USER'S MANUAL

Version 2020

STRAP 2020

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

1 General

This User's Manual provides a concise summary of the main features of the STRAP structural analysis and design program.

Detailed explanations on all of the options and features are available in the 'Help' supplied with the program:
- **To display the entire User's Manual, select the [Help] option in the program menu bar**
- **To display Help for the current option, press [F1]**

STRAP (STRuctural Analysis Programs) is a package of computer programs for the analysis of linear elastic structural models. The system includes static and dynamic analysis.

STRAP programs offer the engineer an easy-to-use tool for analyzing a wide range of frame and truss structures, plates and shells, or a combination of both types.

Frame structures consist of an assemblage of one-dimensional beam elements defined by the cross-section properties. Plate and shell structures are modeled by two- or three-dimensional elements interconnected only at nodal points. The behaviour of a finite element model closely approximates the behaviour of the real structure.

STRAP includes the following elements:
- One-dimensional : Beam element
- Two dimensional : Quadrilateral element & Triangular element
- Three-dimensional : Solid element

All element types may be defined in the same model.

Models are solved using the stiffness method. It assumes a linear elastic model and small displacements. The program solves the equilibrium equation:

[K] * {d} = {P}

where:
[K] = the structure stiffness matrix
{d} = the unknown nodal displacements
{P} = the applied nodal forces

After solving the equation for the unknown joint displacements, the program calculates the internal forces and stresses in the beams and elements.

Although using *STRAP* does not require knowledge of computer structural analysis, it is recommended that the user familiarize himself with the basic theories of the stiffness method and finite elements.

The *STRAP* system consists of the following modules:

- Static Analysis
  - Plane frame analysis:
    Analysis of plane frame models (all loads applied in the plane of the model) composed of beam elements and two dimensional plane stress elements.
  - Plane grid analysis:
    Analysis of plane grid or plate models (all loads applied perpendicular to the plane of the model) composed of beam elements and two dimensional plate bending elements).
  - Space frame analysis:
    Analysis of general space structures composed of beam elements and combined plane stress/plate bending elements. Typical structures are space frames, shear walls combined with frames, roof shells, folded plates, water tanks, etc.
  - Truss:
    Plane and space truss analysis composed of pinned beam elements only.

- Dynamic Analysis
  - Dynamic modal shape analysis and natural frequency calculation.
  - Seismic response spectrum analysis
  - Forced Vibrations and transient Response

- Design Postprocessors
  - Structural steel design and member selection according to:
    Hot-rolled : AISC-LRFD/ASD, Eurocode 3, BS5950, AASHTO-LRFD/ASD, CAN CSA S16.1, IS:800, SABS 0162-2, GBJ17-88,SNIP II-23-81, Nbr
    Composite design is available for all codes
    Cold-formed : AISI-ASD /LRFD, CSA135 , EC1.3, BS5950
  - Reinforced concrete design for beams and columns according to: ACI 318, CSA A23.3, Eurocode 2, BS8110, IS:456, NBr 6118, GB50010, SP52-101 and the associated national seismic codes.

    Column, slab and beam detailing and drafting.
  - Bridge analysis:
    Defines lanes; calculate influence lines and maximum/minimum results for each result type for all points on a bridge model.
  - Post-tensioned concrete beam design.
  - Steel connection design: welded and bolted connections.
  - Foundation design: Design of spread footings.

The most general and comprehensive module in the package is "Space Frame". It enables the engineer to model any type of space structure using of the elements types.

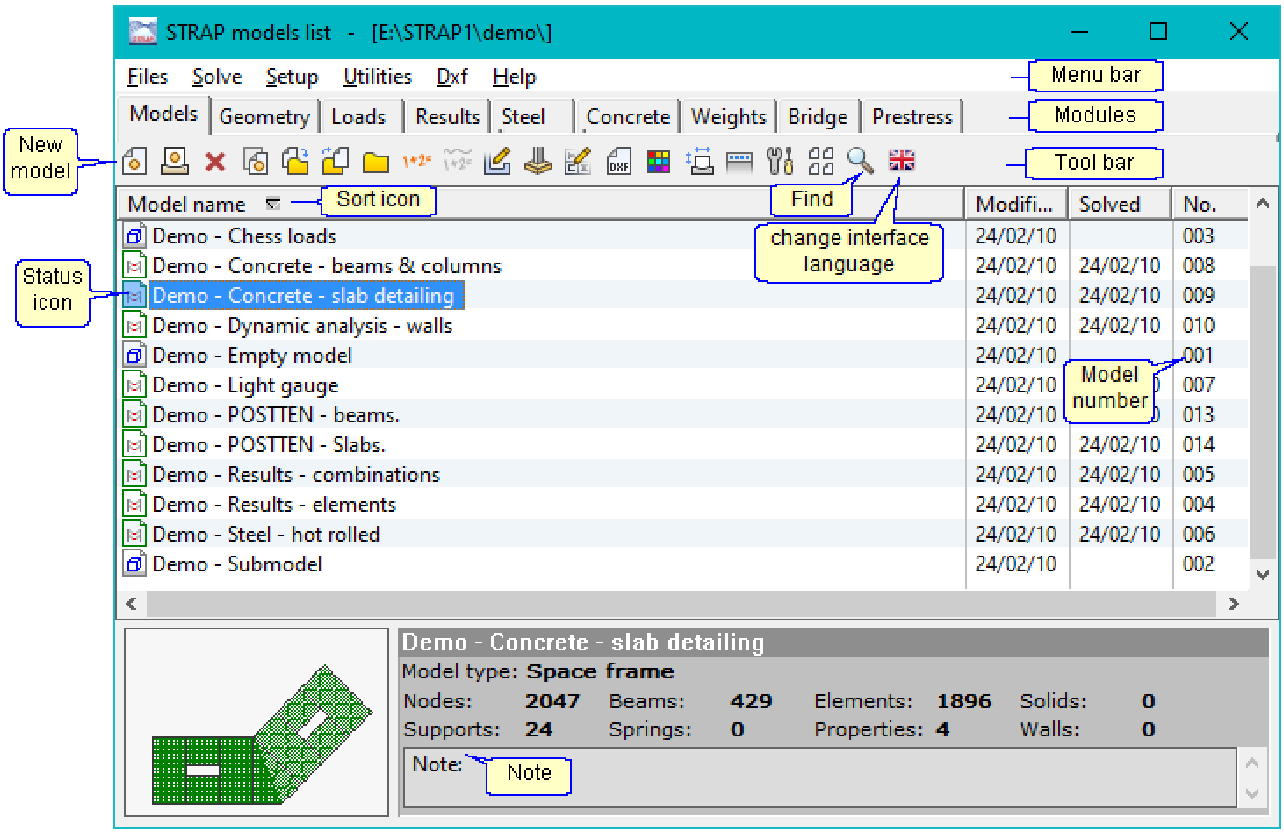
In order to simplify the use of the programs, the system also includes modules for more specific types of structures. For example, the "Plane Frame" module solves plane frames only. "Space frame" can also be used to solve plane frames, but it is more convenient to use "Plane frame" as the data required is simplified and the possibility of user error is reduced accordingly.

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 13
Amend/rerun an existing model 13
File management 14
Utilities 29
DXF Import/export 31
Setup 26



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.

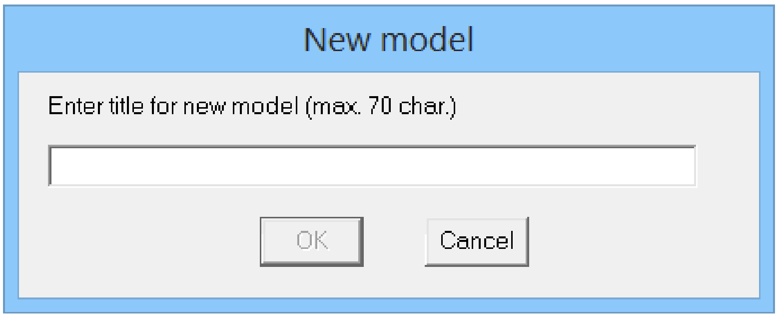
Note:
- corrupted model lists may be restored by using the **Utilities** - Recreate a models list option 29.

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:



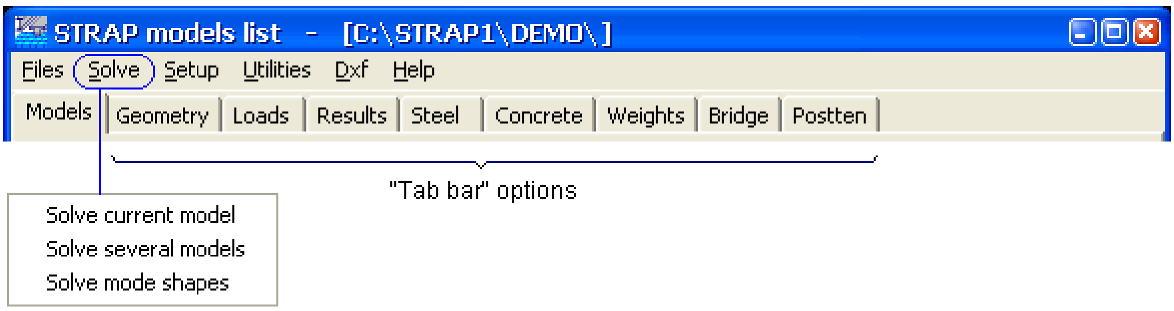
The program then displays the Geometry preliminary menu.

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

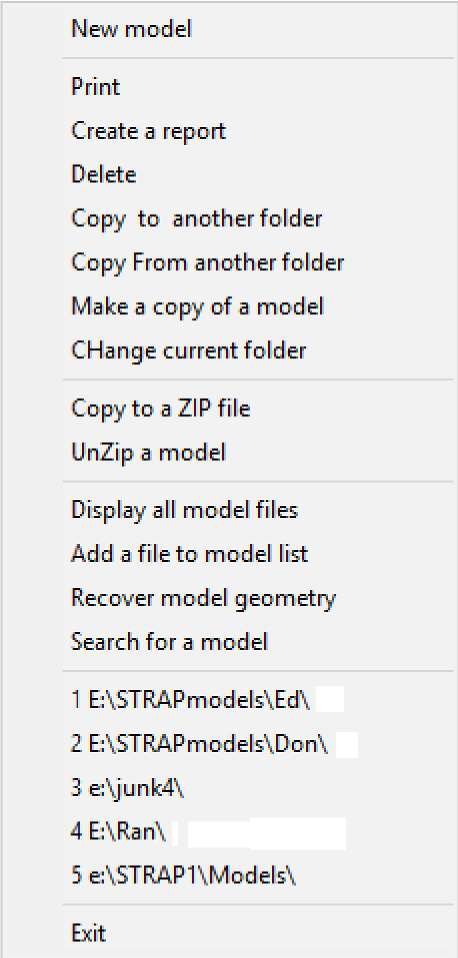
To directly access any other part of the program (loads, results, etc):
- move the cursor to the model name in the list and click the mouse once; the model name is then highlighted.
- click one of the options in the 'tab bar' below the toolbar:



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 25) 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

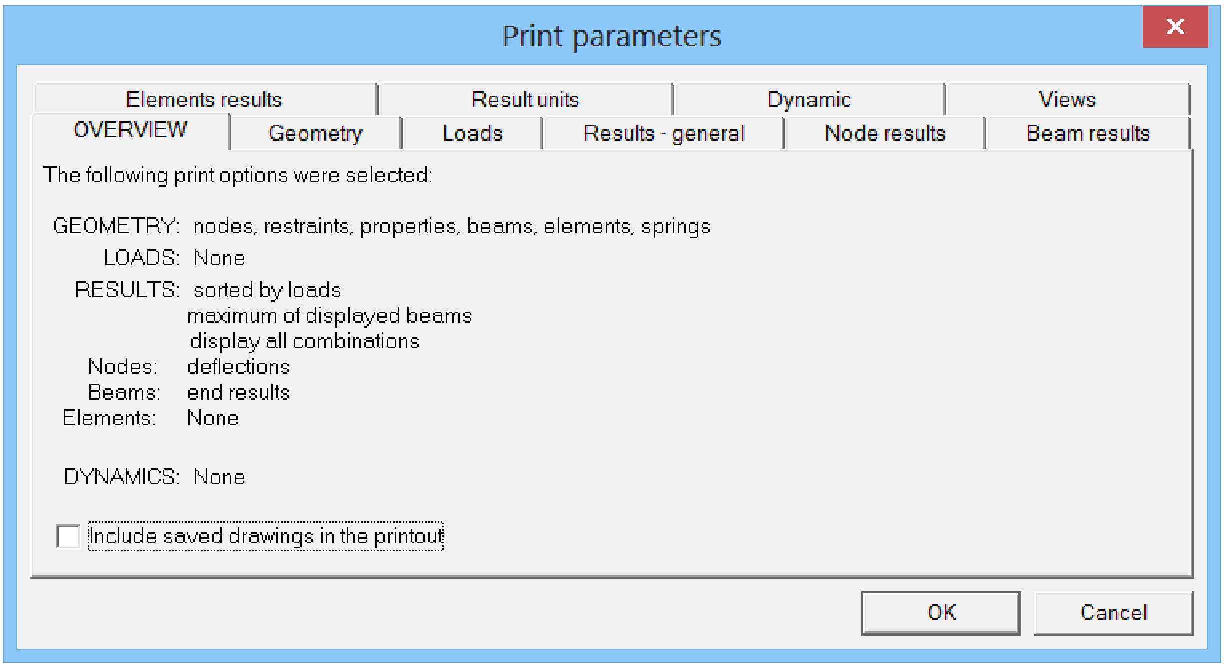
Use this option to print all or selected input, results and saved drawings (in any order) for any model in the list without running the model.

- 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*** 15 ***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.

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.

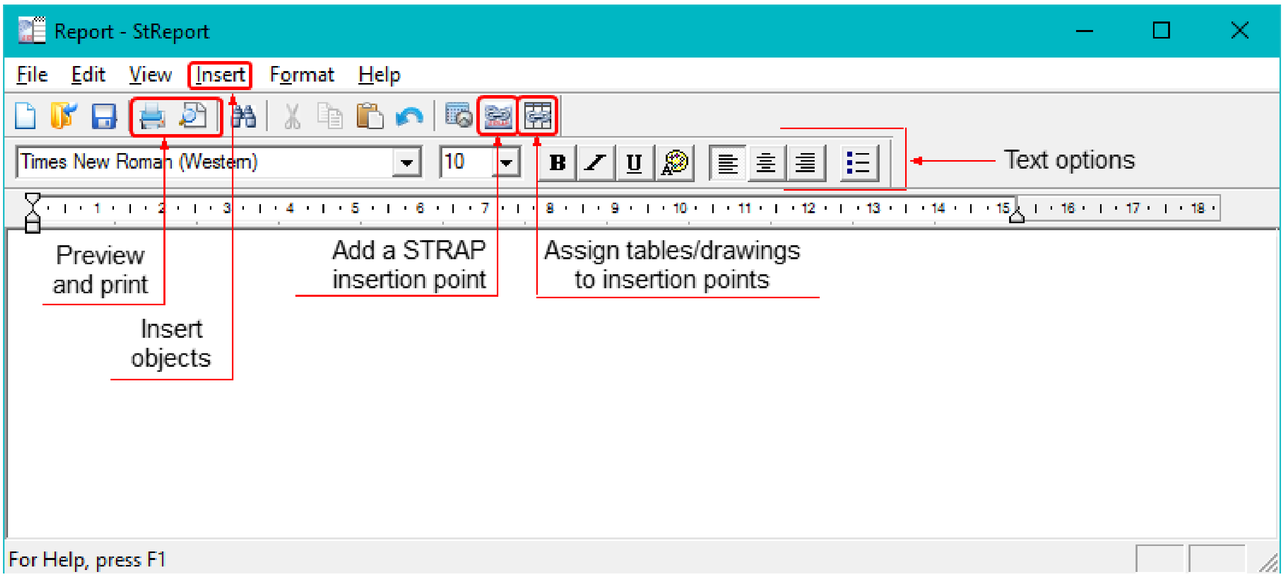
The saved drawings and tables are added to the report at "insertion points":
- saved drawings and tables are assigned by the user to these insertion points.
- the list of drawings & tables assigned to an insertion point is saved with the model and may be updated if the model is revised.

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

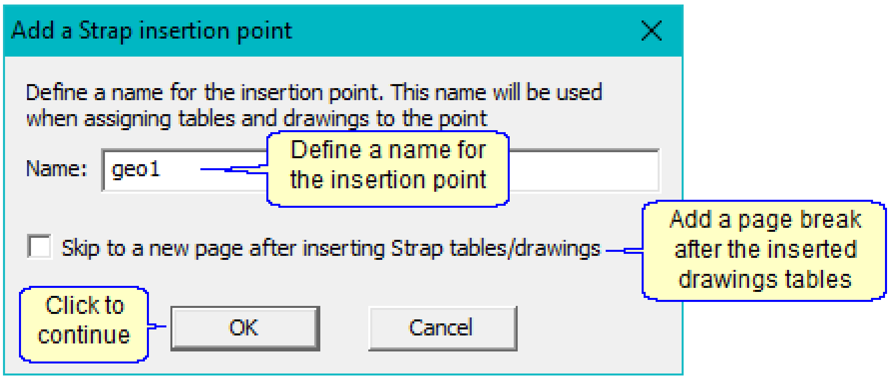
Refer also to Additional notes.

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

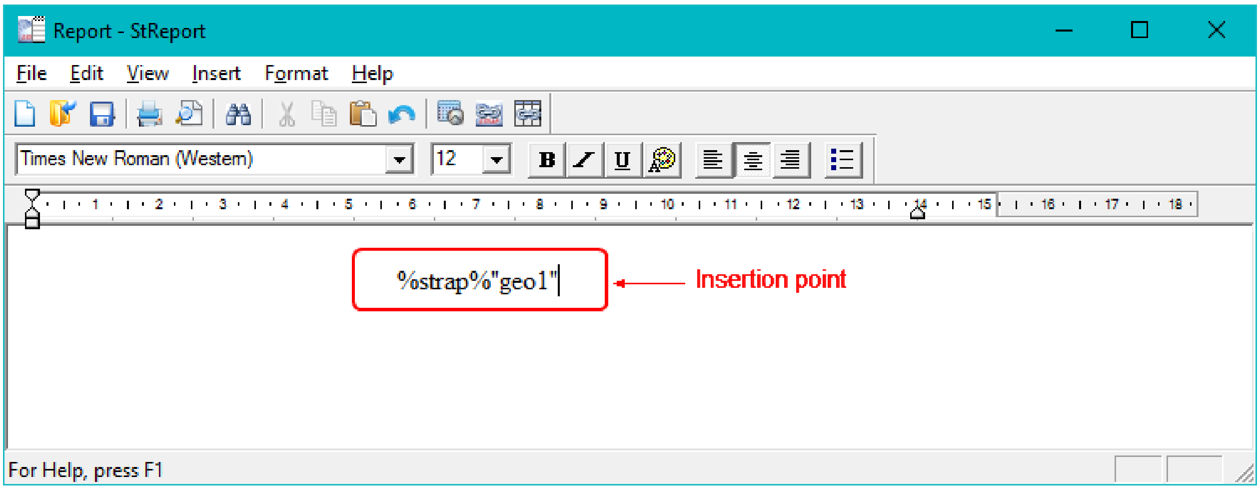


Define an insertion point

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



- the insertion point is displayed in the report document:

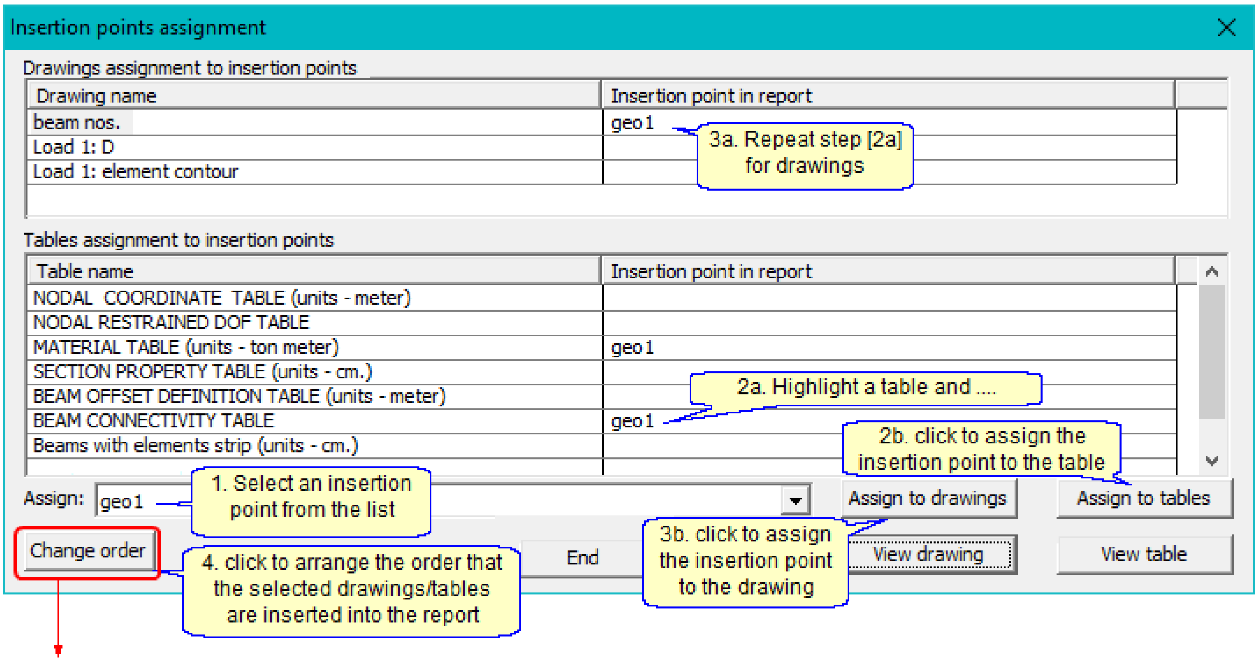


Note:

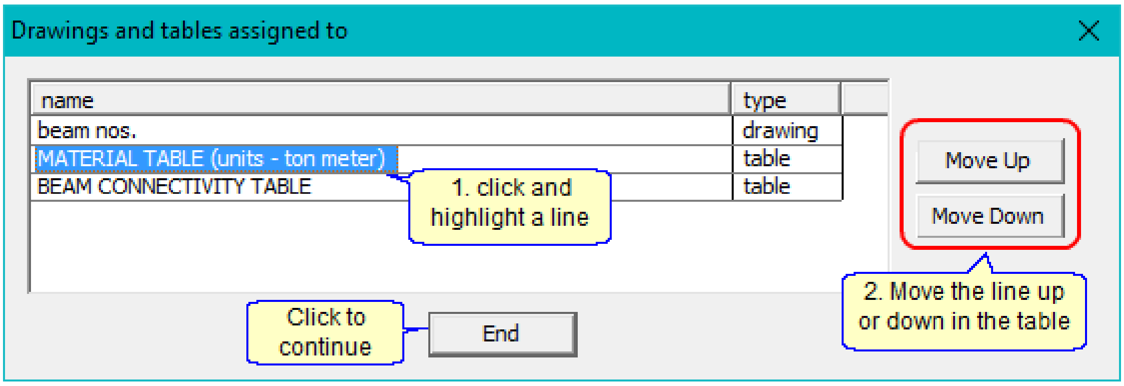
- Insertion points can be typed in manually using the format **%strap%"*name*"**, where ***name*** is the name of the insertion point, e.g. **geo1** in the example above.

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):

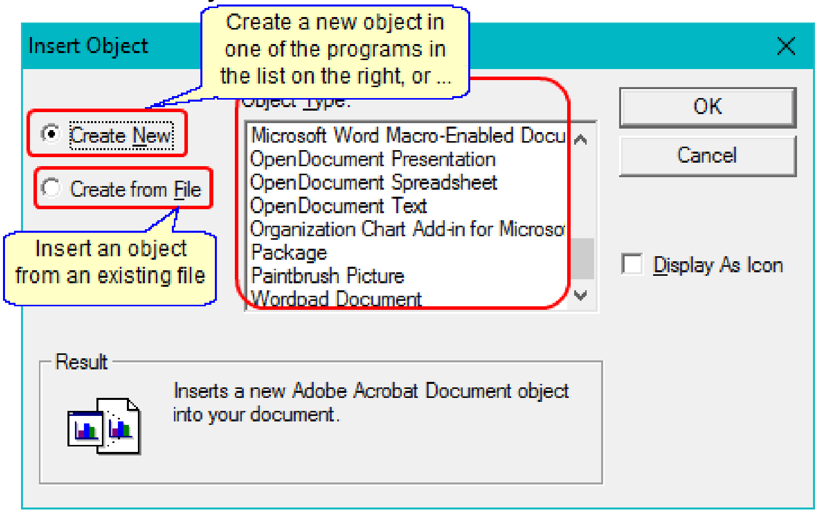


Add miscellaneous texts

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

Add miscellaneous objects

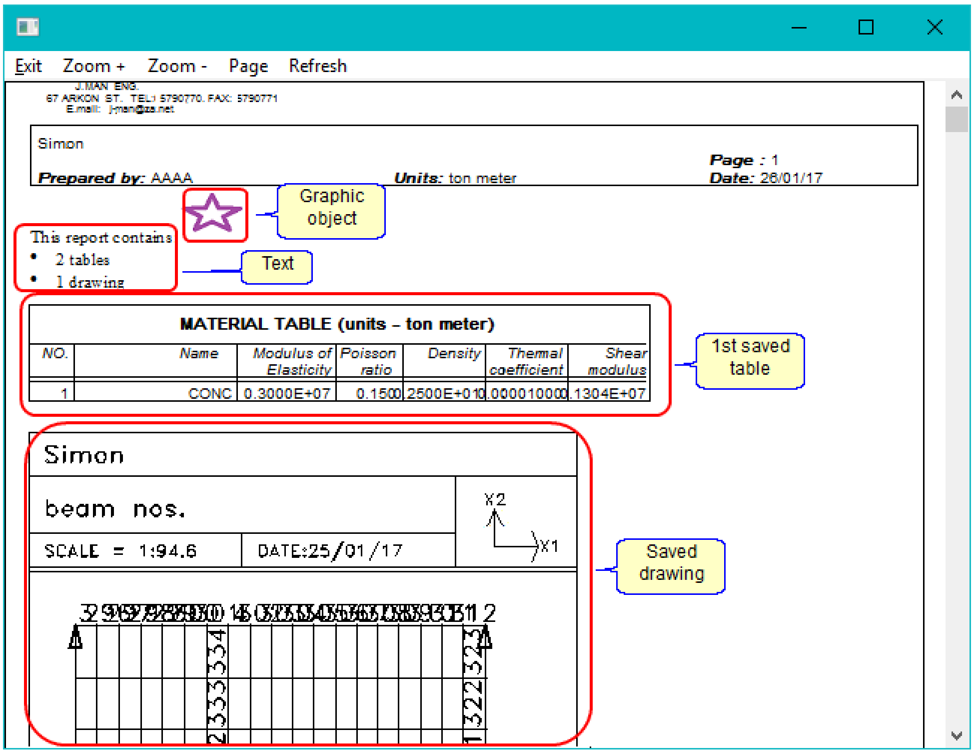
Select **Insert - object**:



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

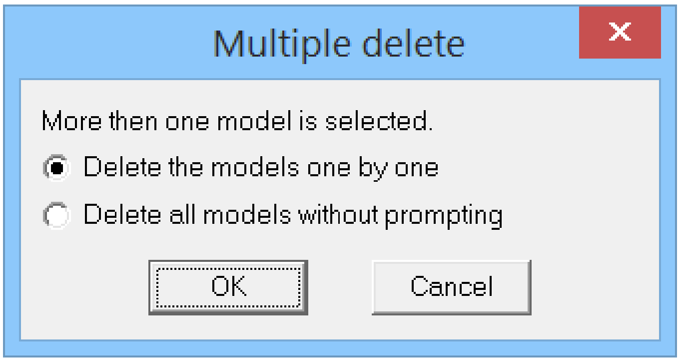
- Report files can be created in any text editor or word processor that can save the file in .RTF format. Insertion points can be typed in manually using the format **%strap%"*name*"**, where ***name*** is the name of the insertion point, e.g. **geo1** in the example above.

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:

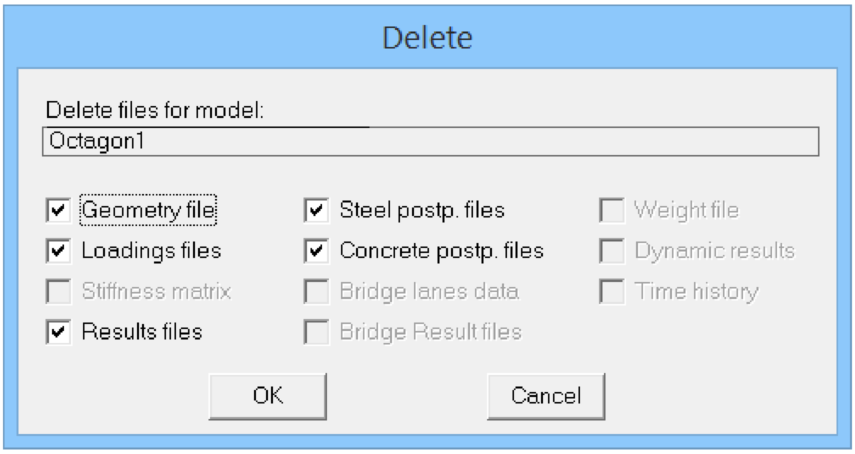
- click and highlight model names in the file list
- if you selected multiple models:



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.

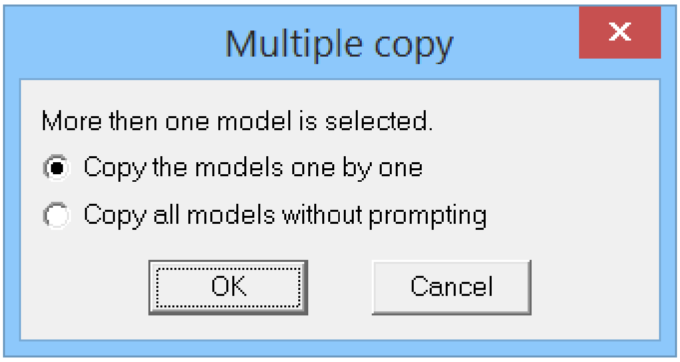
Note:

- 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:

2.3.4 Copy model to another folder

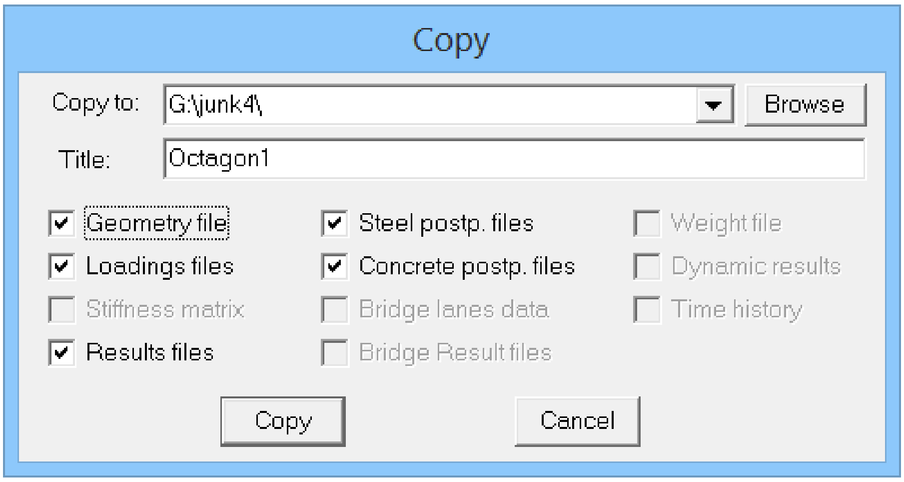
Use this option to copy a model to the current folder or to backup a model to a different volume. Note that multiple models may be selected.

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.

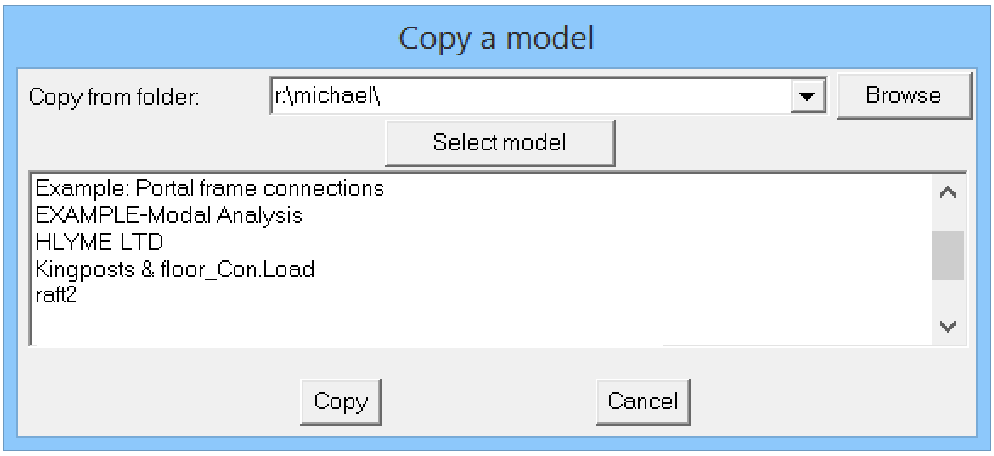
Note:
- the program does not replace an existing model in the destination folder that has the same name; it adds a numerical suffix, e,g. "(1)" to the title of the copied model.
- To create a copy of a model in the current folder, refer to Make a copy of a model 21.

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:



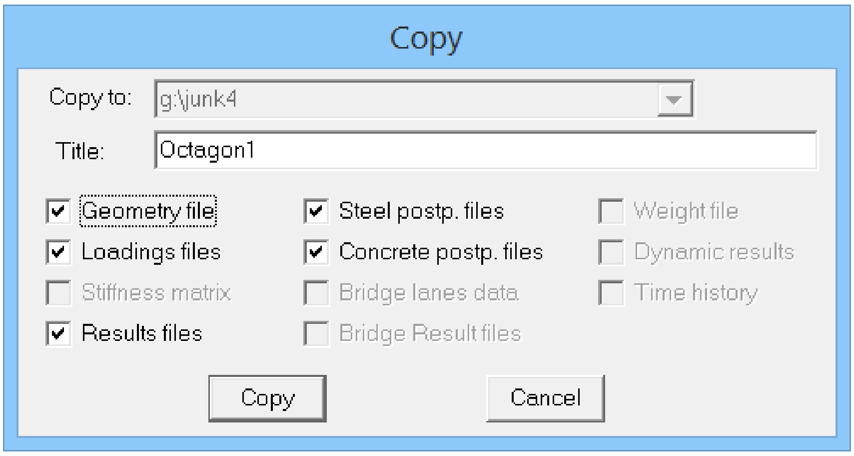
  to select more than one model, press the [Ctrl] key when clicking on the additional models.
- for each model selected, specify the files to be copied from the list displayed; only files that are marked with a ☑ are copied. Click on the file description to toggle the status.
- click the Copy button to 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 21.

2.3.6 Make a copy of a model

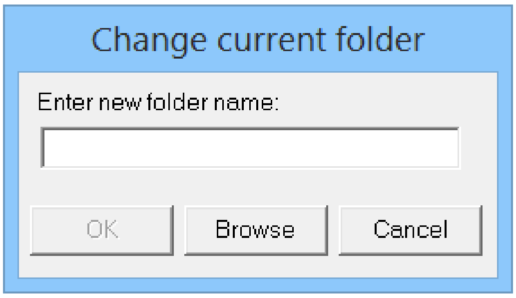
Use this option to create a copy of the highlighted model in the current folder:



- 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.



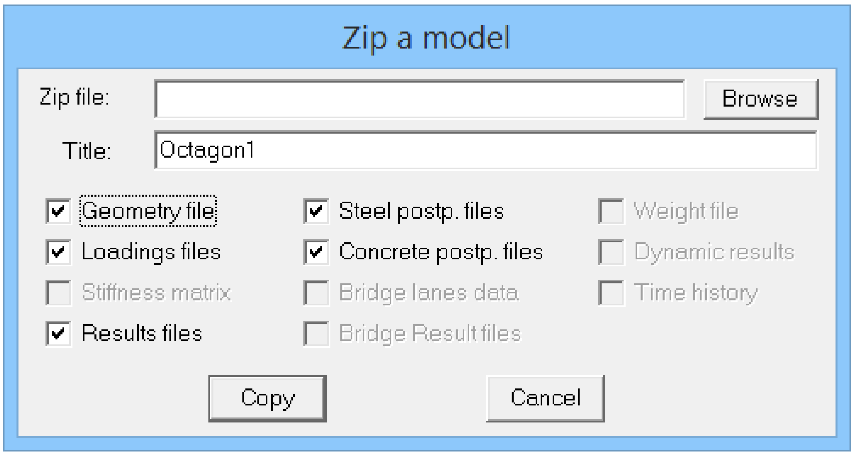
- to select an existing disk folder on any drive type in the name of the path or click the Browse button and select the path in the standard Windows "**Select a folder**" dialog box.
- To create a new folder, click the Browse button and define a new path by clicking the icon ("Create New Folder") in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

Note:
- the last five folders selected in this option are displayed at the bottom of the File menu 14 and may be selected by clicking on them.
- do not select a "Read-only" folder

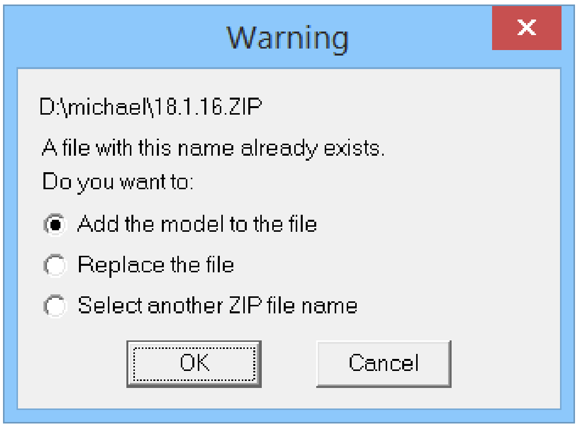
2.3.8 ZIP

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:

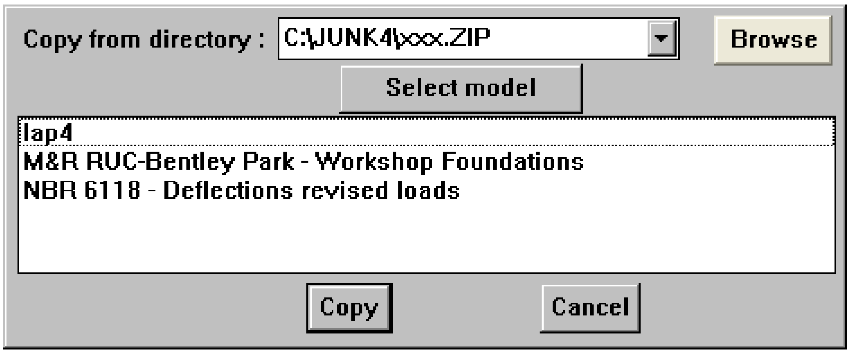
- **Add the model**
  the selected models are added to the file
- **Replace the file**
  the existing file is erased and a new one is created with the selected models
- **Select another ZIP file name**
  Cancel the option and enter a different file name before continuing

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**.

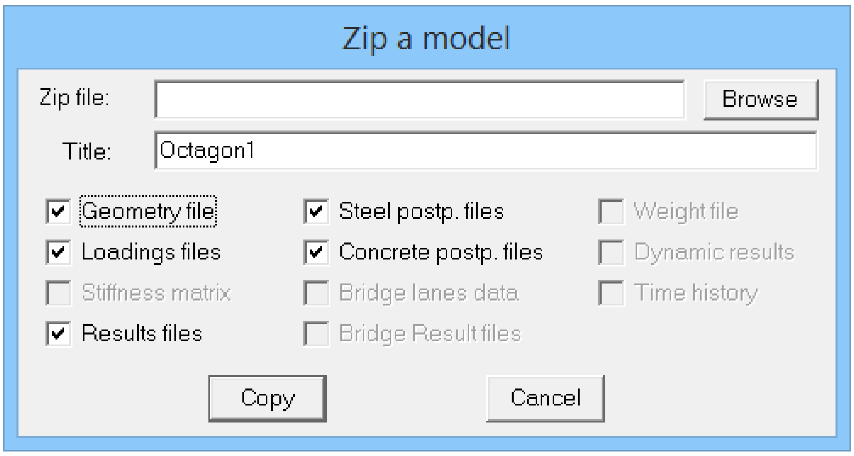
To retrieve a model from a ZIP file:

- select the ZIP file
- click and highlight one or more of the models in the list:



- click 

- select the files to extract:



- click Copy

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.

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.

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)

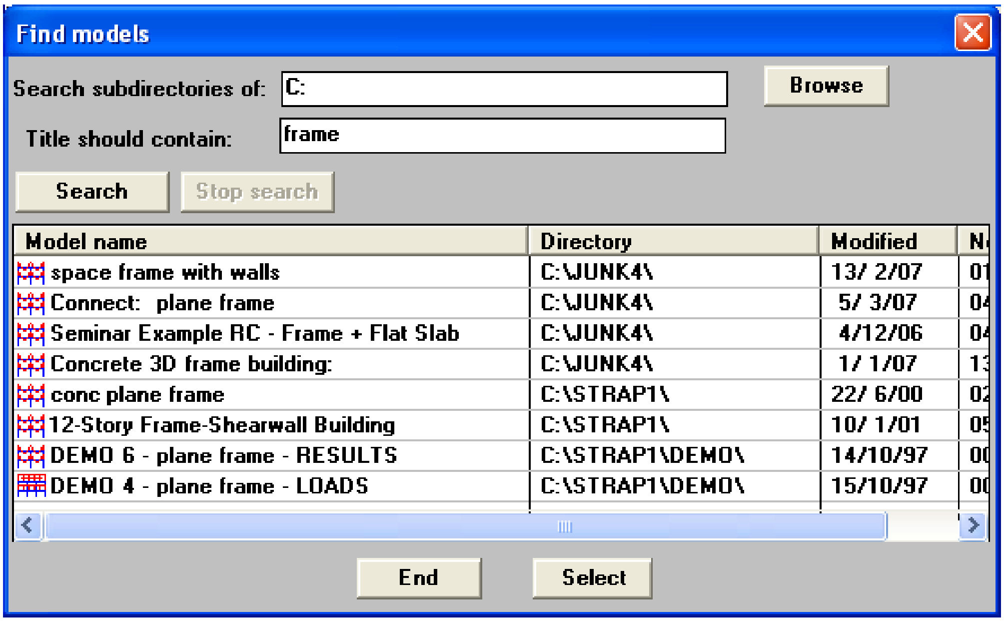
2.3.12 Recover model geometry

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:

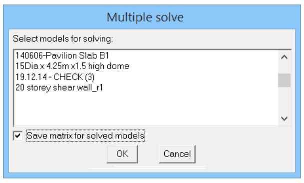
- when you select the results of the previous search are displayed in the list; if you selected the wrong model previously there is no need to search again.

2.4 Solve

There are two options available:

Solve current model
Solve several models
Solve mode shapes

- **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).

2.5 Setup

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

Colors
Miscellaneous
Print parameters
Toolbars
Reset all values

2.5.1 Colours

Specify the permanent screen colours for:

- graphic lines 26, e.g. beams, elements, various loads, etc.
- text 26 associated with each graphic line
- background 26
- contour map 26 fill colors
- steel postprocessor 26 capacity colors

2.5.2 Print parameters

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

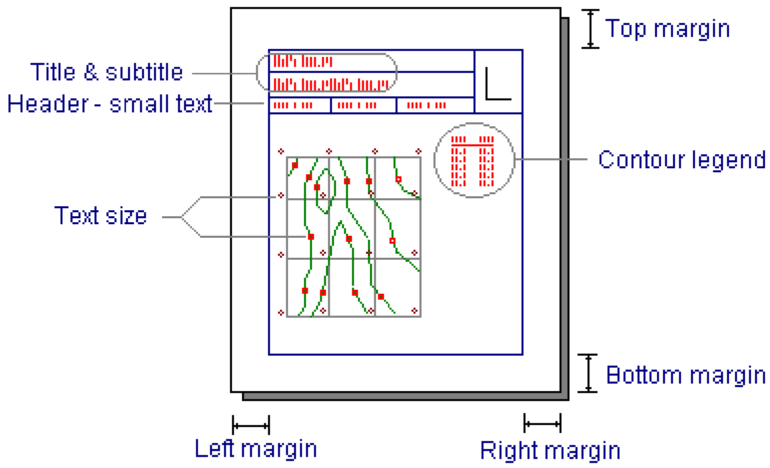
Table print styles

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.

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.

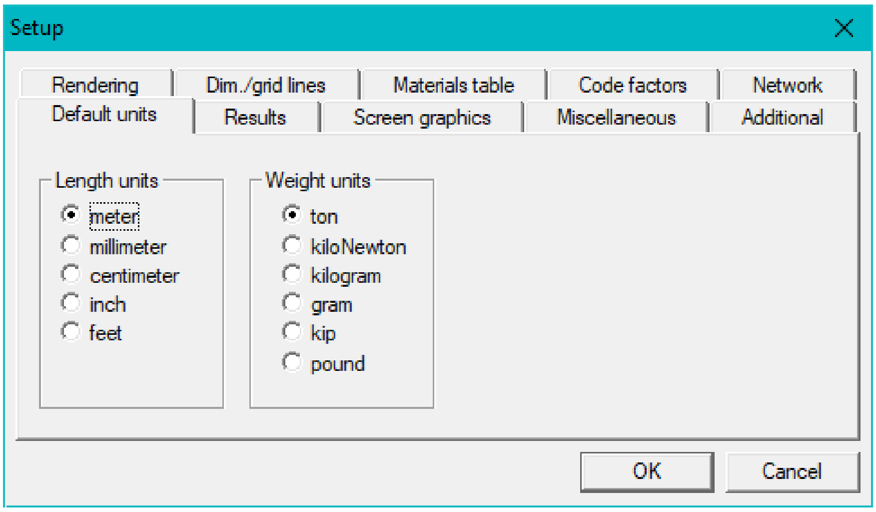
Line / text width

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.

2.5.3 Miscellaneous



Units

Set the default input and result units for ***all*** new models. (if the default units are revised at the start of the geometry definition, the revision is applied only to the current model).

Materials

The properties of 10 materials are permanently stored in the program. Four of these materials are user-defined materials. These 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/°C or 1/°F)

Miscellaneous

Refer to the program "Help" or the full manual.

Rendering

Refer to the program "Help" or the full manual.

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 **Display □ toolbar lines simultaneously**.

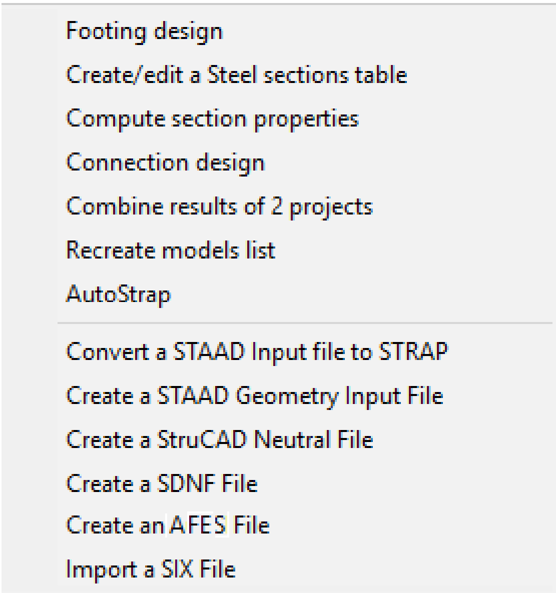
2.5.5 Language

Select an interface language from the list:

Note:
- the characters may not display correctly after selecting a different language. Revise the Windows "Control panel" "Language and regional settings as explained in the menu.

2.6 Utilities

The following utility modules are available:



Note:
- you can add new options 30 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.

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.

Convert a STAAD file to STRAP

Refer to - STAAD file import 30.

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.

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).
- ***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).

2.6.3 Add new options

New options can be added to the Utilities menu. Refer to the full manual or the program Help for instructions.

2.7 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.

2.7.1 Metafile to DXF

Create a two-dimensional DXF file of any ***STRAP*** graphic display.
- the program displays a list of the Metafiles (*.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.

2.7.2 Convert STRAP to DXF

This option converts the ***STRAP*** geometry to a 3-D DXF file. 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/elements are drawn in a separate layer.
- The following is ***not*** transferred to the DXF file:
  Dimension lines, sections, materials, text.

2.7.3 Create a new model from a DXF file

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.
- ***STRAP*** property group numbers may be assigned to the generated beams/elements according to Autocad layer.
- ***STRAP*** property 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 models.
- 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.

3 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 ?

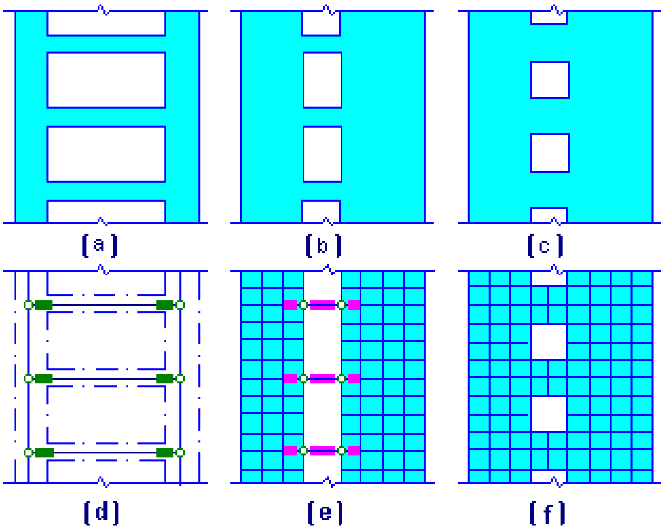
The following table gives results for the centre deflection of the plate : (10x10, 0.2 thick, concrete):

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 ***possible*** 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.

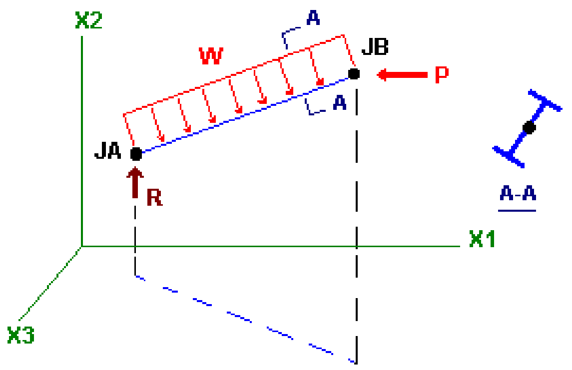
4 Coordinate systems

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. A cylindrical system may be used for node coordinate definition.

Two types of reference systems are used in ***STRAP***. They are:
- the global coordinate system 37, denoted by **X1, X2, X3** (uppercase)
- the local element coordinate system 38, 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.

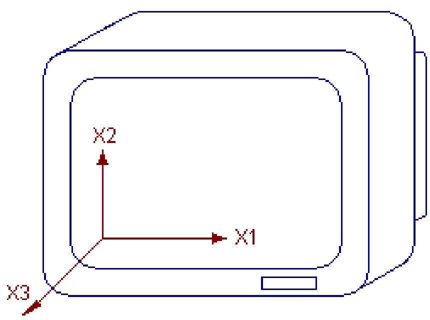
Consequently, the beam is provided with a ***local*** axis system x1,x2,x3; section orientation, direction of loads and results are always relative to this local system.

Note:
- ***each*** element has its own unique local system independent of the local systems for 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 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.
- the directions of loads, forces, moments and stresses are referenced to the global or local axes by standard sign conventions.

4.1 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.

4.2 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.

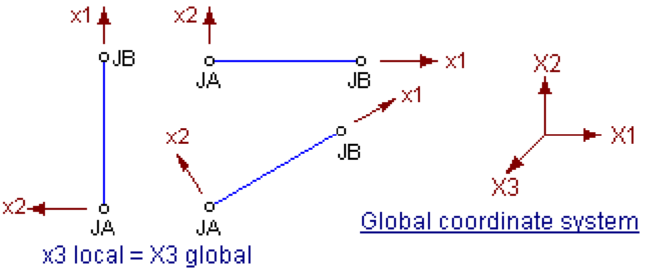
4.2.1 Beams

The directions of the local axis determine:

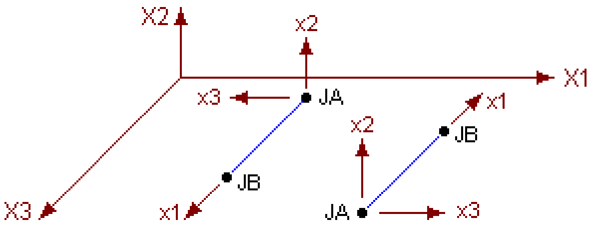
- the orientation of the section major and minor axes, which are ***always*** aligned with the x2, x3 axes
- the direction of the beam loads, when defined parallel to the x2, x3 axes

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.
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.	



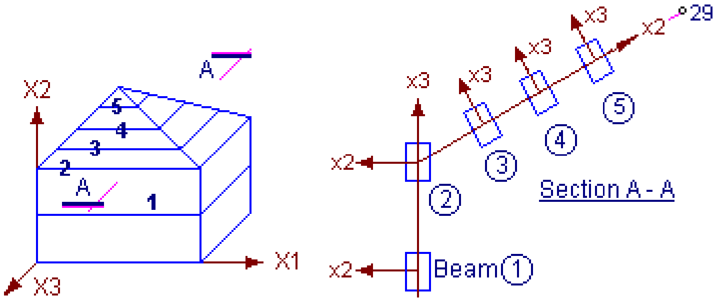
Space models		
Axis	Direction	
	General case: x1 ***not*** parallel to X3	Special case: x1 parallel to X3
x1	Always coincides with the axis of the beam; the positive direction is from the start node (JA) to the end node (JB).	As in 'General case'
x2	As x1 and x3 are known, x2 is determined by the right-hand rule.	***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.
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.

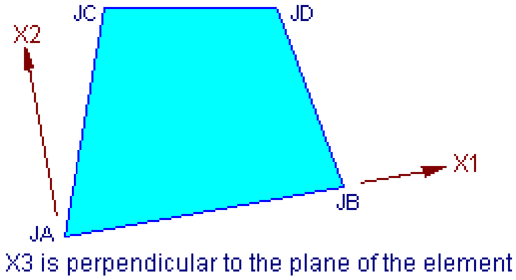
4.2.2 Quad elements

Each two-dimensional finite element has a local coordinate system associated with it.

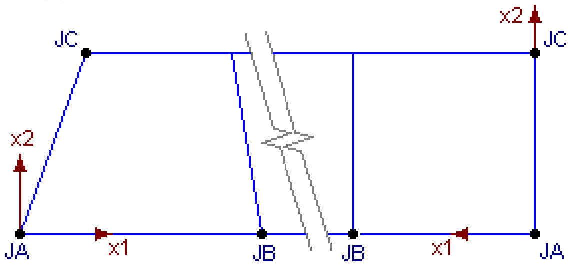
The local x1 and x2 axes ***always*** lie in the plane of the element and x3 is ***always*** perpendicular to the element.

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



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.



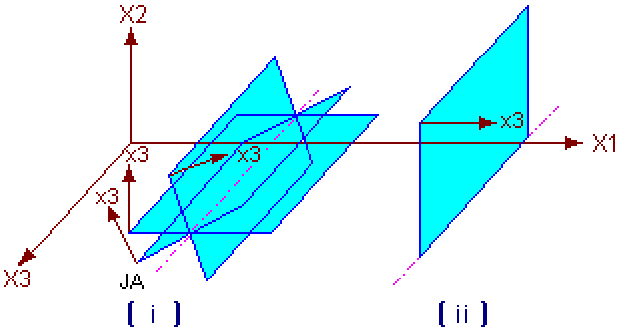
When elements are defined in the Graphic Mode, the program automatically ensures uniformity in the local x3 axis direction for adjacent elements in order to prevent confusion in the results. ***The +x3 direction always points in the general positive direction of the global +X3 axis*** (except for the 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. If +x3 does not point in the direction closest to the global +X3 axis, the program interchanges the JA and JB nodes.

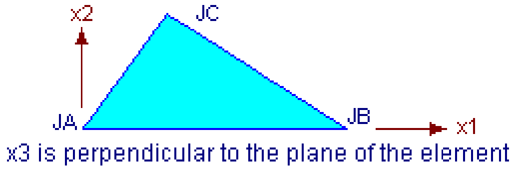
Special cases:

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.



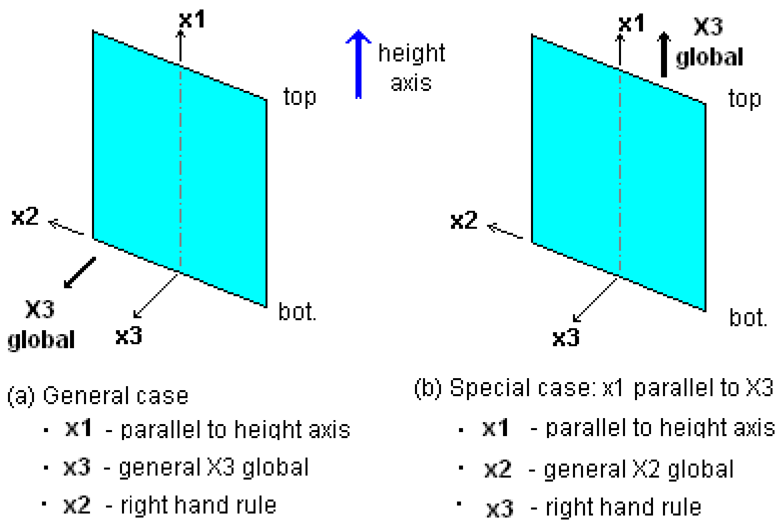
4.2.3 Triangular elements

The definition of local axes is similar to that for quadrilateral elements[39].



4.2.4 Wall elements

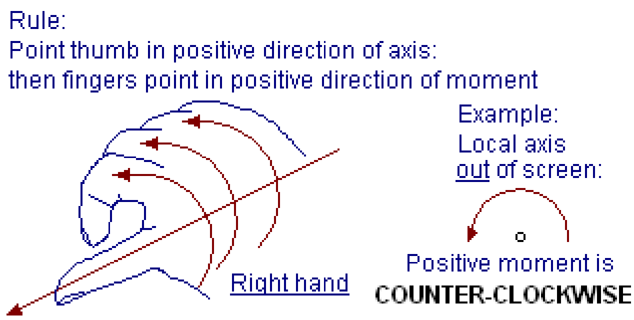
The default local coordinate system for wall element segments is identical to the default system for beams[38]. The program assumes that the wall local x1 axis is parallel to the "height axis" specified when defining the wall and points in the positive direction of the height axis. The default local axes cannot be revised.



4.3 Sign conventions

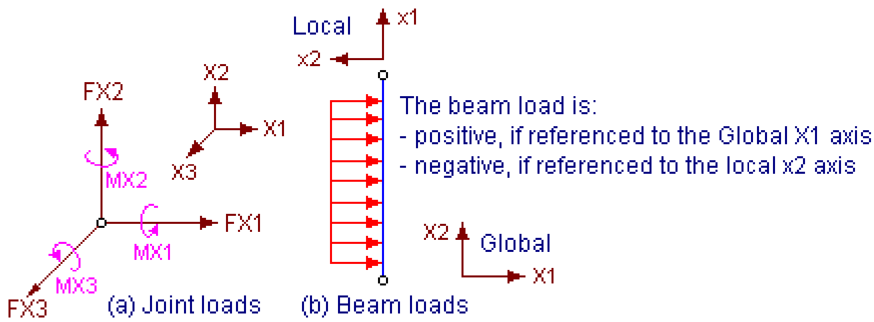
Forces and moments are referenced to a local or global coordinate axis:

- 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:



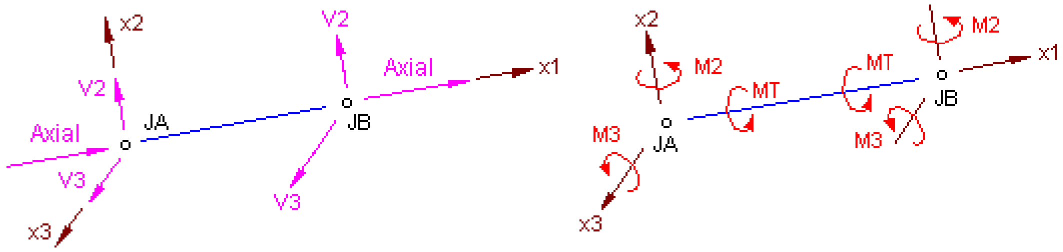
Examples:

- **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

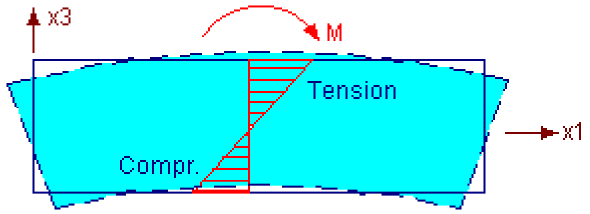
- **Beam Results (Tabular)**
  Moment, shear and axial force results for beam elements are always displayed relative to the beam ***local*** coordinate system. The sign conventions are:



For more information, refer to Beam results sign conventions

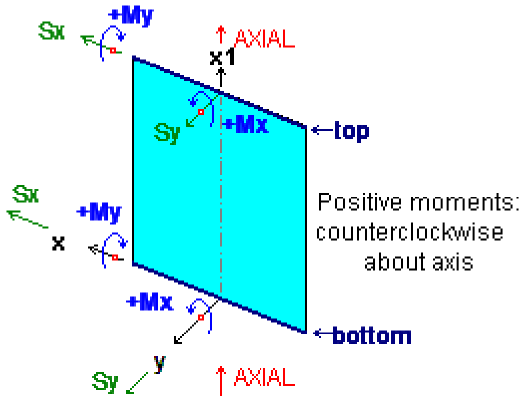
- **Finite Element Results (Tabular)**
  Moments, forces and stresses for finite elements are displayed relative to the element ***local*** coordinate system. Forces and stresses are positive if acting in the positive direction of the parallel local axis.

  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.

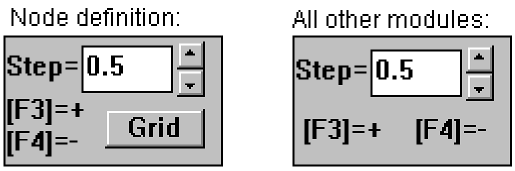
- **Wall elements**
  Moment, shear and axial force results for wall elements are always displayed relative to the wall coordinate system. The sign conventions are:



5 General options

5.1 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 F3 to increase the Step or F4 to decrease the Step.
- Click the buttons.

For node definition only:

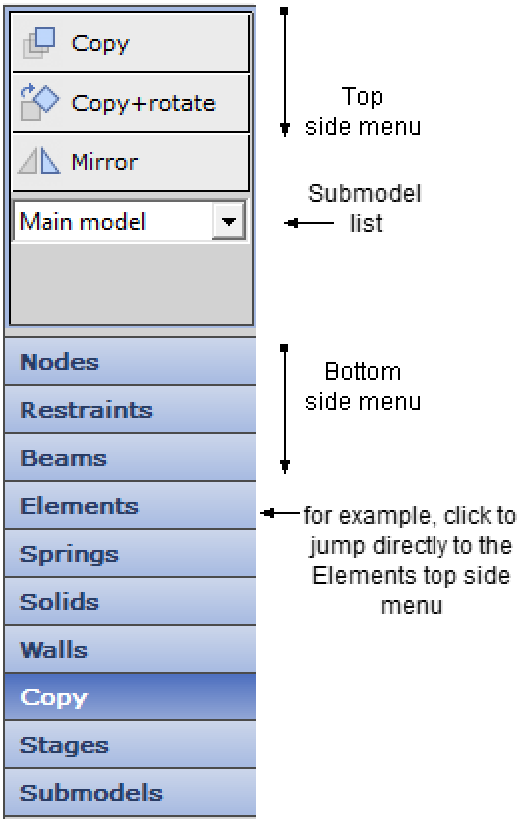
Click the Grid button to display a grid of dots on the screen; the dots will be spaced at the current ***Step*** interval in both screen directions. The jumps from dot to dot when the mouse is moved or when an arrow key is pressed.

- 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).

5.2 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".

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



5.3 Help

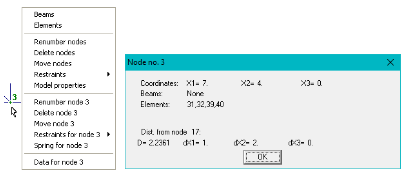
Select **Help** at any time from the menu bar at the top of the screen to display the section of the user's manual pertaining to the current option.

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.

5.4 Shortcut menus (right-click)

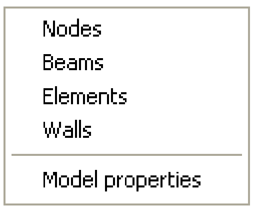
A Shortcut Menu is a useful pop-up menu that provides options specific to a certain object in the model. ***Right-click*** an object (such as a node or element) to display the Shortcut Menu for that particular object.

- ***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 is outside the model boundaries, the following menu is displayed:



Model properties:

- **Weight of displayed elements:**
- **Center-of-gravity for displayed part**

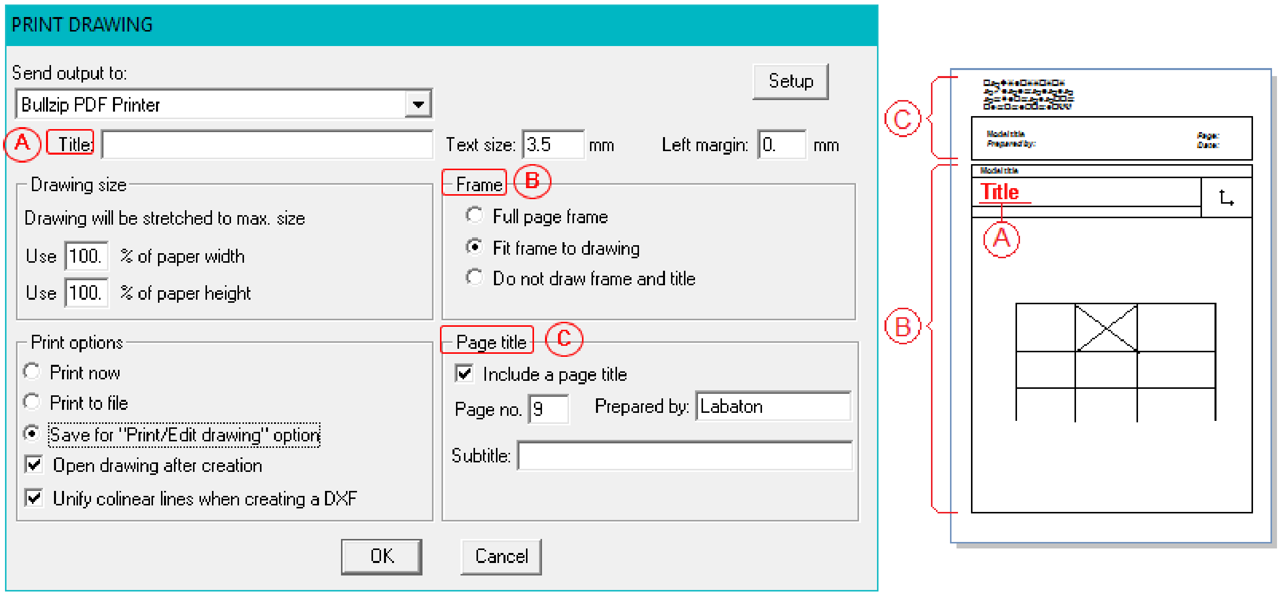
5.5 Print options

Select one of the following options:

5.5.1 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 47.



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 50 option.

Setup

Specify general information for the output device selected:
- paper size
- graphic resolution

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

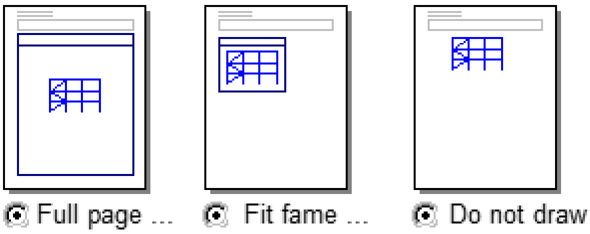
Specify the scale for the drawing. The program initially assumes that the drawing will fit exactly on one page (based on the paper size in the Setup option), calculates the corresponding scale and displays it as the default scale.

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

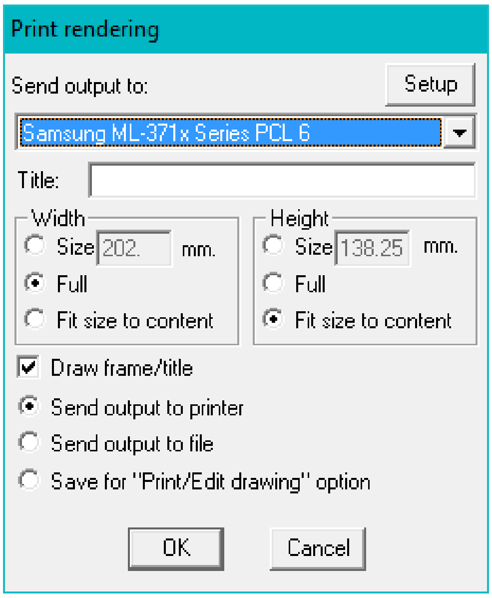
Include a page title

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

5.5.2 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 automatically calculates the dimension required to maintain the drawing 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 45.

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":

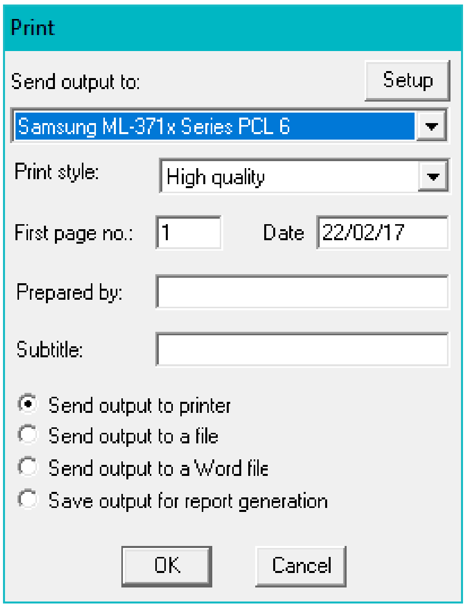
- The program will prompt for a file name.
- Select "Send output to printer" if **Metafile** is selected in **Send output to**:

Note:

- for all other options, refer to Print drawing 45.

5.5.3 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' becuause 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.
- 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 **

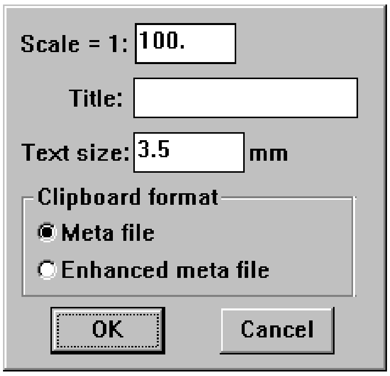
Send output to a Word file
The tables may be saved in a MS Word format file. The file is saved in RTF (Rich Text Format) and may be imported into any word processor or program that recognizes this format. Specify a name for the file. The default extension is .RTF

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

5.5.4 Copy drawing

Use this option to copy the current display to the Windows "Clipboard". The display will be identical to that produced by Print drawing 45, i.e. it will include a frame and a header. The file is transferred to the Clipboard in "Metafile" format.



Clipboard format

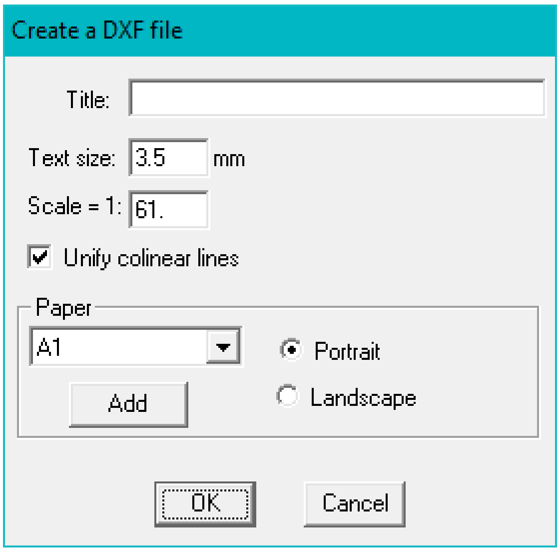
Select one of the following metafile formats:
- **Metafile =** WMF format
- **Enhanced Metafile =** EMF format

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).

5.5.5 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.

5.5.6 Print order

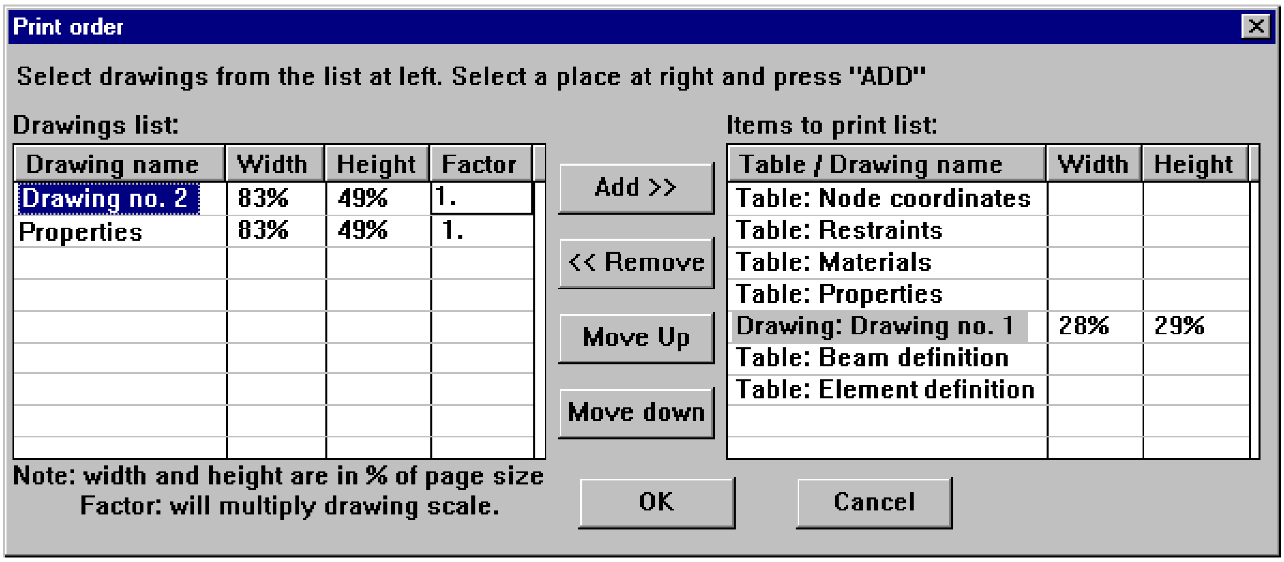
Arrange the order of the printout. Initially, all tables selected in the previous dialog box are displayed in the right list ("Print list"); all drawings are displayed in the left list ("Drawing List") and must be added to the Print List.

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

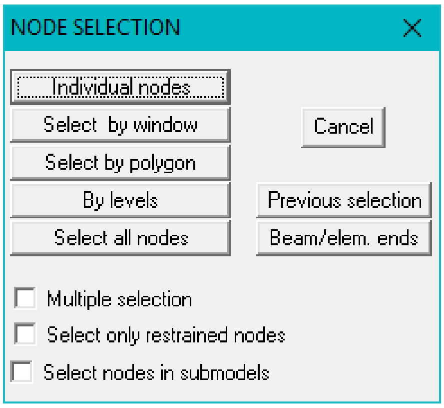


Click OK to start printing.

5.6 Selection options

5.6.1 Node selection

Many options include instructions to select one or more nodes.



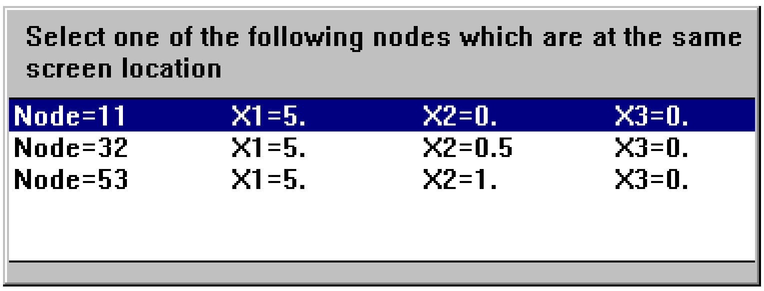
Select individual nodes

Select individual nodes by moving the 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 .

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.

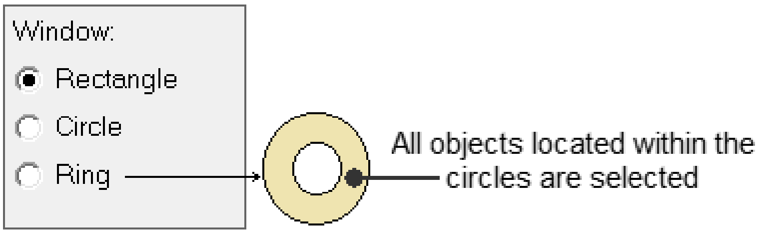


Move the cursor to the line with the correct node and click the mouse.

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.

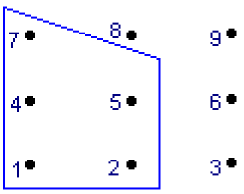
Select by polygon

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

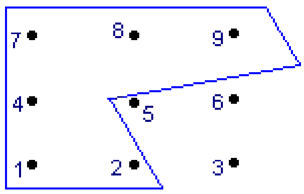
The polygon is constructed as a 'rubber-band' stretched around the defined corners:
- At least three corners must be selected.
- the program automatically connects the last corner defined to the first corner defined.
- click the right mouse button or press [Esc] to delete the previous corner.
- to end 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 :

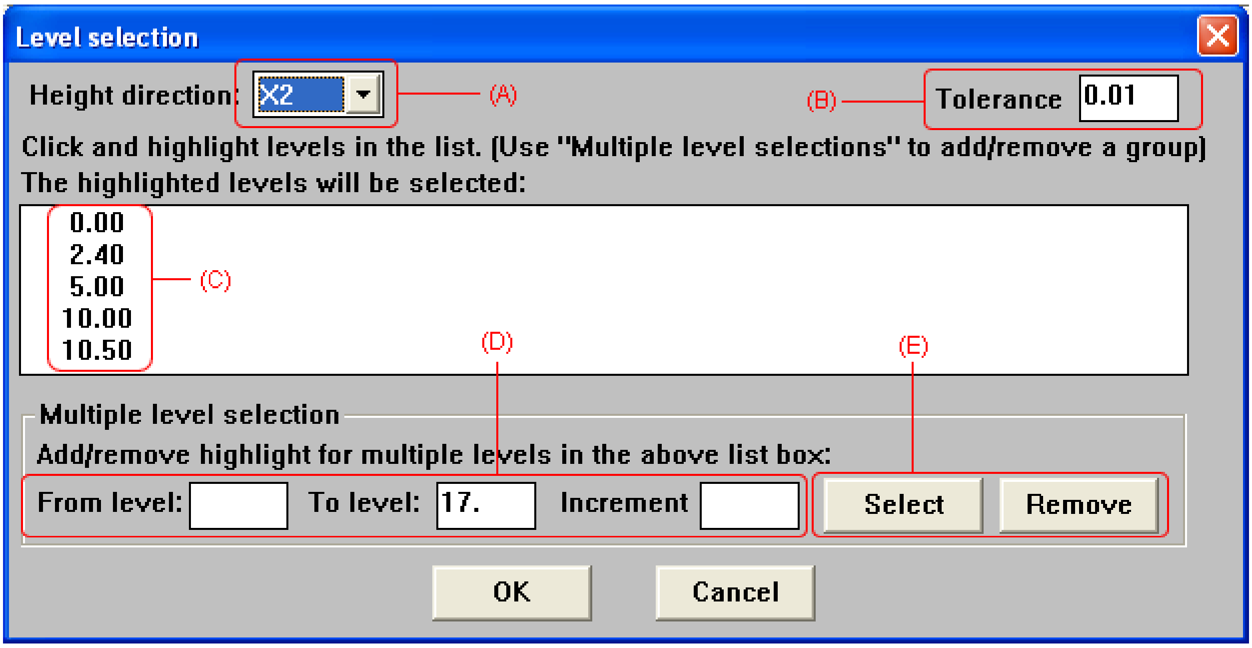


1,2,4,7,8,9:



Select by levels

Select all nodes at selected levels (coordinates):



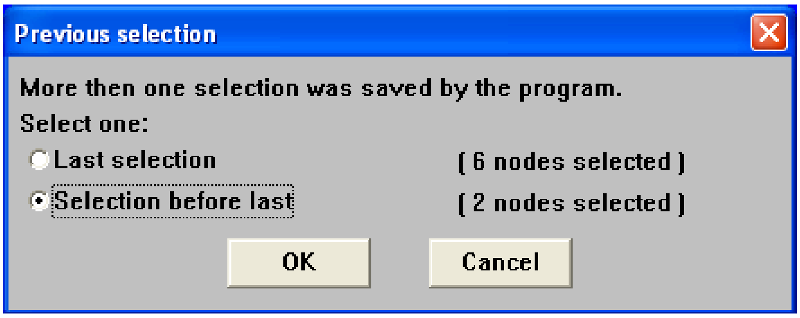
- The program displays a list of the levels parallel to one of the global axes **(A)**. A different global axis may be selected in the Height direction list box **(B)**.
- Select levels by clicking and highlighting them **(A)**.
- Equally spaced levels may be selected by entering the coordinate of the 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

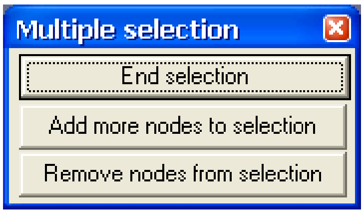
The program remembers the list of selected nodes every time nodes are selected. The nodes from the previous selection are highlighted and the option continues according to **Multiple selection.**

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).

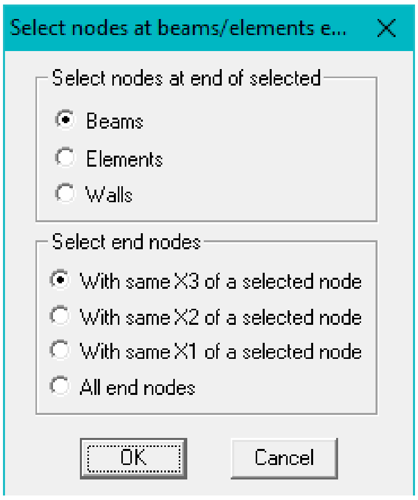
Submodels (results only)

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 -

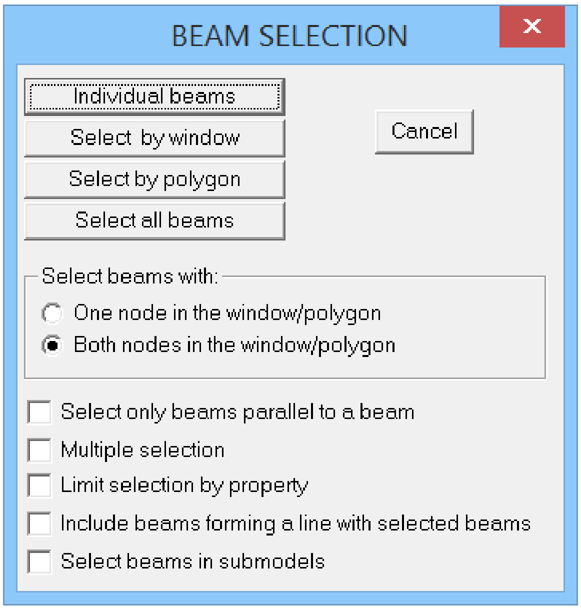
- Select all beams
- select the node at the base of one of the columns

5.6.2 Beam selection

Many options include instructions to select one or more beams.

When selecting a beam/element, the beam nearest to the 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.



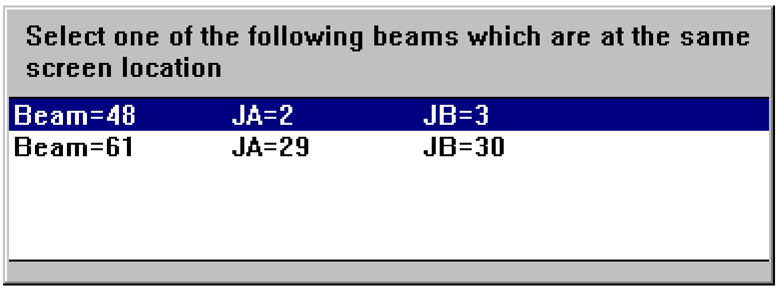
Individual beams

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:

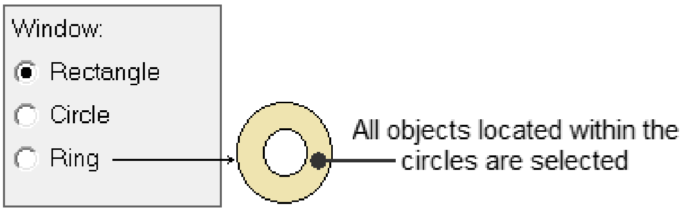


Highlight the line with the correct beam and click the mouse.

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 58)

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 58)

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). The program selects ***all*** of the beams at that location.

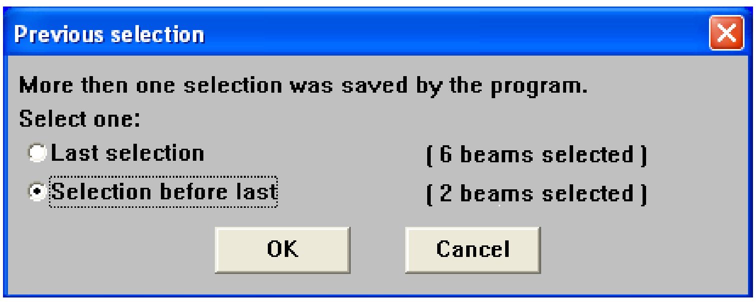
Select all beams

All beams in the model will be selected.

- 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

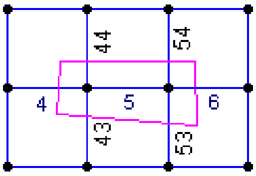
The program remembers the list of beams every time beams are selected. The beams from the previous selection are highlighted and the option continues according to **Multiple selection.** The program remembers up to 5 previous selections and you may select any of them. For example:

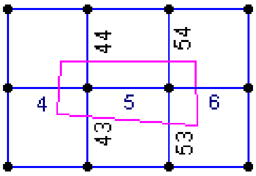


Select beams with

- **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

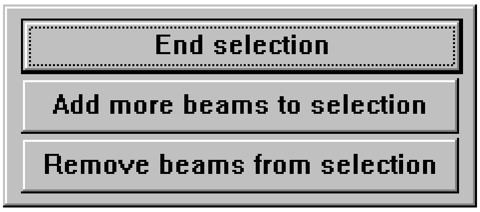
Select only beams parallel to a beam

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:

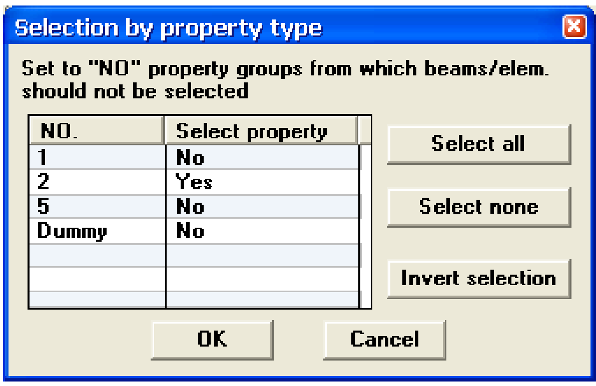
Select:



- 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:

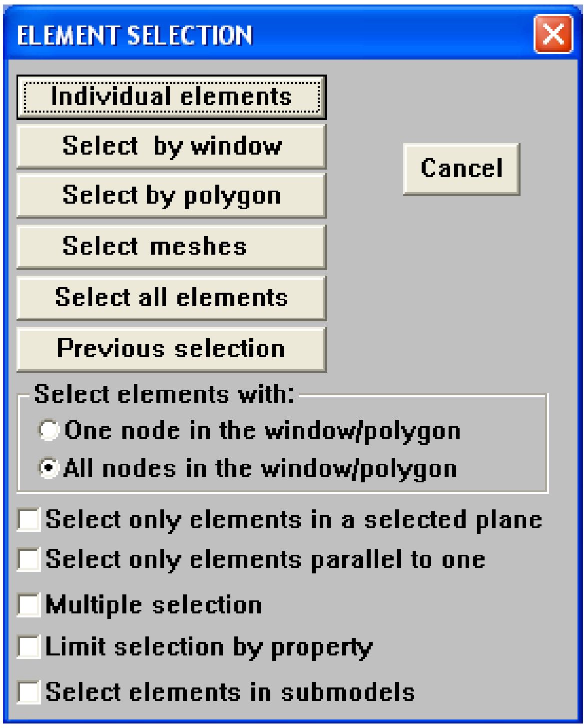
☑ the program allows you to select beams in the main model and the submodels
☐ only beams defined in the main model may be selected.

5.6.3 Element selection

Many options include instructions to select one or more elements.

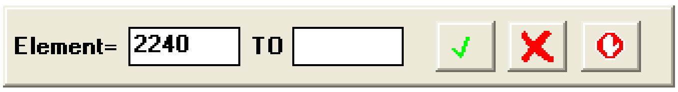
When selecting a element, the element nearest to the is highlighted with a rectangular blip ■. The number of the highlighted element is always displayed at the left-hand side of the Dialog box.

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

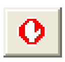


Individual elements

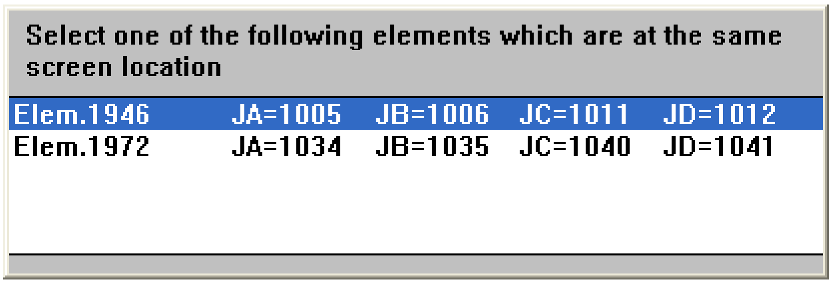
Select a single element only by moving the alongside the element until it is highlighted by a rectangular blip ■; click the mouse. The number of the highlighted element is always displayed at the left-hand side of the Dialog box:



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

Refer to Beams - select by window 57.

Select by polygon

Refer to Beams - select by polygon 57.

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.

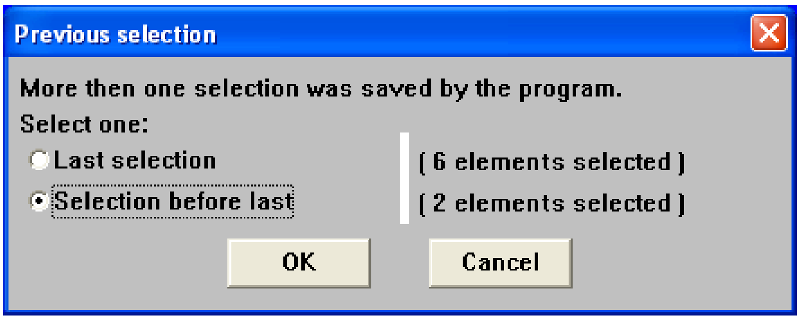
Select all elements

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



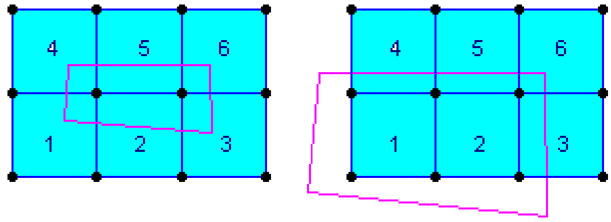
Select elements with:

- **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

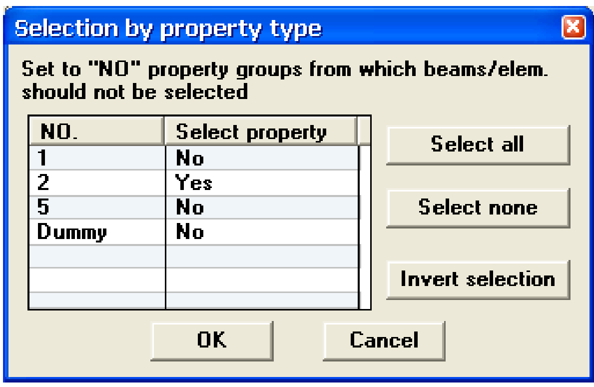
You may impose a further restriction that only the elements parallel to a specified element will be selected. This option allows you, for example, to define a window around several adjacent levels but to select only one of them.

Multiple selection

Refer to Beams - multiple selection 58

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)

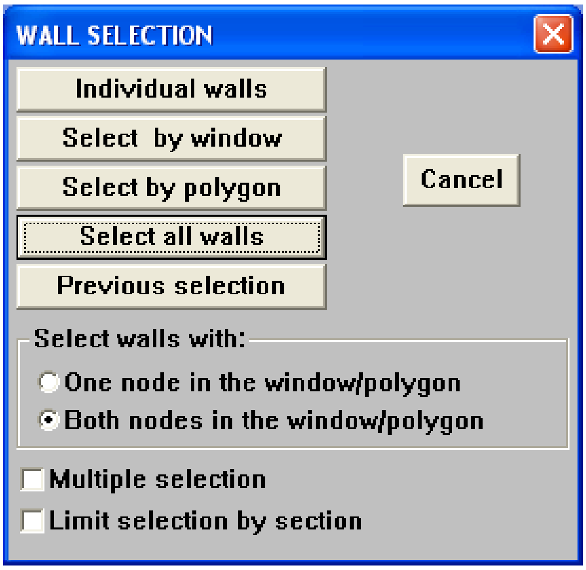
This option is available in results when the Main model is displayed:

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

5.6.4 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

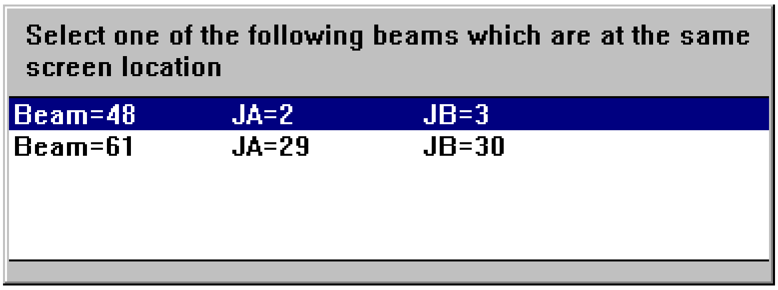
Select a single wall only by moving the cursor alongside the element until it is highlighted by a rectangular blip ■; click the mouse.

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 the button or click the mouse without moving the cursor.

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

Refer to Beams - select by window 57

Select walls by polygon

Refer to Beams - select by polygon 57

Select all walls

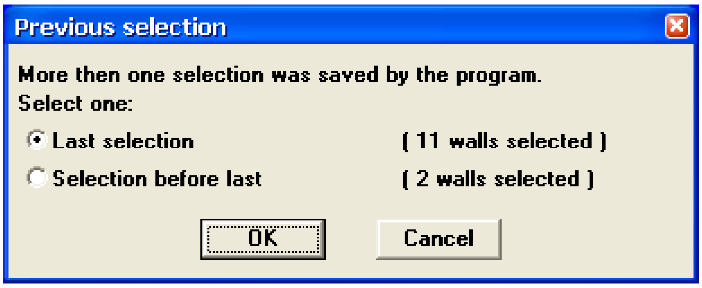
All walls in the model will be selected.

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:

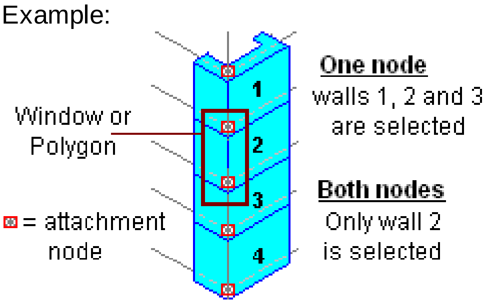


Select walls with:

- **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:

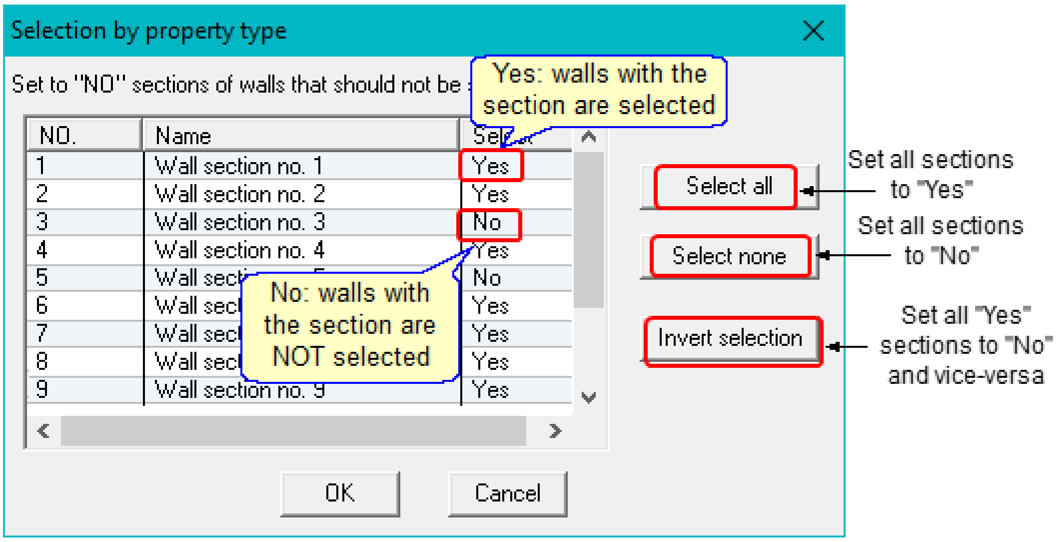


Multiple selection

Refer to Elements - multiple selection 61

Limit selection by section

Further limit the wall selection according to section type:



6 Geometry

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 coordinates
Restraints	Define restrained supports and rigid links.
Beams	Define beam 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, including location of the element, properties, materials (including orthotropic).
Springs	Define elastic supports.
Solids	Define general solid elements, 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 and location
Copy	Duplicate a portion of the model 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'[88]. 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". 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.

6.1 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[66] only.
Line - eq.	Define a series of nodes[71] along a straight line with equal spacing between them. The nodes are equally spaced along an arc when defined using a cylindrical coordinate system.
Line - uneq.	Define a series of nodes along a straight line with varying spacing[71] between them. The nodes are equally spaced along an arc when defined using a cylindrical coordinate system.
Grid	Define a parallelogram grid of nodes[72]. Define a 'base' and a 'height' line (similar to the previous two options) by selecting three corner nodes. When defining nodes using a cylindrical coordinate system, this command creates a series of parallel or concentric arcs.
Equations	Define a series of nodes using an equation, 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 an existing node to a new location.
Delete	Delete an existing node.

Renumber Assign a new node number to an existing node.

System Select a coordinate system 73 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 at those locations.

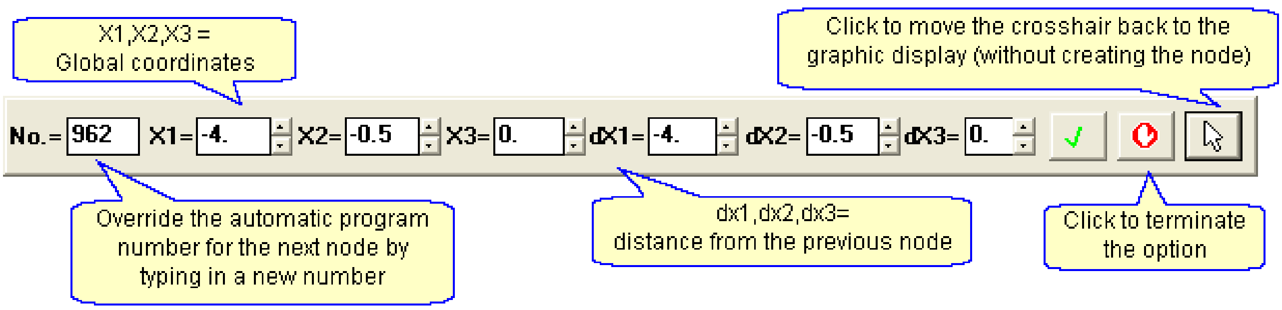
6.1.1 Single Node

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

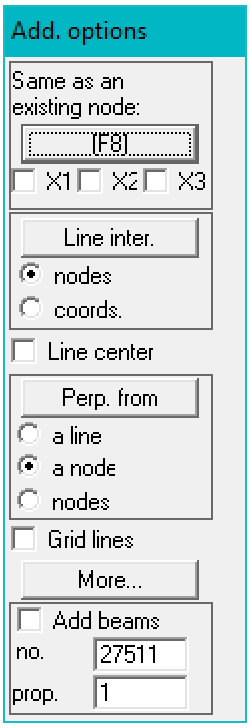
The cursor moves along the current 'working plane' (normally one of the Global planes), even if the the model is rotated. The two coordinates on the plane may be defined as explained for *Plane models*, but the third coordinate must be defined in the Dialog box:

- 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.

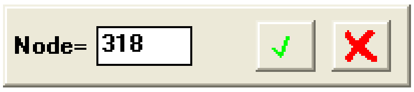
Refer also to:
Additional options 67
Cylindrical coordinate system 73

6.1.1.1 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].

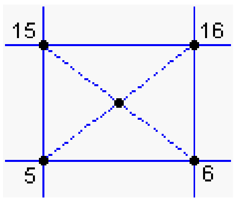


Intersection

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.

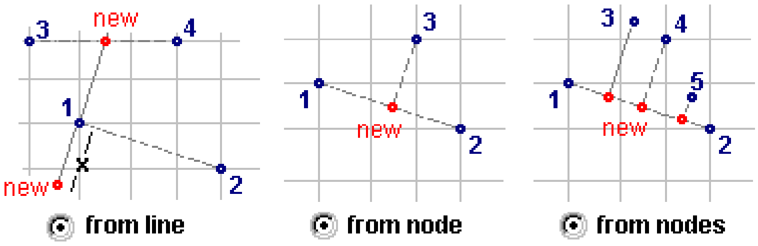
Line center

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

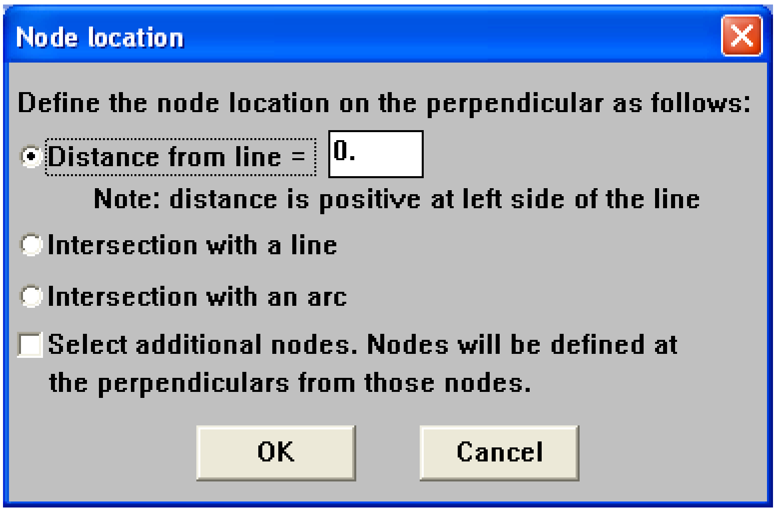
A node (single node, start/end node of a line, etc) may be defined at:

- 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

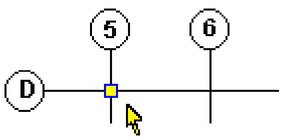
select nodes **1**, **2** and **3**; the new node is created at the perpendicular from node **3** to the line joining nodes **1** and **2**.

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.

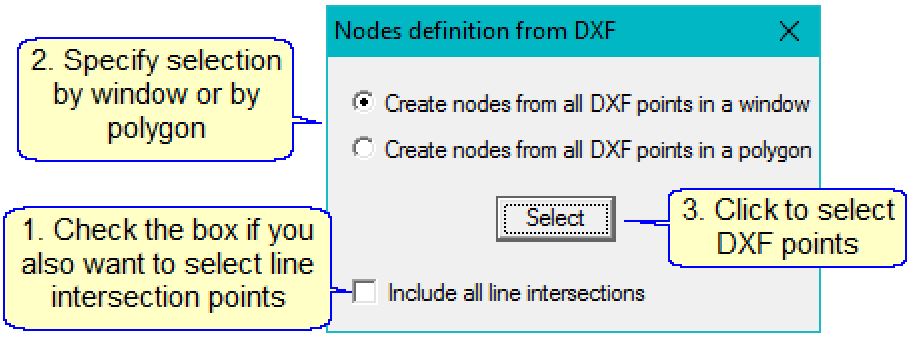
DXF points

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 :



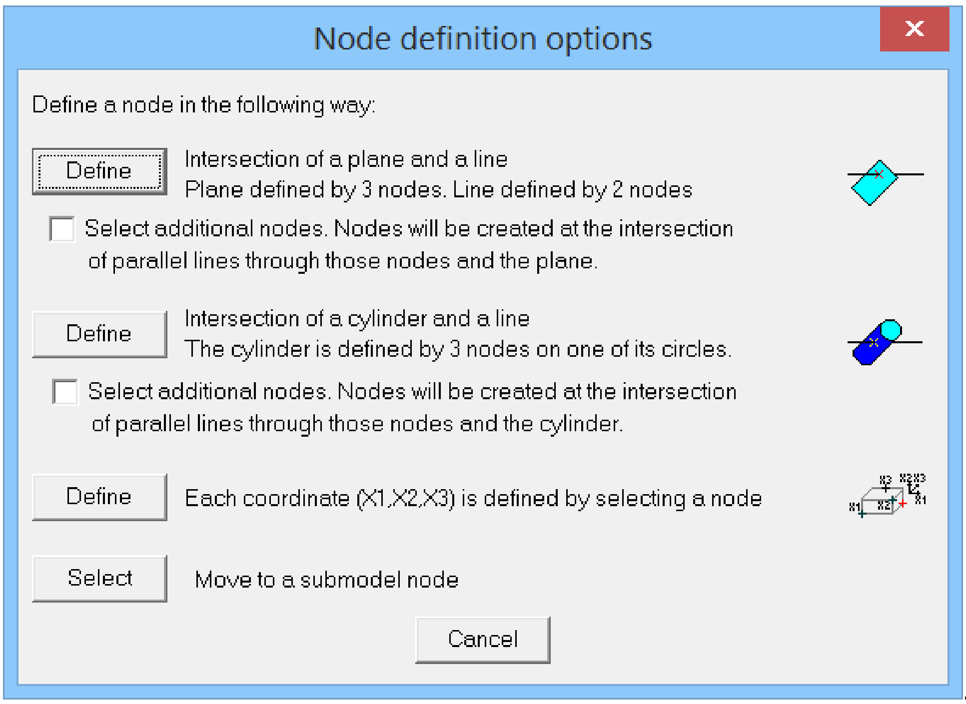
Add beams

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).

6.1.1.2 'More' Additional options

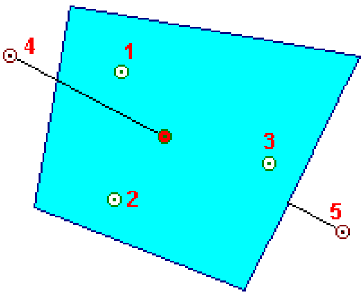


.

Intersection of plane and line

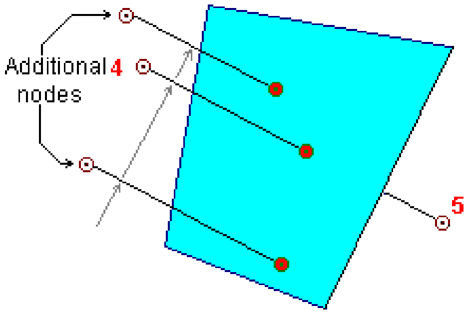
- Select any three existing nodes that define the plane
- Select any two existing nodes that define the line

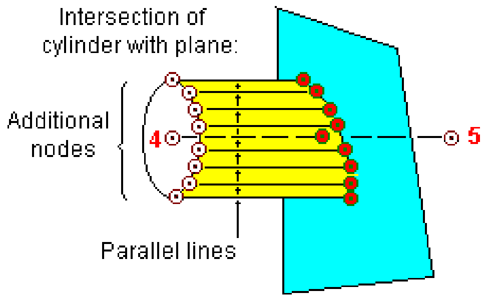
The five nodes cannot lie on the same plane.



☑ **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:

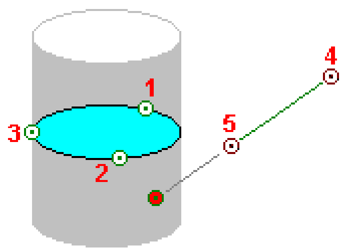




Intersection of cylinder and line

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.

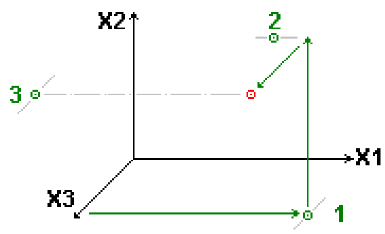
☑ **Select additional nodes**

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

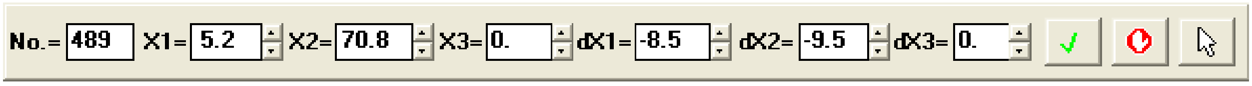
- the program displays a list of submodel instances; select one.
- the program displays the submodel; select a node
- the program restores the previous display and places the crosshair at the selected submodel node location (even if the submodel has been removed from the display). The crosshair can now be moved an additional dX1/dX2/DX3 from the selected node to define multiple nodes.

6.1.2 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 66
- define the end node location:

  The dialog box at the bottom of the screen is:



where:

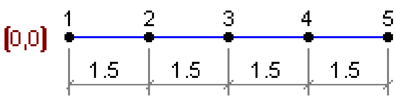
X1,X2,X3 = location on the screen (global coordinates)

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:



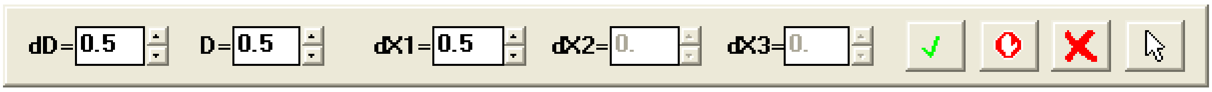
- move to: X1 = 0.0 X2 = 0.0 ; click the mouse
- move to: X1 = 6.0 X2 = 0.0 ; click the mouse
- specify four segments

6.1.3 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 71.
- 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.

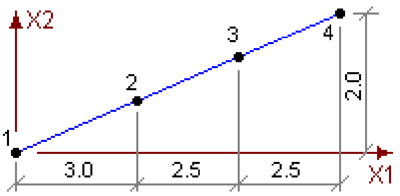
dX1,dX2,dX3 = distance ***from the last node defined on the line*** (global coordinates).

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:



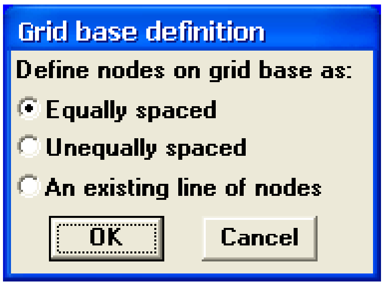
- move the to the location of Node 1 and click the mouse.
- move the to the location of node 4 - dX1 = 8.0, dX2 = 2.0 and click the mouse.
- move the along the line until dX1 = 3.0 is displayed; click the mouse; node 2 is created.
- move the along the line until dX1 = 2.5 is displayed; click the mouse; node 3 is created.
- move the outside the line and click the mouse.

6.1.4 Grid of nodes

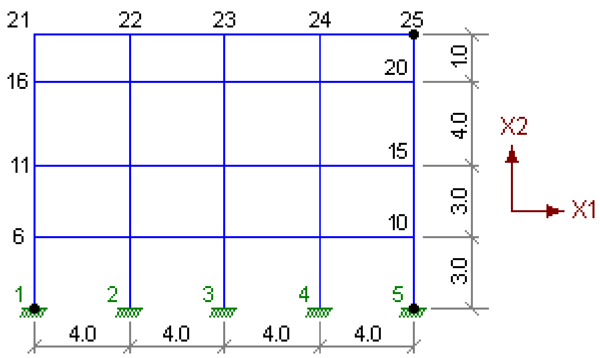
A parallelogram grid of nodes is defined by specifying:
- the nodes on the "base" line of the parallelogram
- the nodes on the "height" line of the parallelogram

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 71 and Line - Unequal 71, 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

6.1.5 Node coordinate system

By default the cursor 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:

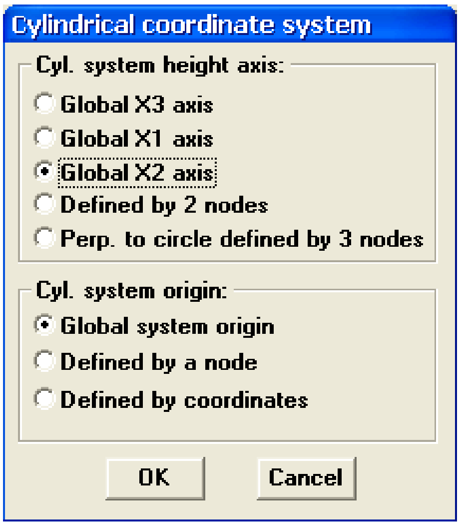
- the cursor will move along the plane of the Cartesian system (referred to as the "Working Plane") or along the arc of the cylindrical system.
- node coordinates are defined relative to the axes of the new system.

6.1.5.1 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

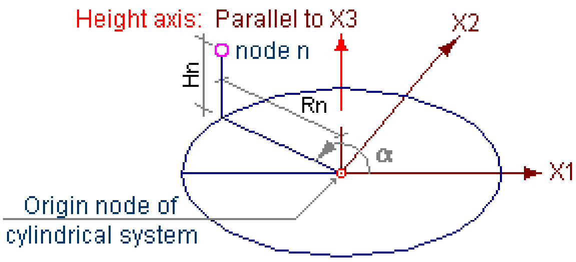
A cylindrical coordinate system has a central non-cylindrical 'height axis' which may point in any direction and which passes through a selected 'origin' node:



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

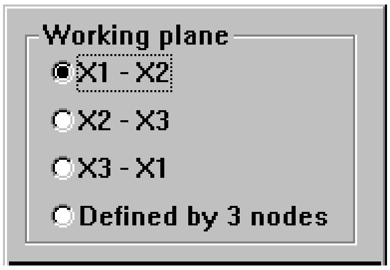
For example, if X3 is chosen as the "height axis" and the origin is set at any arbitrary location, the cylindrical coordinate system is:



The options **Line equal, Line general** and **Grid** may all be used while a cylindrical coordinate system is in effect.

6.1.5.2 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.

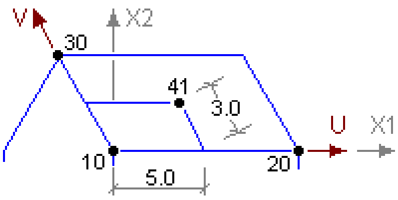
- U is measured from the first node in the direction of the second node.
  The value of U is revised by moving the mouse horizontally.
- V is measured perpendicular to U in the general direction of the third node.
  The value of V is revised by moving the mouse vertically.
- W is the dimension perpendicular 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).

6.2 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.

Fixed	Define supports where all translation ***and*** rotation global degrees-of-freedom are restrained.
Pinned	Define supports where only translation global degrees-of-freedom are restrained.
Other	Define supports with any other combination of restrained global degrees-of-freedom.
Rotate	Define a 'local' support coordinate system (a system for supports not parallel to global axes) and to assign supports to these systems.
Rigid links	Connect selected nodes by means of "Rigid links" (Master-slave nodes).
Delete	Delete supports; Select the nodes with the defined support using the standard Node Selection option.

6.3 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 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. 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 beams that all start on a common line and end on a different common line.
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.
Delete	Delete an existing beam.
Properties	Define section properties (including material) and assign them to beams.
Releases	Release degrees-of-freedom (translational or rotational) at beams.
Offset	Define rigid offsets at the ends of beam elements and assign them to the beams.
Renumber	Renumber a beam element already defined.
Split	Divide an existing beam 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 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.

6.4 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:

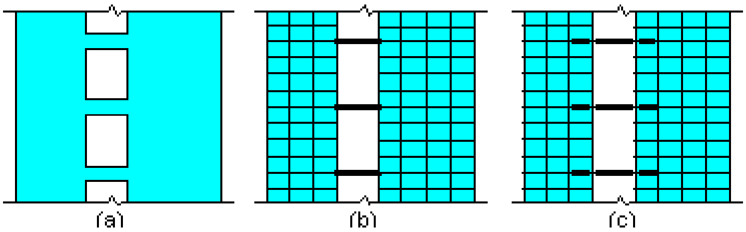
triangle	• Define one triangular element by identifying its end nodes. • Define a chain of triangular elements where only the third node of successive elements need be defined; the program uses two nodes of the previous element (common to both elements) to complete the new triangle.
quad.	• Define one quadrilateral element 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 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: 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 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 elements already defined.
Renumber	Renumber elements already defined.
Properties	Define element properties (including material) and assign them to finite elements.
Local axes	Revise the direction of the local x3 axis. 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".
Offset	Define offsets 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	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).



6.5 Slabs

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:

Define	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	Delete slabs already defined or convert them to regular elements
Revise	Revise the parameters of an existing slab element..
Releases	Define the edges of slabs as "pinned".
Renumber	Renumber slabs already defined.
Show mesh	Display the mesh of elements that the program creates for the slabs. Click End mesh to continue to the slab display.

Click Stages to select a stage other than **Whole model;** different properties may be defined for each stage and slabs may be removed or restored:

Deactivate	'Remove' a slab from the current stage.
Restore	Restore an slab to the current stage.
Properties	Define/revise slab properties and assign them to selected slabs.

6.6 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 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 values on the graphic display (for defined springs).
Unidirectio...	Define translational springs that act in either the positive or negative direction only of the selected axis, but not both.
Area/line	Define spring constant per area/length and then select elements/nodes defining area/length; program will automatically calculate spring constants for relevant nodes.
Non linear	Define non linear springs. These springs have a a spring constant that varies with the deflection.
Gap	Define a gap element. A gap does not transfer forces until it is closed and then it acts as a regular node.

6.7 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 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 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 of the original block about an axis of symmetry.

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.
- 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.

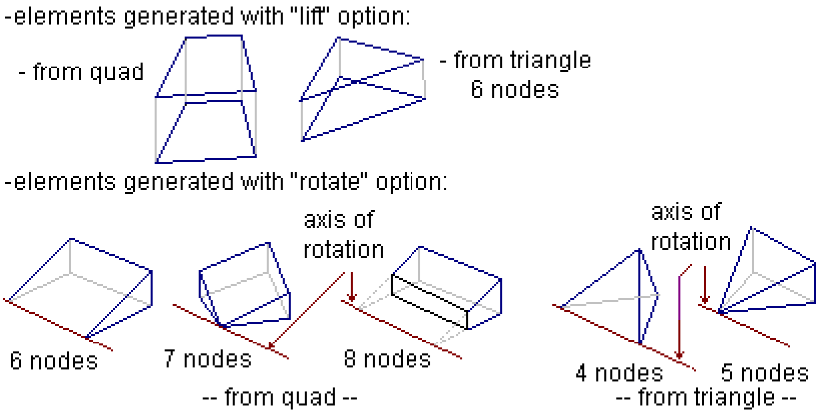
6.8 Solids

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**

Lift	Generate solid elements by "lifting" existing plate elements. Refer to the Figure above for examples.
Rotate	Generate solid elements by "rotating" existing plate elements. Refer to the Figure above for examples.
Delete	Delete existing solid elements; select the solid elements to delete using the standard element selection option.
Renumber	Renumber existing solid elements
Material	Define the material properties for solid elements
Single	Define a single element by specifying the corner nodes

Click Stages to select a stage other than **Whole model;** different properties may be defined for each stage and elements may be removed:

Deactivate	Solids may be removed from the current stage or restored.
Restore	Restore an element to the current stage

6.9 Walls

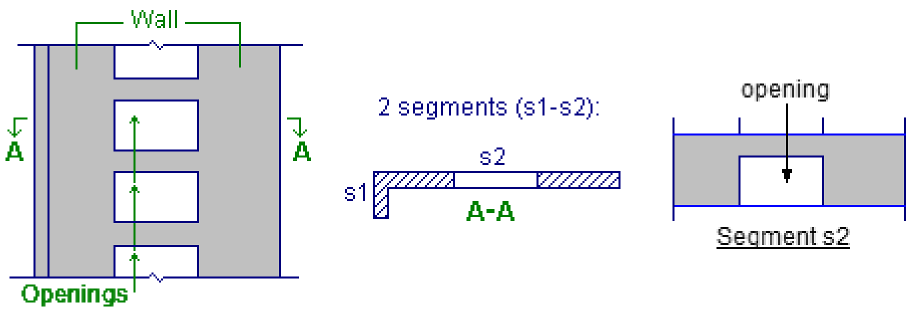
Define wall elements consisting of one or more segments and attach them to nodes in the model. For more information, refer to Walls - general 83:

Single wall	Add a single wall element between two nodes
Line	Add a series of identical wall elements attached to a line of nodes
Section	Define the section of a wall element consisting of a series of connected wall segments and coupling beams
Delete	Delete wall elements from the model; select walls to delete using the standard wall selection option.
Rotate	Rotate wall elements about their "reference point" (elements will remain attached to the same nodes)
Renumber	Renumber wall elements
Link	Create rigid links connecting the wall elements to nodes that are located within the width of the wall segments.

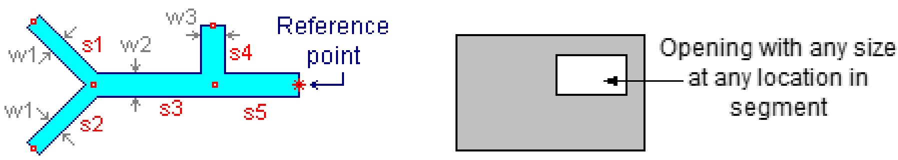
6.9.1 General

The Wall option enables the quick definition of complex walls that extend across multiple levels in the model.

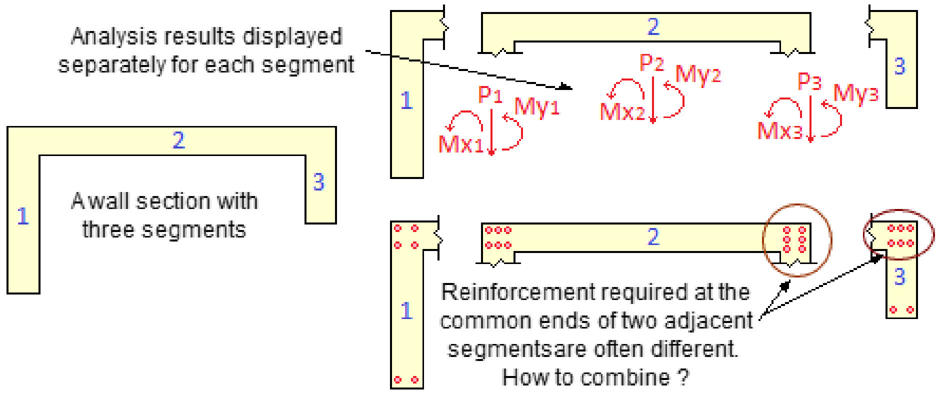
- the wall cross-section is defined first and then is attached to the model by selecting a vertical line of nodes; the program automatically creates any additional nodes that are required
- each wall section may consist of multiple segments together with openings, for example:



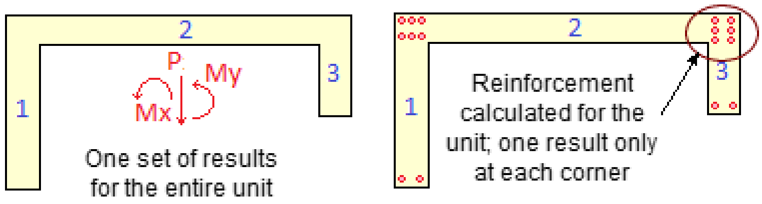
- the number of segments is unlimited, the arrangement and orientation of the segments is not restricted. Each segment may have a different width. For example:



- 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 one 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:

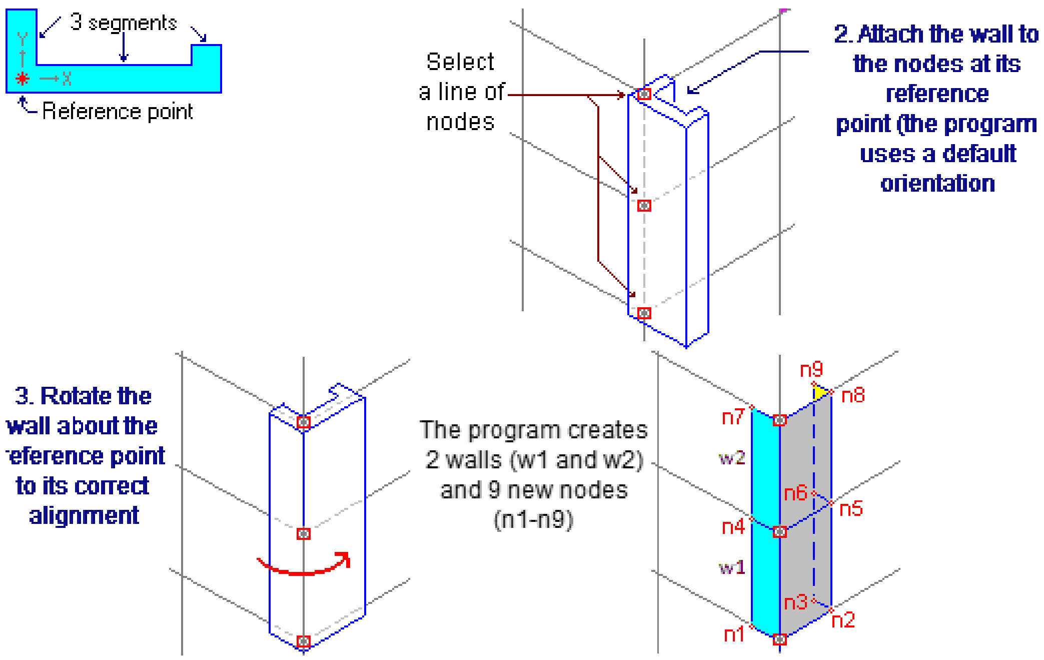


If the segments are combined into one "design unit":



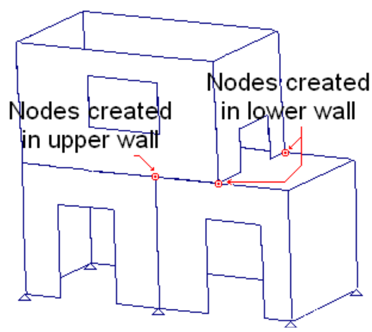
When the wall is attached to the model, the program generates a series of rectangular elements from the segments and creates the necessary nodes at the corners.

Example: the following wall is defined and attached to the model:

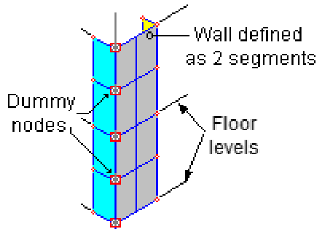


Note:

- 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:

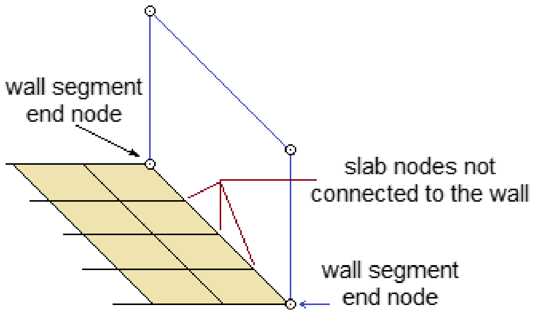


- The program creates a single element between nodes along the height axis for each segment. Normally this implies a single element per storey height. Testing has shown that the results are sufficiently accurate for *typical long multi-storey walls*, i.e. more than 4-5 storeys (refer to the *STRAP* Verification 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.

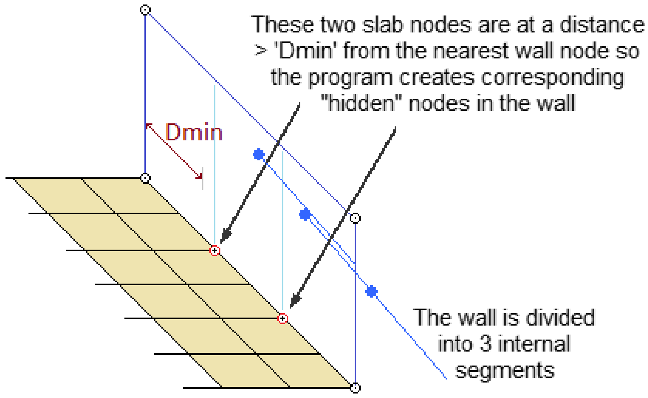


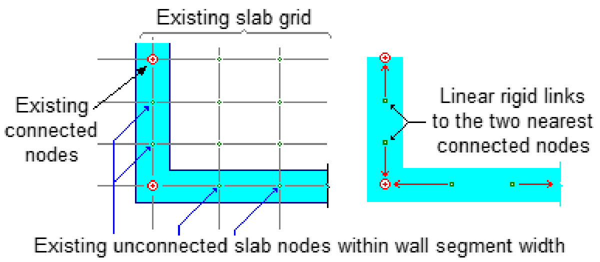
- The wall segments are connected to the adjacent parts of the model only at their end nodes. However there may be existing nodes in the model that lie along the wall segment boundary but are not connected to the wall. For example:

  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") .

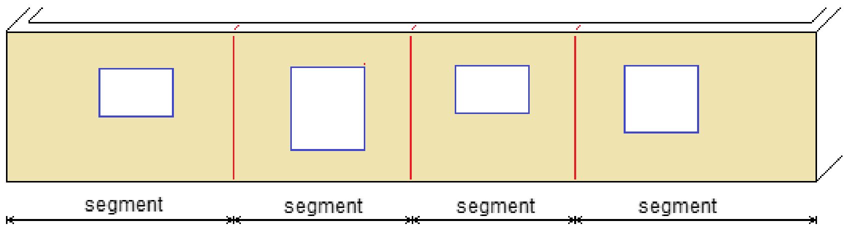




- The results for wall elements are presented in the form of beam results - one axial force, moment and shear value for each generated element.

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:



6.10 Submodels

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:

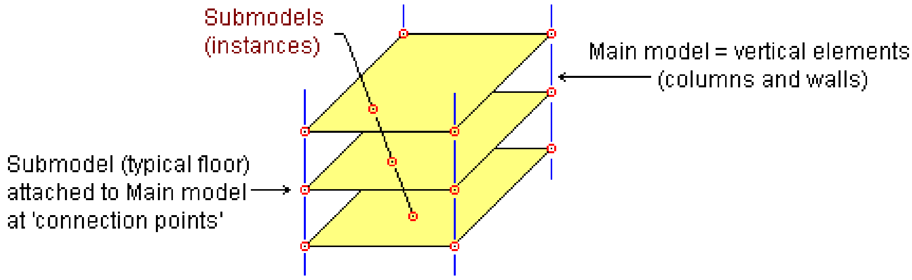
If the submodels are added as ***submodels***:
- the main model and ***each submodel*** may be defined with the maximum number of nodes and/or elements (32,000); multiple instances of each submodel may be added to the main model, i.e. the total size of the model (main model + submodel instances) is ***unlimited***. (see note 87)
- 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.
- 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 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.

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.

6.11 Stages

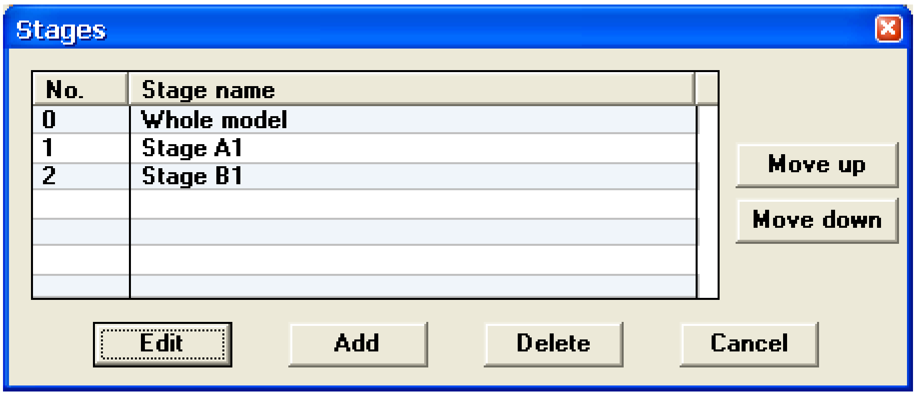
Define construction 'stages' for the current model:

- 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.

For example:



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.

7 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.

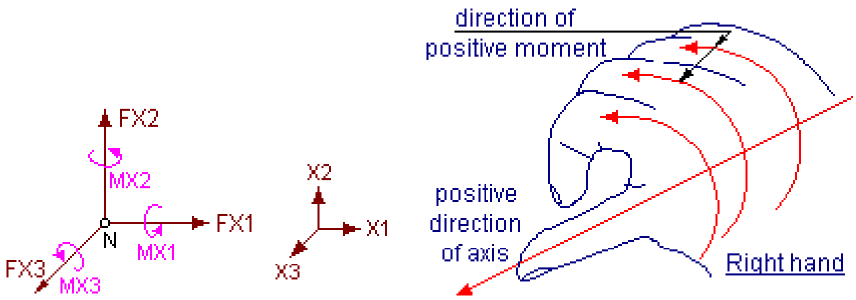
New load	Begin the definition of a new loading case.
Existing load	Revise the loads in an ***existing*** load case.
Delete	Delete an existing load case.
Deactivate	Deactivated load cases are not solved but are not erased.
Moving load	Automatically generate a series of "moving" load cases from a single 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" 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) forces and moments.
Wind load	Define wind 93 load cases according to Code requirements.
Copy load	Copy an entire load case.
Sway	Define sway unit load cases at specified nodes. These load cases are required for the sway/drift control option in the Steel postprocessor.
Combinati...	
Solve	Solve 95 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.

7.1 Joint loads

Define the joint loads - forces and moments.

- 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.

The positive force and moment conventions are:



7.2 Beam loads

Beam loads are uniform, linear or concentrated loads applied anywhere along the length of a 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

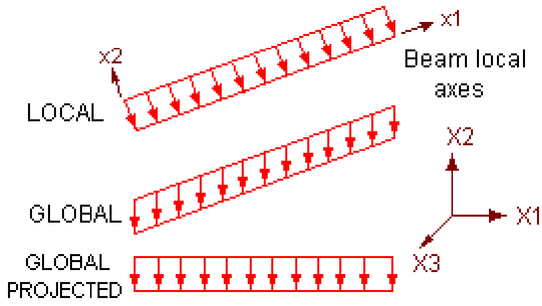
Moments: select the axis that the load acts ***about***.
MX1 - about local x1 - relevant for grids and space frames

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).



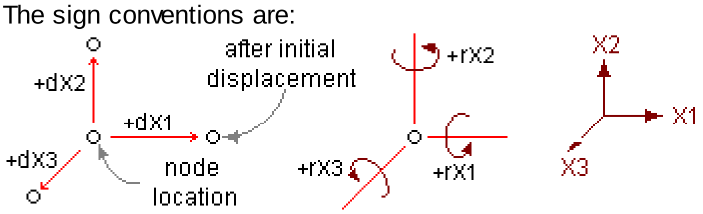
7.3 Element loads

Element pressure are applied to the entire surface of quad or triangular elements. 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.

7.4 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.

The sign conventions are:



7.5 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:
- combinations may also be defined in the results module after the solution. It is strongly recommended that combinations be defined ***after*** the solution rather than at this stage as results can be obtained for new or revised combinations without solving the model again, however -
- 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***, rather than in the post-processor. The same applies for tension/compression only members and for unidirectional springs.
- 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. 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.

7.6 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.

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.

7.7 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.

7.8 Wind loads

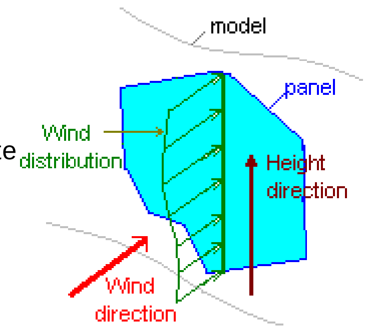
This option generates wind loads according to Code requirements and applies them as loads either on panel or lattice members.

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 Wind load icon
- select "**Define a new wind load case**"
- specify the load application method for the wind loads:
  - contour area applied as beam/element/joint loads
  - selected beams and perpendicular width (for plane models).
- define a contour outlining the first panel / select beams and define width
- define the Code wind load parameters
- repeat for additional panels (all additional panels use the same parameters load application method).
- 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:

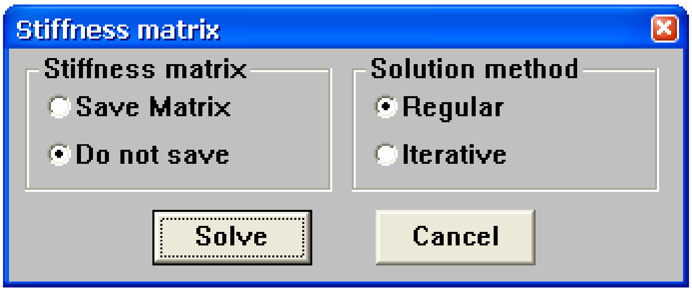
- click the Wind load icon
- select Revise a wind load case and select the case
- for panels defined by contour: revise the load application method (beam/element/joint loads).
- for all wind load types: revise the contour/beam list, the parameters of an existing panel and/or add new panels

Note:

- a wind load case defined by contours cannot be revised to a case defined by beams & width, or vice-versa.

8 Solution

Refer to Solution method 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.

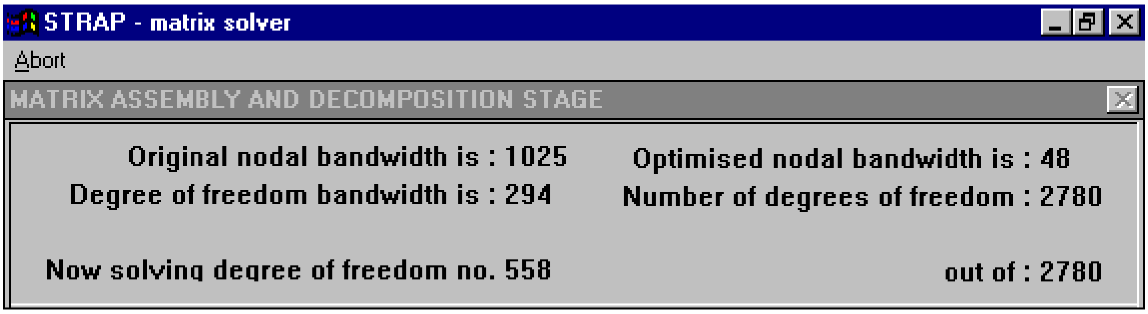
This method is recommended for:
- small and medium sized models
- models with non-linear elements (e.g. one-way springs)
- models with dynamic analysis and a large number of mode shapes
- models with a large number of load cases

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

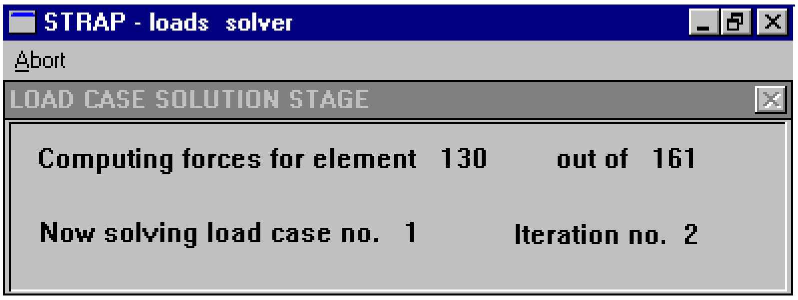
Note
- models with a bandwidth > 16,000 can be solved using this method only.
- the Bridge module does not recognize the matrix saved by the Iterative method.

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 97 or Troubleshooting 97.
- 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:

8.1 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.

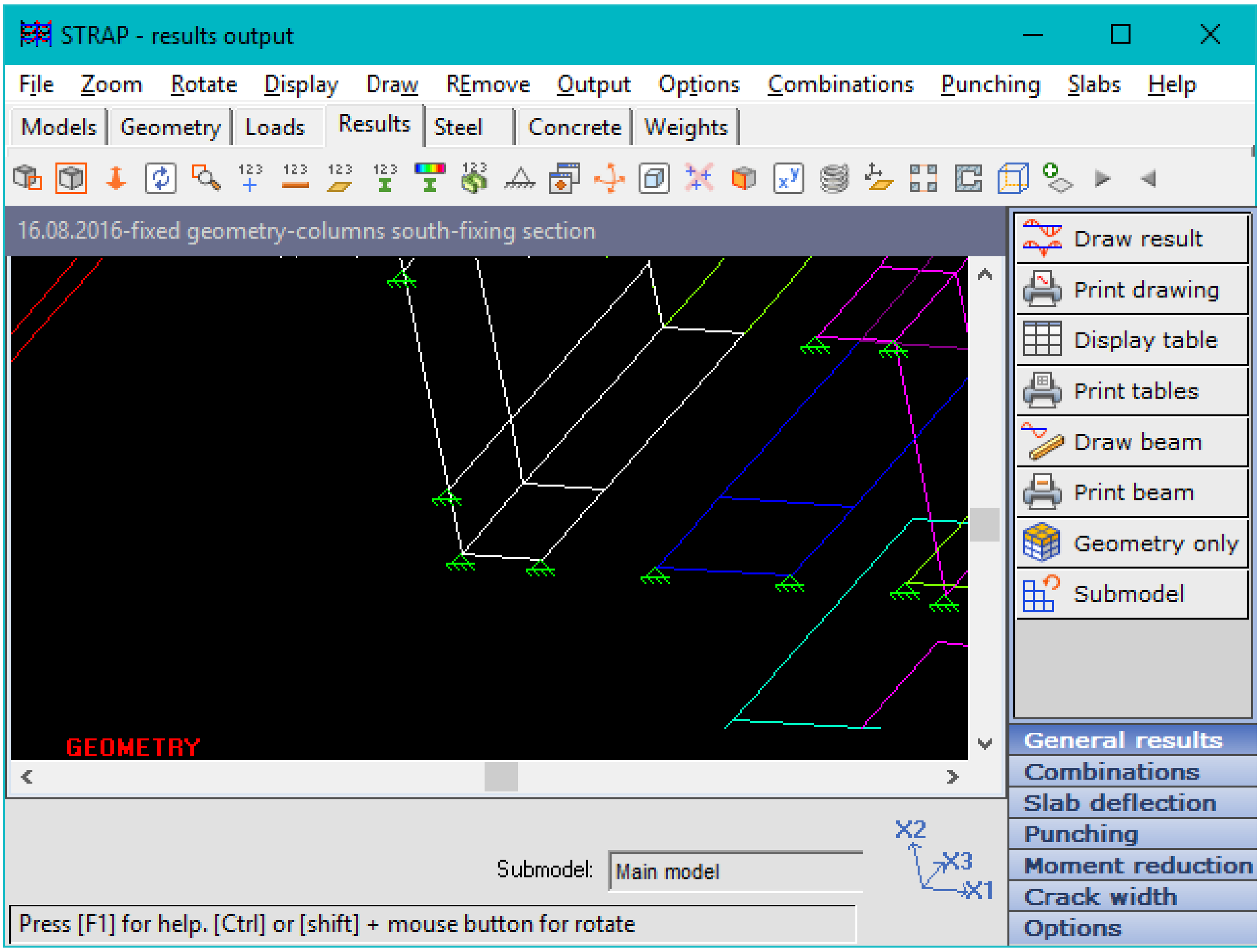
  Note:
  - this singularity message may be displayed when there is a large difference between maximum and minimum moment-of-inertia values defined for property groups.

8.2 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.

9 Results



Display graphic/tabular results:

- Define the load Combinations
- Specify Options, e.g result axes
- Click:

Draw result	display the graphic results.
Display ta...	display the tabular results.
Draw beam	to display the results on a single beam or a line of beams.
Geometry ...	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 Geometry ..., rotate the model (without the results), then select Draw result to add the results.
Submodel	select a submodel instance to display

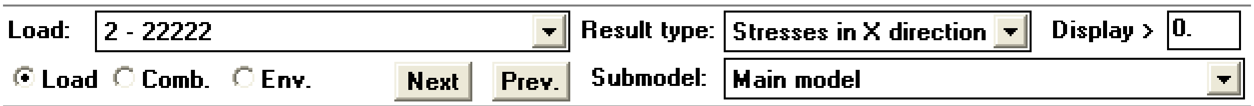
Tabular results for selected beams, elements and nodes may also be displayed using the 'right-click' option.

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 

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 after solving the model using the  option.

9.1 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 102.
- **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°.

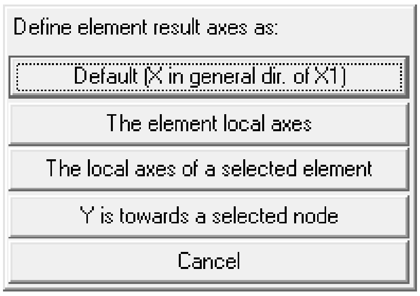
The default Element and Reinforcement Coordinate systems may be revised for each element.

9.1.1 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 102).

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

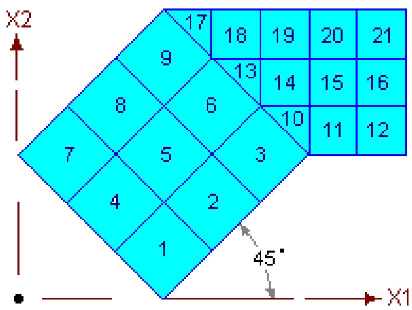
This option restores the default element result axes 102 for the selected elements.

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

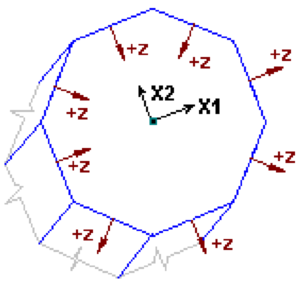
Examples:

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.

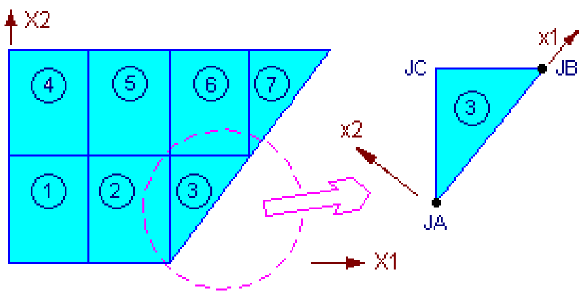
Space frame:



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 101 - for more details.

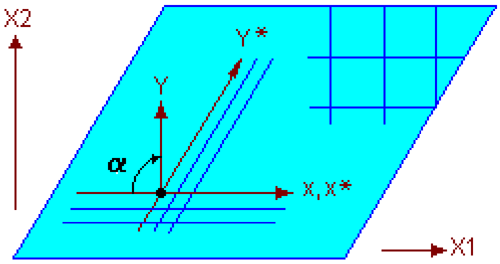
9.1.2 Reinforcement skew angle

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, α = 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 in the direction of X*, Y*.

The following figure shows an example with α ` 90°.



9.1.3 Default system

The default result X and Y axes are determined as follows:

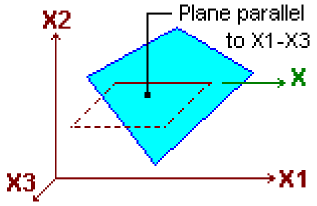
Plane frame, plane grid:
The values plotted are those on the upper surface (+X3 Global). The X direction is always parallel to the global X1 axis and the Y direction is always parallel to the global X2 axis.

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.

9.2 Result combinations

Use this option to define combinations of the load cases for the following output modules:
- tabular results
- graphical results
- design postprocessors

9.2.1 General

- Each "Combination" is defined as a combination of load cases, each of which may be multiplied by a factor.
- Combinations may also be retrieved from a "Combination Library"; combinations defined for the current model may be added to the combination library for use in other models.
- "Groups" of load cases may also be defined. If a group is added to a combination the program either:
  - automatically generates a separate combination for each load in the group, or
  - adds the sum of the load cases in the group to the combination.

Examples:

- **Groups**:
  The following load combination is required: *1.4*Dead + 1.6*Imposed + 1.6*Crane*

  where there are 5 different load cases with Crane loads, each corresponding to a different point of application of the load.

  Instead of defining 5 separate combinations, the 5 crane loading cases may be defined as a Group; then which includes 1.6*"Crane load group" need be defined. The program will then automatically generate 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.

- **Library**:
  Note that standard combinations need not be redefined in every model as 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.

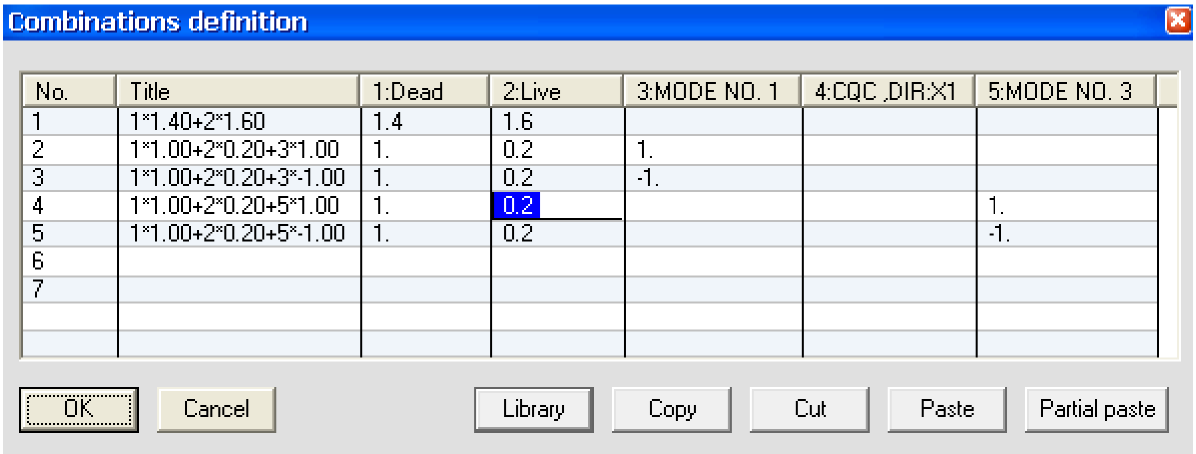
  For each Current Model, retrieve the standard combinations from the Library; then define groups that contain the "dead", "live" and "wind" load cases.

Note:
- the combination library is stored in the file COMB.DAT.
- the COMB.DAT file in the current folder is referred to as the "User library"
- the COMB.DAT file in the program folder is referred to as the "Program library"
- combinations add to the library are always added to the User library (in the current folder)
- create/update the Program library by manually copying the COMB.DAT file from a current folder to the program folder.

9.2.2 Define / revise

The combinations are defined/revised by typing the factors in the appropriate cells in the following list view box. There is a column for each load case (you may have to scroll horizontally if there are many cases). Combinations may also be copied from the Clipboard.

To define a combination:
- move the arrow into the appropriate cell and click the mouse; the entire combination row will be highlighted.
- type in the load factor and press [Enter]; the cursor will move to the next cell in the row.
- the cursor will move to the following row after you press [Enter] in the last cell. The program will automatically generate 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 combination is revised.



Cut / copy / paste

Use these options to:
- delete combinations
- copy combinations
- rearrange the combination list
- copy combination definitions from the clipboard

Delete a combination:

- place the anywhere on the combination line and click the mouse
- click the Cut button

Copy a combination:

- place the anywhere on the combination line and click the mouse
- click the Copy button
- place the anywhere on the line where the copy is to be written and click the mouse (if you select a line with an existing combination, then the copy will be inserted at this point).

- click the Paste button

Rearrange the combination list:

Similar to "Copy a combination", except click Cut after selecting the combination to be moved.

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.

9.2.3 Groups

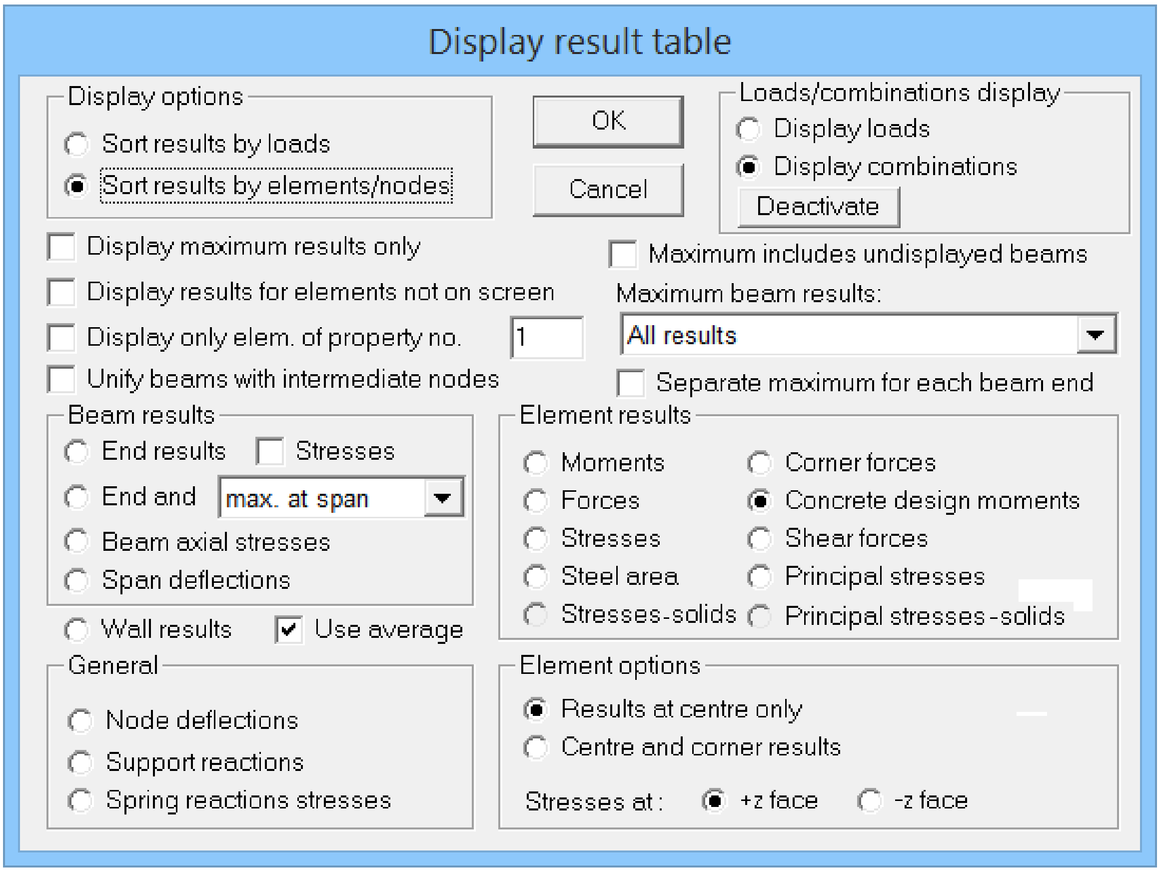
GROUPS of load cases may also be defined. Groups may be defined so that:

- if a group is added to a combination definition, the program will automatically generate a separate combination for each load in the group, or
- if a group is added to a combination definition, the program will automatically add the SUM of the load cases in the group to the combination.

9.3 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 combinations (or load cases, if no combinations were defined); 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.

Example: The following load combinations were defined but the search for the maximum/minimum results should be carried out on load combinations 1,2 and 4 only; load combination 3 must be deactivated:

Delete output for load End

NO.	Deactivated	Title
1		1*1.20+2*1.6
2		2*1.20+3*1.6
3	Inactive	1*1.00
4		2*1.20+3*-1.6

- move the anywhere in line 3 and click the mouse. **Inactive** will be displayed in the "Deactivated" column.
- Repeat for other load combinations when required; click End to return when all combinations have been selected.
- To reactivate a combination, click again with the mouse; the **Inactive** will be removed.

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**"

BEAM RESULTS (Units: ton, ton*meter)

Exit Goto Print Copy

Beam	Axial	V2	V3	MT	M2	M3
8 M3 Max	0.174	-16.786	0.038	-3.055	0.135	37.170
Comb.						4
M3 Min	-0.038	-17.272	-0.004	-7.168	0.012	-33.972
Comb.						2

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:

☐

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

☑

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.

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.

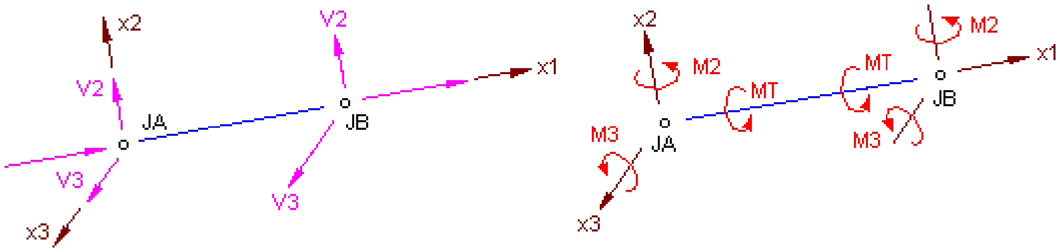
- Element stress results can be viewed either at the top (+z) plane or the bottom plane (-z) of the element. This element is important only in space models.

9.3.1 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.

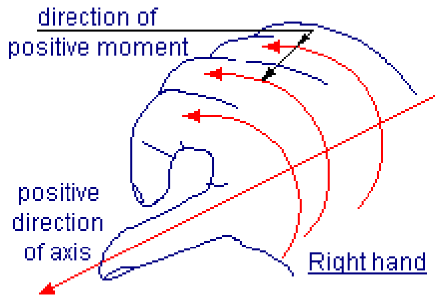
9.3.1.1 Beams

Member results are listed at nodes JA, JB of each element. Results are relative to the local coordinate axes. The positive sign conventions are:

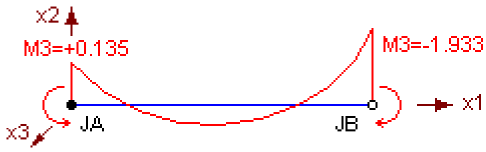


- **Moments:** (M2, M3, MT)

  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).

9.3.1.2 Elements

The following is an explanation of the finite element results:

Note: The results are relative to the element result coordinate system 100.

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:

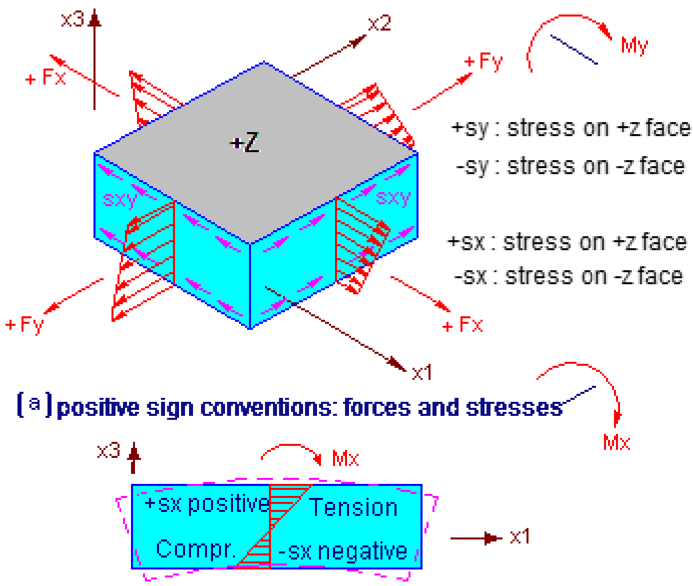
Moments relative to the result coordinate system at the centre of the element. The moments are computed directly from the stresses by:

$$\begin{bmatrix} MX \\ MY \\ MXY \end{bmatrix} = \frac{T^2}{12} x \left(\begin{bmatrix} SX \\ SY \\ SXY \end{bmatrix}_{+Z} + \begin{bmatrix} SX \\ SY \\ SXY \end{bmatrix}_{-Z} \right)$$

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:



[a] positive sign conventions: forces and stresses

[b] positive sign convention: moments

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.

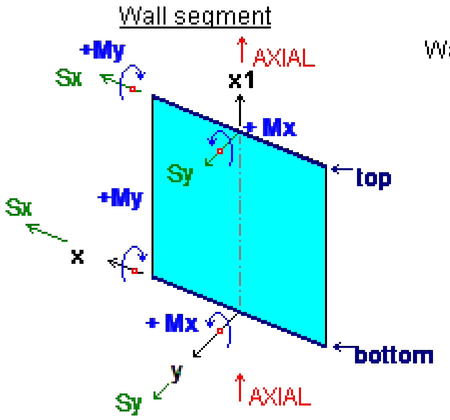
Note:

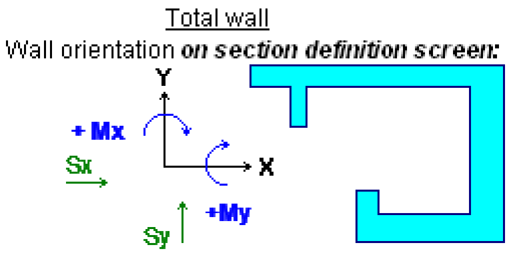
- If results are displayed graphically, the program may modify the sign of the results in order to ensure consistency, etc.

9.3.1.3 Walls

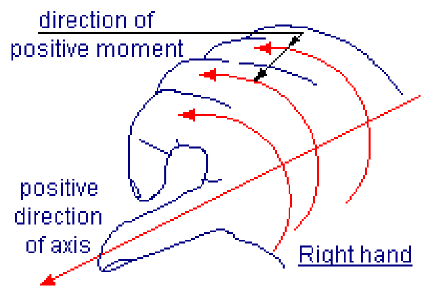
Member results are listed at the top and bottom of each element segment (identified by the corner nodes). Results are relative to the wall local coordinate axes.

The positive sign conventions are:



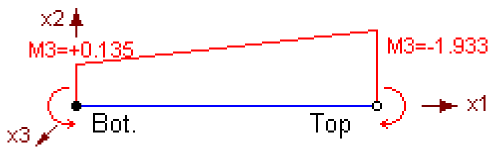


- **Moments:**



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.

9.3.2 Beams

9.3.2.1 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
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 109.

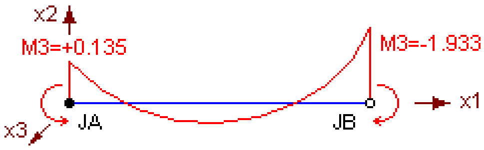
If the **End results and max in span** option is selected, the display is:

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 109, 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.



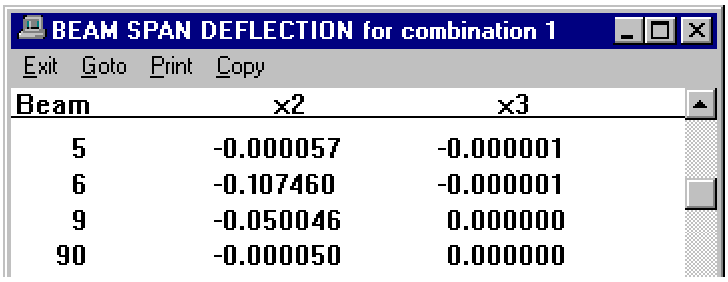
To ensure consistency when calculating the 'maximum/minimum' results, the program ***reverses the sign of the moment at JB***. Therefore, the moment results for the beam in the above Figure are displayed as:

22	1	43	.....		+10.00
			=0.45		-7.50
		44	.....	.....	-20.00
			.....	.....	+20.00
					1
			.....	.....	-7.50
					1

9.3.2.2 Span deflections

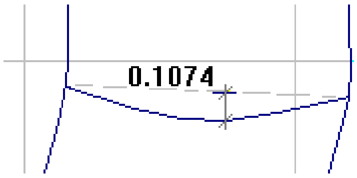
Display the maximum span deflection. The deflection is relative to the final location of the nodes, not the joint coordinates.

For example:



Beam	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:



9.3.3 Elements

9.3.3.1 Moment, force & stress tables

Display :

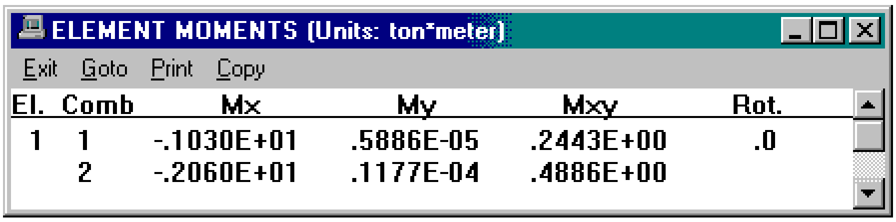
Stresses:

±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 (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 x2 axis.
- a stress distribution with tension on the +z face results in a ***positive*** bending moment; refer to Element sign conventions 109.
- For the equations relating moments and forces to stresses, refer to Element sign conventions 109.

Example (moments sorted by combinations):



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 100 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

9.3.3.2 Shear force results

Display the transverse shear forces Qx, Qy at the element centre.

$$Q_x = \frac{dMx}{dx} + \frac{dMxy}{dy} \qquad Q_y = \frac{dMy}{dy} + \frac{dMxy}{dx}$$

- 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.

9.3.4 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.

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 109

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.

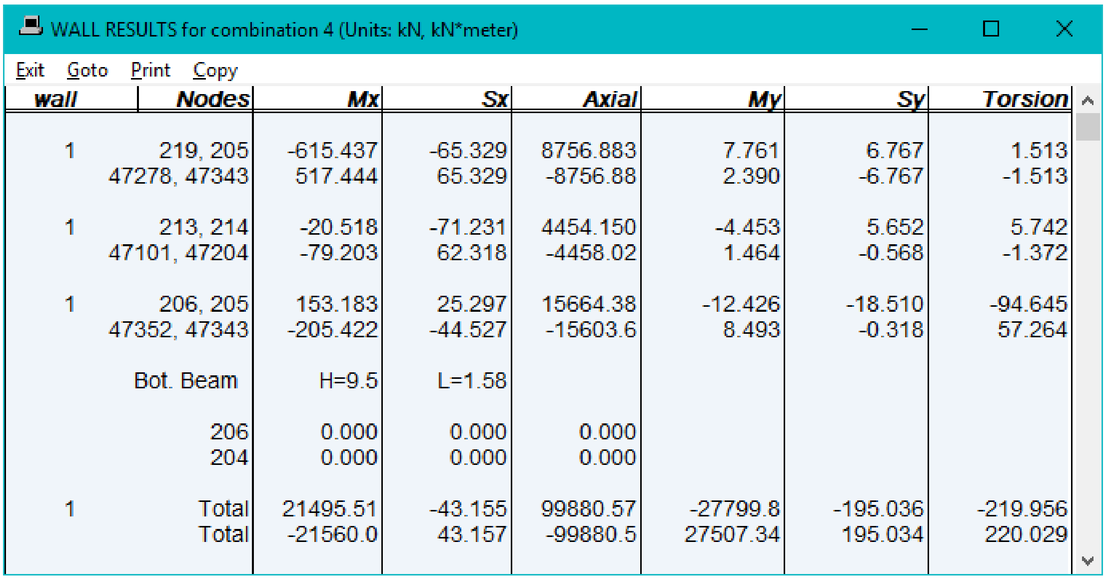
In such cases, the user should check for the following causes:
- Singularity messages during the solution stage.
- Too large a difference between the values of the largest and smallest moment-of-inertia (or area) was defined for the model; The ratio between the largest and smallest values should not 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.

9.3.5 Walls

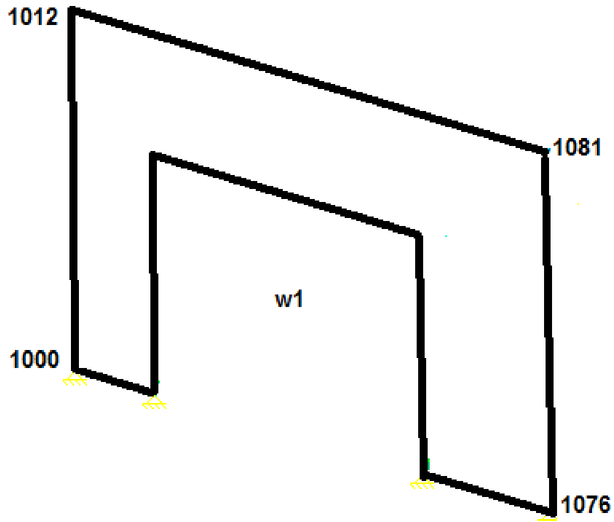
Wall results are displayed for each segment in a wall element, including coupling beams as well as the total wall results. For example:



WALL RESULTS for combination 4 (Units: kN, kN*meter)

Exit Goto Print Copy

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
1	213, 214	-20.518	-71.231	4454.150	-4.453	5.652	5.742
	47101, 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).

- Segments:
  The **Moment** and **Shear** values are segment major axis results; **Mperp** and **Sperp** are the minor axis results and usually will be relatively small. Refer to Wall elements - sign conventions 112.
- Total wall:
  Refer to Wall elements - sign conventions 112.
- Coupling beams:
  Results are identical to regular beams, where **Moment** = **M3, Mperp** = **M2, Shear** = **V2** and **Sperp** = **V3.**

9.4 Graphic results

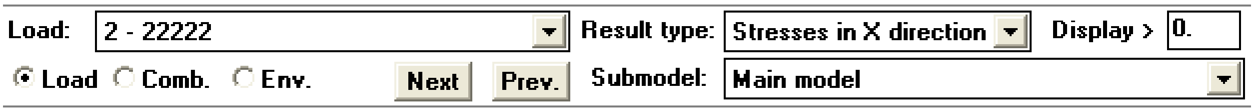
All result types - moments, shear and axial forces, stresses and deflections - for beam elements and finite elements may be displayed graphically on the screen.

The following options are available when **Display types** is selected in the menu:

Geometry
Beam result diagram
Results at element centers
Element results contour map
Element results along a line
Displacements
Reactions
Column/wall results at level
Write beam result
Crack width
Solidelem. contour map
Wall results

The result display may be changed without entering the side menu 

option; the program displays the following menu at the bottom of the screen:



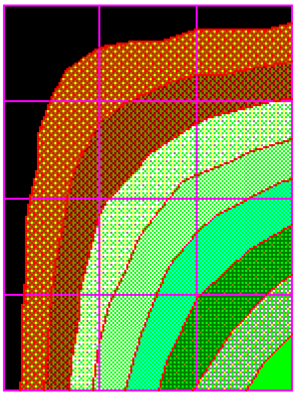
Select a different load/combination, result type or submodel; the display is updated immediately.

9.4.1 Beams

Result values are displayed at the beam ends and at the maximum span result. Refer to Tabular results 113.

9.4.2 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 120,
- Results are displayed according to the Default element result coordinate system 102, unless the result system was revised for specific elements ("Options"/"Element results coordinate system 100" in Menu bar)

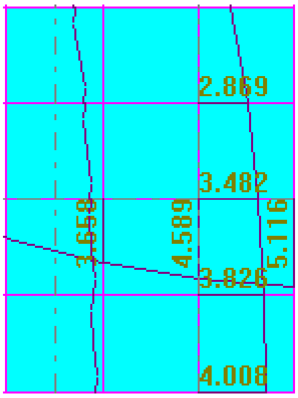
- Reinforcement by diameter and spacing:

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. $v(dX1^2 + dX2^2 + dX3^2)$.
  - **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.

9.4.3 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.***

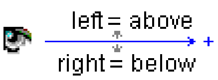
The result coordinate axes are defined as follows:

- **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 are drawn below the line
- moments: moments are positive if they create compression stresses 'above' the element and are drawn below the line

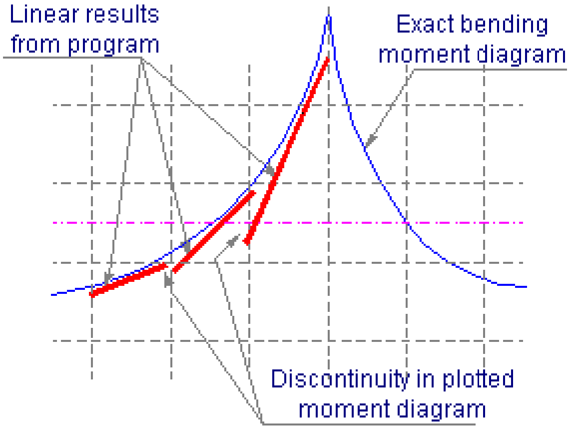
The 'above' side is always to the ***left*** of the line when looking in its ***positive*** direction:



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.



9.4.4 Results at element centres

Display the model geometry with the numerical value of the result written at the centre of each element. For example:

This option superimposes the tabular results at the element Note:

- Results are displayed according to the Default element result coordinate system 102, unless the result system was revised for specific elements ("Options"/"Element results coord. system" in Menu bar)
- the program does not change the sign of the results, ***even if the directions of the local axes are inconsistent***.
- local stress or moment concentrations at element corners, e.g. from a joint load applied at a node or a support reaction, are not displayed. Display Contour map 119 or Results Along a Line 120 for a more complete picture.

10 Dynamic analysis

- This dynamic analysis module analyses 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:

- define the dynamic nodal weights.
- calculate the mode shapes and natural frequency (after nodal weights have been defined).

- display the results of the mode shape and natural frequency calculation.
- carry out a seismic analysis on the model (frame structures).

- 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.

10.1 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.

10.2 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 124 for more information.
- For additional theoretical explanations and background, refer to any textbook covering dynamic response of multi degree-of-freedom systems.

10.3 Method for combining modes

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 = ω_j/ω_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 ($\xi = 0$).

10.4 Forced Vibration & Time-History Response

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 $ß_{cr} = \omega_n$ (ω_n = modal frequency).

11 Steel design

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 - 2010
- CSA/CAN S16-01 - Limit States Design of Steel Structures - 2005.
- 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:
  2007 Edition of the AISI Standard “North American Specification for the Design of Cold-Formed Steel Structural Members” and the 2009 Supplement to the 2007 Edition.
- CSA S136-1994
- Eurocode 3- Part 1.3 - 2003
- BS5950 - Part 5 - 1998 - "Code of practice for design of cold-formed thin gauge sections"

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.

11.1 Sections

The program contains several complete section tables - the "master" tables.
- British sections (CONSTRADO tables)
- American sections (ASTM)
- European sections (Euronorm)
- Canadian
- South African
- Indian

Cold-formed sections may be added to the master tables; 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 for instructions.
- additional "built-up" sections" may be defined by specifying dimensions; the program assumes that these sections are welded shapes.
- Combined (rolled) sections may be defined (2L, I+[,][, etc).

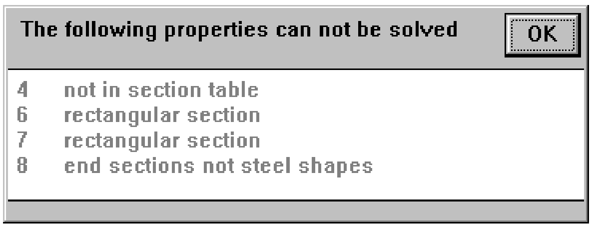
The following terminology is used throughout the program:
- **Master table**: The initial complete table of steel sections (British, American, European, etc. **or** User).
- **Model table**: The section table for the current model, containing sections selected from the master table
- **Section type**: 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 postprocessor
- 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.
  Note that only "Pipe", "Tubes", "L shape" and symmetric "[shape", "I shape" and "T shape" sections is converted to Built-Up sections.
- 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.
- if the property was defined with a steel section not in the current model table.

A warning is displayed at the start of the program. For example:



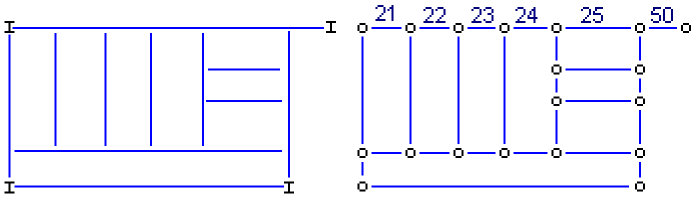
When selecting a suitable section, the program checks sections from a specified list, beginning from the lightest one, until it finds the first adequate section. If the program had to search the complete "master" table when designing each member in the model, the selection process would be very time 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.

11.2 Defining a steel structure from a STRAP model

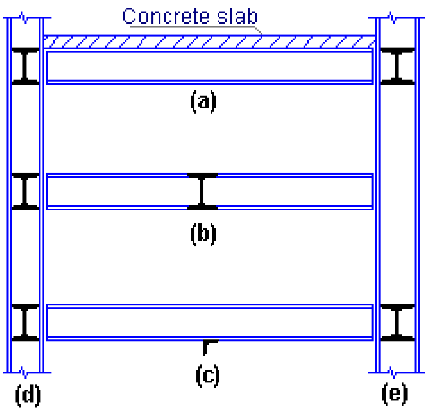
Member selection is automatic; the program designs all beams in sequence without prompting for information. Therefore, all parameters relevant to the design of the members as required by the Code must be specified before member selection begins. In many cases the model geometry as defined in ***STRAP*** 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.

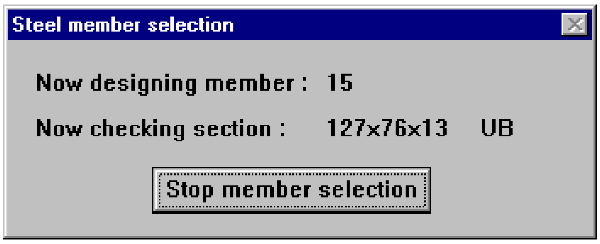
- the section required for each of the beams (a),(b),(c) will be different if the upper flange is entirely in compression. This support data is not available from the ***STRAP*** geometry and must be defined here.
- similarly, column (e) has an unsupported length about its minor axis double that of column (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.

11.3 Compute

When the option "Compute" is selected in the Main Menu, the program begins automatic member selection. 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.

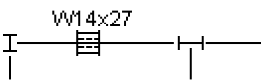
When "Compute" is completed, the program automatically displays the Result Summary.

Note:

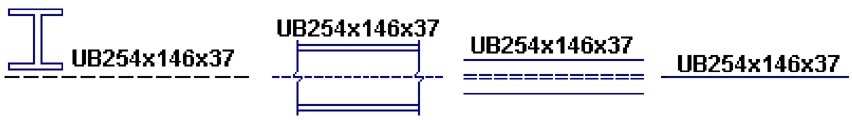
- if the type assigned to a member in the Section option is a cold-formed section, the program automatically designs the member according to the specified cold-formed Code.
- If more than one type is allowed (i.e. if there is more than one type in a group), the program searches for an adequate section 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 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 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 for more details.

11.4 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:



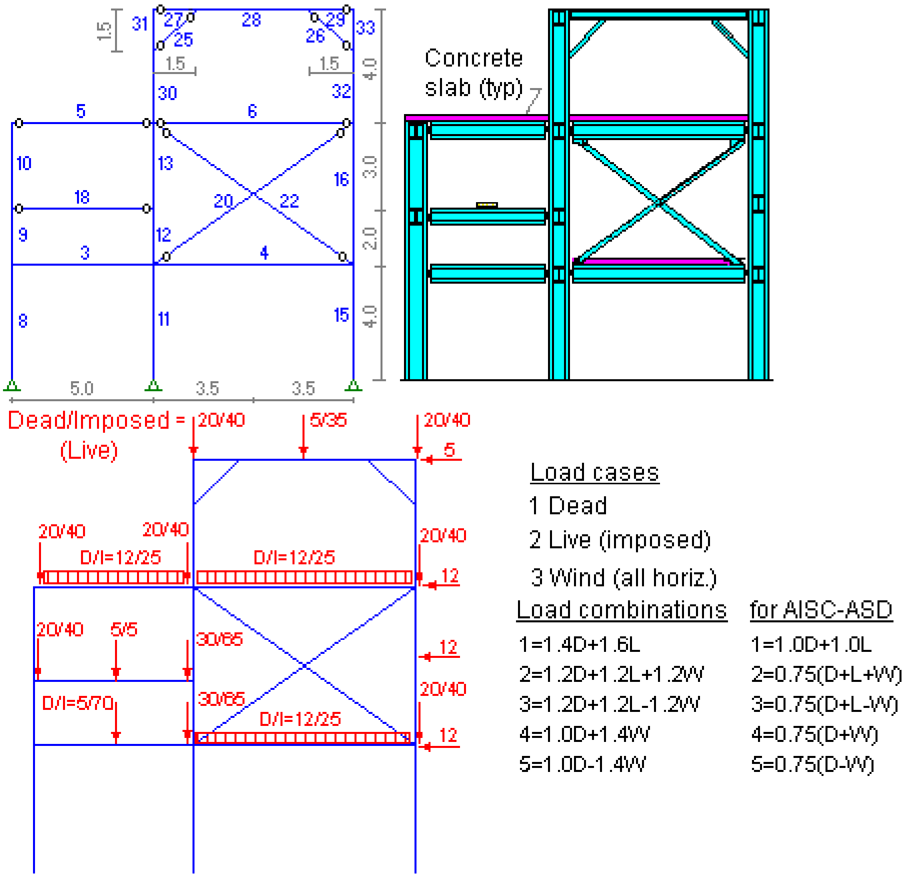
11.5 Example

Design the following plane frame.

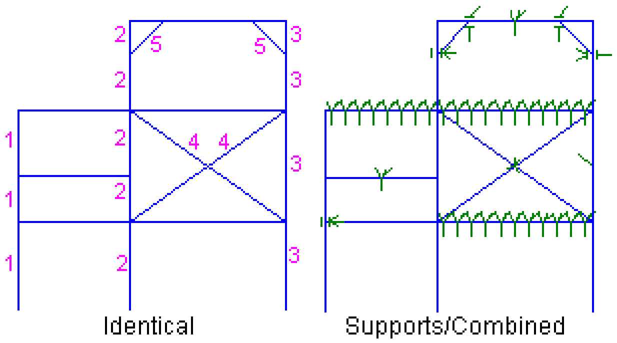
Note:

- This example illustrates the method of application of several Postprocessor 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:



After the geometry and loads were defined in *STRAP* and the model was solved, the following data was defined using the Steel Postprocessor 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.

12 Concrete design

The *STRAP* Concrete Postprocessor 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 6118-2001 (Brazil)
- AS3600 - 2009 (Australia)

Refer to Design assumptions 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 postprocessor. "***Beams***" are defined by the user.
COLUMN	-	a column consisting of a series of connected ***members*** defined in this postprocessor. "***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.

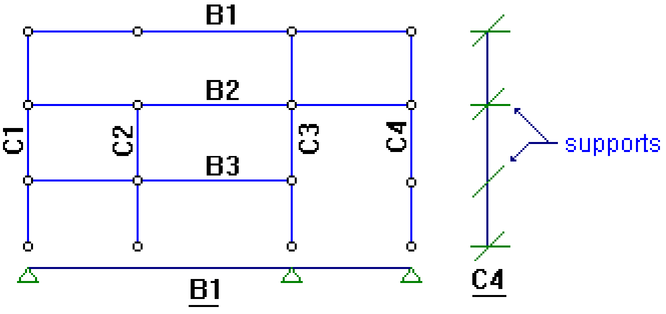
12.1 Create a concrete structure from a STRAP Model

The Concrete Postprocessor 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:

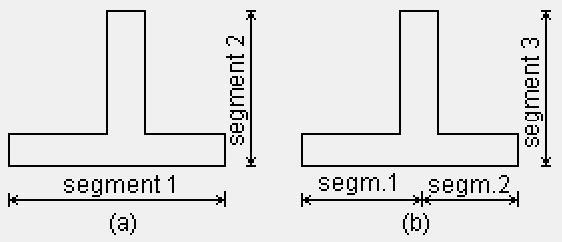
The following figure shows a simple plane frame. In order for the program to design the beams and columns beams B1-B3 and columns C1-C4 must be defined by the user (options for automatic definition of the beams/ columns by the program are available)



- 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):



To design each segment separately, e.g. Figure (b), edit the file STRAP.INI in the program folder and enter the following line in section
[ConcretePP]:
SeismicUnifyPart=TRUE

12.2 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 138 for more details.

Beams:

Moment:

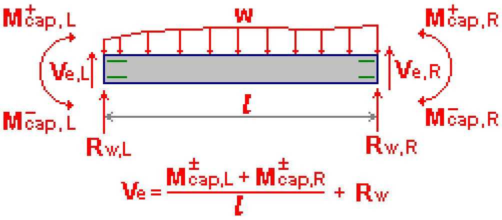
The beams are designed for all load combinations as defined by the user.

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:

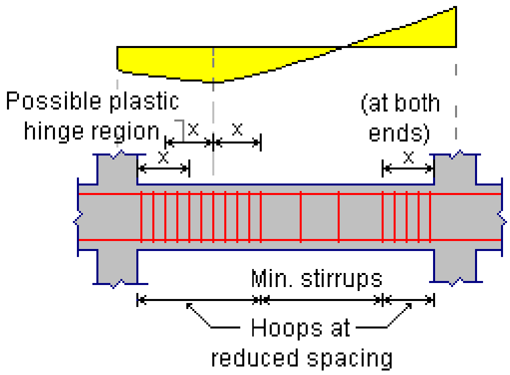
The seismic design shear forces (Ve) are calculated from the probable moment strength of the beam together with the factored beam loads:



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):

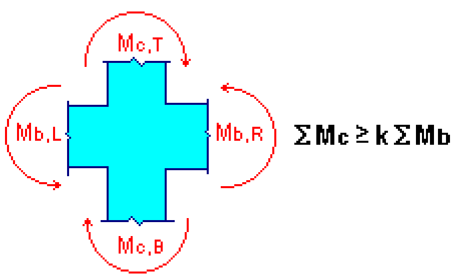


Columns:

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.

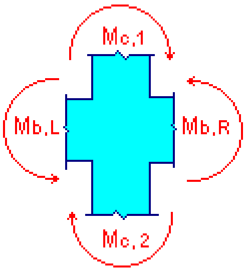
The program ensures that the columns comply with this requirement using the following procedure:

- 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:



$$Mc_{,1} = \Sigma Mb * \frac{\text{column stiffness,1}}{\Sigma\text{column stiffnesses}}$$

and **V_e = ($M_{c,T}$ + $M_{c,B}$)/L**

Stirrups (links) are calculated from **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 * $\phi ye/\phi y$

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

12.3 Design procedure

- For beams and columns designed for seismic loads, refer also to Design procedure - seismic 138
- for slab design and detailing, refer to Draw slabs - general 138

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.
- 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.

12.4 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** and select **Display reinforcement table**.
- Display the reinforcement areas and the corresponding moment and shear capacities: click **Data tables** and select **Display seismic capacity table**.
- Compute columns - select Compute
- To display data used to calculate links/stirrups, click **Data tables** and select **Display column shear table**.
- Check results, revise parameters, properties, etc, if necessary, and compute again.

12.5 Draw slabs

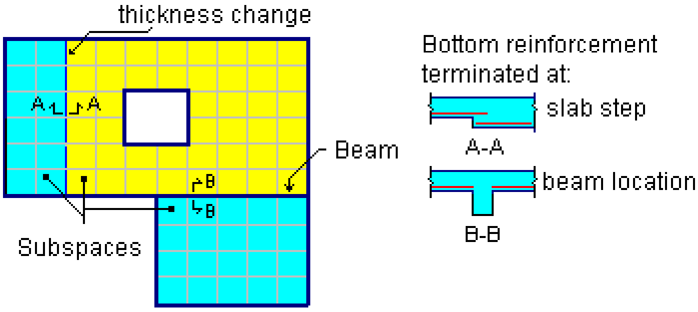
The program calculates and details the slab reinforcement, either as an arrangement of individual bars, a pattern of prefabricated meshes, or a combination of both.

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

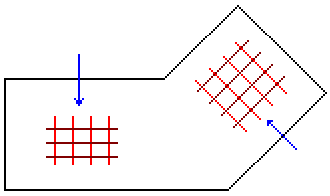
For example:

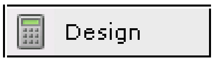


- select 

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 

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 

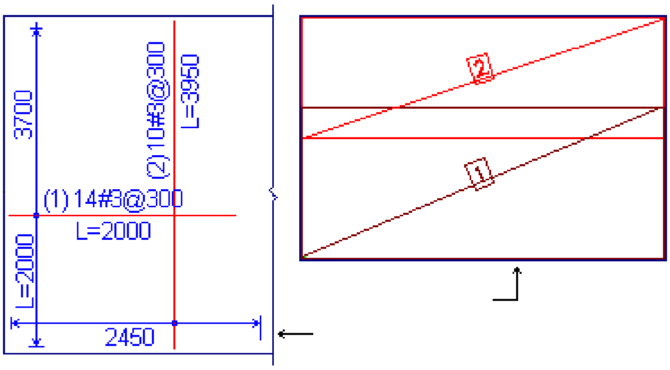
to create drawings and add objects to them
  - select a drawing and click 

  - add slab reinforcement drawing, bar schedules, mesh schedules or mesh details.
- check the reinforcement arrangement and revise parameters if necessary
- select 

to manually edit the drawing: add/delete bars, revise diameter/spacing/length, etc.
- select 

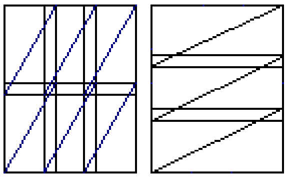
to check that the revised reinforcement provides sufficient area and anchorage.

Examples:



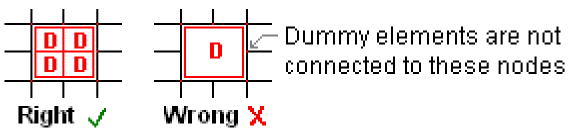
Reinforcement selection method:

- 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.

13 Bridge - general

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
  - lane loads
  - load cases
- 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 that contain max/min 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.

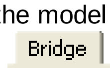
Solution Method and Program Assumptions

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.

13.1 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
- Select 

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.

- 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.

- 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.

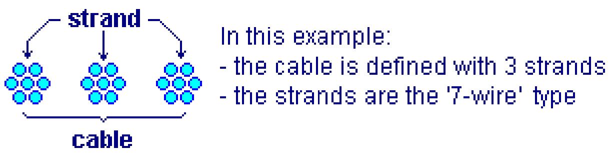
  Note that the transfer for large models with many lanes, strips and load cases may require several hours.

14 Presstres

POSTTEN is a *STRAP* postprocessor program that designs post-tensioned beams and slabs in solved *STRAP* models. The design may be carried out according to one of the following codes:

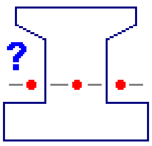
The program uses the following terminology:
- **strand** - the basic prestressing unit, a single wire or a group of wires
- **cable** - a group of strands

For example:



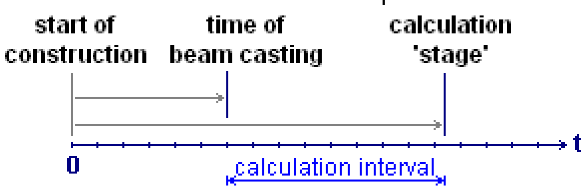
The *STRAP* model must either be a plane frame or a space frame; grids will not be accepted by the program (axial forces from prestressing cannot be added to load cases).

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 that 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:



14.1 How to use the program

POSTTEN is a *STRAP* postprocessor program that designs post-tensioned beams and slabs in solved *STRAP* models. 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 configuration must be defined in a different 'stage'. Refer to 'STRAP stages' 145 below.
- Define load cases with all loads, but without prestressing loads
- Solve the model
- Define load combinations in the results module
- Click the Postten tab.

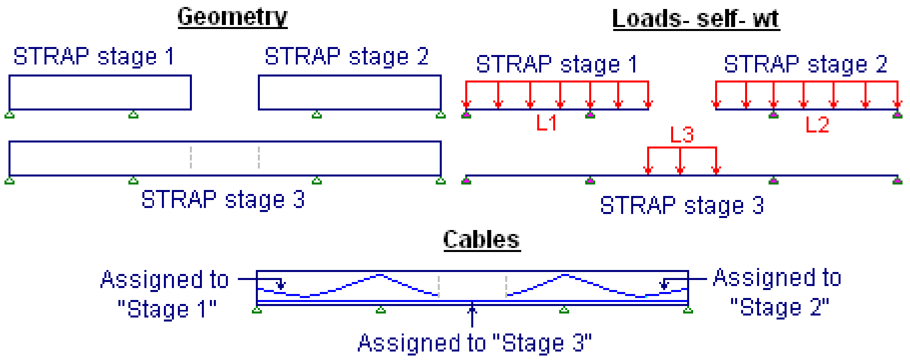
To design post-tensioned beams and slabs:

- Specify the default parameters, e.g. concrete type, prestress loss parameters, etc.
- Define a "Stage table" for models where not all of the cables or beams are prestressed at the same time. The stages are named and defined by the number of days from the start of construction.
- 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:
  - 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.
- Expand the combination table:
  - specify the "Permanent" loads in the combinations (for the calculation of deflections and time losses).
  - assign each load case to specified stage, if relevant. Note that a load case may either be applied or removed at a stage.
- For each beam:
  - specify the type and number of cables and the prestressing force. The program displays a "Magnel diagram" as a design aid for the selection of the prestressing. (refer to How to define cables 146)
  - 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.
- 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.* (Do not create new combinations containing these cable loads - *STRAP* does it automatically).
- Revise the cable details and trajectory, if necessary.
- Display/print tables of stresses, deflections, shear and ultimate moments for each beam.

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 *POSTTEN* module.

Example: segmented construction



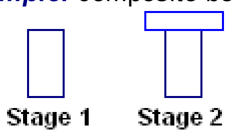
STRAP:

- 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

POSTTEN:

- Assign the relevant *'STRAP* stage' to each POSTTEN 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.

POSTTEN:

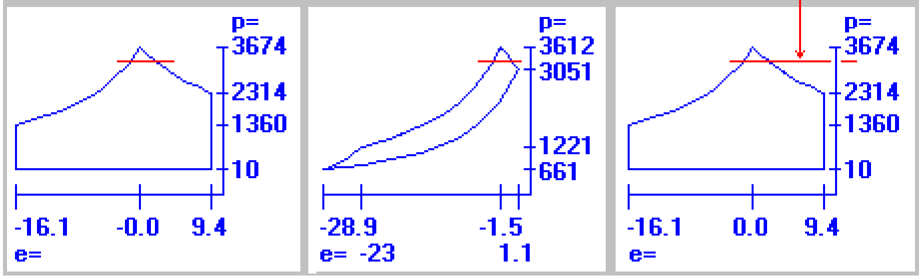
- **Stages - stages** option
  - Assign the relevant *'STRAP* stage' to each *POSTTEN* stage, specify the time when the stage starts and specify the allowable stresses.
  - Set "**Add creep forces..**" to **Yes** for all Composite stages.
- **Defaults - Composite** tab: select ☑ to calculate differential creep and shrinkage.

14.2 How to define cables

- select Define cable
- define cables (each may contain more than one strand so the total force lies within the Magnel diagrams. For example:

Cable No.	No. of strands	Strand type	Start coord. m	End coord. m	jacking percentage %	Total force kN
1	3	3-STRANDS	0.	10.	85.	1638.63
2	2	4-STRANDS	0.	10.	85.	1685.04
3						

Total force of all cables = 3323.67

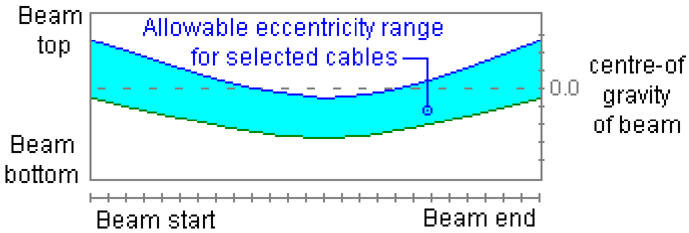


for "slab areas" the header is:

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 [kN]/[m]

where "Spacing of cables" is defined.

- select Cable geom
- double-click one of the cables in the table; the program superimposes the minimum/maximum eccentricity range *for this cable*. For example:



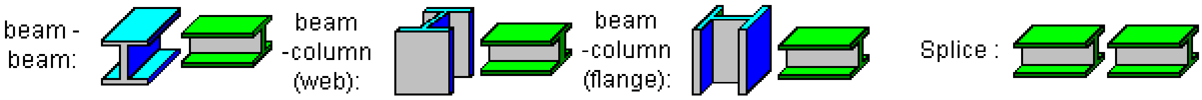
- the cables may be defined with a straight or parabolic trajectory; select one of the options and define the trajectory interactively on the screen; click OK when finished.
- select Cable geom again and choose another cable. The allowable eccentricity range displayed will again be ***only for the selected 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.

15 Steel connections

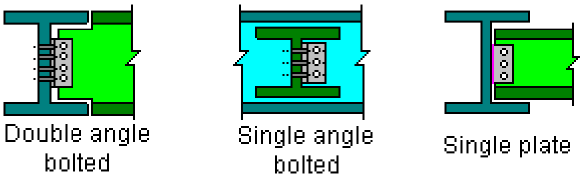
This module designs and details the following standard connections in structural steel models created and analysed with *STRAP*:

- 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.

15.1 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'***.

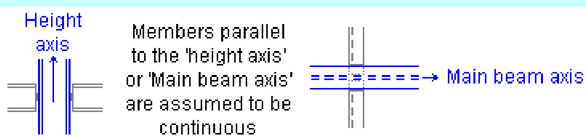
To start the connection design module:

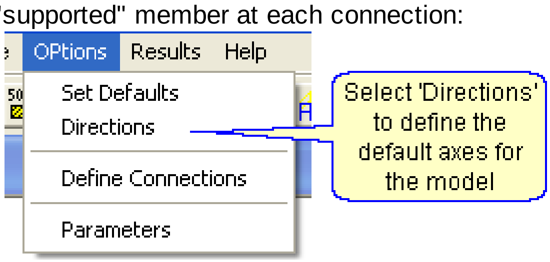
- Steel design module : select **File** and **Design connections**. - *or* -
- *STRAP* main menu : **Utilities** and **Connection design**.

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:



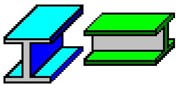
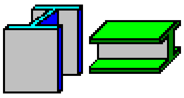
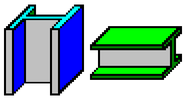
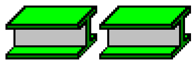


To define the "supporting" and "supported" members for individual connections, select  in the side menu.

Define the connection parameters:

- click  to define default parameters for ***all*** connections in the model:

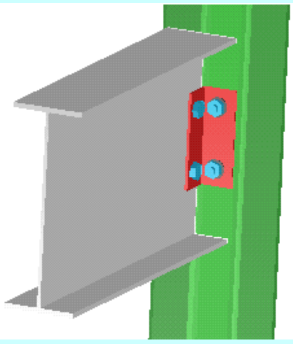
- in the Default connections tab, specify the default connection type for the three connections configurations:

beam - beam:  beam -column (web):  beam -column (flange):  Splice : 

- 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.

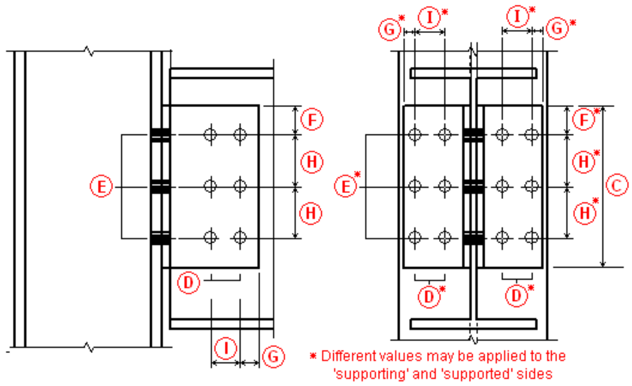
Design the connections and display the results:

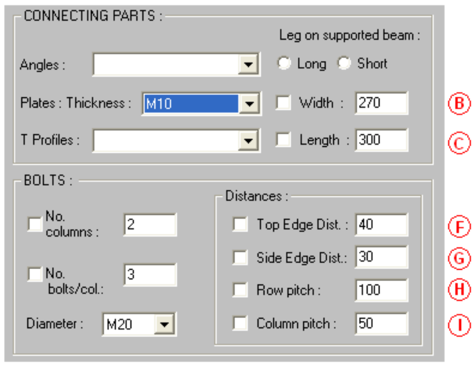


• click Display in the side menu.
• select the two connected members.
• the program displays the connection and the design calculations.
• click Render to display a rendered view of the connection:

Refine parameters or specify parameters details:

• 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.





16 Appendix

16.1 Program capacity

Supported operating Systems: Windows XP / Vista / 7 / 8 /10

Program Capacity	Regular Version (entire model)	Enhanced Version (entire model)	Professional Version (each submodel)
Maximum node number	1000	2500	1,000,000 *
Maximum element number	2500	1000000	1,000,000 *
Maximum property group number	32000		
Maximum number of loading cases	1000		
Maximum number of combinations -	- "solved" combinations : 2000		
	- calculated in results module : 2000		
Maximum number of load groups	100		
Maximum DOF bandwidth after optimization (each submodel)	no limit		
Maximum number of mode shapes	1000		

* the total number of submodels is unlimited, hence the program capacity is unlimited.