulus (Z_x) Major principal axis moment-of-inertia (I_u) Major axis elastic modulus (S_x)		All types, except- - single angles - double angles	cm^3 cm^4 cm^3
10	Minor axis plastic modulus (Z_y) Minor prncipal axis moment-of-inertia (I_v) Minor axis elastic modulus (S_y)		All types, except- - single angles - double angles	cm^3 cm^4 cm^3
11	Buckling parameter (u) Major axis elastic modulus (S_x) Major principal axis moment-of-inertia (I_u)		- I,[,T - single angle - component angle of double angle	 cm^3 cm^4
12	Torsional index (x) Minor axis elastic modulus (S_x) Minor principal axis moment-of-inertia (I_v)		- I,[,T - single angle - component angle of double angle	 cm^3 cm^4
13	Warping constant (H)		Hot rolled: units = Cold-formed : units =	dm^6 cm^6
14	Torsional constant (J)			cm^4
15	Gross section area			cm^2
16	Plastic modulus - major axis: Lower values of n: $S_{xr} = S_x - K2{\cdot}n^2$ Higher values of n: S_{xr}=K3(1-n)(K4+n)	K2	I, [only	cm^3
17		n		
18		K3		cm^3
19		K4		
20	Plastic modulus - minor axis: I - section: as major [- section: Same stress induced in web by moment, axial: S_{yr} = Sy+K2·n(K3-n) Different stress: S_{yr} = Sy-K2·n(K3+n)	K2 K2	I [- low n ; same stress	cm^3 cm^3
21		n K3	I [- low n ; same stress	
22		K3 K1	I [- high n ; same stress	cm^3 cm^3
23		K4 K2	I [- high n ; same stress	 cm^3
24	[- section: Same stress induced in web by moment, axial: S_{yr} = Sy+K2·n(K3-n) Different stress: S_{yr} = Sy-K2·n(K3+n)	K3	[- high n ; same stress	
25		n		
26		K2	[- high n ; opp. stress	cm^3
27		K3	[- high n ; opp. stress	
28	---- Not used ----			
29	Minor axis elastic modulus (S_y)		[	cm^3

For cold-formed sections:

Cold formed sections (type 30)		
Item	**Description**	**Units**
1	Weight per meter * 10.	kg/m·10
2	Section height	cm.
3	Section width (2·width + spacing for double sections)	cm.
4	Web thickness	cm.
5	Internal bend radius	cm.
6	No. of linear segments (max = 14) + : first segment is horizontal – : first segment is vertical	
7	Major axis moment-of-inertia (I_x)	cm^4
8	Minor axis moment-of-inertia (I_y)	cm^4
9	Major axis elastic modulus (S_x)	cm^3
10	Minor axis elastic modulus (S_y)	cm^3
11	x0: distance from centroid to shear centre (major axis)	cm.
12	y0: distance from centroid to shear centre (minor axis)	cm.
13	Warping constant (Cw or H)	cm^6
14	Torsional constant (J)	cm^4
15	Gross section area	cm^2
16 to 29	Segments 1 to 14 : Length - + : up/right direction – : down/left direction	cm.
30	100 * line no. of type in USERSECT.DAT	
31	Distance between sections : for double sections -1 : for single sections	cm.
32	---- Not used ----	
33	Section group no. for Fy calculation	—
34	double sections : height of connection single sections : 'j' constant - about least non-symmetric axis –[min(x_o,y_o)] . Refer to AISI - C3.1.2-11	cm. cm.
35	'j' constant – about most non-symmetric axis –[max(x_o,y_o)]	cm.
36	Iu: moment-of-inertia, minor principal axis	cm^4

13.2.1.4 Cold formed section file

New cold-formed shapes may be created by the user:

- All data for cold formed shapes is stored in the file **USERSECT.DAT** (must be in program directory). The file may be updated using any standard line editor program.
- The information in the file is general and non-dimensional; the actual sections are defined using the Add section 1272 option.

*** ***Create a backup of the file before editing it*** ***

Shapes are defined by specifying "segment" and "parameter" data:

- Each shape must be composed of a series of ***segments*** connected in a chain.
- The sections are defined in the Add section 1272 option by entering data for dimension *parameters* (e.g.

width, height, thickness, etc).

Note:
- The shapes can be duplicated about an axis of symmetry, e.g. double channels (back-to-back or flange-to-flange).

File format:

|
|
n lines : titles of sections types defined in file
|
|
END

For each of the *n* section types:

Line 1[1278]: general data, including:
NSEG = no. segments.
NPAR = no. of dimension parameters
NPAR lines[1279] definition of parameters in terms of segments
NPAR lines[1279] menu prompts
NSEG lines[1279] definition of segments in terms of parameters

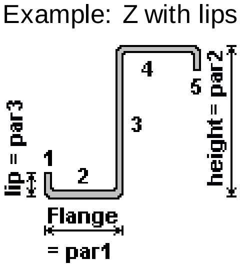
END

Line 1

Item	Column	Description
NSEG	1-3	Number of segments in shape (max=14)
NPAR	4-6	Number of input parameters, not including "thickness" and "bend radius" and "distance between double sections" (max=7)
IDB	7-8	= 0 : single section = 1 : double section
XDIS	9-18	Distance between double sections (mm/inch) = -1 : prompt user for dimension
IFIR	19-20	= 0 : first segment - vertical = 1 : first segment - horizontal
XHT	21-25	height of connection (for double sections) - required for calculating C_w. Enter a fraction between 0.0 and 1.0, where 0.0 indicates the bottom of the section, 1.0 indicates the top of the section.
ITYP	34-37	serial number for PROPTABS.DAT, item 30.

All values must be right justified.

Example: Z with lips



NSEG = 5
NPAR = 3
IDB = 0
XDIS = 0.
IFIR = 0
IHT = 0

NPAR lines - parameters

Define the parameters in terms of the segments.

The parameter = **k1**·seg*1* **+ k2**·seg*2* **+ ... + kn**·seg*n*

where:
ki = any factor (see examples)
seg*i* : indicates the *i*th segment

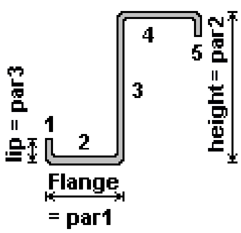
Example:
parameter **i** is equal to the difference between the length of segment 2 and segment 4:
therefore, k1=0, **k2=1**, k3=0, **k4=-1**, k5=0, ...

The k factors are written in the file as follows:

Item	*Column*	*Description*
k1	1-5	factor for segment 1
k2	6-10	factor for segment 2
	.	.
kn	-5·n	factor for segment n

The factors must be right justified

Example: Z+lips



	k values
Parameter 1 = segment 2:	0 1 0 0 0
Parameter 2 = segment 3:	0 0 1 0 0
Parameter 3 = segment 1:	1 0 0 0 0

NPAR lines - menu prompts

Enter the parameter text that will appear in the dialog box when the user defines a new section of this type.

For example: Z with lips - 3 parameters:
- **flange width**
- **height**
- **lip length**

NSEG lines

Define the segments in terms of the parameters.

The length of the segment = **k1**·par*1* **+ k2**·par*2* **+ ... + kn**·par*n*

where:
ki = any number (see examples)
par*i* indicates the *i*th parameter

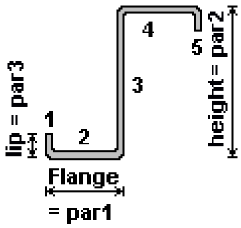
Note that if the end of a segment to below/to the left of the start of the segment, then its length must be negative.

The k factors are written in the file as follows:

Item	*Column*	*Description*
k1	1-5	factor for parameter 1
k2	6-10	factor for parameter 2
.	.	.
kn	-5·n	factor for parameter n

The factors must be right justified.

Example: Z+lips



	k values		
Segment 1 = par3 (down):	0	0	-1
Segment 2 = par1 (right):	1	0	0
Segment 3 = par2 (up):	0	1	0
Segment 4 = par1 (right):	1	0	0
Segment 5 = par3 (down):	0	0	-1

13.3 Compute section properties

13.3.1 Main menu

CROSEC is a program that calculates the properties of geometric sections (area, moment-of-inertia, center-of-gravity, etc).

Refer also to:
[General](1281)
[How to use the program](1282)
[Hints and suggestions](1283)

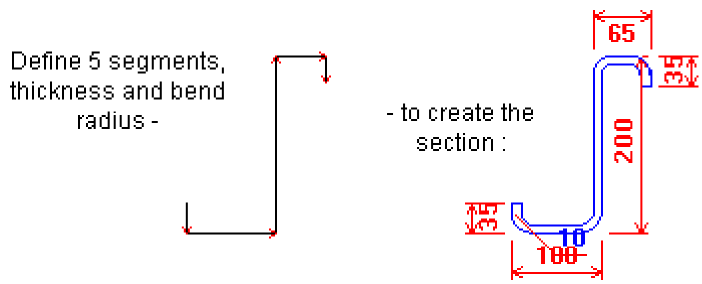
13.3.2 General

CROSEC is a program that calculates the properties of geometric sections (area, moment-of-inertia, center-of-gravity, etc).

The program can calculate all properties that are required for the design of cold-formed (light gauge) sections. All line sections (with any arbitrary shape) defined in this program may be transferred to the *STRAP* steel postprocessor and can be calculated as cold-formed sections.

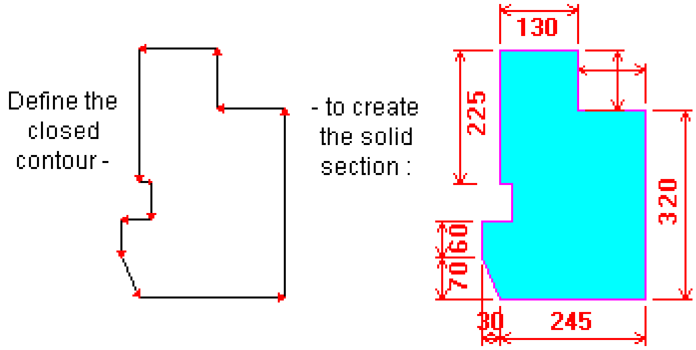
Two general types of sections may be defined:

- **Line sections:**
  Sections composed of a series of connected lines, each with a specified thickness. For example:



  - Segments may be defined in any direction (diagonal).
  - Several separate sections may be defined to form a section; the additional sections are called "subsections". Each subsection may have a different thickness and bend radius.
  - The properties of a Line section may be copied to *STRAP* geometry
  - Tables of cold-formed sections may be added to the *STRAP* property tables; the user first defines the dimensionless general shape and then enters the dimensions for each segment of every section in the table. Different segments may be defined as "equal" (only one of the dimensions must be entered).
  - Properties calculated include: Shear center, warping constant, torsional moment-of-inertia and torsional-flexural buckling constant

- **Solid sections:**
  Sections formed by a closed contour. For example:



- Several separate contours may be defined to form a section; the additional contours may be specified to be "holes".
- The properties of Line sections may be copied to *STRAP* geometry.
- Properties calculated include: Torsional moment-of-inertia (exact).

Refer also to:
[How to use the program] 1282
[Hints and suggestions] 1283

13.3.2.1 How to use the program

- Select **New line section** or **New solid section** in the [Section] 1285 menu in the menu bar:
  New line section: define a series of connected segments to form the section
  New solid section: define a closed contour to form the section

- Add subsections, if necessary to create more complex sections. For example:
  New line section: define a series of connected steel sections to form cold-formed steel sections. The subsections are defined separately and must be "connected".
  New solid section: define a hole in the first contour

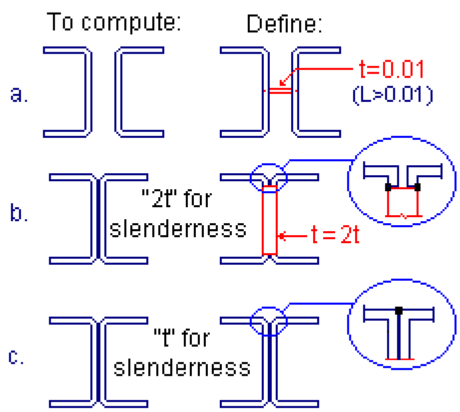
- Select **Display properties** in the [Output] 1305 menu in the menu bar to display the computed section properties (A, I, J, centre-of-gravity, etc) and **Print section** to print the results.
- Transfer the properties of the current section to *STRAP* geometry; select **Copy to clipboard** in the [Output] 1305 menu in the menu bar

For line sections only:
Create a table of cold formed sections for the *STRAP* properties table. The table is created from a Line section by defining sections with similar shapes but with different dimensions:
- select **Create table** in the [Section table] 1302 menu in the menu bar
- select an existing Line section and enter a table of dimensions for the shape.
- select **Transfer to STRAP** in the [Section table] 1302 menu in the menu bar to copy all section in the table to the *STRAP* property files (as cold-formed sections)

Refer also to [Hints and suggestions] 1283.

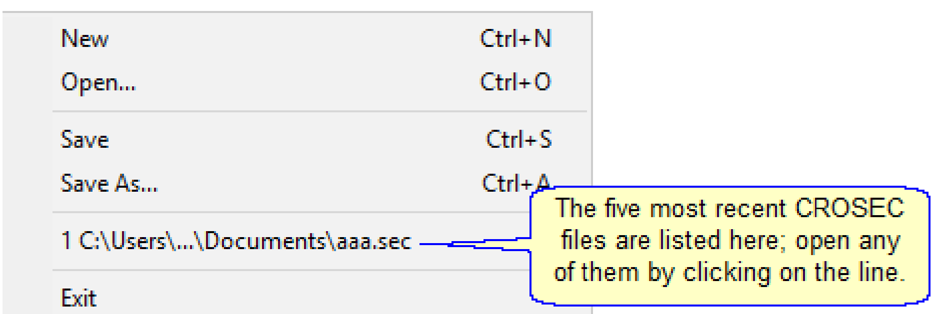
13.3.2.2 Hints and suggestions

- For separated back-to-back channels, angles, etc; connect the two channels with a subsection having the correct length and a thickness = 0.01 (Example a). Refer to Section - connect 1294
- For connected back-to-back channels, etc, where the slenderness of the web must calculated using the ***combined*** web thickness, define the web as a subsection with the double thickness and the flanges by four subsections connected to the corners of the web (Example b).
- For connected back-to-back channels, etc, where the slenderness of the web must calculated using the web thickness of a ***single*** channel, define the second channel as a subsection and connect at one point (Example c).



13.3.3 File menu

All defined sections are saved in a file ***nnn*.SEC**, where ***nnn*** is specified by the user. When the user starts the program, the program automatically opens the last file accessed .



New

Open a new section file. If you have not saved sections defined in the current session, the program asks whether to save them in the current file before opening the new one (the sections are erased if you do not save them).

The program prompts for the file name only when you select **Save**, **Save as** or **Exit**.

Open

Open an existing file. The program displays the list of section files in the current folder:

- double-click on one of the file names to open it, or -

- type in the file name in the **File name** edit box and click Open
- use the << >> options at the top of the dialog box to select a different folder or open a new folder.

Save

Save all sections created in the current session in the current file (the file name is displayed at the top of the screen).

Save as

Save all sections created in the current session **and** all sections in the current file in a **New** file.

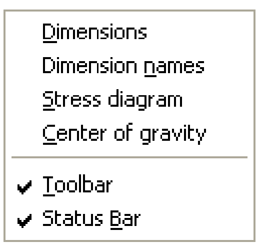
- type in the file name in the **File name** edit box and click Save

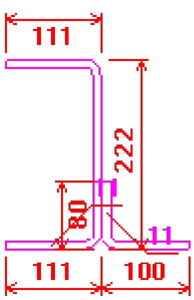
Exit

Exit from the program. If you have not saved sections defined in the current session, the program asks whether to save them before exiting.

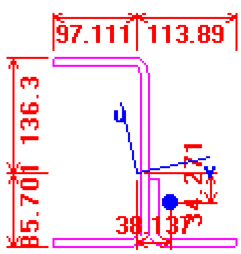
13.3.4 Display menu

Select the display options for the current section.

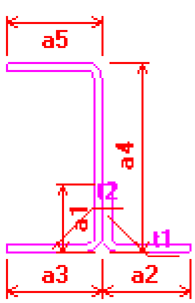




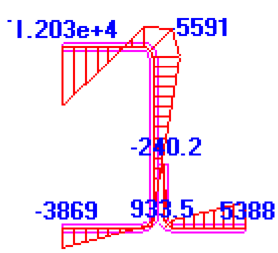
Dimensions



Center-of-gravity



Dimension names

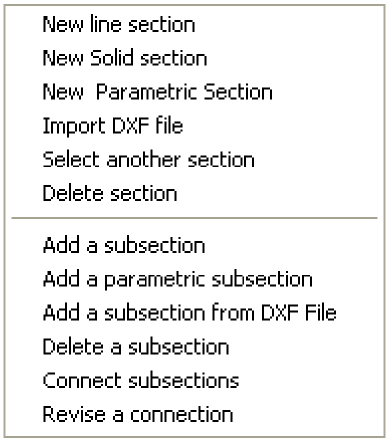


Stress diagram

Note:

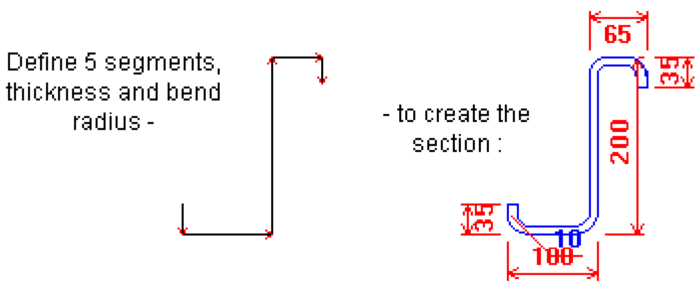
- Dimension names and Stress diagram are not available for Solid sections
- The stress diagram shows the normal stresses required for the calculation of the shear center location.

13.3.5 Section menu



13.3.5.1 New line section

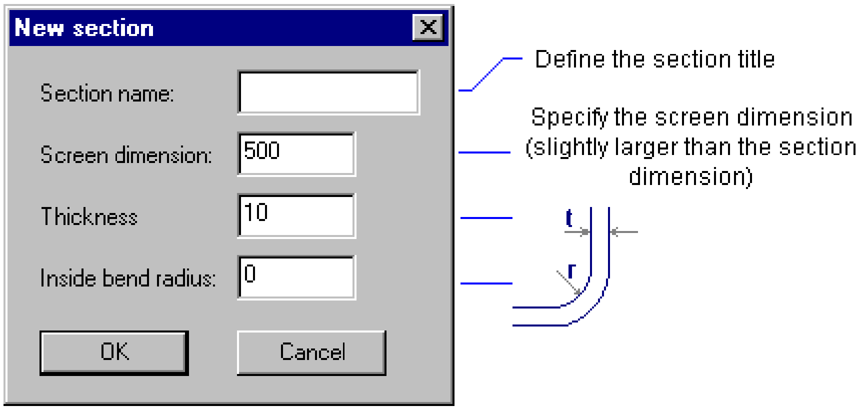
Define a section composed of a series of connected lines, each with a specified thickness. For example:



To display a demo video that explains how to create a cold-formed section table and how to transfer it to STRAP:

- *click on*  *to start the video*
- *then click on*  *to enlarge the display.*

Define the section parameters:

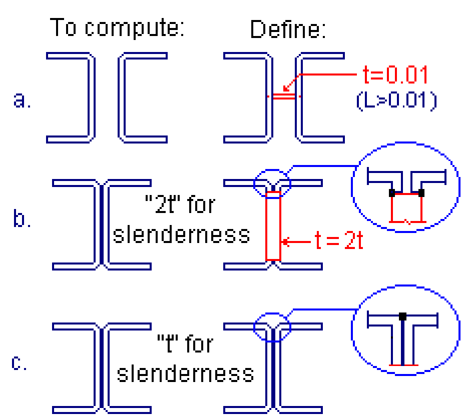


Note:

- refer to Line section - define 1286 for details on how to define the individual line segments.
- screen dimension, thickness and Inside bend radius must be defined according to the same unit (mm, cm, ft, etc).
- segments with different thickness values must be defined as separate subsections and then combined.
- no bend radius is drawn if three segments intersect at the same point

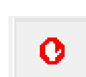
Hints:

- For separated back-to-back channels, angles, etc; connect the two channels with a subsection having the correct length and a thickness = 0.01 (Example a). Refer to Section - connect 1294
- For connected back-to-back channels, etc, where the slenderness of the web must calculated using the combined web thickness, define the web as a subsection with the double thickness and the flanges by four subsections connected to the corners of the web (Example b)

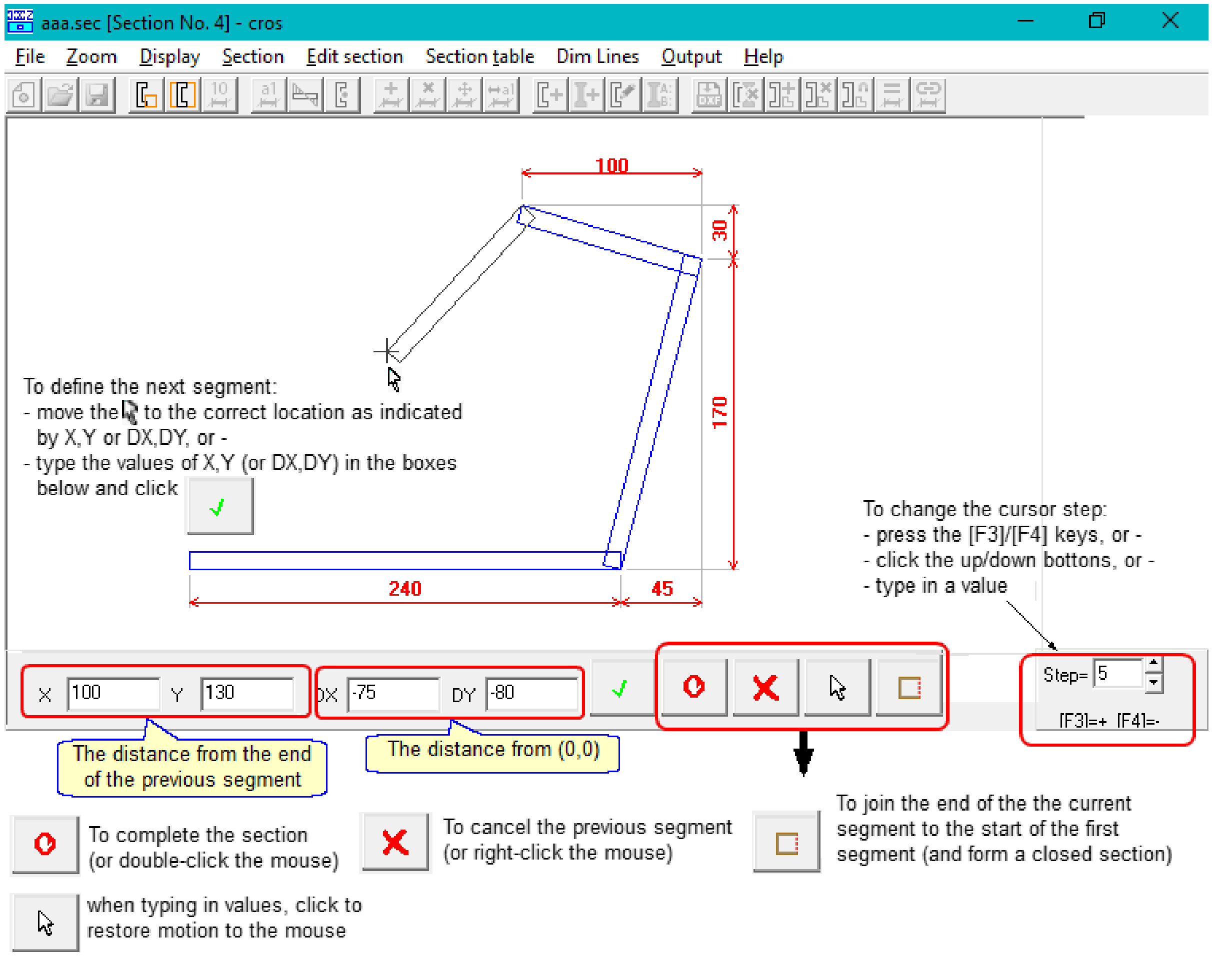


13.3.5.1.1 Define line section

Line sections are created by defining a chain of connected segments (the start of the current segment is the end of the previous segment):

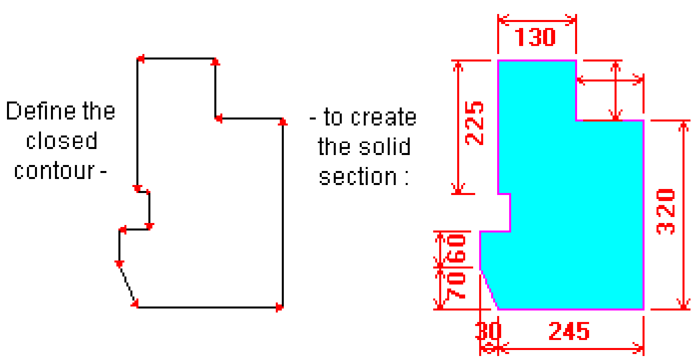
- the end location of each segment is defining by moving the crosshair on the screen or by typing in the coordinate.
- the section is completed by double-clicking the mouse or by clicking the  button

For example:

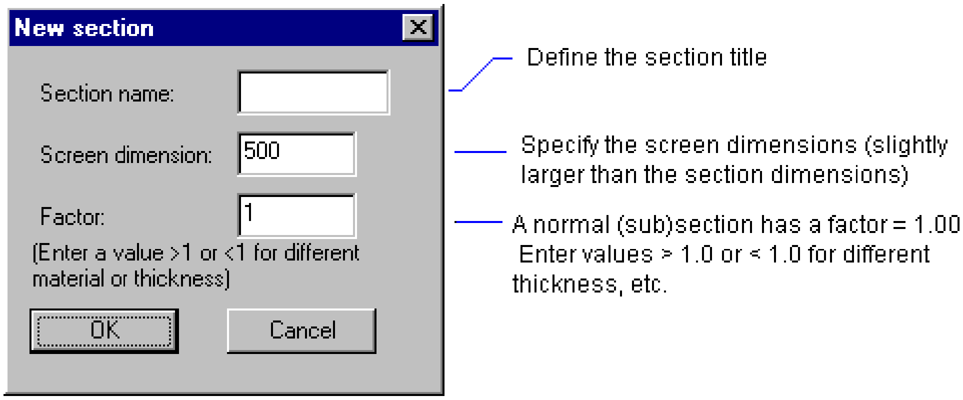


13.3.5.2 New solid section

Define a sections formed by a closed contour. For example:



Define the section parameters:



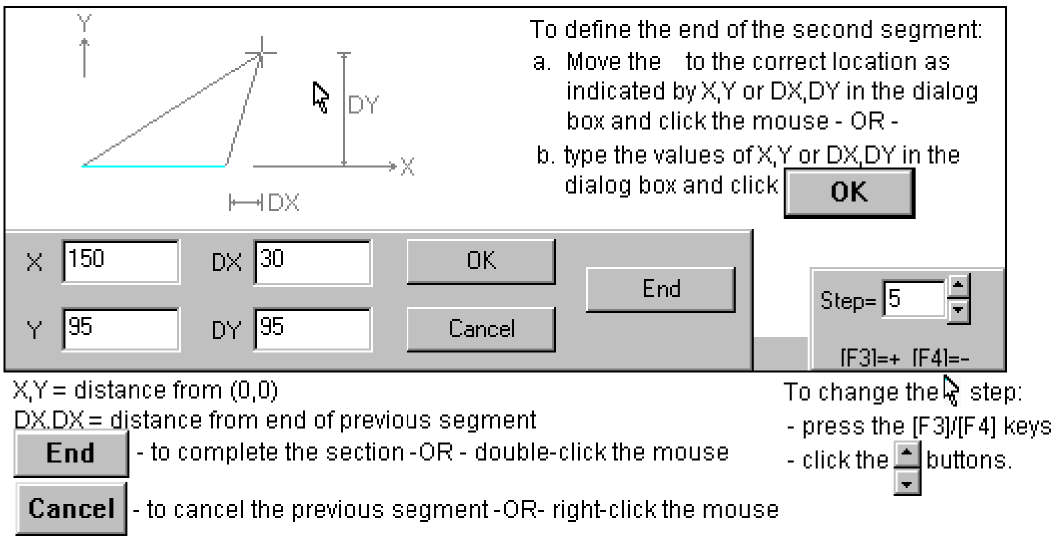
- refer to Solid section - define [1289] for details on how to define the contour line segments.

13.3.5.2.1 Define solid section

Solid sections are created by defining a chain of connected contour segments (the start of the current segment is the end of the previous segment); the program automatically connects the end of the current segment to the start of the 1st segment:

- the end location of each segment is defining by moving the crosshair on the screen or by typing in the coordinate.
- the section is completed by double-clicking the mouse

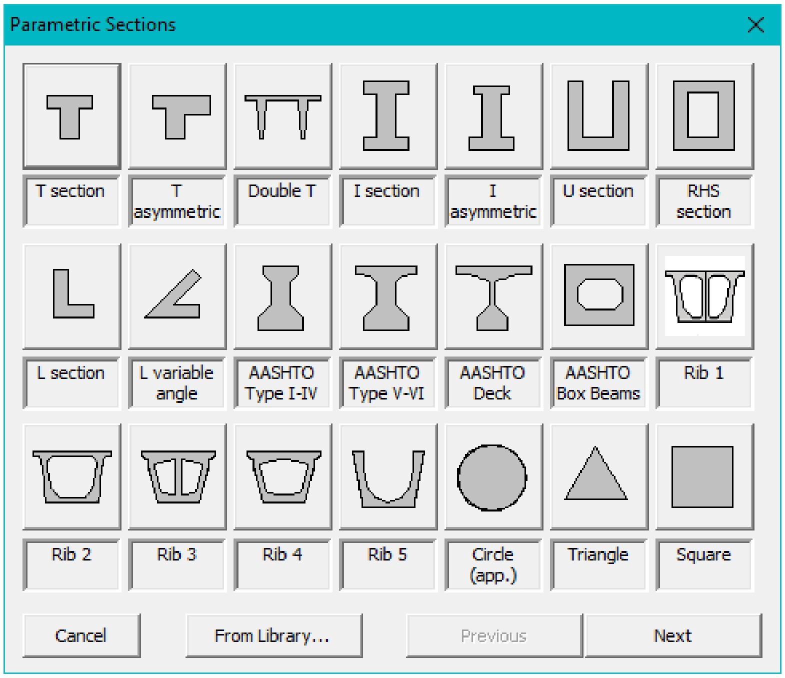
For example:



13.3.5.3 Add parametric section

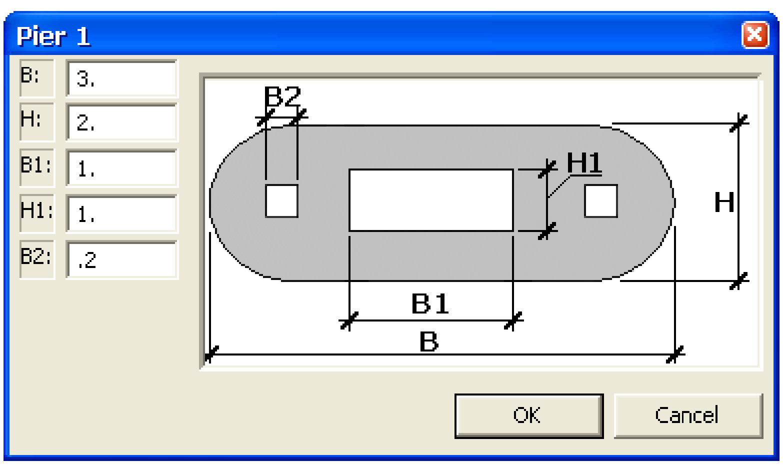
Predefined solid sections may be defined by entering the dimensions:

- select a section by clicking on the image:



Previous and Next are active if more section screens are available.

- enter the dimensions according to the diagram. For example:



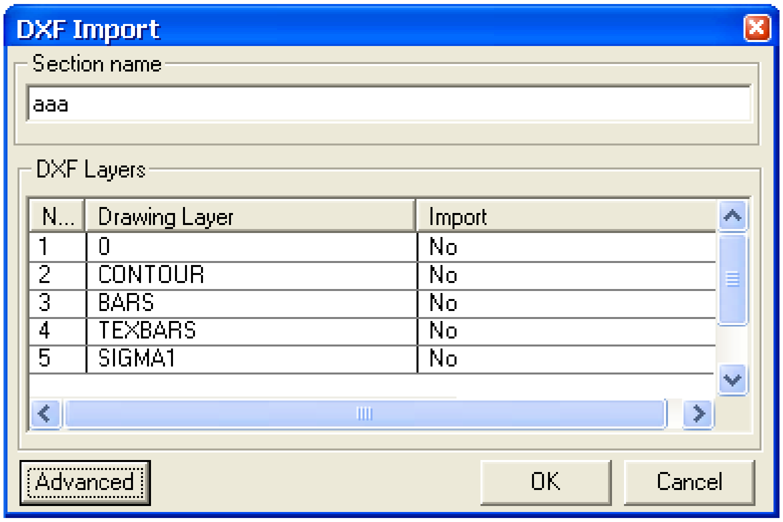
Note:

- Parametric sections can be added as a subsection.
- New section types may be added. Please contact your *CROSEC* dealer.

13.3.5.4 Import DXF file

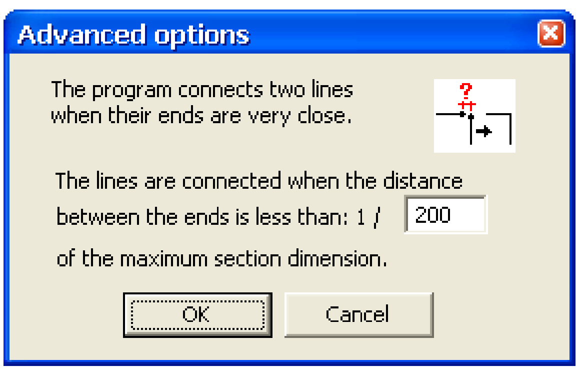
Import a solid section from a DXF file. The program automatically imports all closed sections found in the selected layers, i.e. there may be several sub-sections.

Select the layers:



Move the to the relevant line and click the mouse to toggle the Yes/No in the 'Import' column.

The program can connect lines if their ends are very close together. Click Advanced:

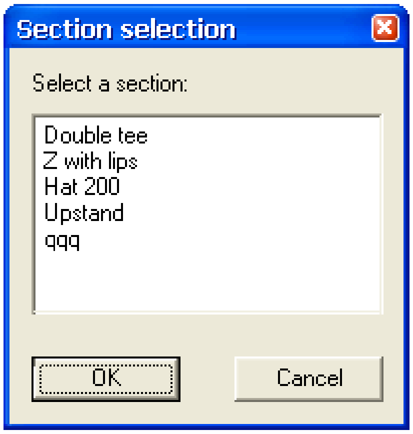


Note:
- a DXF section can be added as a subsection.

13.3.5.5 Select another section

Select and edit an existing section. New subsections may be added and the dimensions of the current section may be revised.

Select a section from the list displayed.

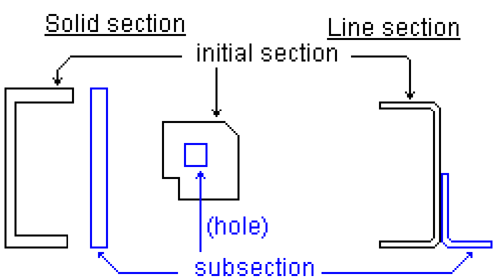


13.3.5.6 Delete section

Select a section to delete from the list displayed.

13.3.5.7 Add a subsection

Combine a new section with the initial one; the program calculates the section properties of the combined section. For example:

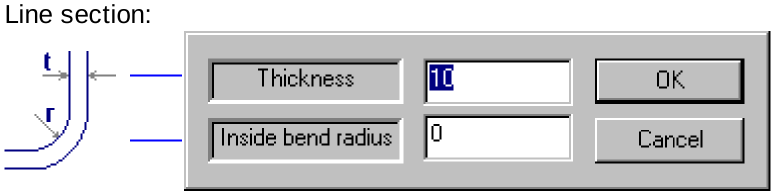


Note:

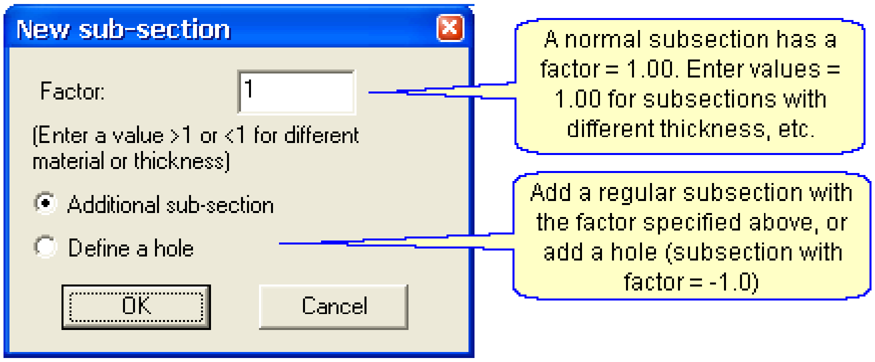
- up to 50 subsections can be added to the initial section
- line sections: the sub- and initial sections must be connected at selected section corners; refer to Connect subsections1299. Define all subsections before combining them.
- solid section: the sub- and initial sections do not have to be connected.

Define the subsection parameters:

- Line section:



- Solid section:



13.3.5.8 Delete a subsection

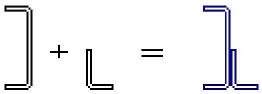
Line sections only: the program refreshes the screen with the subsections displayed separately.

Click on a subsection; confirm to delete.

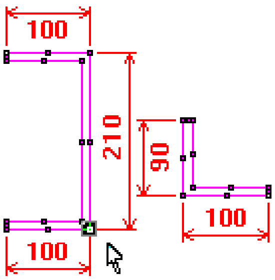
13.3.5.9 Connect subsections

All of the subsections must be connected to form a single final section (the program does not calculate the properties if the subsections are unconnected). Subsections may be connected ***only*** at their corner or mid-points (to connect at any other point, define the segment as two separate ones, joined at the necessary connection point).

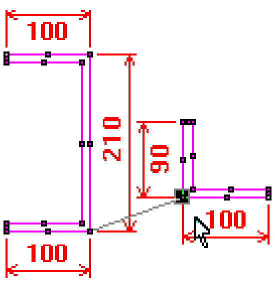
For example:



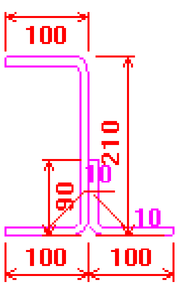
- create the channel section and the angle subsection
- select the "**Connect subsections**" option
- the program displays the connection point on the section - at every corner and at the midpoints of both sides of every segment; select the lower-right corner of the channel:



- select the lower-left corner of the angle:



- the program displays the connected section:

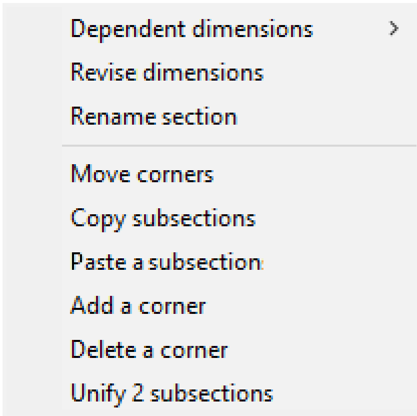


Refer also to Hints and suggestions 1283

13.3.5.10 Revise connections

- select the two connected subsections
- redefine the connection between them, as explained in Connect subsections 1294

13.3.6 Edit section menu



Rename section

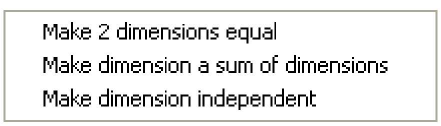
Revise the name for the current section

13.3.6.1 Dependant dimensions

Each segment is automatically assigned a different dimension name by the program (a1, a2, ..., t1, r1). However certain segments may always be equal, particularly in symmetric sections, or the length of a segment may always be equal to the sum of the lengths of other segments.

This option is relevant only if a Section table 1302 is created from the current section (the amount of data that must be typed in may be significantly reduced).

Use this option to define dependent dimensions:

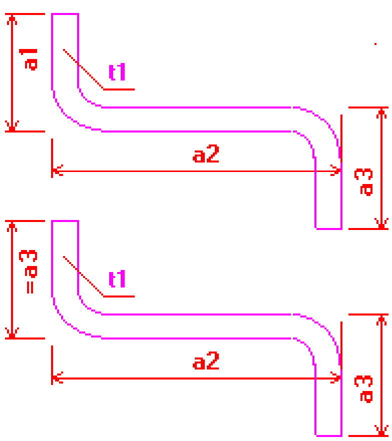


Make 2 dimensions equal

Example:

1. Click on segment "a1"
2. Click on segment "a3"

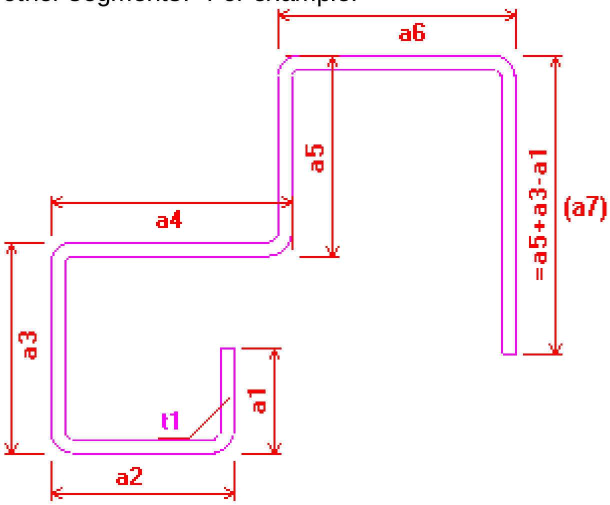
The program redraws the section:



and only the column "a3" is displayed in the "Section table" for this section.

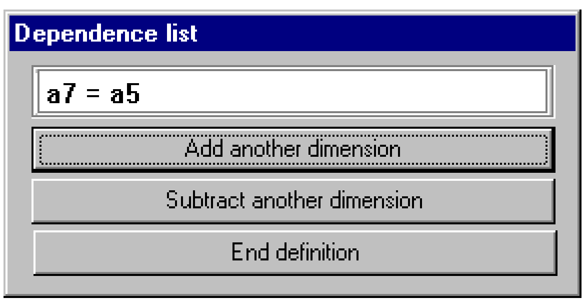
Make a dimension a sum of dimensions

Use this option to specify that the length of a segment must always be equal to the sum of the length of other segments. For example:

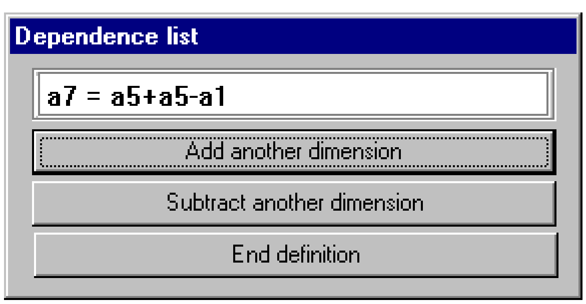


To define **a7=a5+a3-a1**:

- click on the dimension name **a7**
- click on the dimension name **a5**; the program displays the dialog box:



- click "**Add another dimension**"
- click on the dimension name **a3**; the program displays the dialog box with **a7=a5+a3**.
- click "**Subtract another dimension**"
- click on the dimension name a1; the program displays the dialog box:



- click "**End definition**"

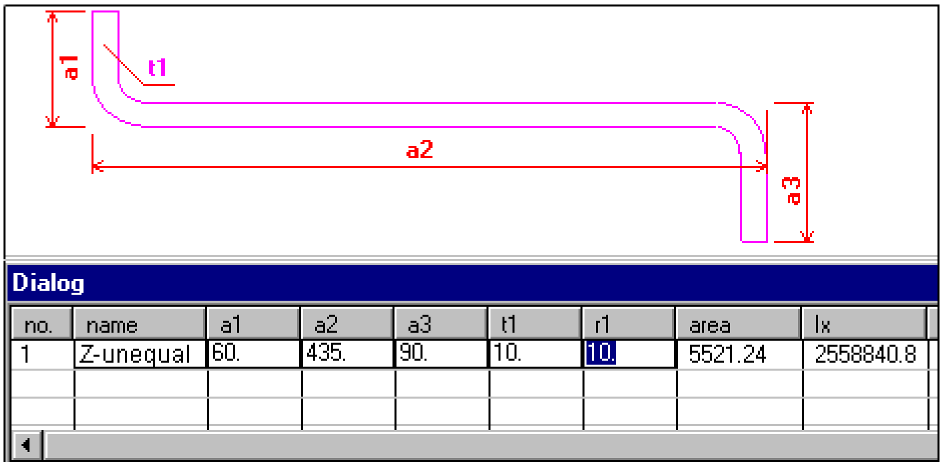
Make a dimension independent

Click on a segment with an "=" dimension name; the segment name reverts to its original one.

13.3.6.1.1 Revise dimensions

Revise dimensions for line sections only:

- the program redraws the current section at the top of the screen with the dimension names (a1, a2, ... an) in place of the dimension values
- the program displays a table at the bottom of the screen with the current value for each dimension



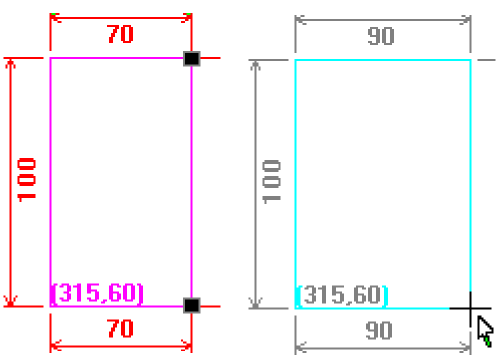
- revise any of the dimensions; the section properties in the table are automatically updated
- click OK to exit and redraw the section

13.3.6.2 Move corners

Use this option to move the corners of an existing solid section.

For example, increase the width of the rectangle below to 90.

- select the two corners on the right edge of the section by highlighting them and clicking the mouse (Figure a).
- drag the mouse to the right until the horizontal dimension is displayed as 90 (Figure b).
- click the mouse; the section is redrawn with the new dimension.



(a) (b)

13.3.6.3 Copy subsections

Copy a subsection to the clipboard. The section can then be added to the current section or another section using the Paste a subsection 1298 option in the same menu.

Note that more than one subsection may be selected at the same time.

- select the first subsection to be copied; the program highlights all corners in the section closest to the crosshair. Click the mouse.
- select additional sections, if required.
- click the mouse again or click 

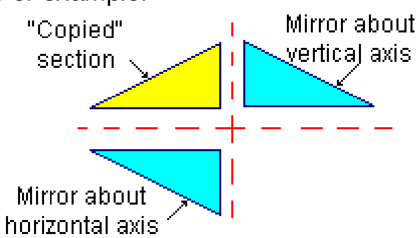
to end.

13.3.6.4 Paste a subsection

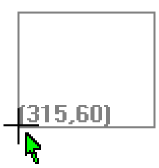
Paste a "Copied" (sub)section to the current section. The section is copied to the clipboard using the Copy a subsection 1298 option in this menu.

The section may be flipped vertically or horizontally when pasted:

For example:

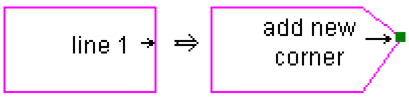


Move the section reference corner to the correct location and click the mouse. Note that the program continuously revises and displays the corner coordinates:



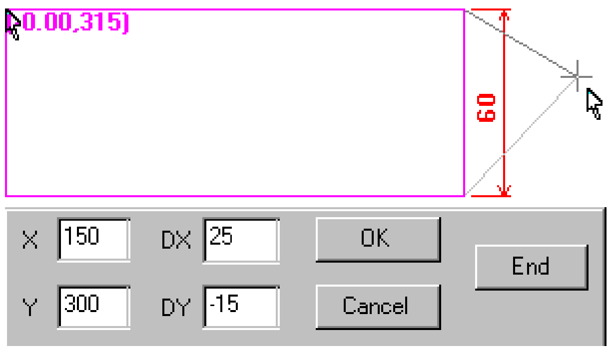
13.3.6.5 Add a corner

Add a corner to an existing section. For example, to revise the rectangle in Figure (a) to the pentagon in Figure (b) and new corner must be added to line '1'.



(a) (b)

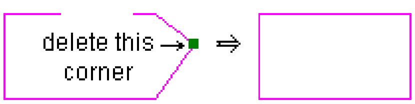
- highlight the line that is to be divided and click the mouse
- the crosshair is placed at the centre of the line; move the crosshair to the location of the new corner and click the mouse:



Note that DX, DY in the box at the bottom on the screen indicates the position of the crosshair relative to the start corner of the existing line.

13.3.6.6 Delete a corner

Delete a corner of an existing section. For example, deleting the corner as shown in Figure (a) creates the section shown in Figure (b).

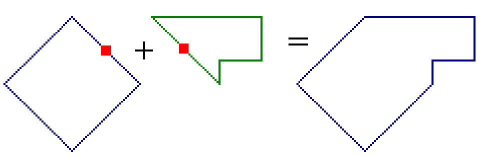


(a) (b)

Select the corner by highlighting it and clicking the mouse; the corner is deleted.

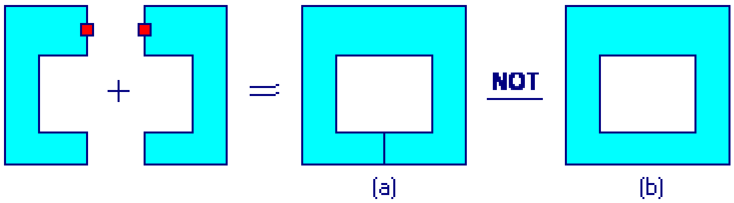
13.3.6.6.1 Unify 2 subsections

Combine two subsections along a common line to form a single section.



Select the two common lines by highlighting them and clicking the mouse

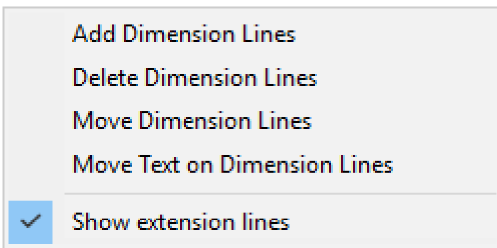
Note:
- the common lines must have the same length and angle; the program does not stretch, squeeze or rotate a subsection when attaching it to the another one.
- if the sections have different factors (thicknesses), the combined section will have the thickness of the ***first*** section selected
- holes may be selected
- lines that are coincidental but were not selected are not combined. For example



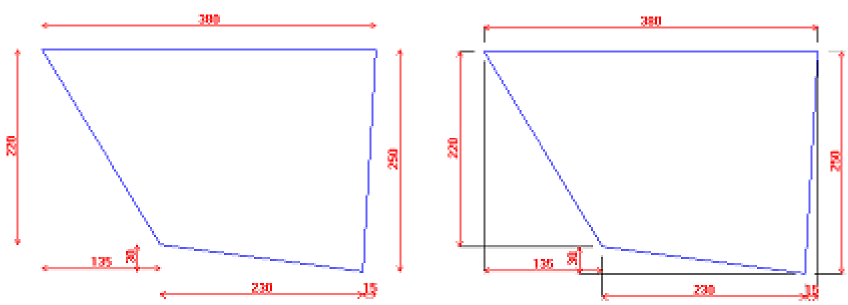
The torsional properties of this section (a) are different from those of the same section attached on the bottom face (b).

13.3.7 Dimension Lines

The program automatically adds dimension lines to the section showing the vertical and horizontal distances between adjacent corners. Use this option to add new dimension lines, erase or move lines or move the text on a dimension line.



Show extension lines



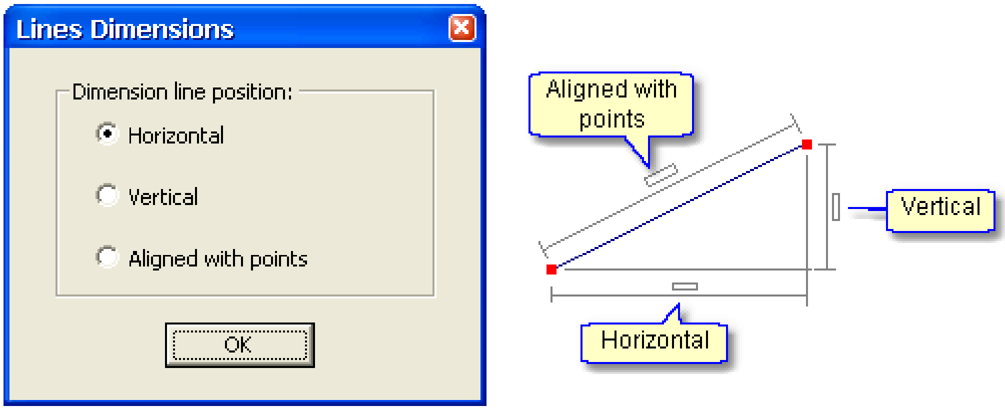
No extension lines With extension lines

13.3.7.1 Add

Dimension lines can be horizontal, vertical or parallel to the corners selected.

To define a dimension line:

- select the series of corners: move the to the relevant corner and click the mouse; repeat for additional corners and click the last corner twice to end the selection.
- select the orientation:



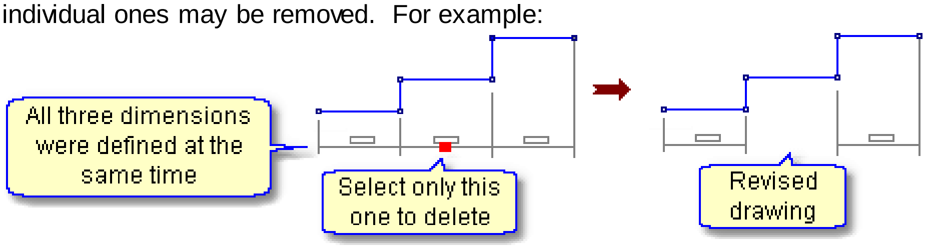
Note that "Aligned with points" cannot be defined if more than two corners were selected.

- Drag the dimension line to its location and click the mouse.

13.3.7.2 Delete

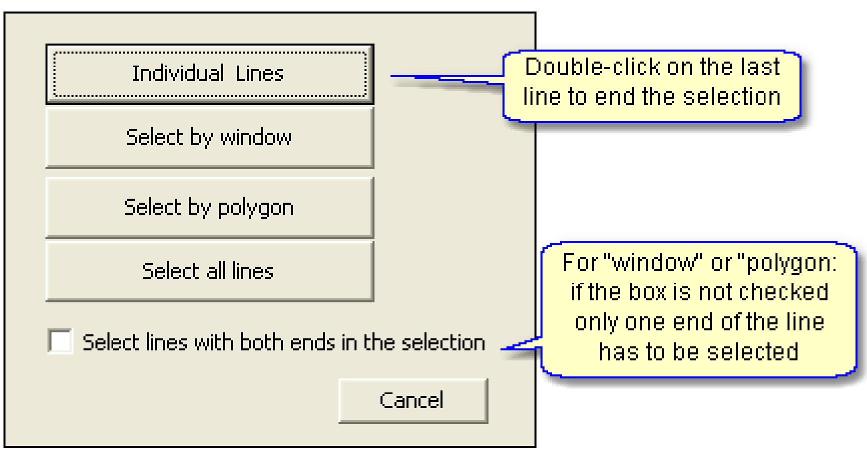
Delete existing dimension lines:

- both the lines added automatically by the program and those added manually may be selected.
- a series of dimensions defined at the same time are considered as separate lines by the program and individual ones may be removed. For example:



Note that all three dimensions must be selected to delete the entire line.

Select the lines to delete:

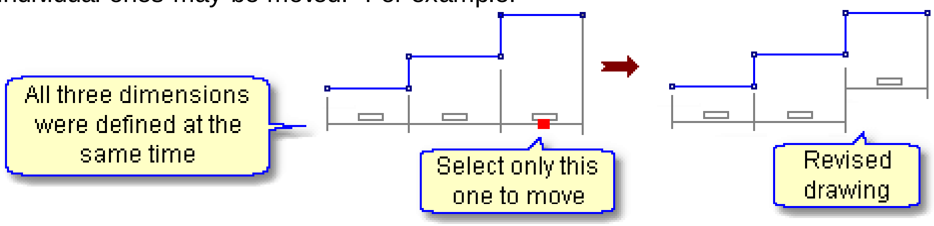


13.3.7.3 Move

Move existing dimension lines:

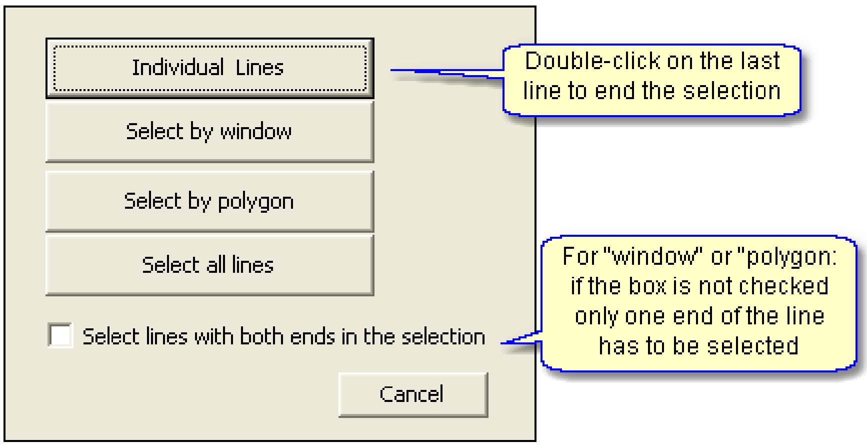
- both the lines added automatically by the program and those added manually may be selected.
- a series of dimensions defined at the same time are considered as separate lines by the program and

individual ones may be moved. For example:



Note that all three dimensions must be selected to move the entire line.

Select the lines to delete:



and drag the lines to the new location.

13.3.7.4 Move text

- Highlight and click on any dimension.
- Drag the dimension to a new location.

13.3.8 Section table menu

Create a table of identical sections with different dimensions. Each table can be transferred as a section type to the *STRAP* properties table as cold-formed sections.

Create table

Refer to Table - create 1304.

Edit table

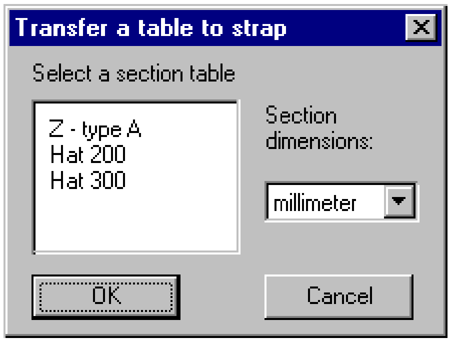
Select a table for the list displayed and edit the dimensions. Refer to Section table - Create 1304.

Delete table

Select a section table from the list displayed; the entire table is deleted

Transfer to STRAP

Add all sections in the current section tables as ***cold-formed*** sections to the *STRAP* property tables.



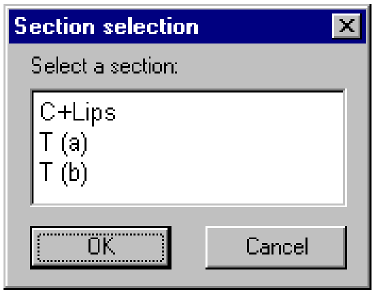
To display a demo video that explains how to create a cold-formed section table and how to transfer it to STRAP:

- *click on* *to start the video*
- *then click on* *to enlarge the display.*

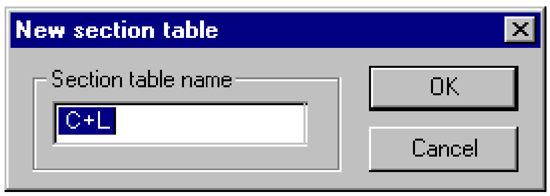
13.3.8.1 Create table

Create a table of identical sections with different dimensions from an existing line section. Each table can be transferred as a section type to the *STRAP* properties table as cold-formed sections.

- select one of the existing line sections:

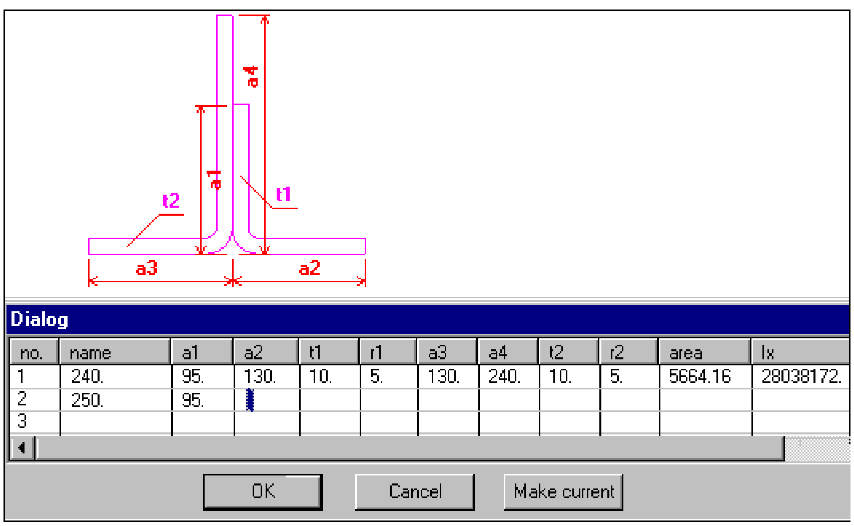


- Enter the name of the table:



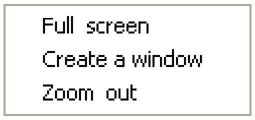
✍more than one table may be created from the same section
✍the name entered here is the type name displayed in the *STRAP* steel postprocessor.

- the program displays the section with the dimension names and a table with a column for each name:



- enter the name and dimensions in each table row; the program automatically calculates the section properties (I, A, etc).
- select Transfer to STRAP 1302 to copy all of the sections in the table as Cold-formed sections to the *STRAP* property tables.
- click Make current to make the highlighted line in the table the new section on the screen (after clicking OK to leave the table).

13.3.9 Zoom menu



Full screen

Display the entire section (maximum size on the full screen).

Create a window

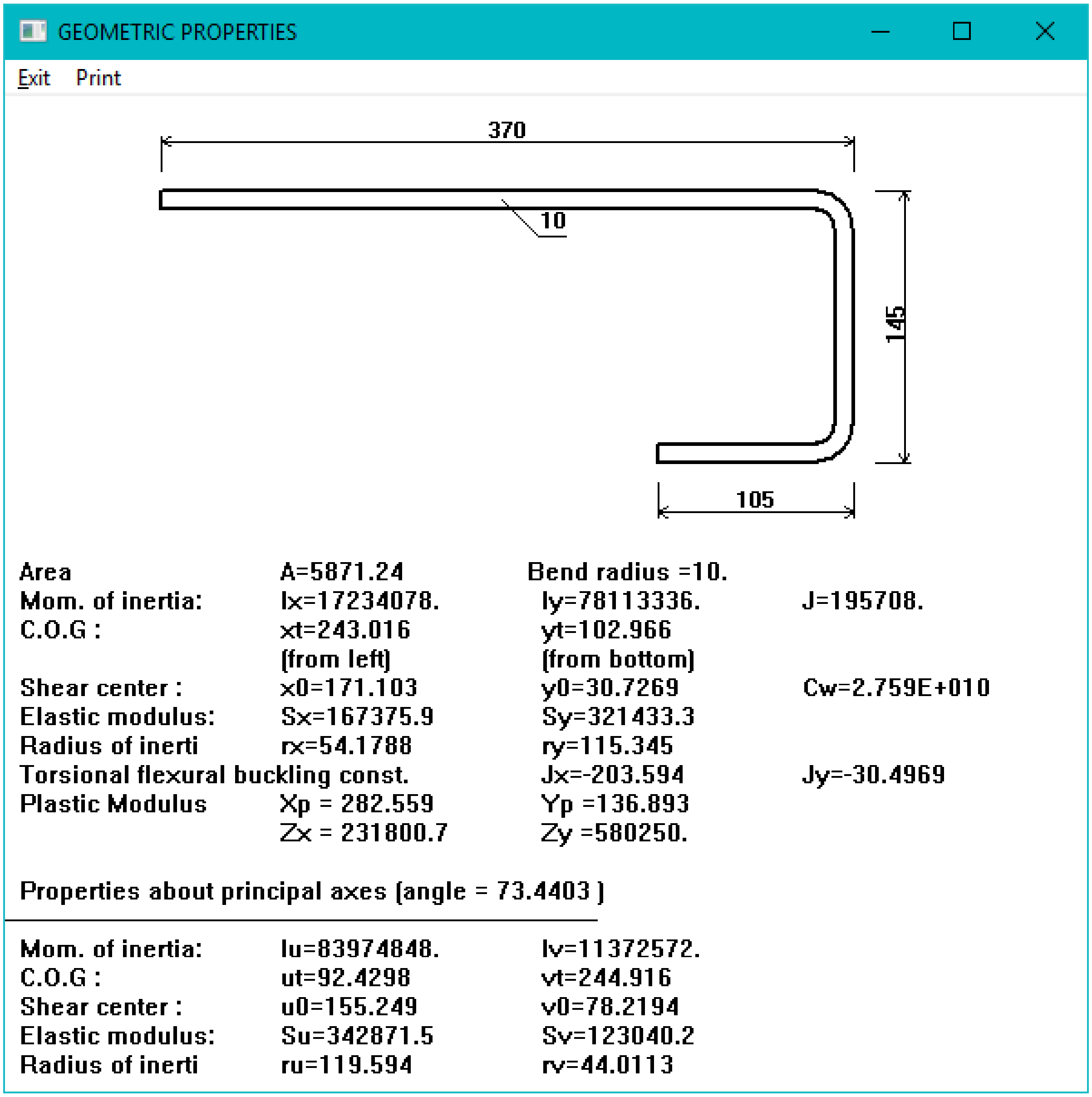
Zoom in on any part of the section by creating a rectangular window; click the mouse with the at the lower-left and upper-right corners of the rectangle.

Zoom out

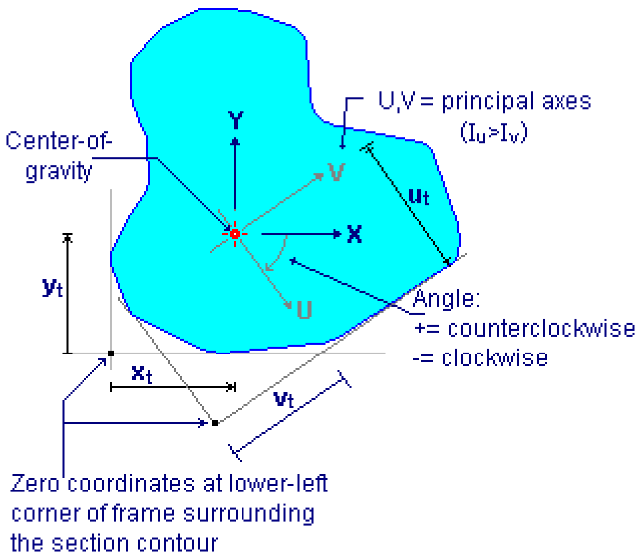
Reduce the size of the section on the screen (in steps of 10%).

13.3.10 Output menu

13.3.10.1 Display properties



Center-of-gravity and principal axes



U,V = principal axes
(Iu>Iv)
Center-of-gravity
Y
V
ut
X
yt
U
Angle:
+= counterclockwise
-= clockwise
xt
vt
Zero coordinates at lower-left corner of frame surrounding the section contour

Moment-of-inertia / Section modulus / Radius-of-inertia

$$I_x = \int dA \cdot y^2 \qquad I_y = \int dA \cdot x^2$$
$$r_x = I_x/A \qquad r_y = I_y/A$$
$$S_x = I_x/y_t \qquad S_y = I_y/x_t$$

$$I_u = \int dA \cdot v^2 \qquad I_v = \int dA \cdot u^2$$
$$r_u = \sqrt{I_u/A} \qquad r_v = \sqrt{I_v/A}$$
$$S_u = I_u/v_t \qquad S_v = I_v/u_t$$

Shear center

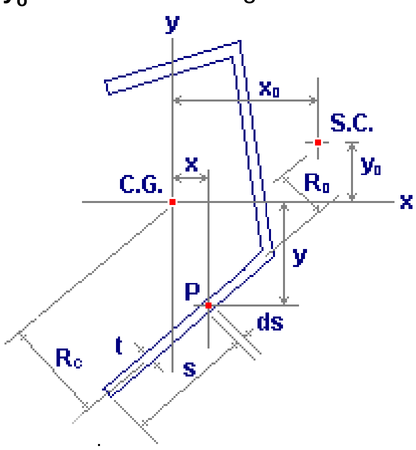
Referring to the AISI - Cold Formed Steel Design Manual - Part III,
Supplementary Information to the August 19, 1986 edition:

Shear center:

$$x_0 = \frac{1}{I_x}\int_0^L w_c\, y\, t\, ds$$
$$y_0 = \frac{1}{I_x}\int_0^L w_c\, x\, t\, ds$$

x_o = distance along the shear centre to the centroid along the principal X-axis.

y_o = distance along the shear centre to the centroid along the principal Y-axis.



Cw - Warping constant:

$$C_w = \int_0^L (w_c)^2\, t - \frac{1}{A}\left[\int_0^L w_o\, t\, ds\right]^2$$

w_c = sectorial coordinate relative to the centroid

w_o = sectorial coordinate relative to the shear center.

Torsional moment-of-inertia

- **Solid sections**
  J is calculated according to the method of finite differences. The method is approximate, but the error for most sections is in the range of 1-2% (4-5% for sections with many holes).
- **Line sections**
  The program calculates J using the membrane analogy method for thin-walled sections (for examples, refer to "Theory of Elasticity", by Timoshenko and Goodier)

Torsional-flexural buckling constant

Referring to the AISI - Cold Formed Steel Design Manual - Part V - 1996 Edition,
Section C3.1.2 - Lateral Buckling Strength:

$$j_x = \frac{1}{2I_y}\left[\int_A x^3 dA + \int_A xy^2 dA\right] - x_o$$

$$j_y = \frac{1}{2I_x}\left[\int_A y^3 dA + \int_A yx^2 dA\right] - y_o$$

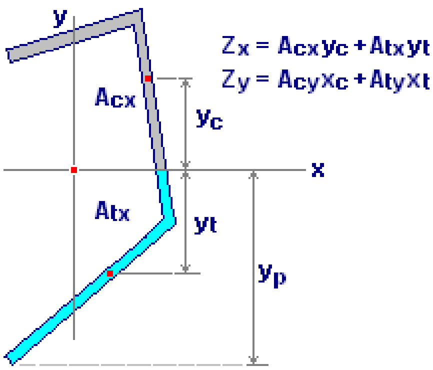
where:

x_o = distance along the shear centre to the centroid along the principal X-axis, taken as negative.

y_o = distance along the shear centre to the centroid along the principal Y-axis, taken as negative.

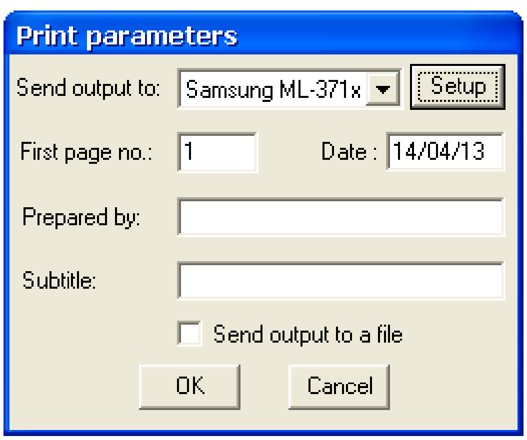
Plastic Modulus

(for line sections only)



13.3.10.2 Print section

Use this option to print the current display directly to the printer or to a file. The printed display is enclosed in a frame and includes a header.



Send output to

Select the output unit, e.g. printer, plotter, etc. The devices must be installed by the "Printers" option in the Windows "Control panel".

Setup

Specify general information for the output device selected:

- paper size
- graphic resolution

- etc.

Options

Define information that will be printed in the header at the top of every printed page:

- **First page no:** page numbering is consecutive
- **Date:** the date format is specified in the Windows "Control panel"
- **Prepared by:**
- **Subtitle:**

Send output to file

☑ to send the drawing to a file.

13.3.10.3 Copy to clipboard

Use this option to copy the properties of the current section to *STRAP*.

- select **Copy to clipboard**
- in the *STRAP* beam properties menu:
  - highlight a property group line and click Define/Revise button
  - click the 

icon.
  - In the next menu select the material and specify the units used when defining the section (the dimensions cannot be converted to any other unit)

The relevant properties (A, I2, etc) are displayed in the section table.

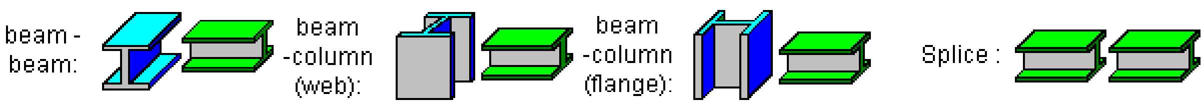
13.4 Connection design

13.4.1 Introduction

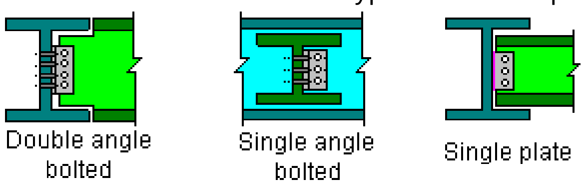
This module designs and details the following standard connections in structural steel models created and analysed with *STRAP*:

- beam-column
- beam-beam
- splices
- base plates

The module is part of the *STRAP* package and cannot run as a stand-alone program.



The user selects the connection type and relevant parameters for each connection. For example:



The program designs the connections, i.e. selects angles, plates, bolts, welds, etc,. and carries out the necessary Code design checks for both the two connected parts and the connector. All load combinations are checked; results are displayed for the critical combination.

For a detailed explanation of the calculation for each connection type, refer to:

AISC 360-05 LRFD/ASD 1334
BS 5950-1 : 2000 1355
EN 1993-1-8 1394

13.4.2 How to use this program

The steel connection design module is part of the *STRAP* package and cannot run as a stand-alone program.

Prior to designing the connections:

- define the model geometry and loads in *STRAP;* solve the model
- complete the design of the structural steel beams and columns in the *STRAP* Steel design module; ***a connection cannot be designed if the connected members have not been 'Computed'***.

To start the connection design module:

- Steel design module : select **File** and **Design connections**. *- or -*
- *STRAP* main menu : **Utilities** and **Connection design**.

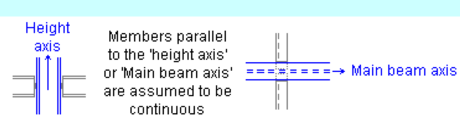
Select 

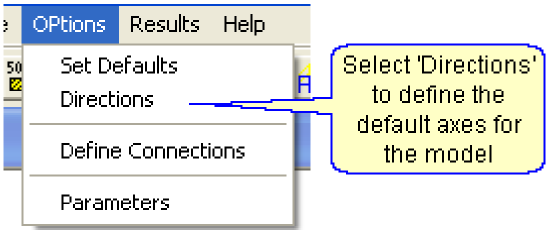
or 

in the bottom side menu.

Define general parameters:

- Define the height axis and the axis of the main beams (girders). This is required by the program to identify the "supporting" member and the "supported" member at each connection:

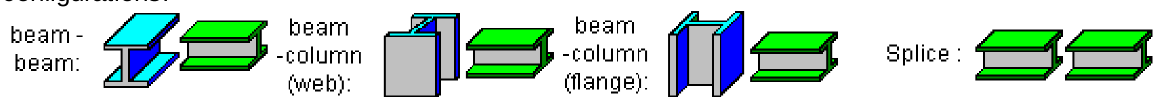




To define the "supporting" and "supported" members for individual connections, select Defaults in the side menu.

Define the connection default parameters:

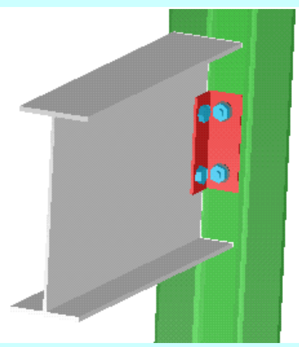
- click Define to define default parameters for ***all*** connections in the model:
  - in the Default connections tab, specify the default connection type for the three connections configurations:



  - in the Connection parameters tab, specify the design code and steel, bolt and weld types.
- click Parameters to define different parameters for specific connections in the model.

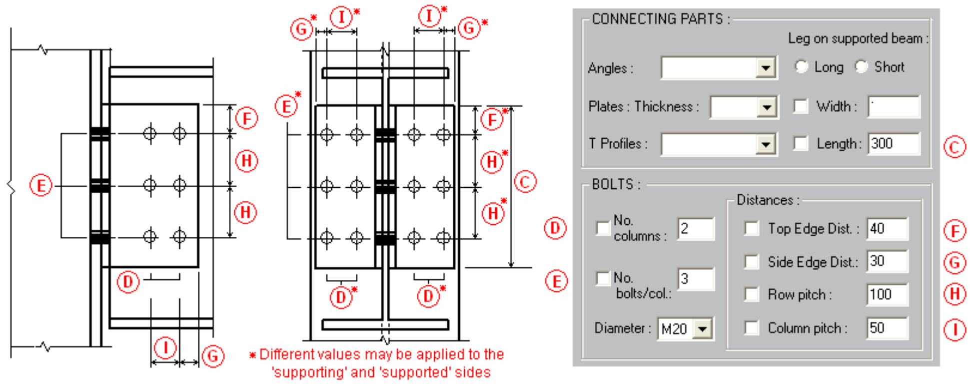
Design the connections and display the results:

- click Display in the side menu.
- select the two connected members.
- the program displays the connection and the design calculations.
- click Render to display a rendered view of the connection:

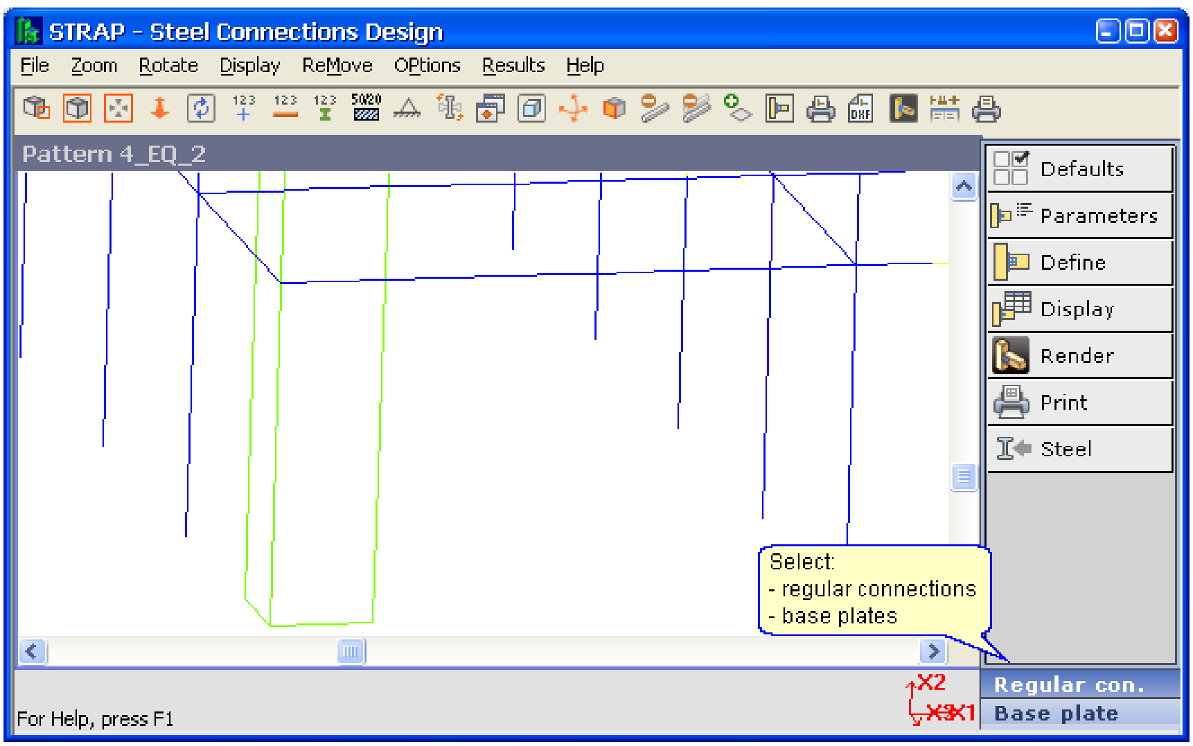


Refine default parameters or specify individual connection parameter:

- click Define or Parameters to refine parameters for the entire model or for specific connections.
- to specify exact plate dimensions, number of bolts, etc., click Parameters and select the Detail connection tab.



13.4.3 Main menu



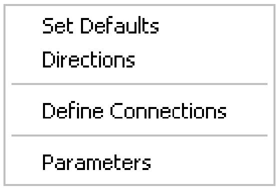
Refer to:

- Toolbar options:
  Options 1314
  Results 1315
- Side menu options:
  Defaults 1317
  Parameters 1322
  Define 1331
  Display 1332
  Render 1333

Refer also to:

How to use this program 1311
Design assumptions 1333

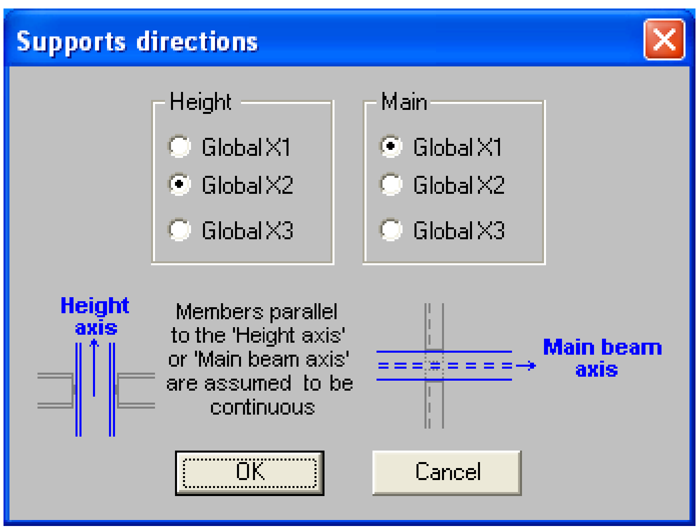
13.4.4 Options



Refer to:
Defaults 1317
Directions 1314
Define connections 1331
Parameters 1322

13.4.4.1 Direction

Define the default directions for the entire model:

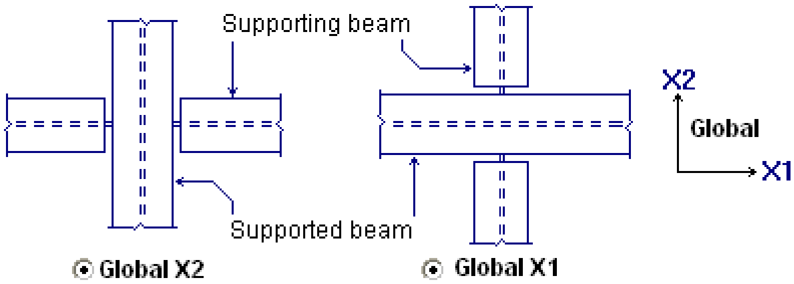


Height axis

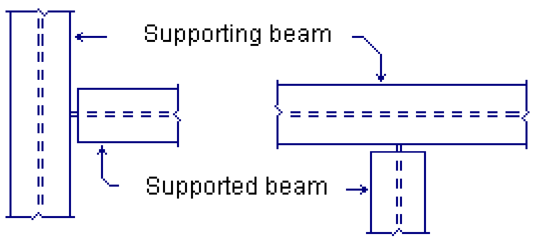
The program assumes that the columns are parallel to the height axis and that all other members are beams supported by the columns.

Main axis

- Node with two or four members:
  The program assumes that the members parallel to the main axis are the supporting (primary) beams and the members not parallel to this axis are the supported (secondary) beams.



- Node with three members ("T" connection):
  The program always assumes that the leg of the T is the supported beam, ***no matter which axis is selected in the menu***.

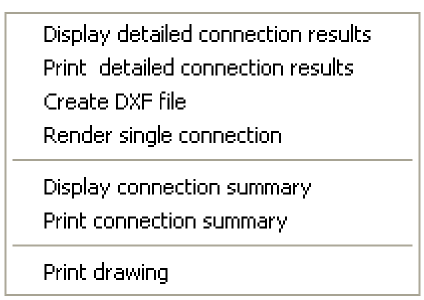


Note:

- to switch the supporting-supported beams for a specific connection, use the 

option in the side menu.

13.4.5 Results



Refer to:
Single connection results 1332
Create DXF file 1316
Render 1333
Connection list 1316

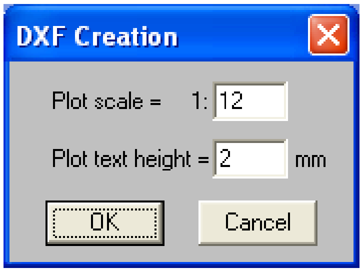
Note:
- The program designs the connections, i.e. selects angles, plates, bolts, welds, etc,. and carries out the necessary design checks for the two connected parts, beam-column or beam-beam.
- All load combinations are checked; results are displayed for the critical combination.

For a detailed explanation of the calculation for each connection type, refer to:
AISC 360-05 LRFD/ASD 1334
BS 5950-1 : 2000 1355

13.4.5.1 Create DXF file

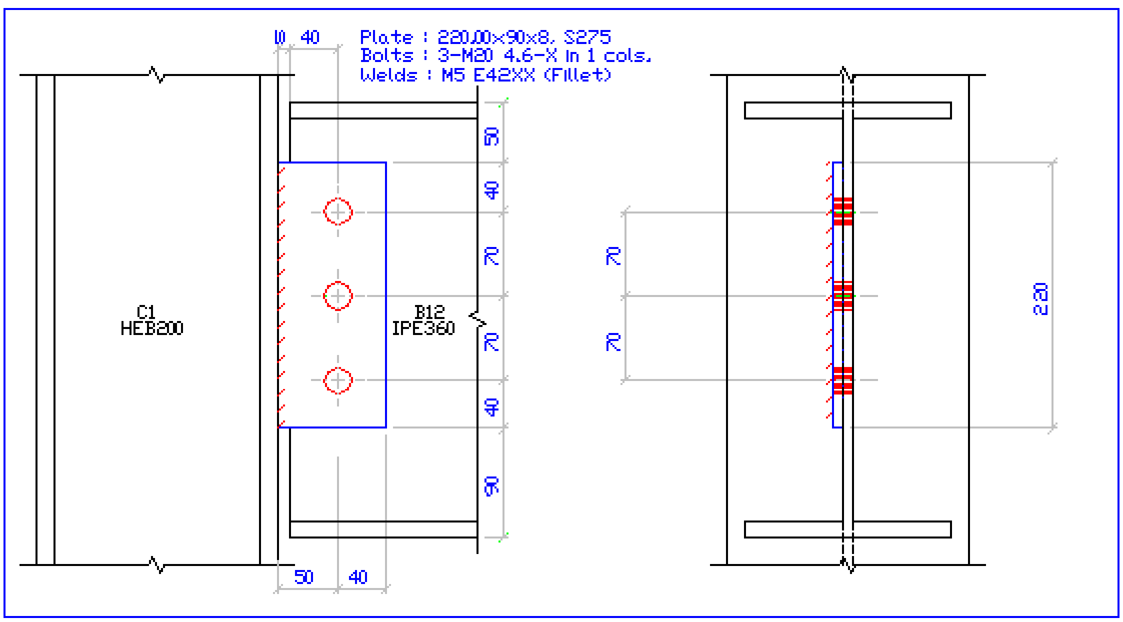
Create a DXF drawing of a single connection:

- select a connection
- enter the name of the DXF file and select a folder; click 
.
- specify the text size



The text will appear with the specified height only if the drawing is plotted with the specified scale; otherwise the text is enlarged/reduced proportionally to the change in the scale.

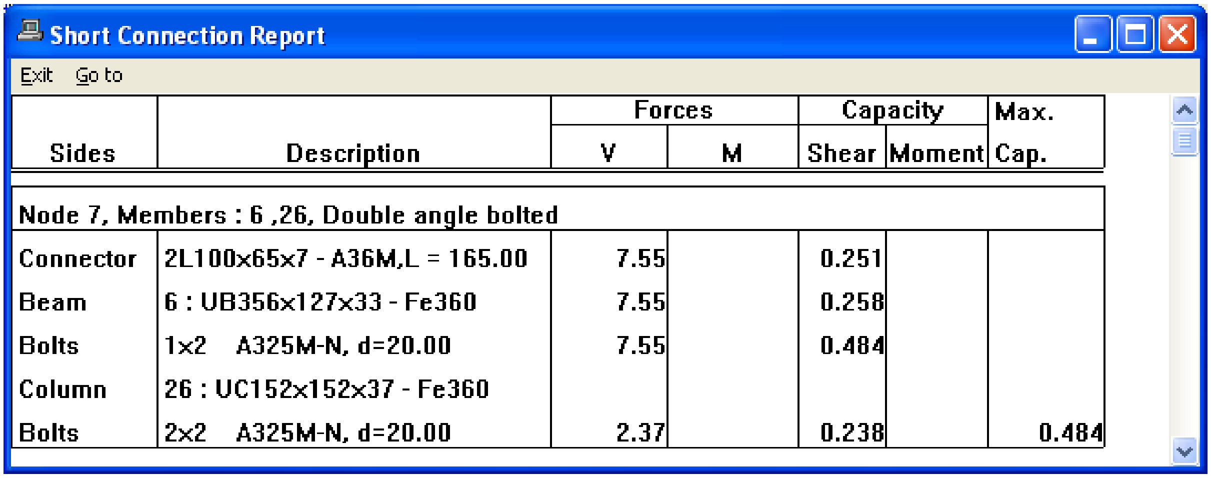
Example:



13.4.5.2 Connection summary

Display a summary table for all connections in the model. For example:

Regular connections:

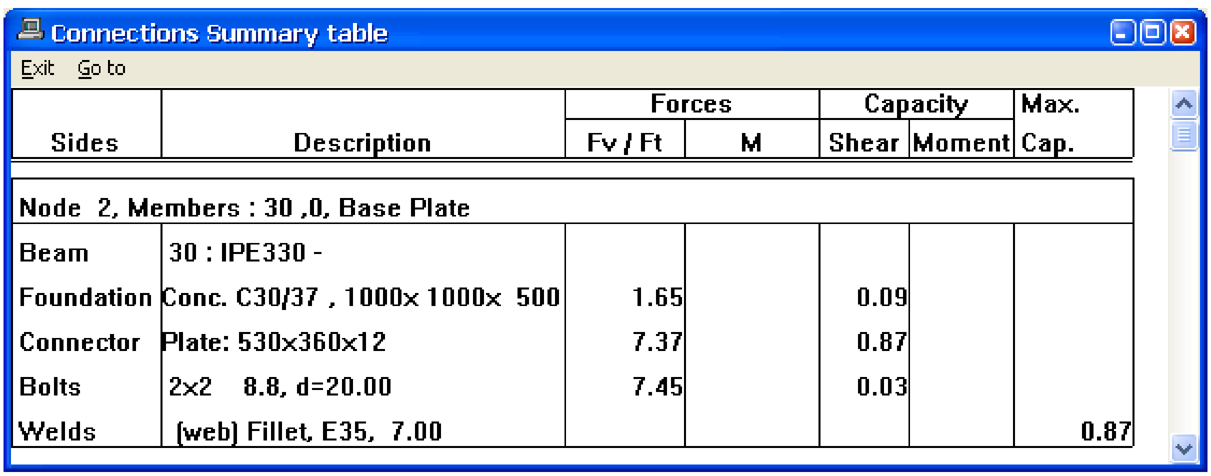


Sides	Description	Forces V	Forces M	Capacity Shear	Capacity Moment	Max. Cap.
Node 7, Members : 6 ,26, Double angle bolted						
Connector	2L100x65x7 - A36M,L = 165.00	7.55		0.251		
Beam	6 : UB356x127x33 - Fe360	7.55		0.258		
Bolts	1x2 A325M-N, d=20.00	7.55		0.484		
Column	26 : UC152x152x37 - Fe360					
Bolts	2x2 A325M-N, d=20.00	2.37		0.238		0.484

The table displays:

- details of all parts of the connection (supporting/supported member, connector, bolts)
- forces (V/M) acting on each part
- capacity factor for each part of the connection
- maximum (governing) capacity factor for the connection.

Base plates:



Sides	Description	Forces Fv / Ft	Forces M	Capacity Shear	Capacity Moment	Max. Cap.
Node 2, Members : 30 ,0, Base Plate						
Beam	30 : IPE330 -					
Foundation	Conc. C30/37 , 1000x 1000x 500	1.65		0.09		
Connector	Plate: 530x360x12	7.37		0.87		
Bolts	2x2 8.8, d=20.00	7.45		0.03		
Welds	(web) Fillet, E35, 7.00					0.87

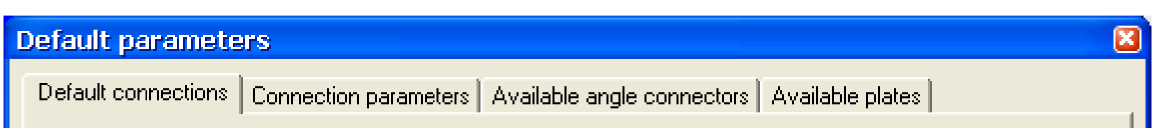
13.4.6 Defaults

Specify the default options for the model:

- Regular connections 1317
- Base plates 1320

13.4.6.1 Connections

Specify the default options for the model:

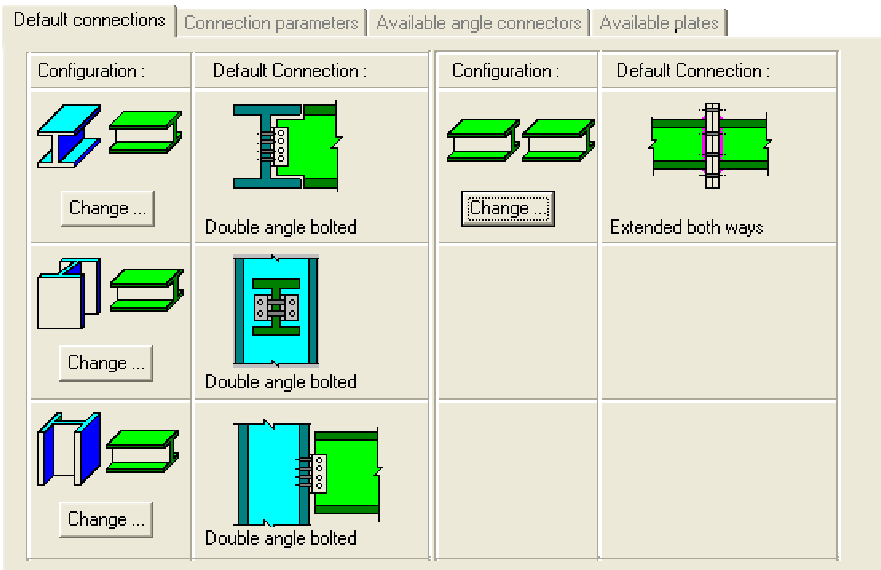


Default connections 1318

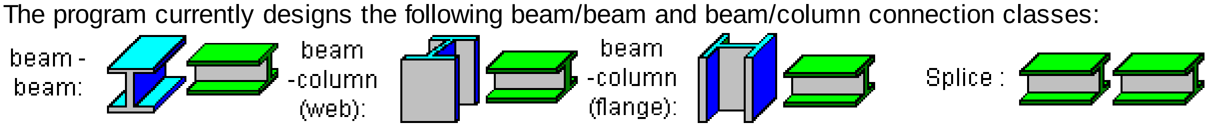
[Connection parameters](1318)
[Available angle connectors](1320)
[Available plates](1320)

13.4.6.1.1 Default connections

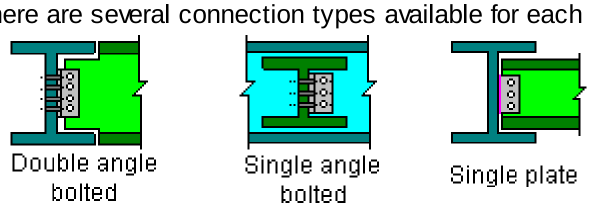
Specify the default connection type for the different connection classes:



The program currently designs the following beam/beam and beam/column connection classes:



There are several connection types available for each one. For example, for beam/beam connections:



To select a different default connection type:

- click 

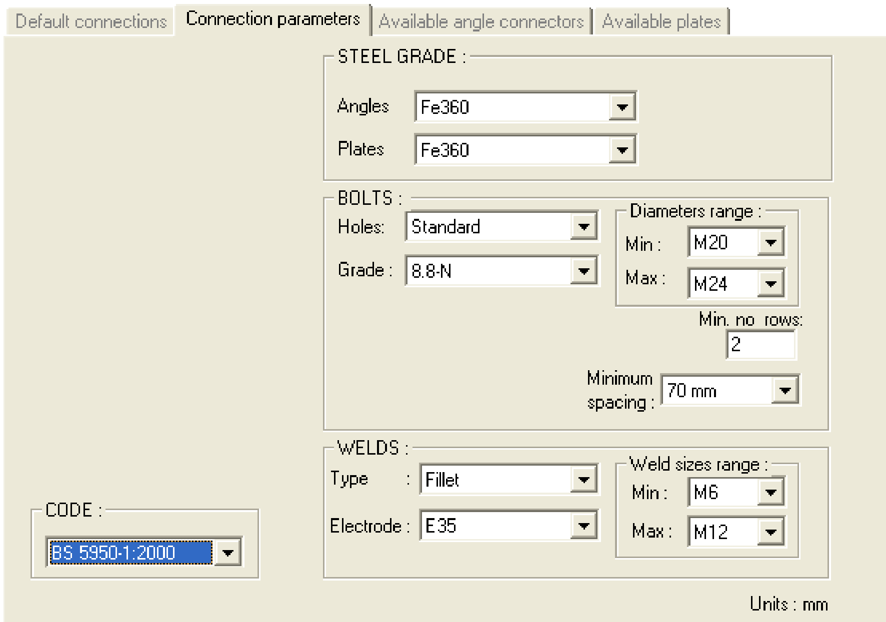
- select ⊙ a connection type from the list.

Note:

- To change the connection type for a specific connection, select [Parameters - connection type](1323).

13.4.6.1.2 Connection parameters

Specify the design code, the default steel grades and default basic parameters for bolts and welds.



Steel grade

- Select the default steel grade for angles and plates from the list.
- If "User-defined" grade is selected, enter the relevant stress values according to the units displayed.

Bolt parameters

Specify the following bolt parameters:
- Steel grade
- Type of holes
- Diameter range; the program selects only bolt diameters within this range.
- Minimum spacing between bolts, either an absolute value of a diameter coefficient
- Minimum number of bolts on either side of any connection

Weld parameters

Specify:
- weld type
- electrode type
- size range; the program selects only weld sizes in this range

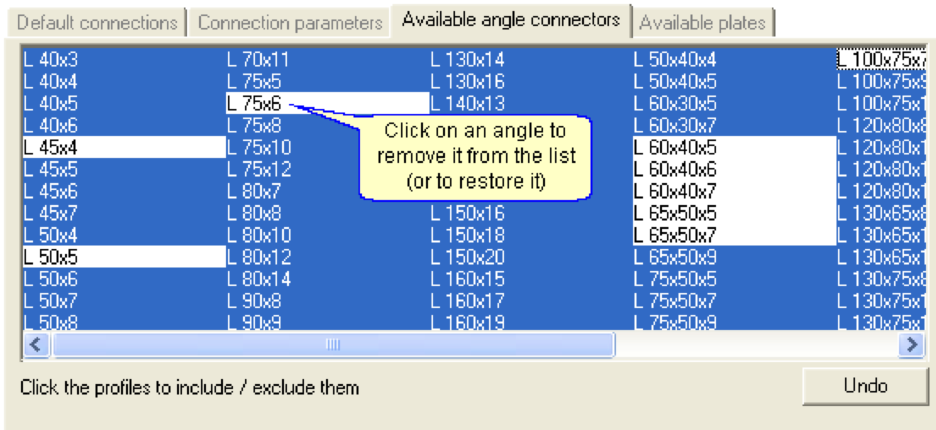
Design code

Select a design code from the list.
- Refer to Design assumptions 1333.

Note:
- To change the parameters for a specific connection, select Parameters - connection parameters 1323 or Parameters - detail connection 1324.

13.4.6.1.3 Available angle connectors

The program automatically selects a suitable angle section for all connection types with an angle connector. The program contains a list of all available angle sections:

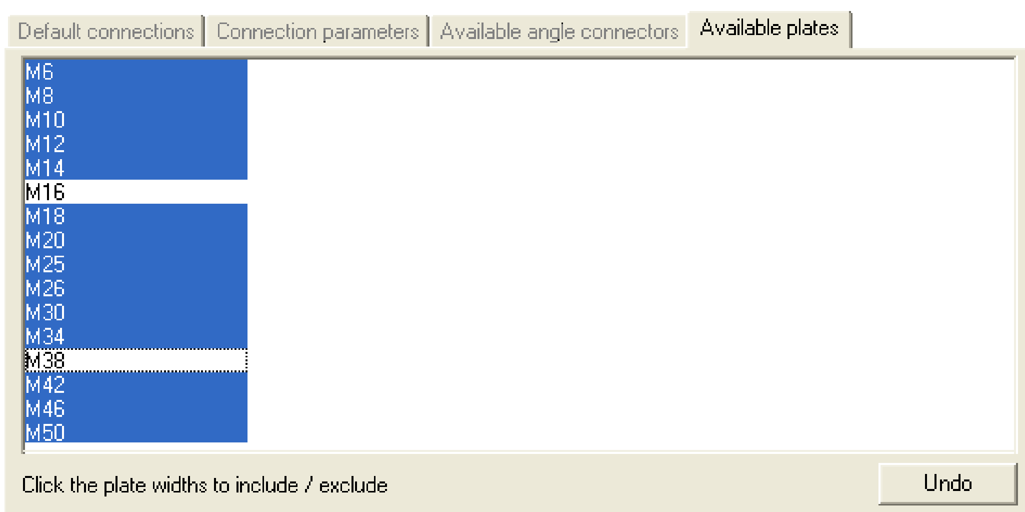


The program selects only angle sections that are highlighted, e.g. L 9x4x5/8.

To remove/restore a section from the list, move the to the section name and click the mouse.

13.4.6.1.4 Available plates

The program automatically selects a suitable plate thickness for all connection types with a plate connector. The program contains a list of all available plate thicknesses:



The program selects only plates that are highlighted, e.g. M20.

To remove/restore a plate from the list, move the to the plate name and click the mouse.

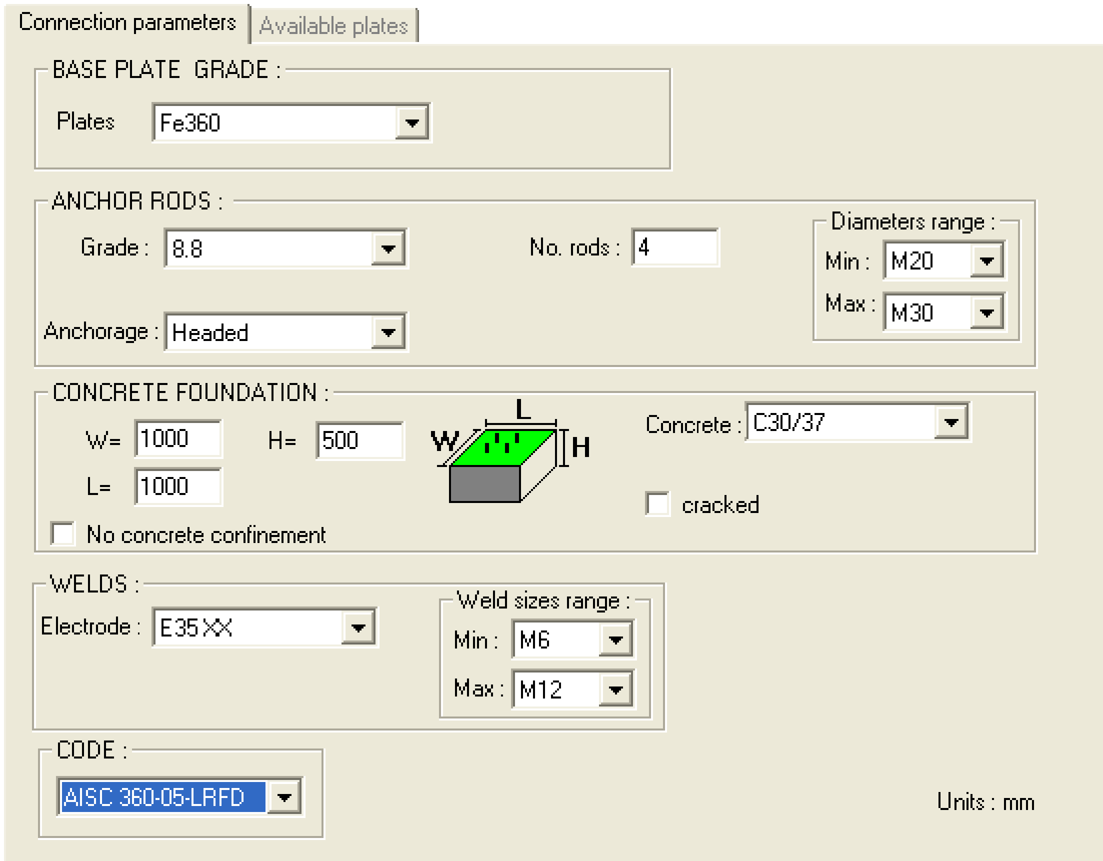
13.4.6.2 Base plates

Specify the default options for the model:

[Connection parameters](1321)
[Available plates](1320)

13.4.6.2.1 Connection parameters

Specify the design code, the default steel grade and default parameters for bolts and welds.



Steel grade

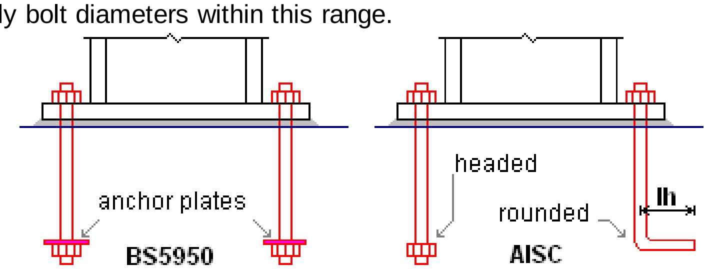
- Select the default steel grade for the base plate from the list.
- If "User-defined" grade is selected, enter the relevant stress values according to the units displayed.

Holding down bolts

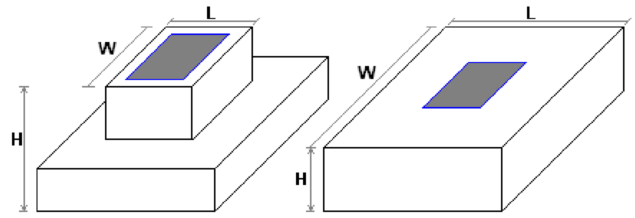
Specify the following bolt parameters:

- Steel grade
- No. of bolts
- Diameter range; the program selects only bolt diameters within this range.
- Anchorage: two options are available:
  BS5950:
  - anchor plates
  - provided by bond

  AISC:
  - headed
  - hooked



Concrete foundation



Enter the dimension of the concrete support:
- H is the height available for the anchor bolts

AISC:
- confined:
  if the concrete is confined the program calculates $(A_2/A_1)^{0.5}$ = 2 when determining $f_{p\,(max)}$
- cracked:
  used for determining ψ_7 in the anchorage calculations.

Weld parameters

Specify:
- electrode type
- size range; the program selects only weld sizes in this range

Design code

Select a design code from the list.
- Refer to Design assumptions 1333.

Note:
- To change the parameters for a specific connection, select Parameters - base plate connection parameters
- in the Parameters menu, ☑ check a box without entering a value to restore the default option to the selected connections.

13.4.7 Parameters

Define different parameters for specific connections; any parameters defined here override those selected in the Define options.
- regular connections 1322
- base plates 1330

13.4.7.1 Connections

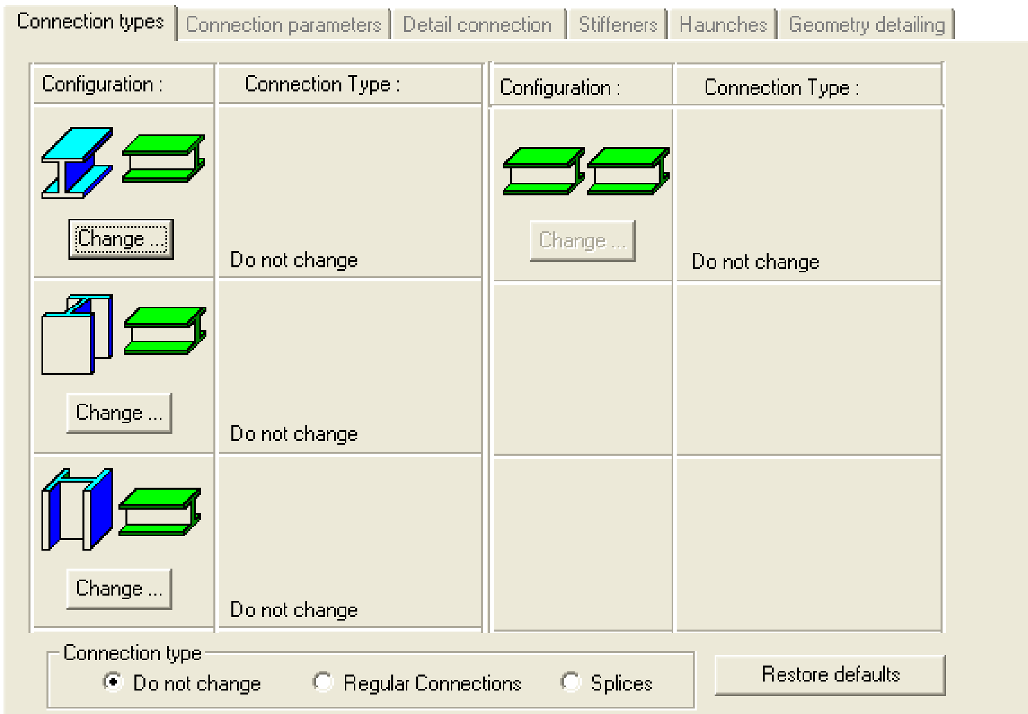
Define different parameters for specific connections; any parameters defined here override those selected in the Define options.

Connection types 1323
Connection parameters 1323
Detail connection 1324
Stiffeners 1328
Haunches 1328
Geometry detailing 1329

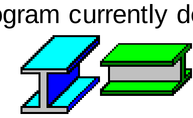
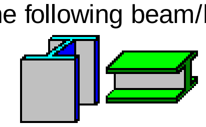
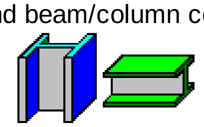
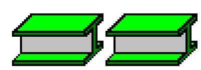
13.4.7.1.1 Connection types

Define different parameters for specific connections; any parameters defined here override those selected in the 

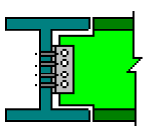
options.

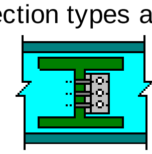


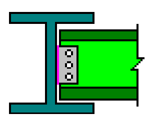
The program currently designs the following beam/beam and beam/column connection classes:

beam - beam:  beam -column (web):  beam -column (flange):  Splice : 

The are several connection types available for each one. For example, for beam/beam connections:


Double angle bolted


Single angle bolted


Single plate

To select a different default connection type:

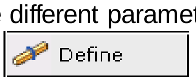
- click 

- select a connection type from the list.

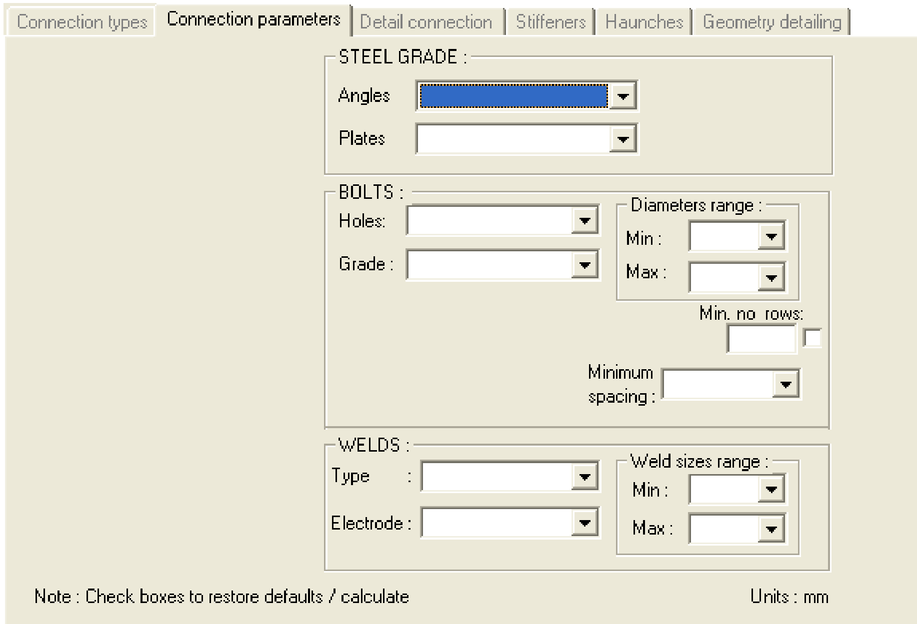
To restore the default parameters to all selected connections, click 

(the defaults from all tabs are restored)

13.4.7.1.2 Connection parameters

Define different parameters for specific connections; any parameters defined here override those selected in the 

options.



Steel grade
Bolts
Welds

Refer to Defaults - connection parameters 1318.

Note:

- check a box without entering a value to restore the default option to the selected connections.

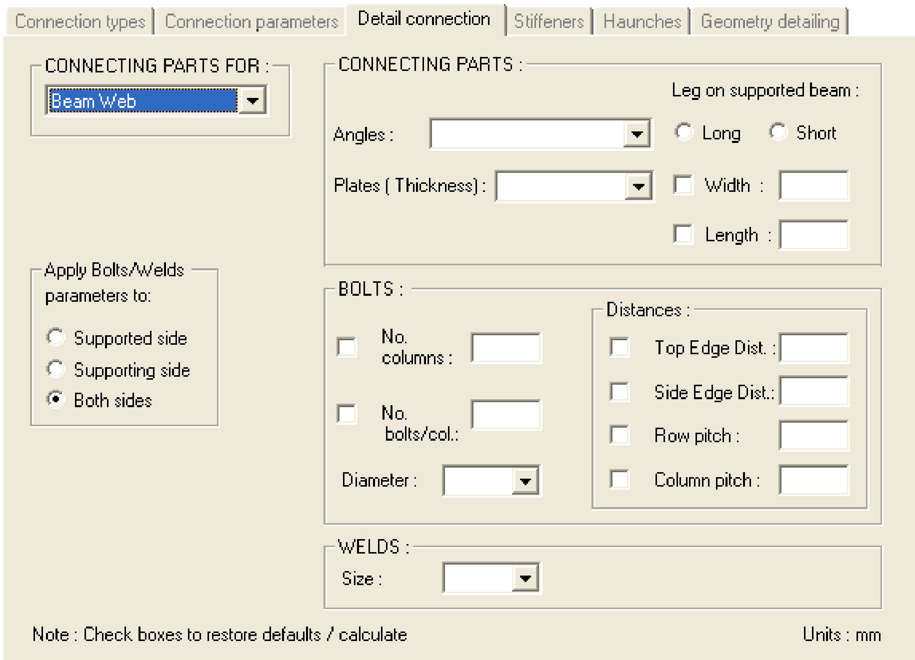
13.4.7.1.3 Detail connection

Use this option to "specify" a particular connection, i.e. define exact parameters for the program to use in designing the connection.

- select the parameters for the connectors, bolts and/or welds
- specify which part of the connection the parameters are applied to.

Refer to the following connection types for an explanation of how the parameters are applied to each connection type:

Double angle 1326
Single (fin) plate 1327
Shear (Flexible) end plate 1327
Moment end plate 1327



Connecting parts

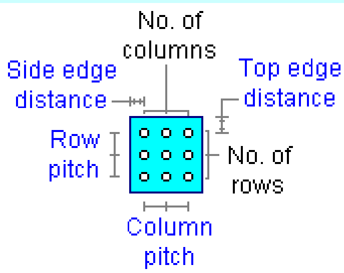
The supporting and the supported members are connected by angles and plates. Specify the section/dimensions/thickness/orientation for these parts:

- Angles
  - select the angle section from the list
  - specify the angle leg attached to the *supported* beam: ⦿ **Long** or ⦿ **Short** (optional)
  - specify the angle **Length** (optional)
- Plates
  - select the plate thickness from the list
  - specify the plate **Width** and **Length** (optional)

Note:

- if ☑ is selected for **Width** or **Length**, the program ignores any values previously specified in this option and uses the calculated dimensions.

Bolts



Note:

- ☑ check a box without entering a value to restore the default option or the calculated value to the selected connections.

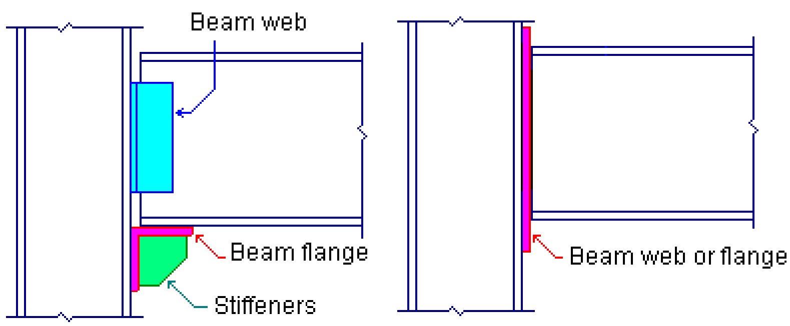
Welds

Select the weld size from the list.

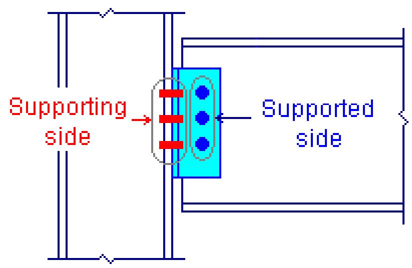
Connecting parts for
Apply parameters to

Any parameter specified in this menu may be applied to any part of the connection:

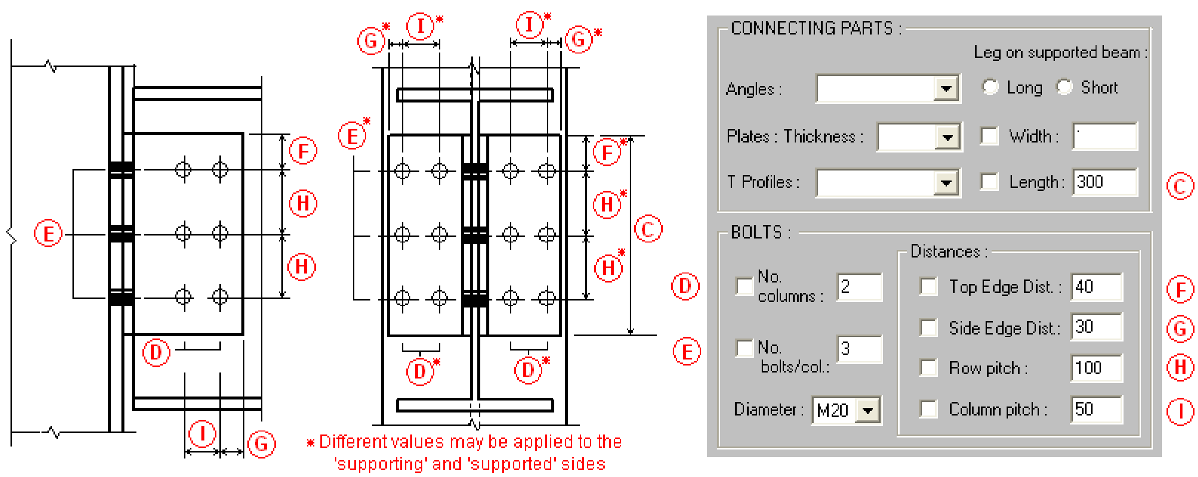
- **Connecting parts for**
  The connection may consist of several parts - connected to the beam web or the beam flange and may include stiffeners. Either option may be selected if the part is connected to both the web and the flange:



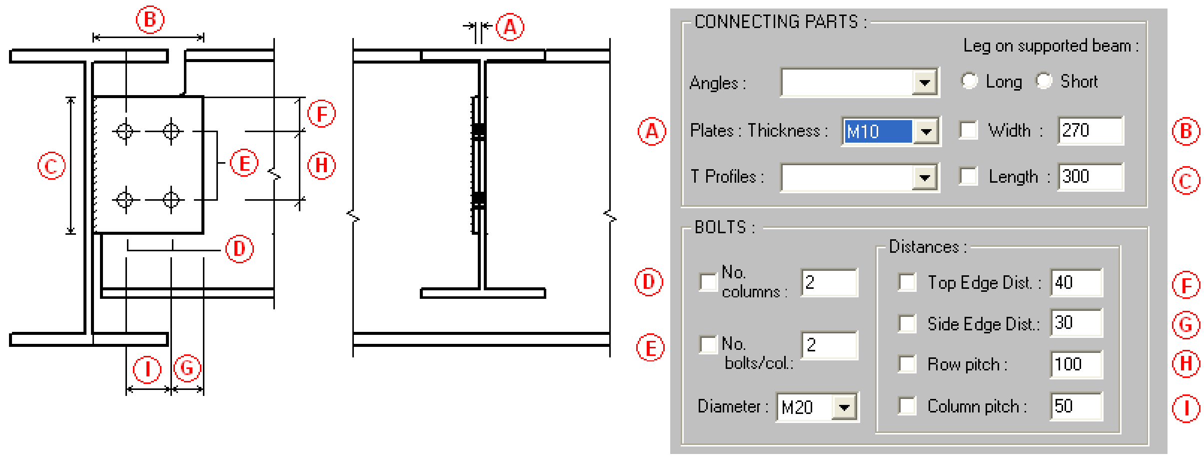
- **Apply parameters to**
  Specify the side of the selected part - the "supported" side, the "supporting" side or both.



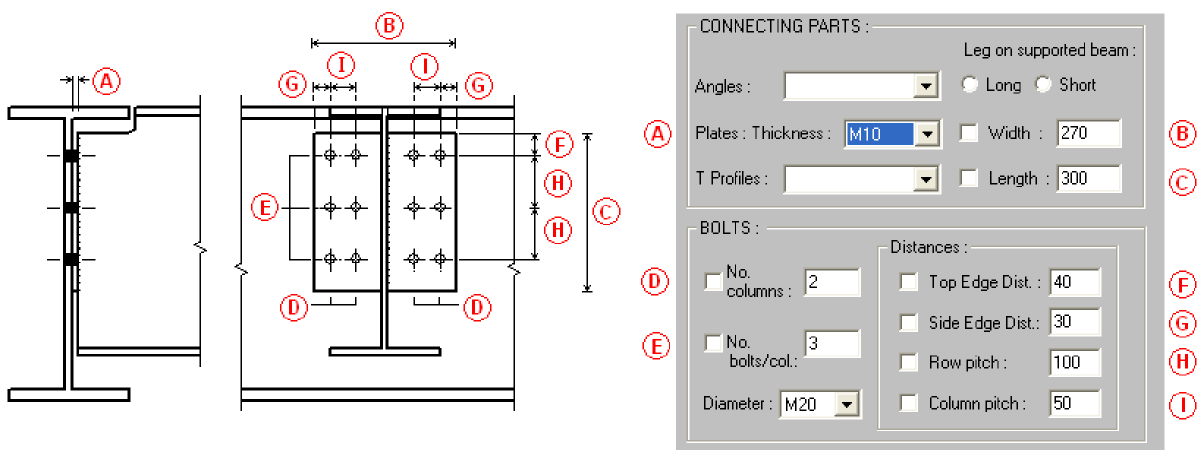
13.4.7.1.3.1 Double angle



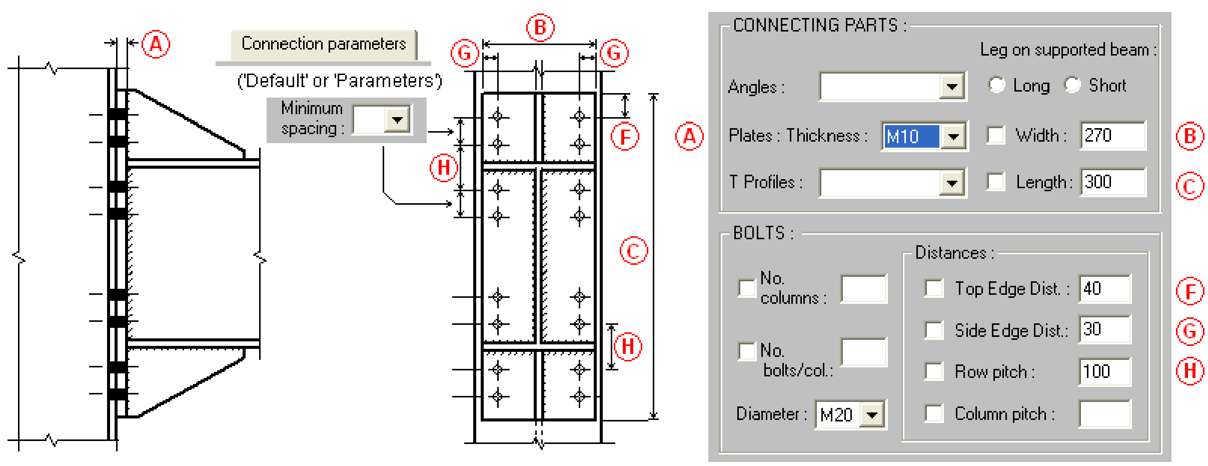
13.4.7.1.3.2 Single plate



13.4.7.1.3.3 Shear end plate

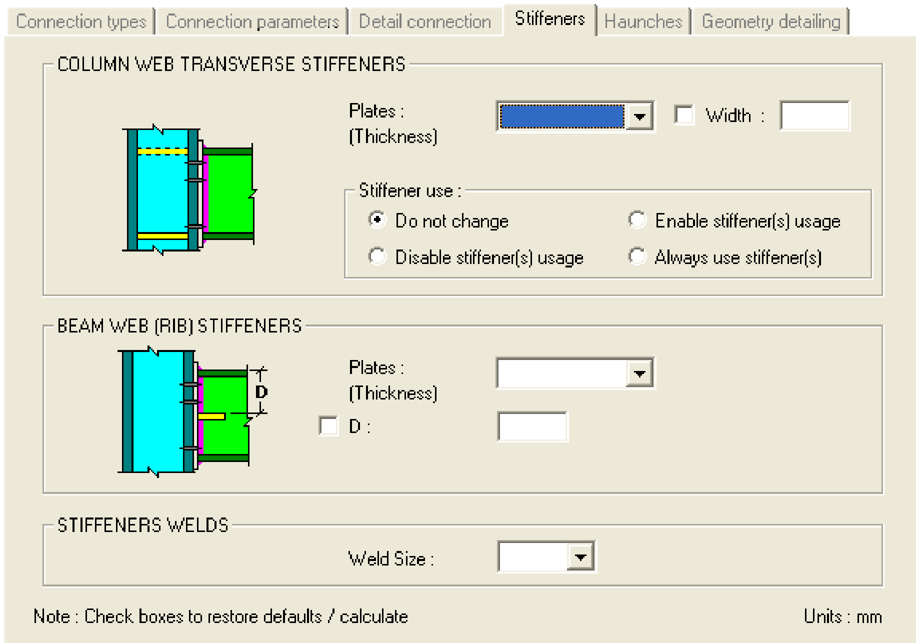


13.4.7.1.3.4 Moment end plate



13.4.7.1.4 Stiffeners

Stiffeners are added automatically to beam-column moment connections where required. Use this option to delete stiffeners or modify the parameters for selected connections.

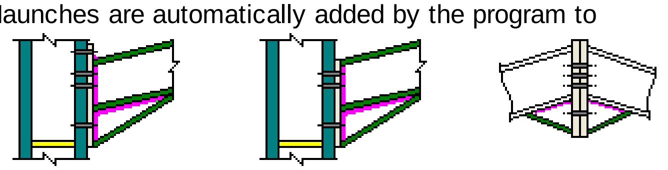


Stiffener use

- **Enable stiffener usage**
  Restore stiffeners to a connection where they were previously disabled.
- **Disable stiffener usage**
  Remove stiffeners from a connection, even if required.
- **Always use stiffeners**
  Detail stiffeners at a connection, even if not required.

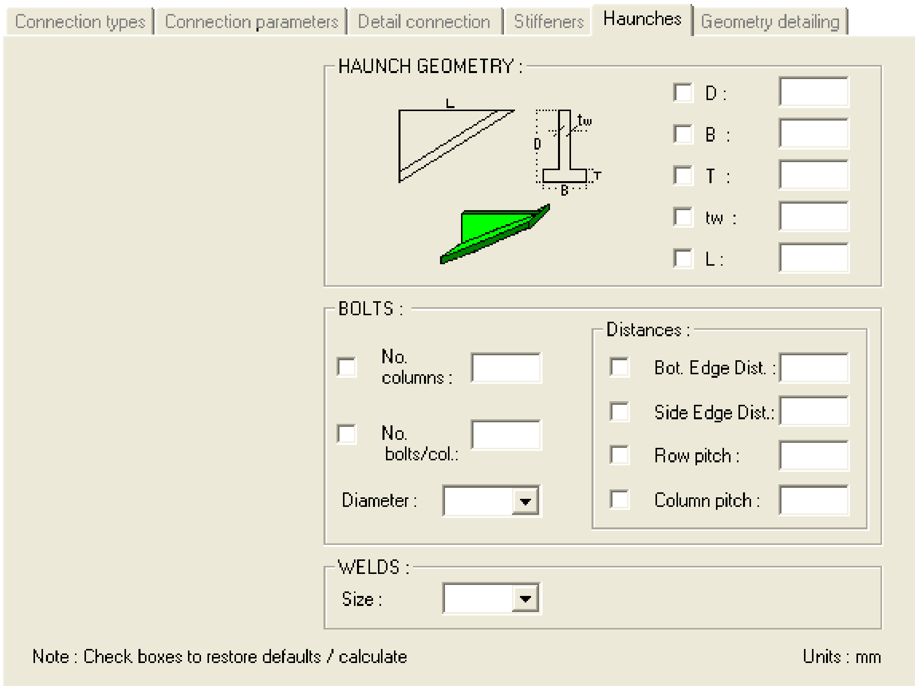
13.4.7.1.5 Haunches

Haunches are automatically added by the program to

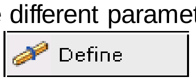


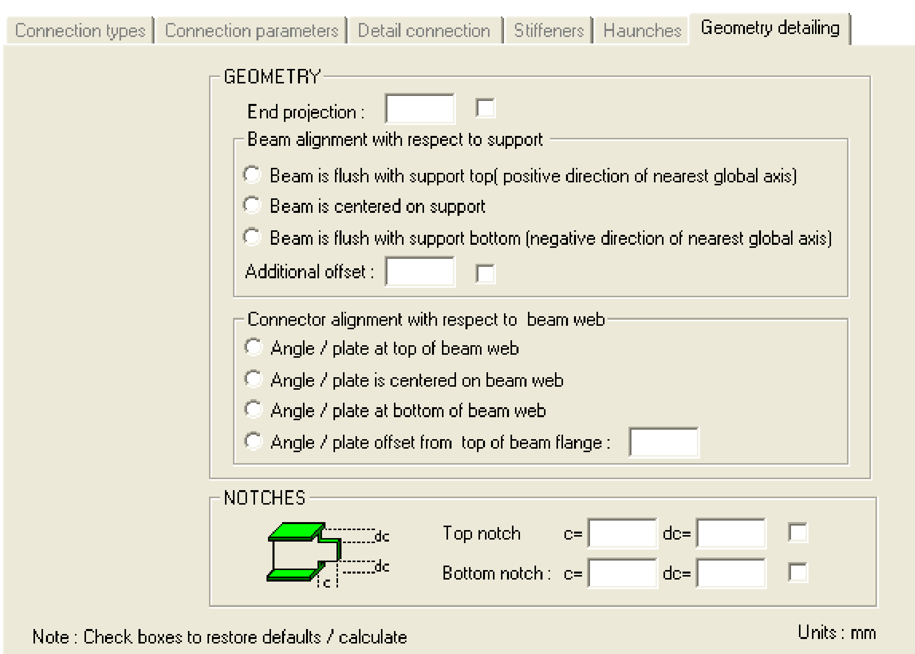
Ext. End Plate Eaves   Flush End Plate Eaves   Portal apex haunch

Use this option to specify dimensions and bolt or weld details for selected connections:

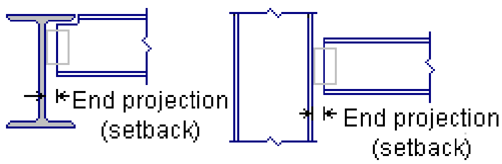


13.4.7.1.6 Geometry detailing

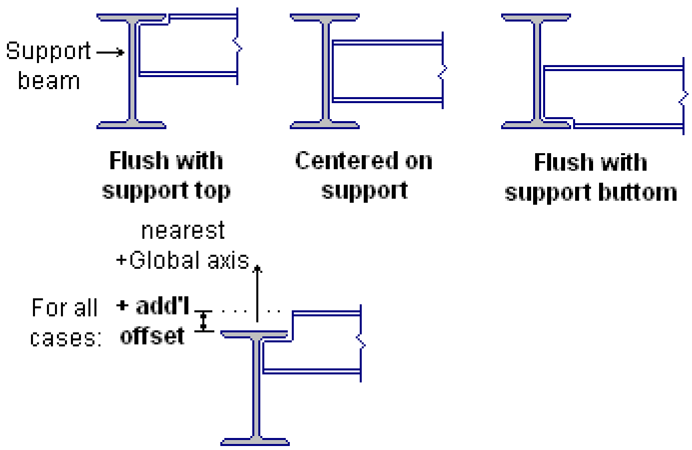
Define different parameters for specific connections; any parameters defined here override those selected in the  options.



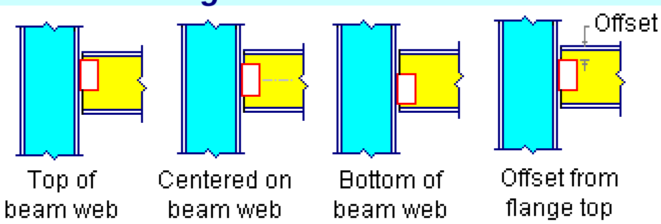
End projection/setback



Align beam

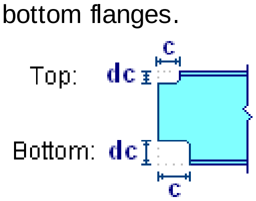


Connector alignment



Notches/copes

Specify the notch/cope dimensions for selected beams. Different values may be specified for top and bottom flanges.



Note:

- ☑ check a box without entering a value to restore the default option to the selected connections.

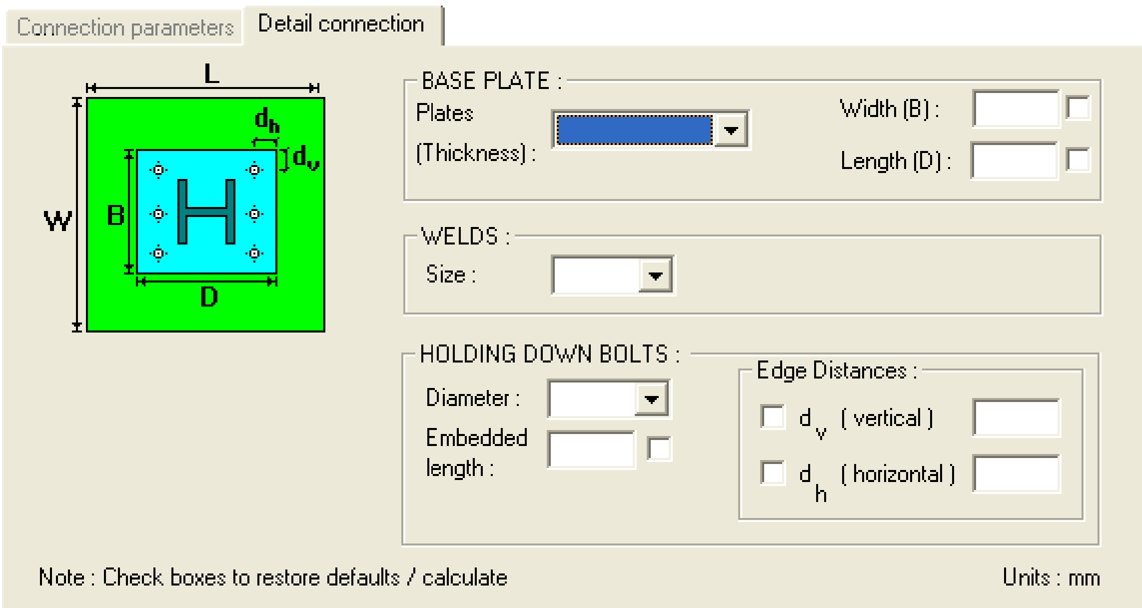
13.4.7.2 Base plates

Define different parameters for specific connections; any parameters defined here override those selected in the [Define] options.

[Connection parameters](1321)
[Detail connection](1331)

13.4.7.2.1 Detail connection

Specify all of the dimensions for a base plate detail and assign the values to a specific location. The program checks the capacity of the base plate for all of the design load combinations:

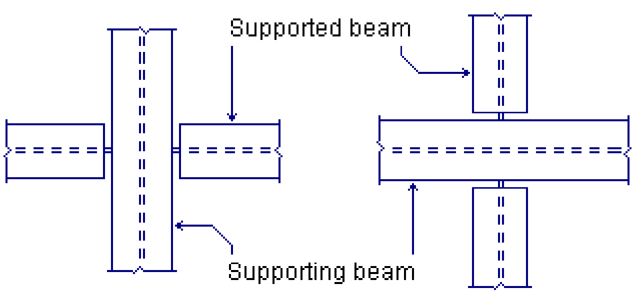


Note:
- check a box without entering a value to use the default values for the selected connections.

13.4.8 Define

Specify the "supported" (secondary) beam and the "supporting" (primary) members in a connection. The orientation defined here overrides the default option specified in the [Options - directions](1314) option.

Select the "supported" beam and the "supporting" beam; there are two possibilities where two beams are connected:



Select the connection using the standard beam connection option, where the supported and the supporting beam are selected separately:

Note:
- a Single connection option is available in the beam selection menu.
- if Individual beams, Select by window, etc. are used to select the members, the program

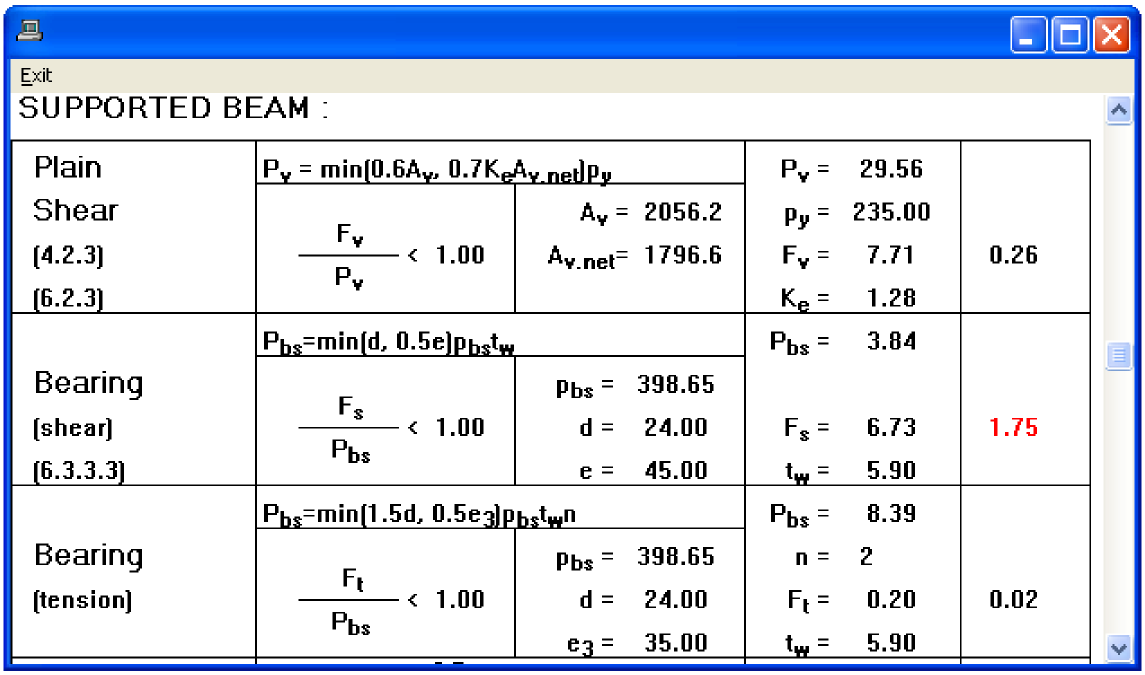
automatically matches up connected members to select the relevant connections.

13.4.9 Display

Display the calculation results for a single connection:

- select the two members (supported and supporting) that form the connection.

The program displays a detailed table listing all of the geometric parameters, load case data and calculations; for example:



Exit

SUPPORTED BEAM :

Plain Shear (4.2.3) (6.2.3)	$P_v = \min(0.6A_v, 0.7K_eA_{v.net})p_y$ $\frac{F_v}{P_v} < 1.00$	$A_v =$ 2056.2 $A_{v.net}=$ 1796.6	$P_v =$ 29.56 $p_y =$ 235.00 $F_v =$ 7.71 $K_e =$ 1.28	0.26
Bearing (shear) (6.3.3.3)	$P_{bs}=\min(d, 0.5e)p_{bs}t_w$ $\frac{F_s}{P_{bs}} < 1.00$	$p_{bs} =$ 398.65 $d =$ 24.00 $e =$ 45.00	$P_{bs} =$ 3.84 $F_s =$ 6.73 $t_w =$ 5.90	1.75
Bearing (tension)	$P_{bs}=\min(1.5d, 0.5e_3)p_{bs}t_wn$ $\frac{F_t}{P_{bs}} < 1.00$	$p_{bs} =$ 398.65 $d =$ 24.00 $e_3 =$ 35.00	$P_{bs} =$ 8.39 $n =$ 2 $F_t =$ 0.20 $t_w =$ 5.90	0.02

Note:

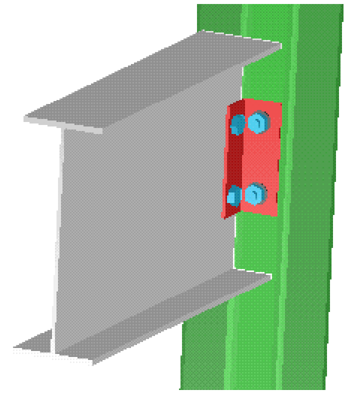
- The program designs the connections, i.e. selects angles, plates, bolts, welds, etc,. and carries out the necessary design checks for the two connected parts, beam-column or beam-beam.
- All load combinations are checked; results are displayed for the critical combination.

For a detailed explanation of the calculation for each connection type, refer to:

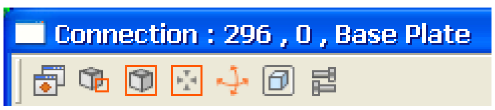
[AISC 360-05 LRFD/ASD](1334)
[BS 5950-1 : 2000](1355)

13.4.10 Render

Display a rendered 3-D drawing of the connection in a separate window. For example:



The following options are available:

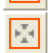


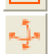
Select:

 Restore the initial view for the selected connection.

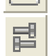
 Zoom in on any part of the connection.

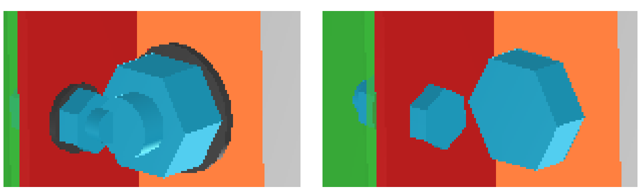
 Display the full drawing (with the current orientation).

 Move the window center; click on any point in the current display.

 Rotate the current display by entering angles about the three display axes (X-Y = screen plane); the angles may be relative either to the initial display axes or to the current orientation.

 Display three different isometric views of the connection; click this icon to toggle through the views.

 Flip the direction of all bolts in the connection. For example:



Note:

- the mouse wheel may be used for zoom in/out, pan and full drawing.
- click and hold the left mouse button for dynamic rotation of the view.

13.4.11 Design assumptions

Select one of the following codes:

AISC 360-05 LRFD/ASD 1334
BS 5950-1 : 2000 1355
EN1993-1-8 1394

13.4.11.1 AISC 360-05 LRFD/ASD

The program designs and checks connections according to AISC-05.

References:
- Steel Construction Manual, 13th Edition
- Manual of Steel Construction, Volume 2
- AISC - Steel Design Guide 4: Extended End Plate Moment Connections.
- AISC - Steel Design Guide 13: Stiffening of Wide-Flange Columns at Moment Connections.
- AISC - Steel Design Guide 16: Flush and Extended Multiple-Row Moment End Plate Connections.

Refer to:
Connection types [1334]
Design checks [1340]

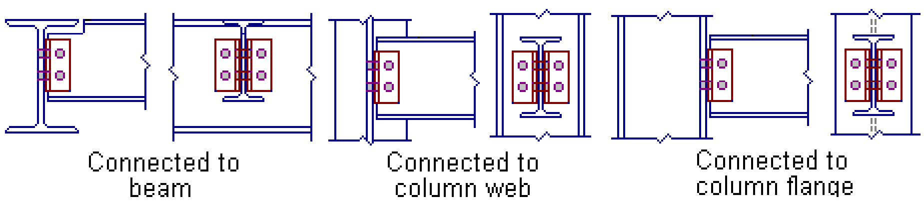
13.4.11.1.1 Connection types

Select one of the following connection types:

- Double angle bolted [1334]
- Double angle welded [1335]
- Shear end plate [1335]
- Single plate shear [1336]
- Moment end-plate [1337]

All other connection types are combinations of the above.

13.4.11.1.1.1 Double angle bolted



The program carries out the following design checks:

Supported beam:
Shear yielding [1344]
Shear rupture [1344]
Bearing strength [1343]
Bolt shear [1340]
Block shear [1344]
Coped beam [1341]

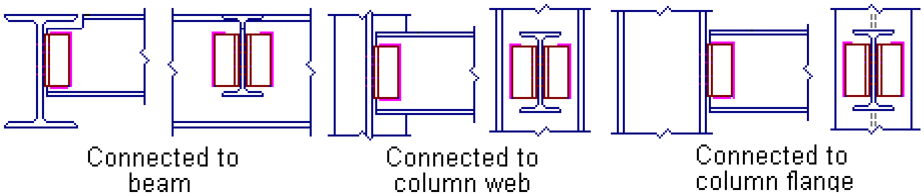
Supporting beam / column
Bearing strength [1343]
Bolt shear [1340]
Bolt shear and tension [1343]

Connecting elements - supported beam side
Shear yielding [1344]
Shear rupture [1344]
Bearing strength [1343]

[Block shear](1344)

Connecting elements - supporting beam / column side
[Bearing strength](1343)
[Block shear](1344)
[Prying action](1342)

13.4.11.1.1.2 Double-angle w elded



- the minimum angle size is 4"x3", where the 3" leg is attached to the web.
- The angle thickness is not less than the largest weld size + $^{1}/_{16}$".

The program carries out the following design checks:

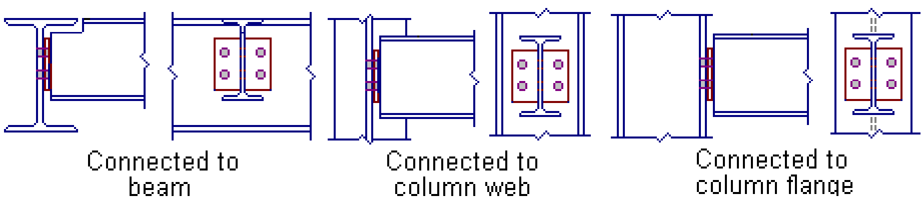
Supported beam:
[Shear yielding](1344)
[Shear rupture](1344)
[Coped beam](1341)

Supporting beam /column
[Bearing strength](1343)
[Bolt shear](1340)
[Bolt shear and tension](1343)

Connecting elements - supported beam side
[Shear yielding](1344)
[Shear rupture](1344)
[Weld group capacity - A](1346)

Connecting elements - supporting beam / column side
[Weld group capacity - B](1346)

13.4.11.1.1.3 Shear end plate



The program carries out the following design checks:

Supported beam:
[Shear yielding](1344)

Shear rupture 1344
Coped beam 1341

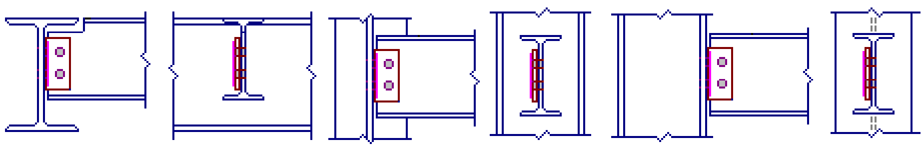
Supporting beam /column

Bearing strength 1343
Bolt shear and tension 1343

End plate

Shear yielding 1344
Shear rupture 1344
Bolt shear 1340
Bearing strength 1343
Block shear 1344
Weld strength 1351
Prying action 1342

13.4.11.1.1.4 Single plate shear



The program carries out the following design checks:

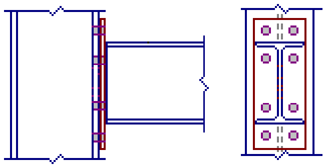
Supported beam:

Shear yielding 1344
Shear rupture 1344
Bearing strength 1343
Bolt shear 1340
Block shear 1344
Coped beam 1341

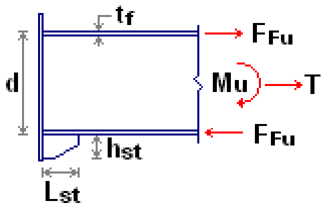
Single plate

Shear yielding 1344
Shear rupture 1344
Bearing strength 1343
Block shear 1344
Maximum plate thickness 1345
Flexural yielding 1344
Flexural rupture 1345
Plate buckling 1345
Weld group strength 1351

13.4.11.1.1.5 Moment end-plate



- If an axial force acts on the beam, the program transforms the force into an equivalent moment which is added/subtracted to the moment in the beam:
  $M = M_u \pm T/2\ (d - t_f)$



- The force in the flange is calculated as:
  $F_{Fu} = M/(d - t_f)$
- For a connection with a stiffener, the stiffener dimensions are calculated according to:
  $L_{st} = h_{st}/\tan 30°$
- The stiffener welds are always $^5/_{16}$" (no calculation)
- The stiffener thickness is calculated as:
  $t_s = t_{wb}\ (F_{yb}/F_{ys})$
  t_{wb} = beam web thickness
  F_{yb} = specified minimum yield stress of the beam
  F_{ys} = specified minimum yield stress of the stiffener

- The program initially assumes a "thin end plate", i.e. a thin plate and thick bolts.
  - The program then calculates the required plate thickness required by the moment **M**.
  - The bolt capacity is then checked according to "prying action"; if the capacity is not sufficient, the program increases the plate thickness and recalculates
  - If the thickness is equal to or greater than the minimum thickness required for the "Thick end plate" method, the program uses the Thick End Plate method to calculate the thickness
- When calculating plate thickness, the program assumes that the plate width is not greater than the beam flange width + 1".

The program carries out the following design checks:

Supported beam:

Shear yielding 1344
Shear rupture 1344

Supporting column

Bearing strength 1343
Bolt shear 1340
Column web yielding 1347
Column web buckling 1347
Column web crippling 1347
Local flange bending 1350

End plate

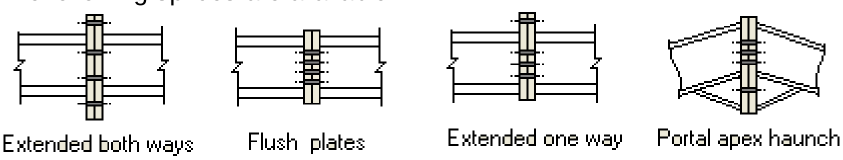
End plate thickness 1348

13.4.11.1.1.6 Splices

The program designs splices at node locations when two *STRAP* beams that form a continuous line are selected. ***Splices can be designed only for two identical sections***.

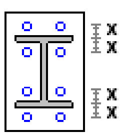
The splices are designed similar to moment end-plate connections 1337.

The following splices are available:



Extended both ways:

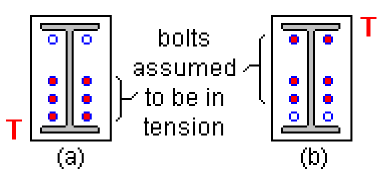
The splice is always symmetric with 8 bolts, 4 on each side, with 2 bolts on each extension.



The program assumes that only the bolts on one side are in tension.

Flush plates:

The program concentrates the required bolts on the tension side and add a pair of bolts on the compression side. Refer to Figure (a). The "tension side" is determined from the largest moment.



The program also checks the capacity of the connection for a moment with the opposite sign. All bolts are assumed to be in tension, except for the pair nearest the compression face. Refer to figure (b).

13.4.11.1.1.7 Base plate

The design of base plates is carried out according to:

AISC Steel Design Guide 1
Base Plate and Anchor Rod Design
Second Edition, 2006
J.M. Fisher & L.A. Kloiber

The base plates can be classified into three groups:

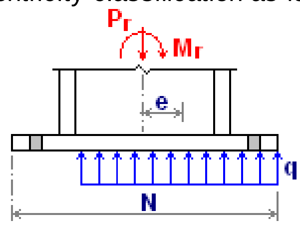
- plates with axial loads only
- plates with small moments (small eccentricity)
- plates with large moments (large eccentricity)

For small eccentricities the axial force and moment are resisted by bearing only. For large eccentricities it is necessary to use anchor rods to resist uplift.

The program determines the eccentricity classification as follows:

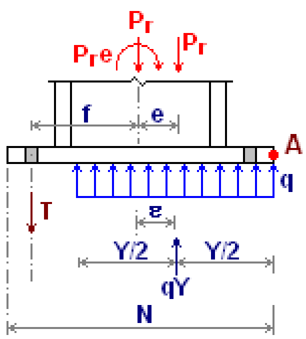
- $\mathbf{e = M_r/P_r}$
- $\mathbf{e_{crit} = N/2 - P_r/(2q_{max})}$

$e = \mathbf{e}_{crit}$: small eccentricity
$e > \mathbf{e}_{crit}$: large eccentricity



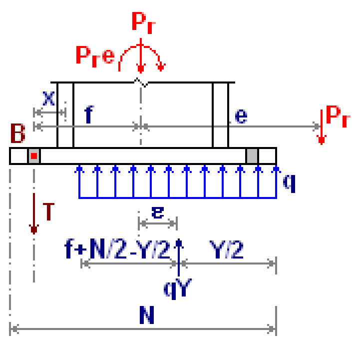
Small eccentricity:

$\mathbf{Y = N - 2e}$
$\mathbf{f_p = P_r/(BY)}$ = bearing stress



Large eccentricity:

$\mathbf{Y = (f + N/2) \pm [(f + N/2)^2 - 2P_u(e + f)/q_{max}]^{1/2}}$
$\mathbf{f_p = P_r/(BY)}$ = bearing stress
$\mathbf{T = q_{max}Y - P_u}$



The maximum allowable bearing stress, $\mathbf{f_{p(max)}}$ is calculated as follows:

$$\mathbf{f_{p(max)} = \phi(0.85f'_c)(A_2/A_1)^{0.5}}$$

where:

ϕ = strength reduction factor for bearing :
ϕ = 0.65 (LRFD)
ϕ = 0.40 (ASD)
$\mathbf{f'_c}$ = specified compressive strength of concrete
$\mathbf{A_2}$ = maximum area of the portion of the supporting surface that is geometrically similar to and concentric with the loaded area
$\mathbf{A_1}$ = area of the base plate

Note:

- $\mathbf{(A_2/A_1)^{0.5}}$ = 2
- $\mathbf{(A_2/A_1)^{0.5}}$ = 1 if ☑ **No concrete confinement** is selected

The program carries out the following design checks:

Bolt tension strength 1340
Breakout strength 1351
Pullout strength 1352
Concrete bearing 1352
Plate yielding 1353
Bolt shear 1340
Breakout strength (shear) 1354
Shear-tension interaction 1354
Weld tension 1354 (column flange)
Weld shear 1354 (column web)

13.4.11.1.2 AISC - Design checks

Select one of the following design checks:

Bolt shear 1340
Coped beams 1341
Prying action 1342
Bolts - combined tension & shear 1343
Bearing strength at bolt holes 1343
Shear yielding and rupture 1344
Block shear strength 1344
Flexural yielding 1344
Maximum plate thickness 1345
Flexural rupture strength 1345
Plate buckling strength 1345
Weld group strength - A 1346
Weld group strength - B 1346
Column web yielding 1347
Column web buckling 1347
Column web crippling 1347
End plate thickness 1348
Bolt rupture with prying 1348
End plate - miscellaneous 1350
Local flange bending 1350
Weld strength 1351

Refer also to:

- AISC - Connection types 1334

13.4.11.1.2.1 Bolt Shear/tension

The program calculates the bolt shear strength, $\boldsymbol{R_n}$, according to section J3.6 of the AISC-05 Code:

$$\boldsymbol{R_n = F_n A_b} \qquad \text{(J3-1)}$$

where:

- $\boldsymbol{F_n}$ **=** nominal tension or shear stress, from Table J3.2
- $\boldsymbol{A_b}$ = nominal unthreaded body area

The program carries out the following design check:

- LRFD: $\boldsymbol{V / (\phi\, n_s\, R_n) = 1.00}$

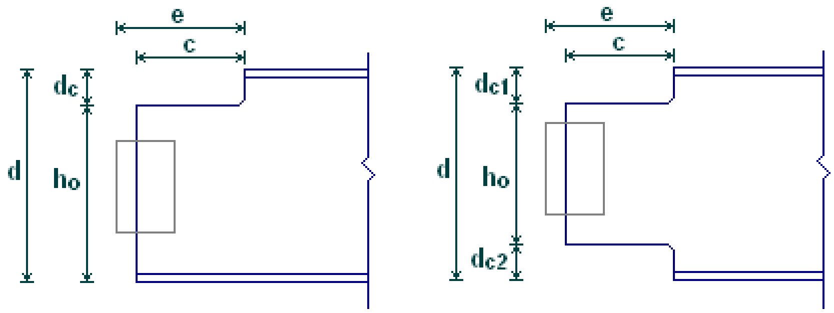
- ASD: $\boldsymbol{V / (n_s\, R_n/\Omega) = 1.00}$

where:
$\boldsymbol{n_s}$ = number of bolt shear planes
ϕ = 0.75
Ω = 2.00
V = the resultant shear force on the bolt group = $\sqrt{(V^2_{vert} + V^2_{horiz})}$

For eccentrically loaded bolt groups, the program uses the "*Instantaneous Center of Rotation Method*" outlined on Page 7-6 of the Steel Construction Manual, 13th Edition.

13.4.11.1.2.2 Coped beams

Coped beam design checks are carried out according to Appendix B of Manual of Steel Construction, Volume 2.



- Beam coped at top flange only

  The program checks the following dimension constraints and displays a warning if they are exceeded:
  c = 2d and **d_c = 0.5d**

  The program calculates the buckling stress of the compressed part of the web in the area of the cope according to:

$$F_{cr} = \frac{\pi^2 E}{12\,(1-\nu^2)} \left(\frac{t_w}{h_o}\right)^2 f\, k$$

  where:
  E = modulus of elasticity
  t_w = web thickness
  υ = Poisson ratio

$$\frac{c}{h_o} \le 1.0 \qquad k = 2.2\left(\frac{h_o}{c}\right)^{1.65}$$

$$\frac{c}{h_o} > 1.0 \qquad k = 2.2\left(\frac{h_o}{c}\right)$$

$$\frac{c}{d} \le 1.0 \qquad f = 2\left(\frac{c}{d}\right)$$

$$\frac{c}{d} > 1.0 \qquad f = 1 + \left(\frac{c}{d}\right)$$

- Beam coped at bottom flange only

  The program calculates the bending capacity only without considering the buckling stress.

- Beam coped at both flanges

  The program checks the following dimension constraints and displays a warning if they are exceeded:

c = 2d and $d_c = 0.2d$

The program calculates the buckling stress of the compressed part of the web in the area of the cope according to:

$$F_{cr} = 0.62\pi E \frac{t_w^2}{c\, h_o} F_d$$

$$F_d = 3.5 - 7.5\left(\frac{d_c}{d}\right)$$

When the top and bottom copes are not identical, the program uses

$d_c = \max(d_{c1}, d_{c2})$

$c = \min(c_1, c_2)$

The capacity for a coped section is calculated from the minimum of:

F_{cr} = buckling stress and F_y = yield stress

LRFD: $\phi R_n = [0.9 \min(F_y,F_{cr})\ S_n]/e$

ASD: $\phi R_n = [0.6 \min(F_y,F_{cr})\ S_n]/e$

S_n = reduced modulus of elasticity of the section in the area of the cope.

13.4.11.1.2.3 Prying action

In connections where the tension force is increased due to prying action, the program calculates the allowable force for the critical bolt. The program also displays the prying force.

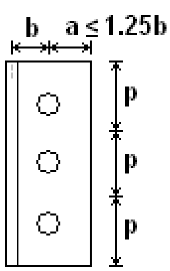
The tension force capacity of the most highly stressed bolt is calculated as:

$$T_n = R_T(1+\delta\alpha')\left(\frac{t}{t_c}\right)^2$$

LRFD: $$t_c = \sqrt{\frac{4.44 R_T b'}{p F_u}}$$

ASD: $$t_c = \sqrt{\frac{6.66 R_T b'}{p F_u}}$$

$$\alpha' = \frac{1}{\delta(1+\rho)}\left[\left(\frac{t_c}{t}\right)^2 - 1\right] \le 1.0$$



p = min length of the flange, parallel to stem or to leg, tributary to each bolt

R_T = $0.75 A_b \min(F_{nt}, F_{nt}')$

b' = $b - d/2$

a' = $a + d/2$

ρ = b'/a'

δ = $1 - d'/p$

d' = width of bolt hole

d = bolt diameter

F_{nt} = nominal tensile stress

F'_{nt} = nominal tensile stress modified to include the effects of shear stress

The program also displays the prying force **Q**:

$$Q = R_T \delta \alpha \rho \left(\frac{t_c}{t}\right)^2$$

$$\alpha = \left[\frac{T\left(\frac{t_c}{t}\right)^2}{NR_T} - 1\right]\frac{1}{\delta}$$

13.4.11.1.2.4 Bolts - combined tension & shear

The nominal tension force in a bolt adjusted for the shear force is calculated as follows:

LRFD: $\mathbf{F'_{nt} = 1.3\ F_{nt}\ \text{-}(F_{nt}/0.75)\ F_v\ \ =\ \ F_{nt}}$
ASD: $\mathbf{F'_{nt} = 1.3\ F_{nt}\ \text{-}(F_{nt}/0.50)\ F_v\ \ =\ \ F_{nt}}$
where $\mathbf{F_v = V/\ (N\ A_b)}$

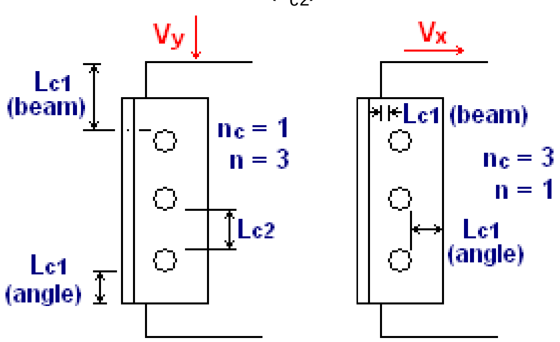
The total tension strength of the bolts in a connection is:
LRFD: $\mathbf{0.75\ N\ F'_{nt}\ A_b}$
ASD: $\mathbf{0.50\ N\ F'_{nt}\ A_b}$

N = number of bolts
V = shear force
$\mathbf{A_b}$ = bolt area

13.4.11.1.2.5 Bearing strength at bolt holes

AISC-05 - Section J3.10

The bearing strength of the bolt holes, $\mathbf{R_b}$, is calculated according to the distance parallel to the shear force from the edge of the holes to edge of the connecting angles or plates (L_{c1}) and the clear distance between the bolt holes (L_{c2}).



LRFD: $\mathbf{R_{b1} = 0.75\ t\ F_u\ 1.2\ n_c\ min(L_{c1},\ 2d)}$
$\mathbf{R_{b2} = 0.75\ t\ F_u\ 1.2\ n_c\ min(L_{c2},\ 2d)\ (n - 1)}$
ASD: $\mathbf{R_{b1} = 0.50\ t\ F_u\ 1.2\ n_c\ min(L_{c1},\ 2d)}$
$\mathbf{R_{b2} = 0.50\ t\ F_u\ 1.2\ n_c\ min(L_{c2},\ 2d)\ (n - 1)}$

t = thickness of part subject to bearing, or sum of thicknesses
$\mathbf{F_u}$ = specified minimum tensile strength of the part subject to bearing
$\mathbf{n_c}$ = number of bolt lines parallel to the shear force
n - number of bolt lines perpendicular to the shear force
d = bolt diameter

Note:
- $L_{c1,max} = 2d$

13.4.11.1.2.6 Shear yielding & rupture

AISC-05 - Section J4.2

The program checks the various elements in shear for Shear yielding and Shear rupture:

Shear yielding:	LRFD:	$1.0 (0.60 F_y A_g)$	(J4-3)
	ASD:	$0.67(0.60 F_y A_g)$	
Shear rupture:	LRFD:	$0.75 (0.60 Fu A_{nv})$	(J4-4)
	ASD:	$0.5 (0.60 Fu A_{nv})$	

$A_g = d\ t$
$A_{nv} = [d - N (d_h + 2mm)]\ t$ (or 1/16 in)
d_h = hole diameter
N = number of bolts
d = height of section

13.4.11.1.2.7 Block shear strength

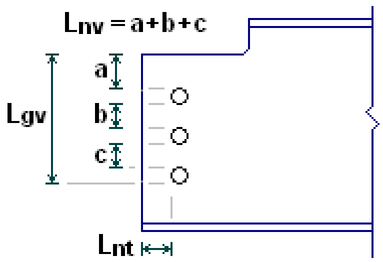
AISC-05 Section J4.3

Block shear strength is calculated as follows:

LRFD: $R_{BS} = 0.75 \min (R_{BN}, R_{BG}) + 0.75 R_{BT}$
ASD: $R_{BS} = 0.50 \min (R_{BN}, R_{BG}) + 0.50 R_{BT}$

$R_{BN} = 0.6 F_u A_{nv}$ $A_{nv} = (L_{nv})t$
$R_{BG} = 0.6 F_y A_{gv}$ $A_{gv} = (L_{gv})t$
$R_{BT} = U_{BS} F_u A_{nt}$ $A_{nt} = (L_{nt})t$

$U_{BS} = 0.5$ or **1.0** , depending on the stress distribution



Note:
- when calculating the net area for shear or tension, the program subtracts d_h + 2 mm ($^1/_{16}$ in).

13.4.11.1.2.8 Flexural yielding

Steel Construction Manual, 13th Edition, p. 10-103.

LRFD: $0.90 M_n$
ASD: $0.60 M_n$

$M_n = F_{cr}\ Z$

$F_{cr} = v\ (F_y^2 - 3F_v^2)$

Z = plastic modulus of the plate

F_v = shear stress in the plate

F_y = yield stress of the plate

13.4.11.1.2.9 Maximum plate thickness

Steel Construction Manual, 13th Edition, p. 10-103.

The program calculates the maximum plate thickness on the assumption that the moment strength of the plate does not exceed the shear strength of the bolts:

$t_{max} = (6\ M_{max}) / (F_y\ d^2)$

M_{max} = $1.25\ F_{nv}\ A_b\ c'$

F_{nv} = bolt shear strength according to Table J3.2

A_b = bolt area

c' = coefficient for strength of a bolt group subject to pure moment acting at the center-of-gravity of the bolt group (refer to p. 7-19)

13.4.11.1.2.10 Flexural rupture strength

AISC-05 Section J4.1.

The flexural (tensile) rupture strength is calculated as follows:

LRFD: $0.75\ M_n$

ASD: $0.50\ M_n$

Mn = $F_u\ Z_{net}$

F_u = minimum tensile strength

Z_{net} = net plastic section modulus (reduction for the area of the holes)

13.4.11.1.2.11 Plate buckling strength

Steel Construction Manual, 13th Edition page 9-8.

The local flexural buckling strength is calculated as follows:

LRFD: $0.90\ M_n$

ASD: $0.60\ M_n$

Mn = $F_{cr}\ S_n$

F_{cr} = $F_y\ Q$

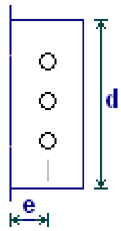
S_n = net elastic section modulus (reduction for the area of the holes)

Q = 1.0 for $\lambda \le 0.7$

1.34 - 0.486 λ for $0.7 < \lambda \le 1.41$

$1.30/\lambda^2$ for $\lambda > 1.41$

$$\lambda = \frac{d\sqrt{F_y}}{10t\sqrt{475 + 280\,(d/e)^2}}$$



13.4.11.1.2.12 Weld group strength - A

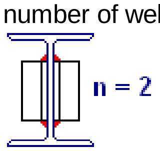
Steel Construction Manual, 13th Edition p. 8-29.

This method is generally used to calculate the weld strength on the supported beam side.

The strength, $\phi R_{n,}$ of an eccentrically loaded weld group is calculated as:

$R_n =$ **n C C_1 D I f**

LRFD: $\phi =$ **0.75**
ASD: $\phi =$ **0.50**
C = coefficient calculated by the "Instantaneous center of rotation" method
C_1 = electrode coefficient (Table 8-3)
D = number of sixteenths of an inch in the weld size
I = characteristic length of weld group
f = **t/t_{min}**
t = beam web thickness
t_{min} = minimum beam web thickness = **(F_{EXX} D□2)/(16F_u)**
(refer to Steel Construction Manual, 13th Edition p.9-5).
n = number of weld groups



13.4.11.1.2.13 Weld group strength - B

Steel Construction Manual, 13th Edition p.10-10 to 10-12.

This method is generally used to calculate the weld strength on the support side.

The weld strength, $\phi R_{n,}$ is calculated according to the following equation

$$R_n = \frac{0.848 F_{EXX} W L f}{\sqrt{1 + \frac{12.96 e^2}{L^2}}}$$

LRFD: $\phi =$ **0.75**
ASD: $\phi =$ **0.50**
F_{EXX} = electrode strength
W = thickness of the weld
L = weld length
e = width of the angle leg on the support side
f = **t/t_{min}**
t = supporting web/flange thickness
t_{min} = minimum supporting web/flange thickness = **(F_{EXX} D□2)/(32F_u)**
(refer to Steel Construction Manual, 13th Edition p.9-5).

13.4.11.1.2.14 Column web yielding

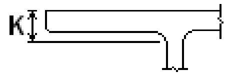
AISC - Steel Design Guide 4: Extended End Plate Moment Connections.

The column web capacity is calculated as $\phi \mathbf{R_n}$, where:

$$\mathbf{R_n = [C_t\ (6K+2t_p) + N]\ F_y\ t_w}$$

LRFD: $\phi = 1.00$
ASD: $\phi = 0.67$

t_w = column web thickness
F_y = specified minimum yield stress of the column
K = distance from the external face of the flange to the far end of the radius on the flange-web connection, calculated from the program section tables as **K = (d-T)/2**, where **T** = "Depth between fillets" (this value may vary slightly from published values of **K**).



N = thickness of the *beam* flange + **2w**
w = leg size of flange weld
t_p = end-plate thickness
C_t = 0.5 if column does not extend beyond the connection, otherwise **C_t** = 1.0

13.4.11.1.2.15 Column web buckling

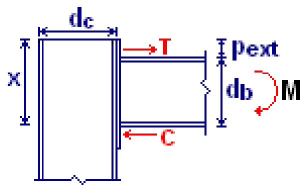
AISC - Steel Design Guide 4: Extended End Plate Moment Connections.

The column web buckling capacity is calculated as $\phi \mathbf{R_n}$, where:

$$\mathbf{R_n = C_c\ [\ 24\ t_w^3\ (EFy)^{0.5}\ ]\ /\ h}$$

LRFD: $\phi = 0.90$
ASD: $\phi = 0.60$

C_c = 1.0 for a continuous column or if if $\mathbf{x \geq p_{ext} + d_c/2}$, otherwise **$C_c$** = 0.5.



where **x** is measured from the compression flange (top flange for reversed moment)
h = "Depth between fillets", taken from the program section tables.
E = modulus of elasticity of the steel (29,000 ksi)
F_y = specified minimum yield stress of the column steel

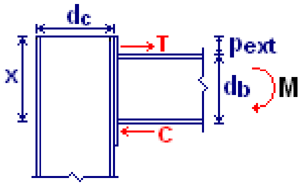
13.4.11.1.2.16 Column web crippling

AISC-05 Section J.10.3.

The column web crippling capacity is calculated as $\phi \mathbf{R_n}$, where:

LRFD: $\phi = 0.75$
ASD: $\phi = 0.50$

$$R_n = k\,t_w^2\left[1+3\left(\frac{N}{d}\right)\left(\frac{t_w}{t_f}\right)^{1.5}\sqrt{\frac{E F_y t_f}{t_w}}\right]$$



x = distance from the beam compression flange to the end of the column

k = 0.4 when $\mathbf{x \le d_c/2 + p_{ext}}$

0.8 when $\mathbf{x > d_c/2 + p_{ext}}$ or when the column is continuous

$\mathbf{t_w}$ **=** column web thickness

$\mathbf{t_f}$ = column flange thickness

N = thickness of the *beam* flange + $\mathbf{(2w+2t_p)}$

$\mathbf{d_c}$ **=** column depth

E = modulus of elasticity of the steel (29,000 ksi)

$\mathbf{F_y}$ = specified minimum yield stress of the column

w = leg size of fillet weld

$\mathbf{t_p}$ = end plate thickness

13.4.11.1.2.17 End plate thickness

AISC - Steel Design Guide 16: Flush and Extended Multiple-Row Moment End Plate Connections.
AISC - Steel Design Guide 4: Extended End Plate Moment Connections.

- Thin plate

$$t_{req} = \sqrt{\frac{M}{\phi_b F_y y_p}}$$

- Thick plate

$$t_{req} = \sqrt{\frac{1.1\phi M_{np}}{\phi_b F_y y_p}}$$

LRFD: $\phi = 0.75$ $\phi_b = 0.90$

ASD: $\phi = 0.50$ $\phi_b = 0.60$

M = beam end moment, including transfer of axial force

$\mathbf{M_{np}}$ = no prying bolt strength moment = $\mathbf{2\,p_t\,(\Sigma h_i)}$

$\mathbf{p_t}$ = bolt tension strength = $\mathbf{F_t\,(\pi\,d_b^2/4)}$

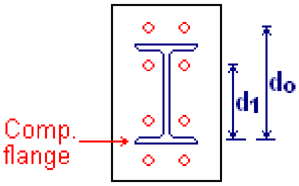
$\mathbf{y_p}$ = a factor dependent of the failure mode of the plate; refer to Tables 3-1, 3-2, 3-3 - AISC - Steel Design Guide 4.

13.4.11.1.2.18 Bolt rupture w ith prying

AISC - Steel Design Guide 16: Flush and Extended Multiple-Row Moment End Plate Connections - Chapter 4.

Thin end plate:

The program calculates ϕM_q, the connection strength for bolt fracture with prying action as follows:
For an 4-bolt connection:



$$M_q = \max \begin{bmatrix} 2(P_t - Q_{max,o})d_o + 2(P_t - Q_{max,i})d_1 \\ 2(P_t - Q_{max,o})d_o + 2(T_b)d_1 \\ 2(P_t - Q_{max,i})d_1 + 2(T_b)d_o \\ 2(T_b)(d_o + d_1) \end{bmatrix}$$

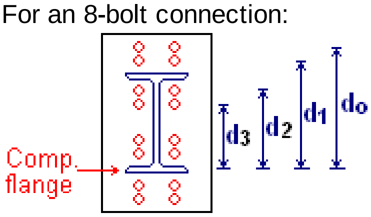
LRFD: $\phi = 0.75$
ASD: $\phi = 0.50$

P_t = bolt proof load = **A_b F_t**
F_t = nominal bolt tensile strength
A_b = bolt area
T_b:
- Fully-tightened bolts
  T_b = specified pretension force in Table J3.1
- Snug-tightened bolts
  T_b = specified pretension force in Table J3.1, reduced by the following factors according to bolt diameter:

d_b = $^5/_8$":	0.75
d_b = $^3/_4$":	0.50
d_b = $^7/_8$":	0.375
d_b = 1":	0.25

$Q_{max,i}$: maximum bolt prying force in bolts between beam flanges; refer to Table 4-1.
$Q_{max,o}$: maximum bolt prying force in bolts outside beam flanges; refer to Table 4-1.
d_1 = distance from beam compression flange to bolt between beam flanges
d_0 = distance from beam compression flange to bolt outside beam flanges

For an 8-bolt connection:



$$M_q = \max \begin{bmatrix} 2(P_t - Q_{max,o})d_1 + 2(P_t - Q_{max,i})d_2 + 2T_b(d_o + d_3) \\ 2(P_t - Q_{max,o})d_1 + 2T_b(d_o + d_2 + d_3) \\ 2(P_t - Q_{max,i})d_2 + 2T_b(d_o + d_1 + d_3) \\ 2T_b(d_o + d_1 + d_2 + d_3) \end{bmatrix}$$

Thick end plate:

The program calculates ϕ**M_n**, the strength of the connection, as follows:

$$\mathbf{M_n = 2\,P_t\,(\Sigma d_i)}$$

LRFD: $\phi = 0.75$

ASD: ϕ = 0.50
d_i = distance from beam compression flange to bolt

13.4.11.1.2.19 End plate - miscellaneous

AISC - Steel Design Guide 4: Extended End Plate Moment Connections.
Chapter 3.

- Shear yielding of extended portion of unstiffened end plate

  $\phi R_n = \phi\ 0.6\ F_y\ b_p\ t_p$

  LRFD: ϕ = 1.0
  ASD: ϕ = 0.67
  F_y = specified minimum yield stress of the plate steel
  b_p = plate width, but not greater than beam flange width + 1 in.
  t_p = plate thickness

- Shear rupture of extended portion of end plate

  $\phi R_n = \phi\ 0.6\ F_u\ A_n$

  LRFD: ϕ = 0.75
  ASD: ϕ = 0.50
  F_u = minimum tensile strength of the plate steel
  A_n = net area of the end plate, reduced for holes.

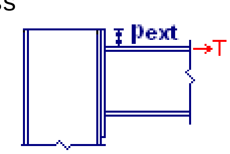
13.4.11.1.2.20 Local flange bending

AISC - Steel Design Guide 13: Stiffening of Wide-Flange Columns at Moment Connections.
Section 2.2.2.

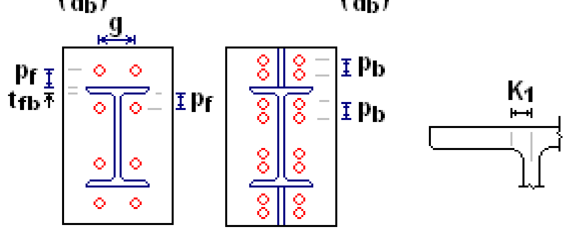
The program calculates the local flange bending strength, ϕR_n, as follows:

$$R_n = \frac{b_s}{\alpha_m p_e} t_f^2 F_y C_t$$

LRFD: ϕ = 0.90
ASD: ϕ = 0.60
F_y = specified minimum yield stress of the column steel ≤ 36 ksi
t_f = column flange thickness
C_t : **$p_{ext} > 10t_f$: $C_t = 1.0$**
$p_{ext} < 10t_f$: $C_t = 0.5$



	4-bolt unstiffened	4-bolt stiffened	8-bolt stiffened
b_s =	$2.5\ (2p_f + t_{fb})$	$2.5\ (2p_f + t_{fb})$	$2pf + t_{fb} + 3.5p_b$
α_m =	$1.36\left(\frac{p_e}{d_b}\right)^{1/4}$	$1.13\left(\frac{p_e}{d_b}\right)^{1/4}$	$1.13\left(\frac{p_e}{d_b}\right)^{1/4}$



$\mathbf{p_e}$ = $\mathbf{g/2 - d_b/4 - k_1}$

$\mathbf{k_1}$: the program calculates the value as $\mathbf{k_{des} + {}^1/_4" - t_f + t_w/2}$, where $\mathbf{k_{des}}$ is calculated from the program section tables as $^1/_2$(d -"Depth between fillets")

13.4.11.1.2.21 Weld strength

- Steel Construction Manual, 13th Edition, p. 8-8.
- AISC - Steel Design Guide 4: Extended End Plate Moment Connections. Chapter 4 - Design examples.

The available strength of a welded joint, $\phi \mathbf{R_n}$, is calculated as follows:

- Beam flange

$$\mathbf{R_n = 0.6363\ F_{EXX}\ W\ L\ f}$$

LRFD: ϕ = 0.75
ASD: ϕ = 0.50
$\mathbf{F_{EXX}}$ = weld strength, ksi
W = weld size
L = weld length
f = $\mathbf{t/t_{min}}$
t = beam flange thickness
$\mathbf{t_{min}}$ = minimum beam flange thickness = $\mathbf{(0.6F_{EXX}\ 2^{0.5}W)/F_u}$
(refer to Steel Construction Manual, 13th Edition p.9-5).

- Beam web - shear

$$\mathbf{R_n = 0.4242\ F_{EXX}\ W\ L\ f}$$

L = one-half of the beam web height
f = $\mathbf{t/t_{min}}$
t = beam web thickness
$\mathbf{t_{min}}$ = minimum beam flange thickness = $\mathbf{(0.6F_{EXX}\ 2^{0.5}W)/F_u}$
(refer to Steel Construction Manual, 13th Edition p.9-5).
and all other symbols are as explained above.

- Beam web - tension

$$\mathbf{R_n = 0.4242\ F_{EXX}\ 2W}$$

$\mathbf{R_n}$ is compared to the beam bending capacity.

13.4.11.1.2.22 Breakout strength - tension

The breakout strength in tension of a bolt group anchored in concrete is calculated according to ACI318, Section 17.4.2:

$$\mathbf{T_R/(\phi N_{cbg}) = 1.00} \qquad \text{(Table 17.3.1.1)}$$

where:

$\mathbf{T_r}$ = design tension force
ϕ = 0.7 (LRFD)
= 0.465 (ASD)

N_{cbg} = $\mathbf{(A_N/A_{No})\ \psi_2\psi_3N_b}$ Eq. (17.4.2.1b)

ψ_2 = 1.00 if $\mathbf{c_{min} = 1.5h_{ef}}$ Eq. (17.4.2.5a)

= $\mathbf{0.7 + 0.3\ (c_{min}/1{,}5h_{ef})}$ if $\mathbf{c_{min} < 1.5h_{ef}}$ Eq. (17.4.2.5b)

ψ_3 = 1.25 (uncracked) Eq. (17.4.2.6)

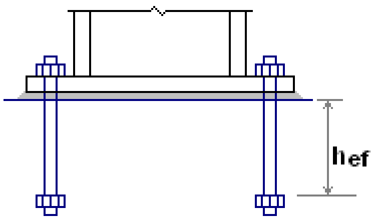
= 1.00 (cracked)

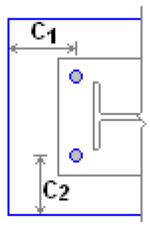
N_b = $\mathbf{24\sqrt{f'_c}\ h_{ef}^{1.5}}$ if $\mathbf{h_{ef}}$ < 11 in Eq. (17.4.2.2a)

= $\mathbf{16\sqrt{f'_c}\ h_{ef}^{5/3}}$ if $\mathbf{h_{ef}}$ = 11 in Eq. (17.4.2.2b)

A_{No} = $\mathbf{9h_{ef}^2}$ Eq. (17.4.2.1c)

A_N = concrete breakout cone area





when $\mathbf{C_1<1.5h_{ef}}$ and $\mathbf{C_2<1.5h_{ef}}$, then $\mathbf{h_{ef} = max(C_1,C_2)/1.5}$

13.4.11.1.2.23 Pullout strength

The pullout strength of a bolt anchored in concrete is calculated according to ACI318, Section 17.4.3:

$\mathbf{T_R/(\phi N_{pn}) = 1.00}$ (Table 17.3.1.1)

where:

T_r = design tension force

ϕ = 0.7 (LRFD)

= 0.465 (ASD)

N_{pn} = $\psi_4 N_p$ Eq. (17.4.3.1)

ψ_4 = 1.00 (cracking) Eq. (17.4.3.6)

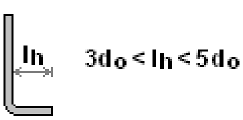
= 1.40 (no cracking)

N_p = $\mathbf{8A_{brg}f'_c}$ for headed bolts Eq. (17.4.3.4)

= $\mathbf{0.9f'_c\ l_h d_o}$ for hooked anchor Eq. (17.4.3.5)

A_{brg} = Bearing area of the head of the anchor bolt

d_o = anchor diameter



13.4.11.1.2.24 Bearing strength

The bearing strength is calculated according to ACI318 - Section 10.17:

$\mathbf{f_p/(0.65f_{p,max}) = 1.00}$

where:

f_p = bearing stress

$f_{p,max}$ = allowable bearing stress. Refer to base plate - bearing 1339

13.4.11.1.2.25 Plate yielding

The base plate flexural yielding limit is calculated according to AISC Steel Design Guide 1, second edition, 3.3.

$$\frac{M_{pl}}{0.9 \cdot R_n} \leq 1.00$$

The plate yielding at bearing interface is caused by the concrete bearing stress that acts on the plate with a cantilever arm l to the column flanges, causing bending moment.

$$M_{pl} = \begin{cases} f_p \cdot \dfrac{l^2}{2} & for\, Y \geq m \\ f_p \cdot l \cdot Y & for\, Y < m \end{cases}$$

$$f_p = \frac{v_x}{B \cdot Y}$$

v_x = the axial compression force.

Y = the length of the compression zone.

B = the width of the base plate.

The cantilever arm l is calculated as follows:

- For I sections:

$$l = \max \begin{Bmatrix} m = \dfrac{N - 0.95 \cdot d}{2} \\ n = \dfrac{B - 0.8 \cdot b_f}{2} \end{Bmatrix}$$

  If only axial load exists , then also:

$$l \geq \lambda \cdot n' = \lambda \cdot \frac{\sqrt{d \cdot b_f}}{4}$$

$$\lambda = \frac{2 \cdot \sqrt{X}}{1 + \sqrt{1 - X}} \leq 1$$

$$X = \frac{4 \cdot d \cdot b_f \cdot P_u}{0.9 \cdot (d + b_f)^2 \cdot f_{p.max} \cdot A_1}$$

- For RHS sections:

$$l = \max \begin{Bmatrix} m = \dfrac{N - 0.95 \cdot d}{2} \\ n = \dfrac{B - 0.95 \cdot b_f}{2} \end{Bmatrix}$$

- For Pipe sections:

$$l = \max \begin{Bmatrix} m = \dfrac{N - 0.8 \cdot D}{2} \\ n = \dfrac{B - 0.8 \cdot D}{2} \end{Bmatrix}$$

The Nominal bending resistance per unit length of the plate:

$$R_n = f_y \cdot \frac{t^2}{2}$$

The plate yielding at tension interface is caused by the tension force in the anchor rods that acts on the plate with a cantilever arm x to the column flanges, causing bending moment.

$$M_{pl} = \frac{T \cdot x}{B}$$

x = the distance from the rod centerline to the center of the column flange
T = the tension force in the anchor rods.

13.4.11.1.2.26 Breakout strength - shear

The breakout strength in shear of a bolt group anchored in concrete is calculated according to ACI318, Section 17.5.2:

$\mathbf{V_y/(2\phi V_{cbg}) = 1.00}$ (Table 17.3.1.1)

where:

$\mathbf{V_r}$ = design shear force
ϕ = 0.7 (LRFD)
= 0.465 (ASD)
$\mathbf{V_{cbg}}$ = $\mathbf{(A_V/A_{Vo})\ \psi_6\psi_7 V_b}$ Eq. (17.5.2.1a)
ψ_6 = 1.00 if $\mathbf{c_2 = 1.5c_1}$ Eq. 17.5.2.6a)
= $\mathbf{0.7 + 0.3\ (c_2/1{,}5c_{1})}$ if $\mathbf{c_2 < 1.5c_1}$ Eq. (17.5.2.6b)
ψ_7 = **1.40** (no cracking) Eq. (17.5.2.7)
= **1.00** (cracking)
$\mathbf{V_b}$ = $\mathbf{7(l/d_a)^{0.2}\ v d_a v f'_c c_1^{1.5}}$ Eq. (17.5.2.2a)
but not greater than $\mathbf{9 v f'_c c_1^{1.5}}$ Eq. (17.5.2.2b)
l **=** $\mathbf{h_{eff} = 8d_a}$
$\mathbf{d_a}$ **=** 1.25 in.

13.4.11.1.2.27 Shear-Tension Interaction

The shear-tension interaction is calculated according to ACI318, Section 17.6:

$\mathbf{(N_u/\phi_1 N_n) + (V_u/\phi_2 V_n) = 1.2}$ (17.6.3)

where:
$\mathbf{N_u}$ = design tension force
$\mathbf{V_u}$ = design shear force
$\mathbf{\phi_1 N_n}$ = $\mathbf{min(nN_s, nN_{pn}, N_{cbg})}$
$\mathbf{\phi_2 V_n}$ = $\mathbf{min(nV_s, 2V_{cbg})}$
$\mathbf{\phi_1, \phi_2}$: taken from the governing axial/shear calculations

13.4.11.1.2.28 Welds - base plate

- Steel Construction Manual, 13th Edition, p. 8-8.

The available strength of a welded joint, $\phi \mathbf{R_n}$, is calculated as follows:

- Column flange - tension

$\mathbf{R_n = 0.6363\ F_{EXX}\ w\ L\ f}$

LRFD: ϕ = 0.75
ASD: ϕ = 0.50
$\mathbf{F_{EXX}}$ = weld strength, ksi
W = weld size
L = weld length

f = **t/t_{min}**
t = beam flange thickness
t_{min} = minimum beam flange thickness = **$(0.6F_{EXX}$ Wv2)/F_u**
(refer to Steel Construction Manual, 13th Edition p.9-5).

- Column web - shear

R_n = 0.4242 F_{EXX} W L f

L = one-half of the column web height
f = **t/t_{min}**
t = column web thickness
t_{min} = minimum beam flange thickness = **$(0.6F_{EXX}$ Wv2)/F_u**
(refer to Steel Construction Manual, 13th Edition p.9-5).
and all other symbols are as explained above.

Pipe sections:

The weld between the base plate and a pipe column is calculated according to the 'Treating weld as a line' approach. The resultant elastic stress in the weld is limited to code allowable stress.

$$\frac{\tau_w}{0.75 \cdot F_{nw} \cdot f} \leq 1$$

where:

$$F_{nw} = 0.6 * F_{EXX}$$

$$\tau_w = \sqrt{\tau_x^2 + \tau_y^2 + \tau_z^2}$$

$$\tau_x = \frac{F_x \cdot \sqrt{2}}{w \cdot D \cdot \Pi}$$

$$\tau_y = \frac{F_y \cdot \sqrt{2}}{w \cdot D \cdot \Pi}$$

$$\tau_z = \frac{F_z \cdot \sqrt{2}}{w \cdot D \cdot \Pi} + \frac{\sqrt{M_x^2 + M_y^2} \cdot 4\sqrt{2}}{w \cdot D \cdot \Pi}$$

W is the weld leg size.
D is the column diameter.

$$t_{min} = \frac{F_{EXX} \cdot w}{\sqrt{2} \cdot f_u}$$

$$f = \frac{t}{t_{min}} \leq 1$$

t_{min} - minimum beam flange thickness (refer to Steel Construction Manual, 13th Edition p.9-5)

13.4.11.2 BS 5950-1 : 2000

The program checks and designs connections according to:

- BS5950-1:2000 - Part 1
- SCI/BCSA - Publication P212 - Joints in Steel Construction - Simple Connections
- SCI - Joints in Steel Construction - Moment Connections

Refer to:
Double angle cleats 1356
Double angle welded 1367

13.4.11.2.1 Double angle cleats

Refer to:

13.4.11.2.1.1 Supported beam

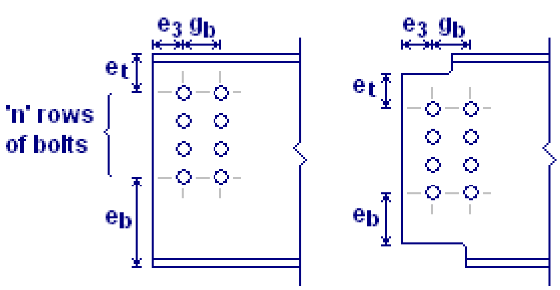
Refer to:

BS5950-1: Sections 4.2.3, 6.2.3

$\mathbf{F_v / P_v < 1.00}$

where:

$\mathbf{F_v}$	=	design shear force
$\mathbf{P_v}$	=	min ($\mathbf{0.6\, p_y\, A_v\, ,\, 0.7 p_y\, K_e\, A_{v,net}}$)
$\mathbf{A_v}$	=	$\mathbf{[e_t + (n-1)p + e_b]tw}$ (un-notched and single notched beams)
		$\mathbf{0.9[e_t + (n-1)p + e_b]t_w}$ (double notched beams)
$\mathbf{A_{v,net}}$	=	$\mathbf{A_v - nD_h t_w}$
$\mathbf{D_h}$	=	diameter of hole
p	=	bolt pitch
$\mathbf{t_w}$	=	supported beam web
$\mathbf{K_e}$	=	1.2 - S275 steel
	=	1.1 - S355 steel
	=	$\mathbf{(U_s/1.2)/p_y}$ for other grades



BS5950-1: Section 6.2.4

$\mathbf{F_v / P_r < 1.00}$

where:

$\mathbf{F_v}$ = design shear force

$\mathbf{P_r}$ = $\mathbf{0.6\, p_y\, t_w\, [\, Lv + K_e\, (L_t - k\, D_h)]}$

$\mathbf{L_v}$ = $\mathbf{e_t + (n - 1)\, p}$

k = 0.5 - single line of bolts

2.5 - double line of bolts

$\mathbf{L_t}$ = $\mathbf{e_3}$ - single line of bolts

$\mathbf{e_3 + g_b}$ - double line of bolts

$\mathbf{D_h}$ = diameter of hole

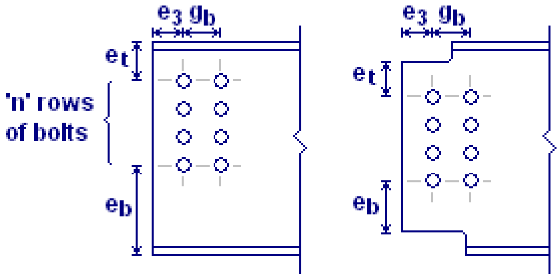
p = bolt pitch

$\mathbf{t_w}$ = supported beam web

$\mathbf{K_e}$ = 1.2 - S275 steel

= 1.1 - S355 steel

= $\mathbf{(U_s/1.2)/p_y}$ for other grades

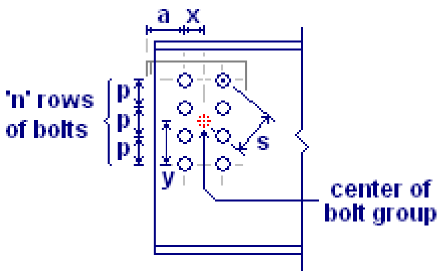


BS5950-1: Sections 6.3.2.1

$\mathbf{F_s / 2P_s < 1.00}$

where:

	Single line	Double line
$\mathbf{F_s}$	= $\mathbf{(F_{sv}^2 + F_{sm}^2)^{1/2}}$	= $\mathbf{[(F_{sv} + F_{smv})^2 + F_{smh}^2\,]^{1/2}}$
$\mathbf{F_{sv}}$	= $\mathbf{F_v/n}$	= $\mathbf{F_v/2n}$
$\mathbf{F_{sm}}$	= $\mathbf{F_v a/Z_{bg}}$	
$\mathbf{F_{smv}}$		= $\mathbf{Mx/I_{bg}}$
$\mathbf{F_{smh}}$		= $\mathbf{My/I_{bg}}$
$\mathbf{Z_{bg}}$	= $\mathbf{n(n+1)p/6}$	
M		= $\mathbf{F_v\,(a + x)}$
$\mathbf{I_{bg}}$		= $\mathbf{\Sigma s^2}$

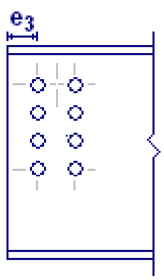


p = bolt pitch
$2P_s$ = $2p_sA_s$
p_s = shear strength of bolt (from Table 30)

BS5950-1: Sections 6.3.3.3

$$F_s / P_{bs} < 1.00$$

F_s = Resultant force
P_{bs} = min ($d\ t_w\ p_{bs}$, $0.5\ e t_w\ p_{bs}$)
p_{bs} = bearing strength of beam web (Table 32)
e = e_3
d = bolt diameter

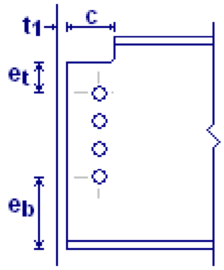


BS5950-1: Sections 4.2.5.4

$$M_n / M_{cn} < 1.00$$

where:

M_n	=	$F_v (t_1 + c)$	
M_{cn}	=	$p_y Z_n$	(low shear)
	=	$1.5\ p_y Z_n [1 - (F_v/P_{vn})^2]^{1/2}$	(high shear)
P_{vn}	=	$0.6\ p_y A_{vn}$	
A_{vn}	=	$[e_t + (n - 1)p + e_b]\ t_w$	



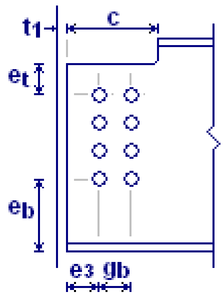
if the notch length $c > e_3 + g_b$:

$$M_v / M_{cc} < 1.00$$

where:

M_{cc}:	<u>Low shear</u>	<u>High shear</u> ($F_v > 0.75P_{v,min}$)
Single notch:	p_yZ	$1.5p_yZ [1- (F_v/P_{v,min})^2]^{1/2}$
Double notch:	$p_yt_w[e_t + (n-1)p + e_b)^2$	$p_yt_w[e_t + (n-1)p + e_b)^2 [1- (F_v/P_{v,min})^2]^{1/2}$

M_v = $F_v(t_1 + e_3 + g_b)$
Z = elastic section modulus at the bolt line
$P_{v,min}$ = min(p_v - plain shear, p_r - block shear)



One flange notched:

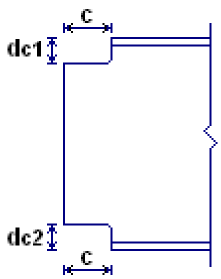
$d_{c1} \le D/2$
$c \le D$ for $D/t_w \le k_2$
$c \le k_1\, D/(D/t_w)^3$ for $D/t_w > k_2$

Both flanges notched

max$(d_{c1},d_{c2}) \le D/5$
$c \le D$ for $D/t_w \le k_2$
$c \le k_1\, D/(D/t_w)^3$ for $D/t_w > k_2$

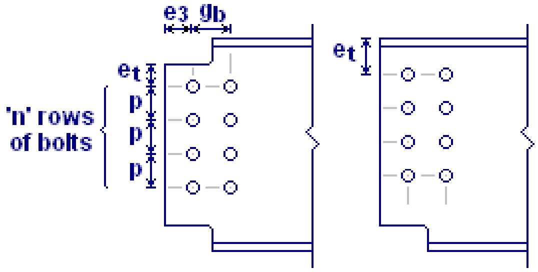
where:

k_1 = $[16 - (p_y - 275)/16] \times 10^4$
k_2 = $54.3 - 6.3(p_y - 275)/80$
t_w = thickness of supported beam web



$F_t / P_t < 1.00$

where:
P_t = $L_e\, t_w\, p_y$
L_e = $2e_e + (n-1)p_e - nD_h$
e_e = min (e_3, e_t) - single line of bolts
= min ($e_3 + g_b - D_h$, e_t) - double line of bolts
p_e = min (p, $2e_3$) - single line of bolts
= min [p, $2(e_3 + g_b - D_h)$] - double line of bolts
t_w = beam web thickness
p = bolt pitch
D_h = diameter of hole



$F_t / P_{bs} < 1.00$

where:

P_{bs} = min (**1.5d, 0.5e$_3$**) **n t_w p_{bs}** - single line of bolts

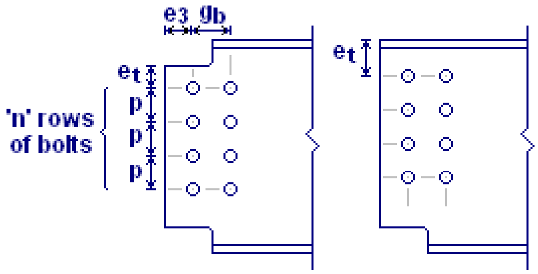
= min (**3d, 1.5d +0.5e$_3$**) **n t_w p_{bs}** - double line of bolts

p_{bs} = bearing strength of beam web - Table 32

t_w = beam web thickness

p = bolt pitch

D_h = diameter of hole



13.4.11.2.1.2 Supporting beam/column

Refer to:

Local shear 1360
Bearing 1361
Bolt shear 1361
Bolt tension 1362
Tying capacity 1362

BS5950-1: Sections 4.2.3, 6.2.3

$0.5\Sigma F_v / P_v < 1.00$

where:

ΣF_v = $F_{v1} + F_{v2}$

P_v = min (**0.6 P_y A_v, 0.7 p_y K_e $A_{v,net}$**)

A_v = $[e_t + (n_2 - 1)p + e_b]\ t_w$

$A_{v,net}$ = $A_v - n_2 D_h t_w$

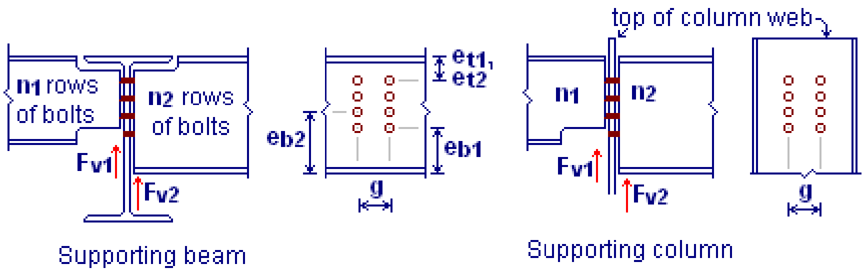
e_t = min (**e_{t1}, 5d**)

e_b = min (**e_{b2}, g/2, p, 5d**) - supporting beam

min (**g/2, p, 5d**) - supporting column

D_h = diameter of hole

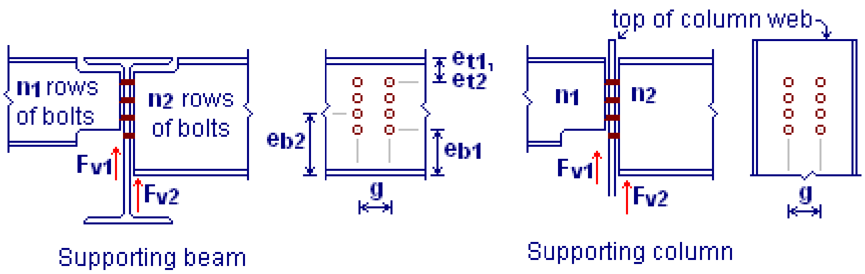
p = bolt pitch
d = diameter of hole
$\mathbf{t_w}$ = supported beam web
$\mathbf{K_e}$ = 1.2 - S275 steel
= 1.1 - S355 steel
= $\mathbf{(U_s/1.2)/p_y}$ for other grades



BS5950-1: Section 6.3.3.3

$$\mathbf{0.5\Sigma F_v / nP_{bs} < 1.00}$$

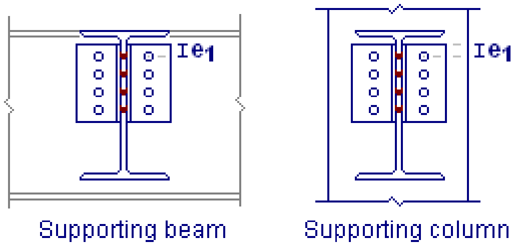
where:
$\mathbf{\Sigma F_v}$ = $\mathbf{F_{v1} + F_{v2}}$
$\mathbf{P_{bs}}$ = $\mathbf{d\ t_w\ p_{bs}}$
$\mathbf{p_{bs}}$ = bearing strength of supporting beam or column
$\mathbf{D_h}$ = diameter of hole
p = bolt pitch
d = diameter of hole
$\mathbf{t_w}$ = supported beam web
$\mathbf{K_e}$ = 1.2 - S275 steel
= 1.1 - S355 steel
= $\mathbf{(U_s/1.2)/p_y}$ for other grades



BS5950-1: Section 6.3.2.1

$$\mathbf{F_v / \Sigma P_s < 1.00}$$

where:
$\mathbf{P_s}$ = $\mathbf{p_s\ A_s}$ - for all rows of bolts, except:
min ($\mathbf{p_s A_s}$, $\mathbf{0.5\ e_1\ t_c\ p_{bs}}$) - for the top row of bolts
$\mathbf{p_s}$ = bolt shear strength
$\mathbf{A_s}$ = bolt shear area
$\mathbf{t_c}$ = cleat thickness

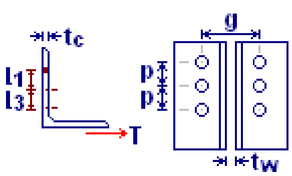


BS5950-1: Section 6.3.4.3

$F_t / P_t < 1.00$

where:

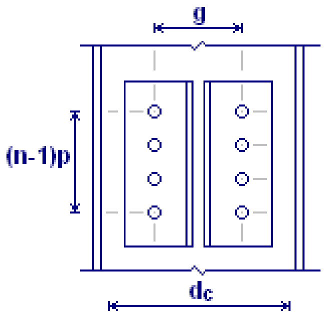
P_t = $2n\ A_t\ p_{tr}$
n = number of rows of bolts
A_t = tensile stress area of bolt
p_{tr} = reduced tension strength of bolt in presence of extreme prying
= (minimum tensile strength) / (1.25 x prying ratio)
prying ratio = $(2t_c + \text{lever arm}) / 2t_c$
lever arm = $l_1 \cos \varphi$
$L_1 = (l_1+l_3)(p-D_h) / (2p-D_h)$
$\varphi = \tan^{-1} [30/(l_1 + l_3)]$
$(l_1 + l_3) = g/2 - t_w/2 - t_c - r$
$(l_1/l_3) = (p-D_h)/p$



$F_t / P_t < 1.00$

where:

P_t = $8M_u [\eta_1 + 1.5(1 - \beta_1)^{0.5} (1 - \gamma_1)^{0.5}] / (1 - \beta_1)$
M_u = $p_u\ t_w^2/4$
η_1 = $[(n-1)p - 0.5n\ D_h] / d_c$
β_1 = g/d_c
γ_1 = D_h/d_c
n = number of rows of bolts
D_h = diameter of hole
p_u = $U_s/1.25$



13.4.11.2.1.3 Connecting element - supported side

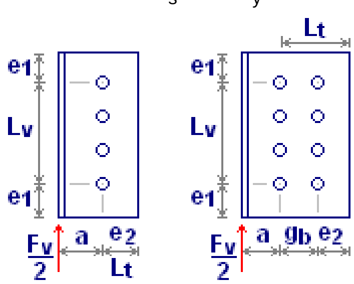
Refer to:
[Plain shear 1363]
[Block shear 1363]
[Bearing 1364]

BS5950-1: Sections 4.2.3, 6.2.3

$0.5F_v / P_v < 1.00$

where:

F_v	=	design shear force
P_v	=	min ($0.6\, p_y\, A_v$, $0.7 p_y\, K_e\, A_{v,net}$)
A_v	=	$0.9[2e_1 + (n-1)p]\, t_c$
$A_{v,net}$	=	$Av - n\, D_h\, t_c$
D_h	=	diameter of hole
p	=	bolt pitch
t_c	=	thickness of cleat
K_e	=	1.2 - S275 steel
	=	1.1 - S355 steel
	=	$(U_s/1.2)/p_y$ for other steels



BS5950-1: Section 6.2.4

$F_v / P_r < 1.00$

where:

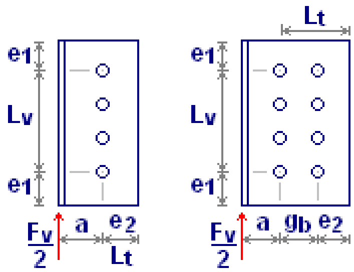
F_v = design shear force

P_r = $0.6\, p_y\, t_c\, [\, Lv + K_e\, (L_t - k\, D_h)]$

L_v = $e_1 + (n - 1)\, p$

k = 0.5 - single line of bolts
2.5 - double line of bolts

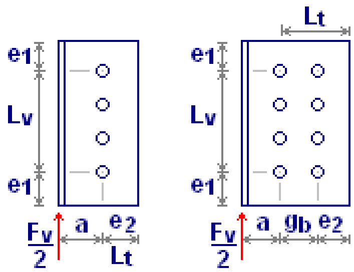
L_t = e_2 - single line of bolts
$e_2 + g_b$ - double line of bolts
D_h = diameter of hole
p = bolt pitch
t_c = thickness of cleat
K_e = 1.2 - S275 steel
= 1.1 - S355 steel
= $(U_s/1.2)/p_y$ for other steels



BS5950-1: Sections 6.3.3.3

$$0.5F_v / P_{bs} < 1.00$$

F_s = Resultant force
P_{bs} = min ($0.5 t_c p_{bs}$, $0.5\ e t_c\ p_{bs}$)
p_{bs} = bearing strength of cleat (Table 32)
e = min (e_1, e_2)
d = bolt diameter



13.4.11.2.1.4 Connecting element - supporting side

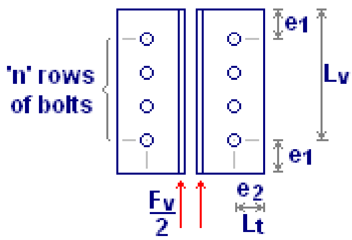
Refer to:
Plain shear 1364
Block shear 1365
Bearing 1365
Tension 1366

BS5950-1: Sections 4.2.3, 6.2.3

$$0.5F_v / P_v < 1.00$$

where:

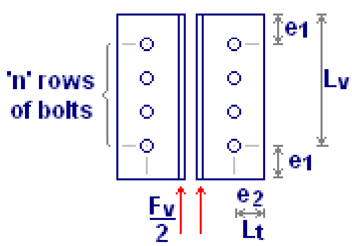
F_v = design shear force
P_v = min ($\mathbf{0.6\ p_y\ A_v\ ,\ 0.7 p_y\ K_e\ A_{v,net}}$)
A_v = $\mathbf{0.9[2e_1 + (n\text{-}1)p]t_c}$
$A_{v,net}$ = $\mathbf{Av - n\ D_h\ t_c}$
D_h = diameter of hole
p = bolt pitch
t_c = thickness of cleat
K_e = 1.2 - S275 steel
= 1.1 - S355 steel
= $(U_s/1.2)/p_y$ for other steels



BS5950-1: Section 6.2.4

$0.5F_v\ /P_r < 1.00$

where:
F_v = design shear force
P_r = $\mathbf{0.6\ p_y\ t_c\ [\ Lv + K_e\ (L_t - k\ D_h)]}$
L_v = $\mathbf{e_1 + (n - 1)\ p}$
k = 0.5
L_t = $\mathbf{e_2}$
D_h = diameter of hole
p = bolt pitch
t_c = thickness of cleat
K_e = 1.2 - S275 steel
= 1.1 - S355 steel
= $(U_s/1.2)/p_y$ for other steels

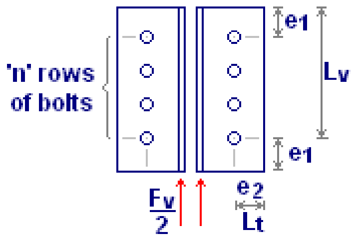


BS5950-1: Sections 6.3.3.3

$0.5F_s\ /P_{bs} < 1.00$

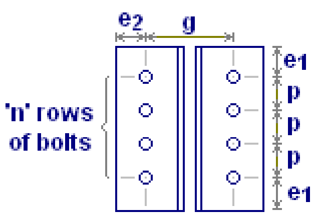
F_s = Resultant force

$\mathbf{P_{bs}}$ = $\mathbf{d\ t_c\ p_{bs}}$
= min ($\mathbf{dt_c\ p_{bs}}$, $\mathbf{0.5\ e_1\ t_c\ p_{bs}}$) for the top bolt
$\mathbf{p_{bs}}$ = bearing strength of cleat (Table 32)
d = bolt diameter



$\mathbf{F_t / P_t < 1.00}$

where:
$\mathbf{P_t}$ = $\mathbf{0.6L_e\ t_w\ p_y}$ - for S275 steel
= $\mathbf{0.6(275/p_y)\ (U_s/430)L_e\ t_w\ p_y}$ - for other steels
$\mathbf{L_e}$ = $\mathbf{2e_e + (n\text{-}1)p_e - nD_h}$
$\mathbf{e_e}$ = min ($\mathbf{e_1, e_2}$)
$\mathbf{p_e}$ = min ($\mathbf{p, 2e_2}$)
$\mathbf{t_w}$ = beam web thickness
p = bolt pitch
$\mathbf{D_h}$ = diameter of hole



13.4.11.2.1.5 Dimensional limitations

Notch depth ≥ max ($\mathbf{T + r_{supported\ beam}}$, $\mathbf{T + r_{supporting\ beam}}$)
Angle thickness : $8 \leq t \leq 10$
Angle length : $0.6D \leq L \leq d$

Supported beam:

Edge/end distance : $\geq 1.25\ D_{hole}$
Bolt row spacing : $\geq 2.5\ d_b$

Supporting beam / column:

Edge/end distance : $\geq 1.25\ D_{hole}$
Bolt row spacing : $\geq 2.5\ d_b$
Bolt column gauge : $90 \leq g < 140$
Column web clearance ≤ d

13.4.11.2.2 Double angle welded

Refer to:
[Connecting element - supported side] 1367
[Connecting element - supporting side] 1367

13.4.11.2.2.1 Connecting element - supported

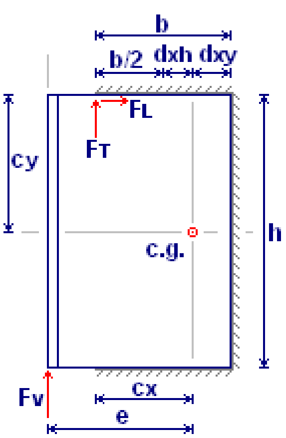
Weld capacity:

BS5950-1: Section 6.8.7.3

$$(F_L/P_L)^2 + (F_T/P_T)^2 < 1.00$$

where:

F_L = F_3
F_T = $F_1 + F_2$
P_L = $p_w a$
P_T = $1.25 p_w a$
F_1 = $F_v/2L$
F_2 = $Mc_x/(2I_p)$
F_3 = $Mc_y/(2I_p)$
M = $F_v L$
a = $0.7s$
I_p = $\Sigma I_x + \Sigma I_y$
ΣI_x = $h^3/12 + 2b(h/2)^2$
ΣI_y = $hd_{xy}^2 + 2(b^3/12 + bd_{xh}^2)$
L = $2b + h$
s = thickness of weld



13.4.11.2.2.2 Connecting element - supporting

Weld capacity:

BS5950-1: Section 6.8.7.3

$$(F_L/P_L)^2 + (F_T/P_T)^2 < 1.00$$

where:

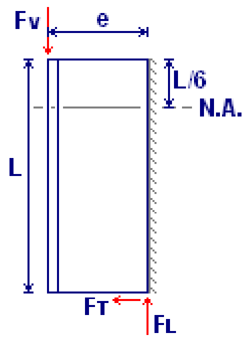
F_L = $F_v/2L$

F_T = $9F_ve/(5L^2)$

P_L = p_wa

P_T = $1.25p_wa$

a = $0.7s$



13.4.11.2.3 Flexible end plate

Refer to:

13.4.11.2.3.1 Supported beam

BS5950-1: Sections 4.2.3

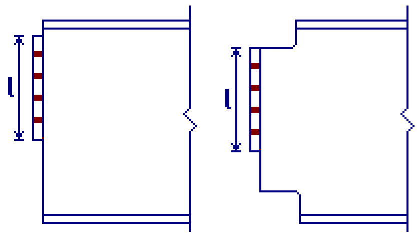
$$F_v / P_v < 1.00$$

where:

P_v = $0.6\ p_yA_v$

A_v = $0.9\ l\ t_w$

t_w = thickness of beam web

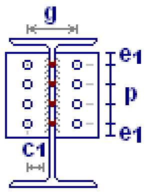


BS5950-1: Section 4.6.1

$F_t / P_t < 1.00$

where:

P_t = $L_e t_w p_y$
L_e = $2e_e + (n-1)p_e$
e_e = min ($e, c_1 + D_h/2$)
p_e = min ($p, 2c_1 + D_h$)
c_1 = $0.5(g - t_w - 2s)$
s = leg length of the fillet weld
t_w = beam web thickness
p = bolt pitch
D_h = diameter of hole

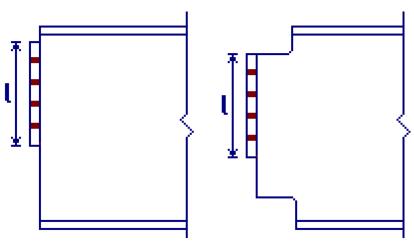


BS5950-1: Section 6.8.7

$F_v / P_v < 1.00$

where:

P_v = $p_w l_{we}\ a$
p_w = design strength of weld (Table 37)
= min **($0.5U_e$, $0.55U_s$)** for steel/electrodes not in the table
l_{we} = $2(l-2s)$
a = $0.7s$
U_e = minimum tensile strength of the electrode
U_s = minimum tensile strength of the parent metal



BS5950-1: Section 4.2.5.4

$M_n / M_{cN} < 1.00$

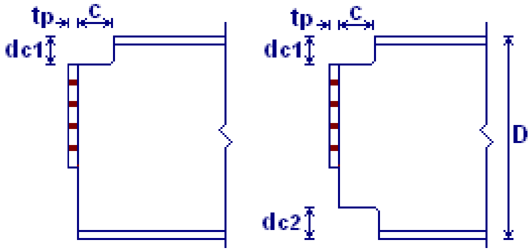
where:

M_{cN}:	Low shear	High shear
Single notch:	$p_y Z_N$	$1.5 p_y Z_N [1 - (F_v/P_{vN})^2]^{1/2}$
Double notch:	$(p_y t_w/6)(D - d_{c1} - d_{c2})^2$	$(p_y t_w/4)(D - d_{c1} - d_{c2})^2 [1 - (F_v/P_{vN})^2]^{1/2}$

P_{vN} = $0.6 p_y A_{vN}$
A_{vN} = $(D - d_{c1})\, t_w$ - single notch

= $\mathbf{0.9(D - d_{c1} - d_{c2})\ t_w}$ - double notch

$\mathbf{Z_N}$ = elastic section modulus at the T-section at the notch



One flange notched:

$\mathbf{d_{c1} \le D/2}$

$\mathbf{c \le D}$ for $\mathbf{D/t_w \le k_2}$

$\mathbf{c \le k_1\ D/(D/t_w)^3}$ for $\mathbf{D/t_w > k_2}$

Both flanges notched

max$\mathbf{(d_{c1},d_{c2}) \le D/5}$

$\mathbf{c \le D}$ for $\mathbf{D/t_w \le k_2}$

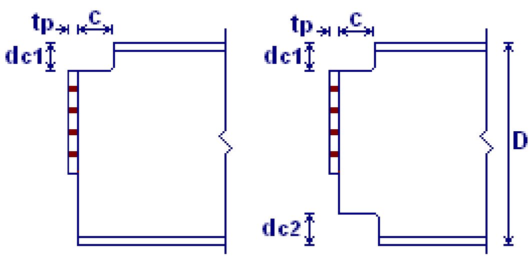
$\mathbf{c \le k_1\ D/(D/t_w)^3}$ for $\mathbf{D/t_w > k_2}$

where:

$\mathbf{k_1}$ = $\mathbf{[16 - (p_y - 275)/16] \times 10^4}$

$\mathbf{k_2}$ = $\mathbf{54.3 - 6.3(p_y - 275)/80}$

$\mathbf{t_w}$ = thickness of supported beam web



13.4.11.2.3.2 Supporting beam/column

Local shear 1370
Bearing 1371
Bolt shear 1371
Bolt tension 1372
Tying capacity 1372

BS5950-1: Sections 4.2.3, 6.2.3

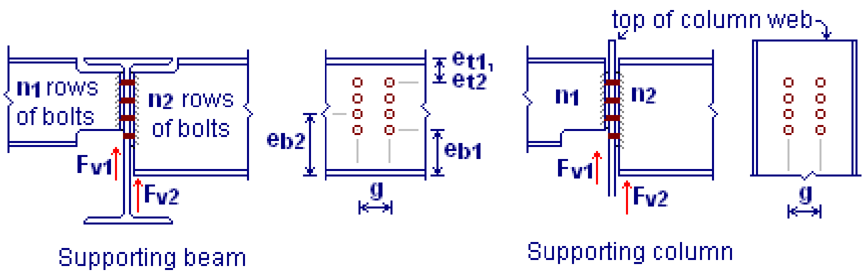
$$\mathbf{0.5\Sigma F_v / P_v < 1.00}$$

where:

$\mathbf{\Sigma F_v}$ = $\mathbf{F_{v1} + F_{v2}}$ (i.e. the program assumes $\mathbf{n_1=n_2}$)

$\mathbf{P_v}$ = min ($\mathbf{0.6\ P_y\ A_v,\ 0.7\ p_y\ K_e\ A_{v,net}}$)

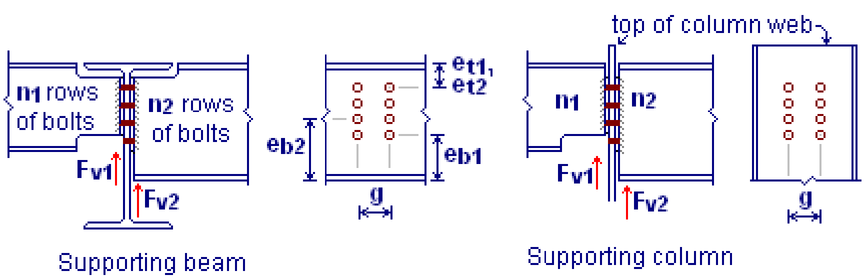
$\mathbf{A_v}$ = $\mathbf{[e_t + (n_2 - 1)p + e_b]\ t_w}$
$\mathbf{A_{v,net}}$ = $\mathbf{A_v - n_2\ D_h\ t_w}$
$\mathbf{e_t}$ = min ($\mathbf{e_{t1}, 5d}$)
$\mathbf{e_b}$ = min ($\mathbf{e_{b2}, g/2, p, 5d}$) - supporting beam
min ($\mathbf{g/2, p, 5d}$) - supporting column
$\mathbf{D_h}$ = diameter of hole
p = bolt pitch
d = diameter of hole
$\mathbf{t_w}$ = supported beam web
$\mathbf{K_e}$ = 1.2 - S275 steel
= 1.1 - S355 steel



BS5950-1: Section 6.3.3.3

$$0.5\Sigma F_v / nP_{bs} < 1.00$$

where:
$\mathbf{\Sigma F_v}$ = $\mathbf{F_{v1} + F_{v2}}$
$\mathbf{P_{bs}}$ = $\mathbf{d\ t_w\ p_{bs}}$
$\mathbf{p_{bs}}$ = bearing strength of supporting beam or column
$\mathbf{D_h}$ = diameter of hole
p = bolt pitch
d = diameter of hole
$\mathbf{t_w}$ = supported beam web
$\mathbf{K_e}$ = 1.2 - S275 steel
= 1.1 - S355 steel

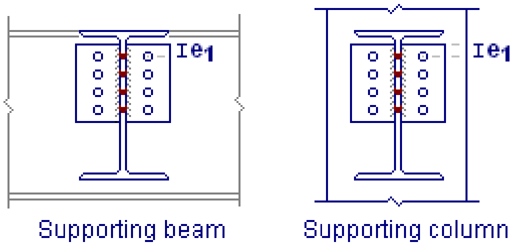


BS5950-1: Section 6.3.2.1

$$F_v / \Sigma P_s < 1.00$$

where:
$\mathbf{P_s}$ = $\mathbf{p_s\ A_s}$ - for all rows of bolts, except:
min ($\mathbf{p_s A_s, 0.5\ e_1\ t_p\ p_{bs}}$) - for the top row of bolts
$\mathbf{p_s}$ = bolt shear strength

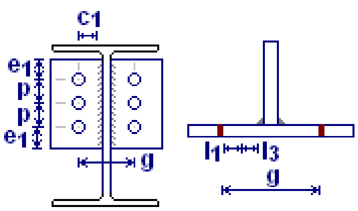
A_s = bolt shear area
t_p = end plate thickness



BS5950-1: Section 6.3.4.3

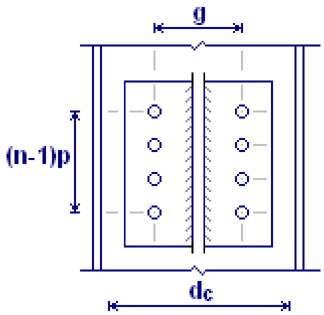
$F_t / P_t < 1.00$

where:
P_t = **$2n\ A_t\ p_{tr}$**
n = number of rows of bolts
A_t = tensile stress area of bolt
p_{tr} = reduced tension strength of bolt in presence of extreme prying
= (minimum tensile strength) / (1.25 x prying ratio)
prying ratio = **$(l_1$ + lever arm) / lever arm**
lever arm = $D_h/2 + 2t_p$
$(l_1 + l_3) = g/2 - t_w/2 - s - D_h/2$
$l_1 = (l_1+l_3)/2$



$F_t / P_t < 1.00$

where:
P_t = **$8M_u\ [\eta_1 + 1.5(1 - \beta_1)^{0.5}\ (1 - \gamma_1)^{0.5}\ ] / (1 - \beta_1)$**
M_u = **$p_u\ t_w^2/4$**
η_1 = **$[(n-1)p - 0.5n\ D_h] / d_c$**
β_1 = **g/d_c**
γ_1 = **D_h/d_c**
n = number of rows of bolts
D_h = diameter of hole
p_u = $U_s/1.25$



13.4.11.2.3.3 End Plate

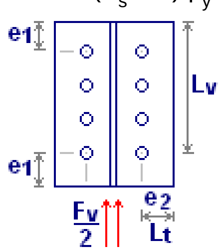
[Plain shear](1373)
[Block shear](1373)
[Bearing](1374)
[Tension](1374)

BS5950-1: Sections 4.2.3, 6.2.3

$0.5F_v / P_v < 1.00$

where:

F_v = design shear force
P_v = min ($0.6\, p_y\, A_v$, $0.7 p_y\, K_e\, A_{v,net}$)
A_v = $0.9[2e_1 + (n-1)p]t_p$
$A_{v,net}$ = $Av - n\, D_h\, t_p$
D_h = diameter of hole
p = bolt pitch
t_c = thickness of plate
K_e = 1.2 - S275 steel
= 1.1 - S355 steel
= $(U_s/1.2)/p_y$ for other steels



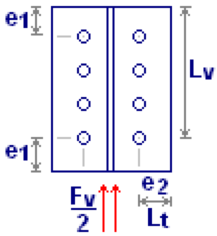
BS5950-1: Section 6.2.4

$0.5F_v / P_r < 1.00$

where:

F_v = design shear force
P_r = $0.6\, p_y\, t_p\, [\, L_v + K_e\, (L_t - k\, D_h)]$
L_v = $e_1 + (n - 1)\, p$
k = 0.5
L_t = e_2
D_h = diameter of hole

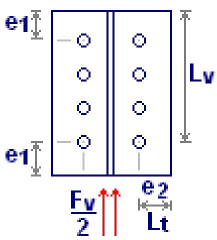
p = bolt pitch
t_c = thickness of plate
K_e = 1.2 - S275 steel
= 1.1 - S355 steel
= $(U_s/1.2)/p_y$ for other steels



BS5950-1: Sections 6.3.3.3

$0.5F_v / P_{bs} < 1.00$

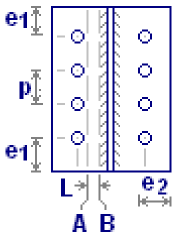
P_{bs} = min (**d, $0.5e_1$**)t_p p_{bs}
p_{bs} = bearing strength of cleat (Table 32)
d = bolt diameter



$F_t / P_t < 1.00$

where:

P_t	=	$2 (M_{uA} + M_{uB}) \varphi / L$
M_{uA}	=	$p_u L_{eA} t_p^2/4$
M_{uB}	=	$p_u L_{eB} t_p^2/4$
L_{eA}	=	$2e_{eA} + (n-1)p_{eA}$
e_{eA}	=	min(e_1, e_2)
p_{eA}	=	min$(p, 2e_2)$
L_{eB}	=	$2e_{eB} + (n-1)p_{eB}$
e_{eB}	=	min$(e_1, c_1 + D_h/2)$
p_{eB}	=	min$(p, 2c_1 + D_h)$
c_1	=	$0.5(g-t_w-2s)$
s	=	leg length of fillet weld
p_u	=	$U_s/1.25$
φ	=	$1 - [2 (F_v/2p_{v,min}) - 1]^2$ for $Fv/p_{v,min} > 0.6$, else:
	=	1.
$p_{v,min}$	=	min (plain shear capacity, block shear capacity)
L	=	$c_1 - D_h/2$



13.4.11.2.3.4 Dimensional limitations

Plate edge distance	:	$\geq 1.25\ D_{hole}$
Plate end distance	:	$\geq 1.25\ D_{hole}$
Bolt gauge on support	:	$90 \leq g < 140$ mm
End plate width	:	$\leq (D - 2T - 2r)_{\text{supporting columns}}$
Plate thickness	:	$8 \leq t_p < 12$ mm
Plate depth	:	$\geq 0.6\ D_{\text{supported beam}}$
Bolt - row spacing	:	$\geq 2.5\ d_b$
Bolt - column spacing	:	$\geq 2.5\ d_b$
Notch depth	:	$\geq (T + r)_{\text{supported beam}}$
	:	$\geq (T + r)_{\text{supporting beam}}$

13.4.11.2.4 Fin plate

Refer to:
Supported beam 1375
Supporting beam 1380
Fin plate 1381

13.4.11.2.4.1 Supported beam

Refer to:
Plain shear 1375
Block shear 1376
Bearing 1376
Bearing - tension 1377
Bolt shear 1377
Punching shear 1378
Tension 1378
Notch - shear & bending 1379
Local stability 1379
Shear & bending 1380

BS5950-1: Sections 4.2.3, 6.2.3

$$\mathbf{F_v / P_v < 1.00}$$

where:
$\mathbf{F_v}$ = design shear force

P_v = min ($0.7\ p_y\ A_v\ ,\ 0.7 p_y\ K_e\ A_{v,net}$)

A_v = $[e_t + (n-1)p + e_b]t_w$ (un-notched and single notched beams)

$0.9[e_t + (n-1)p + e_b]t_w$ (double notched beams)

$A_{v,net}$ = $A_v - nD_h t_w$

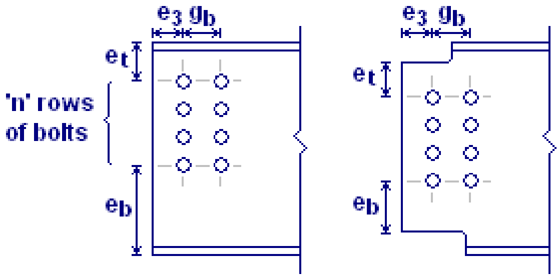
D_h = diameter of hole

p = bolt pitch

t_w = supported beam web

K_e = 1.2 - S275 steel

= 1.1 - S355 steel



BS5950-1: Section 6.2.4

$F_v / P_r < 1.00$

where:

F_v = design shear force

P_r = $0.6\ p_y\ t_w\ [\ Lv + K_e\ (L_t - k\ D_h)]$

L_v = $e_t + (n - 1)\ p$

k = 0.5 - single line of bolts

2.5 - double line of bolts

L_t = e_3 - single line of bolts

$e_3 + g_b$ - double line of bolts

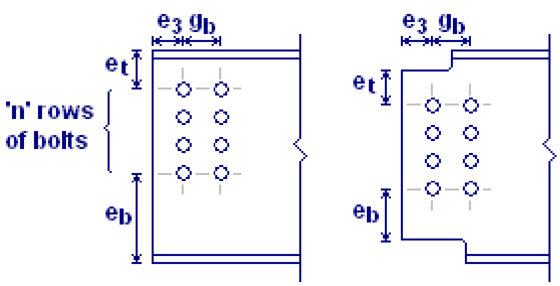
D_h = diameter of hole

p = bolt pitch

t_w = supported beam web

K_e = 1.2 - S275 steel

= 1.1 - S355 steel



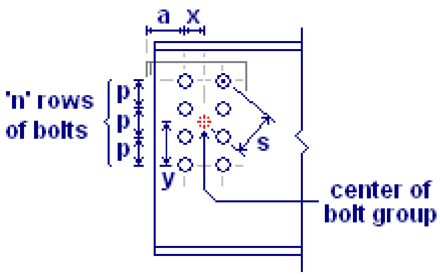
BS5950-1: Sections 6.3.3.3

$F_s / P_{bs} < 1.00$

where:

	Single line	Double line
F_s	$= (F_{sv}^2 + F_{sm}^2)^{1/2}$	$= [(F_{sv} + F_{smv})^2 + F_{smh}^2]^{1/2}$

F_{sv}	$= F_v/n$	$= F_v/2n$
F_{sm}	$= F_v a/Z_{bg}$	
F_{smv}		$= Mx/I_{bg}$
F_{smh}		$= My/I_{bg}$
Z_{bg}	$= n(n+1)p/6$	
M		$= F_v\ (a + x)$
I_{bg}		$= \Sigma s^2$



p = bolt pitch

p_{bs} = min($dt_w p_{bs}$, $0.5e_3 p_{bs} t_w$)

$F_t / P_{bs} < 1.00$

where:

P_{bs} = min (**1.5d, 0.5e$_3$**) **n t_w p_{bs}** - single line of bolts

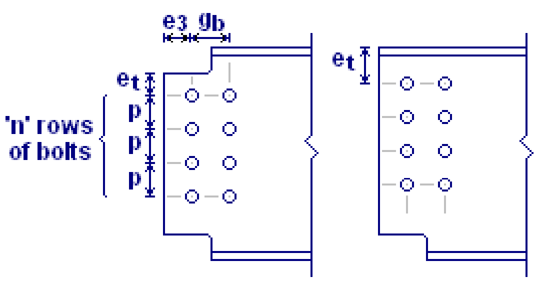
= min (**3d, 1.5d +0.5e$_3$**) **n t_w p_{bs}** - double line of bolts

p_{bs} = bearing strength of beam web - Table 32

t_w = beam web thickness

p = bolt pitch

D_h = diameter of hole



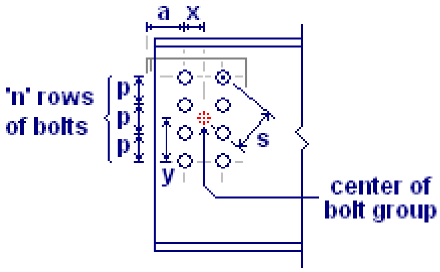
BS5950-1: Sections 6.3.2.1

$F_s / 2P_s < 1.00$

where:

	Single line	Double line
F_s	$= (F_{sv}^2 + F_{sm}^2)^{1/2}$	$= [(F_{sv} + F_{smv})^2 + F_{smh}^2]^{1/2}$
F_{sv}	$= F_v/n$	$= F_v/2n$
F_{sm}	$= F_v a/Z_{bg}$	
F_{smv}		$= Mx/I_{bg}$
F_{smh}		$= My/I_{bg}$
Z_{bg}	$= n(n+1)p/6$	
M		$= F_v\ (a + x)$

$I_{bg} = \Sigma s^2$



p = bolt pitch

$2P_s$ = $2p_sA_s$

p_s = shear strength of bolt (from Table 30)

$t_f / [t_w(U_{sc}/f_b)] < 1.00$

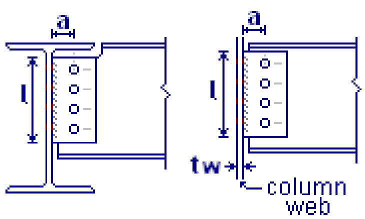
where:

U_{sc} = ultimate tensile strength of supporting member

f_b = min$(F_v a/Z_{gross}, p_{yf})$

p_{yf} = design strength of fin plate

Z_{gross} = $t_f\, l^2/6$



$F_t / P_t < 1.00$

where:

P_t = $L_e\, t_w\, p_y$

L_e = $2e_e + (n\text{-}1)p_e - nD_h$

e_e = min (e_3, e_t) - single line of bolts

= min $(e_3 + g_b - D_h, e_t)$ - double line of bolts

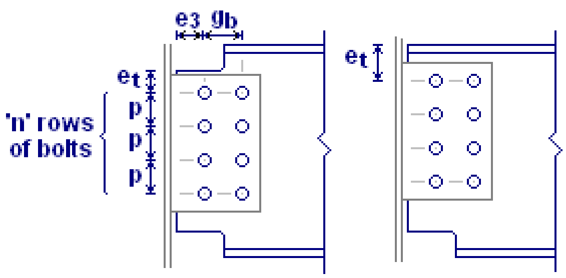
p_e = min $(p, 2e_3)$ - single line of bolts

= min $[p, 2(e_3 + g_b - D_h)]$ - double line of bolts

t_w = beam web thickness

p = bolt pitch

D_h = diameter of hole



BS5950-1: Section 4.2.5.4

M_n/M_{cN} < 1.00

where:

M_{cN}:	Low shear	High shear
Single notch:	$p_y Z_N$	$1.5 p_y Z_N\, [1 - (F_v/P_{vN})^2]^{1/2}$
Double notch:	$(p_y t_w/6)\,(e_t + (n-1)p + e_b)^2$	$(p_y t_w/4)(e_t + (n-1)p + e_b)^2\, [1 - (F_v/P_{vN})^2]^{1/2}$

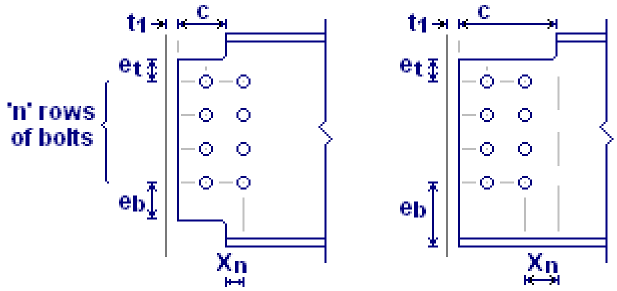
$M_n = F_v(t_1+c)$

$P_{vN} = 0.6 p_y A_{vN}$

$A_{vN} = (e_t - (n-1)p + e_b)\, t_w$ - single notch

$= 0.9(e_t - (n-1)p + e_b)\, t_w$ - double notch

Z_N = elastic section modulus of the T-section at the notch



For double bolt lines, if **X_N < 2d**:

M_n = max$[F_v(t_1+c)\,,\, F_v(t_1+e_3 + g_b)]$

M_{cC}: Low shear	High shear
$(p_y t_w/6)\,(e_t + (n-1)p + e_b)^2$	$(p_y t_w/4)(e_t + (n-1)p + e_b)^2\, [1 - (F_v/P_{v,min})^2]^{1/2}$

One flange notched:

$d_{c1} \le D/2$

$c \le D$ for $D/t_w \le k_2$

$c \le k_1\, D/(D/t_w)^3$ for $D/t_w > k_2$

Both flanges notched

max$(d_{c1}, d_{c2}) \le D/5$

$c \le D$ for $D/t_w \le k_2$

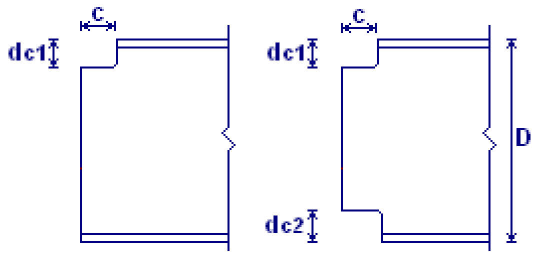
$c \le k_1\, D/(D/t_w)^3$ for $D/t_w > k_2$

where:

$k_1 = [16 - (p_y - 275)/16] \times 10^4$

$k_2 = 54.3 - 6.3(p_y - 275)/80$

t_w = thickness of supported beam web



BS5950-1: Section 4.3.6

$$\mathbf{M/(M_{cBC} + M_{cAB}) < 1.00}$$

where:

$\mathbf{M_{cAB}} = \mathbf{P_{vAB}}(n-1)p$

	Low shear	High shear
$\mathbf{M_{cBC}} =$	$\mathbf{(p_y t_w/6)\,[(n-1)p]^2}$	$\mathbf{(p_y t_w/4)[(n-1)p]^2\,[1-(F_{vBC}/P_{vBC})^2]^{1/2}}$

	Single line of bolts	Double line of bolts
$\mathbf{M} =$	$\mathbf{F_v a}$	$\mathbf{F_v\,(a+g_b)}$
$\mathbf{P_{vAB}} =$	$\min\mathbf{(0.6p_y e_3 t_w,\ 0.7p_y K_e(e_3-0.5D_h)t_w)}$	$\min\mathbf{(0.6p_y(e_3+g_b)t_w,\ 0.7p_y K_e(e_3+g_b-1.5D_h)t_w)}$

$\mathbf{P_{vBC}} = \min\mathbf{(0.6p_y\,(n-1)pt_w,\ 0.7p_y K_e(n-1)(p-D_h)t_w)}$

$\mathbf{F_{vBC}} = \mathbf{F_v - (P_v - P_{vBC})}$ but ≥ 0

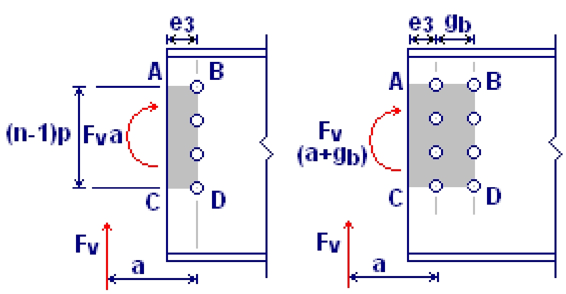
$\mathbf{t_f}$ = fin plate thickness

$\mathbf{t_w}$ = beam web thickness

$\mathbf{p}$ = bolt pitch

$\mathbf{D_h}$ = diameter of hole

$\mathbf{K_e}$ = 1.2 - S275 steel
 = 1.1 - S355 steel



13.4.11.2.4.2 Supporting beam

Refer to:

Local shear 1380

Weld shear & tension 1381

Tying capacity 1381

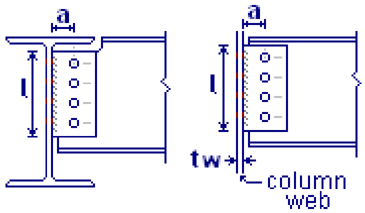
BS5950-1: Sections 4.2.3

$$\mathbf{0.5F_v\,/P_v < 1.00}$$

where:

$\mathbf{P_v} = \mathbf{0.6\,p_y\,A_v}$

A_v = 0.9 l t_w

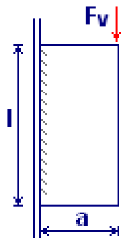


BS5950-1: Section 6.8.2.3

$(f_v/P_v)^2 + (f_t/P_t)^2 < 1.00$

where:

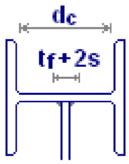
P_v = **p_w a**
P_t = **1.25p_w a**
f_v = **$F_v/(2l_e)$**
F_t = **$(F_v ay)/(2I_w)$ + $F_t/(2l_e)$**
a = **0.7s**
l_e = **l - 2s**
y = **$l_e/2$**
I_w = **$l_e^3/12$**



$F_t / P_t < 1.00$

where:

P_t = **$8M_u [\eta_1 + 1.5(1 - \beta_1)^{0.5}] / (1 - \beta_1)$**
M_u = **$p_u t_w^2/4$**
η_1 = **l/d_c**
β_1 = **$(t_f + 2s)/d_c$**
s = leg length of weld
p_u = **$U_s/1.25$**



13.4.11.2.4.3 Fin plate

Refer to:

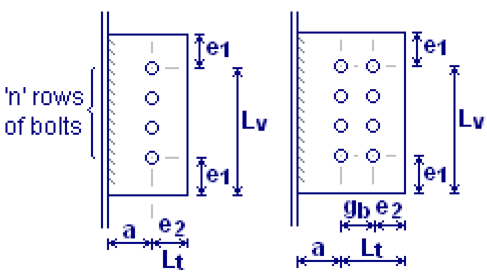
Plain shear 1382
Block shear 1382
Bearing 1383

BS5950-1: Sections 4.2.3, 6.2.3

$\mathbf{F_v / P_v < 1.00}$

where:

$\mathbf{F_v}$	=	design shear force
$\mathbf{P_v}$	=	min ($\mathbf{0.6\ p_y\ A_v\ ,\ 0.7 p_y\ K_e\ A_{v,net}}$)
$\mathbf{A_v}$	=	$\mathbf{0.9[e_t + (n\text{-}1)p + e_b]t_f}$
$\mathbf{A_{v,net}}$	=	$\mathbf{Av - n\ D_h\ t_f}$
$\mathbf{D_h}$	=	diameter of hole
p	=	bolt pitch
$\mathbf{t_c}$	=	thickness of fin plate
$\mathbf{K_e}$	=	1.2 - S275 steel
	=	1.1 - S355 steel
	=	$(U_s/1.2)/p_y$ for other steels

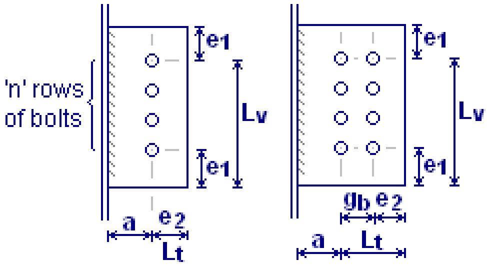


BS5950-1: Section 6.2.4

$\mathbf{F_v / P_r < 1.00}$

where:

$\mathbf{F_v}$	= design shear force
$\mathbf{P_r}$	= $\mathbf{0.6\ p_y\ t_w\ [\ Lv + K_e\ (L_t - k\ D_h)]}$
$\mathbf{L_v}$	= $\mathbf{e_1 + (n - 1)\ p}$
k	= 0.5 - single line of bolts
	2.5 - double line of bolts
$\mathbf{L_t}$	= $\mathbf{e_2}$ - single line of bolts
	$\mathbf{e_2 + g_b}$ - double line of bolts
$\mathbf{D_h}$	= diameter of hole
p	= bolt pitch
$\mathbf{t_w}$	= supported beam web
$\mathbf{K_e}$	= 1.2 - S275 steel
	= 1.1 - S355 steel

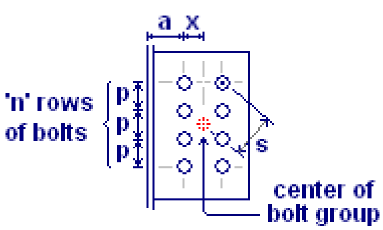


BS5950-1: Sections 6.3.2.1

$F_s / P_{bs} < 1.00$

where:

	Single line	Double line
F_s	$= (F_{sv}^2 + F_{sm}^2)^{1/2}$	$= [(F_{sv} + F_{smv})^2 + F_{smh}^2]^{1/2}$
F_{sv}	$= F_v/n$	$= F_v/2n$
F_{sm}	$= F_v a/Z_{bg}$	
F_{smv}		$= Mx/I_{bg}$
F_{smh}		$= My/I_{bg}$
Z_{bg}	$= n(n+1)p/6$	
M		$= F_v (a + x)$
I_{bg}		$= \Sigma s^2$



p = bolt pitch

P_{bs} = min($d, 0.5e_2$)$p_{bs} t_f$

$F_t / P_{bs} < 1.00$

where:

P_{bs} = min ($1.5d, 0.5e_2$) $n\ t_f\ p_{bs}$ - single line of bolts

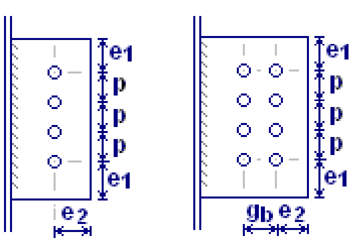
= min ($3d, 1.5d + 0.5e_2$) $n\ t_f\ p_{bs}$ - double line of bolts

p_{bs} = bearing strength of beam web - Table 32

t_f = fin plate thickness

p = bolt pitch

D_h = diameter of hole



BS5950-1: Section 4.2.5.4

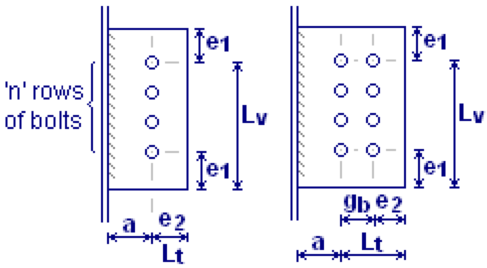
$M/M_c < 1.00$

where:

M_c: Low shear — $(p_y t_f/6)(2e_1 + (n-1)p)^2$; High shear — $(p_y t_f/4)(2e_1 + (n-1)p)^2 [1- (F_v/P_{v,min})^2]^{1/2}$

t_f = fin plate thickness

p = bolt pitch



$F_t / P_t < 1.00$

where:

P_t = $\min(p_y A, K_e p_y A_{net})$

A = $l\ t_f$

A_{net} = $A - nD_h t_f$

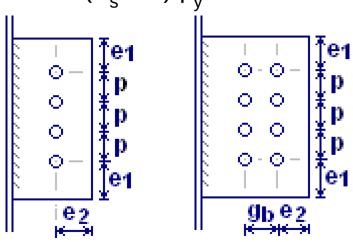
D_h = diameter of hole

t_f = fin plate thickness

K_e = 1.2 - S275 steel

= 1.1 - S355 steel

= $(U_s/1.2)/p_y$ for other steels



BS5950-1: Section 4.3.6

$M/M_{min} \le 1.00$

where:

M = $F_v a$

M_{min} = $\min(M_b/m_{LT}, M_c)$

M_b = $p_b Z_x$

Z_x = $t_f l^2/6$

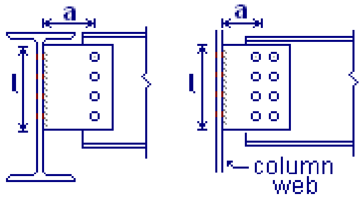
p_b = bending strength of fin plate obtained from BS5950-1, AnnexB, based on λ_{LT}, where:

λ_{LT} = $2.8(al/1.5t_f^2)^{1/2}$

t_f = thickness of fin plate

m_{LT} = 0.6

This check is required for long fin plates, i.e. **a > t_f/0.15**



Dimensional limitations:

End projection	≥	20 for D > 610
	≥	10 for D ≤ 610
Column web clearance	≤	d
Column web slenderness	≤	40ε
Beam edge distance	≥	$2d_b$
Plate/web thickness	≤	$0.5 - [0.08(p_y-275)/80]d_b$
Plate length	≥	0.6D
	≤	d
Plate edge distance	>	$2d_b$
Plate end distance	>	$2d_b$
Bolt row spacing	≥	$2.5d_b$
Weld leg length	≥	$0.8t_f$

13.4.11.2.5 Moment end plate

Refer to:

Moment capacity 1385
Supported beam 1387
Supporting beam/column 1388
End plate 1389

13.4.11.2.5.1 Moment capacity

Refer to:

Web bearing 1385
Web buckling 1386
Flange bearing 1386
Panel shear 1386
Moment capacity 1386

BS5950-1: Section 4.5.2

$$\mathbf{F_c/P_{bw} = 1.00}$$

where:

$\mathbf{P_{bw}}$ = $\mathbf{(b_1+nk)p_yt_w}$

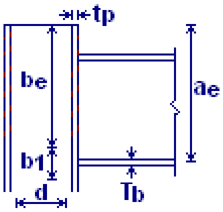
n = 5.0 for a continuous column, else:

= $\mathbf{2 + 0.6b_e/x}$ ≤ 5.0

k = **T + r** for rolled sections, else:

= **T**

b_1 = **$T_b + 1.6S_{wf} + 2t_p$**
S_{wf} = flange weld thickness

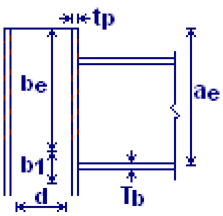


BS5950-1: Section 4.5.3

$F_c/P_x = 1.00$

where:

P_x = **$p_{bw}(25\varepsilon\, t_{wc})/[(b_1+nk)d]^{0.5}$** when **$a_e \geq 0.7d$**
= **$p_{bw}\,[(a_e+0.7d)/1.4d](25\varepsilon\, t_{wc})/[(b_1+nk)d]^{0.5}$** when **$a_e < 0.7d$**
ε = **$(275/p_y)^{0.5}$**
t_{wc} = column web thickness



The program assumes that the column flange is restrained against lateral movement relative to the other flange.

$F_c/P_c = 1.00$

where:

P_c = **$1.4\ p_y\ T_b\ B_b$** for rolled sections, else:
= **$1.2\ p_y\ T_b\ B_b$**
T_b = beam flange thickness
B_b = beam flange width

$F_c/P_v = 1.00$

where:

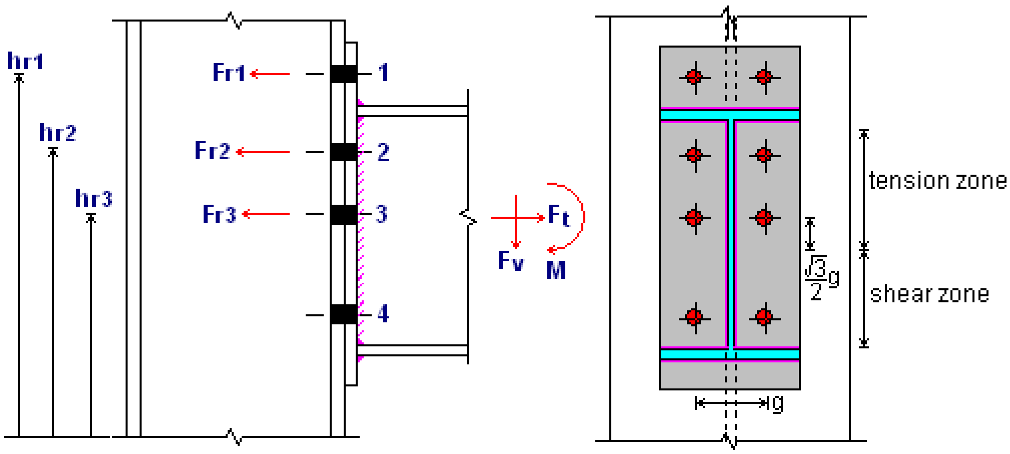
P_v = **$0.6p_yt_wD$**
D = total depth of column
t_w = column web thickness

$M_m/M_c = 1.00$

where:

M_m = **$M + F_th_c$**
F_t = axial force (positive for tension)
h_c = distance between center of section to center of compression flange
M_c = Σ**$(F_{ri}\ h_i)$**

F_{ri} = final force in bolt row
h_i = distance from center of compression to row ***i***
ΣF_{ri}-F_t = min(Σ**p_{ri}-F_t , p_{bw} , p_x, p_c, p_v)**
p_{ri} = potential force in bolt row ***i***
p_{bw} = column web crushing (bearing)
p_x = column web buckling
p_c = beam flange crushing (bearing)
p_v = column web panel shear



13.4.11.2.5.2 Supported beam

Refer to:
[Web shear 1387]
[Tension flange weld 1387]
[Tension zone weld 1388]
[Shear zone weld 1388]

BS5950-1: Section 4.2.3

F_v /P_v < 1.00

where:
F_v = design shear force
P_v = **0.6 p_y A_v**
A_v = **Dt_w**

BS5950-1: Section 6.8.7.3

F_{wt} /P_w < 1.00

where:
P_w = **$2.5p_waL$**
p_w = min**(0.5U_e, 0.55U_s)**
a = **0.7s**
L = **2B - t_w**
F_{wt} = min (**BTp_y, F_{r1}+F_{r2}+F_{r3}**)

BS5950-1: Section 6.8.7.3

$F_y / P_w < 1.00$

where:

P_w = $2.5 p_w a$

p_w = min$(0.5U_e, 0.55U_s)$

a = $0.7s$

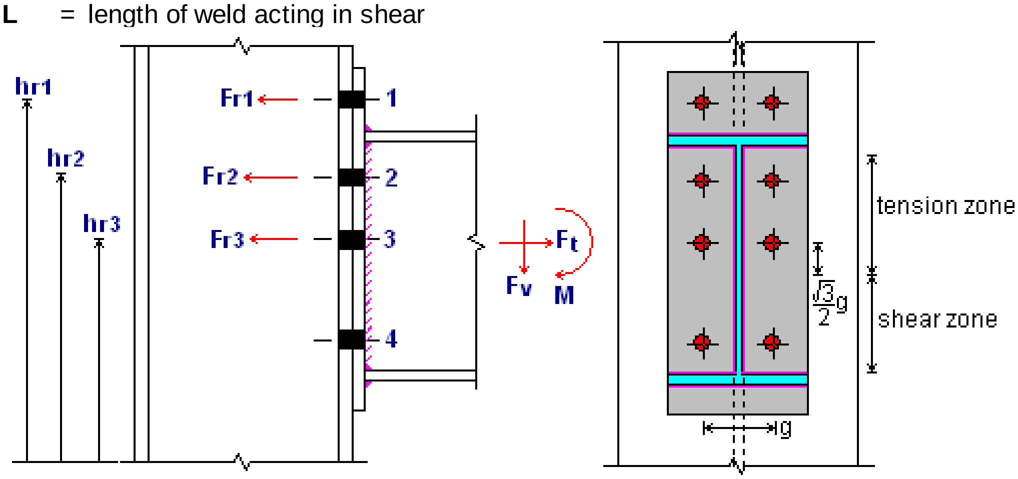
F_y = $p_y t_w$ (full strength)

P_w = $2 p_w a L$

p_w = min$(0.5U_e, 0.55U_s)$

a = $0.7s$

L = length of weld acting in shear



13.4.11.2.5.3 Supporting beam/column

Refer to:

Bearing 1388

Bolt shear 1388

BS5950-1: Section 6.3.3.3

$F_v / NP_{bs} < 1.00$

where:

P_{bs} = dTp_{bs} (column)

d = bolt diameter

T = column flange thickness

p_{bs} = steel bearing strength (Table 32)

N = number of bolts

BS5950-1: Section 6.3.2.1

$F_v / P_s < 1.00$

where:

P_s = $p_s A_s(n_s + 0.4n_t)$

p_s = $0.4U_b$

U_b = specified minimum tensile strength
n_s = number of bolts not in tension
n_t = number of bolts in tension

13.4.11.2.5.4 End plate

Bearing:

BS5950-1: Section 6.3.3.3

$F_v / NP_{bs} < 1.00$

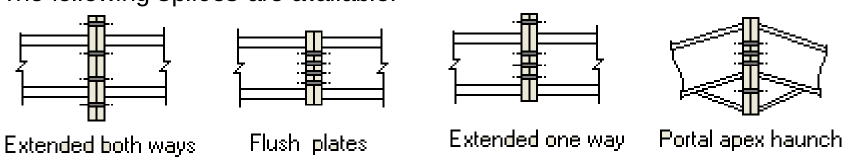
where:
P_{bs} = dTp_{bs} (plate)
d = bolt diameter
t_p = plate thickness
p_{bs} = steel bearing strength (Table 32)

13.4.11.2.6 Splices

The program designs splices at node locations when two *STRAP* beams that form a continuous line are selected. ***Splices can be designed only for two identical sections***.

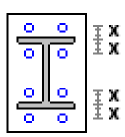
The splices are designed similar to moment end-plate connections 1385.

The following splices are available:



Extended both ways:

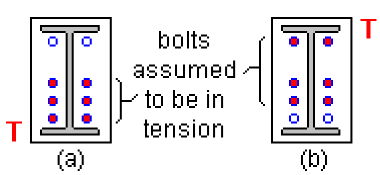
The splice is always symmetric with 8 bolts, 4 on each side, with 2 bolts on each extension.



The program assumes that only the bolts on one side are in tension.

Flush plates:

The program concentrates the required bolts on the tension side and add a pair of bolts on the compression side. Refer to Figure (a). The "tension side" is determined from the largest moment.



The program also checks the capacity of the connection for a moment with the opposite sign. All bolts are assumed to be in tension, except for the pair nearest the compression face. Refer to figure (b).

13.4.11.2.7 Base plate

The program designs two types of base plates:

- Simple connections 1390
- Moment connections 1391

13.4.11.2.7.1 Simple connection

The program carries out the following design checks:

Concrete bearing 1390
Plate strength 1390
Weld shear 1391

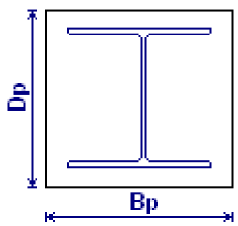
$$(F_c/B_pD_p)/(0.6f_{cu}) = 1.00 \qquad (4.13.1)$$

where:

F_c = design axial load due to factored loads

f_{cu} = characteristic cube strength of the concrete base

and:



$$M_{pl}/M_c = 1.00 \qquad (4.13.2.2)$$

where:

M_{pl} = $0.6f_{cu}c^2/2$

M_c = $P_yt^2/6$

rearranging the terms gives the required plate thickness:

$$t = c[3(0.6f_{cu})/P_y]^{1/2}$$

which is the equation in the Code.

where:

f_{cu} = characteristic cube strength of the concrete base

t = plate thickness

P_y = design strength of the base plate

c = the projection dimension, calculated by solving the following equation:

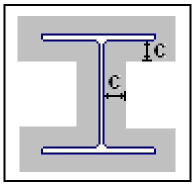
$$A_{req} = F_c/(0.6f_{cu}) = 4c^2 + c(Per_{col}) + A_{col}$$

where:

F_c = design axial load due to factored loads

Per_{col} = column perimeter

A_{col} = cross section area of the column



The effective area is adjusted if there is overlap.

BS5950-1: Section 6.8.7

$F_v / P_v < 1.00$

where:

P_v = **$p_w l_{we}$ a**
p_w = design strength of weld (Table 37)
= min **($0.5U_e$, $0.55U_s$)** for steel/electrodes not in the table
l_{we} = **2(l-2s) =** total effective length of welds in the direction of the shear
a = **0.7s**
U_e = minimum tensile strength of the electrode
U_s = minimum tensile strength of the parent metal

13.4.11.2.7.2 Moment connection

The program carries out the following design checks:

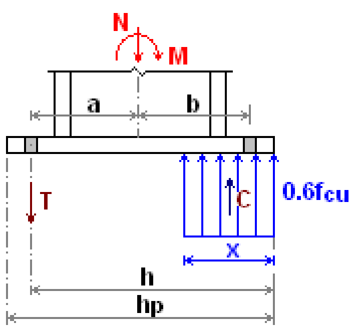
Plate strength 1392
Concrete bearing 1390
Tensile capacity 1392
Anchorage 1393
Shear transfer 1393
Tension flange welds 1393
Shear zone web welds 1394

The program first calculates the length of the compressive stress block, **x**, by solving the following quadratic equation:

$M = 0.6f_{cu}\ b_p\ x\ (h - x/2) - N(h - h_p/2)$

$C = 0.6f_{cu}\ b_p\ x$

$T = C - N$



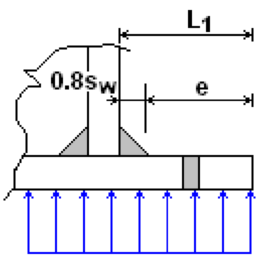
A special case dealt with by the program is when no tension in the bolt is required to resist the moment (the whole area of the base is in compression.

Compression side bending:

M_{pl}/M_c = 1.00 (4.13.2.3)

where:

M_{pl} = **$0.6f_{cu}\ e^2/2$**
Mc = **$P_y t_p^2/4$**
f_{cu} = characteristic cube strength of the concrete base
e = **$L_1 - 0.8s_w$**
P_y = design strength of the plate
t_p = plate thickness

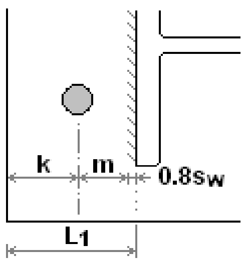


Tension side bending:

M_{pl}/M_c = 1.00 (4.13.2.3)

where:

M_{pl} = **T(m)**
Mc = **$P_y t_p^2/4$**
m = **$L_1 - k - 0.8s_w$**
P_y = design strength of the plate
t_p = plate thickness



T/nP_t = 1.00 (6.3.4.2)

where:

T = tension force
n = number of bolts
P_t = tension strength of bolts, according to Table 34 in the Code. For grades not listed in the Table, **P_t = $0.7U_b = Y_b$**

f_v/v_c = 1.00 (BS8110 - 3.7.7)

where:

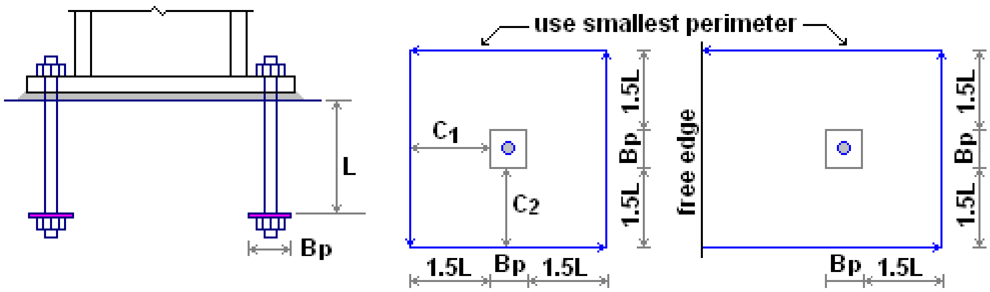
f_v = **T/PL**

v_c = **$0.3358(f_{cu}/25)^{1/3}\ (400/L)^{1/4}$**

L = effective depth of the bolt (see below).
when **C_1<1.5L** and **C_2<1.5L**, then **L = max(C_1,C_2)/1.5**

P = perimeter for punching shear

f_{cu} = characteristic cube strength of the concrete base



F_v/P_s = 1.00

where:

F_v = design shear force

P_s = **$n_sP_{ss} + n_tP_{ts}$**

n_s = number of bolts in the non-tension zone

P_{ss} = shear capacity of a bolt in the non-tension zone

= **$\min(p_sA_s,\ dt_pp_b, 6d^2f_{cu})$**

n_t = number of bolts in the tension zone

P_{ts} = shear capacity of a bolt in the tension zone

= **$\min(0.4p_sA_s,\ dt_pp_b, 6d^2f_{cu})$**

p_s = shear strength of the bolt

A_s = shear area of the bolt

p_b = bearing strength of the base plate

d = bolt diameter

t_p = plate thickness

F_{wt}/P_w = 1.00 (6.8.7.3)

where:

F_{wt} = **$\min[A_f\ p_w\ ,\ M/(D\text{-}T) - N(A_f/A_{tot})]$**

P_w = **$1.25p_waL$**

A_f = area of the column flange

p_w = design strength of the fillet weld

M = design moment

N = design axial force (+ = compression)

D = overall depth of column section

T = column flange thivckness

A_{tot} = column cross-section area
a = effective throat size of the weld = **0.7s**
L = length of the weld

F_v/P_w = 1.00 (6.8.7.3)

F_v = design shear force
P_w = $2p_w aL$
a = effective throat size of the weld = **0.7s**
L = length of weld

13.4.11.3 EN 1993-1-8

The program checks and designs connections according to:

- En 1993-1-8 - Eurocode 3: Design of Steel Structures - Part 1-8 : Design of Joints

The following design aids have been used:

- [1] ECCS Technical Committee 10 Structural Connections - European Recommendations for the Design of Simple Joints in Steel Structures. 1st Edition - 2009
- [2] SCI/BCSA- Joint in Steel Construction – Simple Connections, Publication p212, 2002.
- [3] Access steel NCCI: SN013a
- [4] NCCI: Design model for simple column bases- axially loaded I section columns - SN037a
- [5] NCCI: Design of fixed column base joints - SN043a

13.4.11.3.1 General

13.4.11.3.1.1 Single bolt - shear

The shear capacity of a single bolt is calculated as :

$$F_{v,Rd} = \frac{\alpha_v f_{ub} A}{\gamma_{M2}}$$

where:

A = gross area of the bolt section. When the shear plane passes through the threads the tension area of the bolt A_s is used instead of the gross area.
f_{ub} = the tensile strength of the bolt. Refer to Code Table 3.1.
α_v = a factor that is a function of the bolt type and the location of the shear plane.
When the shear plane passes through the threads:
= 0.6 for bolt types 4.6, 5.6, 8.8
= 0.5 for bolt types 4.8, 5.8. 6.8, 10.9
= 0.5 for all other types and user-defined bolts.
When the shear plane does not pass through the threads:
= 0.6
γ_{M2} = 1.25 (may be specified in the Setup option)

13.4.11.3.1.2 Single bolt - bearing

The bearing capacity of a single bolt is calculated as :

$$F_{b,Rd} = \frac{k_1 \alpha_b f_u \cdot d \cdot t}{\gamma_{M2}}$$

where:

$$\alpha_b = \min\left(\alpha_d; \frac{f_{ub}}{f_u}; 1.0\right)$$

in the direction of the load:

- edge bolt: $\alpha_d = e_1/3d_0$
- interior bolt: $\alpha_d = p_1/3d_0 - {}^1/_4$

perpendicular to the direction of the load:

- edge bolt: $k_1 = \min\left(2.8\frac{e_2}{d_0} - 1.7; 2.5\right)$
- interior bolt: $k_1 = \min\left(1.4\frac{p_2}{d_0} - 1.7; 2.5\right)$

k_1 = a factor that is a function of the distance from the bolt to the edge (in the direction perpendicular to the load)
α_b = a factor that considers the distance from the bolt to the edge, the bolt spacing in the direction of the load and the possibility of failure in bearing of the bolt.
d = diameter of the bolt
d_0 = diameter of the bolt hole
t = thickness of the part with the hole
γ_{M2} = 1.25
f_{ub} = tensile strength of the bolt
f_u = tensile strength of the part being checked
e_1 = the distance to the edge in the direction of the load
e_2 = the distance to the edge in the direction perpendicular to the load
p_1 = the distance between the bolts in the direction of the load
p_2 = the distance between the bolts perpendicular to the direction of the load

Note:

- when there are beams connected to both sides of a column web the program sums the forces transferred to the web from both sides.

13.4.11.3.1.3 Single bolt - tension

The tension capacity of a single bolt is calculated as :

$$F_{t,Rd} = \frac{k_2 f_{ub} A_s}{\gamma_{M2}}$$

where:
k_2 = 0.9
f_{ub} = the tensile strength of the bolt
A_s = the net area of the bolt
γ_{M2} = 1.25

13.4.11.3.1.4 Single bolt - shear & tension

The combined shear & tension capacity of a single bolt is calculated as :

$$\frac{F_v}{F_{v,Rd}} + \frac{F_t}{1.4F_{t,Rd}} \le 1.0$$

13.4.11.3.1.5 Block tearing

Block tearing the failure in shear along a plane passing through the bolt holes accompanied by the tearing along a row of bolts lying on a plane perpendicular to the shear plane.

$$V_{eff,1,Rd} = \frac{f_u A_{nt}}{\gamma_{M2}} + \frac{f_y A_{nv}}{\sqrt{3}\gamma_{M0}}$$

where:

$\mathbf{f_u}$ = plate tensile strength

$\mathbf{f_y}$ = plate yield strength

$\mathbf{A_{nv}}$ = net area subject to shear

= $\mathbf{t_p (h_p - e_1 - (n_1 - 0.5)\, d_0)}$

$\mathbf{A_{nt}}$ = net area subject to tension for two bolt columns

= $\mathbf{t_p (e_2 - d_0/2)}$ for 2 columns of bolts

= $\mathbf{t_p (p_2 + e_2 - 3d_0/2)}$ for 4 columns of bolts

γ_{M2} = 1.25

γ_{M0} = 1.00

Note:

- the capacity is reduced by a factor of 2 when the bolts are loaded eccentrically.

$$V_{eff,2,Rd} = \frac{0.5 f_u A_{nt}}{\gamma_{M2}} + \frac{f_y A_{nv}}{\sqrt{3}\gamma_{M0}}$$

13.4.11.3.1.6 Weld capacity

The forces acting on the weld are divided into their components relative to the axes of the weld:

$$\sqrt{\left[\sigma_{\perp}^2 + 3\left(\tau_{\perp}^2 + \tau_{//}^2\right)\right]} \le \frac{f_u}{(\beta_w \gamma_{M2})}$$

$$\sigma_{\perp} \le \frac{0.9 f_u}{\gamma_{M2}}$$

where:

$\sigma_{\perp}$ = a normal stress perpendicular to the plane of the weld

$\tau_{\perp}$ = a shear stress in the plane of the weld, perpendicular the weld length axis

$\tau_{//}$ = a shear stress in the plane of the weld, along the weld length axis

$\mathbf{f_u}$ = nominal ultimate tensile strength of the weaker part joined.

β_w = a correlation factor. See Code Table 4.1

γ_{M2} = 1.25

13.4.11.3.2 Header plate

13.4.11.3.2.1 Shear resistance

The capacity of a group of bolts is equal to $\mathbf{0.8nF_{v,Rd}}$

13.4.11.3.2.2 Bearing resistance

The resistance of a group of bolts is calculated according to the resistance capacity of the critical bolt, i. e. $\mathbf{nF_{b,Rd}}$

13.4.11.3.2.3 Shear - gross section

The forces transferred from the beam web creates shear forces along two planes. The factor of 1.27 represents the reduction of shear capacity due to the bending moment in the plane of the plate.

$$2F_{v,Rd} = \frac{2A_v f_y}{1.27\sqrt{3}\gamma_{M0}}$$

$$A_v = h_p t_p$$

where:

A_v = gross plate cross-section area

f_y = yield strength of the plate steel

h_p = plate height

t_p = plate thickness

γ_{Mo} = 1.0

13.4.11.3.2.4 Shear - net section

The forces transferred from the beam web creates shear forces along the section that passes through the bolt holes.

$$2F_{v,Rd} = \frac{2A_{v,net} f_u}{\sqrt{3}\gamma_{M2}}$$

$$A_{v,net} = t_p (h_p - n_1 d_0)$$

$A_{v,net}$ = the net area of the plate section that passes through the holes

f_u = tension strength of the plate

n_1 = no.of bolts per line

d_0 = bolt hole diameter

γ_{M2} = 1.25

13.4.11.3.2.5 Block tearing

In header plate connections the tension resistance must be reduced when 1.36 x distance between the bolt columns is greater than the height of the plate, i.e this case must be considered as an external load.

- **$h_p < 1.36p_{22}$**

$$V_{eff,2,Rd} = \frac{0.5 f_u A_{nt}}{\gamma_{M2}} + \frac{f_y A_{nv}}{\sqrt{3}\gamma_{M0}}$$

- **$h_p = 1.36p_{22}$**

$$V_{eff,1,Rd} = \frac{f_u A_{nt}}{\gamma_{M2}} + \frac{f_y A_{nv}}{\sqrt{3}\gamma_{M0}}$$

where:

- for 2 columns of bolts, **p_{22}** = distance between the columns
- for 4 columns of bolts, **p_{22}** = distance between the internal row of bolts and the external row furthest from it, i.e. the distance between two internal columns plus the distance between two bolt columns on the other side of the beam.

f_u = plate tensile strength

$\mathbf{f_y}$ = plate yield strength
$\mathbf{A_{nv}}$ = net area subject to shear
= $\mathbf{t_p (h_p - e_1 - (n_1 - 0.5) d_0)}$
$\mathbf{A_{nt}}$ = net area subject to tension for two bolt columns
= $\mathbf{t_p (e_2 - d_0/2)}$ for 2 columns of bolts
= $\mathbf{t_p (p_2 + e_2 - 3d_0/2)}$ for 4 columns of bolts
γ_{M2} = 1.25
γ_{M0} = 1.00

13.4.11.3.2.6 Plate bending

Bending in the plane of the plate is considered. The calculation is required only when the height of the plate is less than 1.36 x the distance between the rows of bolts.

$$F_{v,Rd} = \frac{2W_{el} f_y}{\frac{(p_{22} - t_w)}{2} \gamma_{M0}}$$

where:
- for 2 columns of bolts, $\mathbf{p_{22}}$ = distance between the columns
- for 4 columns of bolts, $\mathbf{p_{22}}$ = distance between the internal row of bolts and the external row furthest from it, i.e. the distance between two internal columns plus the distance between two bolt columns on the other side of the beam.

$\mathbf{t_w}$ = beam web thickness

$$W_{el} = \frac{t_p \cdot h_p^2}{6}$$

13.4.11.3.2.7 Web shear

Web shear calculations are carried out according to EN1993-1-1 (6.2.6)

$$V_{pl,Rd} = \frac{A_v \cdot f_y}{\sqrt{3}\gamma_{M0}}$$

$\mathbf{A_v}$ = section are subject to shear
= $\mathbf{h_p\ t_w}$
$\mathbf{f_y}$ = yield strength of the beam web steel
γ_{M0} = 1.00

13.4.11.3.2.8 Web tension

Web tension calculations are carried out according to EN1993-1-1 (6.2.3)

$$N_{pl,Rd} = \frac{A \cdot f_y}{\gamma_{M0}}$$

A = gross area subject to tension
$\mathbf{f_y}$ = yield strength of the beam web steel
γ_{M0} = 1.0

13.4.11.3.2.9 Weld capacity - shear

The shear force transferred from the beam web to the plate creates shear stresses in the plane of the weld along the axis of the weld, i.e. the only stress generated is $\tau_{//}$

$$\tau_{//} \le \frac{f_u}{(\beta_w \gamma_{M2})}$$

$$\tau_{//} = \frac{F_v}{2 \cdot a \cdot l}$$

where:

F_v = the shear force acting on the weld
a = the weld throat thickness
l = length of the weld

13.4.11.3.2.10 Weld capacity - tension

The tension force transferred from the beam web to the plate through the weld creates two different stresses:

- $\sigma_\perp$ = a normal stress perpendicular to the plane of the weld
- $\tau_\perp$ = a shear stress in the plane of the weld, perpendicular to the weld length axis.

The general equation is:

$$\sqrt{\left[\sigma_\perp^2 + 3\left(\tau_\perp^2\right)\right]} \le \frac{f_u}{(\beta_w \gamma_{M2})}$$

$$\sigma_\perp \le \frac{0.9 f_u}{\gamma_{M2}}$$

$$\sigma_\perp = \tau_\perp = \frac{F_t}{2 \cdot l \cdot a\sqrt{2}}$$

13.4.11.3.2.11 Dimensional limitations

End plate:

- Minimum end distance: between extreme bolt and end of plate: $e_1 = 1.2d_0$
- Minimum edge distance: between extreme bolt and edge of plate: $e_2 = 1.2d_0$
- Longitudinal bolt pitch: between bolts in direction of load: $p_1 = 2.2d_0$
- Bolt column spacing: between bolts perpendicular to load: $p_2 = 2.4d_0$
- Rotation capacity: $h_p = d_b$
- Plate height:

  $$h_p \ge \max\left(\frac{1900 V_{Ed}}{f_{y,b} t_{w,b}}; 0.6 h_b\right)$$

  V_{Ed} = maximum shear force in the connection
  h_b = height of the beam
- Bolt ductility:
  One of the following two conditions must be complied with:
  1. $$t_p \le \frac{d}{2.8}\sqrt{\frac{f_{ub}}{f_{yp}}}$$
  2. connection to:
     column flange: column web:

     $$t_{cf} \le \frac{d}{2.8}\sqrt{\frac{f_{ub}}{f_{yc}}} \qquad t_w \le \frac{d}{2.8}\sqrt{\frac{f_{ub}}{f_{yw}}}$$

     where:
     t_p = end plate thickness

t_{cf} = column flange thickness
t_w = column or beam web thickness
f_{ub} = ultimate tensile strength of the bolts.
f_{yp}, f_{yw}, f_{ycf} = yield strength of the end plate, column/beam web, and column flange, respectively.

- Weld ductility:
  Weld thickness calculated according to the following equation insures failure in the steel prior to failure in the weld:

$$a \geq \frac{f_y \beta_w \gamma_{M2}}{\sqrt{2} f_u \gamma_{M0}} t$$

where:

a = weld thickness
f_y = yield strength of the beam web
f_u = tensile strength of the welded part
β_w = factor for fillet welds
γ_{M0} = 1.0
γ_{M2} = 1.25

- Plate thickness:
  In order to ensure flexibility of the connection and to classify it as a simple connection, Reference [2] 1394 recommends a plate thickness of 8 or 10 mm. The minimum thickness is intended to prevent damage during fabrication or transport. The maximum thickness may be increased to 12 mm when tension forces act on the connection.

Column:

- minimum edge distance:
  the distance between the extreme bolt and the edge of the flange: **$e_2 = 1.2d_0$**
- Bolt column gauge:
  In order to ensure flexibility of the connection and to classify it as a simple connection, Reference [2] 1394 recommends the following spacing between bolt columns on both sides of the flange: **90 = g = 140** mm.

13.4.11.3.3 Fin plate

13.4.11.3.3.1 Plate - Weld capacity

The general equation is used to calculate the capacity using the shear and tension stresses as well as the stresses generated by the moment.

$$\sqrt{\left[\sigma_\perp^2 + 3\left(\tau_\perp^2 + \tau_{//}^2\right)\right]} \leq \frac{f_u}{\left(\beta_w \gamma_{M2}\right)}$$

$$\sigma_\perp = \tau_\perp = \frac{F_t}{2 \cdot l \cdot a\sqrt{2}} + \frac{F_v z \cdot c}{2 I_p a\sqrt{2}}$$

$$I_p = \frac{l^3}{12}$$

$$c = \frac{l}{2}$$

$$\tau_{//} = \frac{F_v}{2 \cdot a \cdot l}$$

where:

$\sigma_{\perp}$	=	a normal stress perpendicular to the plane of the weld
$\tau_{\perp}$	=	a shear stress in the plane of the weld, perpendicular the weld length axis
$\tau_{//}$	=	a shear stress in the plane of the weld, along the weld length axis
f_u	=	nominal ultimate tensile strength of the weaker part joined.
β_w	=	a correlation factor. See Code Table 4.1
γ_{M2}	=	1.25
F_v	=	shear force acting on the weld
F_t	=	tension force acting on the web
a	=	weld throat thickness
l	=	length of the weld
c	=	the distance between the center-of-gravity of the weld to the further point.

13.4.11.3.3.2 Plate - buckling

The plate buckling capacity must be checked when **z > t_p/0.15**. The calculation is based on Reference [2] 1394 and British Code BS5950-1:2000.

$$F_{v,Rd} = \frac{W_{el} f_{pLT}}{z \cdot 0.6 \cdot \gamma_{M1}} \le \frac{W_{el} f_y}{z \cdot \gamma_{M0}}$$

where:

z	=	the distance between the bolt column closest to the support and the face of the support.
t_p	=	thickness of the plate
γ_{M1}	=	1.00
f_y	=	yield strength of the plate
f_{pLT}	=	the bending strength of the plate, including the reduction for buckling calculated according to BS5950-1:2000:

$$f_{pLT} = \frac{f_e \cdot f_y}{\varphi_{LT} + \left(\varphi_{LT}^2 - f_e \cdot f_y\right)^{0.5}}$$

$$f_e = \frac{\pi^2 E}{\lambda_{LT}^2}$$

$$\varphi_{LT} = \frac{f_y + (\eta_{LT} + 1) f_e}{2}$$

$$\lambda_{LT} = 2.8 \left(\frac{z \cdot h}{1.5 \cdot t_p^2}\right)^{1/2}$$

$$\lambda_{L0} = 0.4 \left(\frac{\pi^2 E}{f_y}\right)^{1/2}$$

- $\lambda_{LT} \le \lambda_{L0} \rightarrow \eta_{LT} = 0$
- $\lambda_{L0} < \lambda_{LT} < 2\lambda_{L0} \rightarrow \eta_{LT} = 14\left(\lambda_{LT} - \lambda_{L0}\right)/1000$
- $2\lambda_{L0} \le \lambda_{LT} \le 3\lambda_{L0} \rightarrow \eta_{LT} = 14\left(\lambda_{L0}\right)/1000$
- $\lambda_{LT} > 3\lambda_{L0} \rightarrow \eta_{LT} = 7\left(\lambda_{LT} - \lambda_{L0}\right)/1000$

13.4.11.3.3.3 Plate - bending

The calculation of the plate bending capacity is required only when:

$$h_p < 2.73z \rightarrow F_{v,Rd} = \frac{W_{el} f_y}{z \cdot \gamma_{M0}}$$

where:

z = distance between the center-of-gravity of the bolts and the face of the support

W_{el} = elastic section modulus of the plate

= $$W_{el} = \frac{t_p \cdot h_p^2}{6}$$

13.4.11.3.3.4 Plate - Block tearing

$$V_{eff,2,Rd} = \frac{0.5 f_u A_{nt}}{\gamma_{M2}} + \frac{f_y A_{nv}}{\sqrt{3}\gamma_{M0}}$$

where:

f_u = tensile strength of the plate steel

f_y = yield strength of the plate steel

A_{nv} = net area subject to shear

= **$t_p (h_p - e_1 - (n_1 - 0.5) d_0)$**

A_{nt} = net area subject to tension:

1 column of bolts:	2 columns of bolts:
$t_p (e_2 - d_o/2)$	**$t_p (p_2 + e_2 - 3d_o/2)$**

γ_{M0} = 1.00

γ_{M2} = 1.25

13.4.11.3.3.5 Plate - Shear - net section

The force transferred from the beam web creates shear stresses along the section that passes through the bolt holes.

$$F_{v,Rd} = A_{v,net} f_u \Big/ \sqrt{3}\gamma_{M2}$$

$$A_{v,net} = t_p (h_p - n_1 d_0)$$

where:

$A_{v,net}$ = net area of the steel section that passes through a column of bolts

f_u = tensile strength of the plate steel

n_1 = number of bolts in one column

d_0 = diameter of the bolt hole

γ_{M2} = 1.25

13.4.11.3.3.6 Plate - Shear - gross section

The force transferred from the web of the beam to the plate generates shear forces.

$$F_{v,Rd} = A_v f_y \Big/ 1.27\sqrt{3}\gamma_{M0}$$

$$A_v = h_p t_p$$

where:

the factor of 1.27 is a modification for the stresses due to bending

A_v = gross area of the plate section

f_y	=	yield strength of the plate steel
h_p	=	height of the plate
t_p	=	plate thickness
γ_{M0}	=	1.0

13.4.11.3.3.7 Plate - Bearing

$$\frac{F_s}{F_{b,Rd}} \leq 1.0$$

This equation for two columns of bolts considers the two vertical components for the bolt bearing as well as the two horizontal components, the vertical components are influenced by the shear force and the vertical component of the moment, the horizontal components are influence by the horizontal tension force and the horizontal component of the moment.

The capacity of a single bolt is calculated as : $\mathbf{F_{b,Rd} = min(F_{b,ver,Rd}\ ,\ F_{b,hor,Rd})}$

For a single column of bolts:

$$F_s = \sqrt{F_{sv}^2 + (F_{sm} + F_{st})^2}$$

$$F_{sv} = \frac{F_v}{n}$$

$$F_{sm} = \frac{F_v z}{W_{el}} = \frac{6F_v z}{n(n+1)p}$$

$$F_{st} = \frac{F_t}{n}$$

For two columns of bolts:

$$F_s = \sqrt{(F_{sv} + F_{smv})^2 + (F_{smh} + F_{st})^2}$$

$$F_{sv} = \frac{F_v}{n}$$

$$F_{smv} = \frac{F_v z \cdot x}{I_{bg}} = \frac{F_v z \cdot p_2}{2I_{bg}}$$

$$F_{smh} = \frac{F_v z \cdot y}{I_{bg}} = \frac{F_v z \cdot (n_1 - 1)p_1}{2I_{bg}}$$

$$F_{st} = \frac{F_t}{n}$$

$$I_{bg} = \sum s^2 = \frac{n_1}{2}p_2^2 + \frac{1}{6}n_1(n_1^2 - 1)p_1^2$$

where:

F_s	=	the equivalent force acting on the extreme bolt
z	=	the distance from the center-of-gravity of the bolts to the face of the support (edge of plate)
F_{sv}	=	shear force acting on the bolt due to the shear force
F_{sm}	=	shear force acting on the bolt due to the moment
F_{st}	=	shear force acting on the bolt due to the tension force
s	=	the distance from the center-of-gravity of the bolts to the bolt
x		the horizontal distance from the center-of-gravity of the bolts to the extreme bolt
y		the vertical distance from the center-of-gravity of the bolts to the extreme bolt

The vertical bearing capacity of a single bolt is calculated as:

$$F_{b,ver,Rd} = \frac{k_1 \alpha_b f_u \cdot d \cdot t}{\gamma_{M2}}$$

where:

$$\alpha_b = \min\left(\frac{e_1}{3d_0};\frac{p_1}{3d_0}-0.25;\frac{f_{ub}}{f_u};1.0\right)$$

$$k_1 = \min\left(2.8\frac{e_2}{d_0}-1.7;1.4\frac{p_2}{d_0}-1.7;2.5\right)$$

The horizontal bearing capacity of a single bolt is calculated as:

$$F_{b,hor,Rd} = \frac{k_1 \alpha_b f_u \cdot d \cdot t}{\gamma_{M2}}$$

where:

$$\alpha_b = \min\left(\frac{e_2}{3d_0};\frac{p_2}{3d_0}-0.25;\frac{f_{ub}}{f_u};1.0\right)$$

$$k_1 = \min\left(2.8\frac{e_1}{d_0}-1.7;1.4\frac{p_1}{d_0}-1.7;2.5\right)$$

$\mathbf{k_1}$	=	a factor that considers the distance from the bolt to the edge (perpendicular to the direction of the load). For a bolt that is far from the edge $\mathbf{k_1}$ = 2.5
α_b	=	a factor that considers the distance from the bolt to the edge, the bolt spacing in the direction of the load and the possibility of failure in bearing of the bolt.
d	=	bolt diameter
$\mathbf{d_0}$	=	bolt hole diameter
t	=	thickness of the steel with the hole for which the bearing capacity is calculated
γ_{M2}	=	1.25
$\mathbf{f_{ub}}$	=	tensile strength of the bolt
$\mathbf{f_u}$	=	tensile strength of the steel
$\mathbf{e_1}$	=	the distance to the edge in the direction of the load
$\mathbf{e_2}$	=	the distance to the edge perpendicular to the direction of the load
$\mathbf{p_1}$	=	the distance between bolts in the direction of the load (vertical distance between bolt rows).
$\mathbf{p_2}$	=	the distance between bolts perpendicular to the direction of the load (horizontal distance between bolt columns).
$\mathbf{n_1}$	=	number of rows of bolts
n	=	the total number of bolts

13.4.11.3.3.8 Plate - Bolt shear

An equivalent force must be calculated when both a tension force and a shear force act on a connection. This equivalent has a vertical component from the shear force and two horizontal components, one from the tension force and the other from the moment that results from the eccentricity. There are two vertical components in connections with two columns of bolts.

$$\frac{F_s}{F_{v,Rd}} \leq 1.0$$

for a single column of bolts:

$$F_s = \sqrt{F_{sv}^2 + (F_{sm} + F_{st})^2}$$
$$F_{sv} = \frac{F_v}{n}$$
$$F_{sm} = \frac{F_v z}{W_{el}} = \frac{6F_v z}{n(n+1)p}$$
$$F_{st} = \frac{F_t}{n}$$

for two columns of bolts:

$$F_s = \sqrt{(F_{sv} + F_{smv})^2 + (F_{smh} + F_{st})^2}$$
$$F_{sv} = \frac{F_v}{n}$$
$$F_{smv} = \frac{F_v z \cdot x}{I_{bg}} = \frac{F_v z \cdot p_2}{2I_{bg}}$$
$$F_{smh} = \frac{F_v z \cdot y}{I_{bg}} = \frac{F_v z \cdot (n_1 - 1)p_1}{2I_{bg}}$$
$$F_{st} = \frac{F_t}{n}$$
$$I_{bg} = \sum s^2 = \frac{n_1}{2} p_2^2 + \frac{1}{6} n_1 (n_1^2 - 1) p_1^2$$

where:

F_s	=	the effective force acting on the extreme bolt
z	=	the distance between the center-of-gravity of the bolts and the face of the support (edge of the plate)
F_{sv}	=	the shear force applied to the bolt by the shear force
F_{sm}	=	the shear force applied to the extreme bolt by the moment
F_{st}	=	the tension force acting on the bolt
s	=	the distance from the center-of-gravity of the bolts to the bolt
x	=	the horizontal distance from the center-of-gravity of the bolts to the extreme bolt
y	=	the vertical distance from the center-of-gravity of the bolts to the extreme bolt

13.4.11.3.3.9 Beam - Tension - net

The tension capacity of the beam is calculated according to EC3 - Part 1 - Section 6.2.3:

$$N_{u,Rd} = \frac{0.9 A_{net} f_u}{\gamma_{M2}}$$
$$A_{net} = t_w h - d_0 n_1 t_w$$

The capacity is reduced when the beam is notched.

13.4.11.3.3.10 Beam - Block tearing

Refer to General - block tearing 1396.

13.4.11.3.3.11 Beam - Web shear - net

The net web shear capacity is calculated according to the steel code provisions:

$$F_{v,Rd} = \frac{A_{v,net} f_u}{\sqrt{3}\gamma_{M2}}$$
$$A_{v,net} = A_v - n_1 d_0 t_w$$

where:

$A_{v,net}$ = net shear area
A_v = gross shear area
f_u = tensile strength of the web steel
t_w = beam web thickness
n = no. of bolt rows
d_0 = bolt hole diameter
γ_{M2} = 1.25

13.4.11.3.3.12 Beam - Web shear - gross section

The overall web shear capacity is calculated in the Steel Postprocessor according to the steel code provisions; this program calculates the reduction due to the notch.

$$F_{v,Rd} = \frac{A_v f_y}{\sqrt{3}\gamma_{M0}}$$

$$A_v = A - 2bt_f + (t_w + 2r)t_f$$

where:

A_v = $(h - d_{c1} - d_{c2})\ t_w$
f_y = yield strength of the web steel
b = flange width
t_f = flange thickness
t_w = web thickness
γ_{M0} = 1.00
d_{c1} , d_{c2} = upper/lower notch height
h = total beam height
r = fillet radius at flange/web connection

13.4.11.3.3.13 Beam - Bearing

The capacity of a group of bolts is calculated according to:

$$\frac{1}{\sqrt{\left(\frac{\frac{1}{n}+\alpha}{F_{b,ver,Rd}}\right)^2+\left(\frac{\beta}{F_{b,hor,Rd}}\right)^2}}$$

- for one column of bolts:

$$\alpha = 0 .... \beta = \frac{6z}{p_1 n(n+1)}$$

- for 2 columns of bolts:

$$\alpha = \frac{z}{I}\frac{p_2}{2} .... \beta = \frac{z}{I}\frac{(n_1-1)}{2}p_1$$

$$I = \frac{n_1}{2}p_2^2 + \frac{1}{6}n_1(n_1^2-1)p_1^2$$

The vertical bearing capacity of a single bolt is calculated according to:

$$F_{b,ver,Rd} = \frac{k_1 \alpha_b f_u \cdot d \cdot t}{\gamma_{M2}}$$

where:

$$\alpha_b = \min\left(\frac{p_1}{3d_0} - 0.25; \frac{f_{ub}}{f_u}; 1.0\right)$$

and in a beam with a notch: $$\alpha_b = \min\left(\frac{e_1}{3d_0};\frac{p_1}{3d_0} - 0.25;\frac{f_{ub}}{f_u};1.0\right)$$

$$k_1 = \min\left(2.8\frac{e_2}{d_0} - 1.7;1.4\frac{p_2}{d_0} - 1.7;2.5\right)$$

The horizontal bearing capacity of a single bolt is calculated according to:

$$F_{b,hor,Rd} = \frac{k_1 \alpha_b f_u \cdot d \cdot t}{\gamma_{M2}}$$

where:

$$\alpha_b = \min\left(\frac{e_2}{3d_0};\frac{p_2}{3d_0} - 0.25;\frac{f_{ub}}{f_u};1.0\right)$$

$$k_1 = \min\left(1.4\frac{p_1}{d_0} - 1.7;2.5\right)$$

and in a beam with a notch: $$k_1 = \min\left(2.8\frac{e_1}{d_0} - 1.7;1.4\frac{p_1}{d_0} - 1.7;2.5\right)$$

k_1	=	a factor that considers the distance from the bolt to the edge (perpendicular to the direction of the load).
α_b	=	a factor that considers the distance from the bolt to the edge, the bolt spacing in the direction of the load and the possibility of failure in bearing of the bolt.
d	=	bolt diameter
d_0	=	bolt hole diameter
t	=	thickness of the steel with the hole for which the bearing capacity is calculated
γ_{M2}	=	1.25
f_{ub}	=	tensile strength of the bolt
f_u	=	tensile strength of the steel
e_1	=	the distance to the edge in the direction of the load
e_2	=	the distance to the edge perpendicular to the direction of the load
p_1	=	the distance between bolts in the direction of the load (vertical distance between bolt rows).
p_2	=	the distance between bolts perpendicular to the direction of the load (horizontal distance between bolt columns).
n_1	=	number of rows of bolts
n	=	the total number of bolts

13.4.11.3.3.14 Dimensional limitations

Plate:

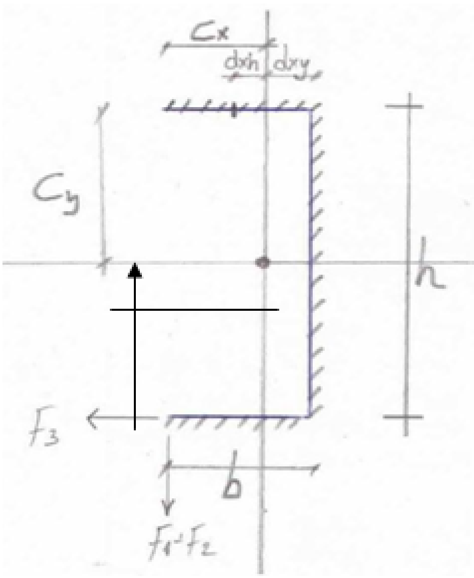
• Minimum end distance:	between extreme bolt and end of plate:	e_1	=	$1.2d_0$
• Minimum edge distance:	between extreme bolt and edge of plate:	e_2	=	$1.2d_0$
• Longitudinal bolt pitch:	between bolts in direction of load:	p_1	=	$2.2d_0$
• Bolt column spacing:	between bolts perpendicular to load:	p_2	=	$2.4d_0$
• Rotation capacity:		h_p	=	d_b
• Plate height:				

13.4.11.3.4 Web cleat - bolted

Supporting side - refer to Header plate 1396.
Supported side - refer to Fin plate 1400.

13.4.11.3.5 Web cleat - welded

13.4.11.3.5.1 Weld capacity - beam



$$L = 2b + h$$

$$\sum I_x = \frac{h^3}{12} + 2b\left(\frac{h}{2}\right)^2$$

$$\sum I_y = h(d_{xy})^2 + 2\left[\frac{b^3}{12} + b(d_{xh})^2\right]$$

$$I_p = \sum I_x + \sum I_y$$

$$\sigma_\perp = \tau_\perp = \frac{F_v}{2L\sqrt{2}a} + \frac{F_v e c_x}{2I_p\sqrt{2}a}$$

$$\tau_{//} = \frac{F_v e c_y}{2I_p a}$$

$$\sqrt{\left[\sigma_\perp^2 + 3\left(\tau_\perp^2 + \tau_{//}^2\right)\right]} \le \frac{f_u}{(\beta_w \gamma_{M2})}$$

$$\sigma_\perp \le \frac{0.9 f_u}{\gamma_{M2}}$$

where:

$\sigma_\perp$	=	normal stresses perpendicular to the axis of the throat of the weld
$\tau_\perp$	=	shear stresses in the plane of the throat of the weld, perpendicular to the weld axis.
$\tau_{//}$	=	shear stresses in the plane of the throat of the weld, along the weld axis.
$\mathbf{f_u}$	=	ultimate tensile strength of the weaker part joined
β_w	=	correlation factor from Table 4.1
γ_{M2}	=	1.25

13.4.11.3.5.2 Shear - angle - beam side

The force transferred from the beam web to the angle creates shear stresses. The factor of 1.27 takes into account the stresses from the bending moment.

$$2F_{v,Rd} = {2A_v f_y}\Big/{1.27\sqrt{3}\gamma_{M0}}$$

$$A_v = h_p t_p$$

where:

A_v	=	gross area of the angle section
f_y	=	yield strength of the angle steel
h_p	=	angle height
t_p	=	angle thickness
γ_{M0}	=	1.00

13.4.11.3.5.3 Shear - web

The shear capacity of the beam is calculated according to the Code and is also displayed in the Steel Postprocessor results. The calculation here includes the reduction in the capacity due to the notch.

$$F_{v,Rd} = {A_v f_y}\Big/{\sqrt{3}\gamma_{M0}}$$

$$A_v = A - 2bt_f + (t_w + 2r)t_f$$

for a beam with a notch: $\mathbf{A_v = (h - d_{c1} - d_{c2})t_w}$

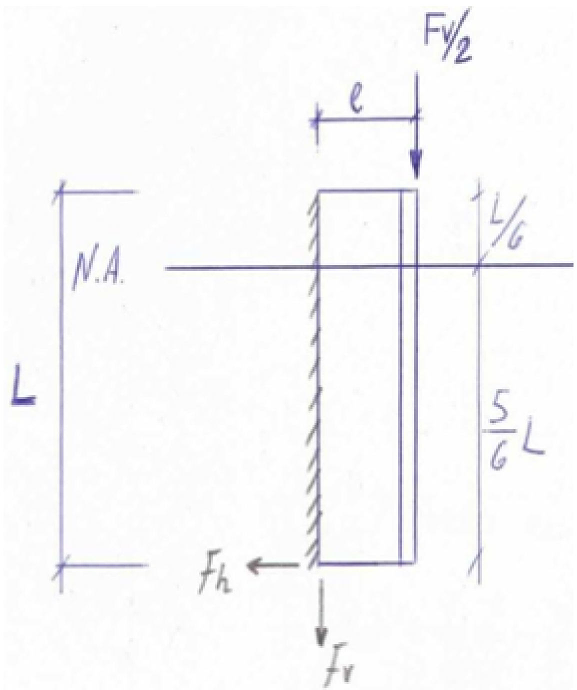
where:

A_v	=	shear area
f_y	=	yield strength of the beam steel
b	=	beam flange width
t_f	=	beam flange thickness
tw	=	beam web thickness
γ_{M0}	=	1.00
d_{c1},d_{c2}	=	upper/lower notch height
h	=	total beam height
r	=	fillet radius

13.4.11.3.5.4 Weld - column

In general, the elastic analysis of weld stresses assumes that the center of rotation is the center-of-gravity. In certain cases this assumption is incorrect and a different center of rotation must be assumed.

In this example it is assumed that the weld is entirely vertical along the length of the angle. The applied load creates torsion in the weld so that the upper portion of the angles are pushed into the beam web while the lower portion is pulled away from the web. This torsion moment is resisted by a small compression area in the upper portion and by shear in the weld that increases towards the bottom. The calculation assumes that the neutral axis is located at L/6 from the top of the angle.



The horizontal force acting on the weld:

$$f_h = \frac{9F_v e}{5L^2}$$

where:

L = length of the weld or length of the angle
e = load eccentricity = angle leg dimension
$\mathbf{F_v}$ = transferred shear force

The horizontal load is divided into two components:

$$\sigma_\perp = \tau_\perp = \frac{f_h}{\sqrt{2}a}$$

The vertical load creates shear stresses along the axis of the weld:

$$\tau_{\parallel} = \frac{F_v}{2La}$$

The calculated stresses are used in the following equation:

$$\sqrt{\left[\sigma_\perp^2 + 3\left(\tau_\perp^2 + \tau_{//}^2\right)\right]} \le \frac{f_u}{(\beta_w \gamma_{M2})}$$

$$\sigma_\perp \le \frac{0.9 f_u}{\gamma_{M2}}$$

where:

σ_1 = normal stresses perpendicular to the axis of the throat of the weld
$\tau_\perp$ = shear stresses in the plane of the throat of the weld, perpendicular to the weld axis.
$\tau_{//}$ = shear stresses in the plane of the throat of the weld, along the weld axis.
$\mathbf{f_u}$ = ultimate tensile strength of the weaker part joined
β_w = correlation factor from Table 4.1
γ_{M2} = 1.25

13.4.11.3.5.5 Shear - angle - column

The force transferred from the beam web to the angle creates shear stresses. The factor of 1.27 takes into account the stresses from the bending moment.

$$2F_{v,Rd} = \frac{2A_v f_y}{1.27\sqrt{3}\gamma_{M0}}$$

$$A_v = h_p t_p$$

where:

A_v	=	gross area of the angle section
f_y	=	yield strength of the angle steel
h_p	=	angle height
t_p	=	angle thickness
γ_{M0}	=	1.00

13.4.11.3.6 Extended end plate

13.4.11.3.6.1 Moment resistance

The beam is connected to the flange of the column by means of a plate welded to the beam and bolted to the column. The calculation is based on the "Basic component" method; the capacity of each component of the connection is calculated - bolts, plates, flanges, webs, etc - and the component with the lowest capacity determines the capacity of the entire connection.

Each row of bolts is checked separately and as part of a group of bolts, once on the beam side and once on the column side. An effective length of the "T-stub" is calculated for each side that represents the failure mode of the column flange or of the end plate.

The moment capacity of a beam connected to a column flange:

$$M_{tr,Rd} = \Sigma h_r F_{tr,Rd}$$

where:

- $F_{tr,Rd}$ is the effective tension capacity of row **r**, calculated separately for each row and as part of a group of rows with the rows above. The program uses the smallest value.
- h_r = the distance from row **r** to the center of the compression flange.

The effective tension capacity of a single row of bolts is calculated as:

$$F_{tr,Rd} = \min(F_{t,wc,Rd}; F_{t,fc,Rd}; F_{t,ep,Rd}; F_{t,wb,Rd})$$

The effective tension capacity of a single row of bolts that is part of a group of bolts is calculated as:

$$F_{tr,Rd} = \min(F_{t,wc,Rd}; F_{t,fc,Rd}; F_{t,ep,Rd}; F_{t,wb,Rd}) - F_{tr-1,Rd}$$

where:

- $F_{tr-1,Rd}$ is the effective tension capacity of the previous row of bolts in the group.

When checking a group of bolts the capacity of the group is reduced by the capacity of the rows above that group.

In certain cases the effective capacity of a bolt row must be reduced in order to comply with the following conditions, the reduction begins with the row closest to the compression flange:

$$\sum F_{r,Rd} \leq V_{wp,Rd} / \beta$$

$$\sum F_{r,Rd} \leq \min(F_{c,wc,Rd}; F_{c,fb,Rd})$$

where:

$V_{wp,Rd}$ = plastic shear capacity of the column web

$F_{c,wc,Rd}$ = compression capacity of the column web

$R_{c,fb,Rd}$ = compression capacity of the beam web and flange

β = correlation factor for the influence of the shear force on the column web, calculated separately for each side.

$$\beta_1 = \left|1 - M_{j2} / M_{j1}\right| \leq 2.0$$

$$\beta_2 = \left|1 - M_{j1} / M_{j2}\right| \leq 2.0$$

β_1 = the β factor for the right side of the connection to the column

β_2 = the β factor for the left side of the connection to the column

M_{j1} = the moment of the beam connected to the right of the column

M_{j2} = the moment of the beam connected to the left of the column

The reduction faction in Table 6.3 is calculated according to the value of β.

The program adds a web stiffener to the column (as an extension to the beam compression flange) whenever there is a reduction due to the strength limits of the column web in compression.

The program adds web stiffeners to the column (as extensions to the beam compression and tension flanges) whenever there is a reduction due to the strength limits of the column web in shear.

Whenever the capacity of one of the bolt rows is greater than 1.9 times the tension capacity of a single bolt, i.e. $F_{tr,Rd} > 1.9F_{t,Rd}$, then the capacity of the row below must be reduced so that

$$F_{tr,Rd} \leq F_{tx,Rd} \frac{h_r}{h_x}$$

$F_{tx,Rd}$ = effective tension capacity of the previous row.

h_x = distance from row X to the center of the compression flange.

x = the row furthest away from the center of the compression flange which meets the condition $F_{tr,Rd} > 1.9F_{t,Rd}$

13.4.11.3.6.2 Column web - tension

The tension capacity of the column web:

$$F_{t,wc,Rd} = \frac{\omega\, b_{eff,t,wc}\, t_{wc}\, f_{y,wc}}{\gamma_{M0}}$$

where:

ω = a reduction factor that considers the interaction with the shear in the column web, calculated according to Table 6.3 in the Code.

$b_{eff,t,wc}$ = the effective tension width of the column web, equal to the effective length of the T-stub calculated for the column flange.

13.4.11.3.6.3 Beam web - tension

The tension capacity of the beam web:

$$F_{t,wb,Rd} = \frac{b_{eff,t,wb}\, t_{wb}\, f_{y,wb}}{\gamma_{M0}}$$

where:

$b_{eff,t,wb}$ = the effective tension width of the beam web, equal to the effective length of the T-stub calculated for the end plate in bending.

13.4.11.3.6.4 Column web - compression

The compression capacity of an unstiffened column web:

$$F_{c,wc,Rd} = Min(\frac{\omega k_{wc} b_{eff,c,wc} t_{wc} f_{y,wc}}{\gamma_{M0}}, \frac{\omega k_{wc} \rho b_{eff,c,wc} t_{wc} f_{y,wc}}{\gamma_{M1}})$$

where:

ω = a reduction factor that considers the interaction with the shear in the column web, calculated according to Table 6.3 in the Code.

$b_{eff,c,wc}$ = **$t_{fb} + 2v2a_p + 5(t_{fc} + s) + s_p$**

t_{fb} = beam web thickness

a_p throat thickness of the weld that connects the beam flange to the plate

t_{fc} = column flange thickness

s = **r_c**

= fillet radius of the column (web-flange)

s_p = **$2t_p$**

t_p = plate thickness

ρ = reduction factor for plate buckling:

$$\text{if } \overline{\lambda_p} \le 0.72 \quad \rightarrow \quad \rho = 1.0$$

$$\text{if } \overline{\lambda_p} > 0.72 \quad \rightarrow \quad \rho = (\overline{\lambda_p} - 0.2) / \overline{\lambda_p}^2$$

$$\overline{\lambda_p} = 0.932\sqrt{\frac{b_{eff,c,wc} d_{wc} f_{y,wc}}{E t_{wc}^2}}$$

E = 210000 mPa

d_{wc} = **$h_c - 2(t_{fc} + r_c)$** = flat portion of the column web

k_{wc} = a reduction factor that reduces the column web capacity in compression when the longitudinal compression stress in the column resulting from the compression force and the moment are greater than 70% of the yield strength of the column web.

$$\text{if } \sigma_{com,Ed} \le 0.7 f_{y,wc} \rightarrow \quad k_{wc} = 1.0$$

$$\text{if } \sigma_{com,Ed} > 0.7 f_{y,wc} \rightarrow \quad k_{wc} = 1.7 - \sigma_{com,Ed} / f_{y,wc}$$

$\sigma_{com,Ed}$ = the longitudinal compression stress in the column resulting from the compression force and the moment calculated at the start of the fillet.

$$\sigma_{com,Ed} = \frac{P}{A} + \frac{My}{I}$$

P = column axial force; compression = negative

A = column section area

M = column bending moment

y = **$d_{wc}/2$** = distance from the center-of-gravity of the column to the start of the fillet.

I = moment-of-inertia of the column about the axis of bending

When the compression or buckling capacity of the column web is less than the sum of the forces in the bolt rows it is necessary to carry out a reduction in the bolt rows, starting with the row closest to the compression flange. This reduction
may be avoided by stiffening the column web at the level of the beam compression flange. The contribution of the stiffeners to the column web capacity is as follows:

- column web compression capacity:
  the addition to the capacity is **$2b_n t_s f_y/\gamma_{Mo}$**
  where:

b_n = contact area between the stiffener and the column flange
t_s = stiffener thickness

- stiffener bucking capacity:

$$F_{c,wc,Rd} = \frac{\chi F_y A_g}{\gamma_{M1}}$$

$$\chi = \frac{1}{\phi + \sqrt{\phi^2 - \bar{\lambda}^2}} \leq 1.0$$

$$\phi = 0.5\left[1 + 0.49(\bar{\lambda} - 0.2) + \bar{\lambda}^2\right]$$

$$\bar{\lambda} = \frac{L_{cr}}{i_y}\frac{1}{93.9\varepsilon}$$

$$i_y = \sqrt{\frac{I_y}{A_g}}$$

The gross area of the section for buckling is equal to the stiffener section area plus the web area on both sides of the stiffener with a length of **$15\varepsilon t_w$**.

$\varepsilon = v(235/F_y)$

The gross area = **$A_g = (30\varepsilon t_w + t_s)t_w + 2b_g t_s$**

The calculation is carried out according to EN1993-1-1, Section 6.3.

The buckling length is equal to the distance between the column flanges:

$L_{cr} = h_c - 2t_{fc}$

The moment-of-inertia about the vertical axis of a cross-shaped section.

13.4.11.3.6.5 Beam flange - comp

The program calculates the capacity conservatively as follows:

$F_{c,fb,Rd} = 1.2t_{fb}b_{fb}f_{y,fb}$

where:

t_{fb} = beam flange thickness, or the haunch flange thickness when a haunch is attached.
b_{fb} = compression flange width, or the haunch flange width when a haunch is attached.

13.4.11.3.6.6 Column w eb panel

Column web capacity in shear:

$$V_{Wp,Rd} = \frac{0.9 A_{vc} f_{y,wc}}{\sqrt{3}\gamma_{M0}}$$

$$A_{vc} = A - 2bt_f + (t_w + 2r)t_f$$

where:

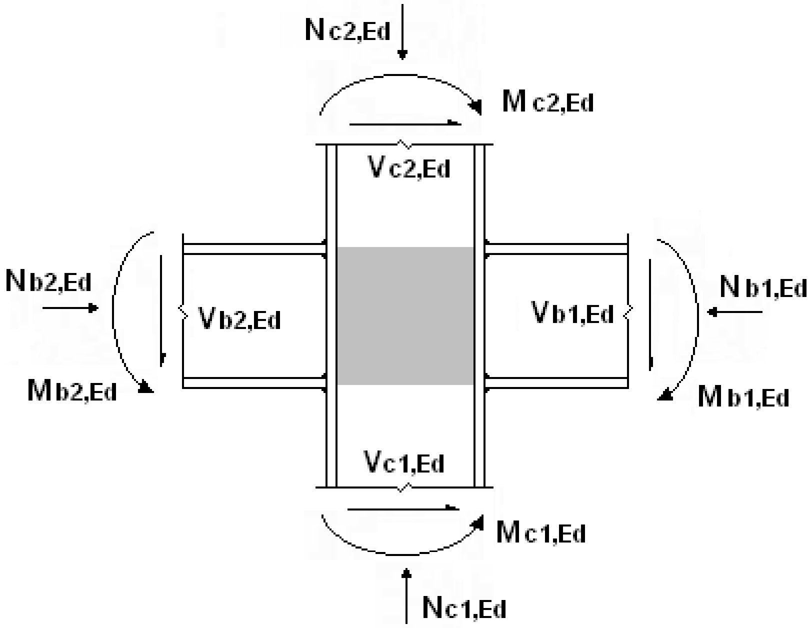
A_{vc} = shear area
$f_{y,wc}$ = column web yield strength
b = column flange width
t_f = column flange thickness
t_w = column web thickness
γ_{Mo} = 1.00
r = fillet radius web/flange

The shear force in the column is calculated as follows:

$$V_{Wp,Ed} = (M_{b1,Ed} - M_{b2,Ed})/z - (V_{c1,Ed} - V_{c2,Ed})/2$$

where:

z = the distance from the center of the compression flange to the mid-point between the two upper bolts in tension. When there is only one row in tension then Z is the distance to this row.



When the column web shear capacity determines the connection capacity web stiffeners may be added at the levels of the beam flanges. The additional capacity is calculated as:

$$V_{wp,add,Rd} = \min\left(\frac{4M_{pl,fc,Rd}}{d_s}, \frac{2M_{pl,fc,Rd} + 2M_{pl,st,Rd}}{d_s}\right)$$

where:

$\mathbf{d_s}$ = distance between the stiffener centers

$\mathbf{b_s}$ = width of the stiffener on each side of the web

$$M_{pl,fc,Rd} = \frac{W_{pl} f_y}{\gamma_{M0}} = \frac{b_{fc} t_{fc}^2 f_y}{4\gamma_{M0}}$$ - plastic moment of the column flange

$$M_{pl,st,Rd} = \frac{W_{pl} f_y}{\gamma_{M0}} = \frac{2 b_g t_s^2 f_y}{4\gamma_{M0}}$$ - plastic moment of the stiffener

13.4.11.3.6.7 Bolt in shear

The calculation of bolts in shear and tension should be done according to the equations in Table 3.4. As a simplification the program assumes that some of the bolts work entirely in tension and are stressed to the tension capacity. The remaining bolts carry the shear along with 28% of the tension bolts:

$$V_{Ed} \le n_c F_{v,Rd} + n_t \frac{0.4}{1.4} F_{v,Rd}$$

where:

$\mathbf{F_{v,Rd}}$ = capacity of a single bolt in shear, according to Table 3.4

$\mathbf{n_c}$ = no. of bolts not required for tension

$\mathbf{n_t}$ = no.of bolts required for tension

13.4.11.3.6.8 Bearing - column

The program assumes that some of the bolts work entirely in tension and are stressed to the tension capacity. The remaining bolts carry the bearing along with 28% of the tension bolts:

$$V_{Ed} \le n_c F_{b,Rd} + n_t \frac{0.4}{1.4} F_{b,Rd}$$

13.4.11.3.6.9 Web weld

When calculating the weld size required it is usually possible to design a weld with a capacity less than the capacity of the welded part, a situation that can lead to brittle failure. Therefore the weld is designed to be equal or greater than the capacity of the welded part in order to prevent this situation, i.e. the welds are designed as "full strength".

The weld size required is:

$$a = \frac{f_y \beta_w \gamma_{M2}}{\sqrt{2} f_u \gamma_{M0}} t$$

where:

a	=	weld throat dimension.
f_y	=	yield strength of the weakest of the welded parts
f_u	=	tensile strength of the weakest of the welded parts
t	=	thickness of the welded part

13.4.11.3.7 Base plate

13.4.11.3.7.1 Compression resistance

At the interface between the steel and the concrete, the "T-stub in compression" model (section 6.2.5 in the Code) is used for the two basic design checks:

- bending in the base plate due to the stresses in the foundation
- bearing on the concrete under the base plate

The base plate compression capacity = $N_{j,Rd} = f_{jd} A_{eff}$

The bearing capacity of the concrete = $f_{jd} = \beta_j f_{cd} \alpha$

where:

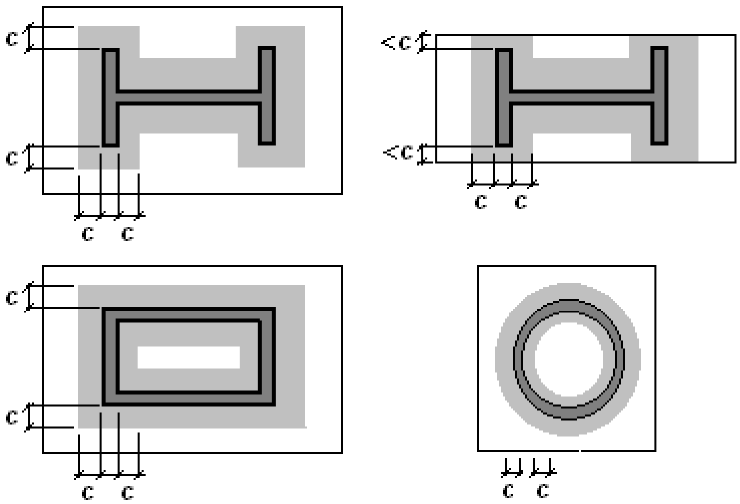
β_j	=	base material factor = 2/3
f_{cd}	=	f_{ck}/γ_d = concrete design strength
γ_d	=	1.5
f_{ck}	=	nominal concrete strength. Refer to EN1992-1-1 - Table 3.1
α	=	a factor that allows the concrete bearing capacity to be increased when the distance from the edge of the plate to the edge of the concrete base and the depth of the base allow for a better distribution of the stresses.
	=	$v(A_{c1}/A_{c0})$

The effective bearing area is a function of several factors - the size of the plate, the plate thickness, and the projection of the plate beyond the column.

The additional bearing width:

$$c = t_p\sqrt{\frac{f_{yp}}{3f_{jd}\gamma_{M0}}}$$

is used to calculate the effective bearing area. The effective area consists of three segments - two under the flanges and one under the web. For example:



13.4.11.3.7.2 Anchor bolt

The anchor bolt capacity is calculated as the minimum of:

- bond strength
- pull-out strength
- bolt tension strength
- cone strength

$\mathbf{F_{t,anchor,Rd} = min[N_{Rd,bond} , N_{Rd,s} , N_{Rd,p} , N_{Rd,c}/n]}$

Bond failure - $N_{Rd,bond}$:

The code does not explain the method for calculating the anchorage capacity of the bolts. In the case of anchorage by bond strength the Code refers to the calculation for deformed bars.

The bond strength of deformed bars is calculated according to EN1992.

The program assumes that the anchorage is provided by a smooth bar, with or without a hook at the end.

The Code refers only to deformed bars; the capacity of smooth bars is calculated by reducing the capacity of deformed bars by a factor of 2.25.

For a bar with a standard hook, the capacity is increased by a factor of 1.0/0.7 = 1.428.

$$\phi \le 32 \quad N_{Rd,bond} = \frac{1}{2.25}(\pi\phi l_b f_{bd})$$

$$\phi > 32 \quad N_{Rd,bond} = \frac{(132-\phi)/100}{2.25}(\pi\phi l_b f_{bd})$$

where:

ϕ = anchor bolt diameter

$\mathbf{l_b}$ = basic anchorage length, measured from the face of the foundation

$\mathbf{f_{bd}}$ = concrete bond strength

$$f_{bd} = 2.25\frac{f_{ctk,0.05}}{\gamma_c} = 2.25\frac{0.7\cdot 0.3f_{ck}^{2/3}}{\gamma_c}$$

γ_c = 1.5
$f_{ctk,0.05}$ = $0.7f_{ctm}$ = concrete tension strength according to EC2, Table 3.1
f_{ctm} = $0.3f_{ck}^{2/3}$
f_{ck} = nominal concrete strength(e.g. f_{ck} =25 for concrete type 25/30)

Steel failure - $N_{Rd,s}$:

$N_{Rd,s}$ = tensile strength of the bolt
= $(f_{ub}A_s)/\gamma_{MS}$
f_{ub} = tensile strength of the bolt
A_s = net area of the bolt
γ_{MS} = $1.2(F_{ub}/F_{yb})$ = 1.4

Pull-out failure :

The pull-out capacity is defined by the stress at the contact between the head of the bolt and the concrete:

$$N_{Rd,p} = N_{Rk,p}/\gamma_{Mp} = (6A_h f_{ck,cube}\psi_{ucr,N})/\gamma_{Mp}$$

where:
A_h = $\pi/4(d_h^2 - d^2)$ - for round or hex bolt heads
= $s^2 - v^2$ - for square bolt heads
d_h = bolt head diameter
d = bolt shaft diameter
s = square bolt head outside dimension
v = square bolt head step dimension (adjacent to the shaft)
$f_{ck,cube}$ = characteristic concrete cube strength
$\psi_{ucr,N}$ = 1.0 - for cracked concrete
= 1.4 - for uncracked concrete
γ_{Mp} = 1.5

Concrete cone failure :

The program checks the concrete cone capacity only when "supplementary reinforcement" is not added, i.e. when the ☐ **Supplementary reinforcement provided** box is not checked.

$$N_{Rd,c} = N_{Rk,c}/\gamma_{Mc} = N^0_{Rk,c}(A_{c,N}/A^0_{c,N})(\psi_{s,N})/\gamma_{Mc}$$

where:
$N^0_{Rk,c}$ = $k_{cr}\sqrt{(f_{ck,cube})}\, h_{ef}^{1.5}$ - the capacity of a single bolt in cracked concrete, not influenced by the distance to the edge of the concrete or an adjacent bolt.
= $k_{ucr}\sqrt{(f_{ck,cube})}\, h_{ef}^{1.5}$ - the capacity of a single bolt in uncracked concrete, not influenced by the distance to the edge of the concrete or an adjacent bolt
γ_{Mc} = 1.5
k_{cr} = 8.5
k_{ucr} = 11.9
h_{ef} = effective embedment depth
$f_{ck,cube}$ = characteristic concrete cube strength

13.4.11.3.7.3 T-stub in compression

The minimum of:

- the bending capacity of the plate together with the concrete bearing calculated for the effective area under the column flange only (column flange T-Stub in compression), i.e. the calculation is identical to a column designed for compression only where the effective area consists of one segment only.
- the compression capacity of the flanges and web of the column:
  $\mathbf{F_{c,fc,Rd} = 1.2 t_{fc} b_{fc} f_{y,fc}}$

where

$\mathbf{t_{fc}}$ = column flange thickness

$\mathbf{b_{fc}}$ = column flange width

13.4.11.3.7.4 T-stub in tension

The capacity is computed on the tension side as for a row of bolts on the extended portion of an extended-plate beam-column connection, according to the T-stub in tension model.

$$l_{eff,cp} = \min[(2\pi m_x),(\pi m_x + w),(\pi m_x + 2e)]$$

$$l_{eff,nc} = \min[(0.5b_p),(4m_x + 1.25e_x),(e + 2m_x + 0.625e_x),(0.5w + 2m_x + 0.625e_x)]$$

Effective length for the failure mode:

Mode 1 effective T-stub length: $\mathbf{l_{eff,1}}$ = $\mathbf{\min(l_{eff,cp}, l_{eff,nc})}$

Mode 2 effective T-stub length: $\mathbf{l_{eff,2}}$ = $\mathbf{\min(l_{eff,nc})}$

The capacity of a row of bolts in tension is calculated as the minimum value of the following modes of failure:

- Mode 1 : plastic mechanism in the plate:
  $\mathbf{F_{t,1,Rd} = (4M_{pl,1,Rd})/m}$
- Mode 2 : a combination of failure of the plate with failure of the anchor bolts:

$$F_{t,2,Rd} = \frac{2M_{pl,Rd,2} + n\sum F_{t,anchor,Rd}}{m+n}$$

If **L > $\mathbf{L_b}$*** prying forces don't develop in the plate at these modes of failure must be replaced with Mode 1-2:

$$L_b > L_b^* = \frac{8.8A_s}{l_{eff,1}}\left[\frac{m}{t_p}\right]^3$$

$\mathbf{L_b}$ = anchor bolt elongation length = 8d + grout thickness + plate thickness

This value is calculated according to Table 6.2 in the Code. The program assumes that the grout thickness = 30 mm.

d = anchor bolt diameter.

Mode 1-2:

$\mathbf{F_{t,1\text{-}2,Rd} = (2M_{pl,1,Rd})/m}$

Mode 3: Anchor bolt failure:

$$F_{t,3,Rd} = \sum F_{t,anchor,Rd}$$

$$M_{pl,1,Rd} = 0.25\sum l_{eff,1} t_p^2 f_y / \gamma_{M0}$$

$$M_{pl,2,Rd} = 0.25\sum l_{eff,2} t_p^2 f_y / \gamma_{M0}$$

13.4.11.3.7.5 Axial + Moment

The calculation for the combination of axial force and bending moment is carried out as follows:

A positive moment acting clockwise: $\mathbf{M_{Ed}} > 0$

A positive axial force creates tension in the column: $\mathbf{N_{Ed}} > 0$

[+] tension
[-] compression

In order to determine the stress acting on each side of the column the axial force is divided into two and the moment is divided by the lever arm between the centers of the two flanges:

Left side: $\mathbf{N_{Ed}/2 + M_{Ed}/z_f}$ (+) (+) (-) (-)
Right side: $\mathbf{N_{Ed}/2 - M_{Ed}/z_f}$ (+) (-) (+) (-)

where:

- z_f = distance between the centers of the flanges
- the eccentricity is calculated as $\mathbf{e_N = M_{Ed}/N_{Rd}}$ and includes the sign of the eccentricity
- the capacity of the tension side $\mathbf{F_{t,Rd}}$ is calculated from the lowest value calculated for the above failure modes.
- the capacity of the compression side $\mathbf{F_{c,Rd}}$ is calculated as explained above.
- the lever arm **z** is calculated according to Table 6.7 and Figure 6.18 in the Code.
- the moment capacity of the connection is then calculated according to the state determined by the preliminary calculation and Table 6.7.

13.4.11.3.7.6 Shear

The shear capacity on the base $\mathbf{F_{v,Rd} = F_{f,Rd} + nF_{vb,Rd}}$ consists of the friction between the bottom of the plate and the grout and the shear capacity of the anchor bolts.

n = the number of bolts when both sides are in compression or the bolts on one side only when one or both sides are in tension.

- friction between the base plate and the grout:
  $\mathbf{F_{f,Rd} = C_{f,d}F_t}$
  where:
  $\mathbf{C_{f,d}}$ = the friction constant between the base plate and the grout, = 0.20 for a water-cement grout.
  $\mathbf{F_t}$ = the axial compression force in the column, taken as 0.0 when the column is in tension.

- Shear capacity:
  $\mathbf{F_{vb,Rd} = \min(F_{1,vb,Rd}, F_{2,vb,Rd})}$
  - $\mathbf{F_{1,vb,Rd}}$ = bearing capacity 1394
  - $\mathbf{F_{2,vb,Rd}}$ = shear capacity:
    the shear capacity of a single anchor bolt is calculated as:

    $$F_{2,vb,Rd} = \frac{\alpha_b f_{ub} A_s}{\gamma_{M2}}$$

    where:
    α_b = $\mathbf{0.44 - 0.0003f_{yb}}$
    $\mathbf{f_{yb}}$ = yield strength of the bolt
    235 = $\mathbf{f_{yb}}$ = 640 mPa

γ_{M2} = 1.25

13.4.11.3.7.7 Flange w eld

When calculating the weld size required it is usually possible to design a weld with a capacity less than the capacity of the welded part, a situation that can lead to brittle failure. Therefore the weld is designed to be equal or greater than the capacity of the welded part in order to prevent this situation, i.e. the welds are designed as "full strength".

The weld size required is:

$$a = \frac{f_y \beta_w \gamma_{M2}}{\sqrt{2} f_u \gamma_{M0}} t$$

where:

a = weld throat dimension.
$\mathbf{f_y}$ = yield strength of the weakest of the welded parts
$\mathbf{f_u}$ = tensile strength of the weakest of the welded parts
t = thickness of the welded part

13.4.11.3.7.8 Code limitations

- EN1993-1-8: Sections 6.2.6.12 (5) -
  does not permit anchorage by bonding when the yield strength of the anchor bolt is greater than 300 Mpa.

- EN1993-1-8: Sections 6.2.6.12 (5) -
  ensures that the anchor bolt yields before a bond failure:
  $\mathbf{N_{Rd,s} = (A_s f_{yb})/\gamma_{Ms} = N_{Rd,bond}}$

- EN1993-1-8: Sections 6.2.5 (7) -
  stipulates that the grout strength be greater or equal to the concrete strength if the grout thickness is greater than 50 mm.
  $\mathbf{f_{ck,g} = f_{ck}}$

- EN1993-1-8: Sections 6.2.5 (7) -
  stipulates that $\beta_j = {}^2/_3$ may be used if the following two conditions are satisfied:
  $\mathbf{f_{ck,g} = 0.2 f_{ck}}$
  $\mathbf{t_g = 0.2 \min(B,D)}$

13.5 AutoStrap

13.5.1 Main menu

Create a *STRAP* model from one or more DXF files. The program automatically identifies beams, columns and walls when importing the drawings and enables the user to modify these elements and to define loads.
For more information refer to General 1422.

Parameters

Define the following parameters:
- element mesh parameters 1453
- slab thickness 1454
- beam dimensions 1455
- wall openings 1457
- DXF files 1458 included in the model
- List of levels 1459 in the model and their elevations
- Origin 1460 (reference point) to align all levels

DXF lines

Revise the definition of the DXF lines 1460 as various element types (beams, columns, walls and contours)

Loads

Define dead and live loads 1461:
- uniform element loads on floor slabs
- uniform and point loads on beams

STRAP

Create the STRAP 1465 model from the DXF files.

Beamd

Create BEAMD 1469 files

Different parameters, loads, etc, may be entered for each DXF drawing. The program displays a list of the files below the menu if more than one DXF is imported into the model, e.g.

IS9

The current DXF is displayed in the box; click on the ▼ to select a different file.

Menu bar options:

File Zoom Display Help

13.5.2 General

AutoSTRAP is a program that converts DXF drawings or IFC models to *STRAP* models:
- DXF:
  - one or more DXF drawings, each of which represents a different floor level, may be imported.
  - the lines that represent beams, walls, columns or general contour line must be in different DXF drawing layers; *AutoSTRAP* automatically identifies the relevant lines and display a drawing showing equivalent *STRAP* beams, walls and columns.
- IFC:
  - the STRAP model is created from a single IFC file.
  - the identifies IFC slab, wall, column and beam elements
  - these IFC elements can be deleted are revised at a preliminary stage.

AutoSTRAP then creates a proposed STRAP model which may be modified and edited.
- element meshes representing floor slabs may also be created.

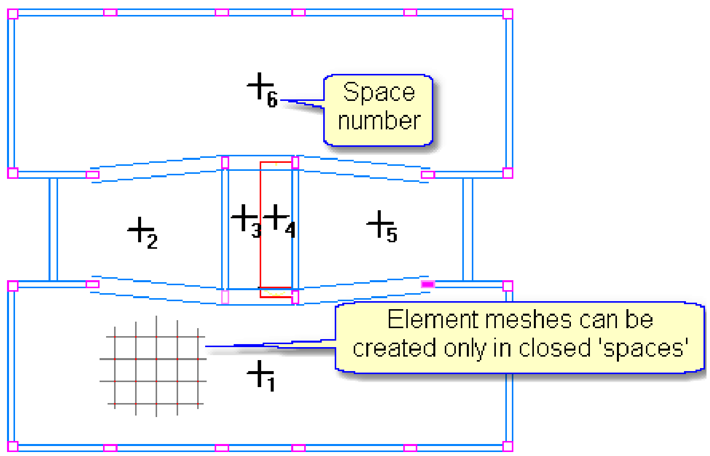
- element and beam loads may be defined.
- the user may define a series of "levels", each at a different elevation, and assign a DXF drawing to each level.
- a *STRAP* model is then created. Each level is transformed to a "submodel" while the columns and walls form the "Main model".

Refer to:

- DXF - procedure 1423
- DXF - import notes 1427
- IFC - procedure 1429
- IFC - Import notes 1432
- STRAP model - notes 1433

Spaces:

- element meshes can only be created in a "***space***".
- a space is an area completely enclosed by beams, columns, walls and or contours.
- spaces are automatically identified and created by the program.

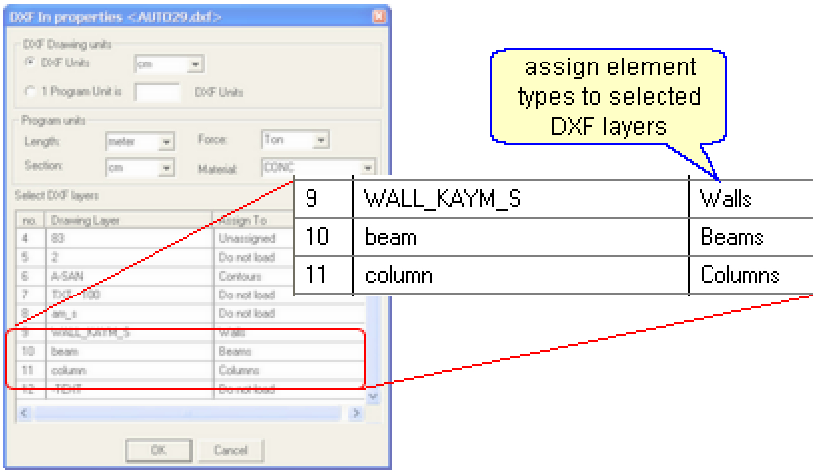


- openings must also be completely enclosed by contour lines.
- spaces with very small dimensions (< minimum element size) will not be created. Refer to DXF Import - notes 1429.

13.5.2.1 DXF - Procedure

The following is a general outline of the basic steps for creating a STRAP model from a DXF file:

- Select the File - new 1440 option, select the DXF file and identify the layers containing beams, columns, walls and contours:

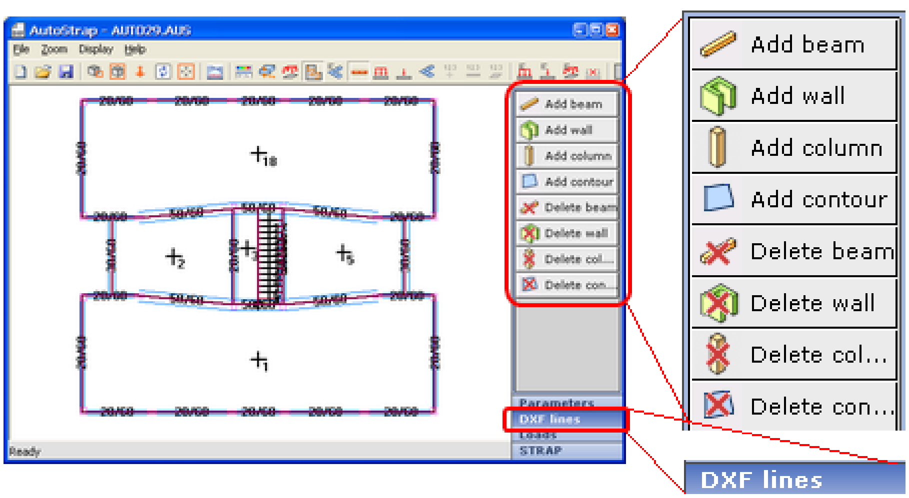


Refer to:

- [DXF import - notes](1427)
- [STRAP model - notes](1433)

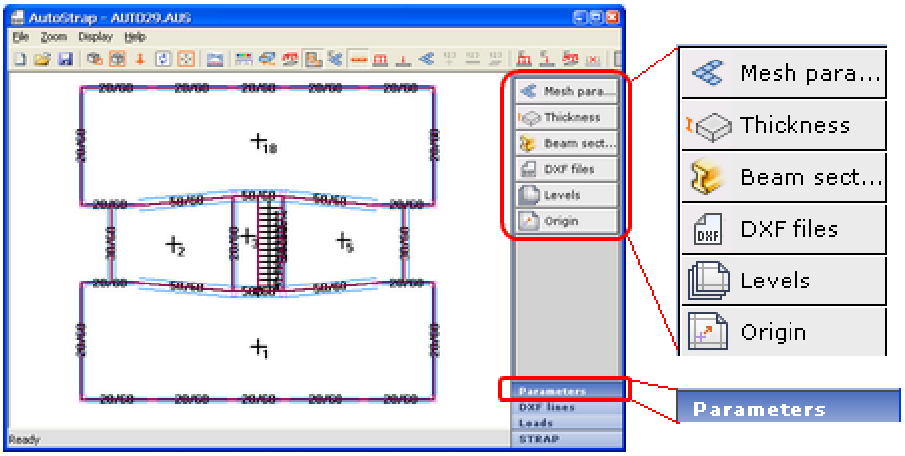
- The program automatically identifies the beams, columns, walls and contours in the assigned layers and creates the structural drawing. However, in many cases the program's conversion of the DXF file to a structural model will not be complete, particularly if the various element types are not assigned to different layers or if extraneous lines are found in a specific element type layer, i.e. the program may either fail to identify beams, columns and walls or it may create these elements where none exist.

  Select the [DXF lines](1460) option in the side menu and add or remove elements from the drawing:



- Modify the drawing element parameters:
  - element mesh parameters 1453
  - slab thickness 1454
  - beam dimensions 1455
  - DXF files 1458 included in the model (a different file may be used for each level)
  - List of levels 1459 in the model and their elevations
  - Origin 1460 (reference point) to align all levels

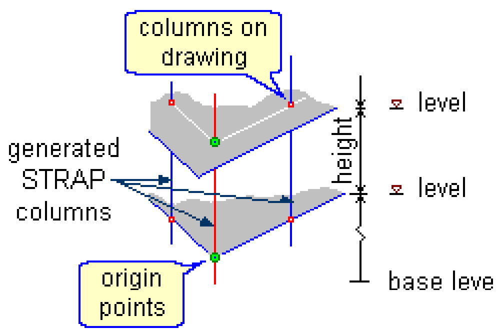
Select the Parameters 1452 option in the side menu:



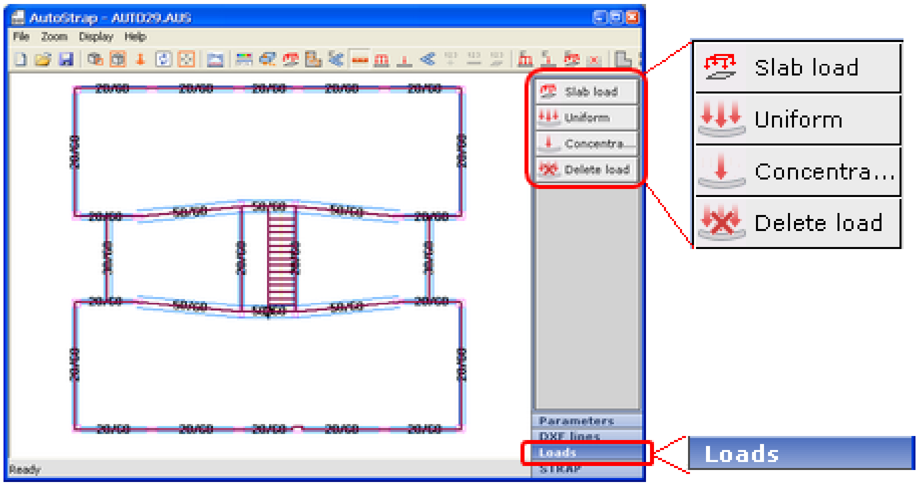
The 

and 

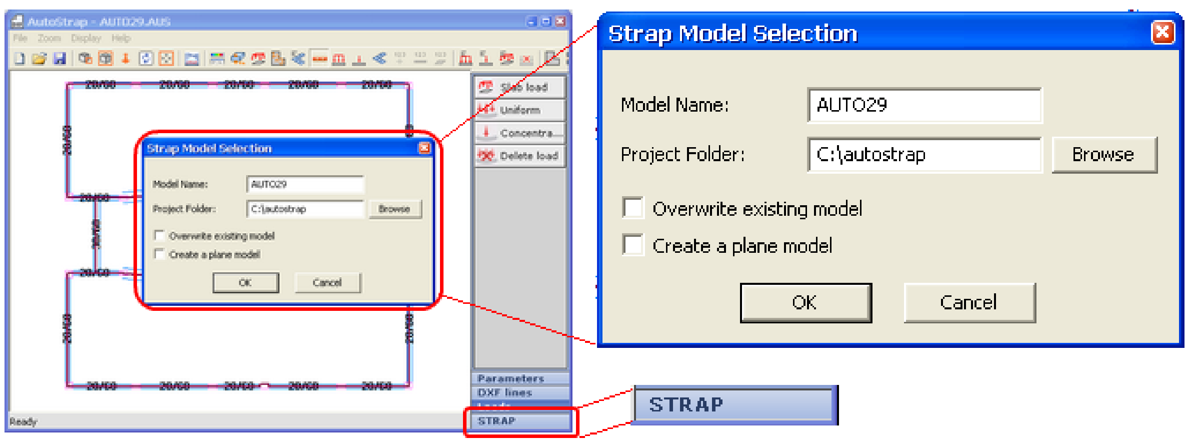
options are important for creating the columns in 3D models. The program looks for column elements at the same coordinate at adjacent levels (relative to the origin point):



- Apply loads 1461 to the slabs and beams in the model (optional):

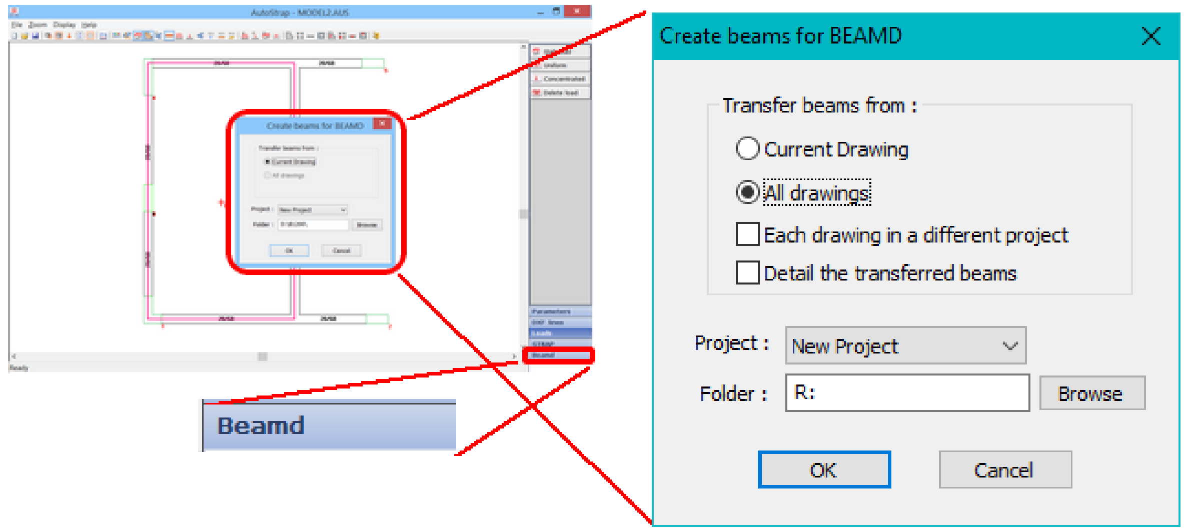


- Create the STRAP model 1465:



- Create BEAMD 1469 files:

  Transfer beam data (geometry & loads) to the BEAMD program to design and detail the reinforcement:



13.5.2.2 DXF Import - notes

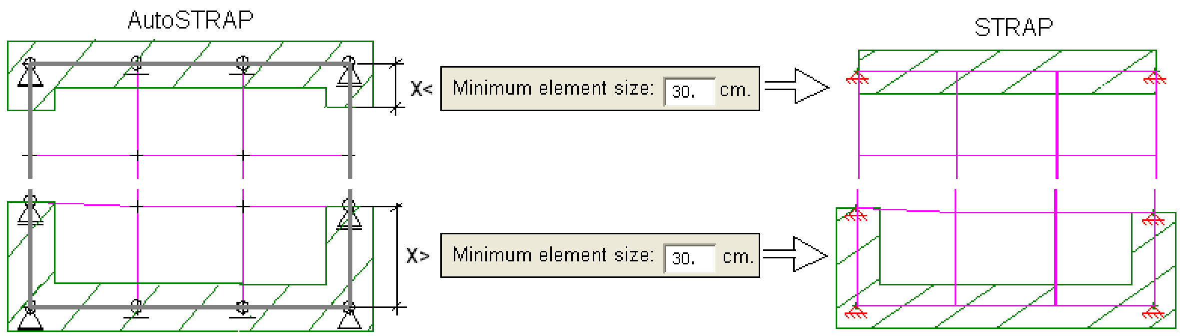
The various drawing elements (beams. walls, columns, etc) will be properly identified by the program if they are drawn properly in the DXF file. It is vital that each element type be drawn in a separate layer.

Beams

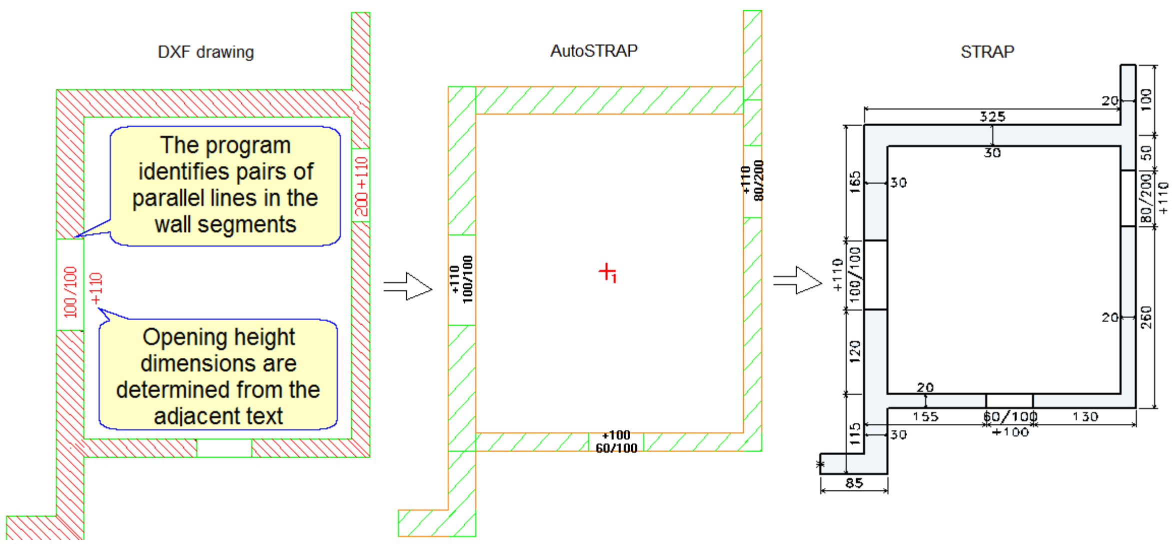
- The program searches for parallel lines in all beam layers and converts them to beams:
  - 2 parallel lines - rectangular beam
  - 3 parallel lines - L-beam
  - 4 parallel lines - T-beam
- Parallel lines that are too close together or too far apart are not converted to beams. The limiting distances are defined in Setup - general [1444].
- Beam height can be determined from the text written on the beam:
  - the second number after the dividing symbol is assumed to be the height, e.g 20/**70** or 20x**70.**
  - the text must be located between the beam lines or not greater than the text height away from one of the lines.
  - the text must be parallel to the beam lines ± 10°.
- The program can identify curved beams and converts them to a series of straight beams. The height derived from the text is applied only to the segment where the text is located.

Walls

- The program searches for parallel lines in all wall layers and converts them to walls.
- Parallel lines that are too close together or too far apart are not converted to walls. The limiting distances are defined in Setup - drawing [1444].
- Wall segments that are shorter than the "minimum element size" are also ignored by the program:



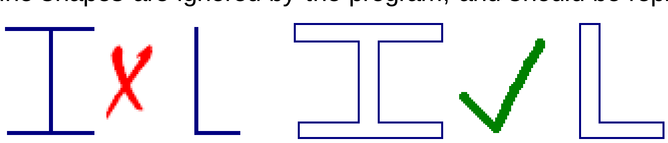
- The program identifies colinear segments and defines them as such in the *STRAP* model.
- The program identifies the wall openings by the two perpendicular lines and the adjacent dimension text:



  - the program takes the horizontal dimension of the opening from the location of the parallel lines (text is ignored).
  - the height of the opening is retrieved from the text:
    - the second number of a pair, e.g 80/**240** or 80x**240,** where any symbol can be used as a divider.
    - the number - if only one is written.
  - the dimension from the floor to the bottom of the opening must be written in the format +nnn, e.g. **+105**. A default value of 100 is used if the dimension is not present on the DXF drawing.
  - dimension opening text must be assigned to the Walls layer.

Columns

- the program searches for ***closed shapes*** in the column layer and converts them to columns.
- line shapes are ignored by the program, and should be replaced by any closed shape, e.g.

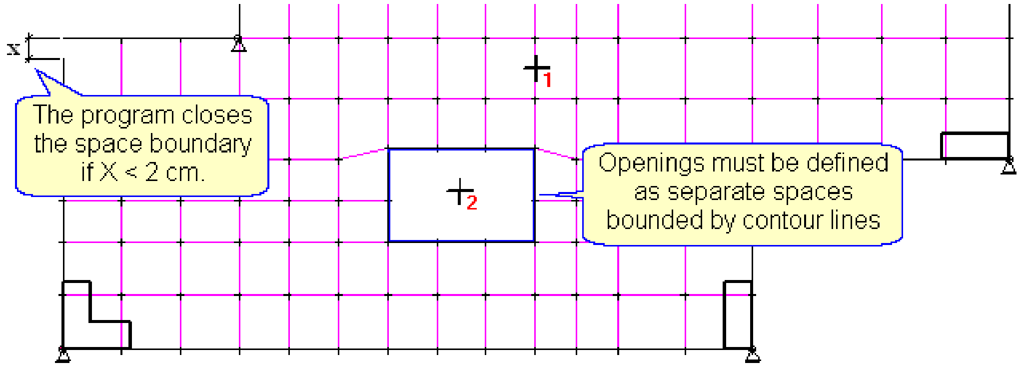


- Rectangular, I, T, L and round sections are converted to equivalent STRAP geometry sections; irregular shaped columns are transferred to *STRAP* as "*CROSEC*" sections and can be designed as "Solid sections" in the Concrete design module.
- "*CROSEC*" sections are defined with J = 0 (torsional moment-of-inertia).

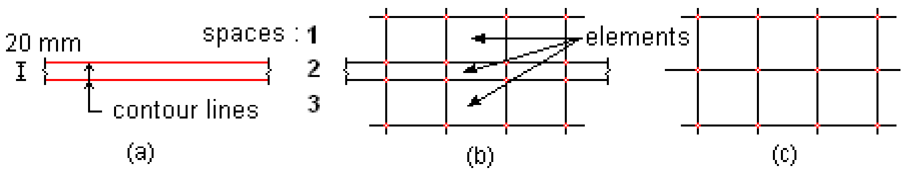
Slabs

The program generates an element mesh in "spaces 1423", areas on the drawing completely enclosed by lines. The element size is determined according to the mesh parameters 1453.

- the program automatically 'closes' spaces if the distance between lines is less than 2 cm.
- the program defines a uniform mesh in each space. If you want different mesh spacing in different parts of a space, then subdivide the area by adding lines to the "contour line" layer in the DXF file.
- Openings must also be surrounded by a contour.



- The program connects the elements on both sides of a space boundary to common nodes along the boundary.
- The program does not create elements smaller than the "Minimum element size" (refer to Setup 1442 and Mesh parameters 1453). For example, there is a 20 mm. expansion joint in a slab and the minimum element size is 100 mm:



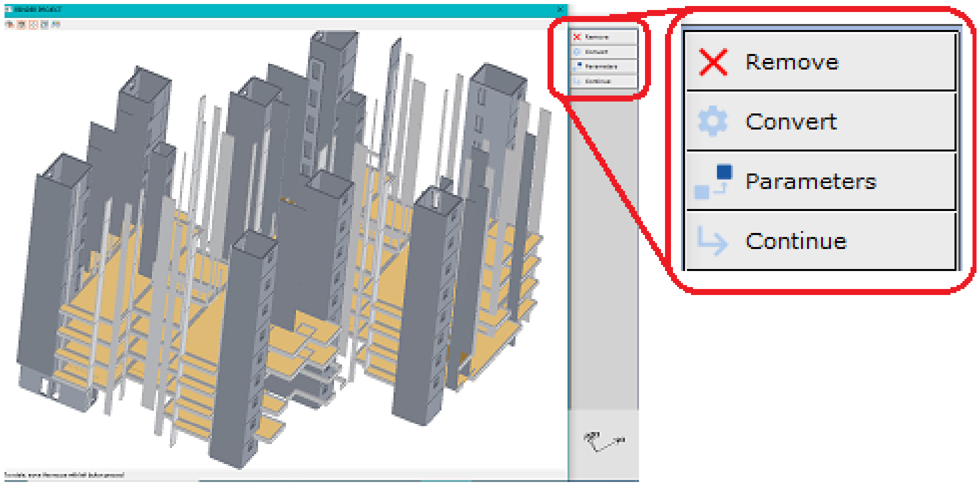
The expansion joint is bounded by contour lines as shown in (a). Theoretically the program should create three spaces,each with an element mesh, as shown in (b). But the element dimension is less than the minimum allowed, so the program automatically eliminates the elements in space 2 and creates the mesh shown in (c).

However this creates an incorrect continuous mesh. To model the expansion joint, the minimum element size must be reduced to less than the gap width; the program will then create the small elements in space 2 which can be deleted.

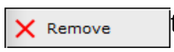
13.5.2.3 IFC - Procedure

The following is a general outline of the basic steps for creating a STRAP model from an IFC file:

- Select the File: New - import an IFC file 1440 option, select the IFC file. The program identifies IFC beam, column, wall and slab elements.
- the program displays a rendered version of the IFC model:

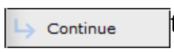


- Select:



to delete beams, columns and/or walls.

to change walls to columns, beams to walls, etc.

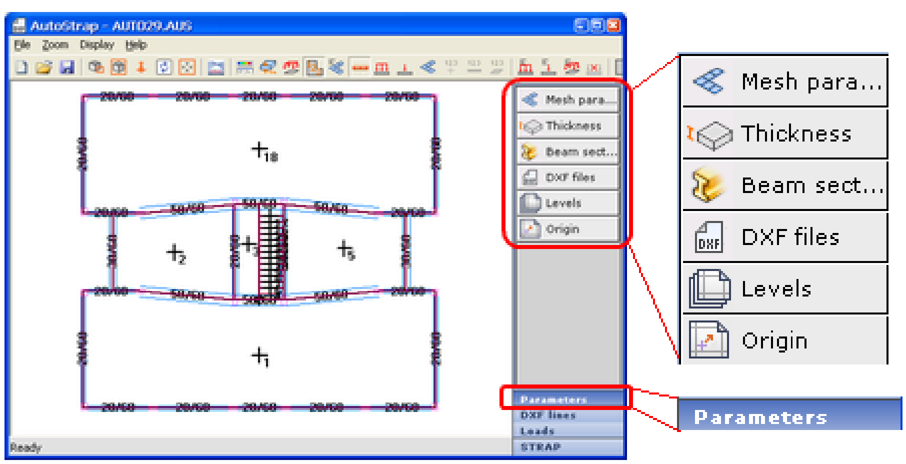


to create the preliminary STRAP model.

The program then displays the preliminary STRAP model:

- Modify the drawing element parameters:
  - element mesh parameters 1453
  - slab thickness 1454
  - beam dimensions 1455
  - DXF files 1458 included in the model (a different file may be used for each level)
  - List of levels 1459 in the model and their elevations
  - Origin 1460 (reference point) to align all levels

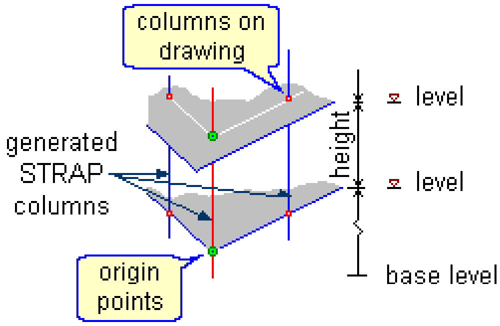
Select the Parameters 1452 option in the side menu:



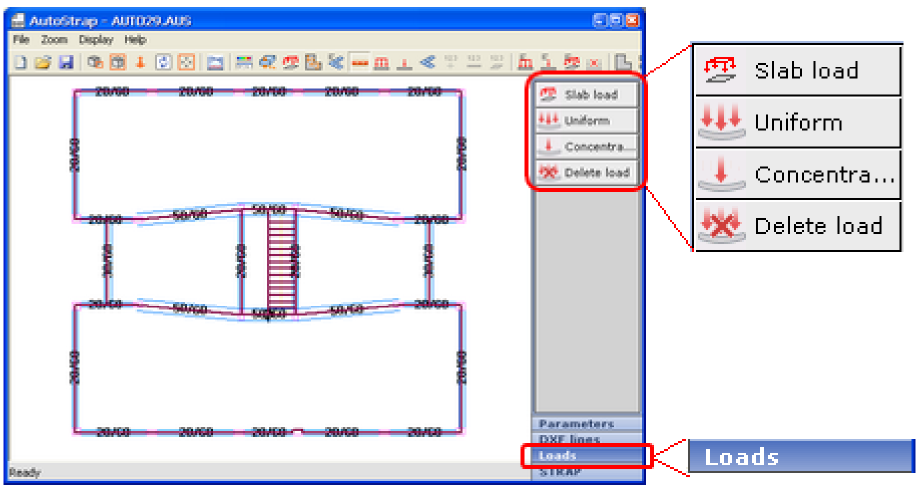
The 

and 

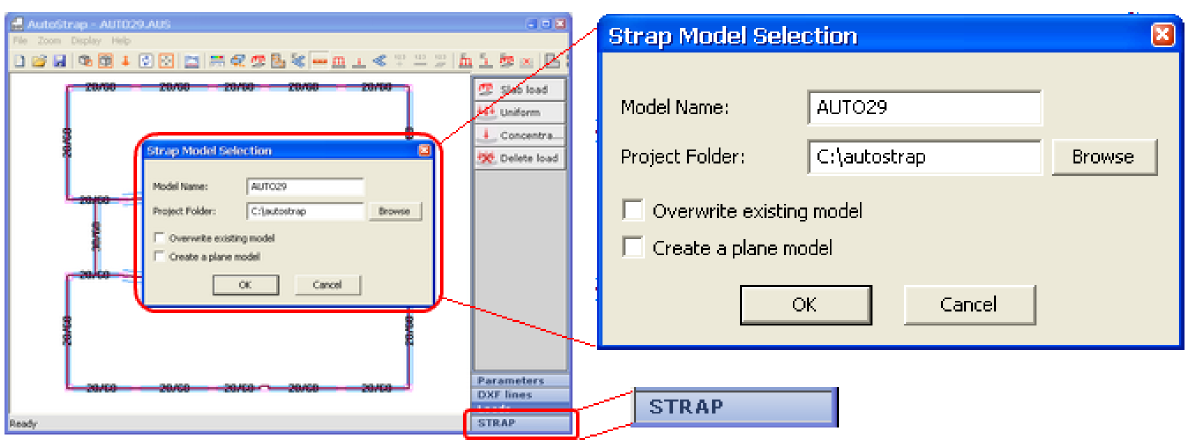
options are important for creating the columns in 3D models. The program looks for column elements at the same coordinate at adjacent levels (relative to the origin point):



- Apply loads 1461 to the slabs and beams in the model (optional):

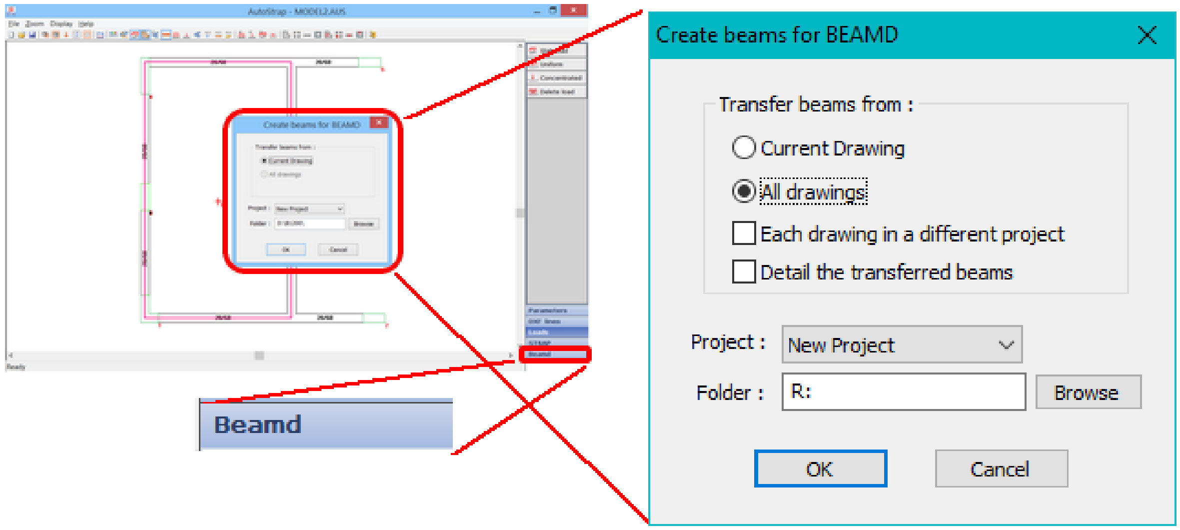


- Create the STRAP model 1465:



- Create BEAMD 1469 files:

  Transfer beam data (geometry & loads) to the BEAMD program to design and detail the reinforcement:



13.5.2.4 IFC - Import notes

IFC is a BIM format for importing three-dimensional drawings. The format is supported by many building softwares such as TEKLA and REVIT.

AutoSTRAP identifies the following objects in IFC files:
- beams (IFCBEAM), columns (IFCCOLUMN) and general beams (IFCMEMBER). These are referred to as "Beams" in the program.
- walls (IFCWALL)
- solid slabs (IFCSLAB)
- steel plates (IFCPLATE). The program does not add them to the *STRAP* model but uses them to decide whether two beams are combined at a common node.
- "Other elements": these are miscellaneous IFC objects that can be converted to beams,walls, slabs or plates.

Levels:
The program reads the levels from the IFC file. In addition, *AutoSTRAP* can create a level at every elevation where it finds a SLAB in the IFC file. A tolerance parameter is used by *AutoSTRAP* to decide whether to combine two levels with a small elevation difference. Refer to Setup - IFC 1445.

Options:
The program identifies two different model types:
- a model with levels:
  the user can:
  - delete columns, walls and/or beams.
  - convert walls to columns, beams to walls, etc.
  - define loads at all levels, specify mesh parameters, etc.
- a model with no levels and no slabs (e.g.a steel hangar):
  the user can:
  - delete columns, walls and/or beams.

Sections:
The program identifies the beam section types from their three-dimensional representation. If the shape resembles a steel section, the program searches for a section with the same dimensions in the steel tables and uses it in the STRAP property table.

Limitations:

- *AutoSTRAP* creates *STRAP* elements according to the way they were defined in the IFC file. For example, if a wall was defined as a beam (IFCBEAM), *AutoSTRAP* will create a deep beam and not a STRAP wall element.
- A wall that is a combination of two IFC wall elements, each with half of the total width, will not be correctly identified by AutoSTRAP.
- a precast hollow-core floor slab defined in the IFC file as a series of adjacent beams will not be correctly identified by AutoSTRAP. They should be defined as a series of slabs.

13.5.2.5 STRAP model - notes

AutoSTRAP can be instructed to create either a ***plane model*** consisting of one level only or a ***space model*** consisting of multiple levels, connected by columns and walls. Therefore the program handles the DXF columns and walls differently when creating the *STRAP* model, depending on whether it is a 2D or 3D model.

Plane model

Beams:

Beams are created along the beam center-lines. Offsets are added where necessary to place the beam in its correct location

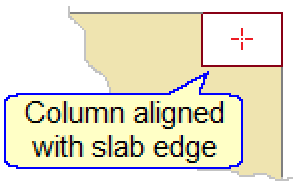
Slabs:

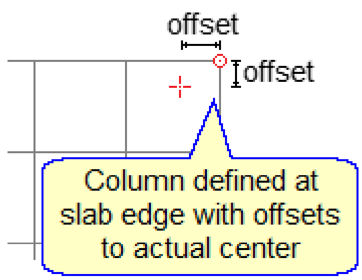
Slabs are created bounded by the beam center-lines, the wall center-lines and the contour lines. The column shape is ignored.

Columns:

A restraint is defined at every column location:

- columns along the slab edge:
  the column is defined connected to the node at the slab edge but with an offset to its correct location:



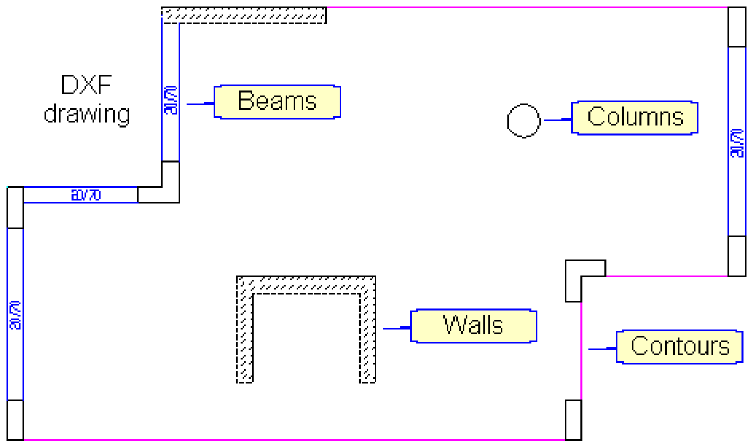


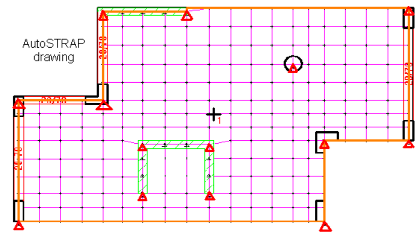
- columns not on the slab edge:
  the restraint is defined at the center of the column.

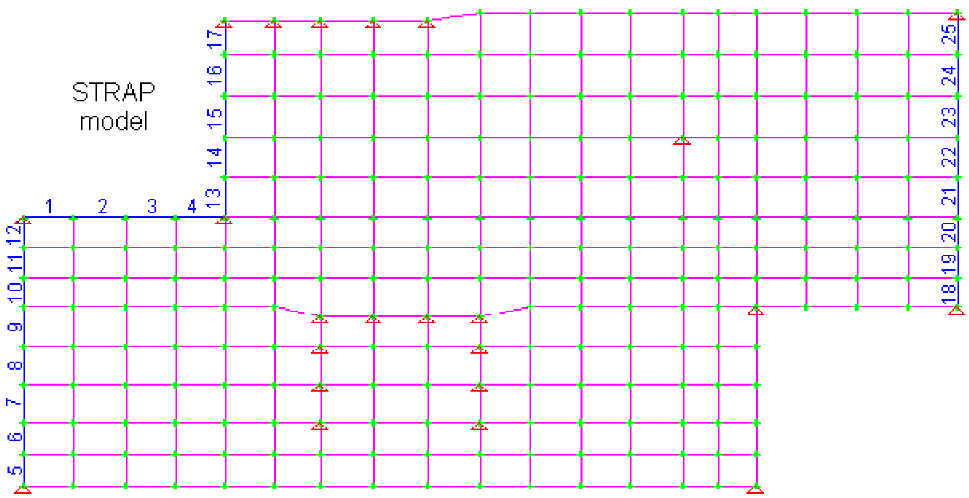
Walls:

The program creates a series of supports at the nodes along the center-line of the wall. Offsets are automatically added where thicknesses vary to place the segments in their correct location .

Example:


DXF
drawing
Beams
Columns
Walls
Contours


AutoSTRAP
drawing


STRAP
model

Space model

- The program can either create a submodel for every level in the model or create a model without submodels. Each support node (column or wall end node) is defined as a "connection point" in the submodel. **By default the program defines the "connection points" as "pinned"**. To revise connection points to "fixed", first create the model, then use the Submodel - connection points 387 option in *STRAP* geometry.
- Walls and columns are defined in the "Main model".

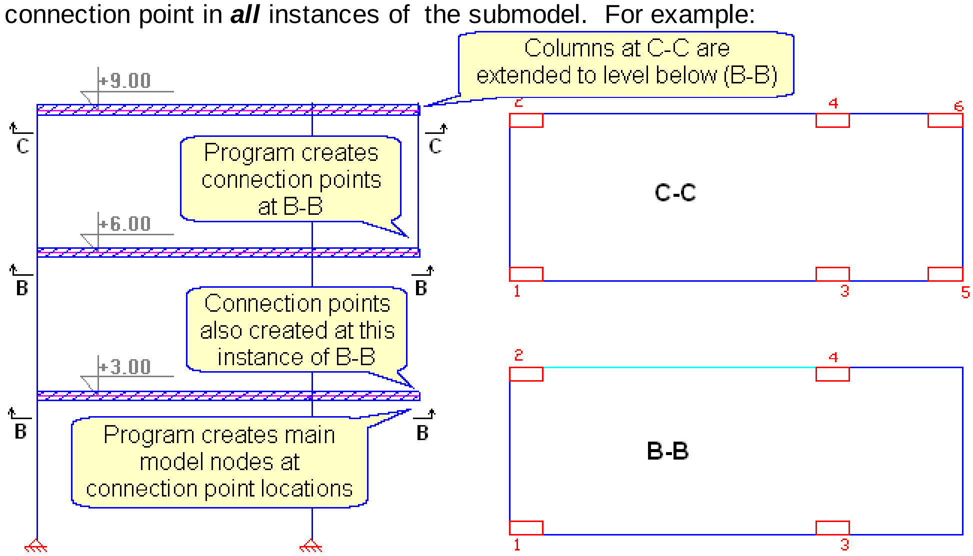
Refer to Procedure 1423 for more information about levels and origin points.

Beams and slabs:

Refer to plane models.

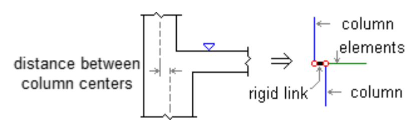
Columns:

- columns are created from the location at any level to the level ***below,*** i.e the program creates nodes at the current level and the level below and connects them with a *STRAP* beam element. Therefore a column that starts at a specific level should be removed from the *AutoSTRAP* drawing at that level and should first appear at the level above.
- If a column starts at a specific level, then the program automatically restrains the connection point at that level for rotation about the column.
- the column at the lowest level is continued down to the base level, where the program automatically creates a fixed support.
- if a connection point is required at one instance of a submodel then the program creates the connection point in ***all*** instances of the submodel. For example:

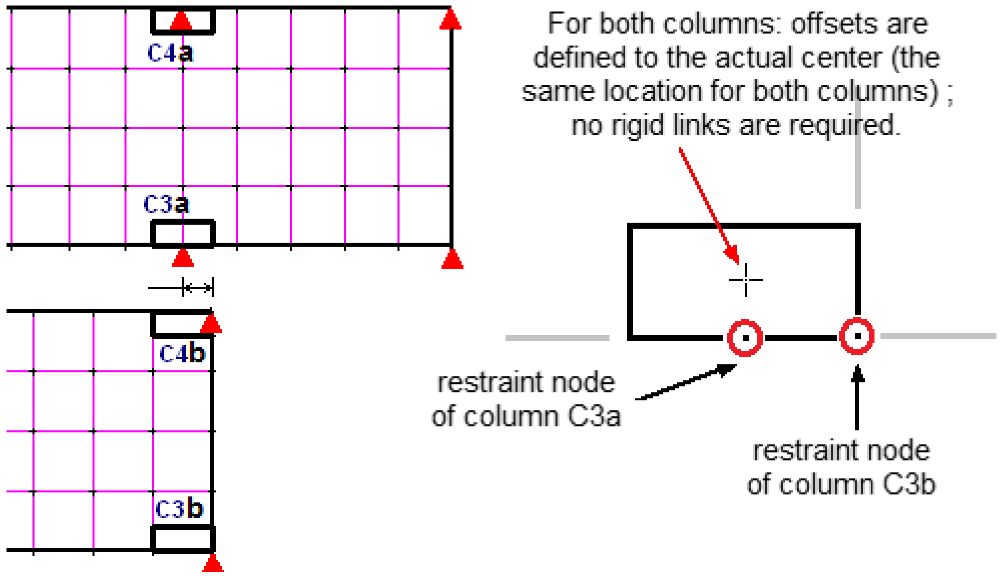


The connection points at the end of the cantilevered slab at level +3.00 are not connected to the main model, which is not allowed. Therefore the program automatically created main model nodes at these locations (restrained against rotation about the height axis). ***Do not erase these main model nodes.***

- the program automatically creates rigid links at locations where the column centers are not aligned (independent of the Create rigid links 1465 option). For example, columns that change size at a floor level:



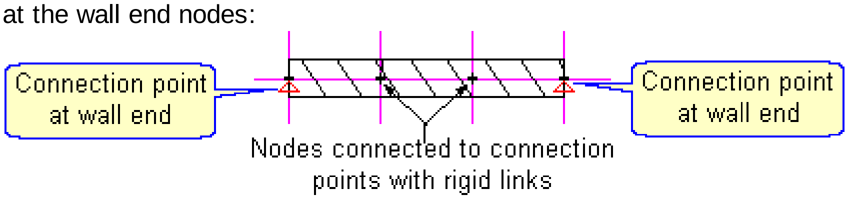
In the following example columns C3 (above) and C4 (below) are at the same physical location but are connected to nodes at different locations:



Walls:

In general, the rules for columns apply to walls:

- offsets are automatically added where thicknesses vary to place the segments in their correct location.
- walls are created from the location at any level to the level ***below,*** i.e the program creates nodes at the current level and the level below and connects them with a *STRAP* wall elements. Therefore a wall that starts at a specific level should be removed from the *AutoSTRAP* drawing at that level and should first appear at the level above.
- the wall at the lowest level is continued down to the base level, where the program automatically creates a fixed support.
- Linear rigid links are automatically defined to connect the wall interior nodes to the connection points at the wall end nodes:

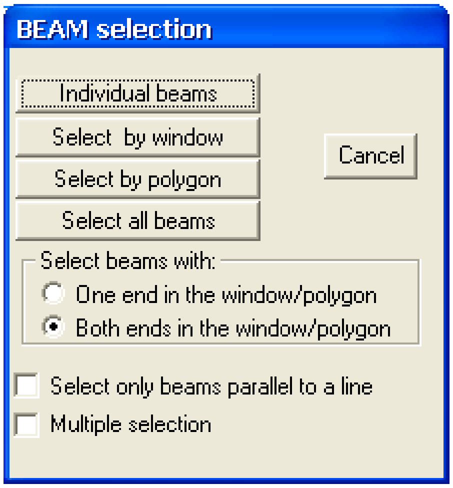


- If a wall starts at a specific level and the wall is shown only on the DXF drawing of the level above it, the program defines the connection points at the start level but does not define the rigid links connecting them to the other wall nodes. The rigid links must be added in *STRAP* geometry (note that rigid links added to one instance of a submodel will be added to all instances, which may be incorrect. In such cases the instances must be separated into different submodels - in *STRAP*).

13.5.2.6 Selection

Many options include instructions to select one or more beams, ribs, columns, walls, contours or spaces. Several options are available for selection one or more of these drawing elements.

For example, select beams (the options are similar for all other drawing elements):



Individual beams

Select a single beam only by moving the alongside the beam until it is highlighted by a rectangular blip ■; click the mouse.

When all the beams have been selected, press End selection or click the mouse without moving the .

Select by window

Define a rectangular window by pointing to its lower-left and upper-right corners with the . The program automatically identifies all beams with either one or both ends located in the window (refer to Select beams with 1438)

Select by polygon

Define a polygon by pointing to its corners with the . The program automatically identifies all beams with either one or both ends located in the polygon (refer to Select beams with 1438)

The polygon is constructed as a 'rubber-band' stretched around the defined corners:

- At least three corners must be selected.
- the program automatically connects the last corner defined to the first corner defined.
- press [Esc] (right mouse button) to delete the previous corner.
- to end the polygon definition, click the mouse without moving the .

Select all beams

All beams in the model will be selected.

Note:

- beams not displayed because of the the **Zoom** option ***are*** selected.

Select beams with:

- **One end in the window/polygon**:
  all beams with at least one end in the window/polygon are selected.
- **Both ends in the window/polygon**:
  only beams with ***both*** ends in the window/polygon are selected.

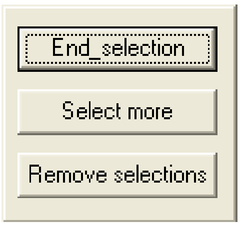
Select only beams parallel to a line

You may impose a further limit that the beams selected are only those parallel to a specified line.

Multiple selection

Turn on the checkbox if you want to define several windows, polygons or lists at the same time. After every window, etc., the following menu is displayed:

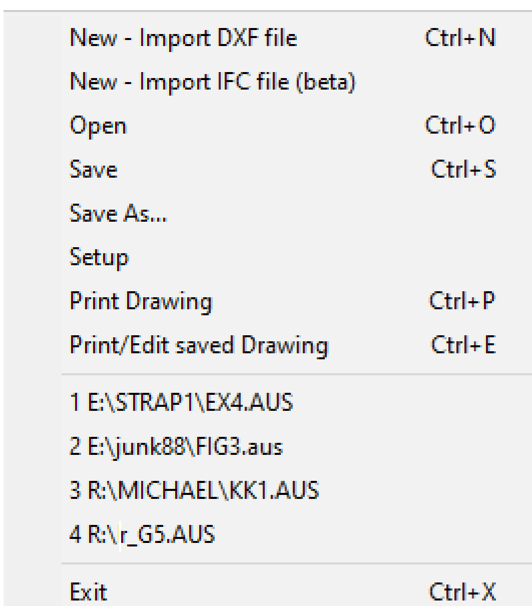
Select:



- Continue without selecting more beams
- Define another window, polygon, etc. for the same command.
- Delete members from the list, window, etc. already defined for this command.

13.5.3 File options

The program imports DXF files and creates an intermediate drawing representing a *STRAP* model. This *AutoSTRAP* drawing can then be modified by the user before the *STRAP* model is generated from it. All modifications are saved in a file *.AUS which may be further modified in *AutoSTRAP* at any time.



New - Import DXF file

Import a DXF 1440 drawing and create a new model. Refer also to:

- DXF - procedure 1423
- DXF - Import notes 1427

New - Import IFC file

Import an IFC[1442] drawing and create a new model. Refer also to:

- IFC - procedure[1429]
- IFC - Import notes[1432]

Open

Open an existing *.AUS drawing. This is the intermediate model drawing that contains are modifications to the structure made in *AutoSTRAP.*

Save

Save the current display as an *AutoSTRAP* drawing (*.AUS) with the same name as the DXF file.

Save as

Save the current display as an *AutoSTRAP* drawing (*.AUS) but with a different name.

Setup

Refer to Setup[1442]

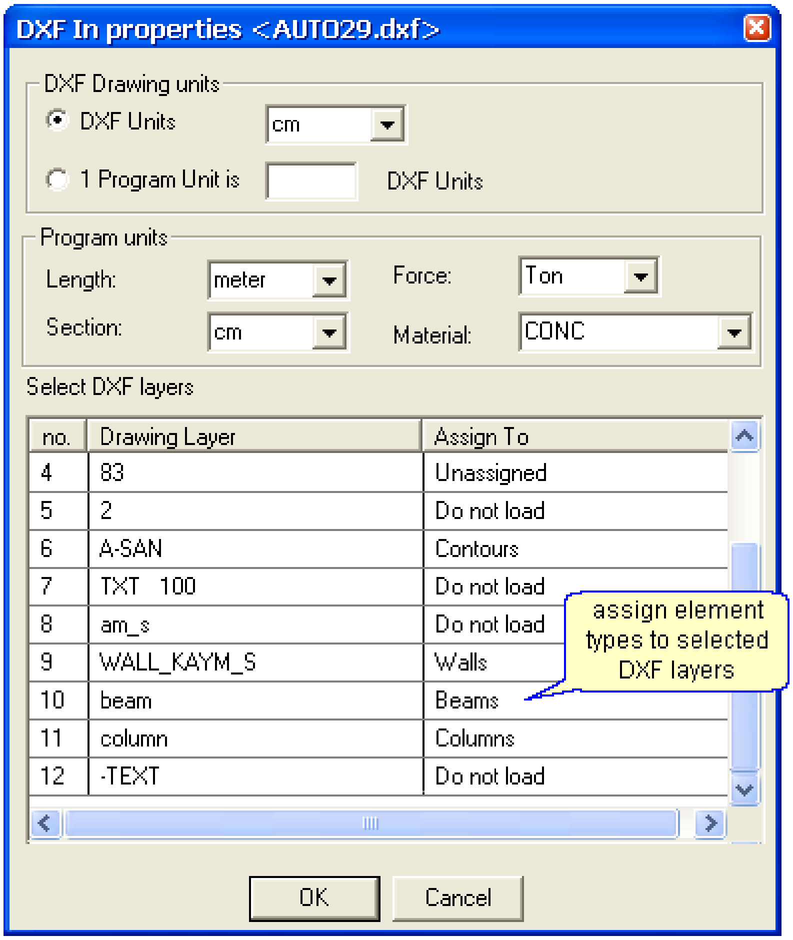
Print drawing

Print the current graphic display.

Print/edit a saved drawing

Refer to Print/edit a saved drawing.

13.5.3.1 New - import a DXF file



DXF - drawing units

The DXF drawing will be scaled according to the *STRAP* geometry units. Two options are available:

- **DXF units:**
  Select a unit from the list box
- **1 STRAP = x DXF units**
  Select the ratio between the units (the DXF dimensions will be divided by the value entered here). For example, *STRAP* units are meter, the DXF units are feet, but you want to double the size of the drawing in the background: Enter 3.281/2 = 1.6405

Program units

Specify the units for the *AutoSTRAP* display and definition and for the generated *STRAP* model.

Material

Select one of the *STRAP* standard materials. The selected material type is assigned to the entire model. The material may be changed later by revising the default material in the Files - Setup - General 1442 option.

DXF layers

The program displays a list of the layers in each DXF layer. Assign one of the following options to each layer:

- Do not load : the layer is ignored by the program and its lines are not displayed.
- Unassigned : the lines in the layer are initially displayed but are not used to create structural elements. The user may then manually create beams/columns/walls/contours from these lines.
- Contours : these lines define "spaces 1423" which are areas in which element meshes may be created
- Beams : parallel sets of lines are converted to beams:
  - 2 parallel lines - rectangular beam
  - 3 parallel lines - L-beam
  - 4 parallel lines - T-beam

  Refer to Setup - General - Dimensions 1443.
- Wall : parallel pairs of lines are converted to walls. Refer to Setup - General - Dimensions 1443.
- Column : closed shapes (standard or irregular) are converted to columns.

Note:

- In many cases the program's conversion of the DXF file to a structural model will not be complete, i.e. the program may either fail to identify beams, columns and walls or it may create these elements where none exist. The DXF lines 1460 option allows you to correct the drawing, e.g. delete beams/walls/columns from the drawing or create them from unassigned lines.

13.5.3.2 New - import an IFC file

Select an IFC file. The program imports the model and creates the various structural elements; refer to IFC Import notes 1432.

The program identifies two different model types:

- a model with levels:
  the user can:
  - delete columns, walls and/or beams.
  - convert walls to columns, beams to walls, etc.
  - define loads at all levels, specify mesh parameters, etc.
- a model with no levels and no slabs (e.g.a steel hangar):
  the user can:
  - delete columns, walls and/or beams.

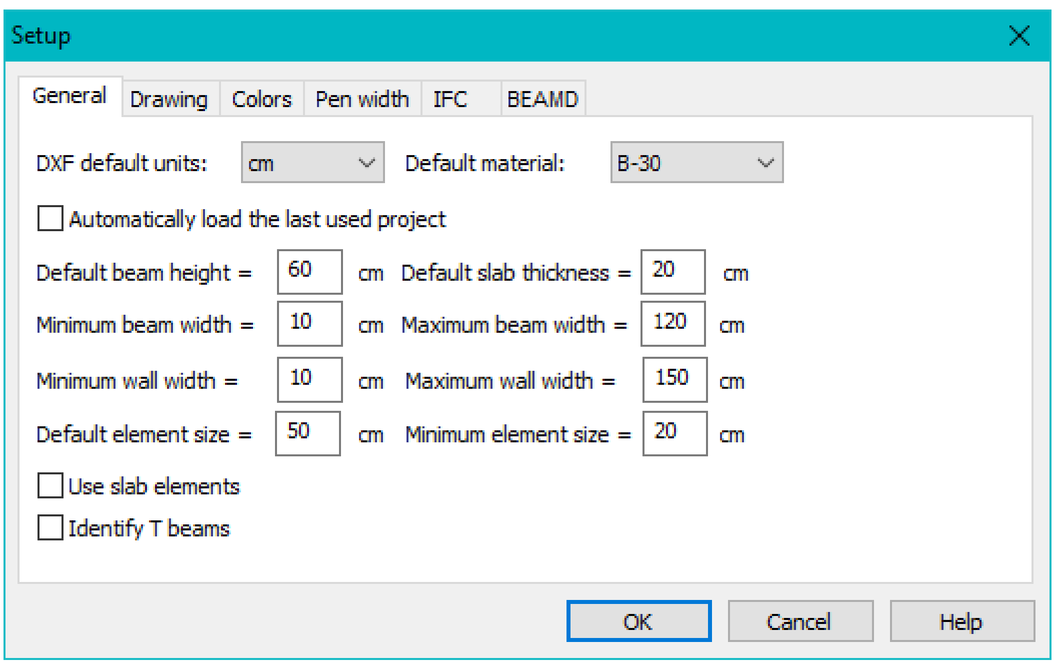
The program displays a rendered version of the model. Beams, columns, walls and slabs are displayed with different colours.

The model can be modified before the AutoSTRAP model is created. The following options are available:

13.5.3.3 Setup

13.5.3.3.1 General

Define the general default parameters for all new models:



DXF default units

The unit specified here is displayed in the menu when a new 1440 DXF file is imported.

Default material

The default material for all elements, transferred to properties table in the *STRAP* model. The material specified here is displayed in the menu when a new[1440] DXF file is imported.

Automatically load ...

☑ the previous *AutoSTRAP* project is automatically displayed upon running the program.

Dimensions

Specify the default dimensions for:

- dimensions that cannot be obtained from the DXF file.e.g. beam height, slab width, etc.
- element mesh parameters
- dimension limits for beams and walls:

  Beams and walls are created from pairs of parallel lines in the relevant layers. The program ignores lines that are too close together or too far apart. Specify the following parameters:
  - minimum beam/wall width: parallel lines closer than this dimension are ignored
  - maximum beam/wall width: beams/walls are not created if the lines are further apart than this dimension.

Use slab elements

When transferring to *STRAP* -

☑ - the program generates a single *STRAP* slab element for each space in a level

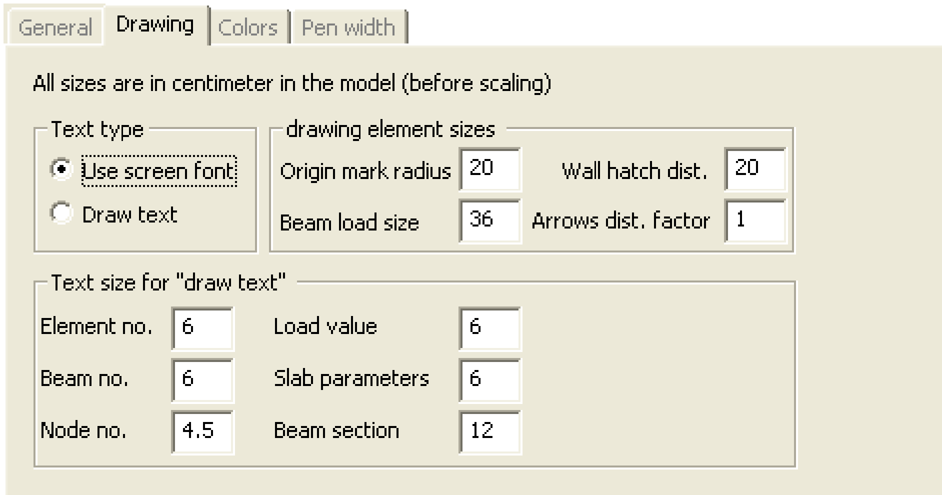
☐ - the program creates a mesh of rectangular and triangular finite elements.

Identify T beams

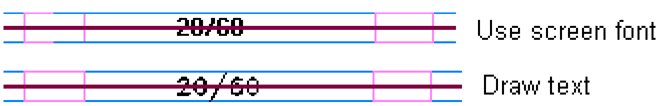
☐ the program creates all beams as rectangular and ignores the lines that define the flanges of T and L beams.

13.5.3.3.2 Drawing

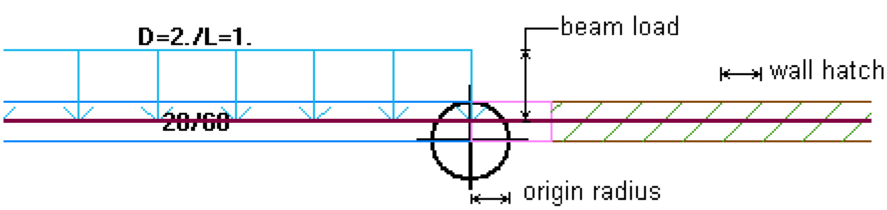
Specify the default drawing parameters. Note that if the parameters are revised the changes apply also to the current drawing and all existing drawings:



Text type

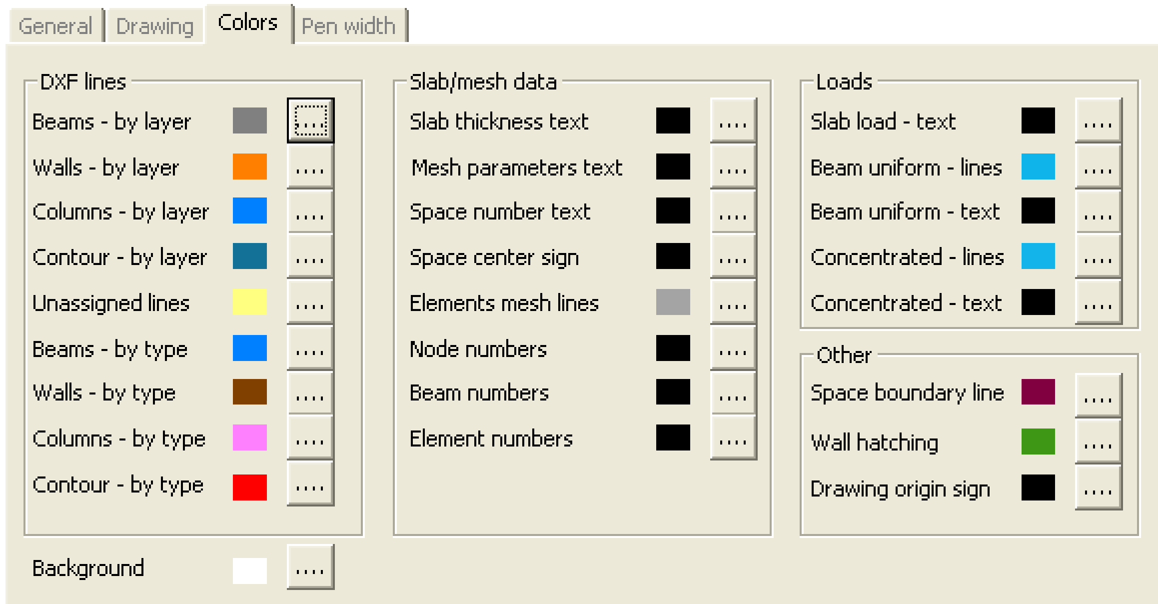


Element sizes



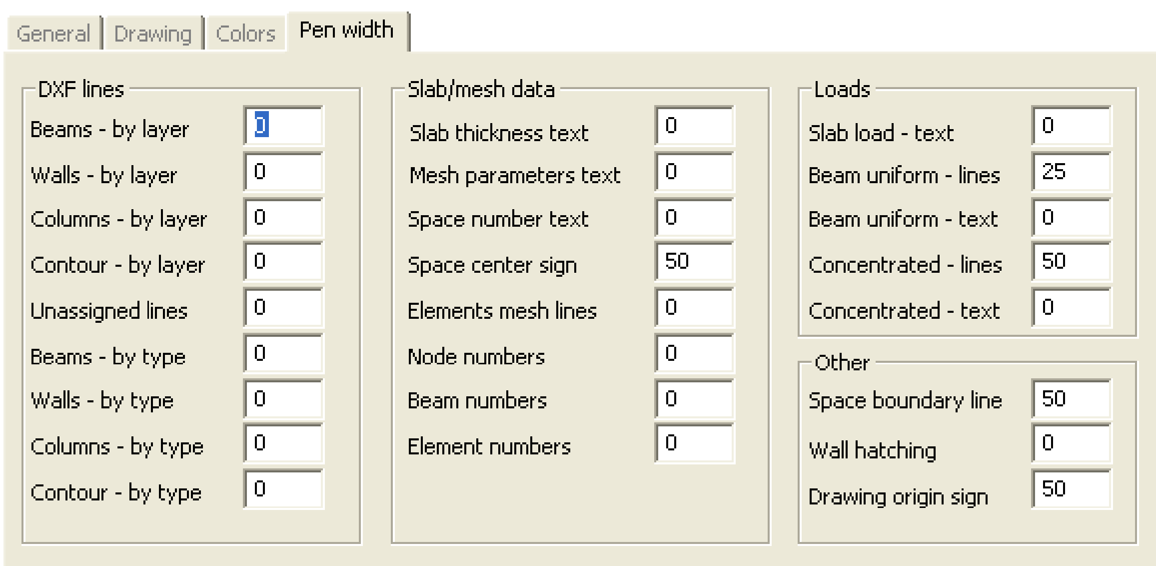
13.5.3.3.3 *Colors*

Specify the display color for the various line and text types on the drawing.



13.5.3.3.4 *Pen width*

Define the line thickness for the screen display:

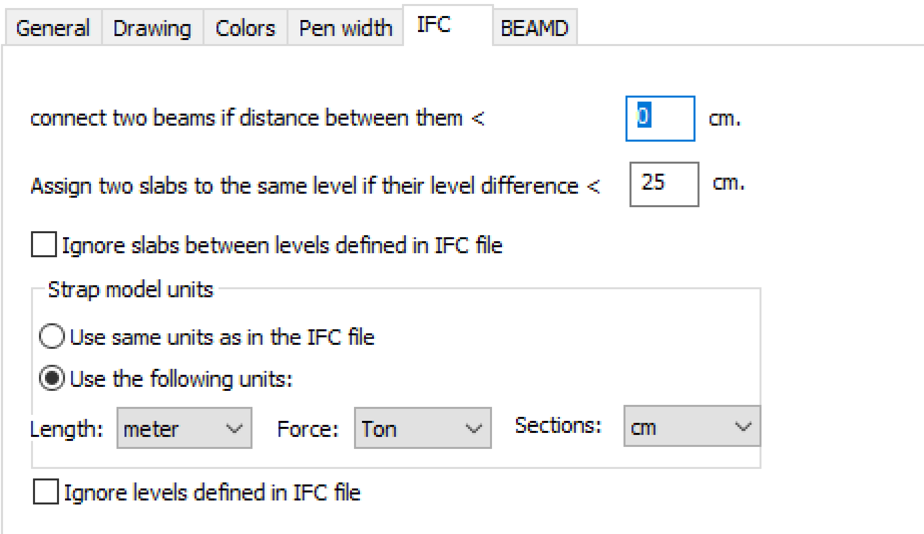


width = 0 signifies 1 pixel

13.5.3.3.5 *IFC*

Define the parameters for IFC Import. Refer also to IFC - Import notes 1432.

Note that revising the parameters will not affect any existing models; the IFC drawings must be imported again into a new project.



Connect two beams if the distance between them ...

Some BIM programs create beams with a small gap at their ends, therefore AutoSTRAP will not be able to identify two connected beams as being connected at a common node. In cases like this, increase the tolerance to the value that corrects the problem.

Assign two slabs to the same level ...

AutoSTRAP will create different levels in the STRAP model if the imported slabs are not exactly at the same elevation. Define a tolerance; slabs with an elevation difference less than this value will be defined by AutoSTRAP a the same level in the model.

Ignore slabs between levels

AutoSTRAP can detect IFC "slabs" at elevations that are not defined as "levels" in the IFC files. Select one of the following:

☑ - ignore the slabs (do not transfer to AutoSTRAP)

☐ - create an AutoSTRAP level at that elevation (and generate a plane of STRAP elements).

STRAP model units

Select the default units for the STRAP model: Use the same units as in the IFC file or specify units for length,force and section dimensions.

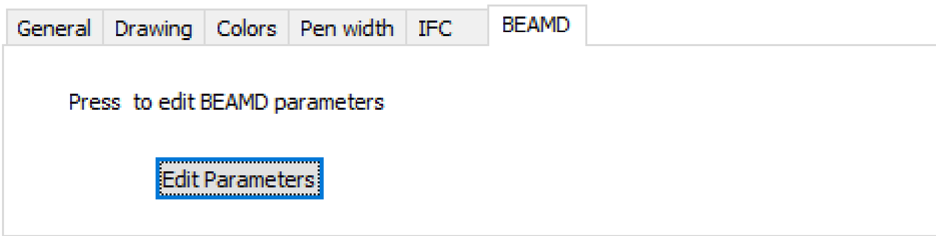
Ignore levels defined in IFC files

AutoSTRAP checks whether IFC "slabs" were created at each IFC "level". Select one of the following:

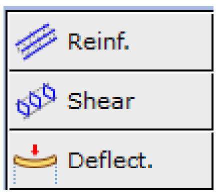
☑ - ignore all IFC objects attached to all IFC levels, including wall objects..
☐ - generate STRAP beams, walls, etc for the relevent IFC objects attached to IFC levels.

13.5.3.3.6 BEAMD

Define the default parameters for BEAMD reinforcement, shear and deflections:



- click on Edit parameters; the program displays the BEAMD design screen
- click on one of the icons in the side menu to display the options:

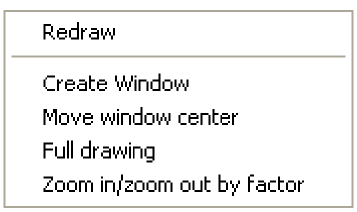


Refer to Reinforcement, Shear, Deflection for more information on the parameters.

- Click Back at the bottom of the side menu to return to AutoSTRAP.

13.5.4 Zoom options

Select one of the following options:



Create a window

Create a window by defining the lower-left and upper-right corners; the contents of the window will be displayed over the entire screen.

Move window centre

Relocate the centre of the current window to a new location. The relocated window is drawn with current options and scale.

Full drawing

Display the entire model according to the current display options.

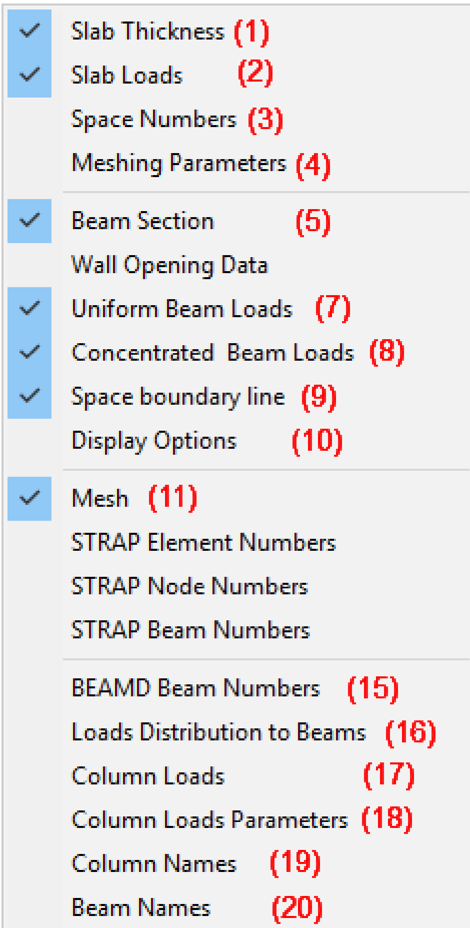
Zoom in/out by factor

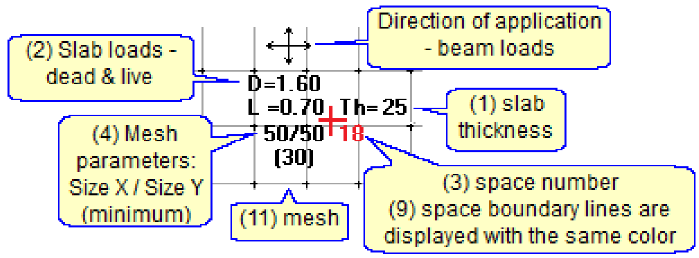
Zoom in on the current display; the center remains in the same location but the scale is changed. The degree of zoom is defined by entering the percentage scale change (a factor between 0 and 100%). Note that the percentage is the ***ratio between the change in scale to the final scale***.

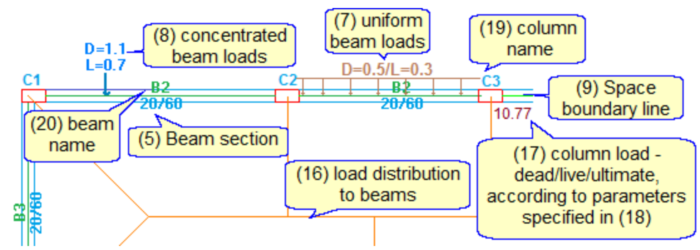
For example, to reduce the scale by half, select 100%.

13.5.5 Display options

Add/remove any of the following from the display:

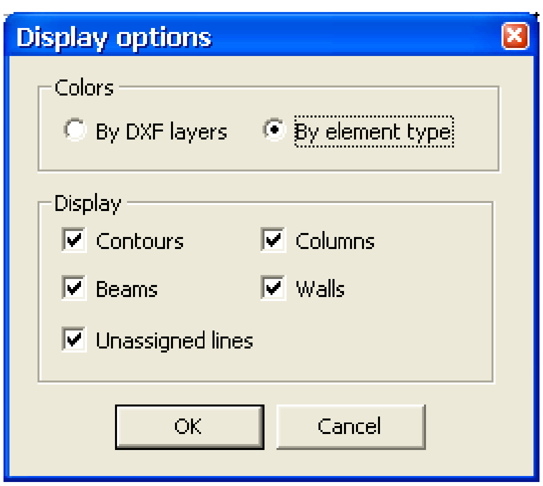






The (20) "beam name" can be replaced with the (15) "BEAMD beam number" - the index number of the beam in the BEAMD main screen beam list.

(10) Display options



Colours:

- By DXF layers:
  Display the lines in each DXF layer with a different color. The colors do not change when the drawing is modified.
- By element type:
  Display the lines of each element type (beams, columns, walls, etc) with a different colour. The colors are revised when the drawing is modified.

To define the colors, refer to File - Setup - Colors 1445.

Display:

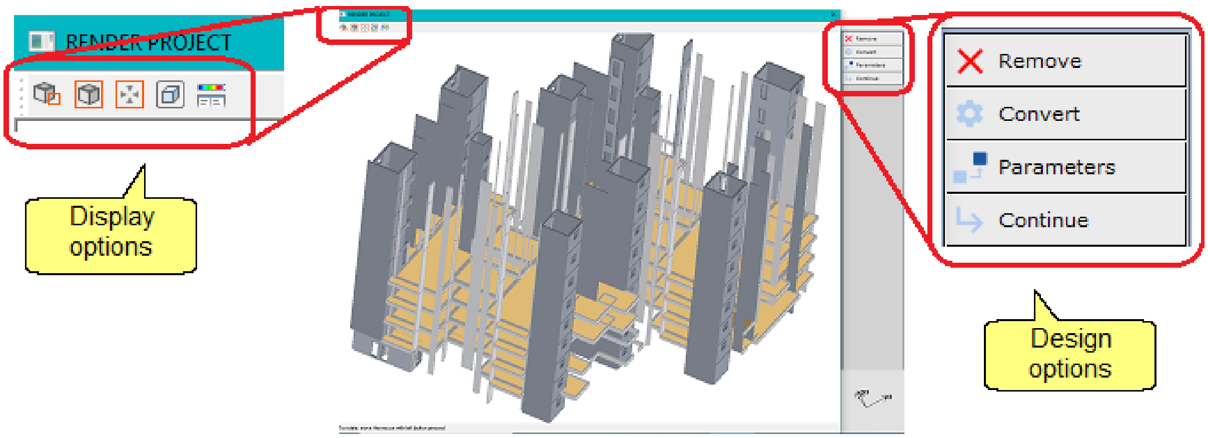
Selected element types may be removed from the drawing.

13.5.6 IFC options

After importing the IFC file, the program displays a rendered version of the model showing the following identified IFC objects: beams, slabs, plates and walls.
- Each object type is displayed with a different colour.
- Other object types are classified as "Removed elements". They may be added to the display and converted to identified elements, etc.
- Columns are classified as beam objects.

Options are available to:
- remove any of these objects from the *STRAP* model
- convert beams (columns) to walls, etc. Convert 'Removed elements' to beams, walls, slab, plates
- define various parameters for creating the model. Parameters for 'levels' defining the submodel locations are specified here:

For example:

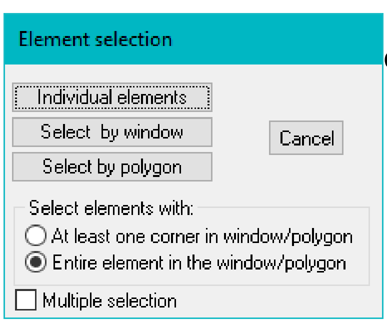


13.5.6.1 Design

Specify the elements to be included in the *STRAP* model and define design parameters for them. Select:

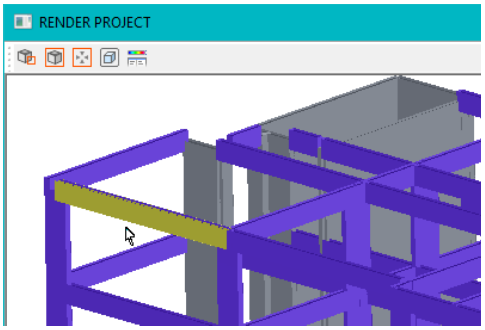
Remove

Remove beams, walls, plates and/or slabs from the STRAP model:



Options are similar to the standard STRAP selection options 1437.

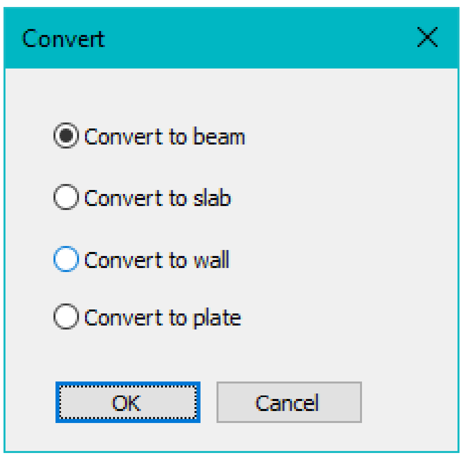
When selecting 'Individual elements', the program highlight the object nearest the : For example:



To restore deleted objects, first ☑ Display removed elements 1452, the use the Convert option

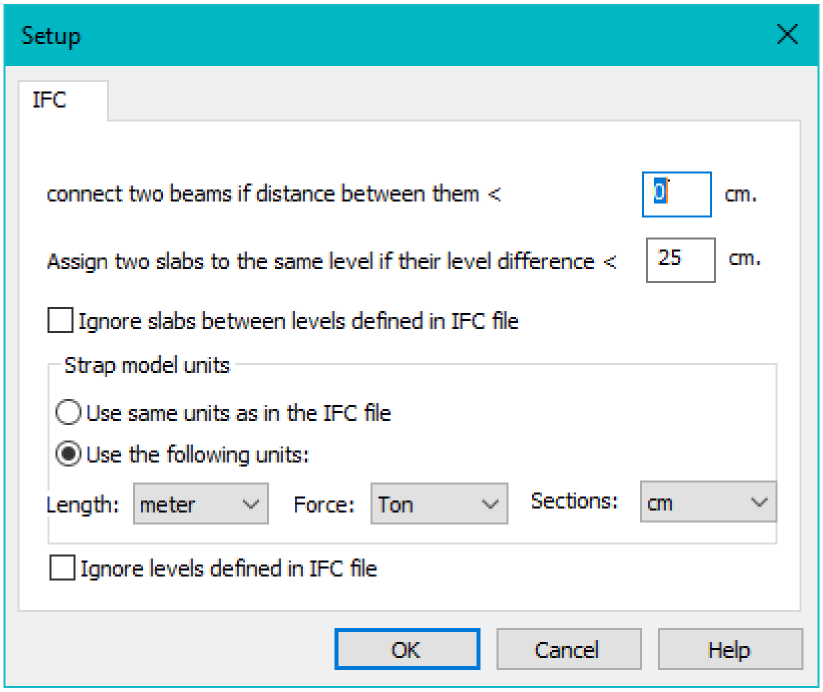
Convert

Convert walls to columns, beams to walls, etc. Specify the new type for the selected objects:



Removed elements can be restored using this option but first they must be displayed 1452.

Parameters



- ***Connect two beams if ...***

  Some BIM programs create beams with a small gap at their ends, therefore AutoSTRAP will not be able to identify two connected beams as being connected at a common node. In cases like this, increase the tolerance to the value that corrects the problem.

- ***Assign two slabs to the same level ...***

  AutoSTRAP will create different levels in the *STRAP* model if the imported slabs are not exactly at the same elevation. Define a tolerance; slabs with an elevation difference less than this value will be defined by AutoSTRAP at the same level in the model.

- ***Ignore slabs between IFC levels***

  IFC slab objects are sometimes not attached to IFC levels. For example: stair landings. Select one of the following:

  ☑ - ignore the slabs (do not transfer to AutoSTRAP)

  ☐ - create an AutoSTRAP level at that elevation (and generate a plane of STRAP elements).

- ***Ignore IFC levels***

  ☑ - ignore all IFC objects attached to all IFC levels, including wall objects..
  ☐ - generate STRAP beams, walls, etc for the relevent IFC objects attached to IFC levels.

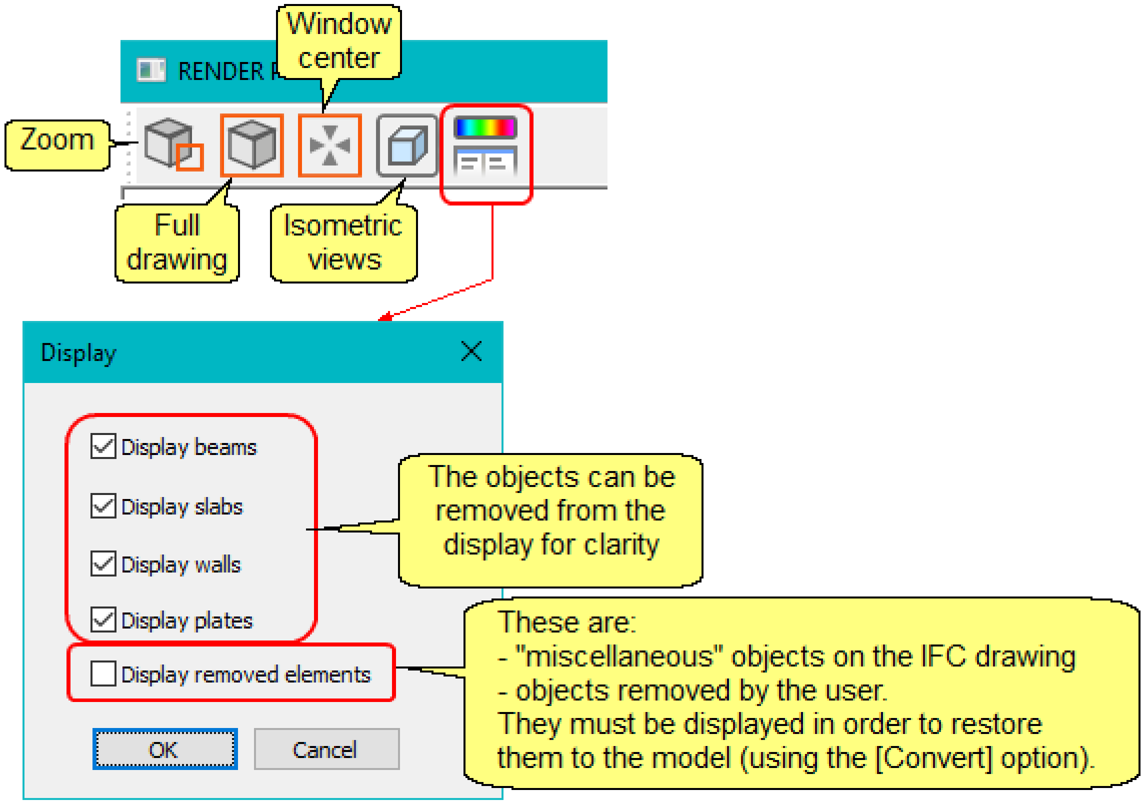
- ***STRAP model units***

  Use the same units as in the IFC file or select different units from the listboxes.

Continue

Create the preliminary *STRAP* model, The program displays the Parameters 1452 option (similar to those available for a model created from a DXF file).

13.5.6.2 Display



13.5.7 Parameters

Modify the drawing parameters:

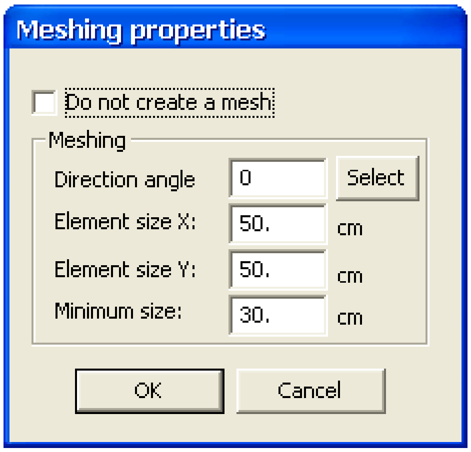
Mesh para...	The program generates an element mesh in the selected spaces 1423. The element size is determined according to the mesh parameters 1453
Thickness	Define the slab thickness 1454 for selected spaces 1423.
Beam sect...	The program can only determine the web and flange widths from the DXF drawing but not the beam depth and flange thickness. In addition the flange width required for analysis and design may not be available on the DXF drawing. Use this option to complete the beam dimension 1455 data.
Wall openings	The program identifies the wall openings in the DXF file. Use this option to modify these openings or to add new ones.
DXF files	Create a list of DXF files 1458 that are used to create the model. Each of the DXF files is used to create one or more levels in the structure.
Levels	For a 3D model: • define a series of floor levels 1459 and their elevations

- assign a DXF drawing to each level

For 3D drawings it is important to define the Origin 1460 point at each level; the program aligns all of the origin points at the same location along the height axis.

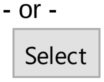
13.5.7.1 Mesh parameters

The program generates an element mesh in the selected spaces 1423. The element size is determined according to the following mesh parameters:



Direction angle

The element edges are drawn by default parallel and perpendicular to the X axis. To create the elements at a different angle:

- type in the value of the angle in the text box; the angle is measured counterclockwise from the base line. - or -
- click 

 and select any line on the drawing; the element edges are created parallel and perpendicular to the selected line.

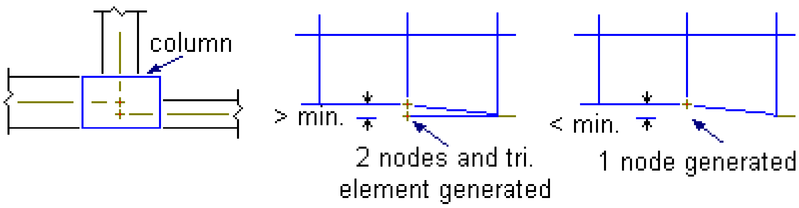
Element size

Specify the optimum size of the elements in the space. For example, if 1.0 x 1.0 elements are adequate for our example, set **Element size X** and **Element size Y** equal to 1.00.

Minimum size

Increasing this value prevents the generation of small elements; if a generated node on the mesh is less than the distance specified in this option from an existing node (x or y projection), then the program does not generate the node.

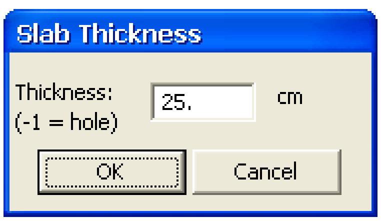
For example, offset beams meeting at a column:



13.5.7.2 Thickness

Define the slab thickness for selected spaces 1423:

- select 1437 one or more spaces
- enter the thickness (enter **-1** to delete all of the elements in a space)



13.5.7.3 Beam section

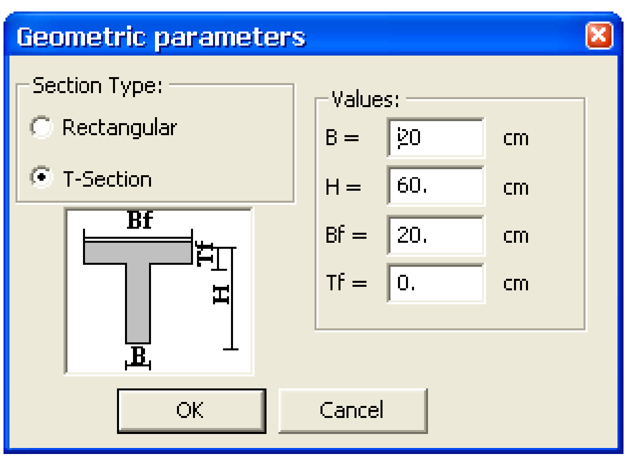
The program initially converts parallel sets of lines in the designated layers to beams:
- 2 parallel lines - rectangular bean
- 3 parallel lines - L-beam
- 4 parallel lines - T-beam

The program can only determine the web and flange widths from the DXF drawing the depth dimensions are not always available (note that the program can retrieve the beam depth from the drawing text; refer to DXF import - notes 1427) . In addition the flange width required for analysis and design may not be available on the DXF drawing.

Use this option to complete the beam dimension data:
- select one or more beams using the standard selection option 1437
- select one of the following options:

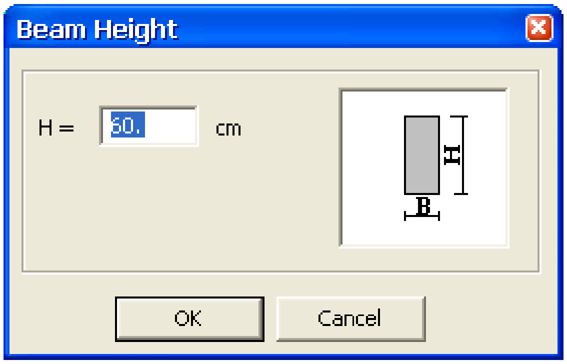
Define a section

Revise any or all of the section dimensions:



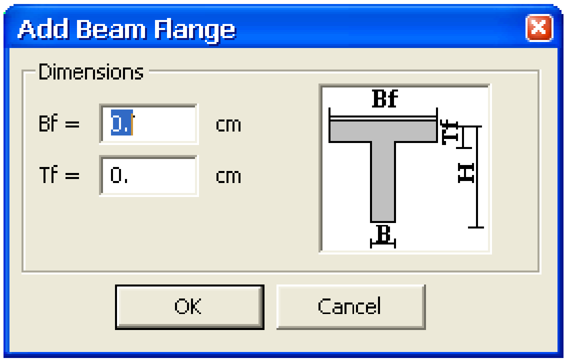
Change beam height

Define/revise the beam/flange height dimensions of the selected beams; beam/flange width dimensions are not modified.



Add a flange

Create a flange for a beam; beam width/height dimensions are not modified.

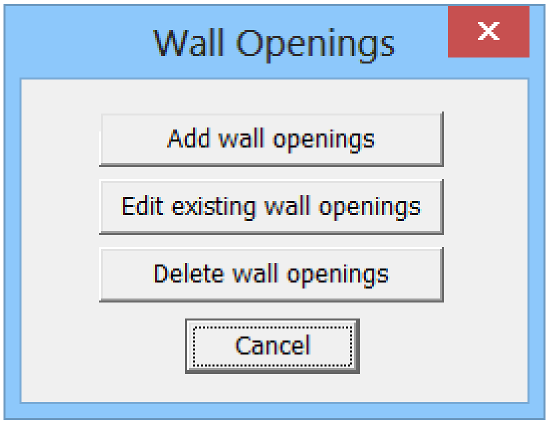


Remove added flanges

Remove the flanges from the selected beams, the web width/height dimensions are not modified.

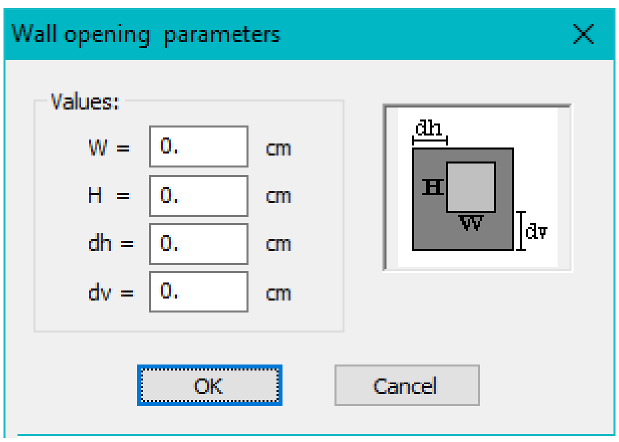
13.5.7.4 Wall openings

Add new wall openings or edit the dimensions of existing ones:



Add/edit wall openings:

- select 1437 one or more existing wall segments
- define the dimensions of the opening according to the sketch in the dialog box:

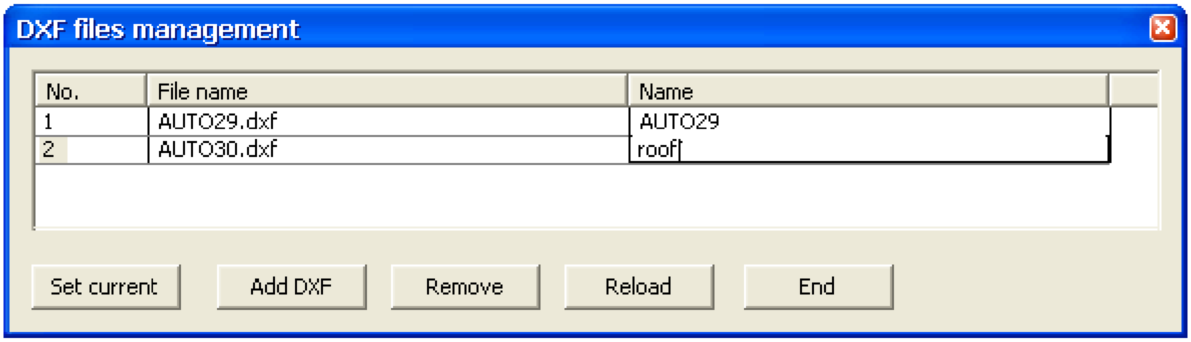


Delete wall openings

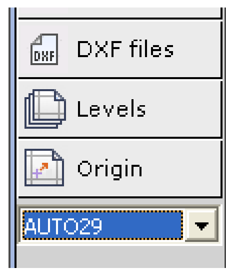
- select 1437 one or more existing wall segments

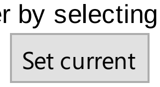
13.5.7.5 DXF files

Create a list of DXF files that are used to create the model. Each of the DXF files is used to create one or more levels in the structure:



The program displays a list of the DXF files in the side menu if more than one is added to the table in this option:



The display can be switched from one DXF to another by selecting one for this list or by selecting 

in this option.

Name

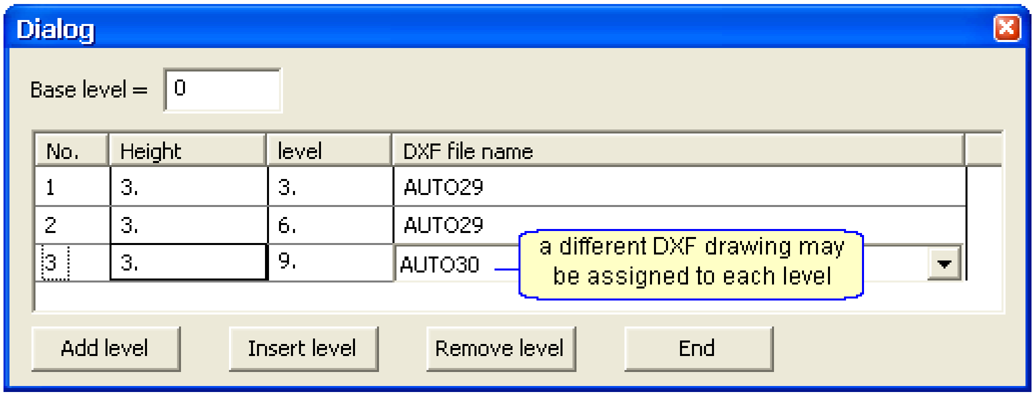
Enter a name for the DXF drawing. The name is displayed in the small DXF list below the side menu and is used as the submodel name in 3D *STRAP* models.

Reload

Use this option to reload a DXF drawing that has been modified. AutoSTRAP tries to maintain all revisions made in this program.

13.5.7.6 Levels

For a 3D model:
- define a series of floor levels and their elevations
- assign a DXF drawing to each level



DXF file name

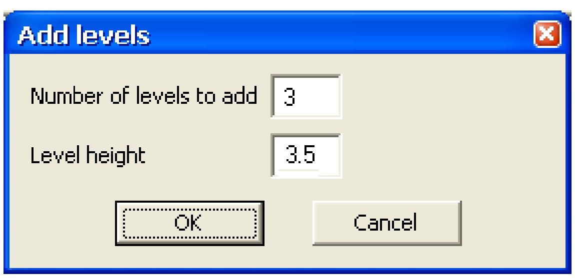
Assign a DXF to the highlighted level. Note that only DXF files added in the Parameters - DXF files 1458 option may be selected here.

Base level

All levels in the table are measured from this base. For example, if the base level in the table above is revised to **-2** the program automatically updates the levels from **3, 6, 9** to **1, 4, 7**

Add level

Levels are add to the end of the table. For the table above -



adds **12.5, 16, 19.5** to the end of the list.

Insert level

Levels are inserted ***before*** the highlighted line. For example, in the table above highlight line 2 and add 2 levels with height = 3.7 : The table is revised to 3.0, **6.7, 10.4**, 13.4, 16.4.

Remove level

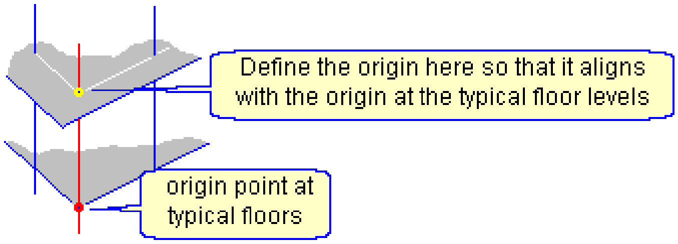
Highlight and click one row in the table; the following lines are updated, i.e, if the current levels are 3, 6.7 **, 10.4**, 13.4, 16.4 and 10.4 is deleted, the list is revised to **3.0, 6.7, 9.7, 12.7**.

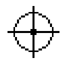
Note:
- the levels are aligned relative to each other in the Origin 1460 option

13.5.7.7 Origin

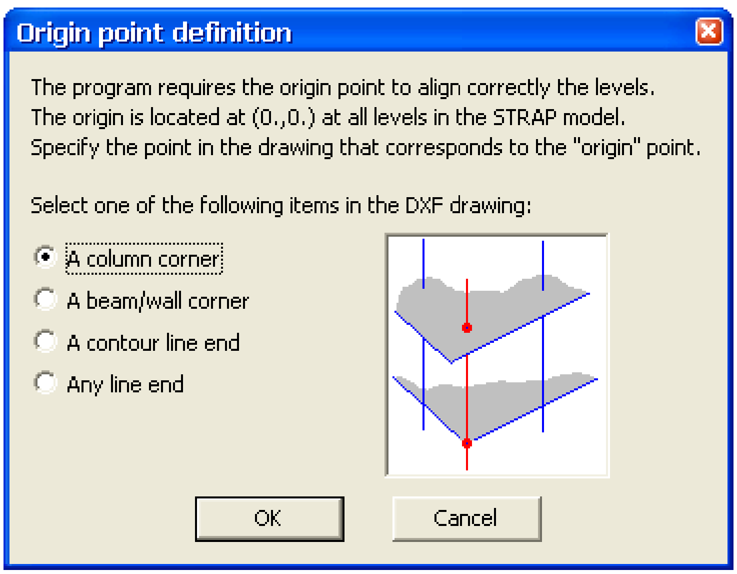
For 3D drawings it is important to define the ***Origin*** point at each level 1459; the program aligns all of the origin points at the same location along the height axis.

For example, one level in the model has a cantilevered balcony:



The origin point is identified by a  symbol on the drawing (the size of the symbol may be modified in the Files - Setup - Drawing 1444 option.

The origin point may be located at the end of any line on the drawing. Specify the line type to select:



13.5.8 DXF lines

In many cases the program's conversion of the DXF file to a structural model will not be complete, particularly in irregular structures, i.e. the program may either fail to identify beams, columns and walls or it may create these elements where none exist.

These options allow you correct and complete the model by assigning an element type to any lines on the drawing or by removing the element type from selected lines. For example:

- a beam may be converted to a wall
- unassigned lines may be defined as a beam, column, etc.

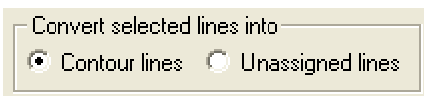
Note that the lines are not deleted from the drawing; only the type assignment is deleted.

Select one of the following options:

Select ***pairs*** of lines and define them as beams. If the lines are currently a different type (walls, columns, etc), the program prompts for confirmation.

Add wall	Select ***pairs*** of lines and define them as walls. If the lines are currently a different type (beams, columns, etc), the program prompts for confirmation.
Add column	Select all lines forming the boundary of the column. If the lines are currently a different type (beams, walls, etc), the program prompts for confirmation.
Add contour	Select lines to convert them to "contour lines". The program recalculates the space 1423 boundaries.
Delete beam	Select one line in a beam to delete it. The program converts the line to "contour lines" or "unassigned lines" (see below) to maintain the spaces.
Delete wall	Select one line in a wall to delete it. The program converts the line to "contour lines" or "unassigned lines" (see below) to maintain the spaces.
Delete col...	Select one line in a column to delete it. The program converts the line to "contour lines" or "unassigned lines" (see below) to maintain the spaces.
Delete con...	Select contour lines. The program converts the lines to "Unassigned".

Note:
- when deleting beams/columns/walls, the element lines may be converted to either contour lines or unassigned lines; the following option is displayed in addition to the regular selection options 1437:



13.5.9 Loads

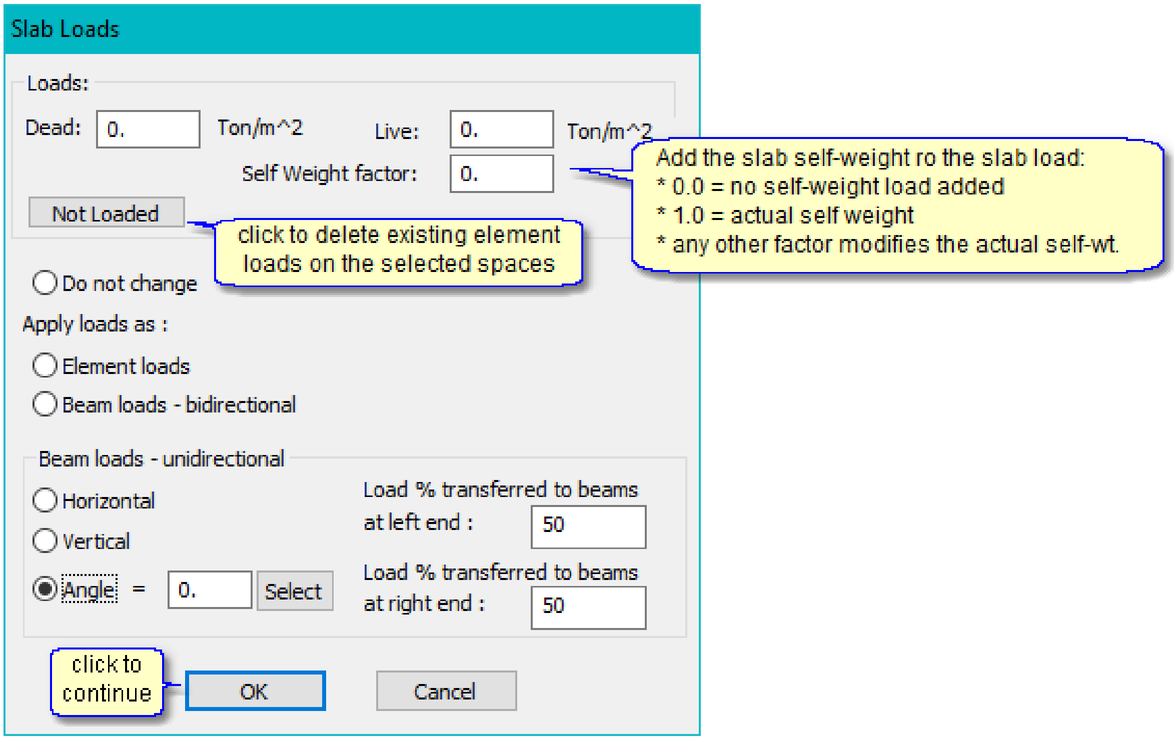
Two *STRAP* load cases are created:
- Dead loads
- Live loads

Select one of the following load types:

Slab load	Define a uniform element load on selected "Spaces 1423". The load is applied to all elements in the space.
Uniform	Define a uniform beam load on selected beams.
Copy load	Define a point load at any location a long a beam
Deactivate	Delete any existing load.

13.5.9.1 Slab load

Define a uniform element load on all elements in a selected 'space' or beam loads on the surrounding beams. The self-weight of the slab (modified by a factor) can also be applied in addition to the uniform load:
- Select the spaces
- Define the dead and live load values.



The slab loads can be applied to the elements or to the supporting beams (on all edges or the edges in one direction only). Select one of the following:

Element loads

The defined loads are applied to all the elements in the space.

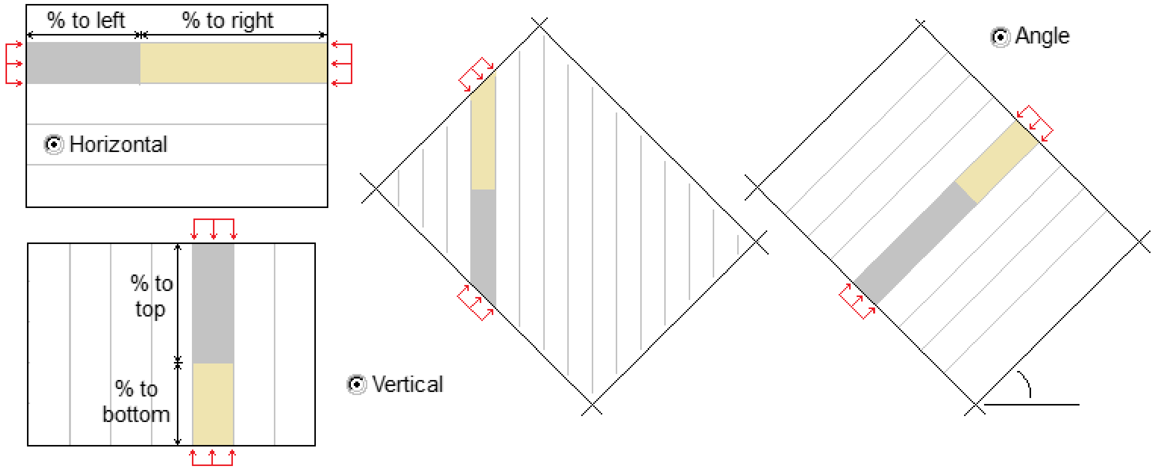
Beam loads - bidirectional

The loads are applied to the surrounding beams using the same method that global loads are converted to beam loads. Refer to Global loads - Method of application.

Beam loads - unidirectional

The program calculates the beam loads by dividing the slab area into a series of strips which many be defined in any direction:

- "Horizontal" and "Vertical" refer to the screen directions; while the "angle option allows strips to be defined in any direction.
- By default 50% of the load is applied to the support at either end but any distribution may be defined.

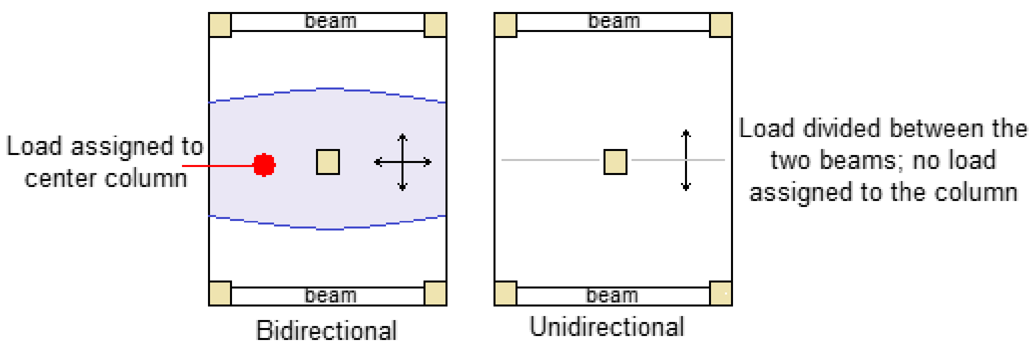


To define an angle:

- enter a value (measure counter-clockwise from the horizontal), or -
- click Select and select a DXF line; the strips will be parallel to this line.

Note:

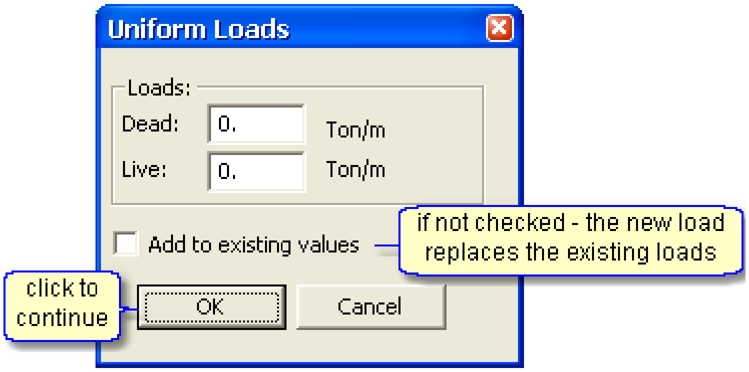
- if **Beam loads - unidirectional** is selected, the program will not assign the load correctly to columns that are not attached to beams. For example:



13.5.9.2 Uniform load

Define a uniform ***beam load*** on all selected beams:

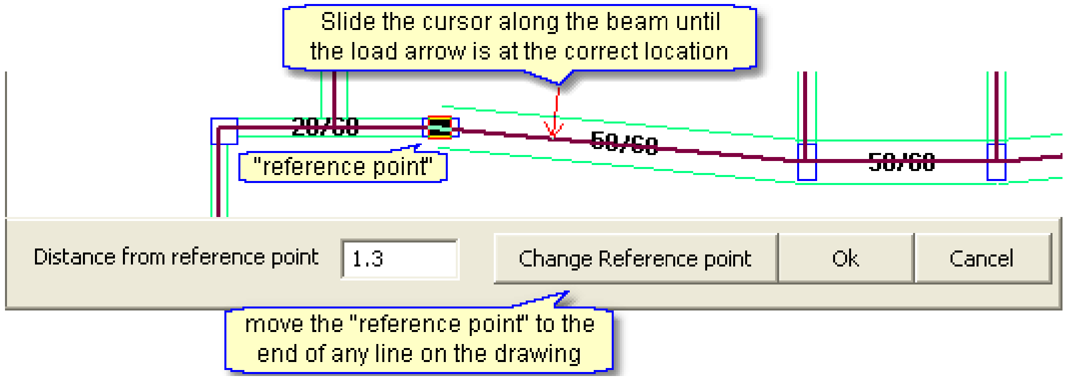
- Select the beams
- Define the dead and live load values.


Uniform Loads
Loads:
Dead:
0.
Ton/m
Live:
0.
Ton/m
Add to existing values
if not checked - the new load replaces the existing loads
click to continue
OK
Cancel

13.5.9.3 Point load

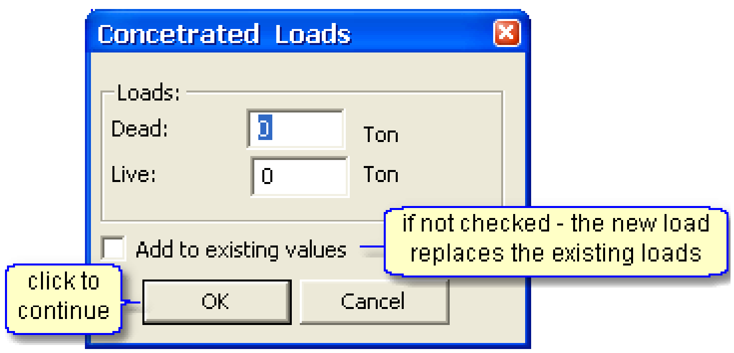
Define a beam point load on all selected beams:

- Select the beams
- Define the load location (for the first beam selected):



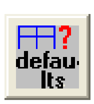
Note that the program applies the load at the same distance along all of the selected beams, even if they have different lengths

- Define the dead and live load values. Note that the program creates two *STRAP* load cases.

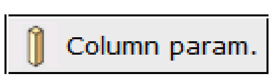


13.5.10 Create the STRAP model

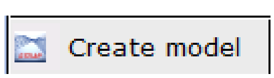
Create a *STRAP* model from the current drawing(s):



Specify default parameters for rigid links, offsets, supports, etc in the model that is created by the program. The defaults apply to all columns, beams, walls, etc.



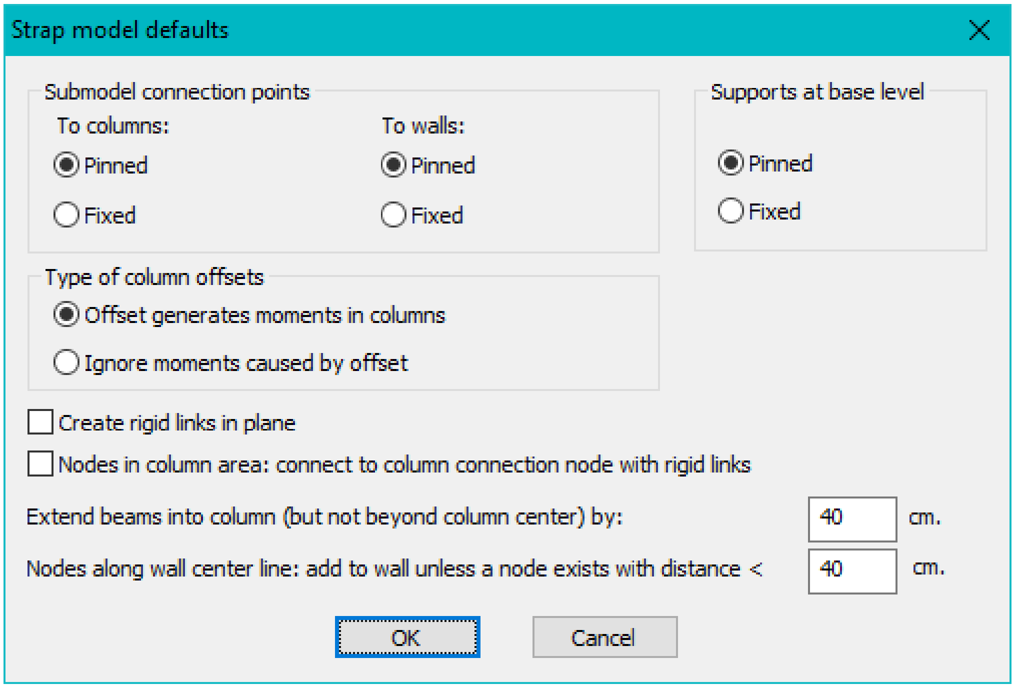
Specify parameters for selected columns. The program will use the default parameters 1465 if no parameters are specified using this option.



Create a *STRAP* model from the current drawing(s).

13.5.10.1 Defaults

Specify default parameters for rigid links, offsets, supports, etc in the model that is created by the program. The defaults apply to all columns, beams, walls, etc. To specify different parameters for selected columns, refer to Column parameters 1468.



Submodel connection points

The submodel is connected to the main model at common nodes called ***Connection points***. Use this option to define the types of connection - fixed or pinned - to columns and walls. To specify different parameters for selected columns, refer to Column parameters 1468.

Supports at base level

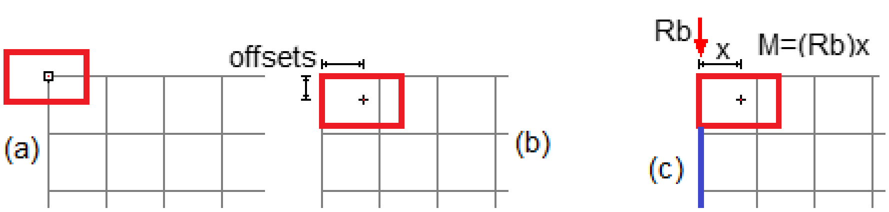
The program extends all columns and walls found at the lowest level to the base level 1459 and defines supports. ***All*** of the supports may defined as either fixed or pinned.

Note:

- Individual supports can be revised only in *STRAP* geometry.

Column offsets

Offsets sometimes generate unwanted moments in attached beams and columns. For example, the column in Figure (a) is defined at the slab corner node but the column faces are in reality aligned with the slab edge as shown in (b):



If the offset as shown in (b) is defined, the column will appear at the correct location in all of the drawings but an additional column moment will be generated equal to product of the slab reactions and the offset distance.

To ignore these additional moments, select **Ignore moment caused by offset** .

However if there are beams attached to the column as shown in (c), the beam reactions are the main column loads and the moment caused by the eccentric beam reaction should be applied.

To specify different parameters for selected columns, refer to Column parameters 1468.

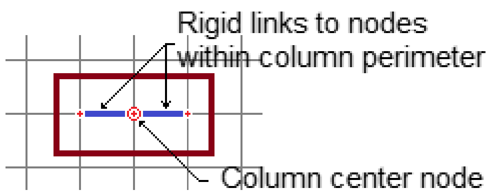
Create rigid links in plane

☑ Link ***all*** nodes in the plane with rigid links. This option is useful for models where dynamic analysis is required. Do not use this option is the level contains two or more unconnected parts.

☐ The rigid links required for offset columns and for interior wall nodes are still created by the program.

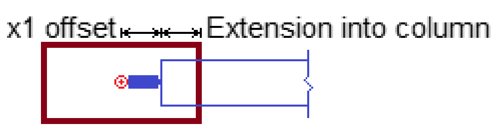
Nodes in column area

This option applies to grid nodes that lie within column perimeters and only when there is a fixed connection to the column. Rigid links that connect these nodes to the column center node may be created. For example:



Extend beams

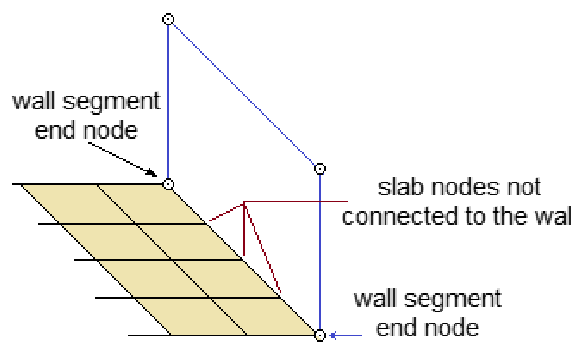
Create an x1 offset at the end of beams by defining the distance the beams extend into the columns:



To specify different parameters for selected columns, refer to Column parameters 1468.

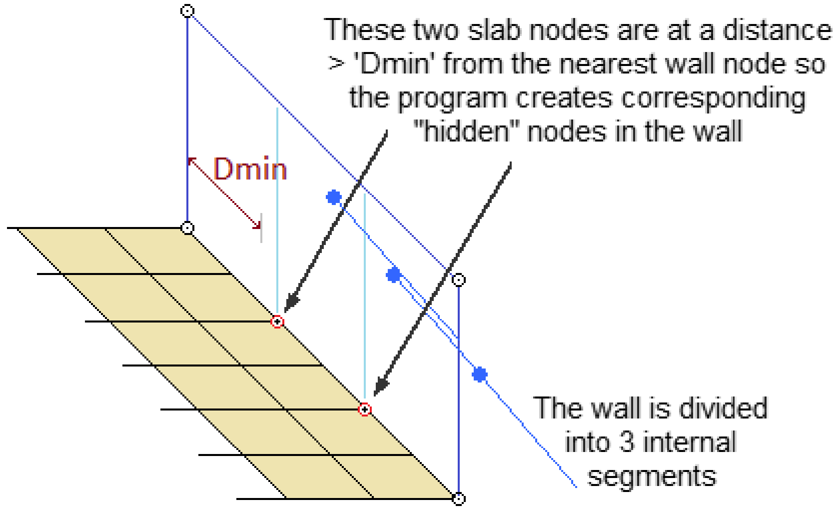
Nodes along wall

The wall segments are connected to the adjacent parts of the model only at their end nodes. However there may be existing nodes in the model that lie along the wall segment boundary but are not connected to the wall. For example:



The wall may not move together with the slab along the boundary when either is loaded. To deal with this problem the program can create 'hidden nodes' in the wall segment at the unattached model node locations, and then connect the wall to the model at those nodes.

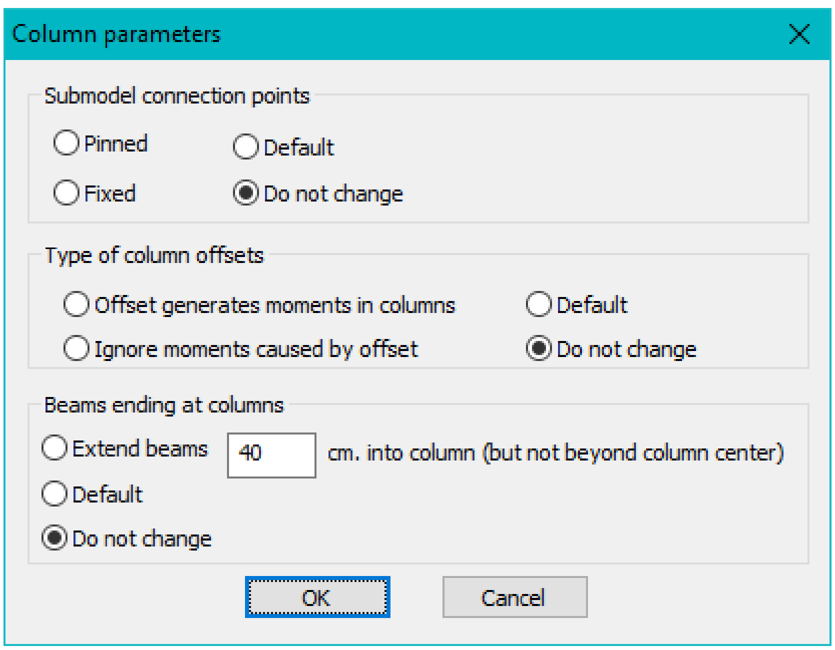
These 'hidden nodes' are placed so that no unattached slab node is further than "**Dmin**" from a wall node (a corner node or another "hidden node") .



- The default value of **Dmin** = 0.4 m;

13.5.10.2 Column parameters

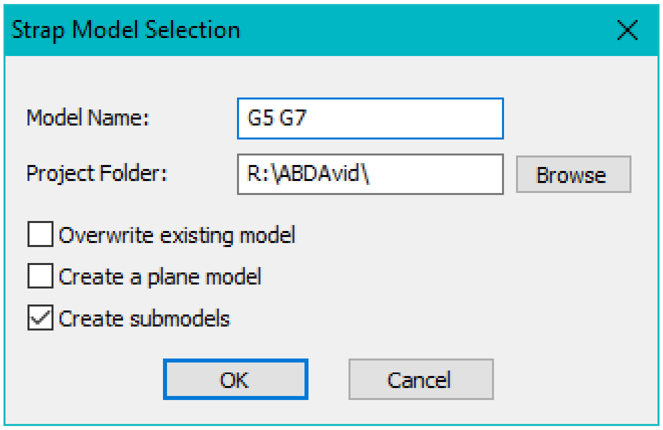
Use this option to specify parameters for selected columns. The program will use the default parameters 1465 if no parameters are specified using this option.



Select **Default** to restore the default parameters to selected columns.

13.5.10.3 Create model

Create a *STRAP* model from the current drawing(s):



Model directory/name

The program creates the model in the selected folder and adds the "Model name" to the model list.

Overwrite existing model

The program searches the selected folder for an existing model that has the identical "Model name".

Create a plane model

☑ - the program creates a plane model from the currently displayed DXF drawing only and ignores other "levels".

☐ - the program creates a 3D drawing consisting of all levels and the columns that connect them.

Create submodels

☑ - the program creates a submodel for every level (this is the default option)

☐ - the program creates a model without submodels (the entire structure is in the 'Main model').

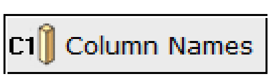
Note:

- the levels are always created on the *STRAP* X1-X2 plane and X3 is always the height axis.
- irregular shaped columns are transferred to *STRAP* as "*CROSEC*" sections and can be designed as "Solid sections" in the Concrete design module.

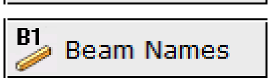
For more detailed information, refer to STRAP model - notes [1433].

13.5.11 BEAMD

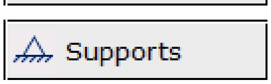
Beams are automatically identified and created by the program and can be desined and detailed by he BEAMD program. Select one of the following options:



revise column names [1470] (column names can be used when creating beam names)



revise beam names [1471]



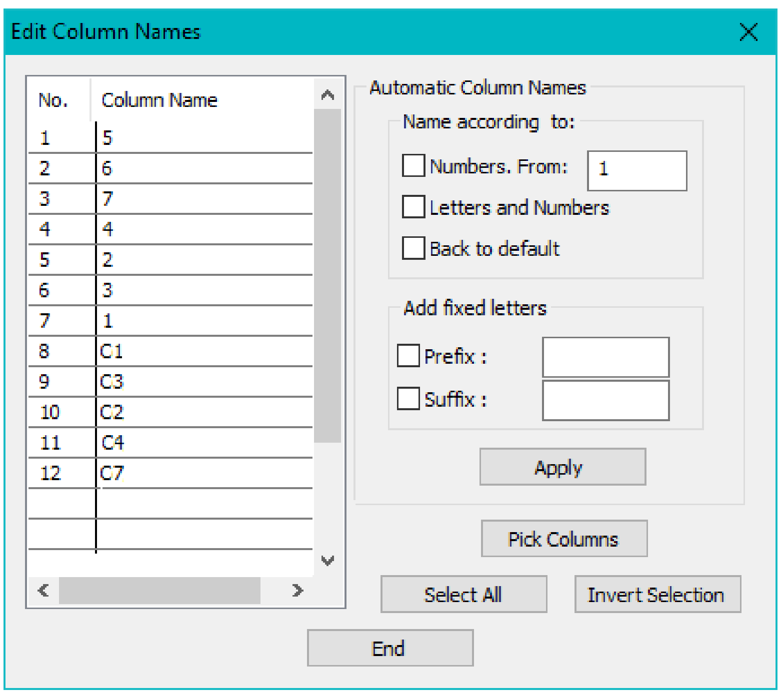
modify or delete beam supports [1472]

Edit Beams	Edit[1473] beams: split or combine beams.
Delete	View the beams in BEAMD and modify the design.
Detail	Detail[1474] the reinforcement in beams transferred to BEAMD.
Col. Reactions	Display a table showing the loads applied to columns[1475]
Ribs	Define a typical rib[1476] spanning between two beams/walls
Beamd	Transfer beams to BEAMD[1478] (create the files)

13.5.11.1 Column names

The program can identify column names in the DXF drawing. It searches for text in the column layer that is adjacent to the column. If names are not found, the program assigns the names1, 2, 3,....., etc, to the columns, starting from the lower-left corner of the drawing.

Revise the column numbers, add a prefix, suffix, etc.



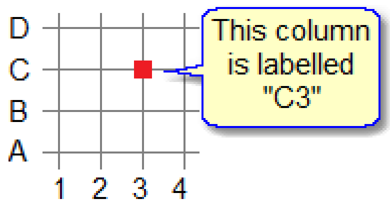
To revise individual column names:
- click and highlight a name in the table and type in a new one.

To revise several names systematically:
- highlight the rows in the table where the names are to be revised. There are several methods:
  - click and highlight the rows in the table

- click Pick columns and select columns on the drawing (the program will renumber columns in the order that you select them)
- click Select all to revise the names for all columns

- Select one of the following numbering methods:
  - **Numbers** to renumber sequentially and enter the start ("from") number
  - **Letters and numbers**:

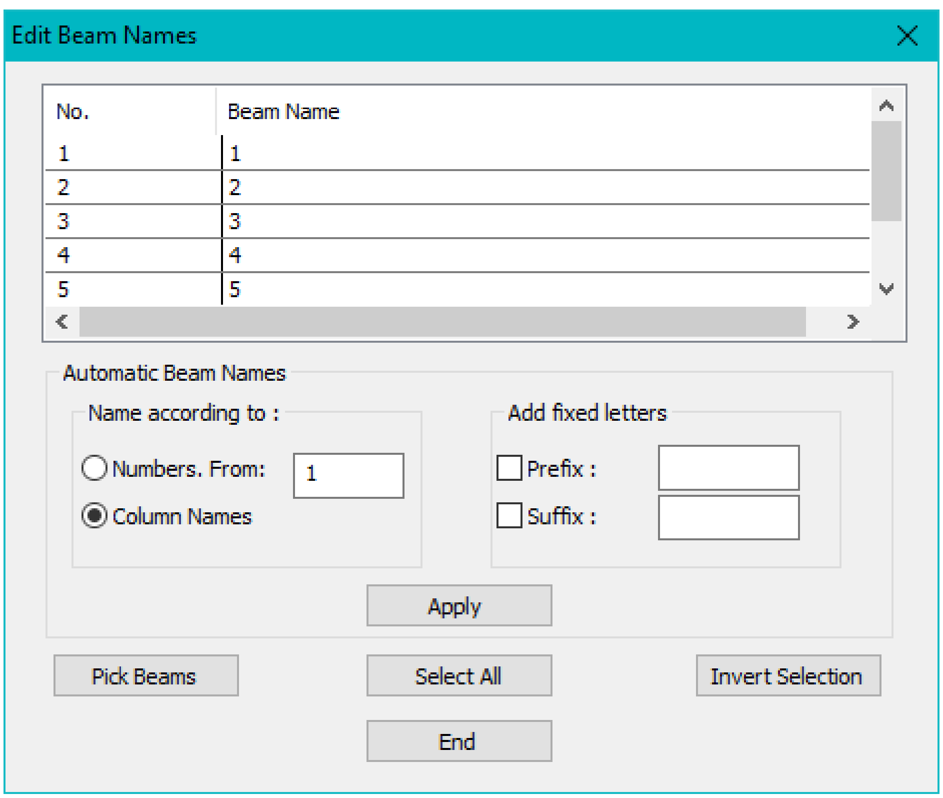
    the program scans the drawing and identifies all column locations, then assigns grid names as follows:



  - **Back to default** - to restore the original column names (1,2,3,... or the names in the DXF drawing).
- Add a **Prefix** or a **Suffix** string to the names of all the selected columns
- click Apply

13.5.11.2 Beam names

By default the beams are named 1, 2, 3,....., etc. Revise the beam numbers, add a prefix, suffix, etc.



To revise individual beam names:

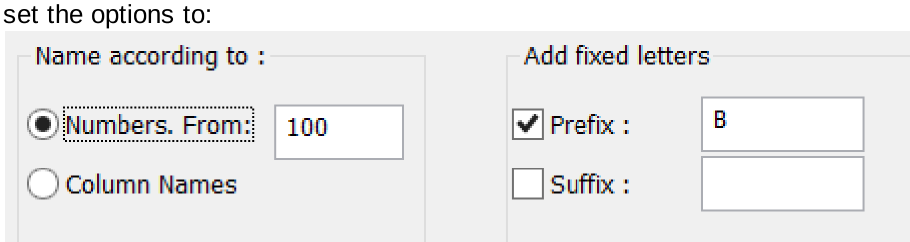
- click and highlight a name in the table and type in a new one.

To revise several names systematically:

- highlight the rows in the table where the names are to be revised. There are several methods:
  - click and highlight the rows in the table
  - click Pick beams and select beams on the drawing (the beams will be renumbered in the order you select them).
  - click Select all to revise the names for all beams
- Select **Numbers** to renumber sequentially and enter the start number
  Select **Column names** to rename with a list of column names 1470 at the supports
- Add a **Prefix** or a **Suffix** string to the names of all the selected beams
- click Apply

For example:

- click and highlight rows 2 and 4
- set the options to:



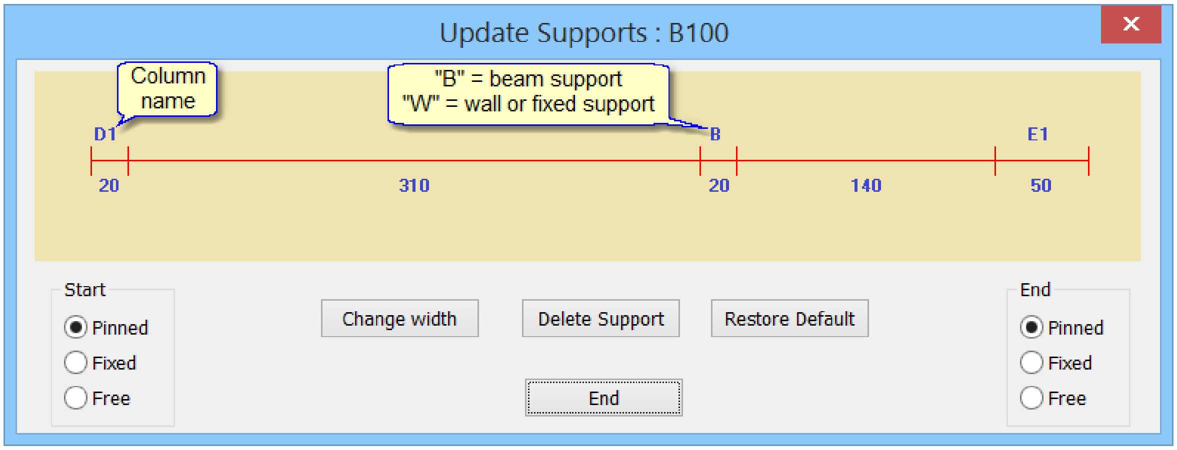
- the names in the table are revised to:

No.	Beam Name
1	1
2	B100
3	3
4	B101

13.5.11.3 Supports

Revise support width or delete supports for selected beams:

- select one or more beams using the standard beam selection option. The program will display the following screen for each beam selected.
- change the end support, revise the support width or delete a support:



- click Change width , Delete support or Restore default and select a support.
- change the exterior supports to Pinned, Fixed or Free

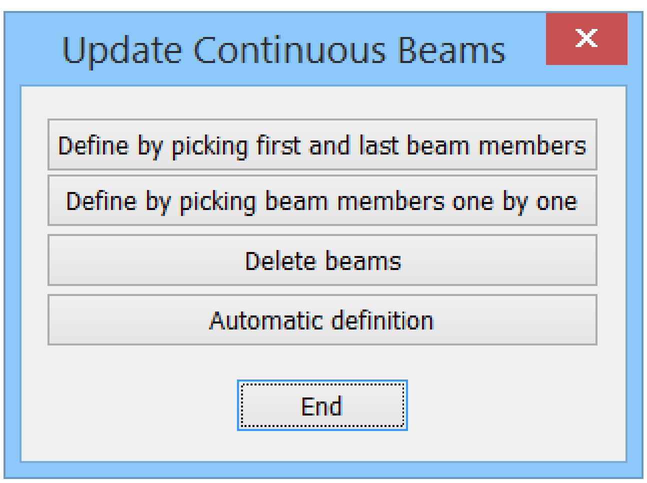
Note:
- a beam end supported on a wall is initially assumed to be "fixed"; all other exterior supports are assumed "pinned"/

13.5.11.4 Edit beams

The program automatically creates continuous beams by identifying connected spans that form a continuous line.

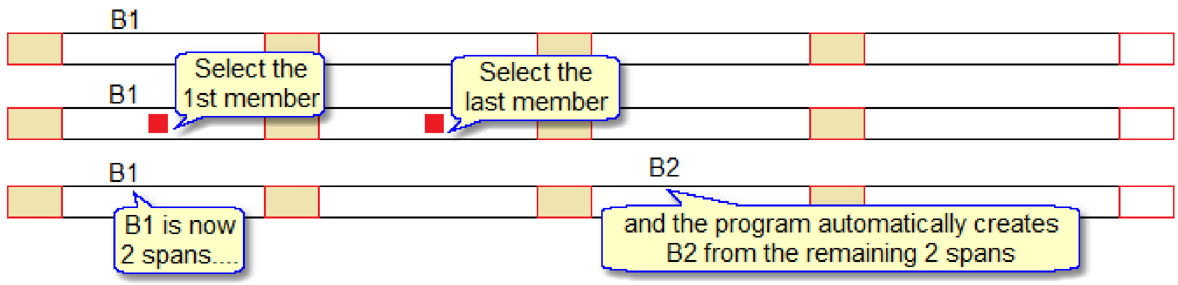
Note:
- a span is considered as a continuation of the previous span if the angle between the axes of the two members is less than 30°.

Select one of the following options:



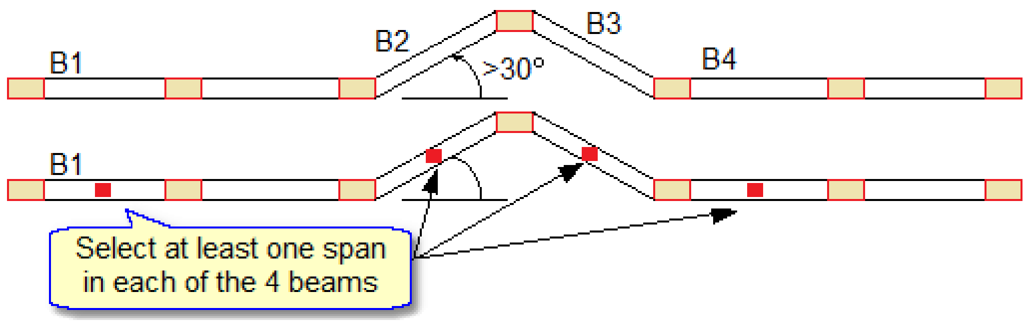
- Define by picking first and last beam members

  Use this option to split beams. For example, split the following beam B1 into two beams, each with 2 spans:



- Define by picking beam members one by one

  Use this option to combine beams. For example, the beam initially creates 4 beams because the angle is > 30°:



- Automatic definition

  Restore the default definition of selected beams.

- Delete beams

  Select beams to delete; they will not be transferred to *BEAMD*.

13.5.11.5 Design

Design and detail one of the beams in the *BEAMD* program:

- move the adjacent to any beam and click the mouse:

Note:

- the *BEAMD* files must first be created using the Beamd option.

13.5.11.6 Detail

The program details the reinforcement in the beams according to the Code requirements and user-defined parameters.

Select one or more beams and specify the detailing parameters.

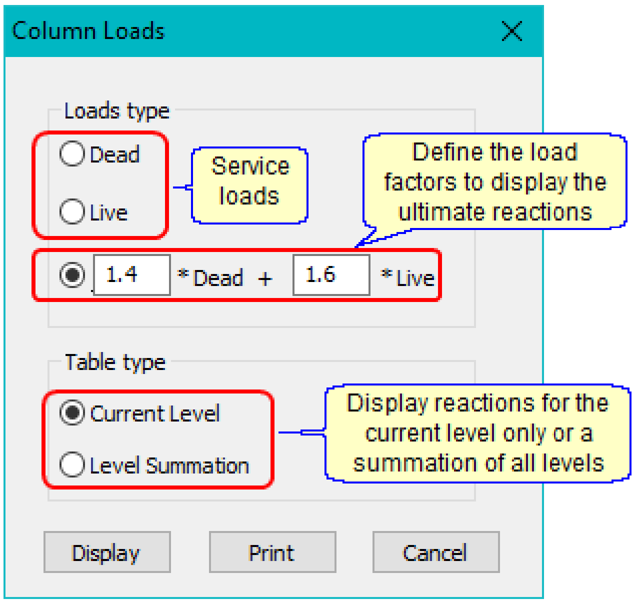
Note:

- beams can also be detailed when they are transferred to BEAMD 1478 if the ☑ **Detail the transferred beams** option is selected.
- The detailing can be viewed and revised in the *BEAMD* program.

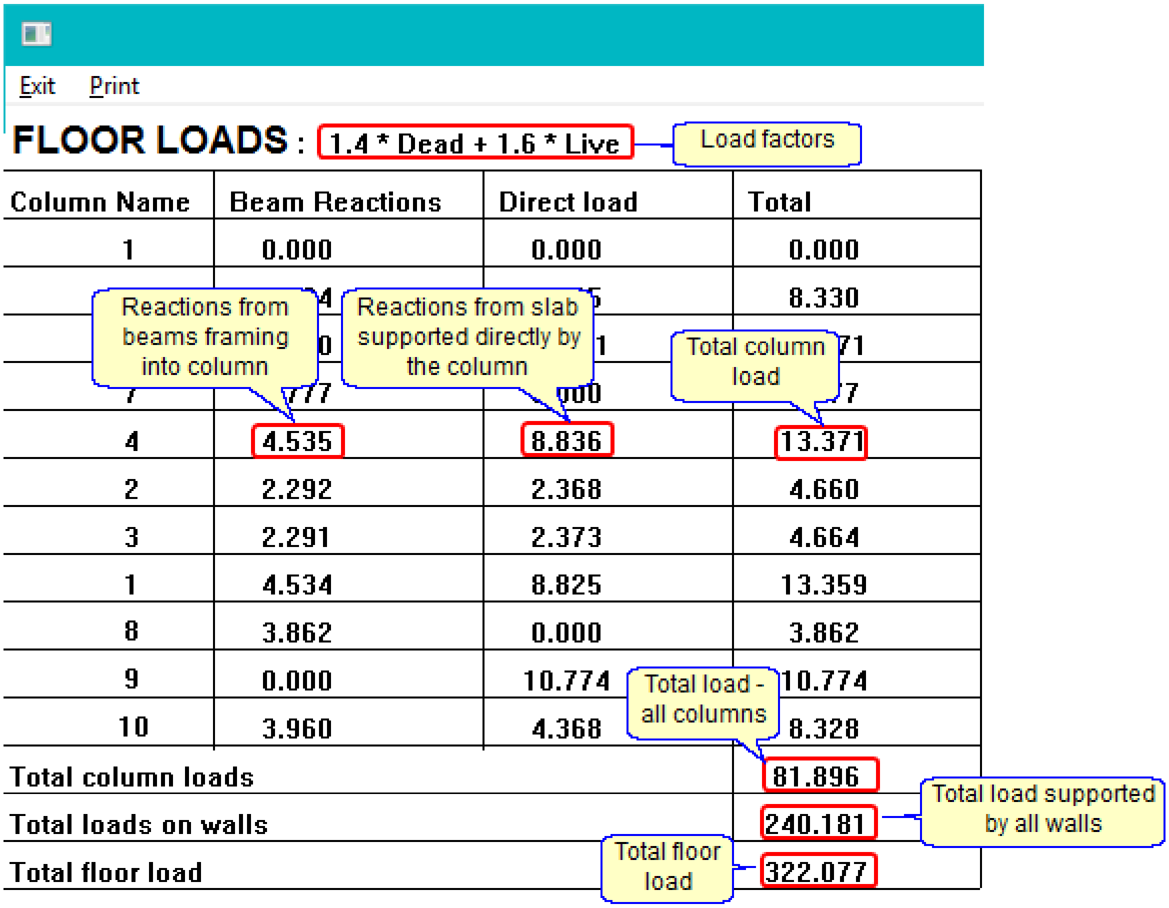
13.5.11.7 Column reactions

Display a table of the load reaction at every column

- the reactions from both supported beams and slabs are calculated.
- the reactions at the current level or the sum of the reactions for all levels may be displayed.
- service dead load, service live load or ultimate total load reactions may be displayed.
- the sum of the loads supported by the walls are also calculated.
- the column reactions may also be added to the graphic display by selecting Display - column loads 1448.



For example:



FLOOR LOADS : 1.4 * Dead + 1.6 * Live

Column Name	Beam Reactions	Direct load	Total
1	0.000	0.000	0.000
[illegible]	[illegible]4	[illegible]5	8.330
[illegible]	[illegible]0	[illegible]1	[illegible]71
7	[illegible]777	[illegible]000	[illegible]7
4	4.535	8.836	13.371
2	2.292	2.368	4.660
3	2.291	2.373	4.664
1	4.534	8.825	13.359
8	3.862	0.000	3.862
9	0.000	10.774	10.774
10	3.960	4.368	8.328
Total column loads			81.896
Total loads on walls			240.181
Total floor load			322.077

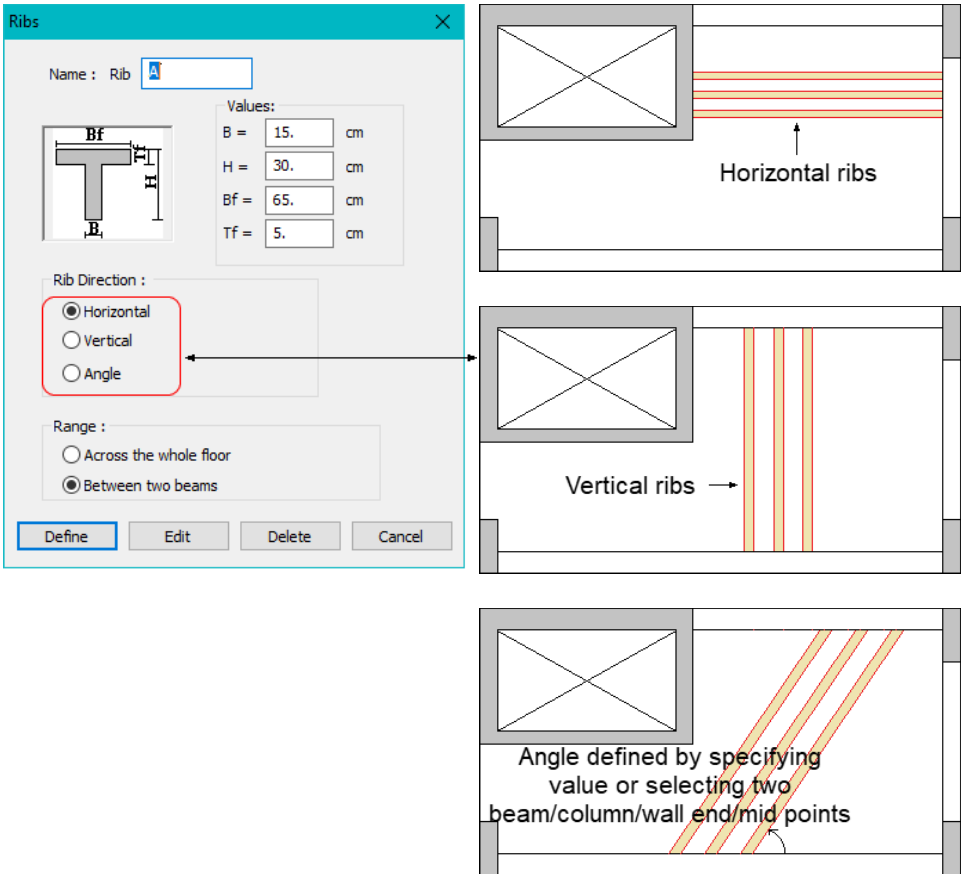
13.5.11.8 Ribs

Define a typical rib spanning between two beams/walls or across the entire slab.

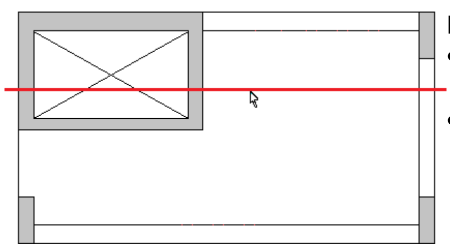
- the program calculates the load on the typical rib and the rib can be transferred to *BEAMD* for designing and detailing.

To define a rib:

- click Ribs in the side menu
- specify the dimensions and parameters:

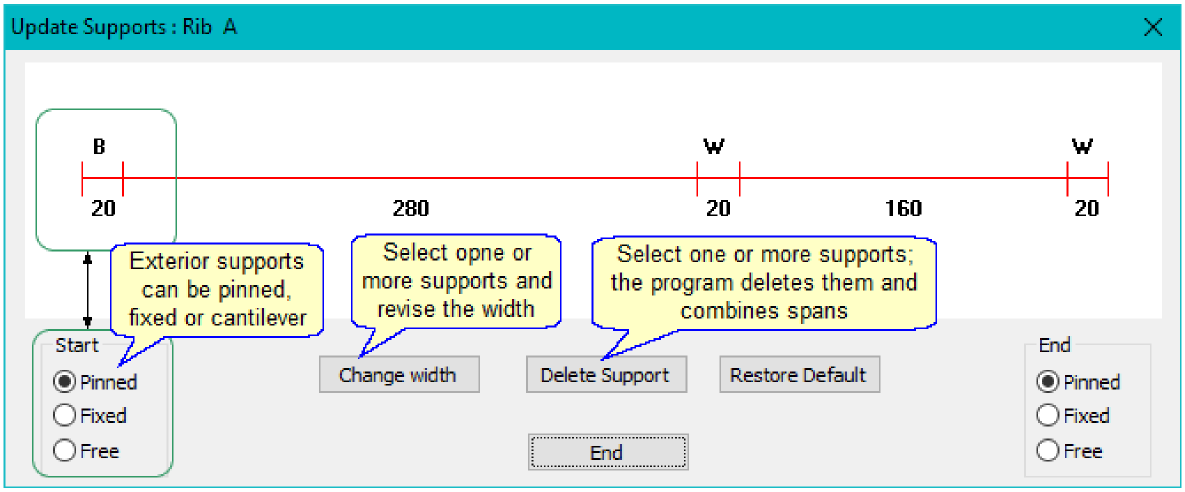


- click Define and select the rib location. For example, the "horizontal rib" shown above:



Note:
- if ◉ **Across the whole floor** is specified, the program will create a two-span rib, including the opening at the left
- to create the rib as shown (one span), specify ◉ **Between two beams** and select the wall at the left and the beam at the right

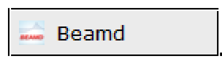
- the program then displays the span lengths and the supports width. Support widths can be revised, internal supports can be deleted, and end support conditions can be revised:



To revise the supports after the rib is defined, select 

in the side menu.

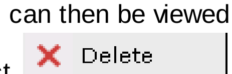
Note:

- the program assumes that the spacing between ribs is "Bf" and calculated the load on the rib accordingly.
- to create BEAMD files 1478 and detail the ribs, select 

.

13.5.11.9 BEAMD

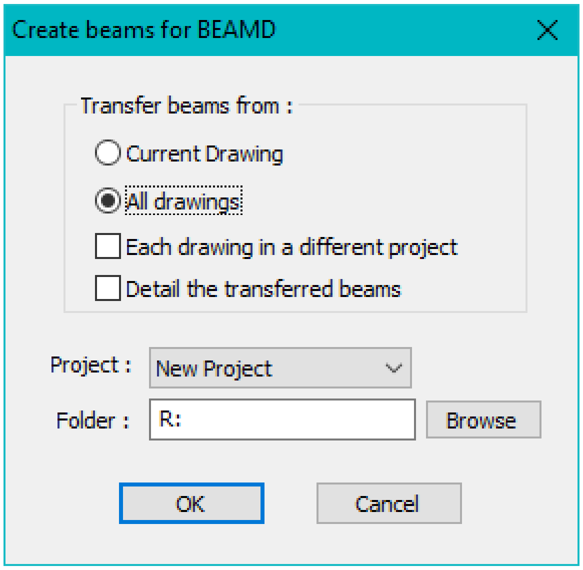
Create *BEAMD* files for beams and ribs on the drawings.

- the program transfers the beam geometry and the applied loads to the files.
- the beams and ribs can then be viewed and modified in the *BEAMD* program to complete the design and detailing; select 

Note:

- after a beam is transferred, the program display a **green** line on the beam center line together with the BEAMD beam number.
- if changes are made to the beam loads, parameters, etc, the line color changes to **red**
- the line reverts to **green** if the beam is transferred again/

Select the drawings and the *BEAMD* project:



Transfer beams from

Create *BEAMD* files for either the current drawing or all the drawings in the model.

- If you select **All drawings**, the beams on each drawing can be created in separate *BEAMD* projects or they can all be created in a single project.
- If you select **Detail the transferred beams**, the program also details the reinforcement in the beams according to the Code requirements and user-defined parameters.

The designed (and detailed) beams and ribs can be viewed and revised in *BEAMD*.

Project

Select an existing project from the list or create a new project; enter the new project name.

If you select **Each drawing in a different project**, the program automatically takes the project names from the list of drawing names in the Parameters - DXF 1458 files option.

Folder

The beams can be added to those in an folder with existing beams (created either by *AutoSTRAP, BEAMD* or the *STRAP* concrete design module) or can be created in an empty folder.

- Click OK to continue
- select the beams and/or ribs to transfer
- Run the *BEAMD* program to complete the design of the beams

Note:

- all slab loads 1461 must be applied as ***beam loads*** (bidirectional or unidirectional), not element loads.
- Select Display - BEAMD beam numbers 1448 to display the beam number in the *BEAMD* beam list.
- adjacent beam segments with identical sections and with a "no support" between them are transferred as a single span.
- The program creates end supports as follows:
  - supported by another beam or column : pinned
  - supported by a wall - long wall dimension parallel to beam : fixed
  - supported by a wall - short wall dimension parallel to beam : pinned
- the program calculates reactions from a supported beam as a load on the supporting beam at supports defined in the Supports 1472 option.
- when a beam is created again in the same folder (any project), the program ***replaces*** the previous beam.
- when a beam previously transferred is created in a different folder, the existing *BEAMD* file is not deleted. The number (***Bnnn***) of the last created version is shown on the *AutoSTRAP* display.

Index

- . -

- A -

- B -

- C -

- D -

- E -

- F -

- G -

- H -

- I -

- J -

- L -

- M -

- N -

- O -

- P -

- Q -

- R -

- S -

- T -

- U -

- V -

- W -

- Z -

Back Cover