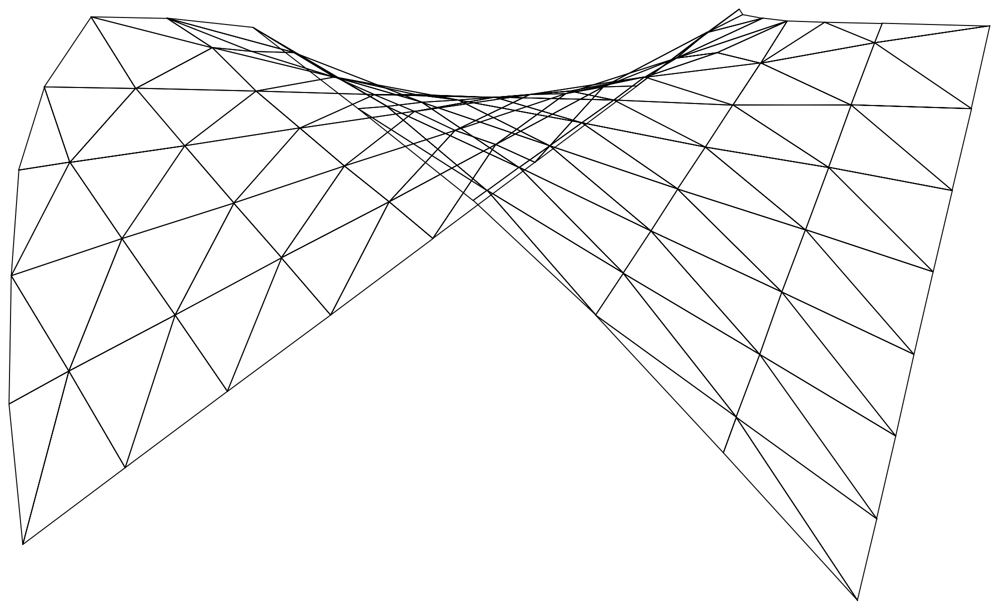
* * * A T I R * * *



S T R A P

for

Windows

STRUCTURAL ANALYSIS
PROGRAM

USER'S MANUAL

Examples

Version 11.5

December 2004

** This page is deliberately blank **

Disclaimer

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

The user must verify his own results

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

General

This manual explains the method for defining the geometry and loading of several typical structure models. The examples are intended as an aid to learning the proper use of the *STRAP* program options, **and are not intended as a guide to proper engineering judgment in the construction of a model for analysis.**

Note that only the main input options are listed in the examples.

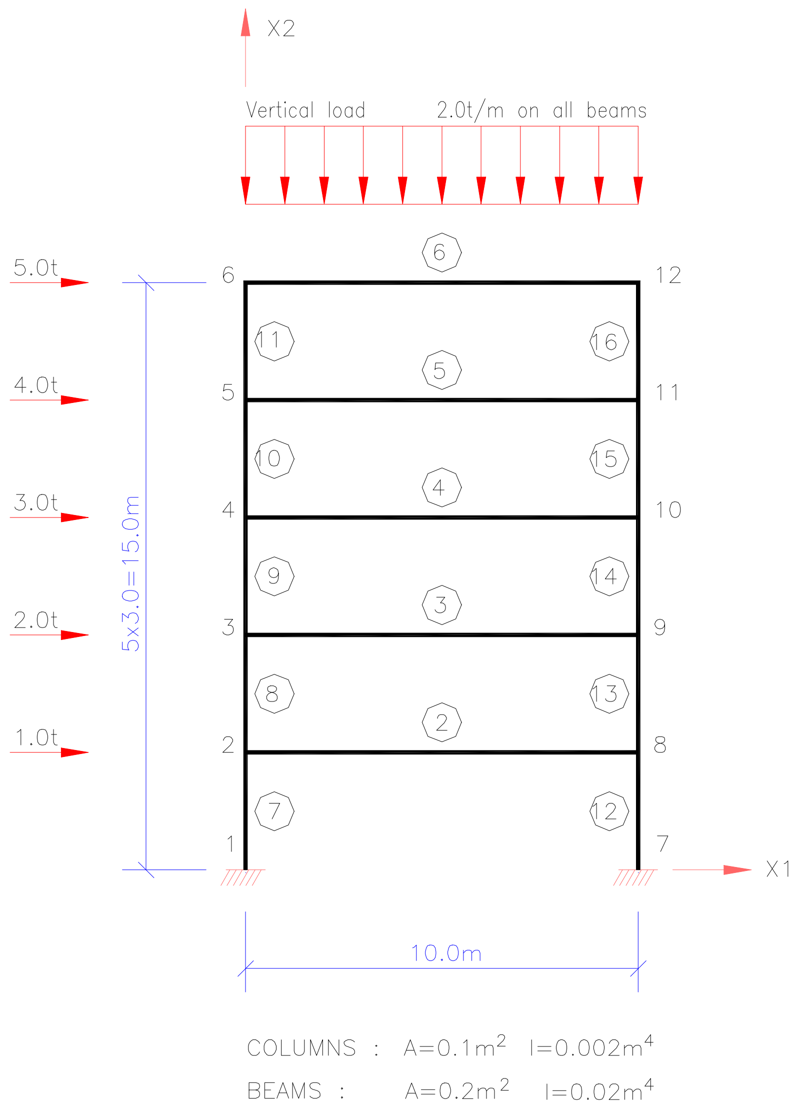
** This page is deliberately blank **

Table of Contents

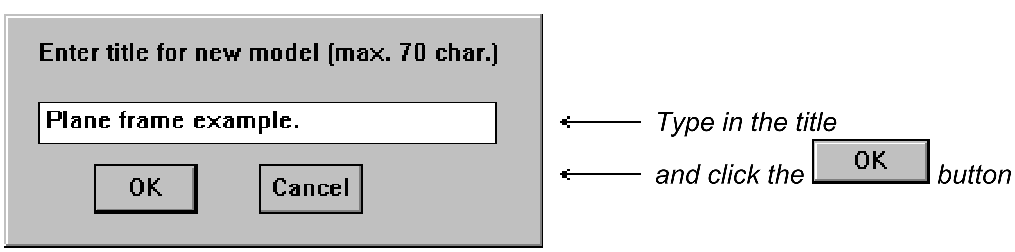
** This page is deliberately blank **

1 Plane Frame

Define vertical and horizontal loads in separate loading cases and create a combination 1.4*Vertical + 1.6*Horizontal.

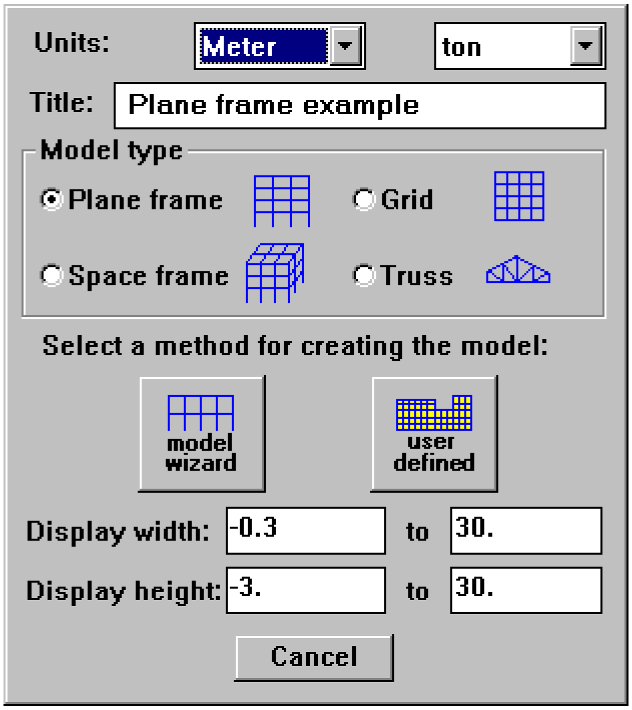


- **Main Menu**
  - select **Files** in the menu bar
  - select **New model** in the pull-down menu
  - enter the model title:



- **Preliminary Menu**:

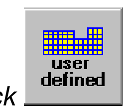
  Arrange the menu as follows:



← *Select the model units*

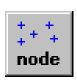
← *Enter the model title*

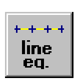
← *Set the model type to* **Plane frame**

← *Click*  *to proceed to geometry*

- **Nodes**:

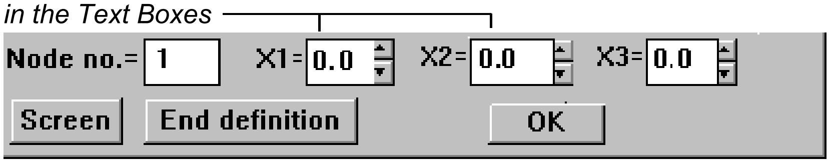
  Define the nodes with two **Line - equal** commands:

- click the  icon

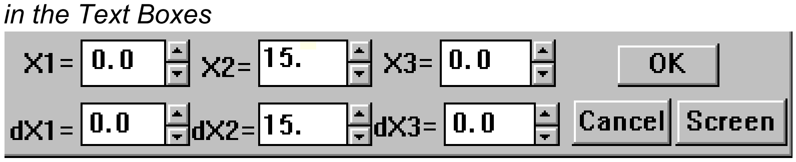
- click the  icon

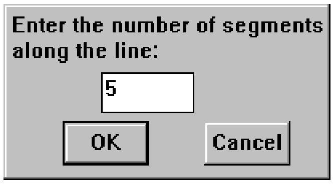
- define line 1 – 5:

  *Move the crosshair using the mouse to **X1=0, X2=0,** or type in the values directly in the Text Boxes*



  *Move the crosshair to **X1=0, X2=15.0** (or **dX2=15.0**), or type in the values directly in the Text Boxes*





Specify 5 segments

Click the OK *button*

Nodes 1 to 6 are created.

- Repeat for nodes 7 to 12.
- Click the  icon.

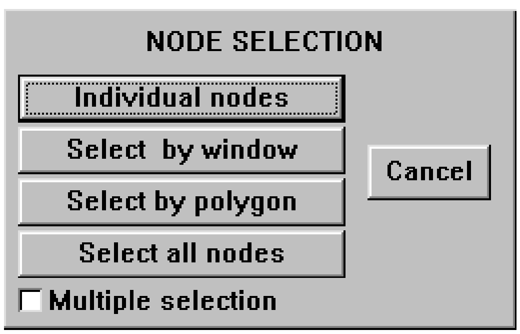
The nodes may also be defined with the **Grid** command, where the "base line" contains one segment only.

- **Restraints**

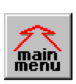
- Click the  icon.

- Click the  icon.

- Select the two bottom nodes:

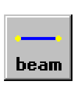


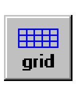
Click **Individual nodes**

- Move the crosshair adjacent to node 1 and click the mouse, then move the crosshair adjacent to node 7 and click the mouse: click the End selection button.
- Click the  icon.

- **Beams**:

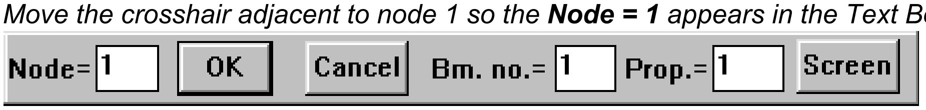
Define all beams with a grid command -

- Click the  icon

- Click the  icon

- Select Node 1 as the start node of the base line:

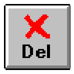
Move the crosshair adjacent to node 1 so the ***Node = 1*** *appears in the Text Box and click the mouse.*

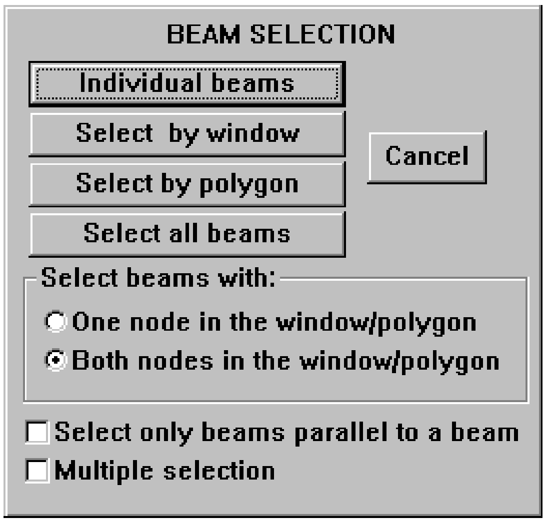


- Select Node 7 as the end of the base line.

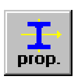
- Select Node 12 as the end of the height line.

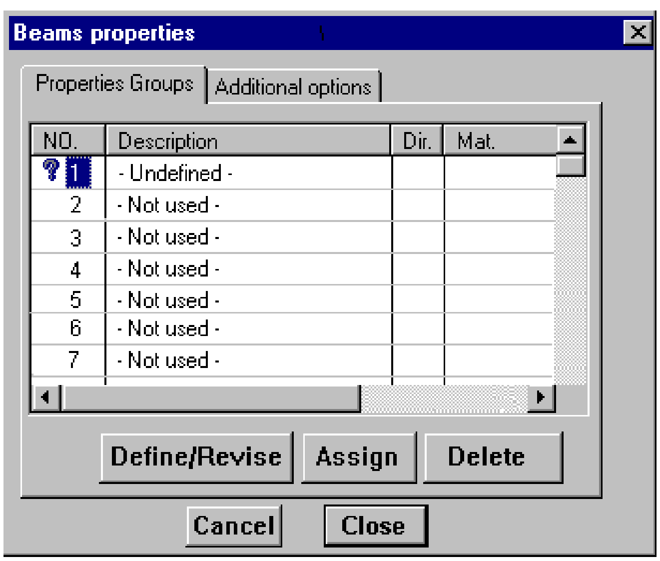
  The program draws a complete grid, including a beam between 1 and 7 ; to delete this beam

- click the  icon.



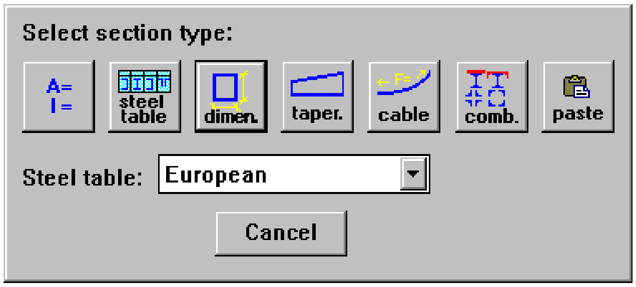
Click **Individual beams**

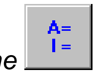
- Move the crosshair adjacent to the beam so that it is highlighted with the small rectangular blip.
- Click the mouse. To complete the selection click the mouse again or click the  button.
- Click the  icon.



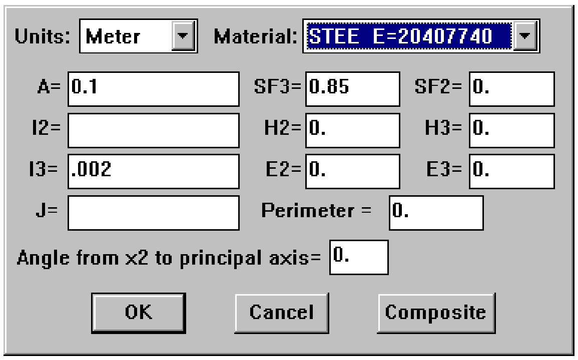
Click property no. 1

Click the **Define/Revise** *button*



Click the  *icon*

- Enter the constants:

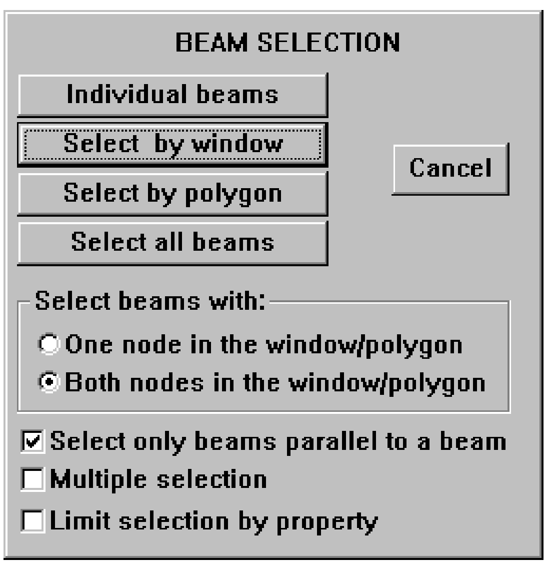


Set units to **Meter**
Select **Steel**
Define **A**
Define **I3**
Define ***SF3***
Click the  *button*

- Similarly define property group 2 with values **A=0.2** and **I3=0.02.**

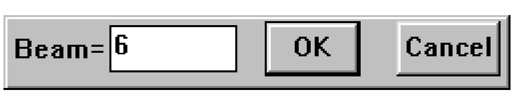
Property 1 was automatically assigned to all of the beam elements. To assign property 2 to the horizontal elements:

- Select Property 2 (click and highlight)
- Click the  button.



Click **Select all beams**
Set to 

- Select the “parallel” beam.:

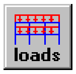


Move the crosshair adjacent to any horizontal beam

Property 2 is assigned to the selected beams.

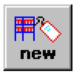
- Click the  button to complete the property definition.

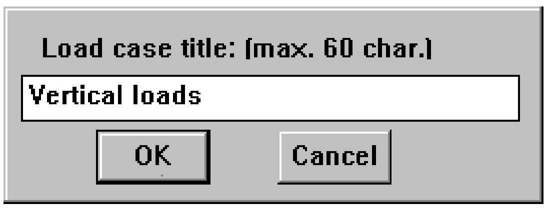
The geometry definition has been completed; click the  icon.

- Click the  icon.

- **Loads**:

  Define the vertical loads as load case 1:

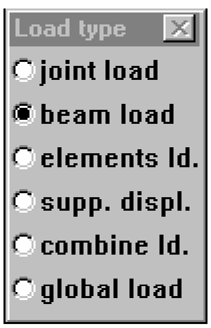
  - Click the  icon.
  - Define the load case title:



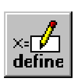
Type in the title

and click the  *button*

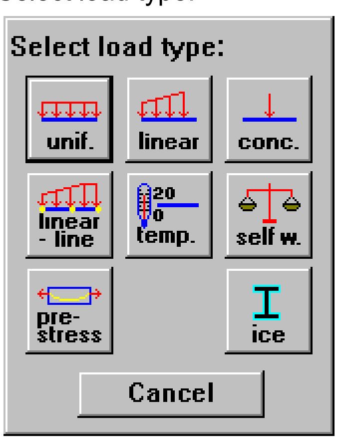
  - Select the load type:

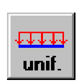


Set the **Load type** menu to **beam load**

and click the  icon.

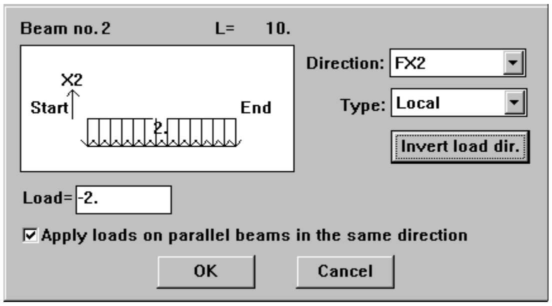
  - Select load type:



Click the  *icon*

  - Select all horizontal beams as explained in Properties.

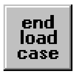
- Define the load:



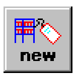
Set the **Direction** *to* **FX2**
Set **Type** *to* **Local** *or* **Global**

Type in the load value

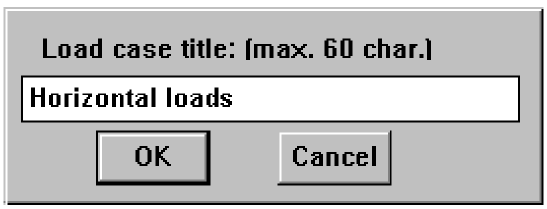
Click the  *button*

- Click the  icon.

Apply the horizontal loads in a separate loading case:

- Click the  icon.

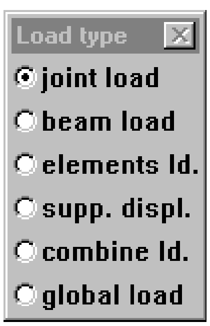
- define the load case title:



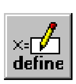
Type in the title

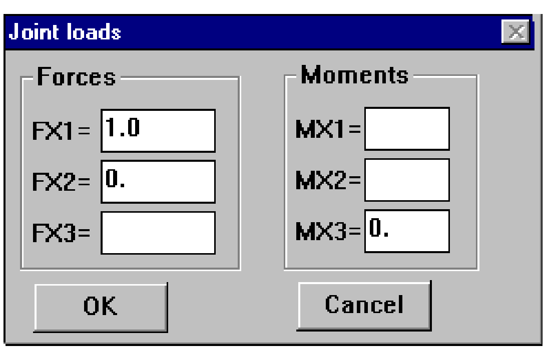
and click the  *button*

- Select the load type:



Set the **Load type** *menu to* **joint load**

- and click the  icon.



Type in the load value

and click the  *button*

- Select node 2 as explained in “Restraints”.

The load is displayed graphically on the node.

- Repeat the process for the loads on nodes 3,4,5 and 6.
- Click the 

icon.

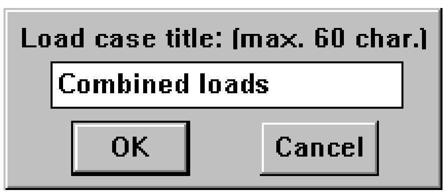
- **Combination**

The combination can be created in **Loads** as a separate load or can be defined after the solution.

<u>Combination defined as a load case</u>:

- Click the 

icon.
- Define the load case title:

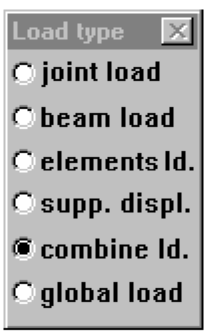


Type in the title

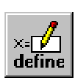
and click the 

button

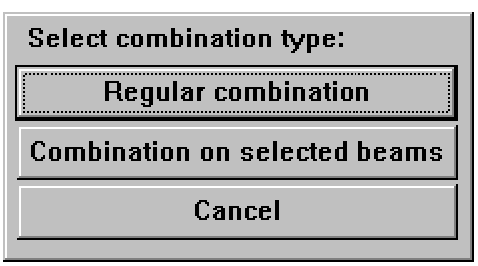
- Select load type:



Set the **Load type** *menu to* **combine ld.**

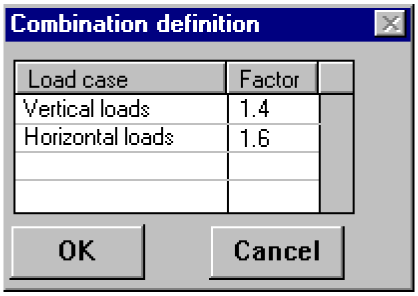
- And click the 

icon.



Click **Regular combination**

- Select load cases and enter the load factor so that the table appears as:

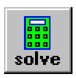


- Click the 

button.

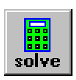
The loads are displayed graphically.

- Click the 

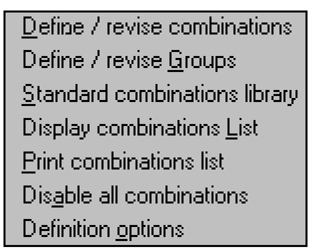
icon.
- Click the 

icon.
- Click the **No** button when the program asks **Do you want to save the matrix ?**

<u>Combinations defined after solution:</u>

- Click the 

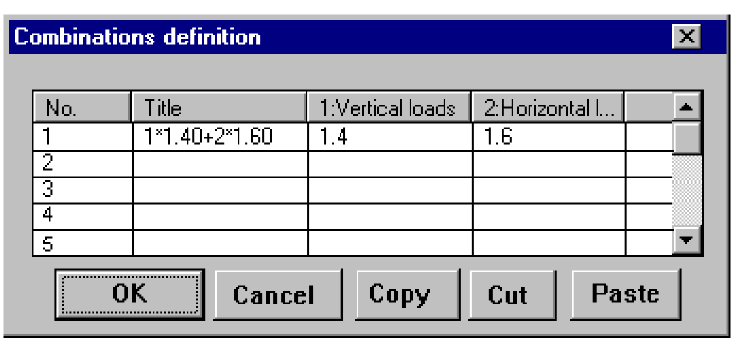
icon.
- Click the **No** button when the program asks **Do you want to save the matrix ?**
- Select **<u>C</u>ombinations** in the menu bar and **Define/Revise** in the combination menu.



Define/Revise combinations

- type in the load factors for each load case (the program will automatically create the Title).
- Click the 

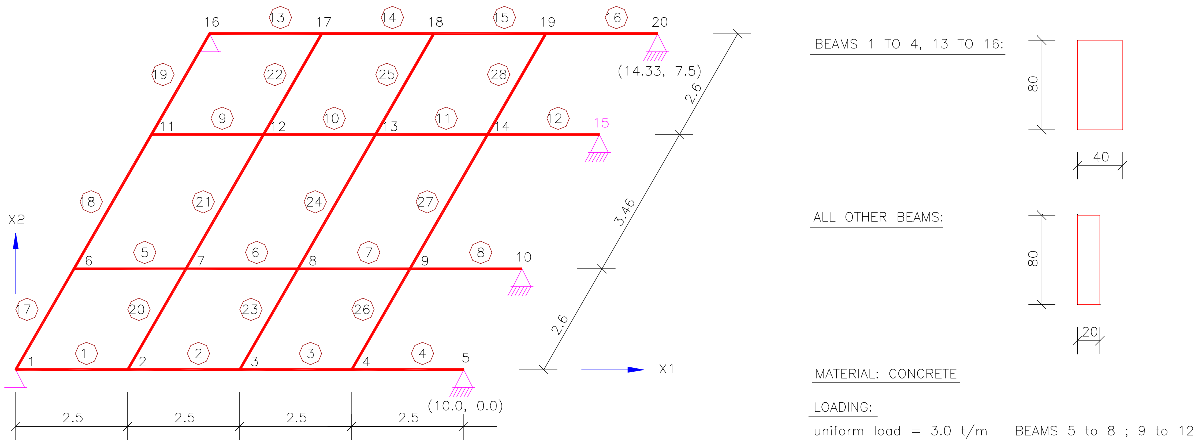
button.



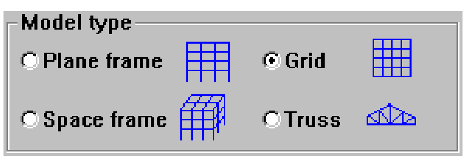
** This page is deliberately blank **

2 Plane Grid

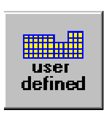
Solve the following plane grid of beams:



- **Main Menu**
  - select **Files** in the menu bar
  - select **New model** in the pull-down menu
  - enter the model title
- **Preliminary Menu**:

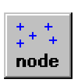


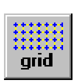
Set the model type to ***Plane grid***

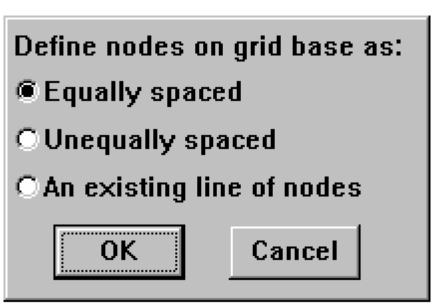
- Click  to proceed to geometry

- **Nodes**:

  Define the nodes with a **Grid** command:

  - click the  icon

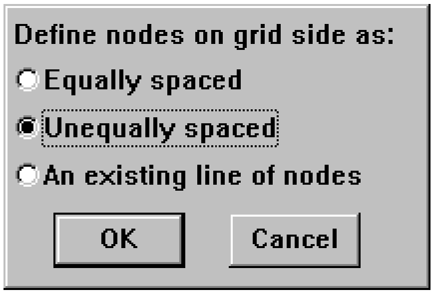
  - click the  icon



Select ***Equally spaced***

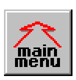
and click the  *button*

- Move the crosshair to **X1** = **0.0, X2** = **0.0** and click the mouse.
- Move the crosshair to **X1** = **10.0, X2** = **0.0** and click the mouse.
- Specify: **4** segments.



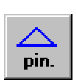
Select **Unequally spaced**

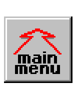
and click the OK button

- Move the crosshair to **X1** = **14.33, X2** = **7.5** and click the mouse. (You will have to decrease the **Step** to 0.01 to set the X1 coordinate).
- Move the crosshair along the line to the intermediate nodes and click the mouse. In this skew grid it is easier to type in **dD** rather than moving the crosshair:
- Move the crosshair into the **dD** text box.
- Type **2.6** and click OK, nodes 6 to 10 are displayed.
- Type **3.46** and click OK; nodes 11 to 15 are displayed.
- Move the crosshair outside the line 5-20 and click the mouse or click End; nodes 16 to 20 are displayed.
- Click the  icon.

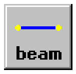
- **Restraints**
  - Click the  icon.

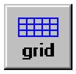
  - Click the  icon.

  - Select **Individual nodes**
  - Move the crosshair adjacent to node 5, 10, 15 and 20 so that the nodes are highlighted with the rectangular blip; click the mouse. After the node 20 has been selected, click the mouse again without moving the crosshair.
  - Click the  icon.

  - Select **Individual nodes.**
  - Move the crosshair adjacent to nodes 1 and 16 so that the nodes are highlighted with the rectangular blip; click the mouse. After the node 16 has been selected, click the mouse again without moving the crosshair.
  - Click the  icon.

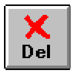
- **Beams:** Define all beams with a grid command –

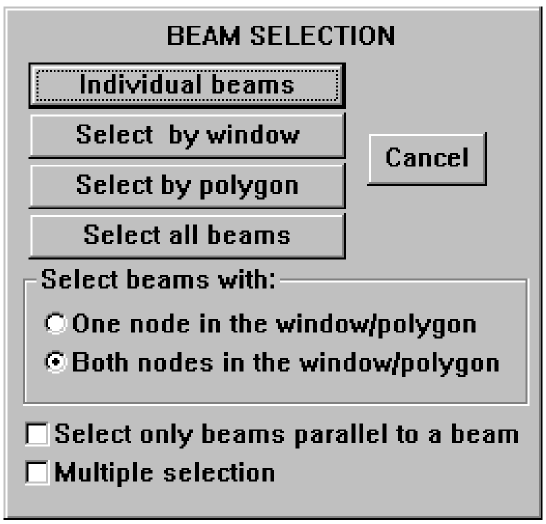
  - Click the  icon

  - Click the  icon

  - Select Node 1 as the start node of the base line:
  - Select Node 5 as the end node of the base line
  - Select Node 20 as the end node of the height line.

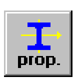
  The program draws a complete grid, including beams between nodes 5-10, 10-15, 15-20;to delete these beams -

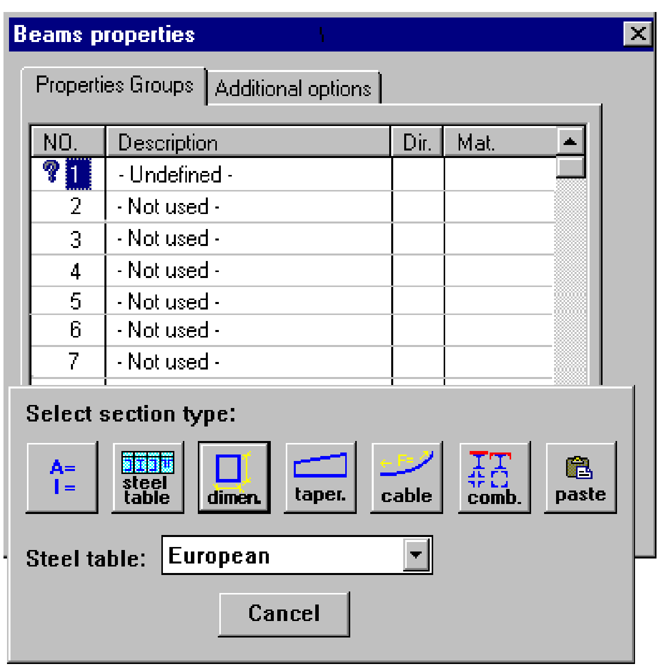
  - click the  icon.



← *Click* **Select by polygon**

← *Select* **Both nodes in the window/polygon**

  - Click the  icon.

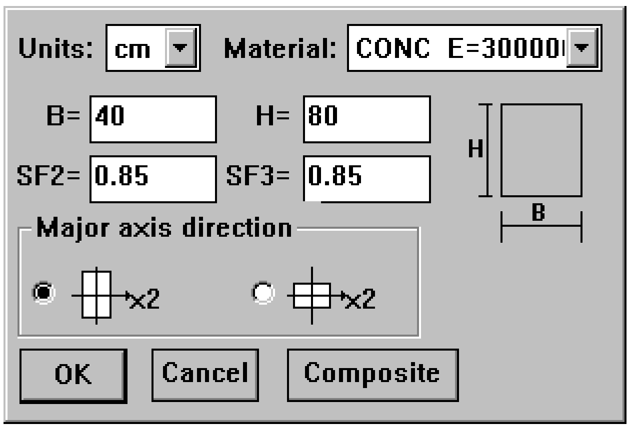


← *Click property no. 1*

← Click the **Define/Revise** *button*

← *Click the* dimen. *icon*

- Click the icon
- Enter the dimensions, as follows:



← *Select **Conc***

← *Define **B,H***

← *Define **SF2***

← *Define **x2** as the major axis*

← *Click the* OK *button*

Note that in plane grids I2 corresponds to the axis of bending.

- Similarly define property group 2 as a rectangular section with dimensions **H** = **80**; **B** = **20**.

Property group 1 was automatically assigned to all of the beam elements. To assign property group 2 to beams 5 to 12, 17 to 28:

- Select **2 – Rectangle H = 80 B = 20 MAJOR CONC**
- Click the Assign button.
- Set **One node in the window/polygon** and define a window enclosing nodes 6 to 15.
- Click the Close button to complete the property definition.
- Click the main menu icon.

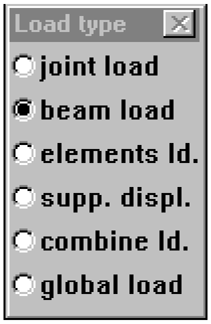
The geometry definition has been completed.

Click the loads icon.

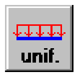
- **Loads**:

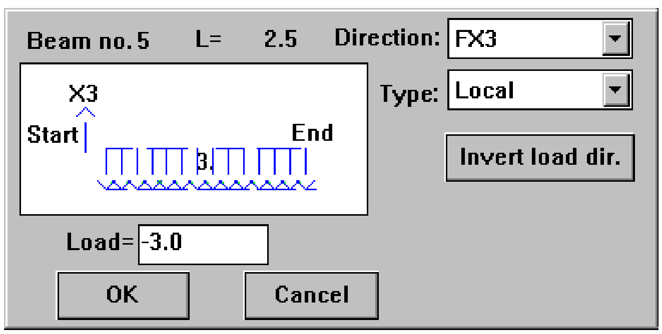
  Define all loads in load case 1

  - Click the new icon.
  - Define the load case title
  - select the load type:



*Set the **Load type** menu to* **beam load**

- click the  icon.
- click the  icon
- Select beams 5 to 8, 9 to 12.
- Define the load:

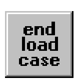


*Set the **Direction** to FX3*

*Set **Type** to **Local** or **Global***

Type in the load value

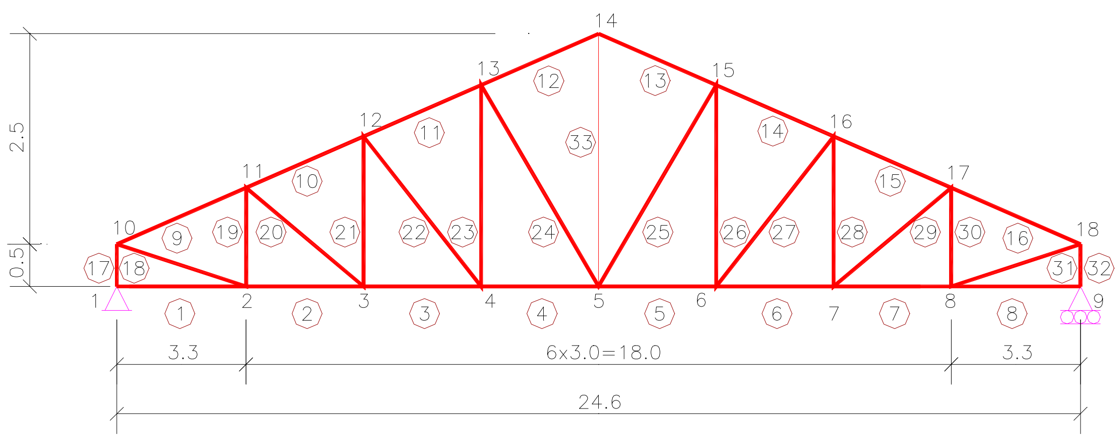
Click the  *button*

- Click the  icon.

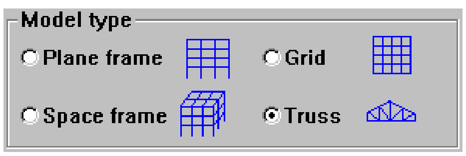
** This page is deliberately blank **

3 Plane Truss

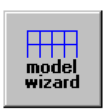
Define the geometry of the following plane truss using a “Model wizard”.

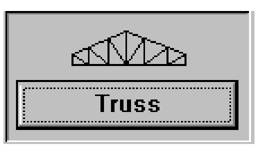


- **Preliminary Menu**:

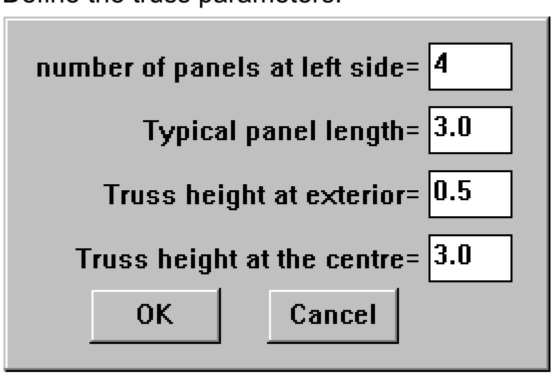


Set the model type to ***Truss***

- Click 

- Click 

- Define the truss parameters:

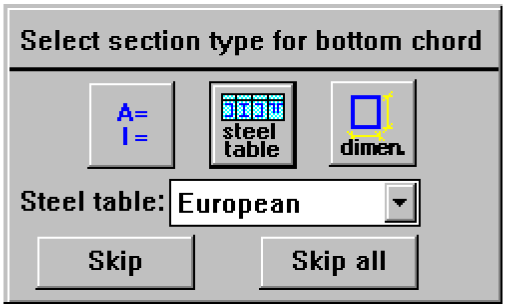


Define the length of the interior panels
(Exterior panel dimensions will be revised later)

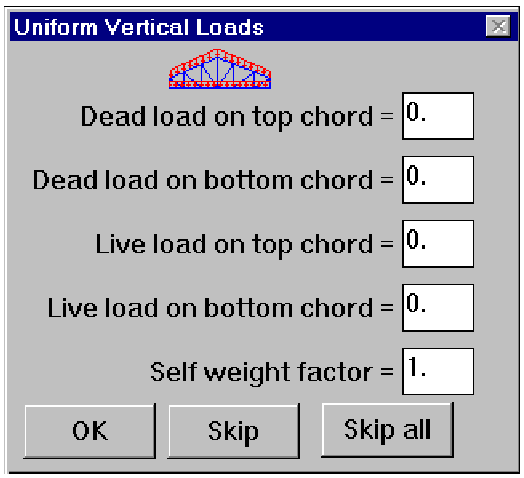
Press the 

button

- The following menu is displayed:



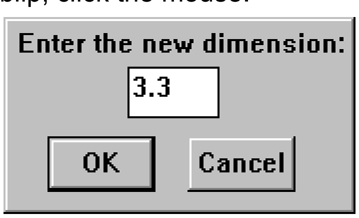
Click the Skip all button to exit from the menu.



Click the Skip all button to exit from the menu.

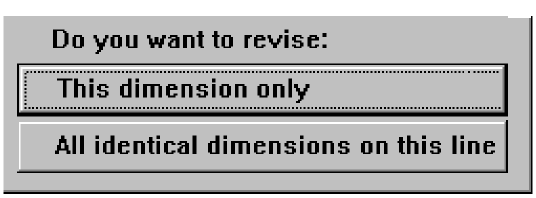
The program creates the model according to the parameters and displays it on the screen :

- To revise the exterior panel lengths, click the Dimension button.
- Move the cursor until the dimension of the left exterior panel (beam 1) is highlighted with the rectangular blip; click the mouse.



Enter the new dimension

and click the OK *button*

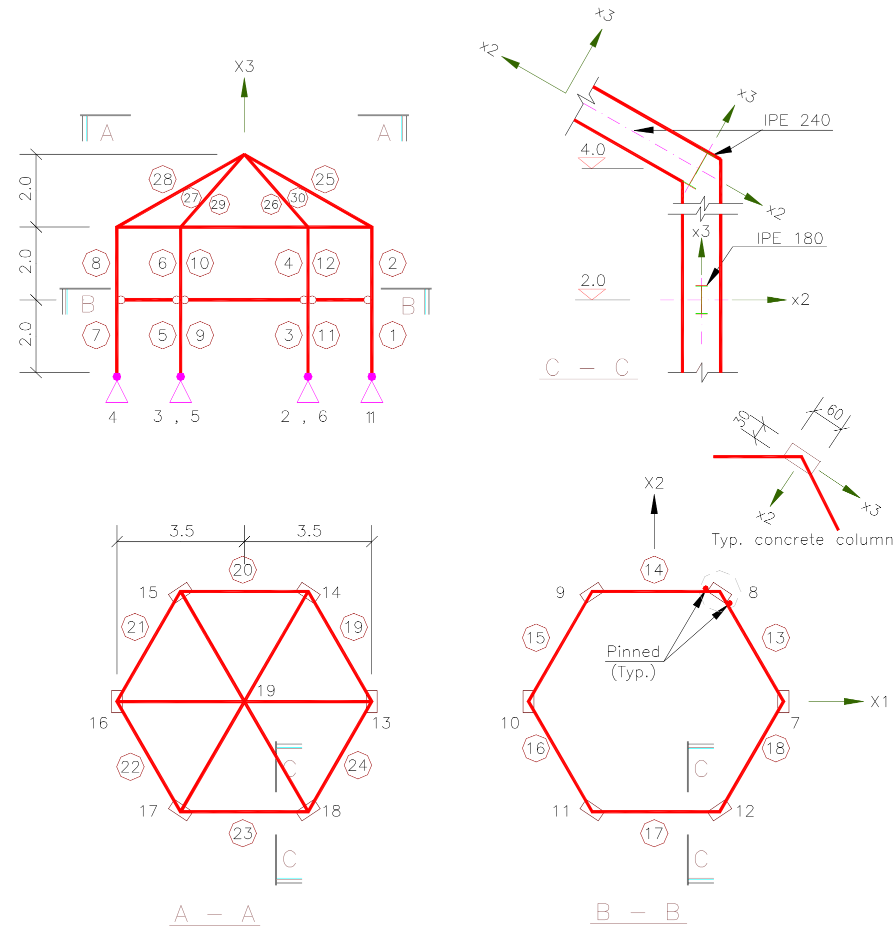


Revise ***This dimension only***

- Repeat for the right exterior panel (beam 8).
- Press **Esc** to end the revision of dimensions.

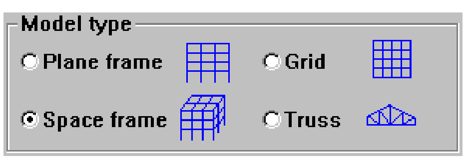
- Click the button to leave Model Wizard. The program now enters the regular geometry module.

4 Space Frame

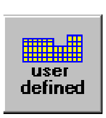


Load Case 1: -1.2 t/m distributed vertical load on roof beams
Self-weight of entire structure
Load Case 2: 1 cm. vertical initial displacement of one support node.

- **Preliminary Menu**:

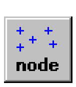


Set the model type to **Space frame**

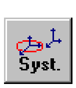
- Click 

to proceed to geometry

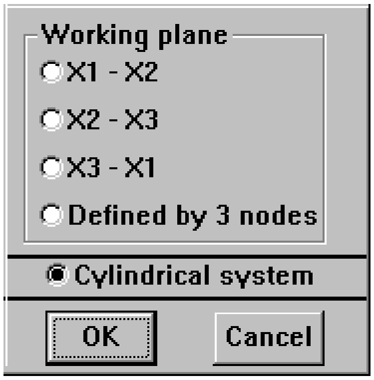
- **Nodes**:

- click the 

icon

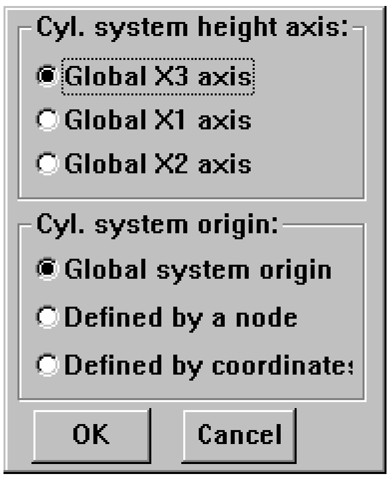
- click the 

icon



*Select a **cylindrical system***

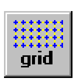
and click the  *button*

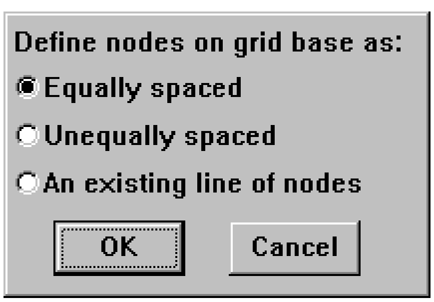


Select the **Global X3 axis**

and the **Global system origin**

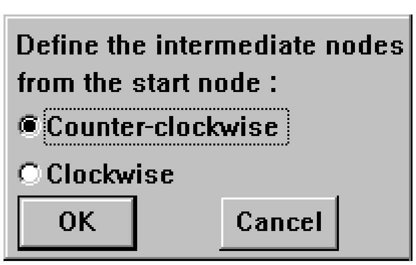
and click the  *button*

- click the  icon



Select **Equally spaced**

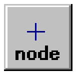
- Move the crosshair to **R = 3.5, f = 0,** **H = 0.0** and click the mouse.
- Move the crosshair to **R = 3.5, f= 300,** **H = 0.0** and click the mouse.
- Specify **5** segments.
- Specify :



Nodes 1 to 6 are created.

- Define the nodes on the grid side as **Equally spaced**
- Move the crosshair to **R = 3.5, f = 300, H = 4.0** and click the mouse.
- Specify **2** segments.

Nodes 7 to 18 are created (rotate the model to view all of the nodes).

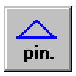
- click the 

icon
- Move the crosshair to **R = 0.0, f = 0, H = 6.0** and click the mouse.

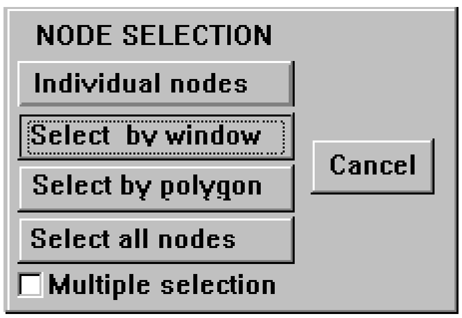
Node 19 is created.

- **Restraints**

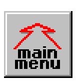
- Click the 

icon.
- Click the 

icon.

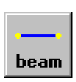


Click the **Select by window**

- Create a window enclosing nodes 1 to 6 (even though nodes 2 and 3 are hidden they will be included in the window). A pinned restraint is assigned to the selected nodes.
- Click the 

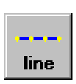
icon.

- **Beams**

- Click the 

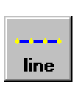
icon

Define the vertical beams:

- Click the 

icon
- Define beams 1-2 as a line: Select node 1 as the start node of the line and node 13 as the end node of the line; beams 1 and 2 are created.
- In a similar manner, define beams 3-4, 5-6, 7-8, 9-10, 11-12.

Define the horizontal rings:

- Click the 

icon

- Set **Define an arc**.
- Select node 7 as the start node of the arc, node 12 as the end node and any of nodes 8-12 as the third node; beams 13-17 are created.
- Click the icon
- Define beam 18 between nodes 12 and 7.
- In a similar manner, define the upper ring with beams 19-24.

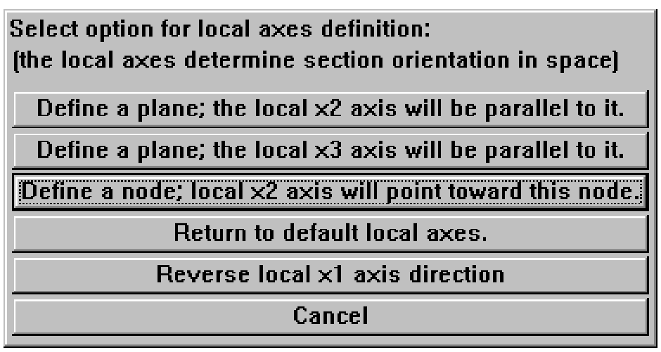
Define the roof beams:

- Click the icon
- Define beam 25 between nodes 13 and 19.
- In a similar manner, define the upper ring with beams 26-30.

Define the section orientation:

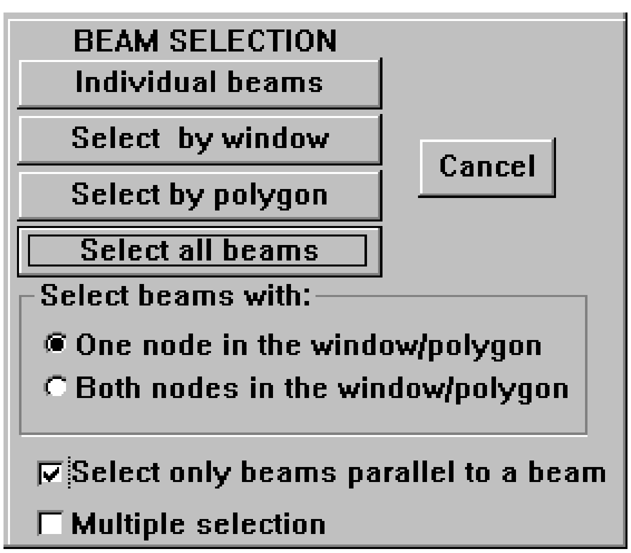
Rotate the local x2 axes of the columns so that they point towards the center of the circle:

- Click the icon



Click **Define a node**

- Move the crosshair until the rectangular blip highlights node 19; click the mouse.



Click **Select all beams**

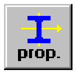
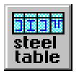
Select this option

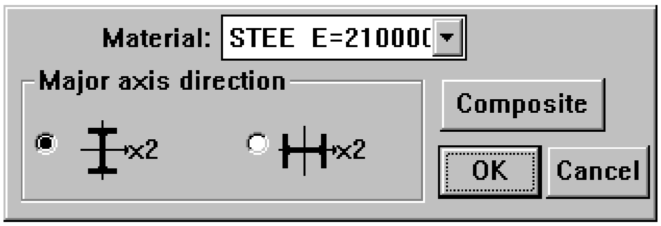
Select this option

and select any of beams 1 to 12 as the parallel beam.

- Similarly rotate the local **X2** axes of the top ring beams 19-24 so that they point towards the peak:

Define properties:

- Click the  icon.
- Select property group **1 – UNDEFINED-**
- Click the Define/Revise button.
- Click the  icon
- Click **IPE** in the section type list and **240** in the section list.



Select **Steel**

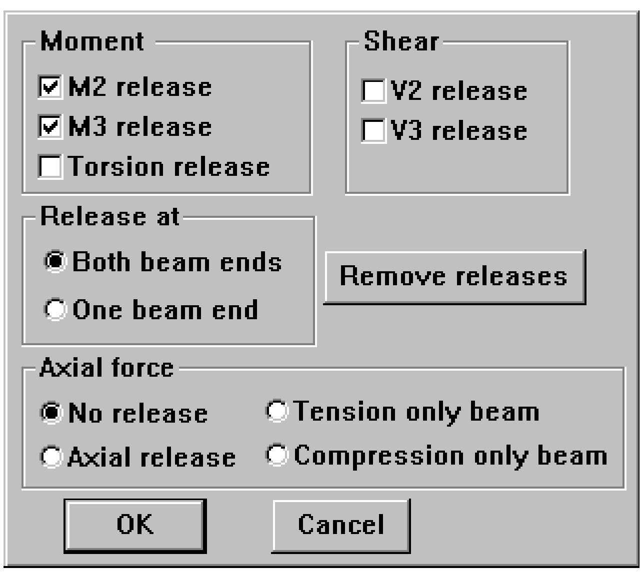
Define ***X2*** *as the major axis*

Click the OK *button*

- Similarly, define Property group 2 as **IPE 180** and Property group 3 as a 30x60 rectangular concrete section.
- Select property group **2 IPE 180.**
- To assign property 2 to the ring beams, click the Assign button.
- Select ring beams 13-18.
- Similarly assign property group 3 to the vertical columns.

Define Releases:

- Click the  icon



Release both moments

At both ends

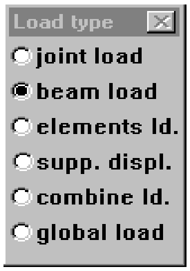
Click the OK *button*

- Select property group **2 IPE 180.**
- Select the lower ring beams 13 to 18.

- **Loading:**

  - Click the 

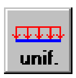
icon.
  - Define the load case title
  - Select load type:

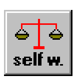


Set the Load type menu to **beam load**

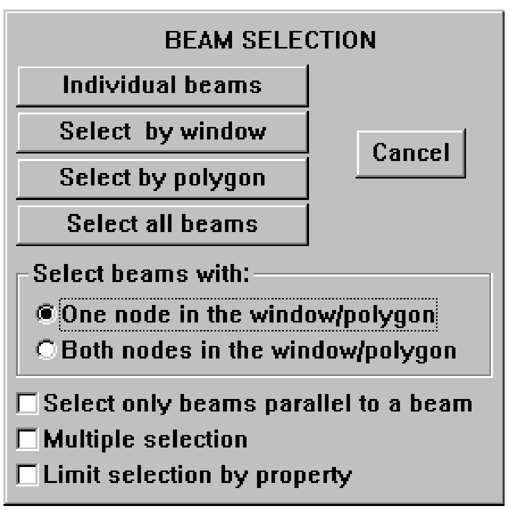
and click the 

icon.

  - Click the 

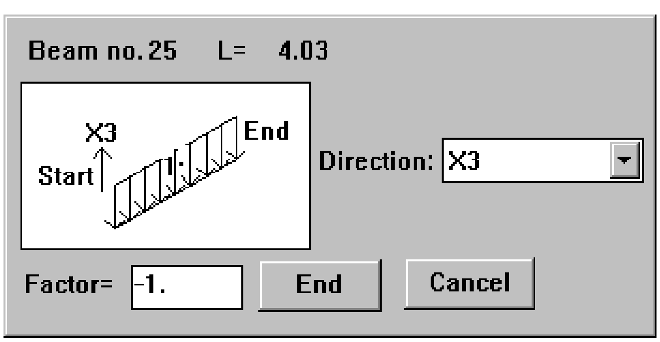
icon
  - Select all roof beams 25 to 30.
  - Define a load of -1.2 t/m in the Global X3 direction.
  - Click the 

icon



Click the **Select all beams**

  - Define the load:



Set the Direction to **X3**

Type in the factor and click 

- Click the 

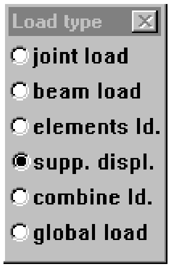
icon.

- Click the 

icon.

- Define the load case title.

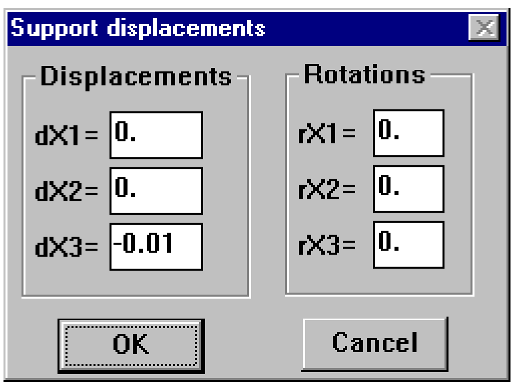
- Specify the load type:



set the **Load type** *menu to* **support displacement**

- click the 

icon.



Define a vertical displacement of 1 cm

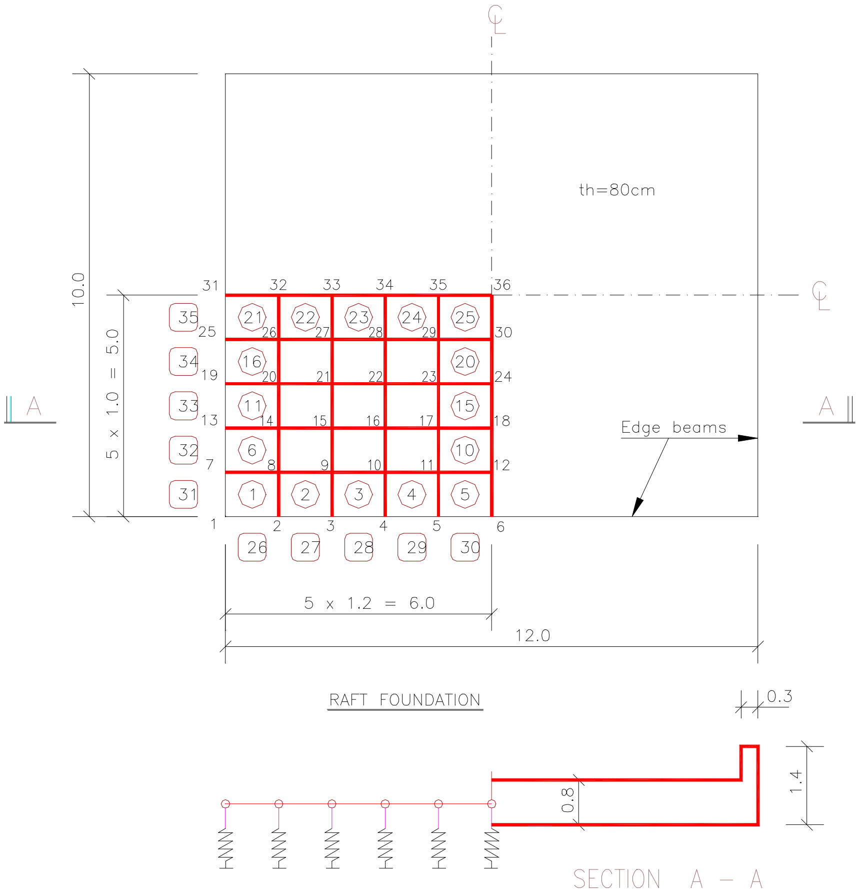
and select node 1.

- Click the 

icon.

** This page is deliberately blank **

5 Plate Bending

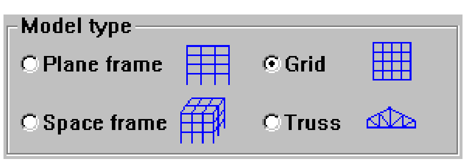


This raft foundation the use of elastic supports and the solution of a symmetric structure where only one-quarter of the structure is entered.

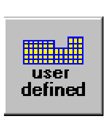
Vertical Loads:	Element pressure	=	-3.0 t/m^2
	Uniform load on edge beam	=	-20.0 t/m
	Concentrated load at nodes 15, 17, 27	=	-200.0 t

- **Main Menu**
  - select **Files** in the menu bar
  - select **New model** in the pull-down menu
  - enter the model title

- **Preliminary Menu**:



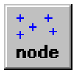
← *Set the model type to* **Grid**

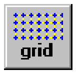
- Click 

 to proceed to geometry

- **Nodes**:

  Define the nodes with a **Grid** command:

  - click the 

 icon
  - click the 

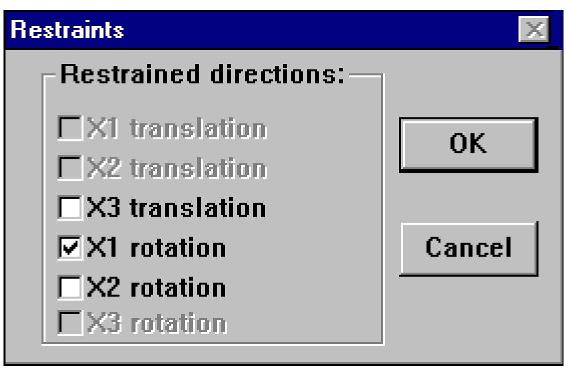
 icon
  - Define the base line as  **Equally spaced**
  - Move the crosshair to **X1** = **0.0, X2** = **0.0** and click the mouse.
  - Move the crosshair to **X1 = 6.0**, **X2** = **0.0** and click the mouse.
  - Specify: **5** segments.
  - Define the height line as  **Equally spaced**
  - Move the crosshair to **X1 = 6.0**, **X2** = **5.0** and click the mouse.
  - Specify: **5** segments.

- **Restraints**

  - Click the 

 icon.
  - Click the 

 icon.



Restrain rotation about ***X1***

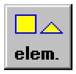
  - Use **Select by Window** and create a window that includes nodes 31 to 35.
  - Click the 

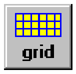
 icon.
  - Arrange the menu so that ☑ X2 rotation is restrained.
  - Use **Select by Window** and create a window that includes nodes 6,12,18, 24, 30.
  - Click the 

 icon.
  - Arrange the menu so that ☑ X1 rotation ☑ X2 rotation is restrained.
  - Use **Individual nodes** and select node 36
  - click the 

 icon.

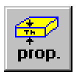
- **Elements**: Define all elements with a grid command –

  - click the 

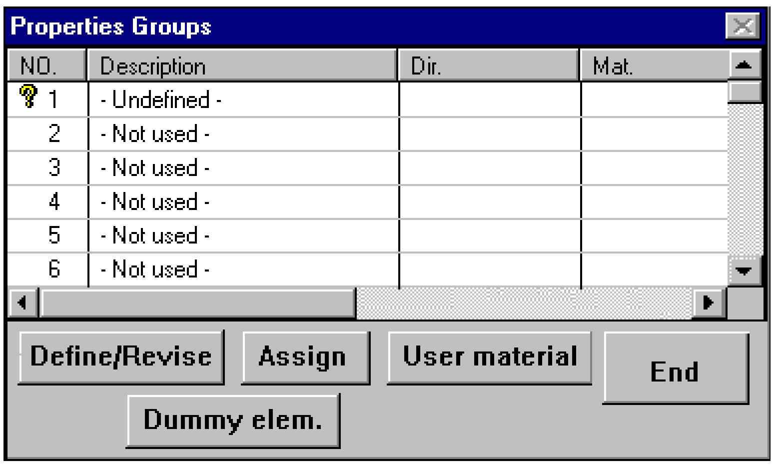
icon.
  - click the 

icon.
  - Select Node 1 as the start node of the base line
  - Select Node 6 as the end node of the base line
  - Select Node 36 as the end node of the height line

  Define the element thickness:

  - click the 

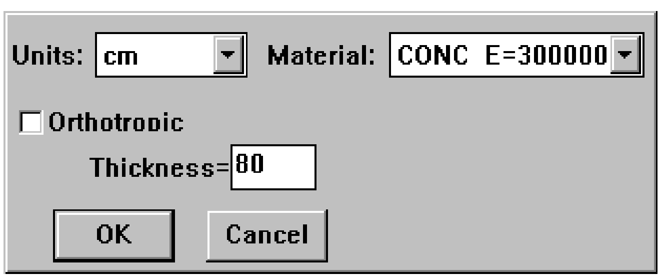
icon.



*Select **1 -Undefined-** (assigned by default to all elements)*

Click the Define/Revise *button.*

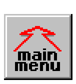
  - Enter the property values:



Select **Concrete**

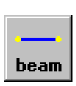
Type in **Thickness = 80**

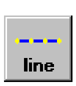
and click the OK *button*

  - Click the End button.
  - click the 

icon.

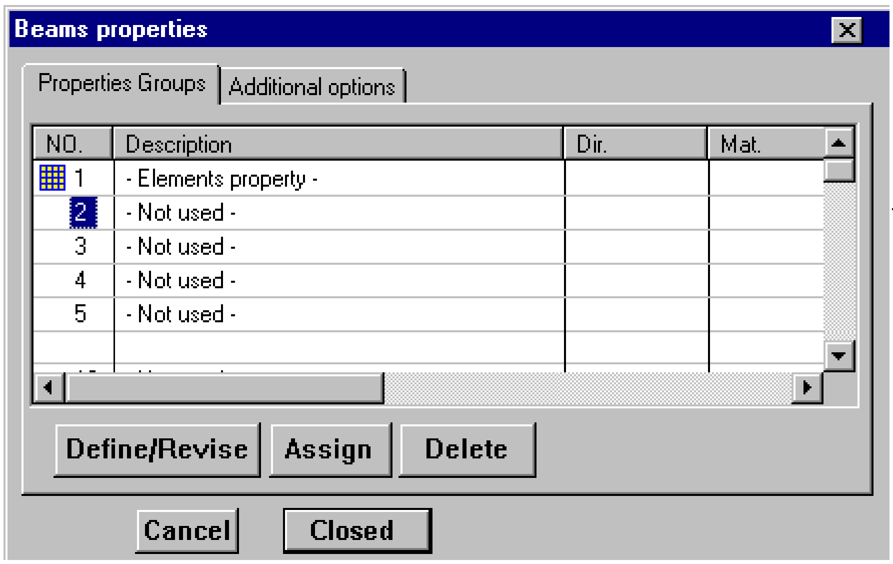
Beams: Define all beams with line commands:

  - Click the 

icon
  - Click the 

icon
  - Select node 1 as the start node of the line and node 6 as the end node of the line; beams 26 to 30 are created.

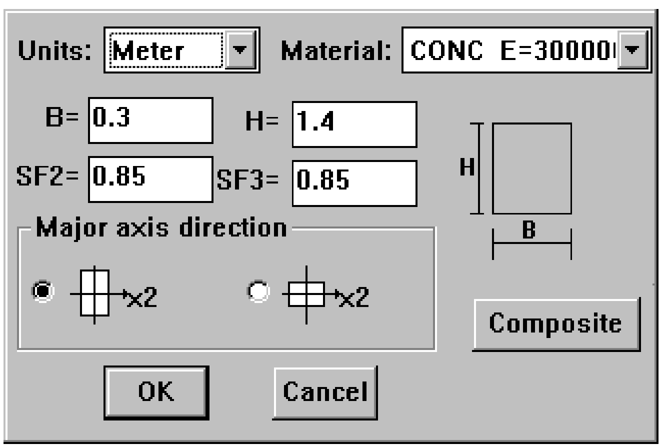
- Click the line icon
- Select node 1 as the start node of the line and node 31 as the end node of the line; beams 31 to 35 are created.
- Click the prop. icon.



← *Select* **2 -Not used-**

← *Click the* Define/Revise *button.*

- Click the dimen. icon and the icon



← *Select* **Conc**

← *Type in section dimensions*

← *Select major axis directions*

← *Click the* OK *button*

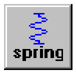
Note that in plane grids **I2** corresponds to the axis of bending.

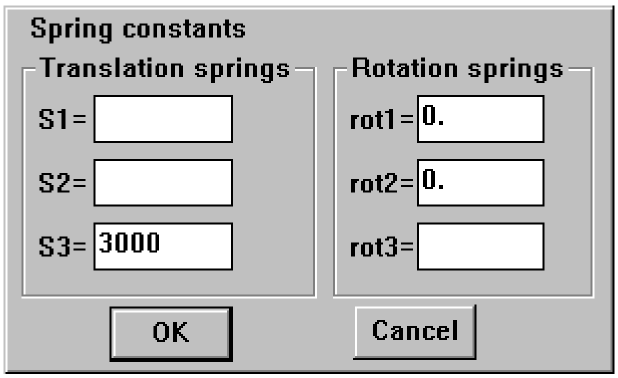
Property 1 (the element property) was automatically assigned to all of the beams. To assign property 2 to the beams:

- Click the Assign button.
- Select **2 – Rectangle H = 1.40 B = .30 MAJOR CONC**
- Create a rectangular window enclosing beams 26 to 35
- Click the Close button.

- click the  icon.

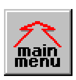
- **Springs**:

  - click the  icon.
  - click the  icon.
  - Enter the spring constant:

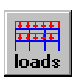
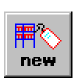


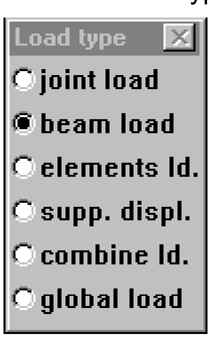
Define **S3 = 3000**

Click the  *button*

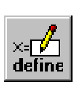
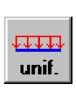
  - Select all nodes in the model.
  - click the  icon.

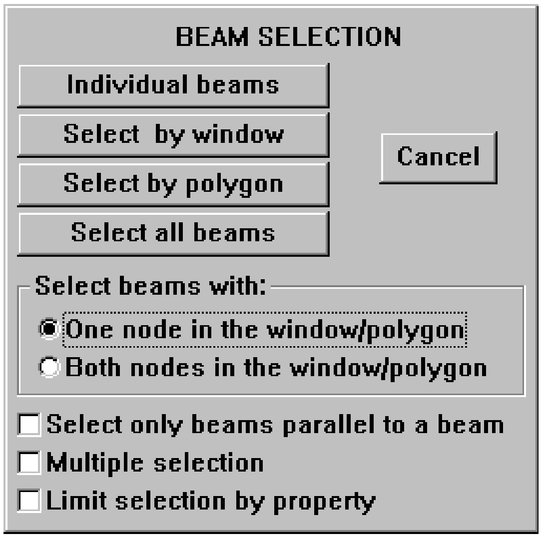
- **Loads**:

  - Click the  icon.
  - Click the  icon.
  - Define the load case title
  - Select the load type:



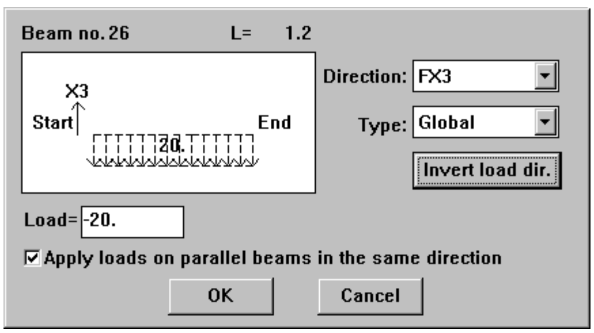
Set the **Load type** menu to **beam load**

  - and click the  icon.
  - Click the  icon



Click **Select all beams**

- define the load:



Set direction FX3

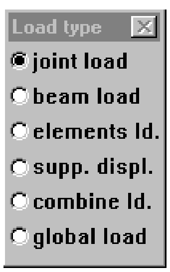
Set type ***Global***

Set ***load*** *=-20*

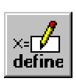
Click the 

button

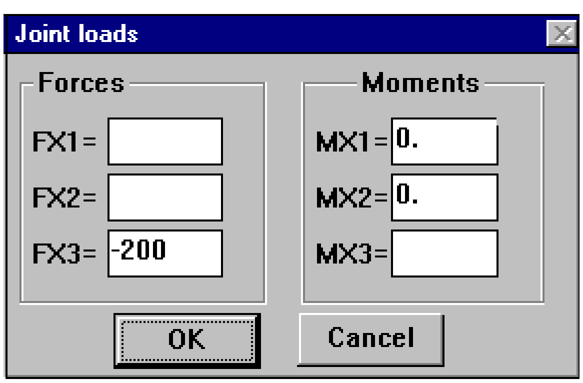
- specify the load type*:*



Set the **Load type** *menu to* **joint load**

- click the 

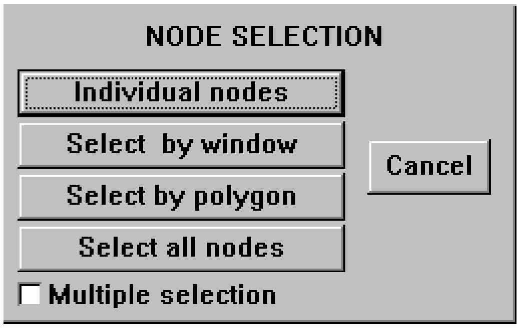
icon.



Type in the load value **FX3=-200**

Click the 

button.

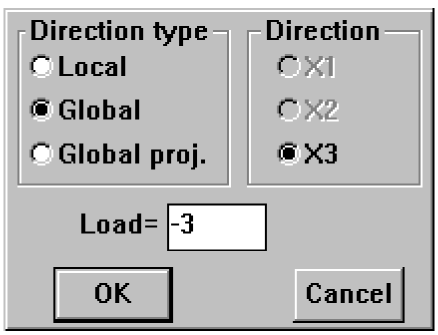


← *Click* **Select individual nodes**

- Select nodes 15, 17, and 27; after the node 27 has been selected, click the mouse again without moving the crosshair. The load is applied to the three nodes.

Define the element loads:

- set the **Load type** menu to elements ld. and click the define icon.
- click the unif. icon.
- Select the load direction and enter the load value:

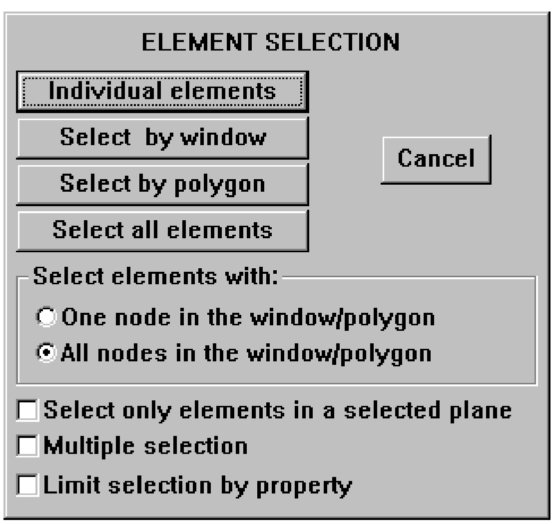


← *Set* **type** to **Global** *or* **Local**

← *Set* **Direction** *to* **X3**

← *Define* **Load** = **-3.0**

← *Click the* OK *button*



← *Click* **select all elements**

- Click the end load case icon.

** This page is deliberately blank **

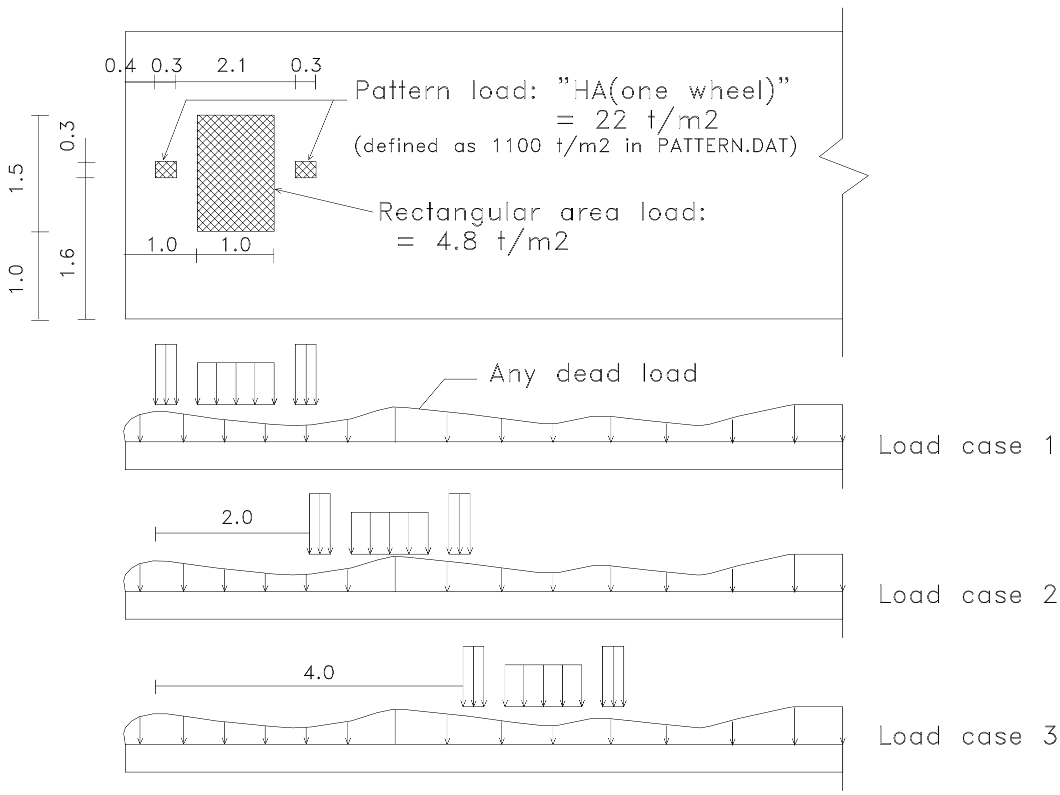
6 Moving Loads

Use the Moving Load option to generate a series of load cases where the live load each case is offset by a constant increment relative to the previous case.

Note that the Moving Load option applies only to loads defined as Global Loads.

The Figure below shows an arbitrary element grid for which three load cases must be defined. In all three cases the loads are identical, but the live load in any case is offset 2.0 m from the live loads in the previous case.

Two types of Global Loads are defined:
- a pressure on a rectangular area defined by the user.
- a pressure on a rectangular area retrieved from the "Pattern' file.



Two methods for generating the loads will be illustrated:

- Option (a):
  - Define the dead and live loads in load case 1. Generate the two additional load cases using the moving loads option. Note that the dead loads will be copied automatically to load cases 2 and 3.

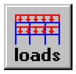
- Option (b):
  - Define the dead loads in load case 1 and the live loads in load case 2.
  - Generate the two additional load cases using the moving loads option. Note that the dead load will be not be included in any of these three load cases.
  - After solving the model, combine the dead load with each of the three live load cases using the Group" option in the "Combination" module.

- **Geometry**

  Define the geometry of the grid as outlined in Example 5.

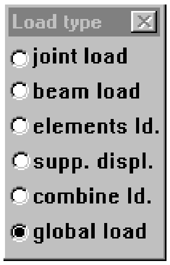
- **Loads**

<u>Option (a)</u>

- Click the 

icon.
- Click the 

icon.
- Enter a load case title
- Select the load type:

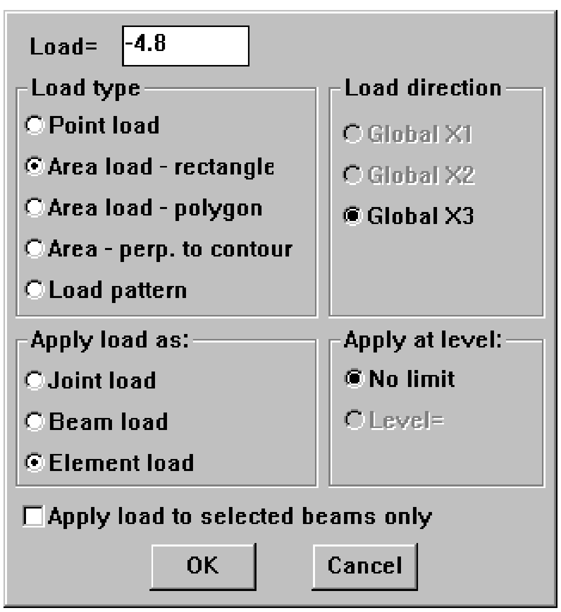


Set the **Load type** *menu to* **Global load**

- and click the 

icon.

Define the rectangular load:



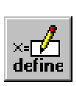
Select **rectangle**

Apply to **elements**

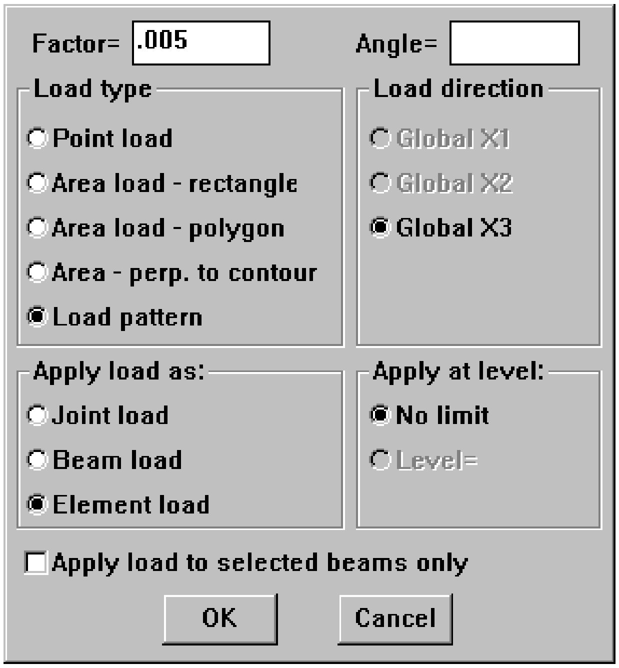
Click 

- Define the area using **Select by window**; click the mouse at the lower-left and upper-right corners of the rectangle at the coordinates show in the Figure.

Define the "Pattern" load:

- click the 

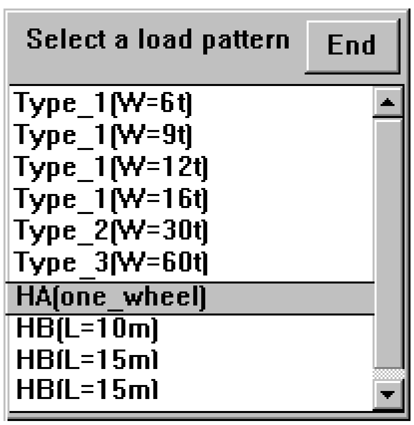
icon.



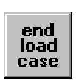
← *Select* **Load pattern**

← *Apply to* **elements**

← *Click* 

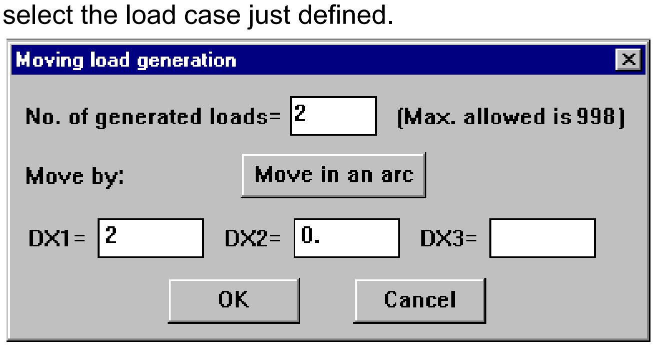


← *Select this load*

- Move the crosshair to the coordinates of the lower-left corner of the small rectangle as shown in the Figure and click the mouse.
- Repeat for the second wheel load.
- Click the 

 icon.
- Click the 

 icon.
- select the load case just defined.



← *Generate* **2 add'l loads**

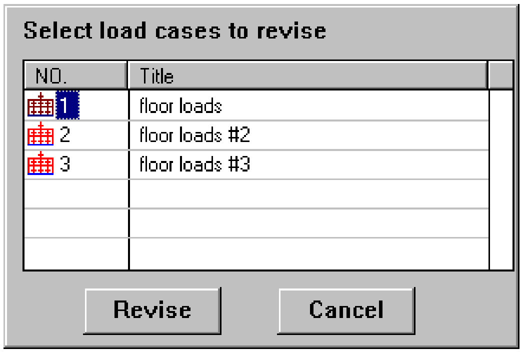
← *Move 2 m each load case.*

← *Click* 

- Click **End** in the following menu (there are no more moving load cases are required).

To check that the two cases were generated:

- Click the **revise** icon. The load cases now appears as:



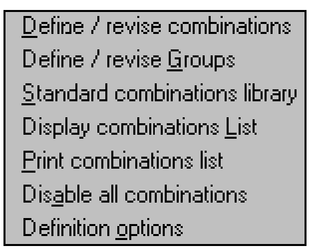
← *first load case*
← *generated load case #1*
← *generated load case #2*
← *Click* **Cancel**

Note that all load cases contain the same dead loads.

- Solve the model.

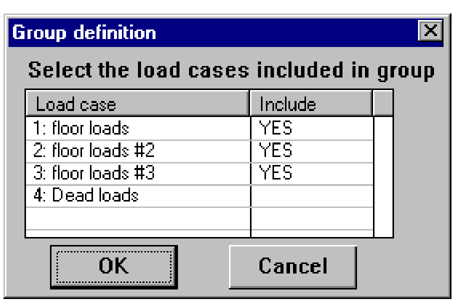
Option (b)

- Define the dead loads in load case 1.
- Define the global loads in load case 2.
- Generate the moving loads as load cases 3 and 4, as outlined above.
- Solve the model.
- Select Combination in the Menu bar.



← *Select* **Define/revise groups**

- click the **Add / revise a group** button
- Click **-UNDEFINED-** and enter a name for group.



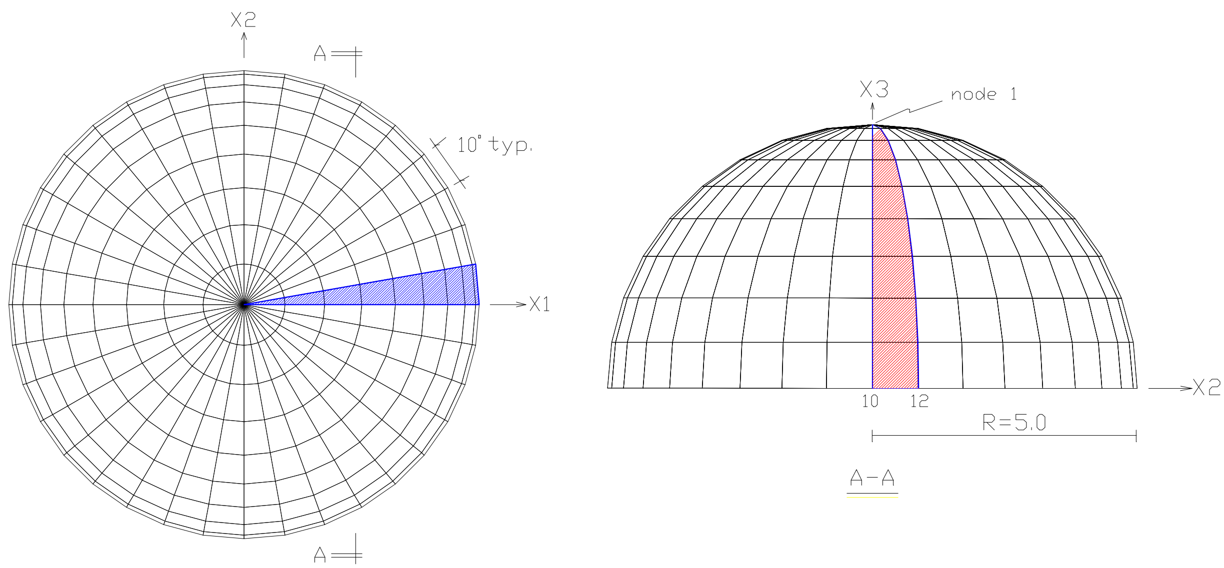
← *Include these cases in the group*
← *and click the* **OK** *button*

- Click the **Generate a combination for each load in the group** button.
- Click **End**

7 Dome Shell

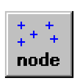
Define the dome shell shown in the Figure below using either of two options:

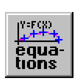
- Equations
- Copy and Rotate



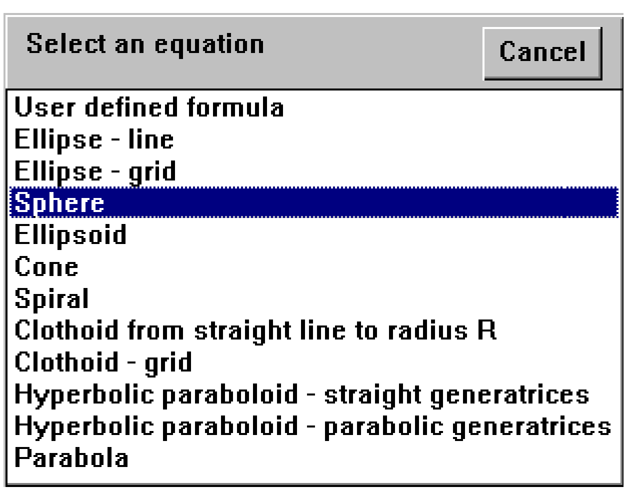
- Method 1 - **Equations**

  Define the nodes using a predefined equation:

  - click the 

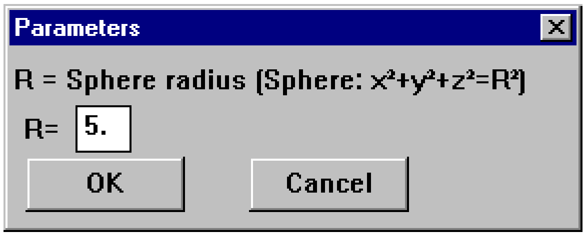
 icon
  - click the 

 icon



Highlight ***Sphere*** *and click the mouse*

Refer to the program **Help** for a detailed explanation of the sphere parameters **U,V** and **R.**

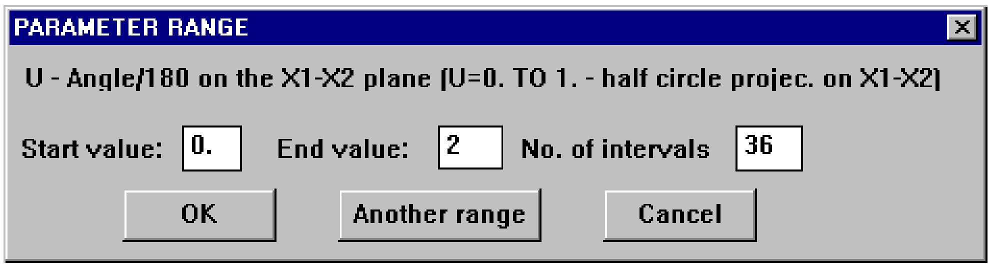


Define **R= 5.0**

and click OK

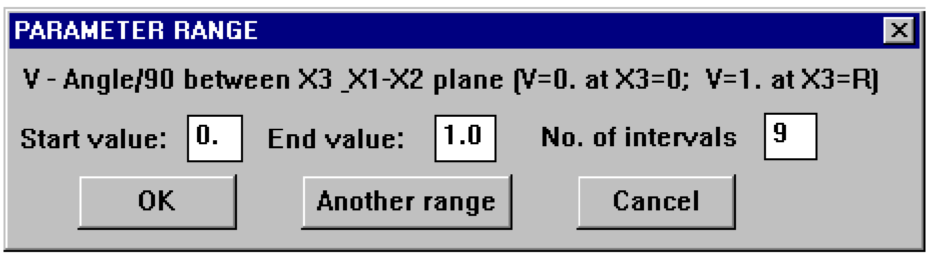
Define the variable ranges as follows:

- **U** from 0 to 1 represents a half circle of the base:

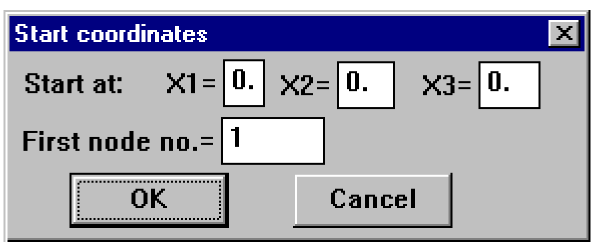


click OK

- **V= 0 or 1** represents the full height of a half – dome:



click OK



Specify location of dome center

Click OK

- Click the Define elements button to define all of the elements in the dome.

Two sets of nodes will be generated along line 1-10 and 36 nodes will be generated at the top of the dome.

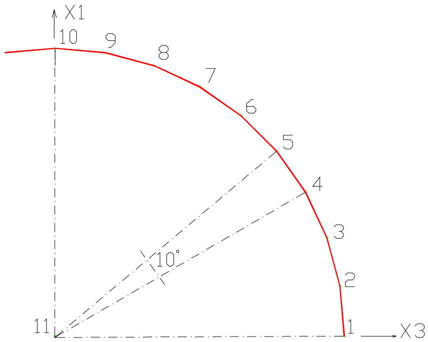
To delete the superfluous nodes, click the unify icon.

- Method 2 - **Copy**

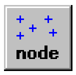
This method makes extensive use of the Copy option using a cylindrical coordinate system..

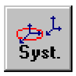
To create the dome, define the wedge 1-10-12 (shaded in the Figure) and then copy it 35 times using the **Copy – translate and rotate** option.

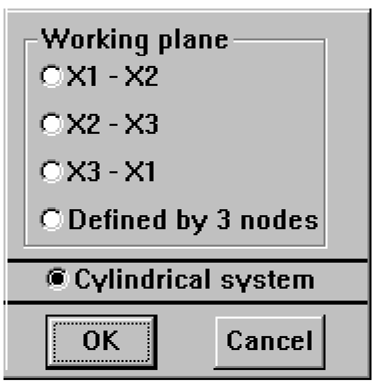
Define the arc 1-10:



X2 is the height axis of this arc (the axis perpendicular to the plane of the cylinder)

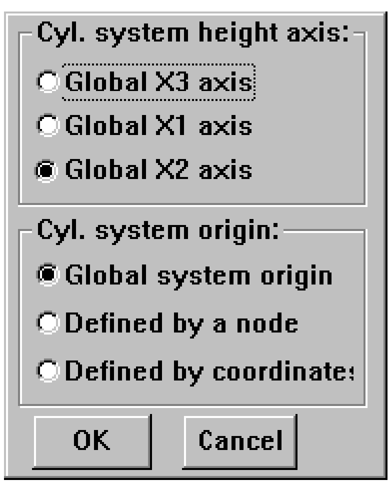
- click the  icon

- click the  icon



Set **Cylindrical system**

click the  button

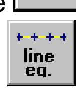


Set **Global X2 axis**

Set **Global System origin**

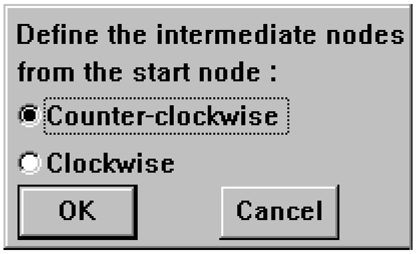
Click  *button*

- Click the  button to rotate the model to the screen plane.

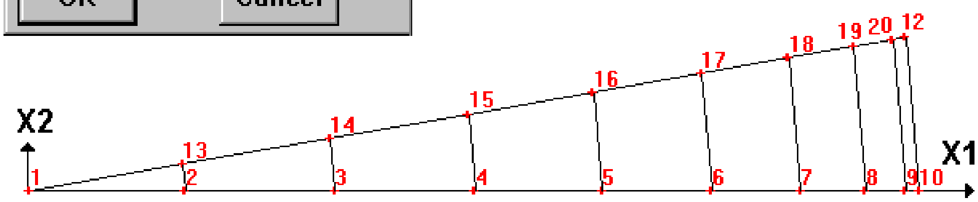
- click the  icon

- define the line start (node 1) at **R = 5.0 f =0.0 H = 0.0**
- define the line end (node 10) at **R = 5.0 f =90.0 H = 0.0**

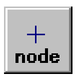
- set the **number of segment = 9** (10° each)

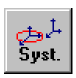


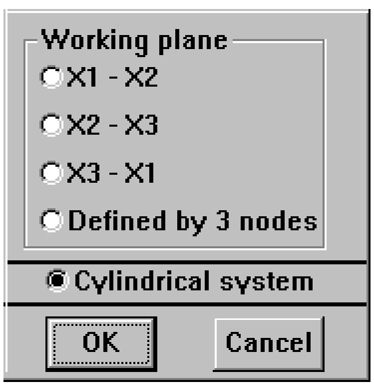
Set **Counterclockwise**



Generate the arc 1-12 by **Copy** arc 1-10.

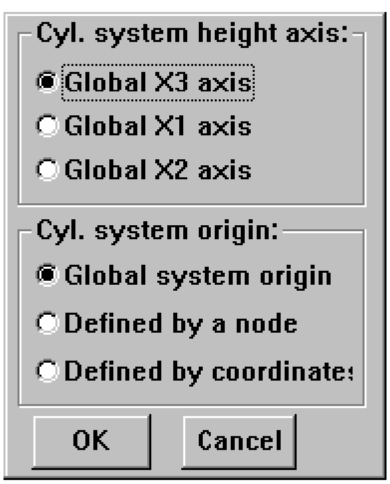
- click the  icon

- define a node at: **R = 0.0 f =0.0 H = 0.0** i.e. at the system origin**.** This node is required for the following **Copy + rotate** command.
- click the  icon



Set **Cylindrical system**

and click the OK *button*

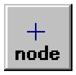


Set **Global X3 axis**

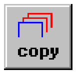
Set **Global system origin**

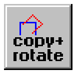
and click the OK *button*

- Click the OK button to rotate the model to the screen plane.

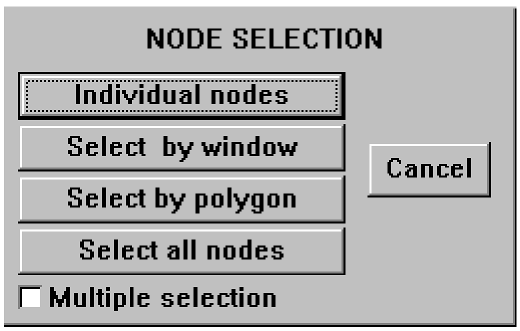
- click the 

icon
- define node 12 at **R = 5.0 f = 10.0 H = 0.0-**
- to facilitate node selection, click **Rotate** in the menu bar and rotate the model to: **X = -10 , Y = -10 , Z = 0.-**
- click the 

icon
- click the 

icon
- click the 

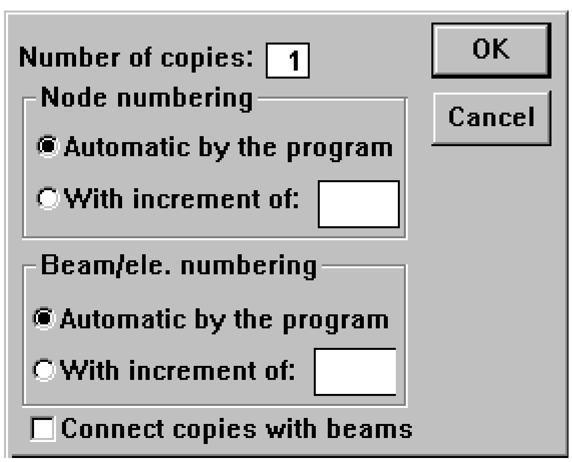
icon



Click **Select by polygon**

- define a polygon enclosing arc 1-10.
- select the 3 reference nodes and their new location:
  node 10 to node 12
  node 1 to node 1
  node 11 to node 11
- Select the new location by clicking the 

button.



Set **Number of copies** *to* **1**

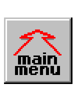
Select **Automatic numbering**

Select **Automatic numbering**

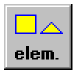
- click 

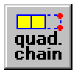
The program now generates arc 1-12.

Define the elements in wedge 1-10-12:

- click the 

icon

- click the 

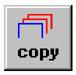
icon
- click the 

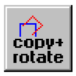
icon and define all the quad elements.
- click the 

icon and define all the triangular element 1-2-13.

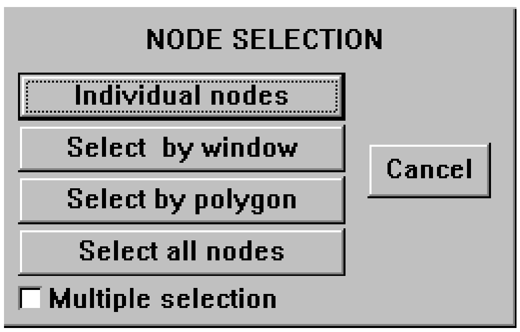
Generate the remaining wedges to create the entire dome:

- click the 

icon
- click the 

icon
- click the 

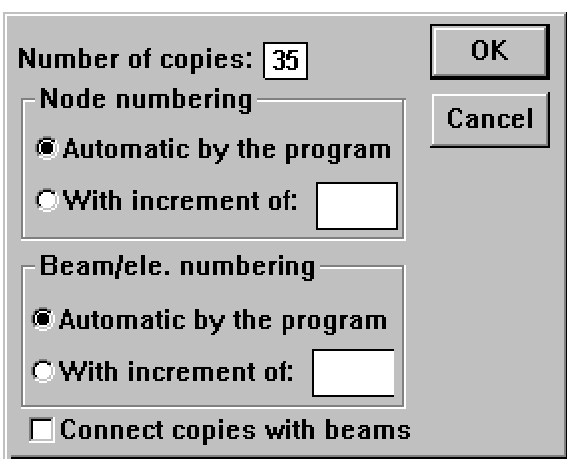
icon



← *Click* **Select by polygon**

- define a polygon enclosing the entire wedge
- select the 3 reference nodes and their new location:
  node 10 to node 12
  node 2 to node 13
  node 1 to node 1
- Select the new location by clicking the 

button.



← *Set* **Number of copies** *to* **35**

← *Select* **Automatic numbering**

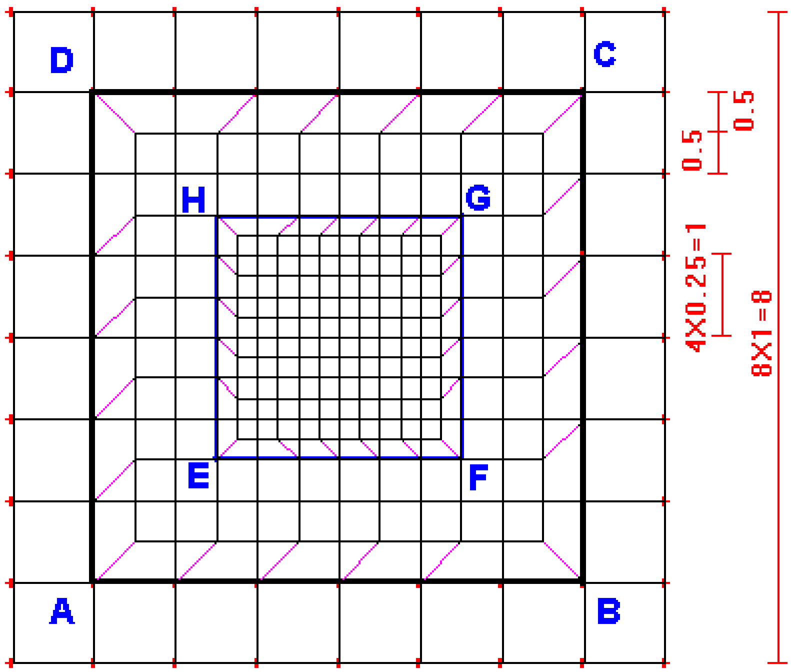
← *Select* **Automatic numbering**

The program now generates the entire dome.

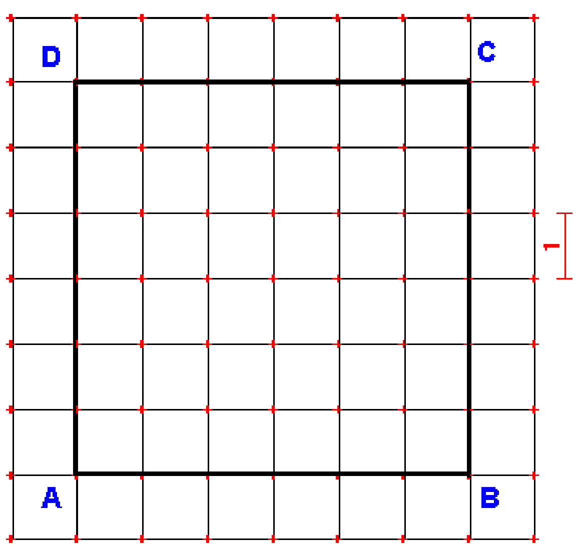
- select **Zoom** and **Full Drawing** to display the entire model.

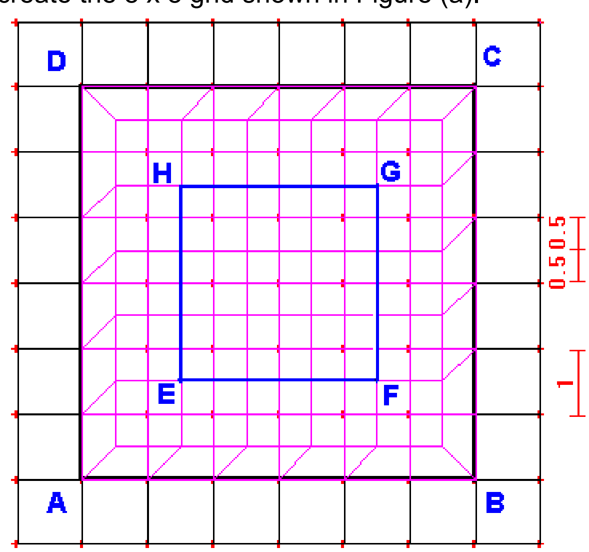
8 Refined Mesh

Define the gradually refined mesh:

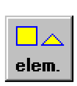
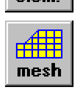


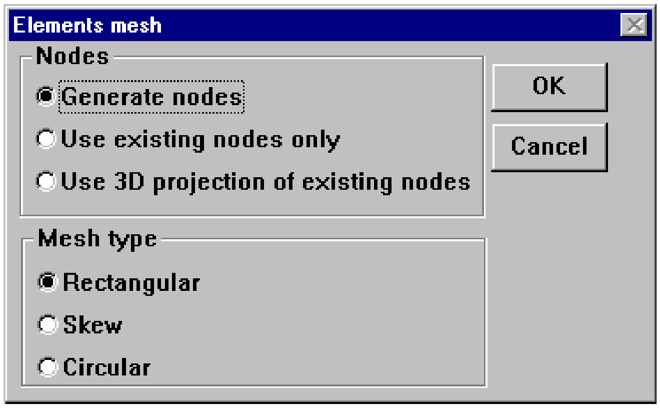
Use the Node “Grid” and Element “Grid” options to create the 8 x 8 grid shown in Figure (a).





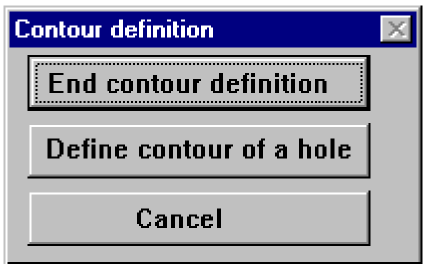
(a) (b)

- click the  icon
- click the  icon



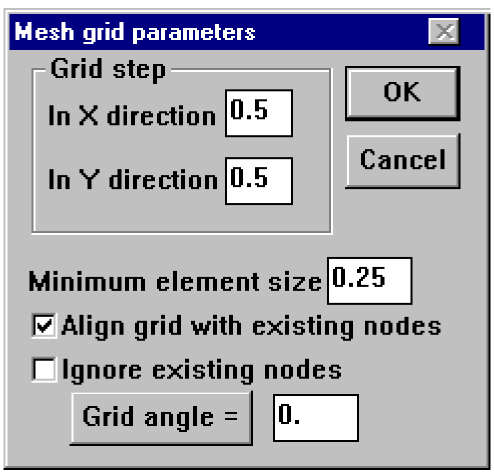
← *Select* **Generate nodes**

- Define the contour by selecting nodes A, B, C,D and A again to close the contour.



← *Select* **End contour definition**

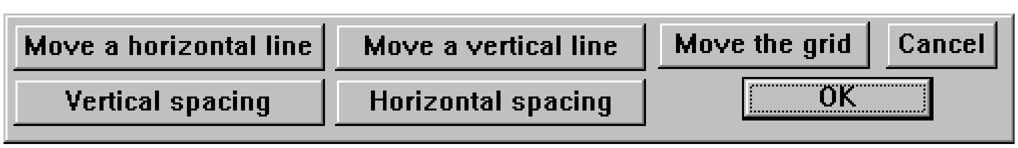
- Arrange the mesh parameters menu as follows:



← *Set the* **Grid step** *to* ***0.5*** *in the both directions*

← *Click the* OK *button*

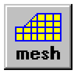
- The program superimposes the preliminary mesh (0.5 x 0.5) on the area defined by the contour.

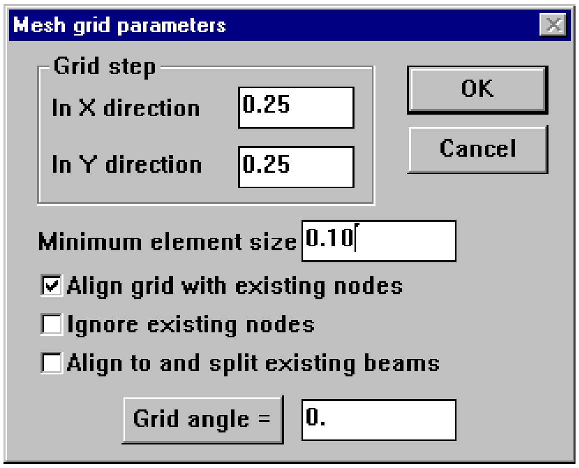


- Click the OK button.

The elements as displayed in Figure (b) are generated. Note that the existing elements in the contour area are erased.

- Click the OK button when the program asks **Is the mesh OK?**

- click the  icon
- Select **Generate nodes.**
- Define the contour by selecting nodes **E, F, G, H** and **E** again to close the contour.
- Select **End contour definition.**
- Arrange the mesh parameters menu as follows:



Set the **Grid step** *to* **0.25** *in both directions*

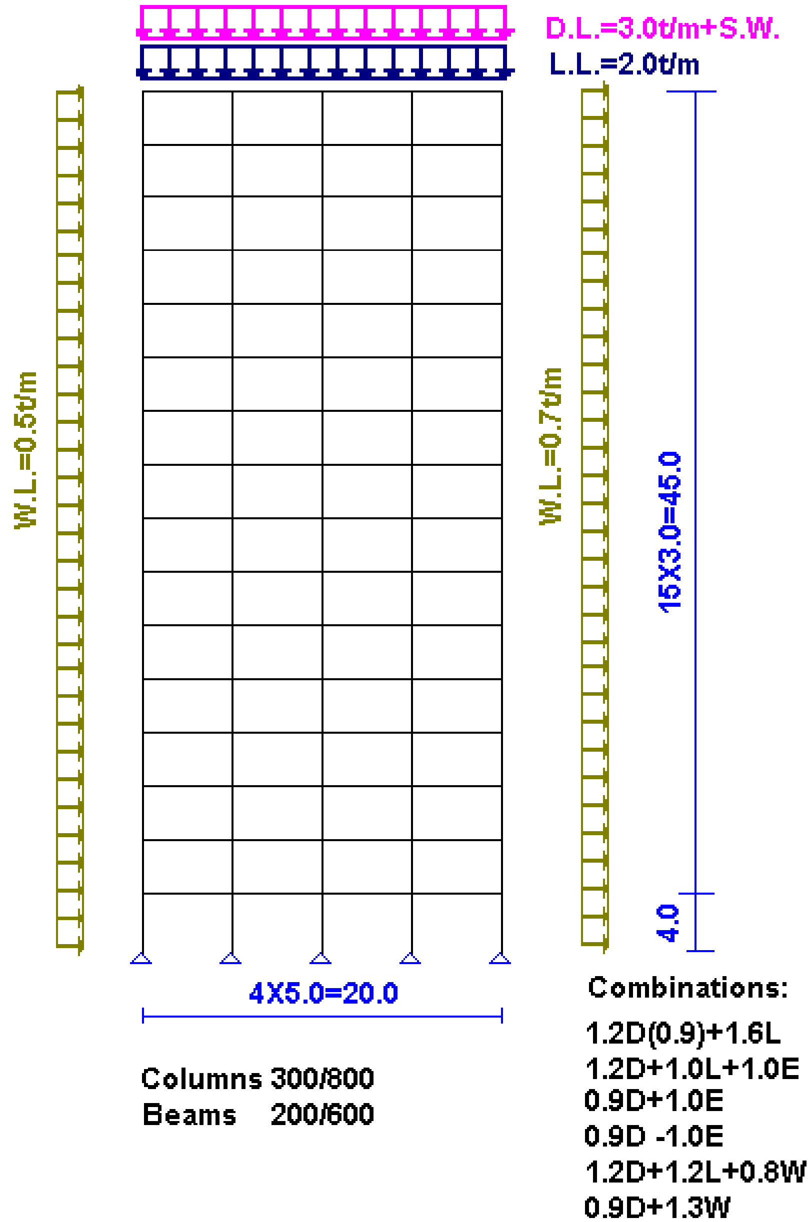
- Click the  button

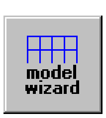
The program superimposes the preliminary mesh (0.25 x 0.25) on the area defined by the contour; select “Continue”. The elements as displayed in the Figure are generated.

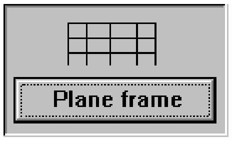
** This page is deliberately blank **

9 Dynamic Analysis

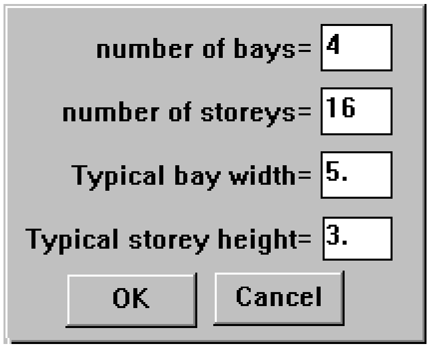
Calculate the seismic loading cases for the following model and add them to the *STRAP* static results.



- Preliminary Menu:
  - Enter the model title
  - Set the model type to **Plane frame**
  - Click 

- Click 

- Define the plane frame parameters:



Type in the number of bays

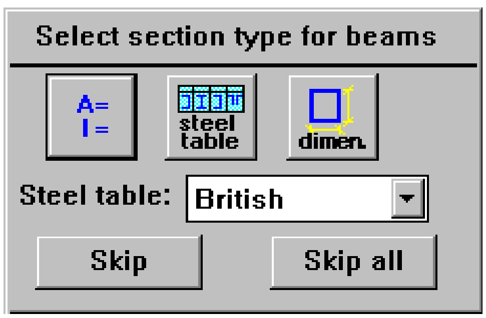
Type in the number of storeys

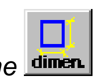
Type in the bay width

Type in the storey height

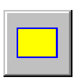
Click the  *button*

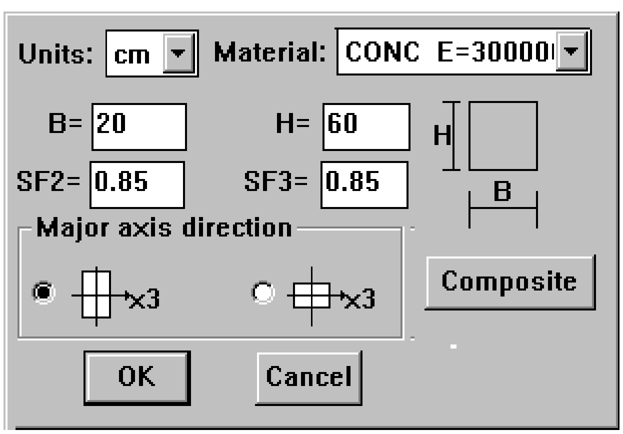
- Define the section type for beams:



Click the  *icon.*

Select section shape:

- Click the  icon.
- Enter the dimensions as follows :

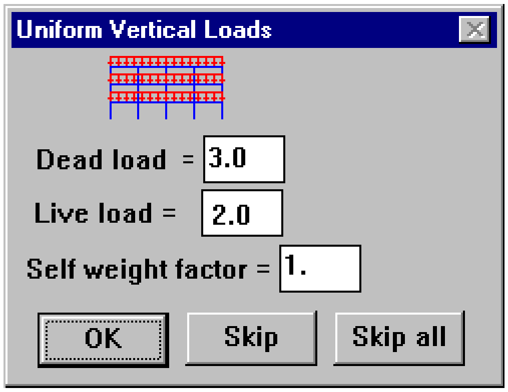


Click the  *button*

Similarly define the property for columns as a rectangle section with dimensions **H = 80; B = 30**.

Define Dead, Live and Wind loads and load factors for combination.

Dead and Live load:



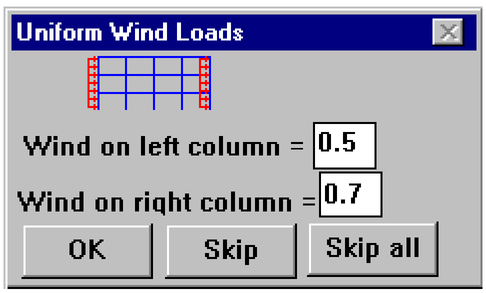
Type in the dead load value

Type in the live load value

Type in the self weight factor

Click the OK *button*

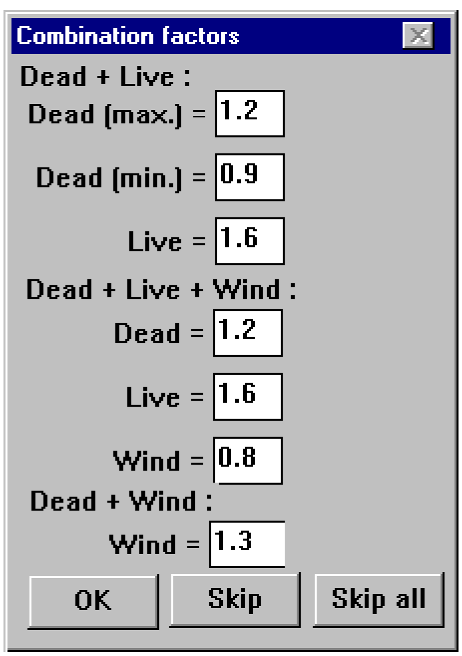
Define Wind loads:



Type in the wind loads

Click the OK *button*

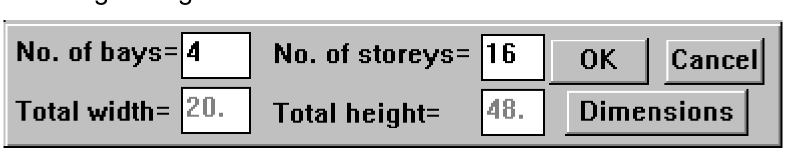
Define combination factors:



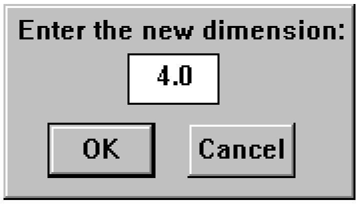
Enter the load factors

Click the OK *button*

The program now creates the model according to the parameters and displays it on the screen with the following Dialog Box at the bottom:

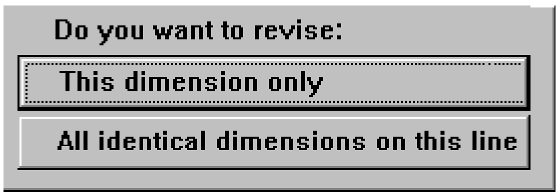


- To revise the bottom storey height to 4.0m, click the **Dimension** button.
- Move the cursor to the vertical dimension line so that the dimension of the first storey is highlighted with the rectangular blip; click the mouse.



Enter the new dimension

Click the OK *button*



*Select **Revise this dimension only***

- Press **Esc** (right click the mouse) to end the revision of dimensions.
- Click the OK button to leave the Model Wizard.

The program now enters the regular geometry module.

- click the loads icon

- **Loads**

All loads were defined in the Model Wizard. Click the revise icon to review and check the loads.

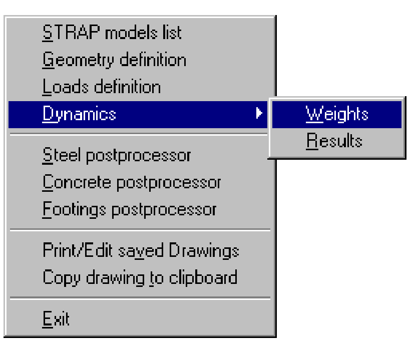
- Click the solve icon on the Side menu to solve the model.

*Click the **No** button*

After completing the solution, the program enters the results module of program.

- **Results**

Display graphic and tabular results for the static load cases. To continue to Dynamic Analysis select **Files** in the menu bar.

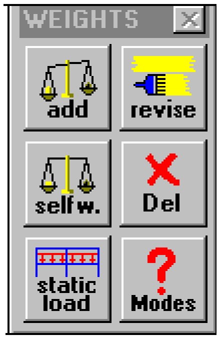


*Select **Dynamics** the pull-down menu and click on **Weights**.*

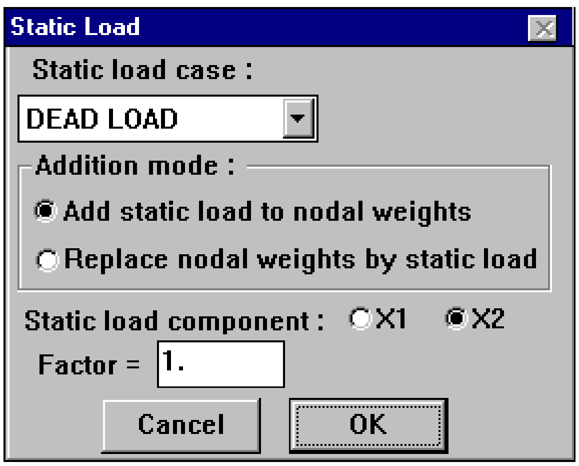
- **Dynamic analysis**

Define the nodal weights.

The program calculates the natural frequencies and the corresponding mode shapes. Assume that the dynamic nodal weight **= 1.0 x Dead loads + 0.2 x Live load** Retrieve the loads from the static load cases:



Click **static load**



Select **Dead Load**

Select **Add static load**

Select **X2**

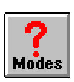
Define factor **= 1.0**

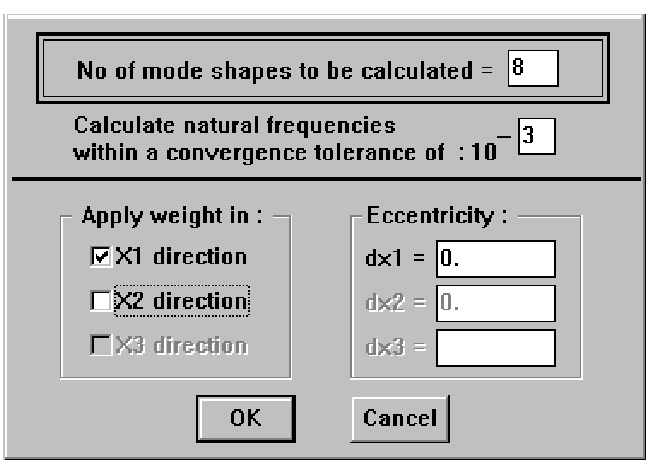
Click  *button*

Similarly add the **Live Load** to the static load with the Factor = 0.2.

Nodal weights will be automatically assigned to the nodes in the model.

Solve natural frequencies and mode shapes:

- click the  icon



Specify ***No of mode shapes***

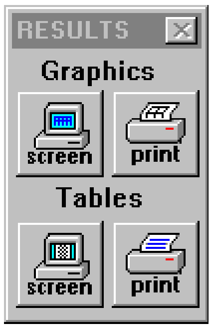
Apply weights in global **X1 direction**

Click  *button*

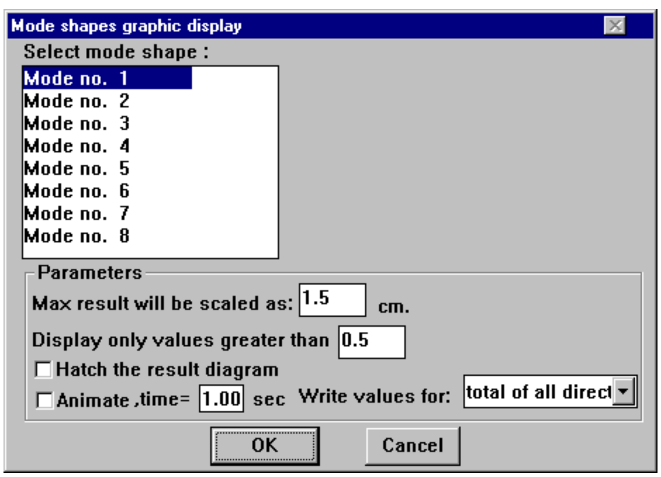
- Select **File** in the Menu bar.
- Select **Solve the Model** in pull down menu. The program solves for the natural frequencies and the corresponding mode shapes.

Click the **No** *button*

- Display the results of the mode shape analysis:

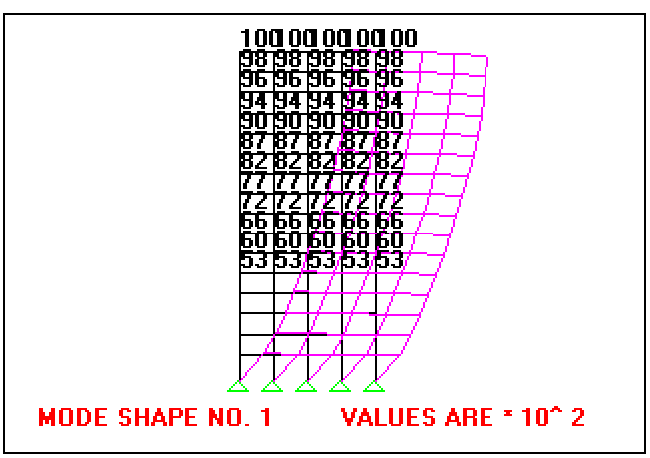


Select **Graphics - Screen** *to display graphic results.*
Select **Tables - Screen** *to display tabular results*



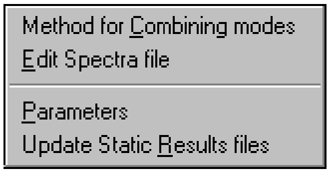
Select **Mode no. 1** *to display*

The program displays the normalized mode shape (maximum deflection = 1.0):



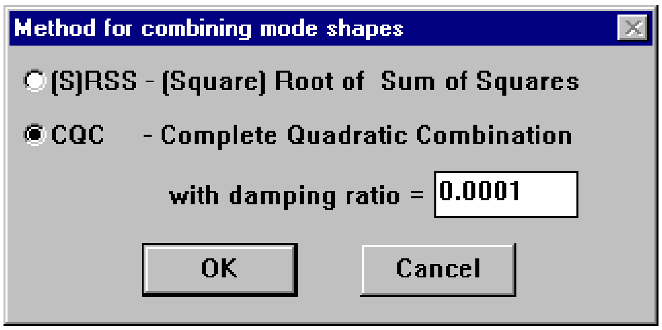
- **Seismic analysis**

  - Select **Seismic analysis** in the menu bar.



← *Select* **Method for Combining modes**

The program calculates the response that for each mode separately and combines them according to a formula that accounts for the fact that when one mode achieves its maximum response, the other modes are less than their individual maximum.



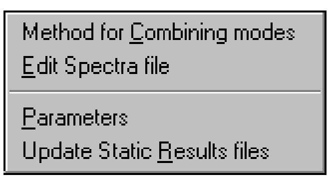
← *Select* **CQC**

← *Click the* OK button

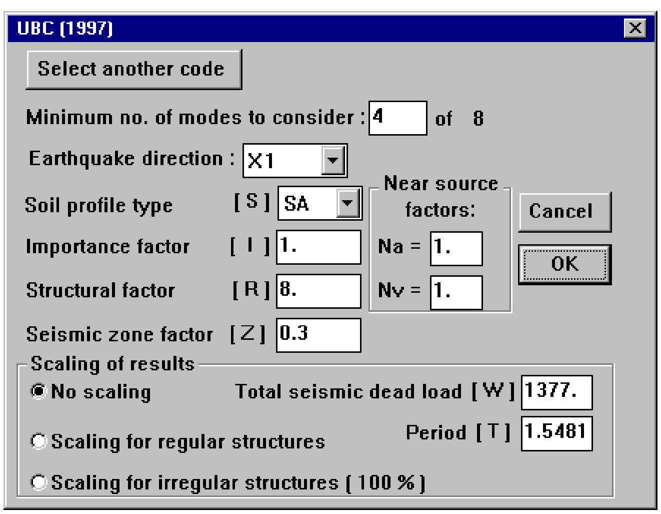
Define the **Code** parameters:

All seismic design codes modify the analysis results by factors that represent the framing method, soil type, importance, etc.

Refer to the Code and the program Help/manual for a detailed explanation on the factors used in the following Dialog boxes:



← *Select* **Parameters** *in pull down menu*



← *Specify the calculation code*

← *Type in minimum number of modes*

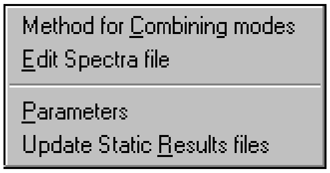
← *Select earthquake global direction* **X1**

← *Click the* OK *button*

← *Select* **No Scaling**

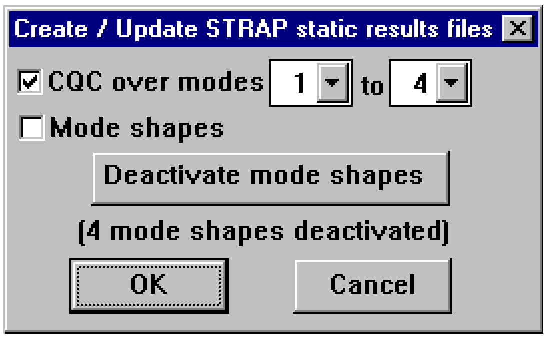
- **Create equivalent static load cases**

  - Select **Seismic Analysis** from the Menu bar.



Select **Update Static Result**

Write the dynamic results to the static results file.



Select **CQC over modes**

Select the range **1** *to* **4** *of modes to be included in the calculation*

Click the 

button

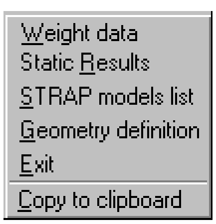
The program updates two **STRAP** files:

- Results file: each dynamic load cases is appended to the file
- Applied forces file: in order that the number of load cases in this file will correspond to the number of load cases in the results file, “zero” load cases are appended to the end of the file.

- **Static results**

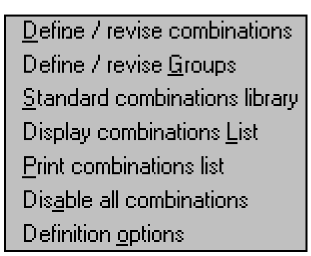
Display static results for dynamic load cases.

- Click **File** in the menu bar.

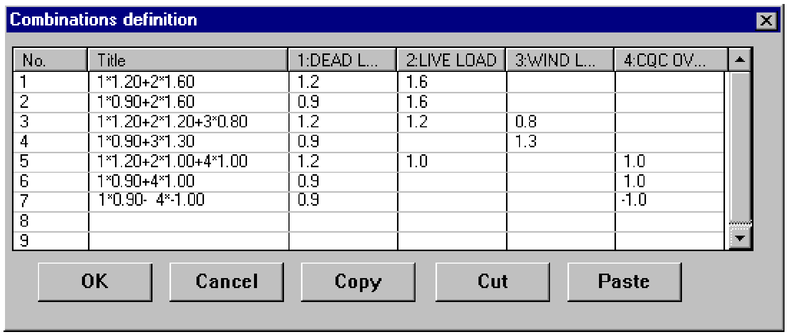


Select **Static Results** *in the pull down menu*

- Click **Combination** in the menu bar



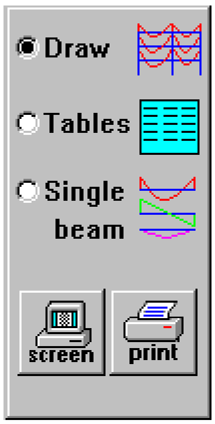
Select **Define/Revise combination**



Enter/revise the combination factors

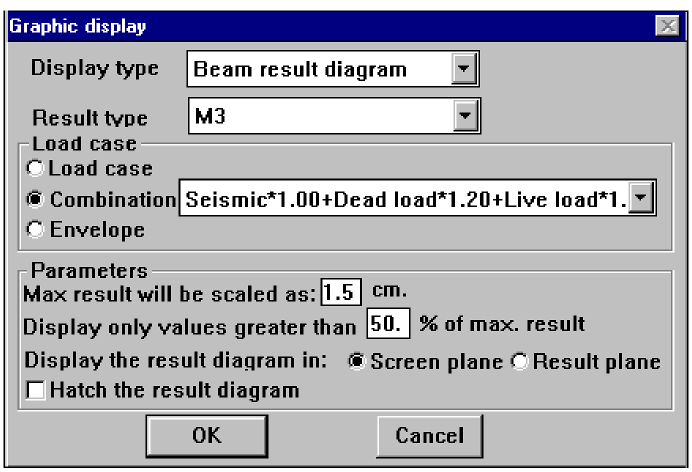
Click the OK *button*

Display the results:



Click **Draw**

Click **screen**



Select a **Display type**

Select a **Result type**

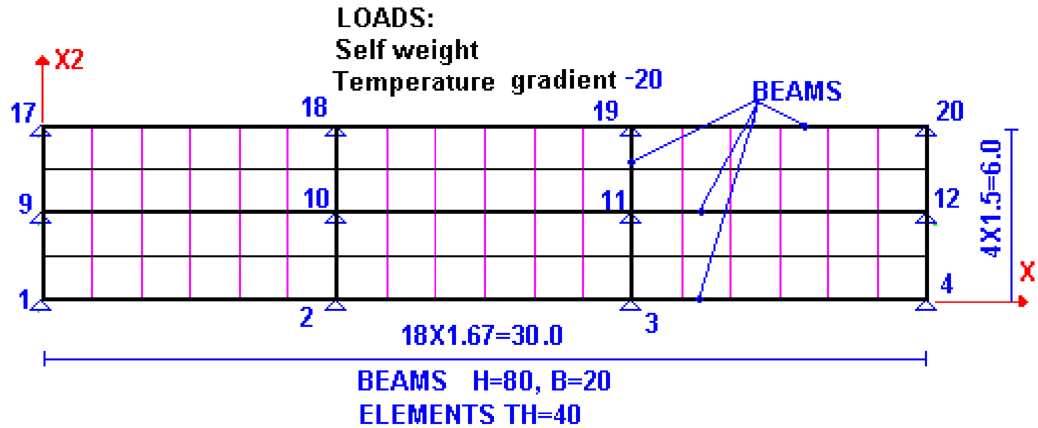
Click the OK *button*

The dynamic analysis has been completed.

** This page is deliberately blank **

10 Bridge Analysis

Use the Bridge Postprocessor to calculate the max-min results for the following two lane bridge:

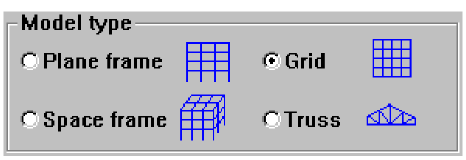


- The self-weight and temperature gradient loads will be defined in the regular STRAP loading module.
- The results corresponding to the vehicle loading patterns will be calculated in the Bridge Postprocessor.
- All results will be combined in the **Results** module.

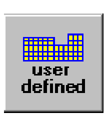
Define a regular *STRAP* model of the bridge consisting of beams and plate elements, loaded at this stage by self weight and temperature gradient load and solve the model for these loads.

- **Main Menu**
  - select **Files** in the menu bar
  - select **New model** in the pull-down menu
  - enter the model title

- **Preliminary Menu**:

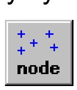
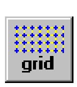


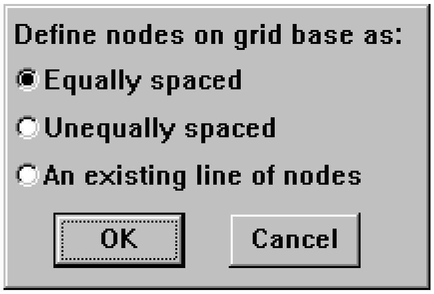
Set the model type to **Plane grid**

  - Click  to proceed to geometry

- **Nodes**:

  Define only the nodes of the beam grid using the **Grid** command; the remaining nodes will be generated automatically by the Element **Mesh** command.

  - click the  icon
  - click the  icon

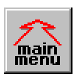


← *Select* **Equally spaced**

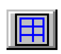
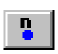
← *and click the* OK *button*

- Move the crosshair to **X1** = **0.0, X2** = **0.0** and click the mouse.
- Move the crosshair to **X1** = **30.0, X2** = **0.0** and click the mouse.
- Specify: **3** segments.

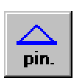
Similarly define the nodes parallel to X2:

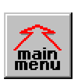
- Select **Equally spaced**
- Move the crosshair to **X1** = **30, X2** = **6.0** and click the mouse.
- Specify: **4**segments
- click OK
- Click the  icon.

- **Restraints**

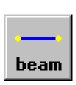
- to display the full drawing, click  in the menu bar
- click  in the menu bar to display the node numbers

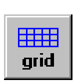
- Click the  icon.

- Click the  icon.

- Select **Individual nodes**
- Move the crosshair adjacent to node 1,2,3,4,9,10,11,12,17,18,19,20 so that the nodes are highlighted with the rectangular blip; click the mouse. After the node 20 has been selected, click the mouse again without moving the crosshair.
- Click the  icon.

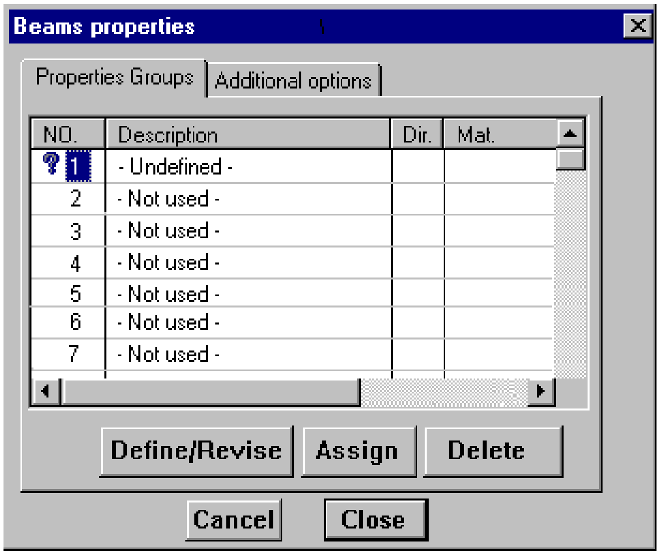
- **Beams:** Define all beams with a grid command –

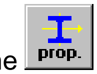
- Click the  icon

- Click the  icon

- Select Node 1 as the start node of the base line:
- Select Node 4 as the end node of the base line
- Select Node 20 as the end node of the height line.

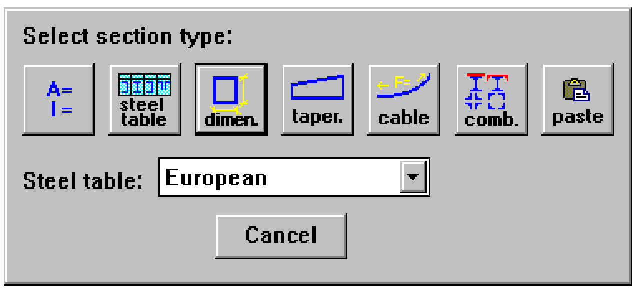
Properties:

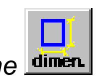


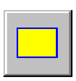
- Click the  icon.

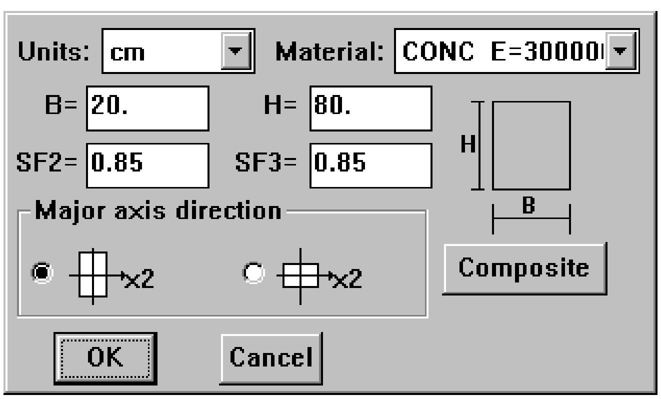
Click property no. 1

Click the ***Define/Revise*** *button*



Click the  *icon*

- Click the  icon.
- Enter the dimensions, as follows:



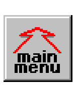
Select ***Conc***

Define ***B,H***

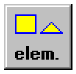
Define ***SF2***

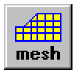
Define ***x2*** *as the major axis*

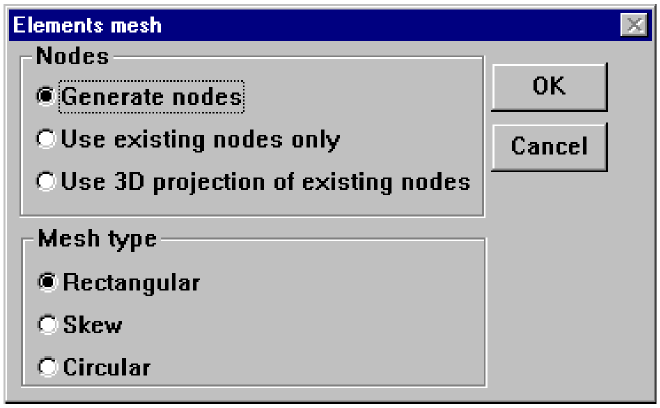
Click the  *button*

- Click the  button to complete the property definition.
- Click the  icon.

- **Elements**

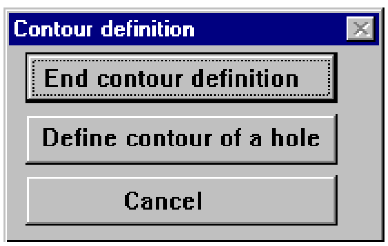
  - click the  icon

  - click the  icon



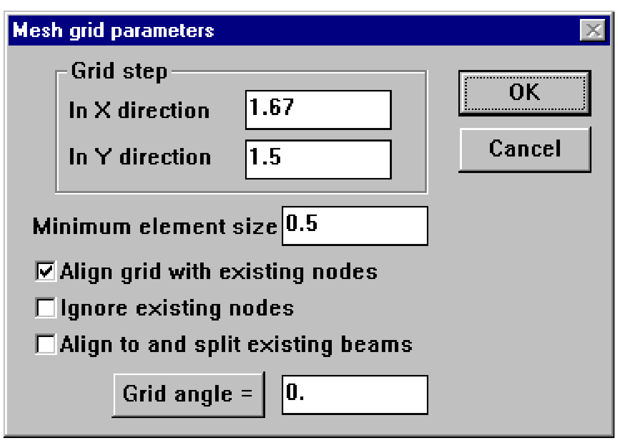
← *Select* **Generate nodes**

  - and click the OK button
  - Define the contour by selecting nodes 1, 4, 20, 17 and 1 again to close the contour.



← *Select* **End contour definition**

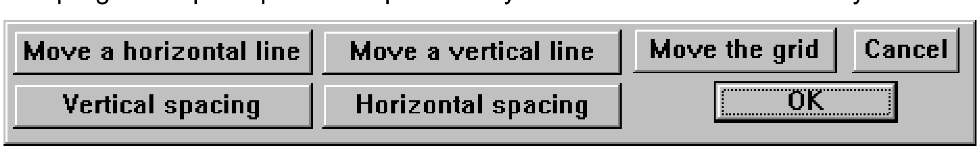
  - Arrange the mesh parameters menu as follows:



← *Set the* **Grid step** *in both directions*

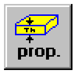
← Click the OK *button*

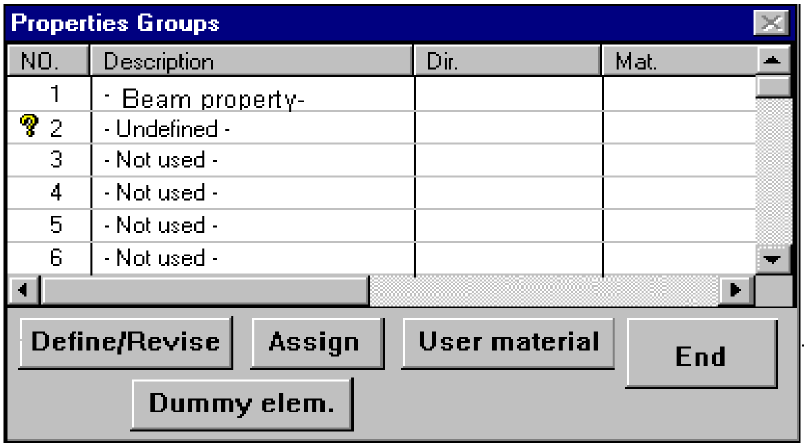
  - The program superimposes the preliminary mesh on the area defined by the contour.



  - Click the OK button.
  - Click the OK button when the program asks **Is the mesh OK?**

Define the element thickness:

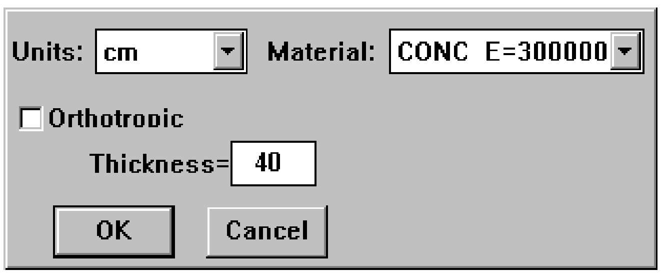
- click the  icon.



*Select **2 -Undefined-** (assigned by default to all elements)*

Click the Define/Revise button.

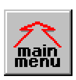
- Enter the property values:



Select **Concrete**

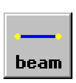
Type in **Thickness = 40**

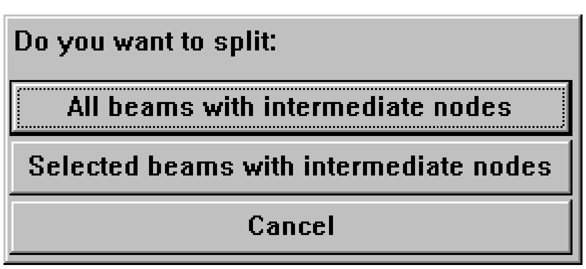
and click the OK *button*

- Click the End button. In the **Properties Groups** dialog box
- click the  icon.

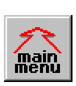
- **Beams**:

Note that the beams are not connected to the elements at the new nodes between the supports.Use the Beam **Split** option to divide up the beams

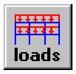
- Click the  icon
- Click the  icon



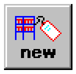
Click **all beams**

- click the  icon.

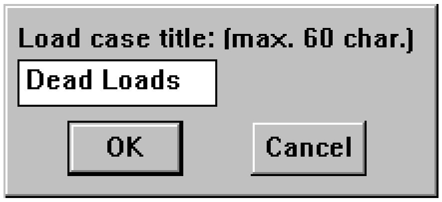
- **Loads**:

Click the  icon.

Define the self-weight and temperature loads in separate load cases:

- Click the  icon.

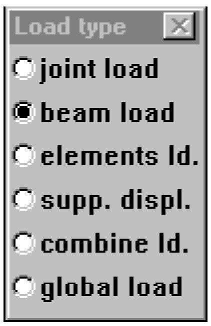
- Define the load case title:



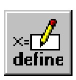
Type in the title

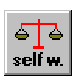
and click the  *button*

- Select the load type:

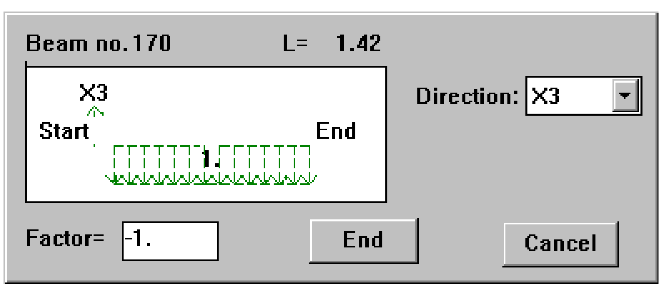


Set the **Load type** menu to **beam load**

and click the  icon.

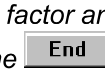
- Click the  icon

- Click **Select all beams**
- Define the load:

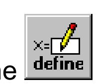


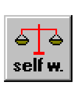
Set the **Direction** *to* **X3**

Set the factor and

Click the  *button*

Define the self-weight for the elements:

- set the **Load type** menu to  and click the  icon.

- click the  icon and set the **Factor** = -1.
- click **Select all elements**

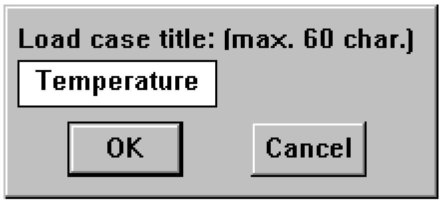
- Click the 

icon.

Define the temperature loads in Load case 2:

- Click the 

icon.
- Define the load case title:

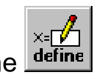


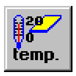
Type in the title

and click the 

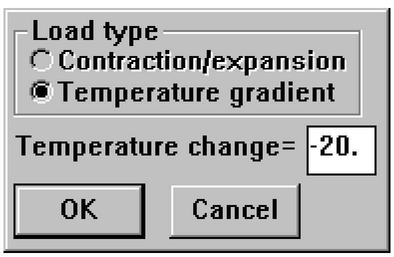
button

- set the **Load type** menu to 

and click the 

icon.
- click the 

icon.



Select ***gradient***

Type in the gradient value

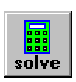
and click the 

button

- click **Select all elements**
- Click the 

icon.

• **Solve**

- Click the 

icon on the Side menu to solve the model.

Click the ***No*** *button*

- Click **File** in the menu bar.
- Click **STRAP Models list** to return to Main menu.
- Select **Design** in the menu bar and **Bridge module** in the pull-down menu.

- **Bridge Module-general**

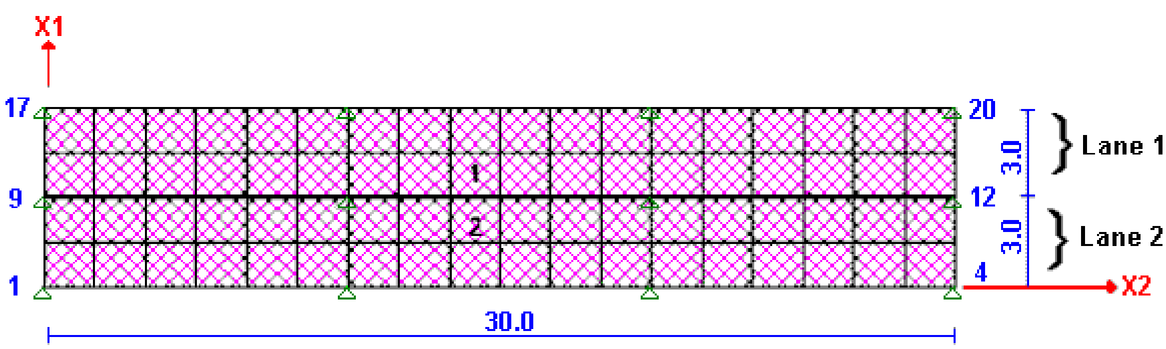
The bridge module automatically calculates the critical loading pattern that generates the max/min results for any result type at any point on the bridge.

The procedure is as follows:

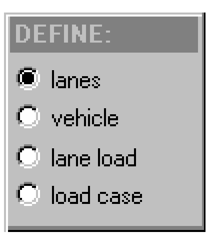
- Divide the bridge into lanes and then divide each lane into strips perpendicular to the axis of the lane.
- Solve the model: the program automatically applies a unit load to each strip in a separate load case (no. of load cases = no. of strips). The program uses the results of those cases to calculate worst case effects by means of superposition.
- Define lane loads: specify the vehicle loads and the distributed loads required by the codes on each of the lanes.
- Create load cases: tell the program how to arrange the various lane loads to create the design load cases.
- Transfer results to *STRAP*: append a load case to the *STRAP* results files that contains an envelope of the maximum results for vehicle load.
- Create combinations of the vehicle results and the self-weight and temperature loads.

- **Bridge Module- Lanes**

The bridge consists of 2 parallel lanes, each 3.0 meters wide.

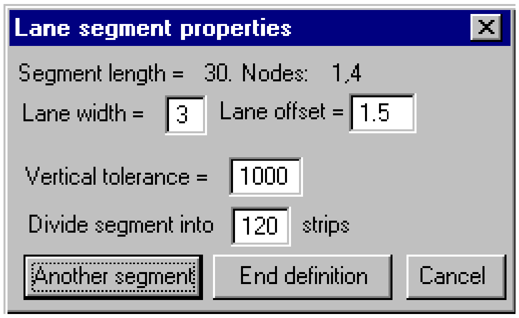


The lanes are defined by specifying the center line axis and width.



Select **lanes**

- Click the  icon to define lane 1.
- Click Node 1 to specify the start of lane.
- Click Node 4 to specify the end of lane.



Type in lane **Width** *and the center line* ***Offset*** *(the offset of the lane center from the line connecting start and end nodes*)

Type in the number of strips

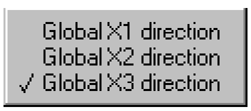
Click ***End definition***

- Similarly define lane 2 between nodes 9 and 12.

• **Load distribution**

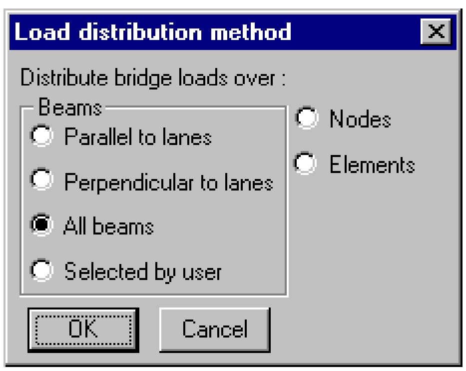
The loads may be applied to nodes, elements or selected beams.

- Select **Options** in the menu bar.
- Select **Load direction** to specify the Global direction of applied loads.



Select **Global X3 direction**

- Select **Loads distribution** in the pull down menu.



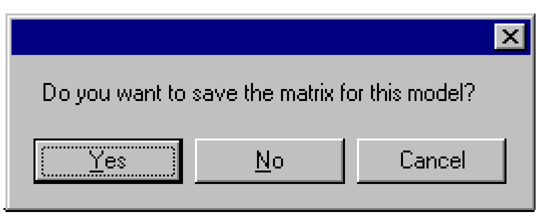
Click **All beams**

Click the OK *button*

• **Solve the model**

STRAP will create one load case for each strip in the model (a unit load is applied to the strip); 240 load cases will be solved in this model.

- Select **File** in the menu bar and click **Solve** in the pull down menu.



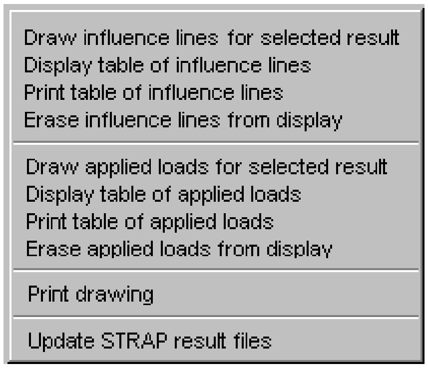
Click the ***No*** *button*

The program returns to the bridge module after completing the solution.

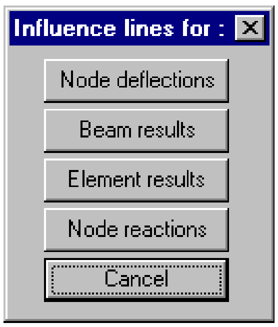
• **Display influence lines**:

Display influence lines for any *STRAP* beam ,element or node for any result type.
For example, display the influence line for **M2** moment at the center of beam 111.

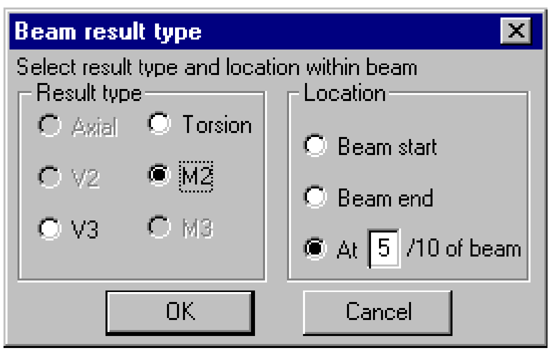
- Select **Results** in the menu bar:



Select **Draw influence lines**



Select **Beam results**



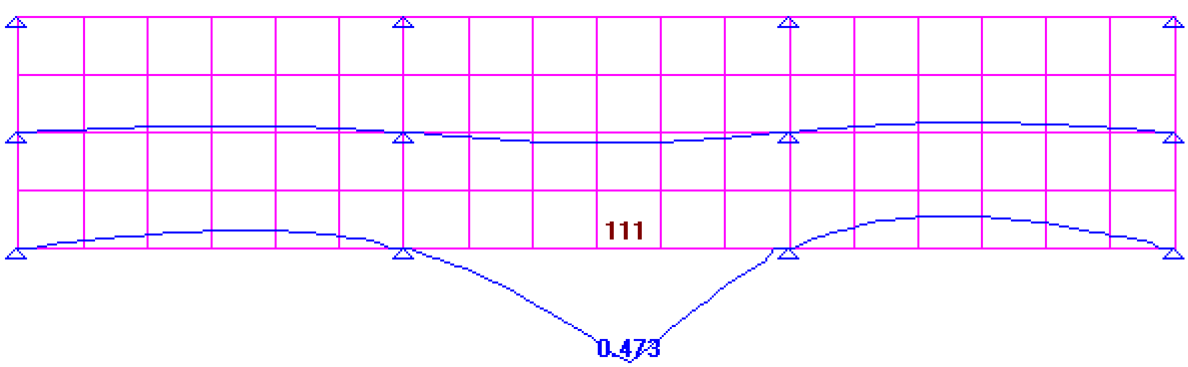
Select **Result type**

Click the 

button

- To select beam 111, highlight it with the ■ and click the left button of the mouse.

The program displays the following influence line:

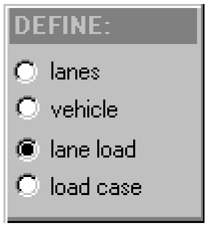


INFLUENCE LINES,BEAM111, 0.5 OF SPAN , M2

- **Define lane loads.**

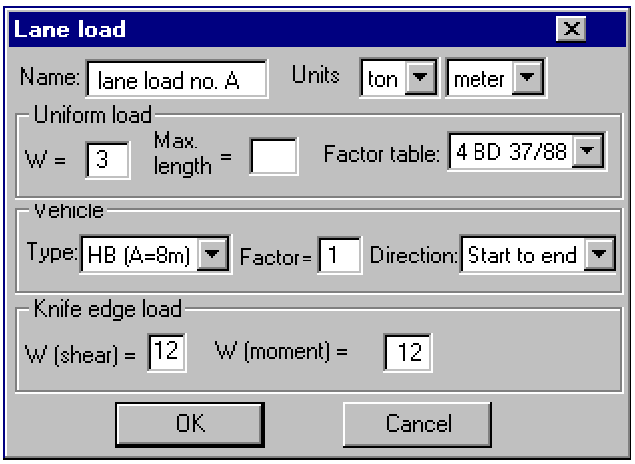
  Specify the vehicle loads applied to the lanes.

The following lane loads may be defined: uniform, vehicle, knife-edge loads.



Select **lane load**

- Click the  icon.



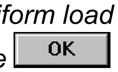
Type in **Load case** *name*

Type in **Uniform load** *value*

select load reduction (vs. length) factor table

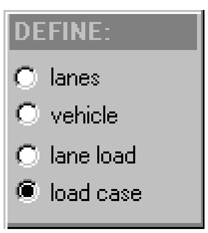
Select **Vehicle** *group*

Type in **Knife edge load** *value (the program applies the knife-edge load as a uniform load to a strip)*

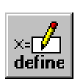
Click the  *button*

- **Define load cases**

Assign a lane load to each lane to define a load case. The program applies the load to each of the strips along the length of the lanes; only those loads that contribute to the requested maximum/minimum result are used.



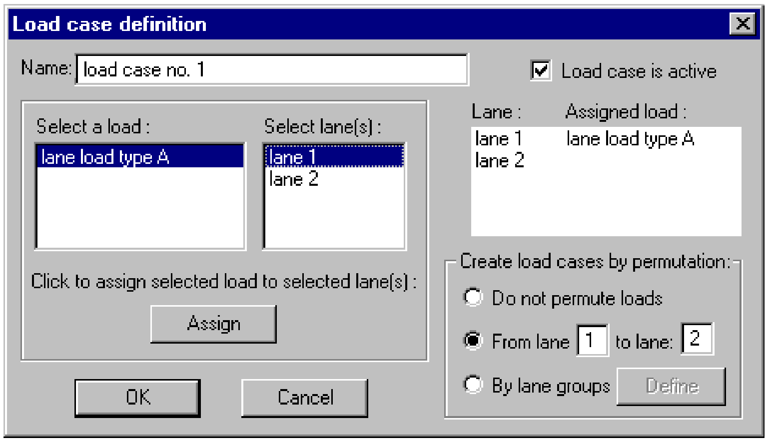
Select **load case**

- Click the  icon.

Define three load cases:
- lane load A is applied to lane 1 only.
- lane load A is applied to lane 2 only.
- lane load A (defined in the previous step) is applied to both lane 1 and lane 2.

- Define load case 1 and 2:

  Generate all possible permutations of the lane loads on the selected lanes. The program will create load cases by interchanging the lane loads.

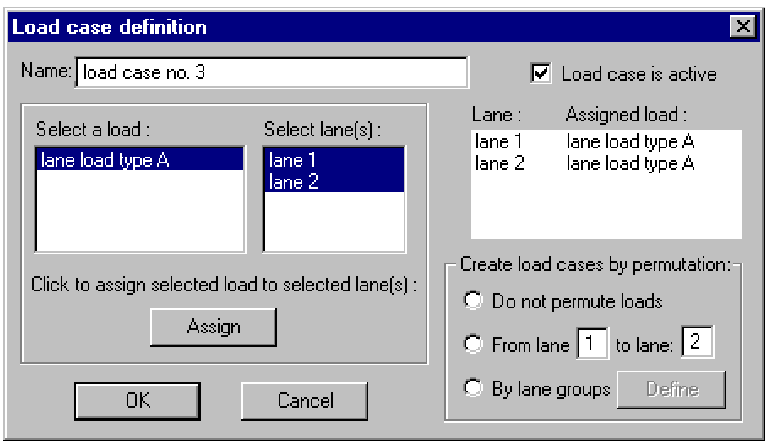


Select lane load A
Select lane 1

Specify all possible permutations on lanes 1 and 2

Click the OK *button*

- Define load case no. 3.



Select lane load A
Select lane 1 and 2

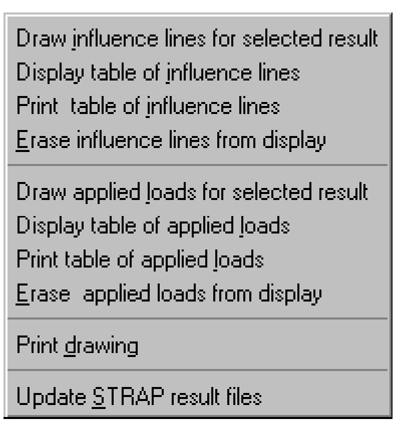
Click the OK *button*

- **Display selected results**

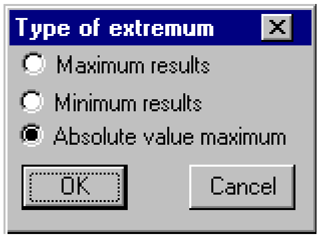
  The program displays the loads applied to the various strips that are required to generate the maximum/minimum result.

  For example, display the maximum absolute value for **Mx** result in the center of element 58.

- Select **Results** in the menu bar.

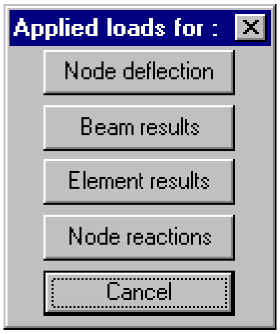


Select **Draw applied loads for selected results**

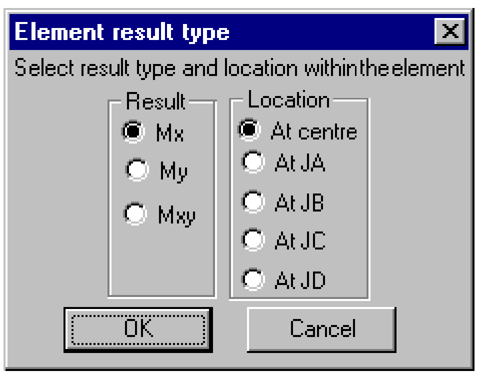


Select **Absolute value maximum**

Click the  *button*



Select **Element results**

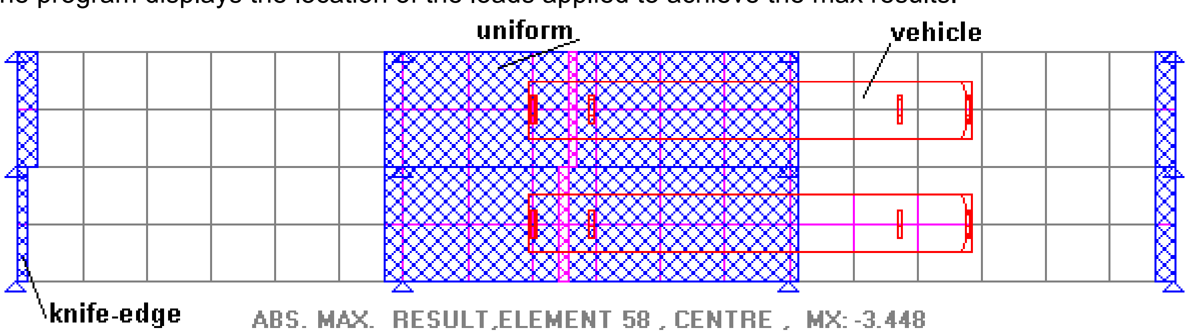


Select **Result type**

Click the  *button*

- To select element 58 click the mouse.

The program displays the location of the loads applied to achieve the max results.

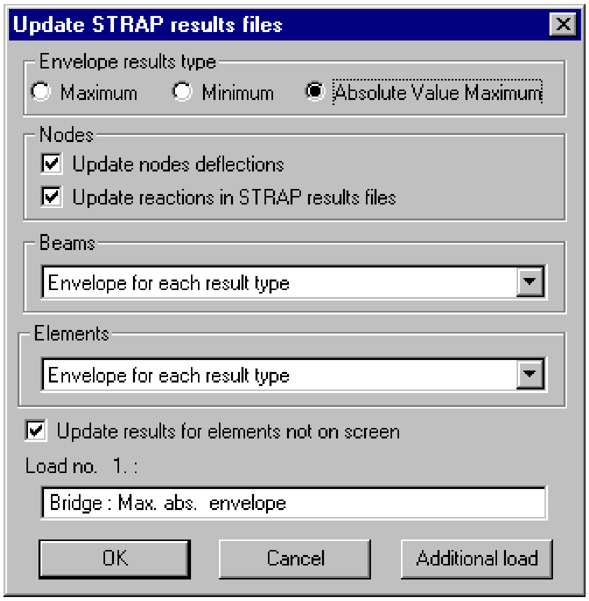


- **Transfer results to STRAP**

  Create a single *STRAP* load case consisting of a maximum result envelope for all results types (moment, shear, etc.). Note that you can create separate load cases for each result type.

  The program will search for the critical load pattern for each result type for each node, element and beam ($1/10^{th}$ of span).The results will be transferred to the *STRAP* results file.

  - Select **Results** in the menu bar.
  - select **Update STRAP** result file.

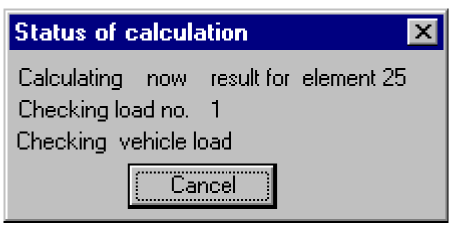


Select envelope result type

Type in name

Click the OK *button*

The program runs about one MILLION comparisons for each beam in this model for computing the envelope and takes a few minutes to complete the calculation.

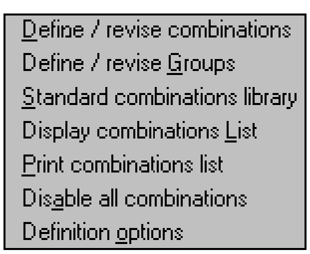


- Select **File** in the menu bar.
- Select **STRAP results** to return to the *STRAP* results screen.

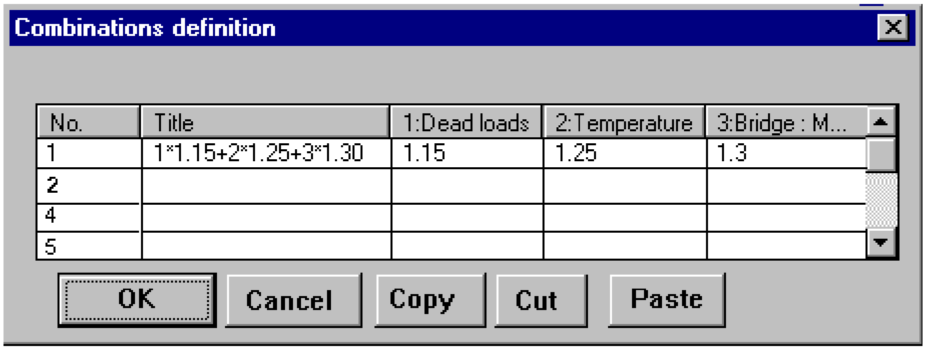
- **Combinations**

The model also contains other load types e.g. self-weight and temperature (already defined). Define factored loading combinations of these loads with the vehicle loads envelope.

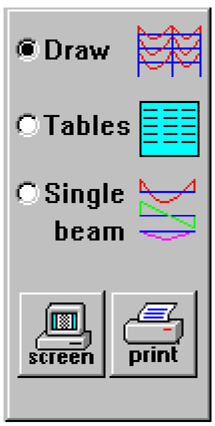
- Select **Combinations** in the menu bar.



Define/revise combinations

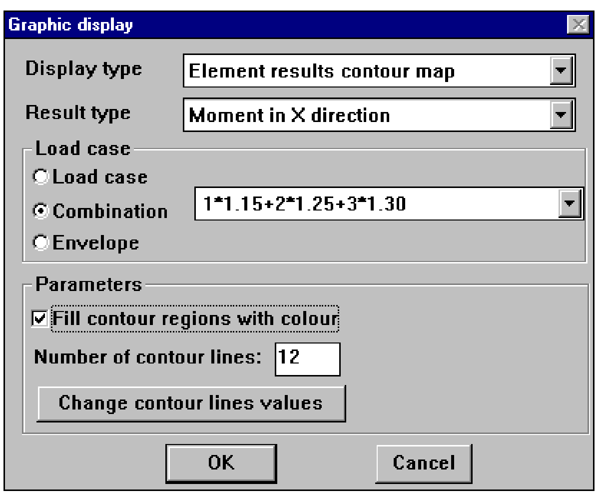


- Select the load case and enter the load factor.
- Click the OK button to exit from table.
- To display graphic results:



← *select* **Draw** and

← click the **screen** icon.

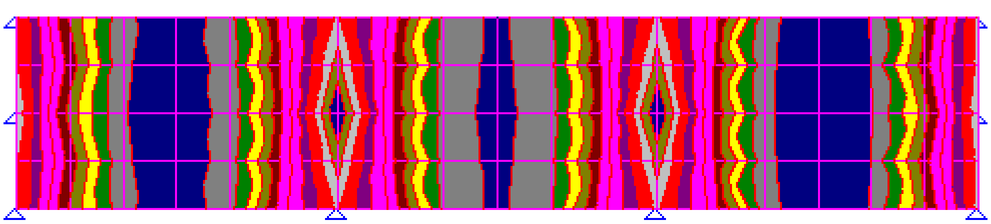


← *Select* **Display type**

← *Select* **Result type**

← *Select* **Load combination**

← *Click the* OK *button*

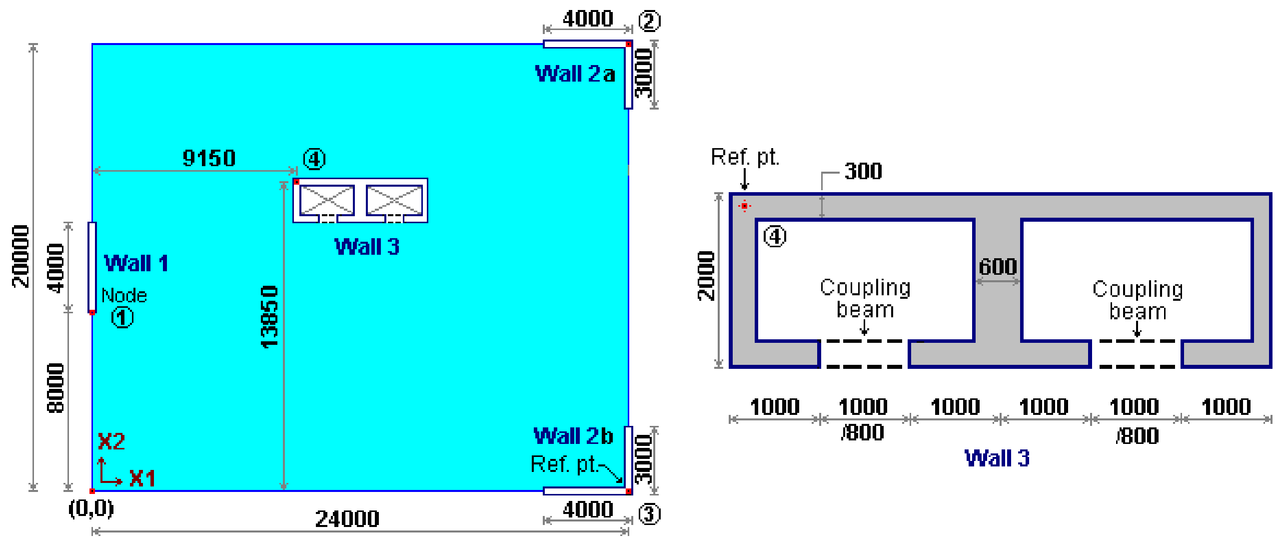


MX CONTOUR LINES COMB. NO. 1 1*1.15+2*1.25+3*1.30

** This page is deliberately blank **

11 Space Frame with Wall Elements

Define the geometry of the following 10-storey building that includes four walls extending the full height of the structure:

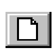


Note:

- this example explains how to define the geometry; for dynamic analysis and interpretation of the results, refer to the user's manual.
- The model is used for dynamic analysis only:
  - the slabs are defined with dummy elements; the in-plane rigidity is provided by rigid links.
  - columns are not defined because their contribution to the lateral stiffness is negligible.

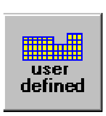
To define the model:

- **Main Menu**

- click the **New model**  icon

- **Preliminary Menu**

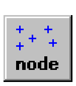
- Set the **Model Type** to  **Space frame**

- click the 

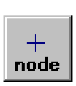
 button

- **Geometry**

Define the wall reference nodes 1, 2, 3, 4 at ground level (X3=0.0):

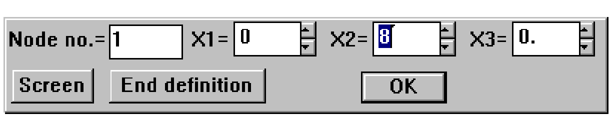
- click the 

 icon

- click the 

 icon

- Node 1: Move the crosshair to **X1 = 0.0, X2 = 8.0, X3 = 0.0** and click the mouse.



- Node 2: Move the crosshair to **X1 = 24.0, X2 = 20.0, X3 = 0.0** and click the mouse.
- Node 3: Move the crosshair to **X1 = 24.0, X2 = 0.0, X3 = 0.0** and click the mouse.
- Node 4: Move the crosshair to **X1 = 9.15, X2 = 13.85, X3 = 0.0** and click the mouse.
- Click **End definition**

Define the supports:

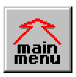
- Click the 

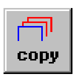
icon.
- Click the 

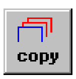
icon.
- Click the 

icon.
- Click the **Select all nodes** button

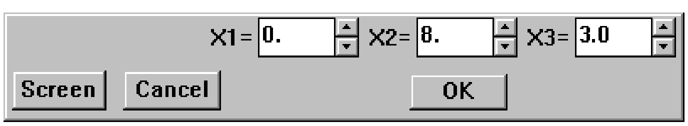
Copy the nodes to all 10 floors:

- Click the 

icon
- Click the 

icon
- Click the 

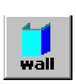
icon in the Copy options menu at the side of the screen
- Click the **Select all nodes** button
- select any of the 4 nodes as the reference node
- Click the **by coordinates** button
- Define the floor levels at 3.0 m intervals (X3): Press the left mouse button and do not release; move the  into the **X3** edit box (note that the coordinates will not change) and type **3.0**:



- set **Number of copies: 10** and click OK
- click the Isometric View icon  in the icon bar to display the entire model.

Define the wall sections:

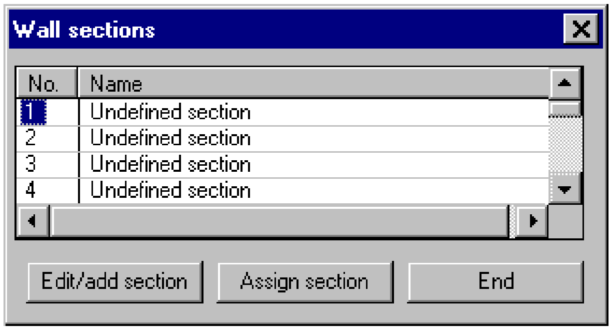
- Click the 

icon
- Click the 

icon
- Click the 

icon in the menu at the side of the screen

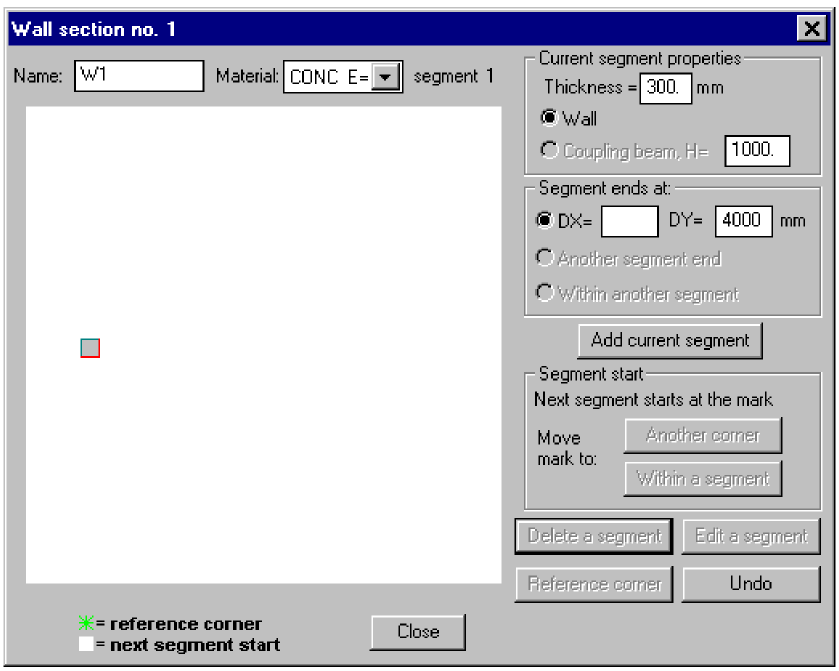
- Select the section:



← *Select section* **1**

← *Click* **Edit/add section**

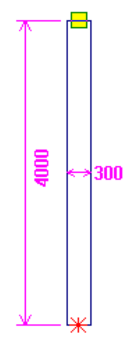
- Define the properties and dimensions of Wall 1; the wall consists of one segment only and we will define the segment end at DX=0, DY=400:



← *Specify thickness* = **300**

← *Specify* **DY = 4000**mm and press [Enter]

The segment is drawn in the center box as

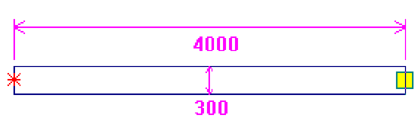


← *Click* **Close**

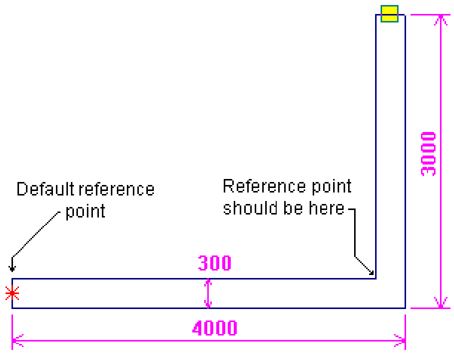
Define Wall 2 section:

- Click the  icon in the menu at the side of the screen

- Highlight section no . **2** and click **Edit/add section**
- The wall is composed of two segments; the first ends at DX=4000; DY=0. Set the Thickness = **300**, type in DX = **4000** and press [Enter]; the first segment is drawn in the center box:



- the second segment is offset DX=0, DY=3000 *from the end of the first segment*. The cursor is already in the DY= box; type 3000 and press [Enter]; the second segment is added to the display:



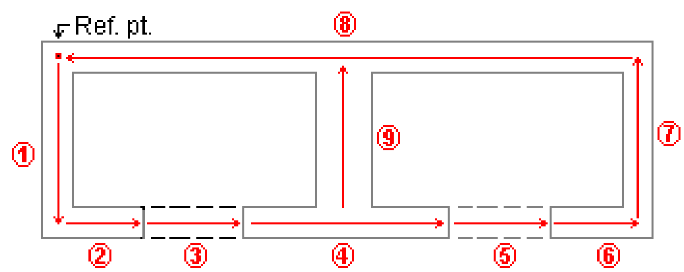
Note the “reference point” at the first corner denoted by the ✳. This is the point where the wall section will be attached to nodes in the structural model. Referring to the floor plan at the beginning of this example, it is apparent that Walls 2a/2b will be attached at the corner joining the two segments in the section.

To move the reference point, click **Reference corner**; move the cursor to the new location so that it is highlighted with the ■; click the mouse. The ✳ will now appear at the correct location.

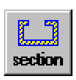
- Click **Close**

Define Wall 3 section:

- Wall 3 is created by defining nine segments in the following order:

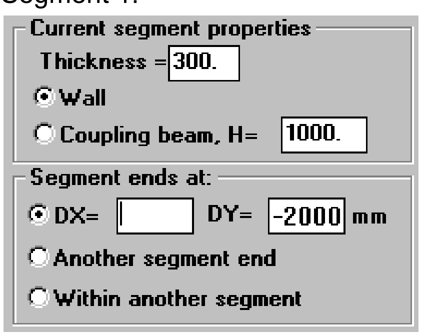


Note:
- segments 3 and 5 are coupling beams
- segment 9 bisects segments 4 and 8 and divides them each into two new segments.

- Click the 

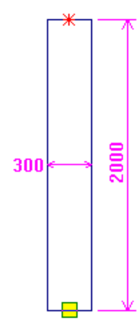
icon in the menu at the side of the screen
- Highlight section no . **3** and click **Edit/add section**

- Segment 1:



Specify thickness = **300**

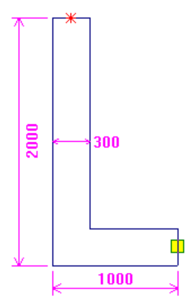
Specify **DY = -2000**mm and press [Enter]



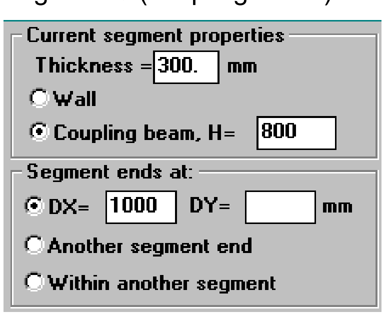
The segment is drawn in the center box:

- Segment 2:

  Specify **DX = 1000**mm and press [Enter]. The segment is drawn:

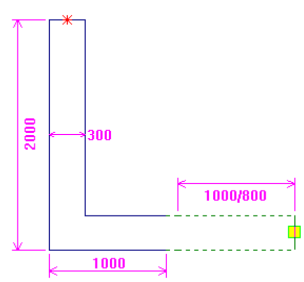


- Segment 3 (coupling beam):

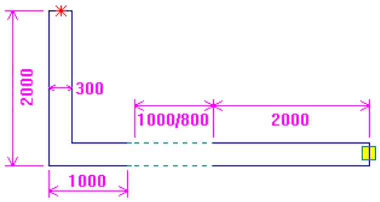


Specify ◉ **Coupling beam H= 800** mm

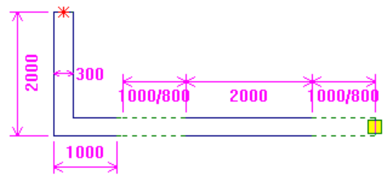
Specify **DX = 1000**mm and press [Enter]



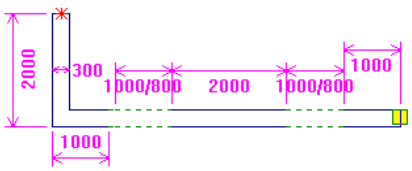
- Segment 4:
  Specify **Wall** and **DX = 2000**



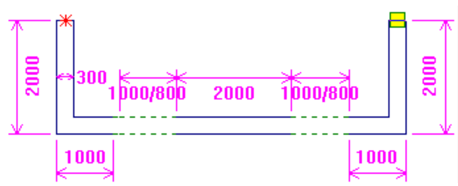
- Segment 5:
  Specify **Coupling beam** and **DX = 1000**



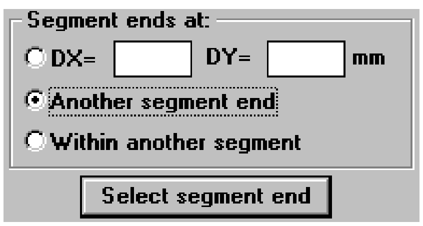
- Segment 6:
  Specify **Wall** and **DX = 1000**



- Segment 7:
  Specify **Wall** and **DY = 2000**



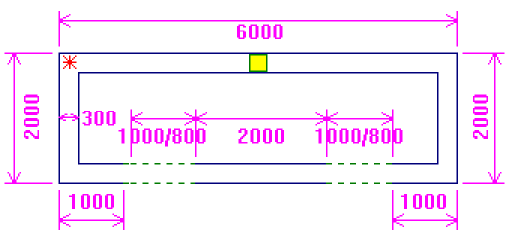
- Segment 8:
  This segment ends at the start of segment 1



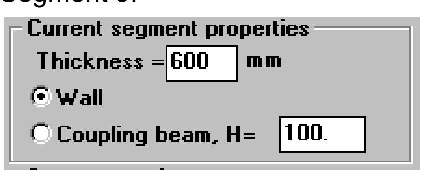
← *Specify* **Another segment end**

← *Click* **Select segment end**

move the to the start of Segment 1 so that it is highlighted with the ; click the mouse.

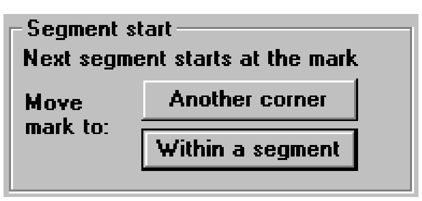


- Segment 9:

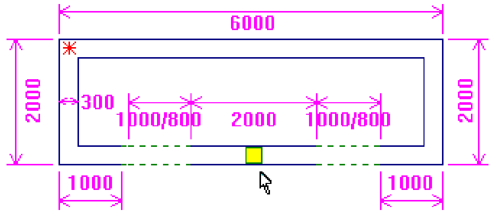


← *Specify* **Thickness = 600 mm**

This segment starts at the mid-point of segment 4 and ends at the mid-point of segment 8

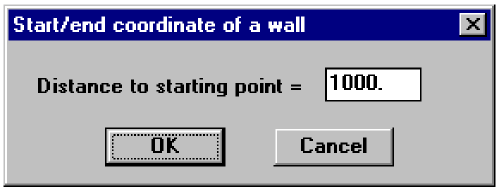


← *Click* **Within a segment**



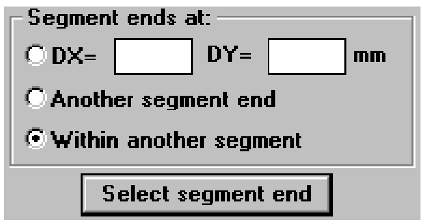
← *move the adjacent to segment 4 so that it is highlighted with the ■; click the mouse.*

Specify the location along segment 4:



← *Enter the distance from the start of the segment to the mid-point.*

Now define the end-point of the segment:



← *Specify* **Within another segment**

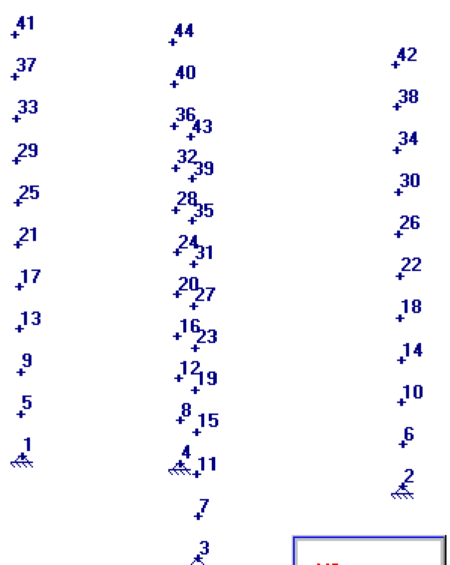
← *Click* **Select segment end**

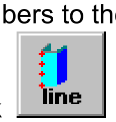
Move the adjacent to segment 8 so that it is highlighted with the ■; click the mouse. Specify the location of the end point of segment 9 at the mid-point of section 8, as explained above for the start point.

- Section 3 is now complete; Click **Close**.

Now we will attach the three wall sections to the model:

- click the node number icon in the icon bar to add the node numbers to the display:



- click 

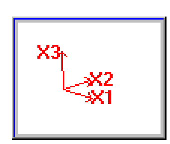
- select the wall section:

- attach section 1 to line 1-41:

  move the adjacent to node 1 so that it is highlighted with the ■; click the mouse; move the adjacent to node 41 so that it is highlighted with the ■; click the mouse;

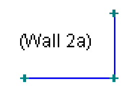
  The program adds the wall section to the line 1-41.

- attach section 2 to line 2-42:
  Specify **Sect =2** and select nodes 2 and 42. Note that the orientation of the wall is not correct. We will rotate it after the other three walls have been attached.

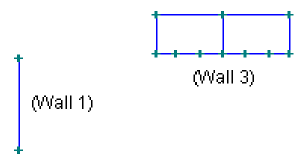


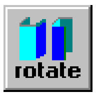
- attach section 2 to line 3-43:
  Specify **Sect =2** and select nodes 3 and 43
- attach section 3 to line 4-44:
  Specify **Sect =3** and select nodes 4 and 44
- click the node number icon, the Isometric View icon and the Restraints icon. The model is displayed on theX1-X2 without numbering or restraints:

  All of the walls are oriented correctly except **Wall2a** (check the coordinates of the wall corners by clicking on the data Data icon)

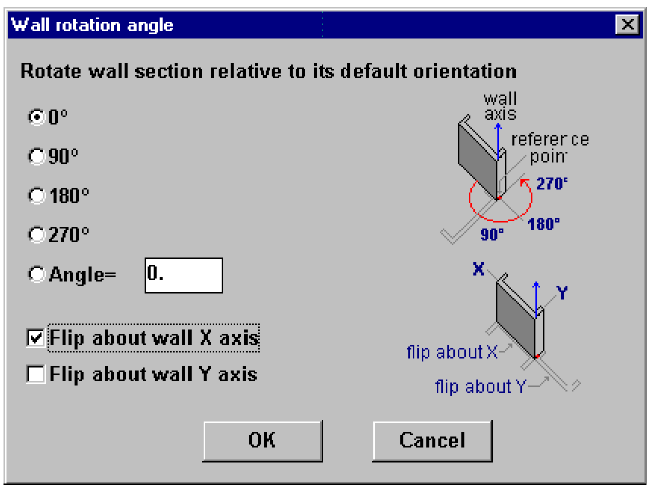


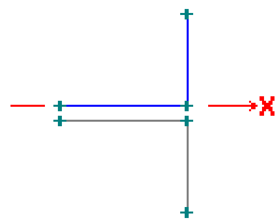
- Rotate Wall 2a:
  click the Isometric View icon



- click  icon

- the wall may be flipped or rotated:





Specify **Flip about X axis**

- Click **Select by window**
- Create a window around Wall 2a; the program flips it into its correct position.

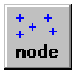
Define the floor slabs:

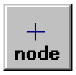
The floor slabs are modeled by dummy elements since the model will be solved only for dynamic loads. The elements are necessary in order to define the masses, but the size of the elements is not important.

We will define the slab at X3=3.00 and copy it to the other 10 levels.

- Click on **Remove** in the toolbar at the top of the screen
- Select **Limit display to a plane**
- Select any three nodes at X3 = +3.00
- click the Isometric View icon to display theX1-X2 plane

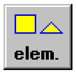
Define the nodes at the X1=0 corners of the slab

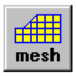
- click the 

icon
- click the 

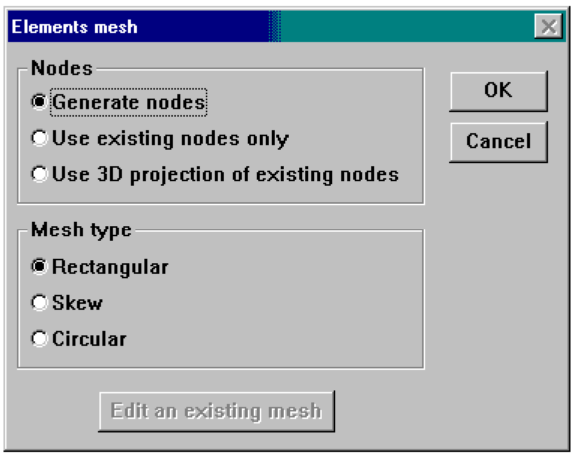
icon
- Type in the coordinates **X1 = 0.0, X2 = 0.0, X3 = 3.0** in the dialog box at the bottom of the screen
- Type in the coordinates **X1 = 0.0, X2 = 20.0, X3 = 3.0** in the dialog box

Define the element mesh:

- click the 

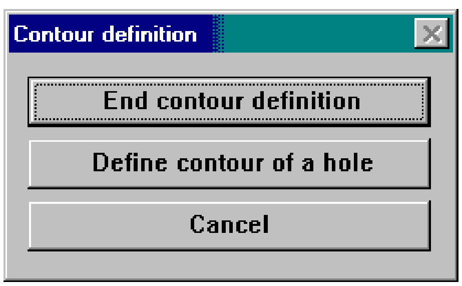
icon
- click the 

icon

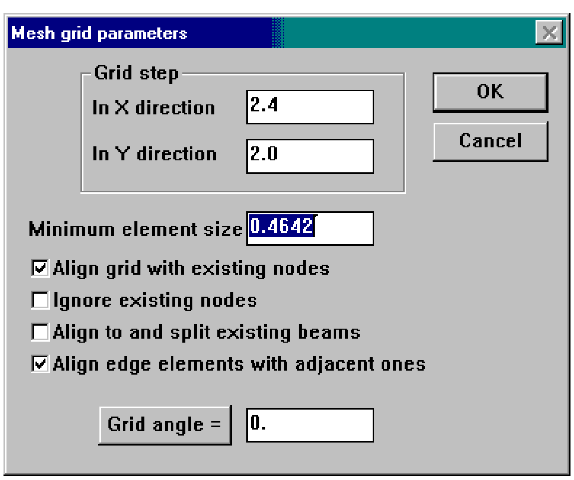


Click **OK**

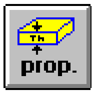
- Select the four corner nodes and end the selection by clicking the first corner again



Select **End contour definition**

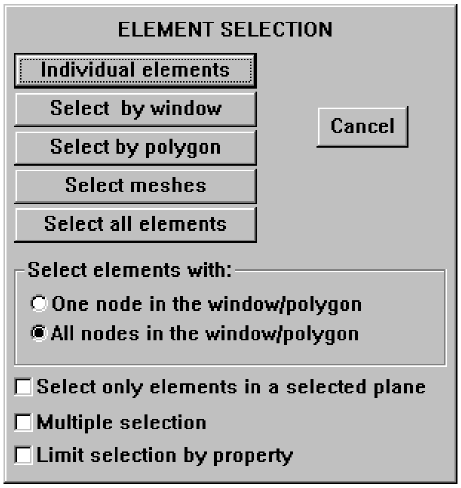


Specify Grid steps: X = 2.4, Y=2.0 *and click* OK

- click 

in the Elements menu

- click **Dummy element**



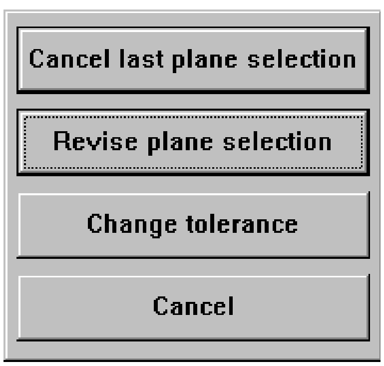
-

Select meshes

- select any two nodes in the mesh

- Click on **Remove** in the toolbar at the top of the screen

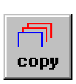
- Select **Limit display to a plane**



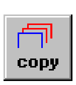
Cancel last plane selection

- click the Isometric View icon 

Copy the slab to the other nine levels:

- Click the 

icon in the main menu.

- Click the 

icon in the copy options menu

- Click the **By levels** button

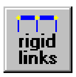
- Select level X3 as the height axis and select X3 = +3.00

- Select a reference mode: move the adjacent to the node in the left corner so that the node is highlighted with a ■; click the mouse.

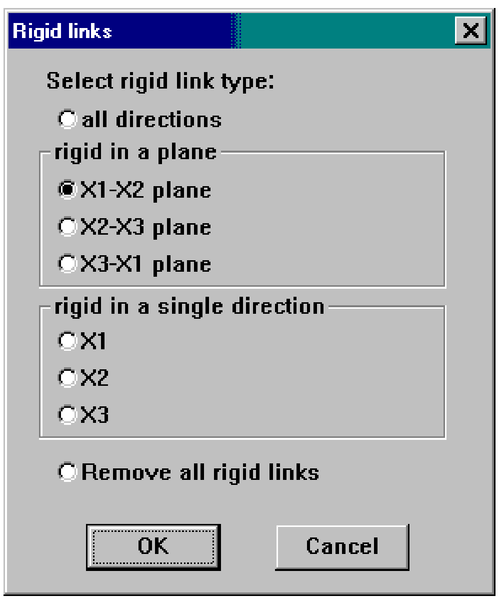
- Define the new location of the reference node:
- Click the **by coordinates** button
- Click and hold the left mouse button and drag the cursor into the dialog box at the bottom of the screen - the coordinate values in the box will not change as the cursor moves. Type the coordinates **X3=6.0**.
- specify **No. of copies = 9**
- Click OK to continue

Define Rigid links in the slab plane:

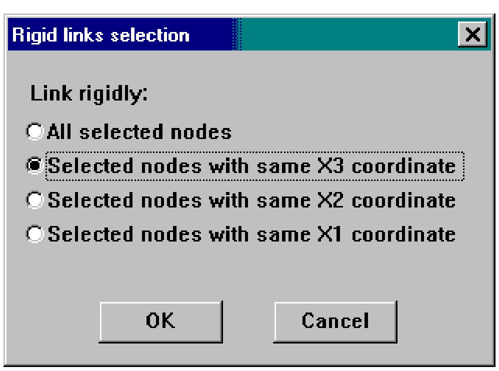
- Rotate the model so that it is displayed on the X1-X3 plane. Select **Rotate** in the toolbar, specify **X1-X2** and enter the rotation values **X1 = -90, X2 = 0, X3 = 0**
- Click the 

icon in the main menu.
- Click the 

icon in the Restraints options menu



Select **X1-X2 plane**

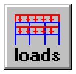


Select **Selected nodes with same X3 coordinates**

- Select the nodes in the entire model, except the support nodes at X3=0.0

The geometry of the model is now complete.

- **Loads**

- click the 

icon

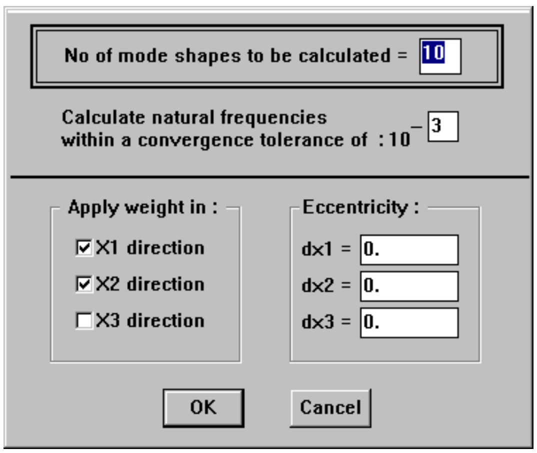
- apply the following loads to all slabs on all levels:
  Dead = 8.0 kN/m2
  Live = 1.5 kN/m2

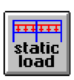
The mass used for the dynamic analysis will be based on these static loads.

- **Dynamic analysis**

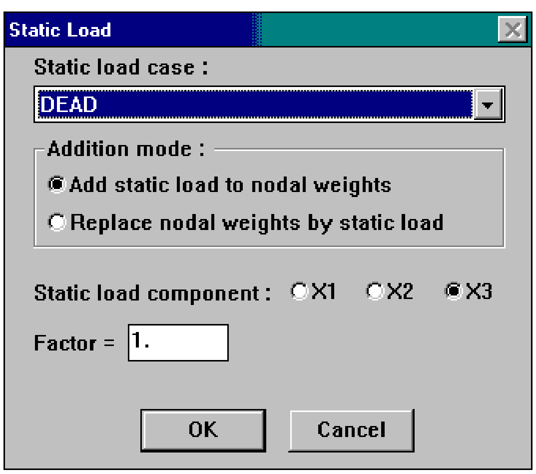
- return to the Main menu and model list
- click on **Dynamics** in the toolbar and **Weight data** in the pulldown menu.
- click the 

icon



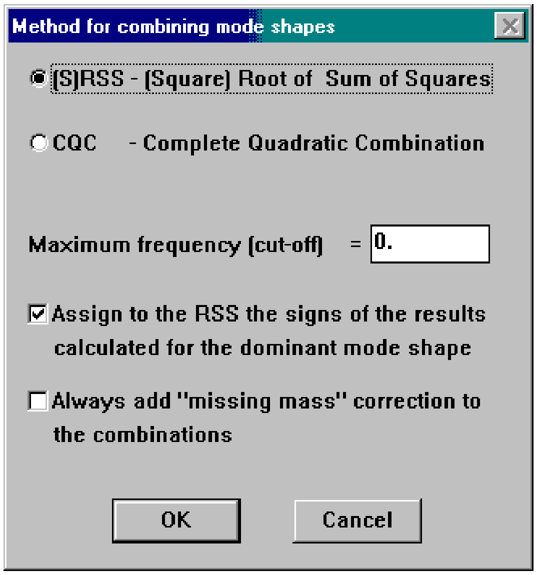
- click the 

icon

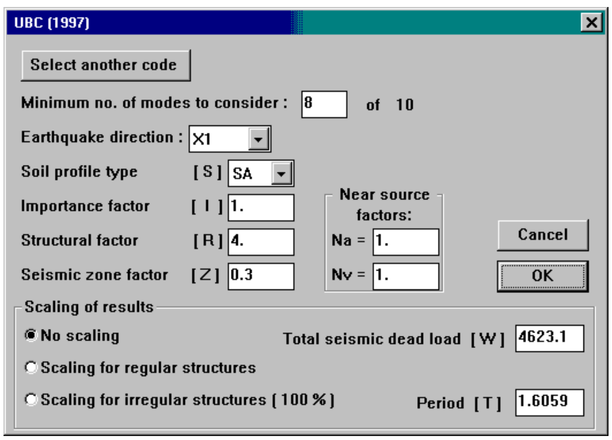


- Select **File** in the toolbar and **Solve the model** in the pulldown menu.

- Select **Seismic analysis** in the tool bar and **Method for combining modes** in the pulldown menu



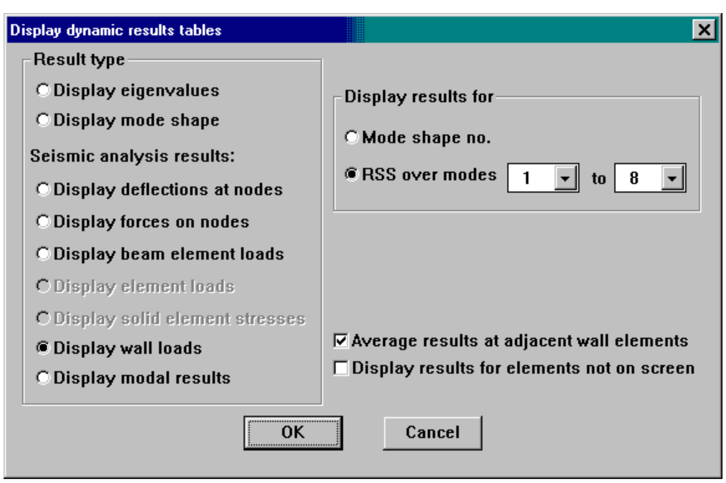
- Select **Seismic analysis** in the tool bar and **Parameters** in the pulldown menu



(If **UBC** is not displayed in the title of the box, click **Select another code** and select **UBC 1997**)

- click the **Tables** - 

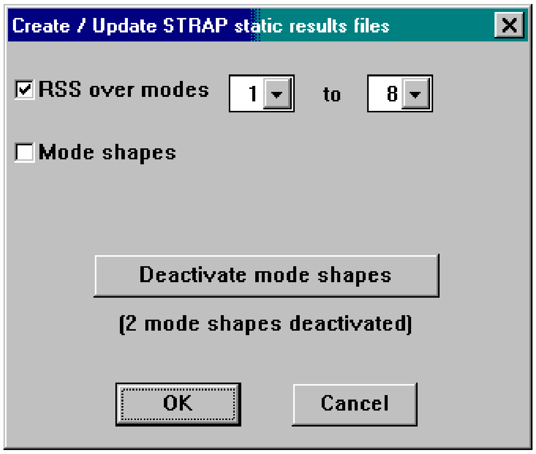
icon



The results for all wall segments are displayed on the screen.

Add the dynamic results to the static results file:

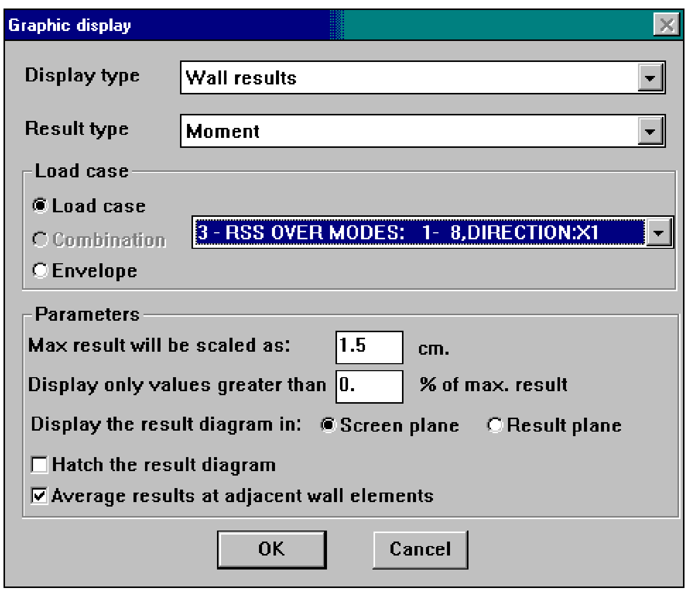
- Select **Seismic analysis** in the toolbar and **Update static results files** in the pulldown menu



Display the results in STRAP:

- Select **File** in the toolbar and **Static results** in the pulldown menu

- click the **Graphics** -  icon



The moments in the walls are displayed.

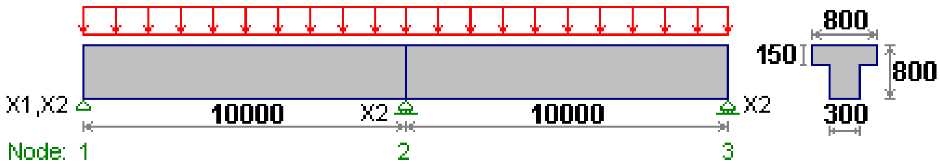
** This page is deliberately blank **

12 Prestressed beam

Define the following two-span continuous post-tensioned concrete beam:

STRAP:

- Geometry



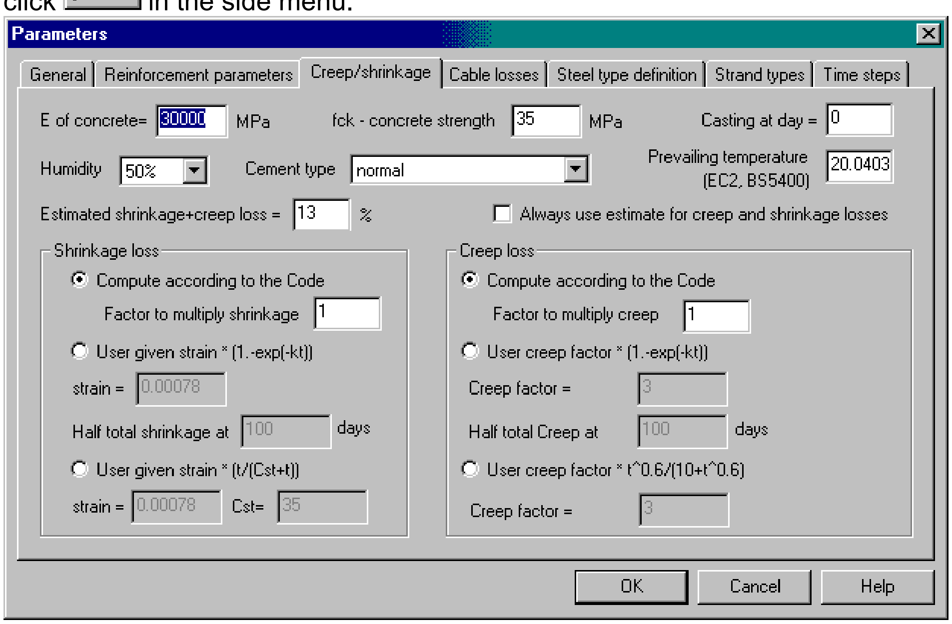
- Load cases
  1 - Self-weight
  2 - Dead load = -5.0 kN/m (service)
  3 - Live load = -7.5 kN/m (service)
- Solve the model
- Combinations
  1 - 1 x 1.00
  2 - 1 x 1.40 + 2 x 1.40 + 3 x 1.60
  The first combination corresponds to the "stage" where the prestressing is applied and is used to calculate service results (stresses, deflections). The second combination is used to calculate both service and factored results with total loads(ultimate moment, shear); the program ignores the load factors when calculating service values.
- Select **Files** - **STRAP model list** to return to the main menu.
- Select **Design** - **Posttensioning module**.

POSTTEN:

Refer also to the Help/Users Manual topics: "How to use the program" and "How to define cables".

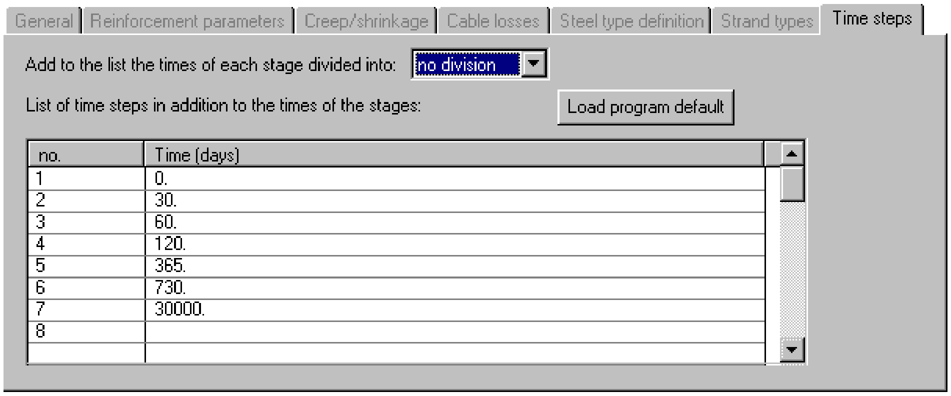
Specify the default parameters, e.g. design code, revise the loss parameters, etc.

- click  in the side menu.



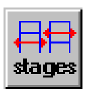
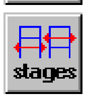
Select concrete grade ***35***

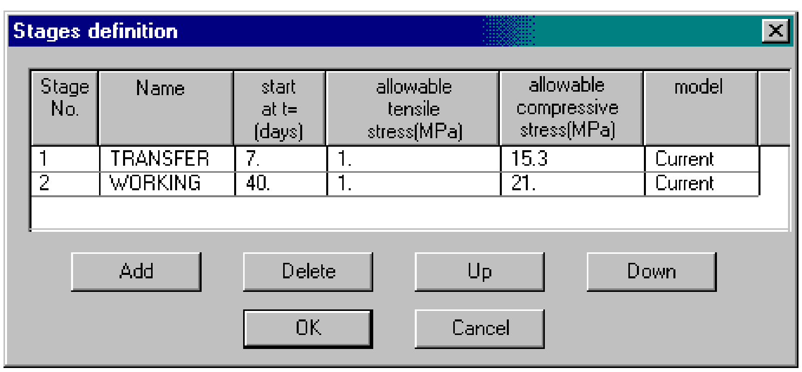
Define time steps (days) where the program calculates the prestress load, including losses. The resulting load is automatically applied at the start of the next time step.



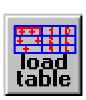
Note:
- the program default time-steps are usually sufficient
- additional times are defined in the "Stages" option.

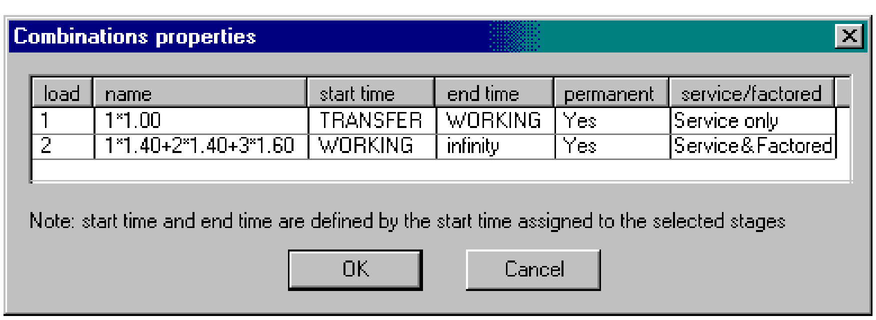
Define the design "stages". Stages are defined at dates where prestressing and/or loads are applied. The program calculates results at all stages and at the "time-step"dates.

- click  in the main side menu.
- click  in the stages side menu.
  The program checks stresses at each design stage. Define two stages:
  - "transfer"- the stage at which the prestressing is applied and only self-weight acts on the beam.
  - "working"- the stage at which all loads are applied.



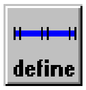
for each stage, specify:
- the allowable stresses
- the start day

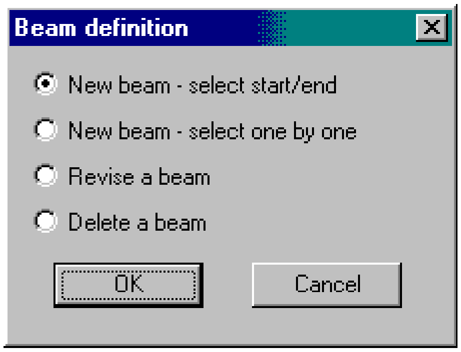
- click  in the main side menu.



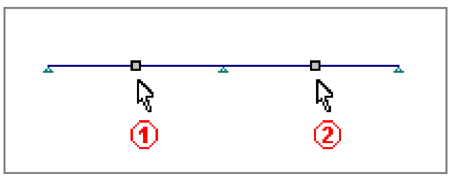
Specify:
- *time range where each combination is applied*
- *combination used for "service" calculations (stresses, deflections) or "factored" (ultimate moment, etc) or both.*

Define the beam:

- click  in the side menu.

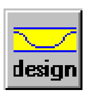
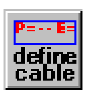


Select ◉ **New beam - select start/end**



Start: highlight member **1** *and click the mouse*
End: highlight member **2** *and click the mouse*

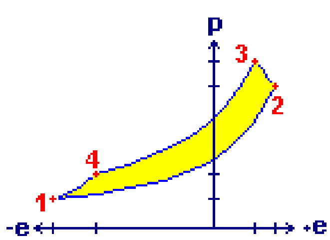
Define the prestressing:

- click  in the side menu.
- click  in the Design side menu to define the cables.

The program displays the “Magnel diagrams” at the start, middle and end of the beam. The stresses at the extreme fibres are limited to the Code values and are checked at every stage for maximum and minimum moments with the actual prestressing force (after losses) . This gives four limiting stress conditions:

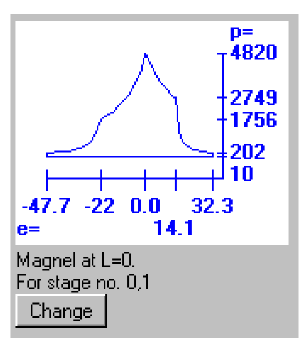
- minimum moment - top fibre	- maximum moment - top fibre
- minimum moment - bottom fibre	- maximum moment - bottom fibre

The equations are in the form $\sigma_e - \frac{P}{A} \pm \frac{Pey}{I} \leq \sigma_{all}$ where **se** is the relevant stress from external loads.



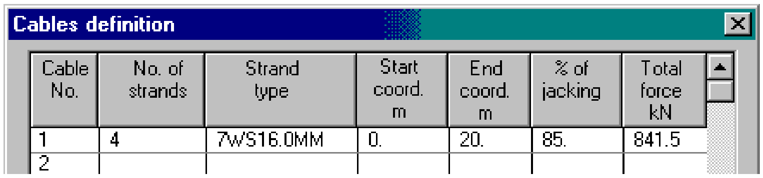
Plotting all of the possible solutions for each of the four conditions gives four intersecting lines: 1-2, 2-3, 3-4, 4-1 in the adjacent diagram; any combination of **P** and **e** within the enclosed area will result in acceptable stresses in the specific cross-section.

The program creates an ‘envelope’ of diagrams for all combinations, all stages and for all locations over a specified portion of the beam length. For example, at the start of the beam, the Magnel diagram is:



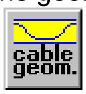
From examination of the allowable ranges of prestress force and eccentricity, try a cable with four 16 mm. strands.

- Enter the data in the first row of the table:



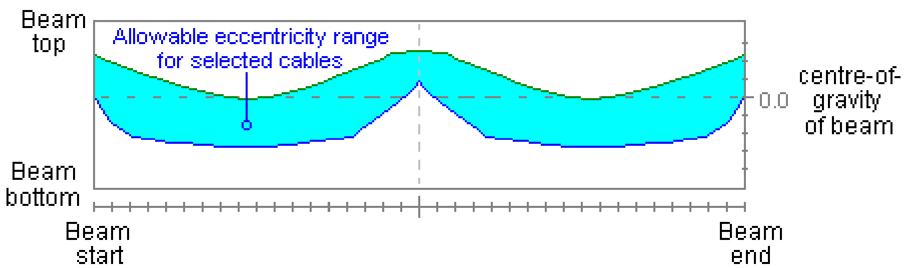
Cable No.	No. of strands	Strand type	Start coord. m	End coord. m	% of jacking	Total force kN
1	4	7WS16.0MM	0.	20.	85.	841.5
2						

Define the geometry for the cable:

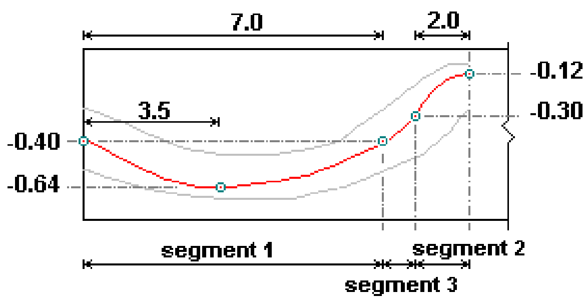
- click  in the side menu.

- click and highlight Cable no. 1 and click OK

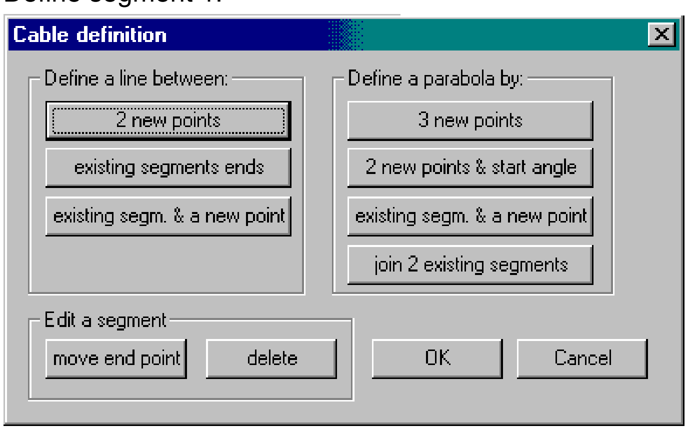
The program superimposes the eccentricity limits for these cables on the display:



Define a symmetric cable; each half of the cable will be defined with three segments, two parabolic and a third segment joining them:

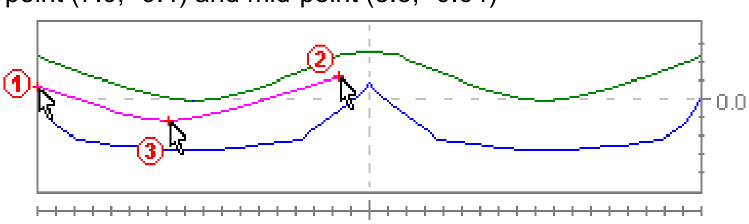


Define segment 1:



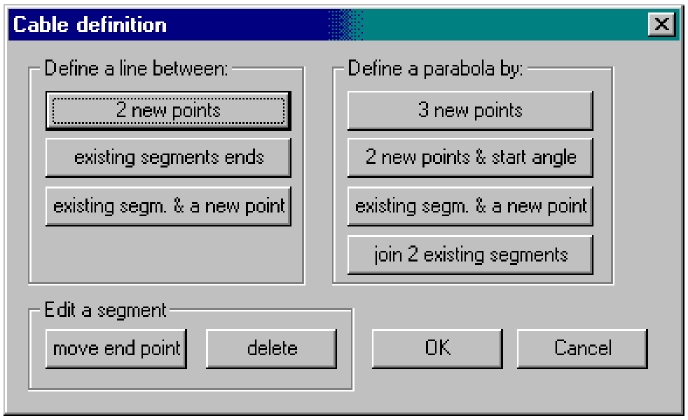
← *Select* **3 new points**

Move the cursor to the start point of the parabola (coordinates 0.0, -0.4) and click the mouse; repeat for the end point (7.0, -0.4) and mid-point (3.5, -0.64)

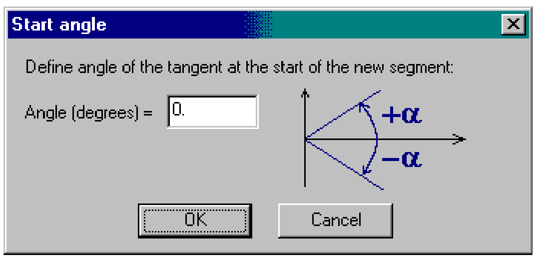


Alternatively, the coordinates may be typed in the dialog box at the base of the screen. For example:

Segment 2 is tangent to the horizontal at the span centre-line



← *Select* **2 new points & start angle**



← *Enter angle* = **0.**

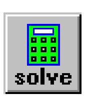
Move the cursor to the start point of the parabola (coordinates 10.0, -0.12) and click the mouse; repeat for the end point (8.0, -0.3)

Define segment 3 by joining segments 1 and 2:

- Move the cursor to Segment 1 so that it is highlighted with the rectangular blip; click the mouse
- Move the cursor to Segment 2 so that it is highlighted with the rectangular blip; click the mouse

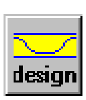
The program automatically draws Segment 3, modifying the end points of segments 1 and 2 to create a smooth curve.

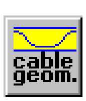
Repeat for the right side of the beam.

- click 

in the side menu; 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*

The program now is able to calculate the stresses accurately at all locations in the beam at any time, taking into account the calculation of losses, secondary moments, etc. To view the change in the allowable eccentricity envelope:

- click 

in the side menu.
- click 

in the side menu.

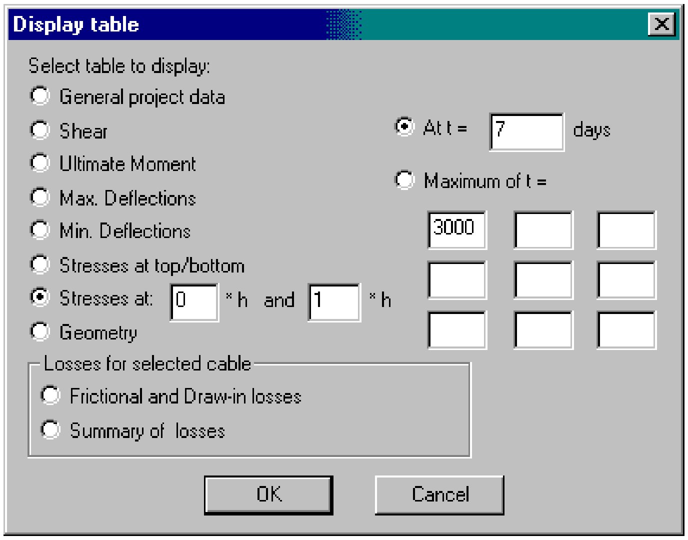
- click and highlight Cable no. 1 and click OK

To display results and a summary of the input data for the beam:

- click 

in the side menu.

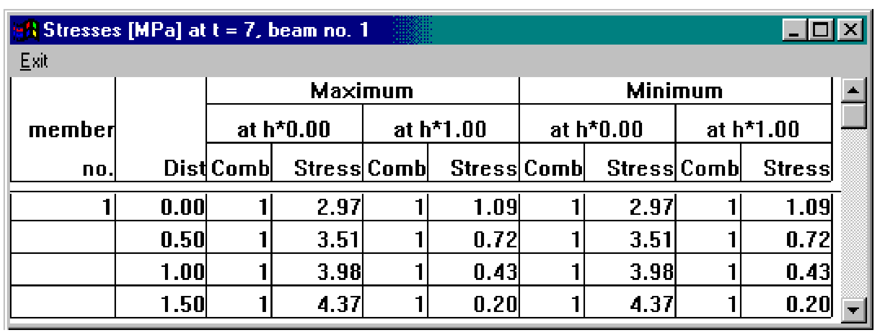
For example, to view the stresses at the top and bottom of the beam at time **t =7**



*Enter **t = 7** days*

Select **Stresses**

The program displays the maximum/minimum stresses at every $^{1}/_{20}$ of each span. For example:



member no.	Dist	Maximum at h*0.00 Comb	Maximum at h*0.00 Stress	Maximum at h*1.00 Comb	Maximum at h*1.00 Stress	Minimum at h*0.00 Comb	Minimum at h*0.00 Stress	Minimum at h*1.00 Comb	Minimum at h*1.00 Stress
1	0.00	1	2.97	1	1.09	1	2.97	1	1.09
	0.50	1	3.51	1	0.72	1	3.51	1	0.72
	1.00	1	3.98	1	0.43	1	3.98	1	0.43
	1.50	1	4.37	1	0.20	1	4.37	1	0.20

The program lists the combination associated with each stress.