Atir Software Development LTD

STRAP - Submodel

Step by step

© Atir Engineering software LTD

13 Khilat sloniki, Tel Aviv, Israel

Tel: +972-3-6480129

Website: http://www.atir.com

Email: atir@atirsoft.com

# Table of contents

1. Abstract

This example demonstrates how to define a submodel and add it to the main model. Submodels are efficient when there is a portion of the structure that is repeated more than once in the model, e.g. the typical floor in a high-rise building. The advantages of the submodel are:

- each submodel may contain up to 32,000 nodes, i.e. the total model sizes - comprised of the "main model" and all the submodels - is virtually unlimited.

- When a submodel is revised each "instance" (occurrence) of the submodel in the structure is automatically revised.

- The solution time is much faster than that required for an identical structure without submodels.
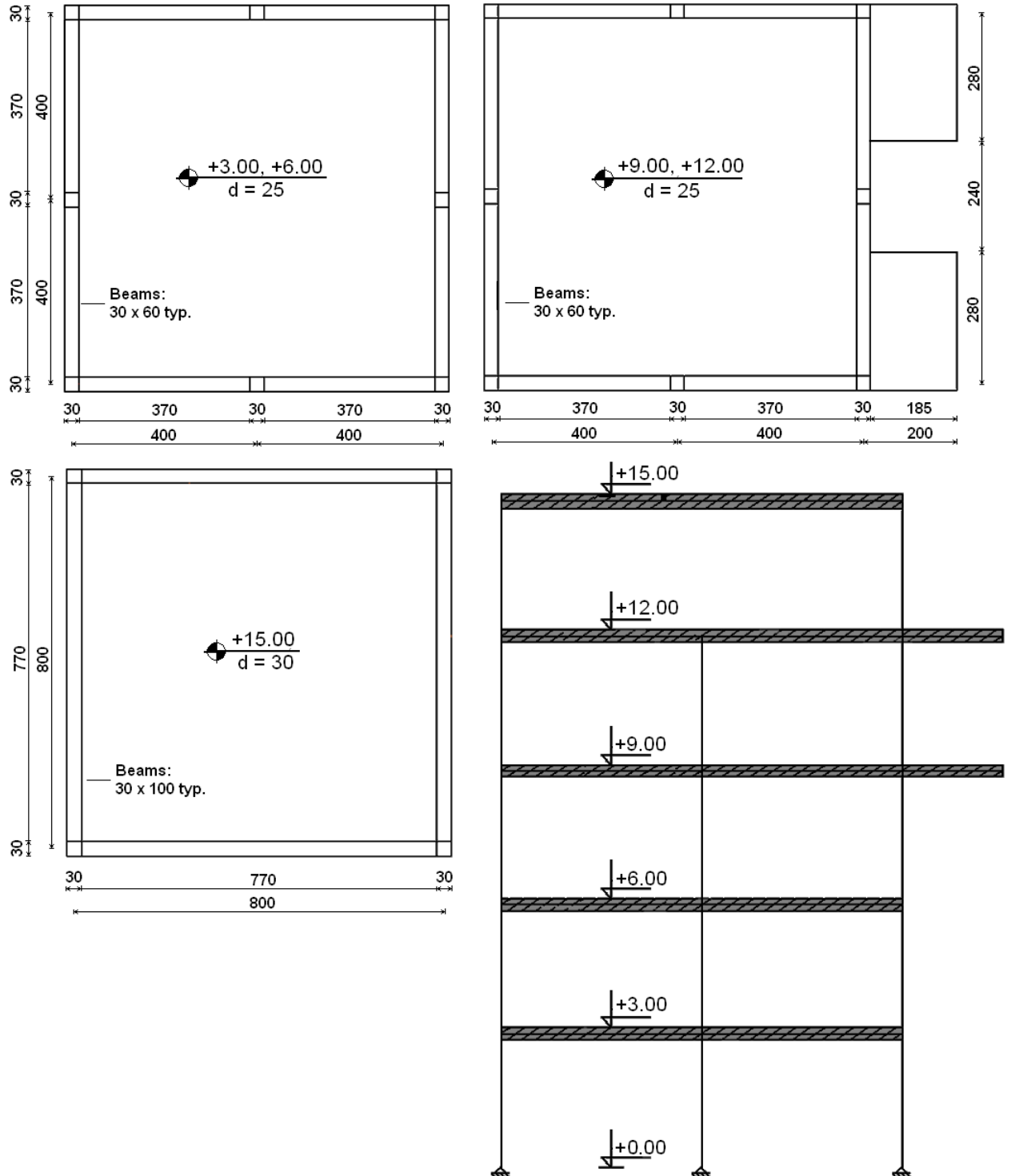
We will define the following structure:
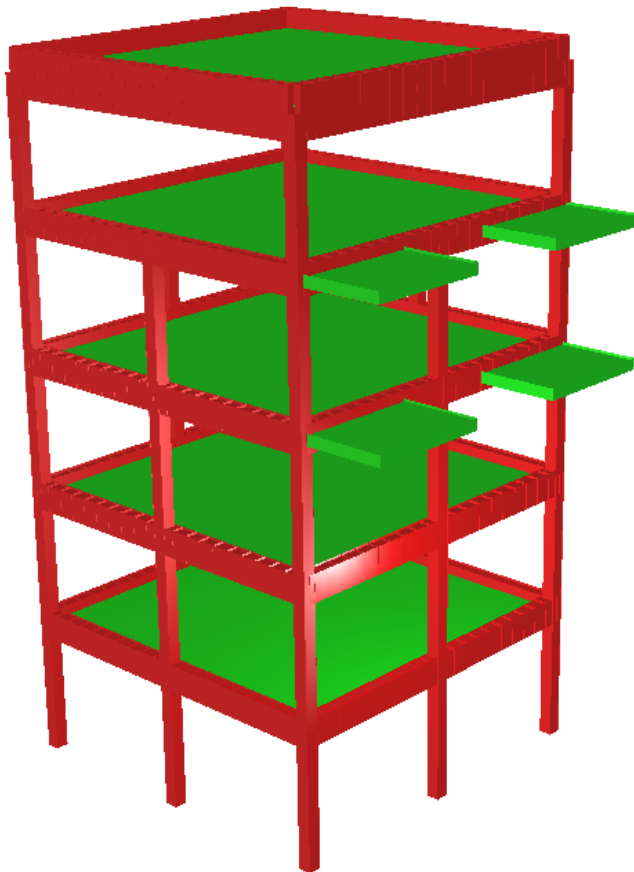
The structure consists of:

- the "Main model" - the columns.

- Three different submodels:

  o the two lower floors

  o the two upper floors (with the balconies)

  o the roof

Each of the first two submodels has two "instances"; the roof submodel has one instance.

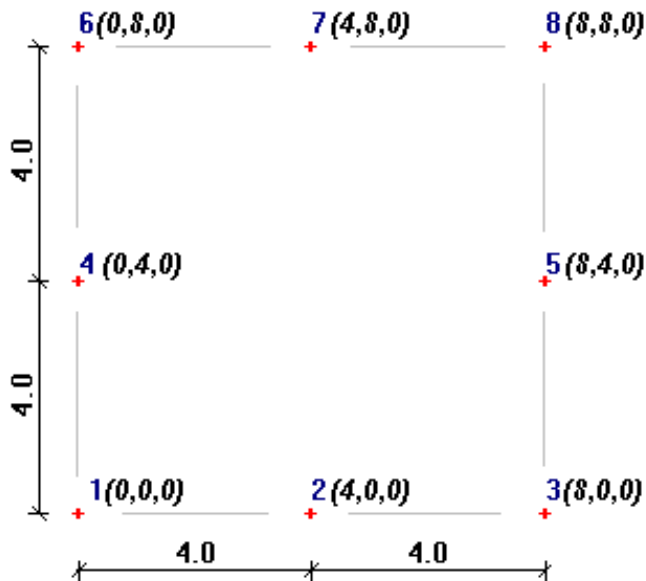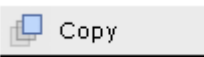Each submodel instance is joined to the Main model at "connection points".

+3.00, +6.00
d = 25

Beams:
30 x 60 typ.

+9.00, +12.00
d = 25

Beams:
30 x 60 typ.

+15.00
d = 30

Beams:
30 x 100 typ.

+15.00

+12.00

+9.00

+6.00

+3.00

+0.00

2. Main model

- click the 🔲 new model icon.

- select ⊙ Space Frame and click 🔲 **user defined**
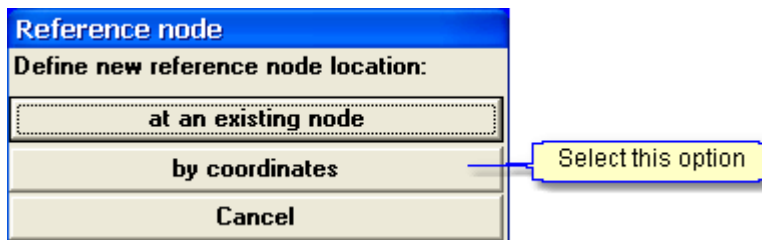
- click **Nodes** in the side menu and define the following nodes:
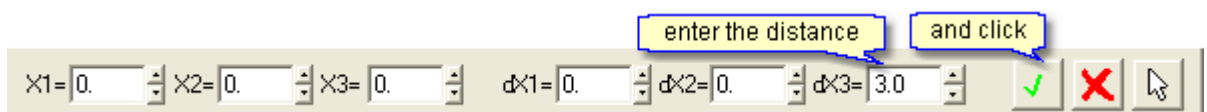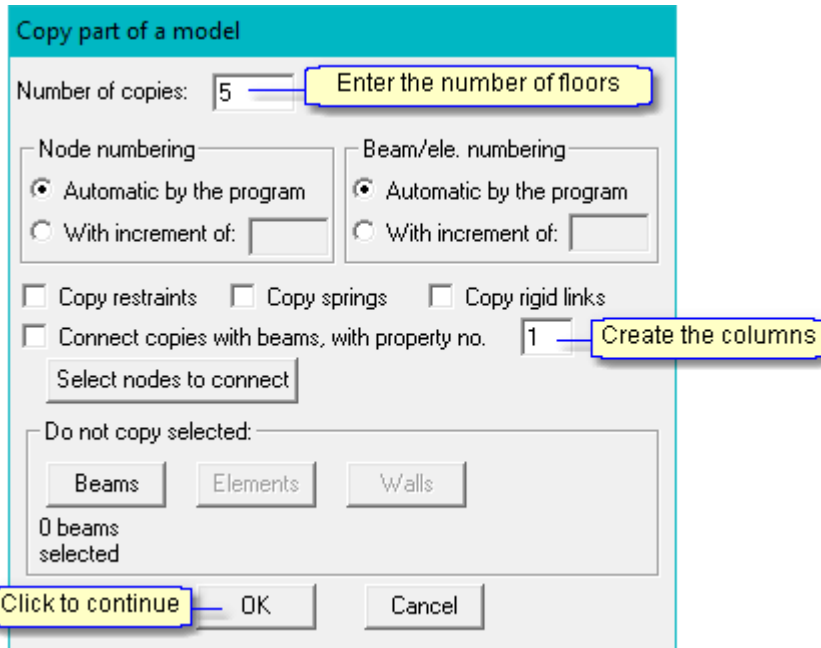

Copy the nodes to the other levels and create the columns:

- select **Copy** in the bottom side menu and then click on 🔲 Copy

- select all of the nodes.

- select Node 1 as the reference node.

- specify the new location of the reference node by coordinates;

Reference node
Define new reference node location:

- at an existing node
- by coordinates   Select this option
- Cancel

and enter the new coordinate value:

enter the distance   and click

X1=0.   X2=0.   X3=0.   dX1=0.   dX2=0.   dX3=3.0   ✓ ✗ ⬚

Copy part of a model

Number of copies: `5` — Enter the number of floors

Node numbering
- ● Automatic by the program
- ○ With increment of: [ ]

Beam/ele. numbering
- ● Automatic by the program
- ○ With increment of: [ ]

☐ Copy restraints ☐ Copy springs ☐ Copy rigid links

☐ Connect copies with beams, with property no. `1` — Create the columns

Select nodes to connect

Do not copy selected:

| Beams | Elements | Walls |

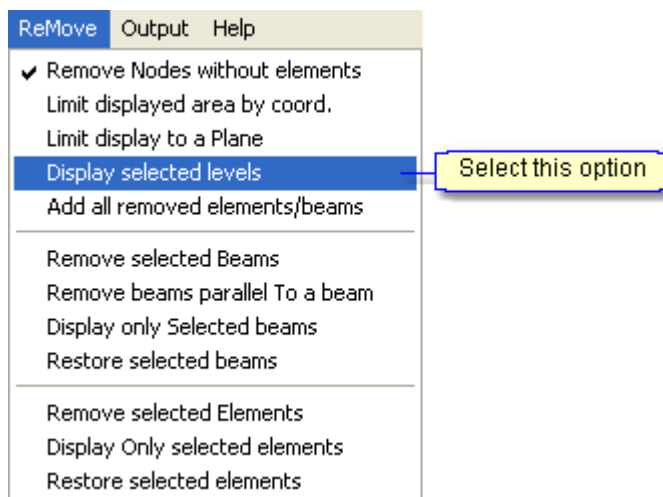0 beams selected

Click to continue — OK   Cancel

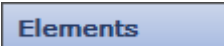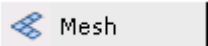- click 🔲 in the icon bar to display an isometric view of the model.

Erase the four intermediate columns at the top floor:

- click **Beams** and then select ✕ Delete in the side menu.

- select the four columns.

Define the floor slab at level +3.00:

- display only the level at +3.00:

ReMove   Output   Help

- ✔ Remove Nodes without elements
- Limit displayed area by coord.
- Limit display to a Plane
- **Display selected levels** — Select this option
- Add all removed elements/beams

- Remove selected Beams
- Remove beams parallel To a beam
- Display only Selected beams
- Restore selected beams

- Remove selected Elements
- Display Only selected elements
- Restore selected elements

- select "+3.00" in the list and click "OK".

- click 🔲 in the icon bar to cancel the isometric view.

- click **Elements** and then select ≪ Mesh in the side menu.

- click "OK" in the following menu.

- select the four corner nodes to define the floor contour and close by selecting the first node again.

- Select "End contour definition".

- click "OK" in the following menu to accept the default mesh parameters (check that Property 3 is specified); the program creates and displays the floor slab.

Define the perimeter beams:

- click **Beams** and then select ✎ Define in the side menu.

-
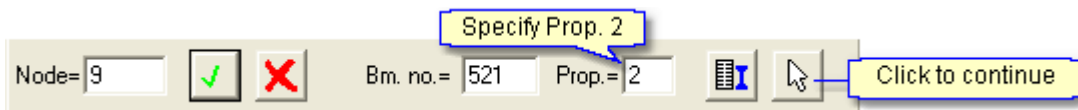
- Define the beams along the perimeter.

- Display the entire model: Again select Remove and Display selected levels, but this time click on +3.00 in the list to remove the highlighting.

- click ▣ in the icon bar to display the isometric view of the model:
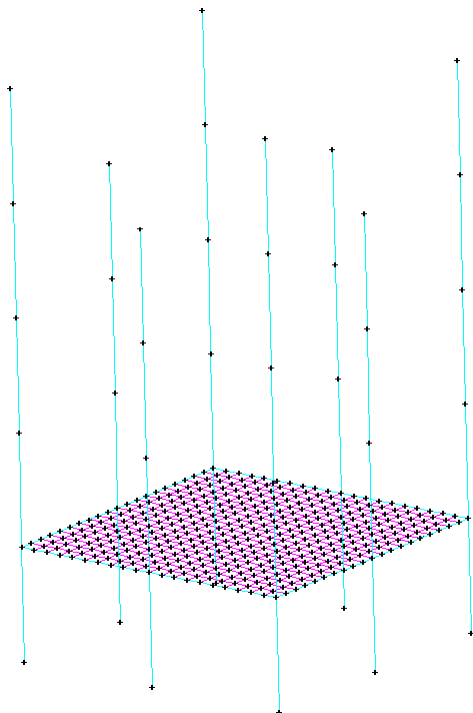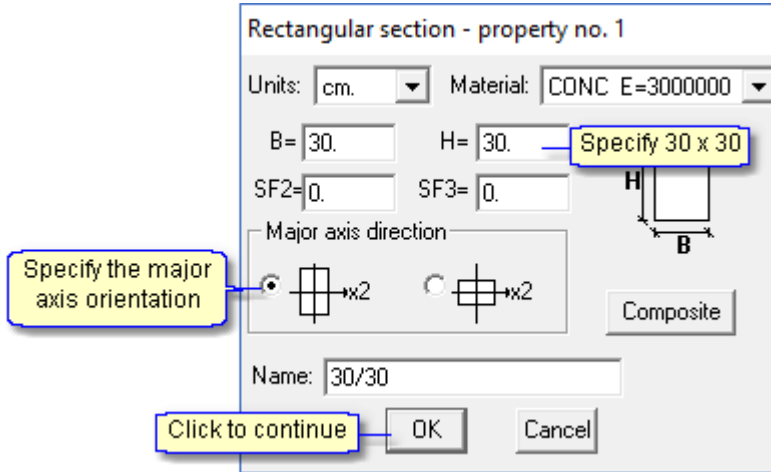

Define the beam and column sections:

- click **Beams** and then select 🐾 Properties in the side menu.

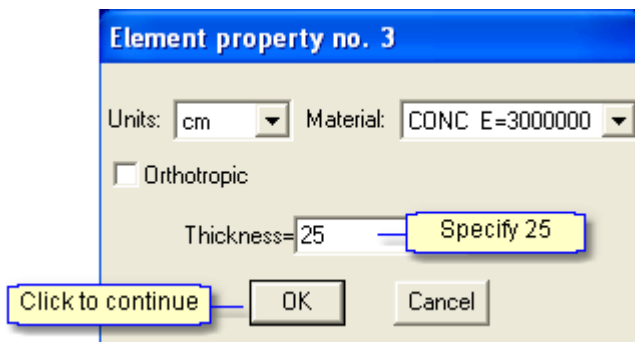- click and highlight Property 1 in the table (the columns); click "Define/revise", select [dimen.]

  and [▢] and [▢] and define the section dimensions:

- repeat for Property 2 (the beams) - 30 x 60 cm.

- click [Elements] and then select [Properties] in the side menu.

- click and highlight Property 3 in the table (the floor slab); click "Define/revise":

Define the supports:

- click [Restraints] and then select [△ Pinned] in the side menu.
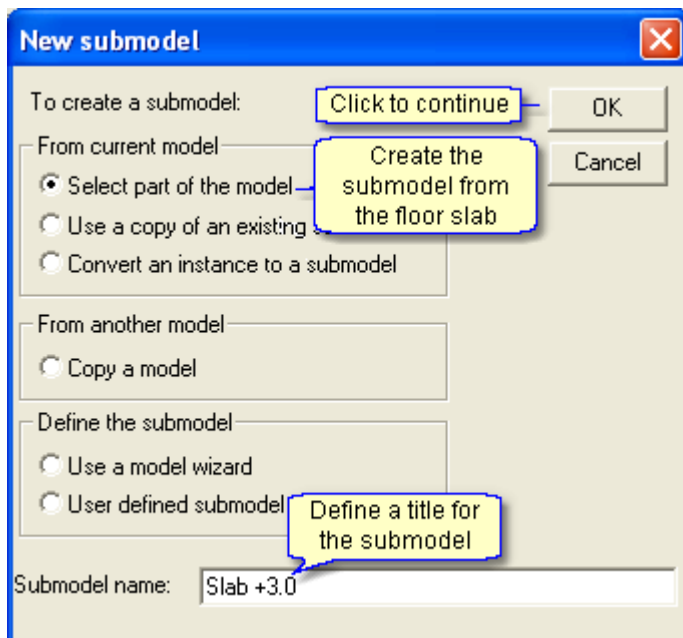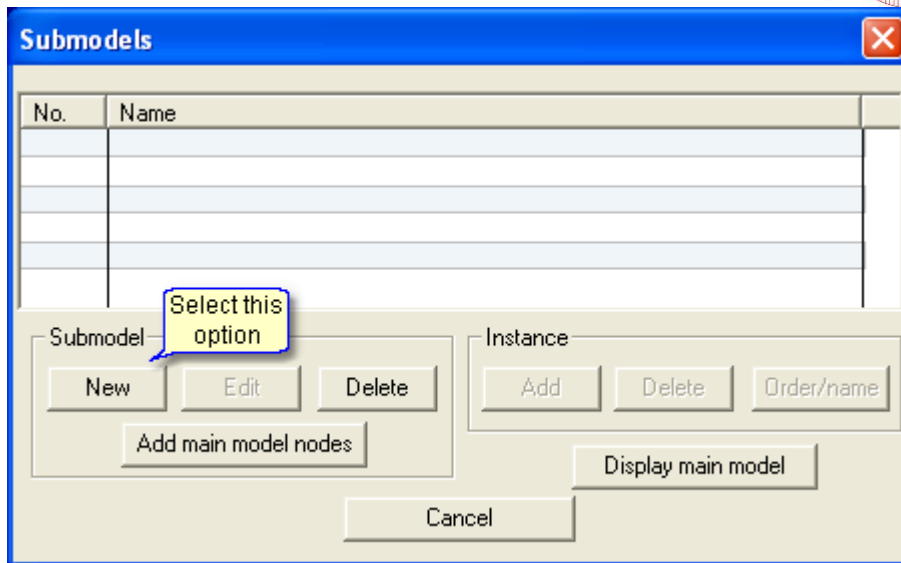
- select all nodes at level +0.00.

3. Submodels

3.1 Definition

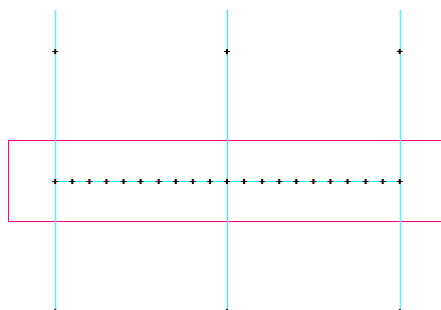Create the first submodel from the floor slab at +3.00:

- click the [X1 X3] icon to display the X1-X3 plane on the screen.

- click [Submodels] in the side menu:

Submodels

No.	Name

Submodel

New   Edit   **Delete**

Select this option

Add main model nodes

Instance

Add   Delete   Order/name

Display main model

Cancel

New submodel

To create a submodel: *Click to continue* — **OK**

From current model
- Select part of the model — *Create the submodel from the floor slab*
- Use a copy of an existing
- Convert an instance to a submodel

Cancel

From another model
- Copy a model

Define the submodel
- Use a model wizard
- User defined submodel   *Define a title for the submodel*
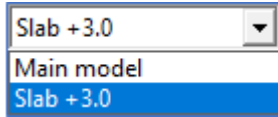
Submodel name: Slab +3.0

- Select the all the nodes at the floor level (use the Select by window option):
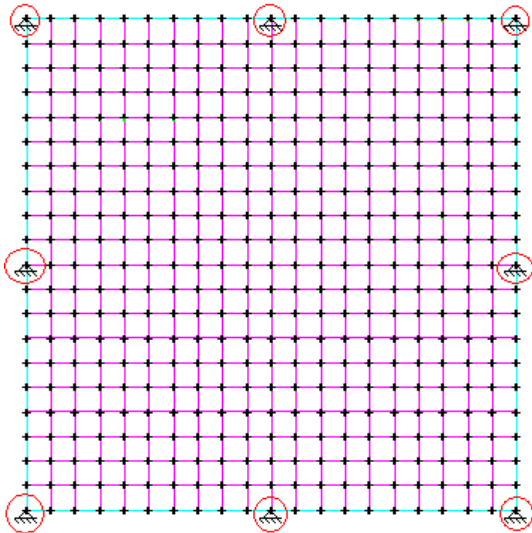
The program now displays a list of all submodels and instances in the side menu:

This list is displayed in all program modules (geometry, loads, results, etc) and is used to transfer from one submodel to another.

- select the submodel from the list; the program displays it on the screen:


All nodes common to to submodel and the Main model are automatically defined as "Connection points"

The connection are by default restrained in all 6 d.o.fs but the user may modify these.

Create the second instance of the submodel:

- select "Main model" in the submodel list.

- click in the icon bar to restore the isometric view of the model.

- click **Submodels** in the side menu:

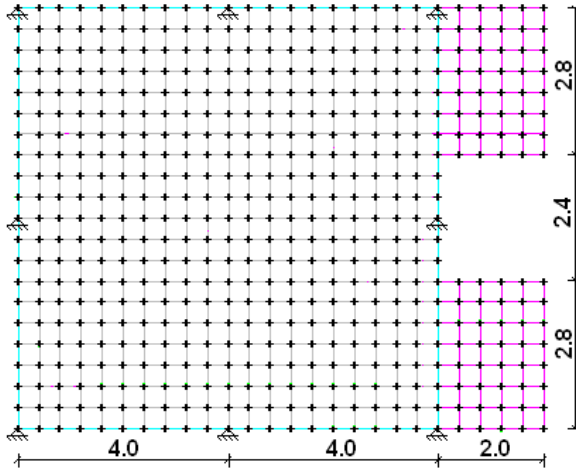Sect "Add" (an 'instance', not a submodel)
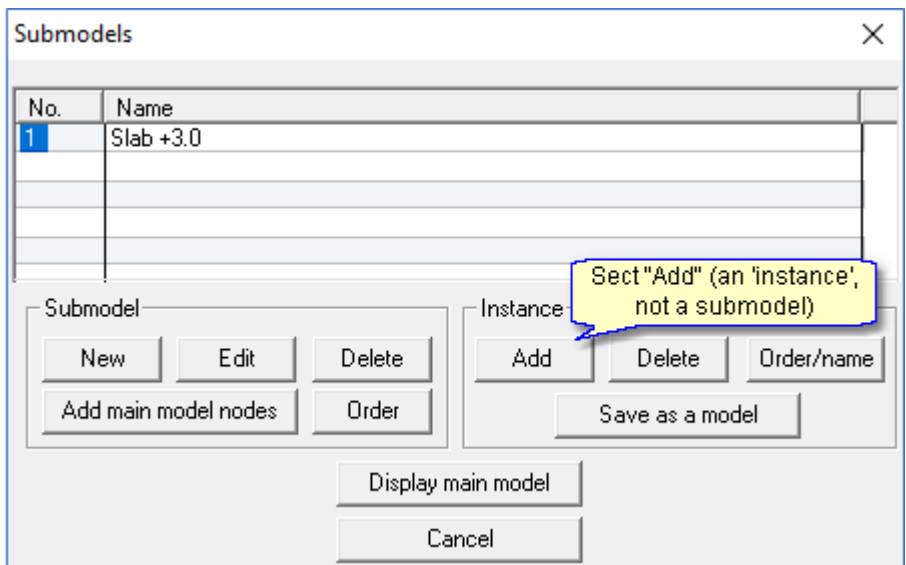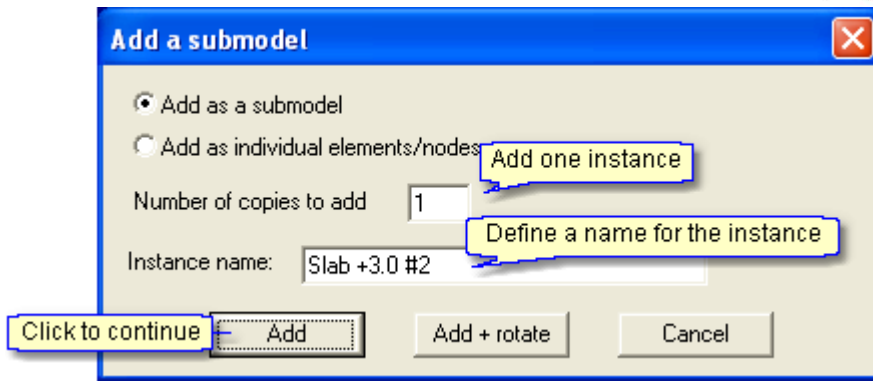
- Select the lower-left corner of the submodel as the "reference point"
- Select the location of the reference point in the Main model:
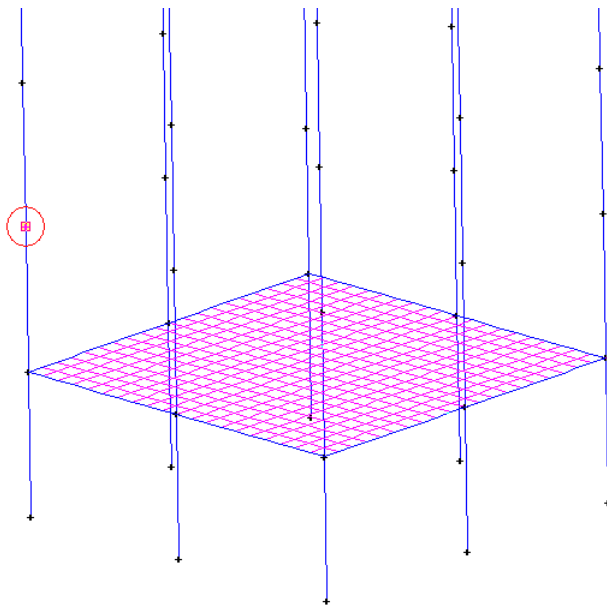
Create the submodel at +9.00:

The second submodel is identical to the first one with the addition of the balconies. We will create it by copying and modifying the first submodel.

- click **Submodels** in the side menu:
- click "New" (submodel) in the following menu and -
  - ○ select ⊙ Use a copy of an existing submodel and define the submodel name as "Slab +9.00".
  - ○ select Slab +3.00 from the list of existing submodels.
  - ○ the program displays the submodel; add the balcony elements as shown:
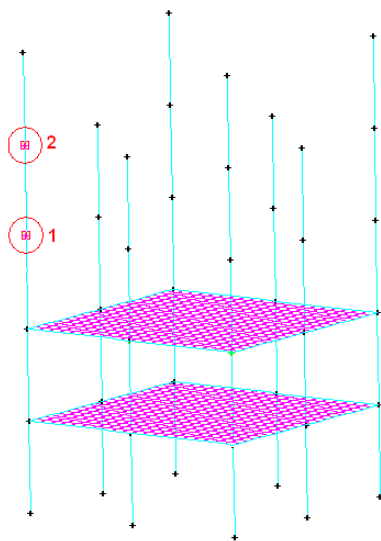
Add both instances of the second submodel:

- click **Submodels** in the side menu:

- Select Slab +9.00 in the submodel list and click "Add" (instance).
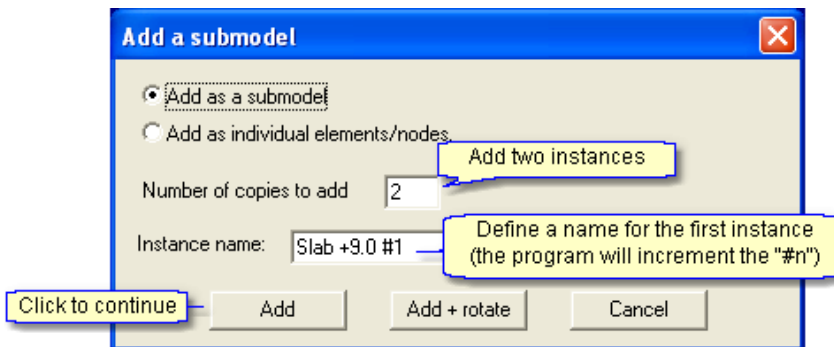
- select the lower left corner of the submodel as the "reference point"

- the program then displays the Main model; select the two reference points defining the location of the two instances of the submodel:

Define the third submodel at +15.00:

The roof slab is identical to the first submodel but with different slab and beam dimensions. There are also different connection points because not all of the columns go up to the roof.

- click **Submodels** in the side menu:

- click "New" (submodel) in the following menu and -

- select ⊙ Use a copy of an existing submodel and define the submodel name as "Slab +15.00".

- select Slab +3.00 from the list of existing submodels.

- the program creates and displays the submodel.
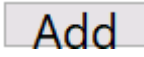
Revise the geometry as follows:

- click **Beams** and then select **Properties** in the side menu.

- click and highlight Property 4 in the table; click"Define/revise", select **dimen.** and **▢** and define the section dimensions: 30 x 100

- assign this property to all beams in the submodel.

- click **Elements** and then select **Properties** in the side menu.

- click and highlight Property 5 in the table 5; click "Define/revise" and define the slab thickness: 30 cm.

- assign this property to all elements in the submodel.

Remove the unnecessary connection points (note that connection points without an exact corresponding Main model node are automatically connected by Rigid links to the closest Main model node).

- select **Connections** in the side menu and click on **✗ Delete**

- select the four intermediate connection points.

The program assumes by default rigid connections between the submodel and the main model. Assume that the roof beams are pinned to the columns, i.e. release the rotational degrees-of-freedom at the connection points:

- select **Connections** in the side menu and click on **△ Pinned**

- select the four corner connection points.

- Add the third submodel to the structure:

- click **Submodels** in the side menu:

- Select 3 Slab +15.00 in the submodel list and click **Add** (instance).

- Specify 1 copy and define the instance name.

- select the lower left corner of the submodel as the "reference point"

- the program then displays the Main model; select the top of the lower-left column as the location of the reference point in the Main model.

3.2   Connection points - additional options

There is an option to automatically define the connection points for a submodel to the main model. The program searches for Main model nodes that are within a "tolerance" distance of the selected submodel nodes.

Note:

this option does not revise existing connection points
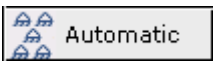
connection points without an exact corresponding Main model node are automatically connected by Rigid links to the closest Main model node.

For example, use this option to redefine the connection points for the submodel Slab +3.00:

- select the submodel in the small window:

Submodel
Slab +3.0
Main model
Slab +3.0
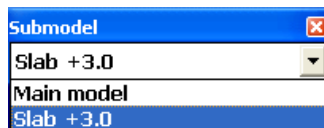
Delete the existing connection points because the program will not revise them.

- select **Connections** in the side menu and click on **✕ Delete**

- select the four connection points.

- click on **Automatic**

Automatic connection points de...

Height direction: X3

Tolerance
In height direction: 0.01
In submodel plane: 1.

set the tolerance values - in the plane and perpendicular to it

Connection point type
- Fixed
- ● Pinned, fixed around height
- Pinned

Select the connection type

☑ Change elements to coincide with main model nodes (when necessary)
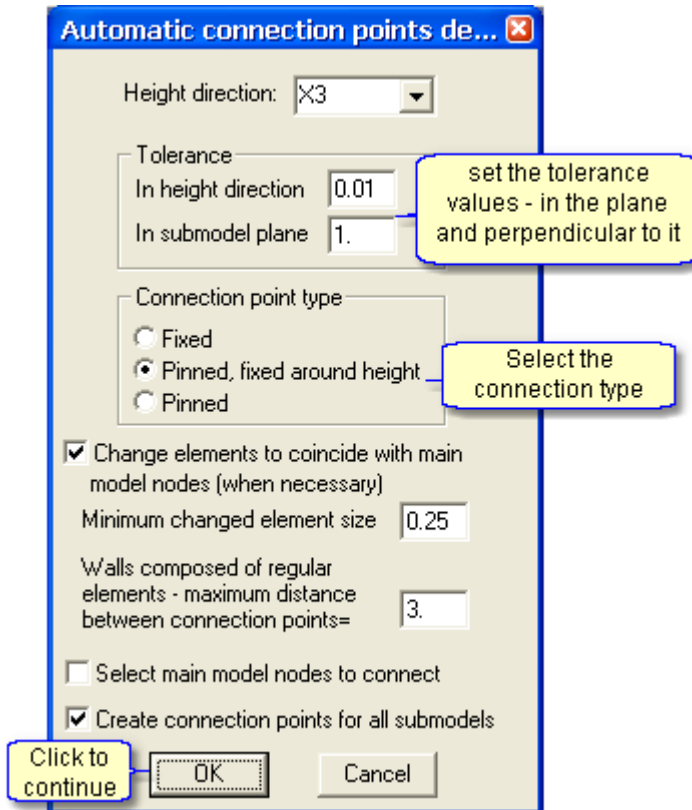Minimum changed element size: 0.25

Walls composed of regular elements - maximum distance between connection points= 3.

☐ Select main model nodes to connect
☑ Create connection points for all submodels

Click to continue

OK   Cancel

return to the Main model.

3.3   Submodels - tables

Display a table with the submodel data:

Output   Help

Display Node coordinates
Display node Restraints
Display Properties
Display Beams

Display solid elements
Display walls list
Display wall sections
Non linear
Display submodels connections
Display Weights

Output format

SHow data

Print data tAbles
Print Drawing
Create a DXF file

Print/Edit saved Drawing

Select this option
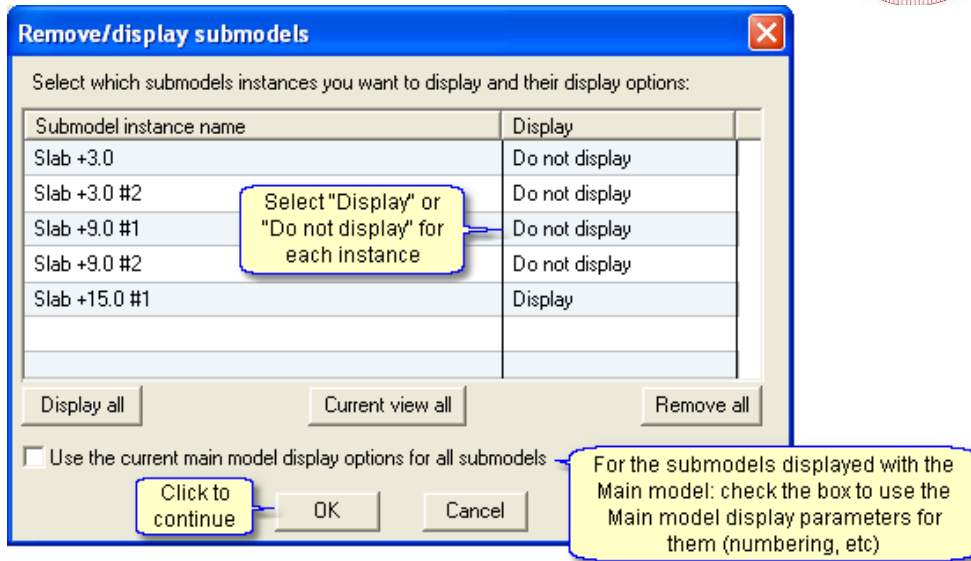
3.4 Submodels - display options

Selected submodels may be displayed/removed from the main model display. This option is useful when creating the geometry or displaying the results.

Remove/display submodels

Select which submodels instances you want to display and their display options:

Submodel instance name	Display
Slab +3.0	Do not display
Slab +3.0 #2	Do not display
Slab +9.0 #1	Do not display
Slab +9.0 #2	Do not display
Slab +15.0 #1	Display

Select "Display" or "Do not display" for each instance

Display all   Current view all   Remove all

☐ Use the current main model display options for all submodels

For the submodels displayed with the Main model: check the box to use the Main model display parameters for them (numbering, etc)
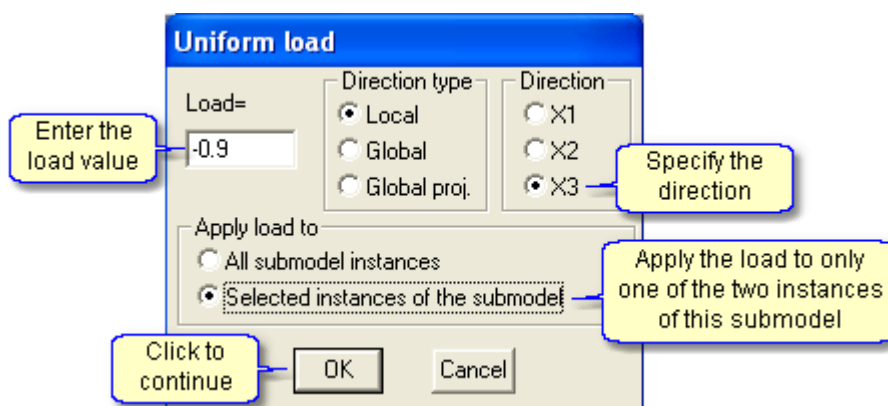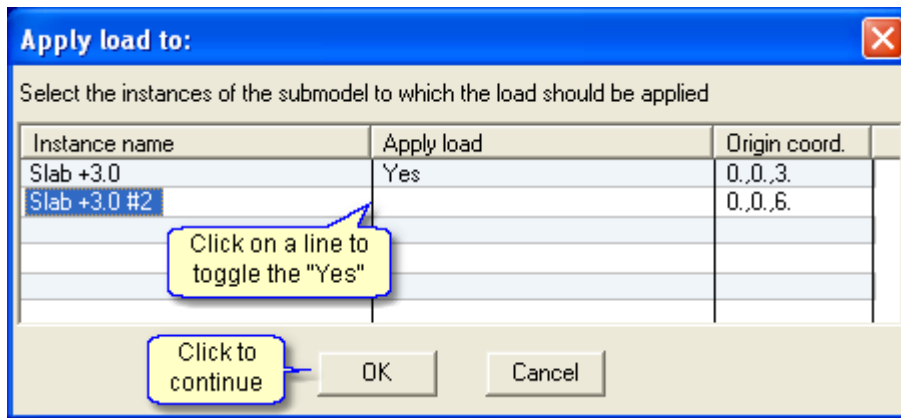
Click to continue   OK   Cancel

4. Loads

Loads applied to a submodel may be applied to all instances or different loads may be defined for each instance of the submodel:

- click the ⎯ Loads ⎯ tab.

- select 🔽 New load in the side menu and enter a name for the load case.

- select the Slab +3.00 in the small window.

- select **Element loads** in the side menu.

- select 🔽 Define and 🔺 unif. and define the loads:

Uniform load

Load=

Enter the load value

-0.9

Direction type
- ⦿ Local
- ○ Global
- ○ Global proj.

Direction
- ○ X1
- ○ X2
- ⦿ X3

Specify the direction

Apply load to
- ○ All submodel instances
- ⦿ Selected instances of the submodel

Apply the load to only one of the two instances of this submodel

Click to continue   OK   Cancel

- select the submodel instance:

- apply the load to selected elements.

Continue to define loads, solve the model and display the results.