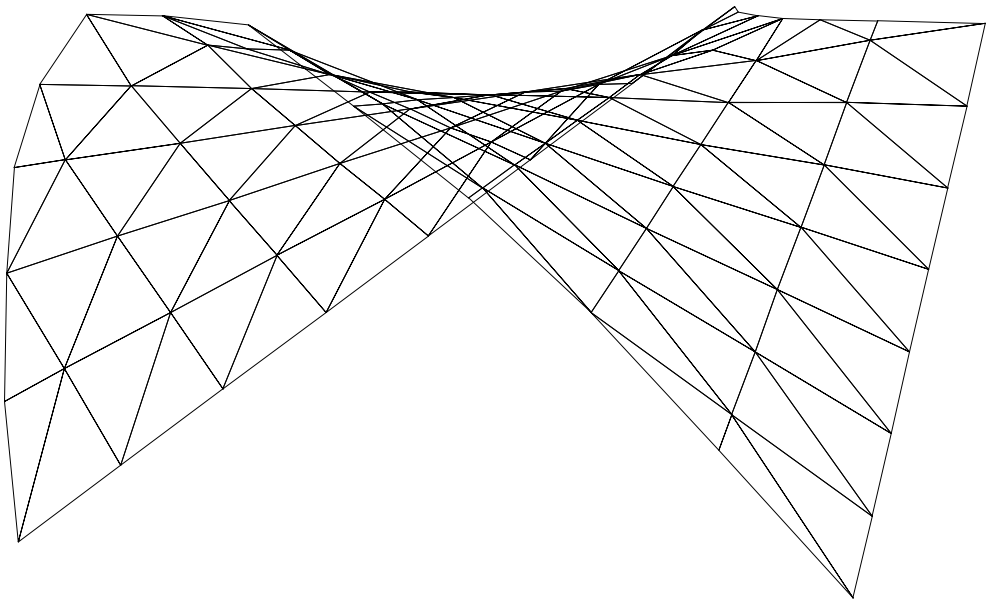


S T R A P

for

Windows

STRUCTURAL ANALYSIS
PROGRAM

TUTORIAL
&
DEMO
Examples

- **Metric units** -

Version 11.5

December 2004

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 following examples demonstrate various program options used for defining the geometry and loads for structural models and for displaying their results. These examples were designed as teaching aids only and should not be considered as a guide for accurate modeling of similar structures

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.

The user must verify his own results

Table of Contents

1 General 1-1
1.1 Installing the Program 1-2
1.2 Hints 1-3

2 Demo examples 2-1
2.1 Demo 1 - Plane Frame 2-2
2.2 Demo 2 - Plane Grid 2-8
2.3 Demo 3 - Space Frame 2-12
2.4 Demo 4 - Chess Loads 2-17
2.5 Demo 5 - Finite element results 2-20
2.6 Demo 6 - Result Combinations 2-22
2.7 Demo 7 - Structural steel design 2-25
2.8 Demo 8 - Reinforced concrete design 2-29
2.9 Demo 9 - Light gauge steel design 2-32
2.10 Demo 10 - Bridge loading analysis 2-37
2.11 Demo 11 - Bridge analysis – Results 2-39

Miscellaneous 2-58
3.1 Defining new models 2-58
3.2 How to Use HELP 2-59

1 General

This DEMO version is a limited version of the *STRAP* analysis and design program.

New models with **up to** 12 nodes may be defined, solved and saved. For instructions, refer to section 3.1.

The DEMO includes:
- the files of 11 existing models; these models already contain more than 12 nodes and additional nodes may be defined. **However, revisions to the geometry and loading will not be saved**.
- detailed instructions for defining the geometry or loading of models or viewing their results (Demo 2.1 to Demo 2.11).

The 11 existing models are:

- **"DEMO - Empty model"** :
  an 'empty' model i.e. it contains no existing geometry. It may be used to define any model up to the program capacity (32,000 nodes), e.g. the models in Demo 2.1, Demo 2.2 and Demo 2.3. However the geometry will not be saved.

- "**DEMO 4...**" to **"DEMO 11..."** :
  the models corresponding to Demo 2.4 to Demo 2.11. These Demo examples describe the definition of loads, viewing of results and use of the design postprocessors.

- "**Ramp**", "**Stadium**", etc.
  4 large solved models, intended primarily for displaying results.

Note:
- The Windows version of *STRAP* provides the user with extensive on-line Help; the **entire User's Manual** is an integral part of the program and may be displayed at any time. For instructions, refer to section 3.2.

1.1 Installing the Program

Before starting the program installation, make sure that you have all the hardware required to run *STRAP*:
- A personal computer with Windows 95/98/2000/NT
- At least 35 MB of free space on your hard disk

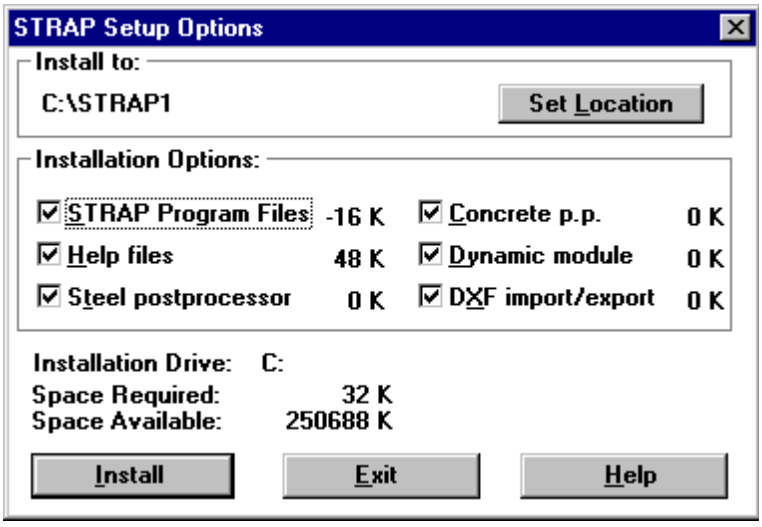
The files on the STRAP installation diskettes or CD are in compressed format; they are copied to the hard disk by an installation program. The program cannot be installed by copying the files from the diskettes or CD.

To install the program::

- Insert the CD in its drive.
- Follow the instructions on the CD case back cover
- Follow the instructions displayed on the screen:

Then define the program directory and select the modules to be installed:

The program should be installed in a **separate** directory (the installation program will open a new directory if the directory with the name entered does not exist).



To change the program folder, click Set Location

Set the required program modules to ☑, Note that the entire program does not have to be installed, a useful feature for updates.

Click Install to begin the installation.

1.2 Hints

- **Help**
  Note that Help is available at any point by pressing the F1 key; the program will display the manual page for the current option. Refer to 3.2.

- **Undo**
  If you made a mistake, simply click the **Undo** option in the menu bar at the top of the screen; the program will delete the previous definition step.

- **Icon bar**
  Note the row of small icons below the menu bar at the top of the screen. Clicking an icon provides access to many of the program options. Place the on an icon to display a 'hint' explaining its function.

- **Zoom**
  Click the **Zoom** option in the menu bar at the top of the screen and try the **Number of Windows** option. This option creates a 'split screen' showing different views of the model. Each view can be rotated to a different angle, have a different zoom and display different information.

  Any of the windows may be the 'active' one; move the into a window and double click the mouse. When you define part of the geometry in the active window, all of the other windows are updated simultaneously.

- **Display**
  Click the **Display** option in the menu bar at the top of the screen to display various information about the model - node, beam and property group numbers, support location and type, spring location, etc.

  Verify node coordinates by selecting **Define/revise dimension lines**; the program will display the distances between nodes that you select. The lines may be vertical, horizontal or parallel to any pair of nodes.

- **Print graphics**
  Select the **Output** option on the menu bar and click on **Print drawing**. The current display will be printed to the scale and with the title that you define.

- **File management**
  The program provides complete file management capabilities (backup, restore, delete, etc.); select the **Files** option in the menu bar in the program's initial screen (**STRAP models list**).

- **Create your own model**
  To create a new model of your own, simply **Exit** to the **STRAP models list**, select **Files** in the menu bar and click **New model**. Remember that only models with 12 nodes or less can be saved in the Demo version. Refer to 3.1.

Try the **Models wizard** option - you only have to type in a few parameters to create standard structural models. Note that the models available in the preprocessor are dependant on the **Model type** specified - click the **Wizard** button and the program will display the models that may be defined.

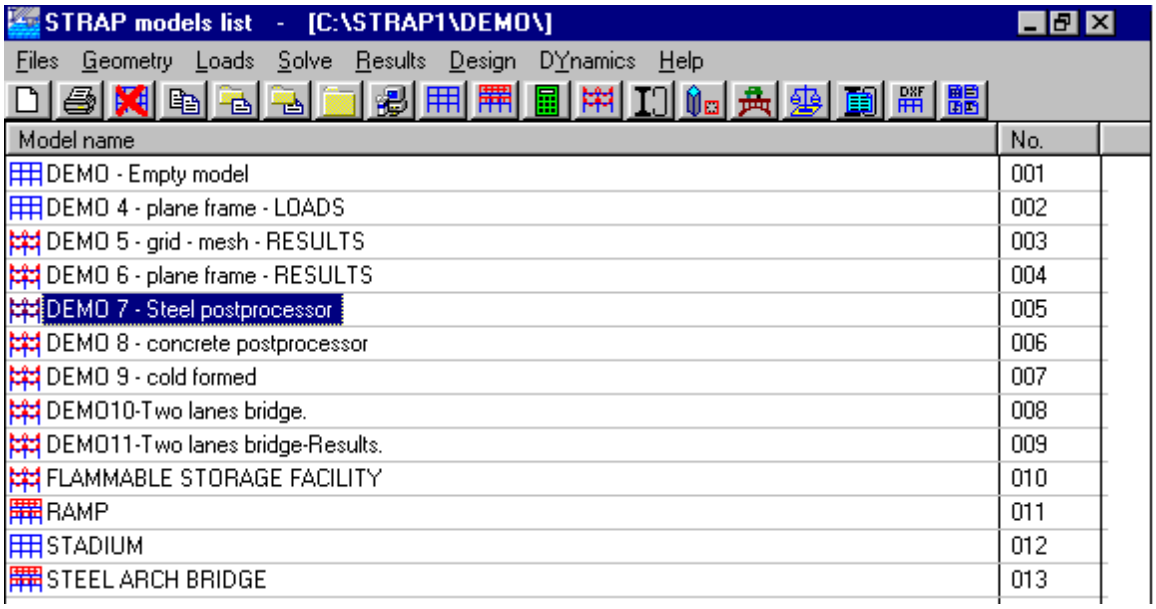
- Beam/element properties
  Beam properties (section properties, dimensions or steel section) are defined for property groups; each beam is then assigned to a group. Revise the whole group instead of making tedious member by member changes.

  Click the **Display** option in the menu bar and select **Property name** to verify that the correct property was assigned to all of the beams in the model.

2 Demo examples

Select the *STRAP* icon in the Windows Program Manager screen:

The program Main Menu and model list is displayed. The model list initially contains the names of the models supplied with the Demo. Remember that the geometry and loads of these models may be revised but the revisions will not be saved.

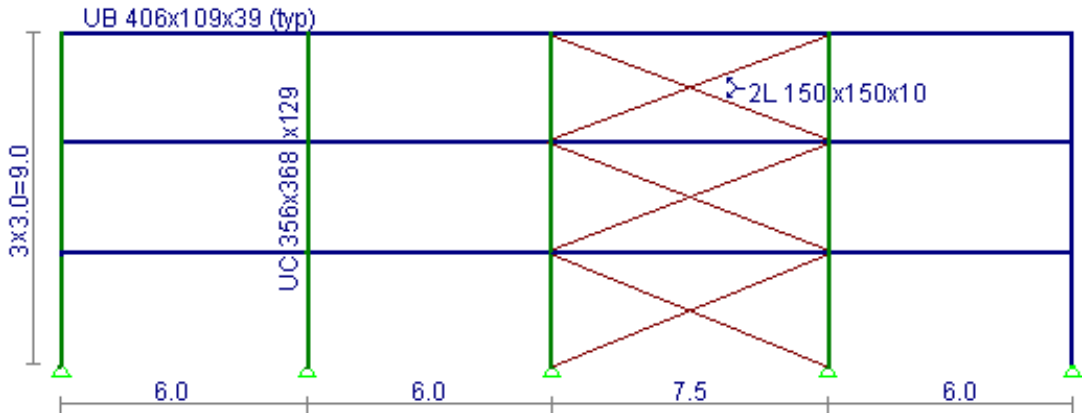


To select an existing model:

- move the to the model name in the list and click the mouse; the model name will be highlighted.
- follow the detailed instructions in each of the Demo examples.

2.1 Demo 1 - Plane Frame

Define the following plane frame using the Models wizard.



Demo 1 is a simple example that demonstrates the following:

- Definition of a simple plane frame, using *STRAP*'s model wizard library.
  STRAP contains a library of standard structures that enables the user to easily and quickly define the model geometry and loads by entering a small number of parameters. The library can be extended to include additional model types.
- Utilization of *STRAP*'s "copy" feature for defining the diagonals in the upper stories.
  STRAP's "copy" commands are powerful and sophisticated. The program refrains from duplicating existing beams. In our example, the program will duplicate only the diagonals and not the other existing members (beams and columns).

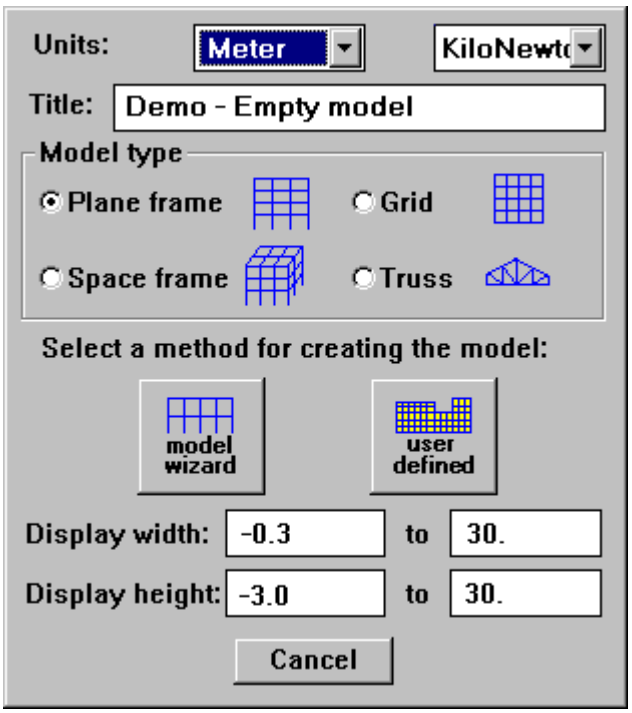
Define the model:

- **Main Menu**

- double-click **DEMO - Empty model** in the model list. This is an empty demo model that allows you to define any size model up to the program capacity.

- **Preliminary Menu**:

  Arrange the menu as follows:

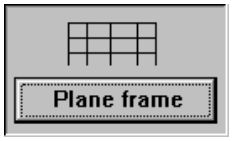


← *Revise the model units (if necessary)*

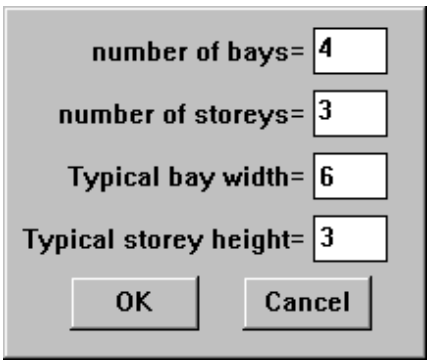
← *Set the model type to* **Plane frame**

← *Click* model wizard *to continue*

- Model wizard:

- Click 

- Define the plane frame parameters:



← *Type in the number of bays*

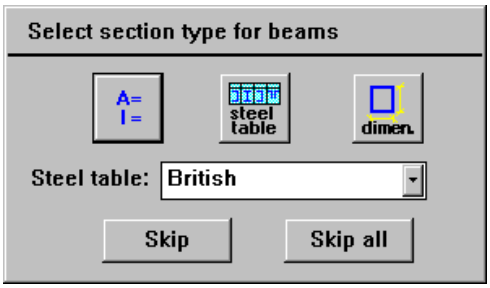
← *Type in the number of storeys*

← *Type in the bay width*

← *Type in the storey height*

← *Click the* OK *button*

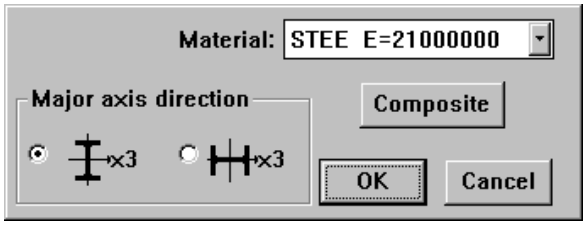
- Define the section type for beams:



← *Click the* steel table *icon*

← *Select* **American** *(if necessary)*

- Click **UB** in the section type list and **406x140x39** in the section list.



← *Select* **Steel**

← *Define* **X3** *as the major axis*

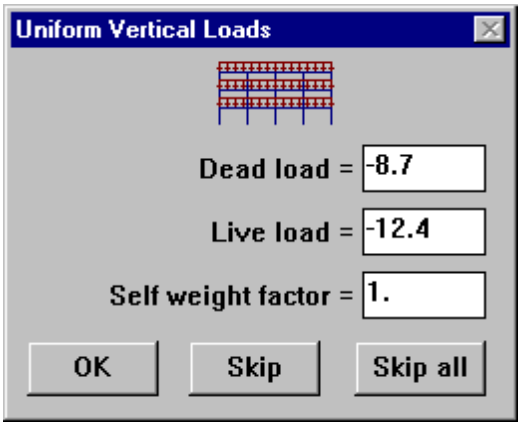
← *Click the* OK *button*

- For columns, click the steel table icon and select **UC** and **356x368x129.**
- Define dead, live and wind loads and load factors for combinations.

  The model wizard creates standard load cases from the basic load data. For the plane frame wizard, nine cases are created - Press F1 to display more details.

  Note that loads defined by a wizard in the Demo version are not saved and cannot be displayed. Loads for this frame are also defined in Demo 4.

- Define dead and live load:



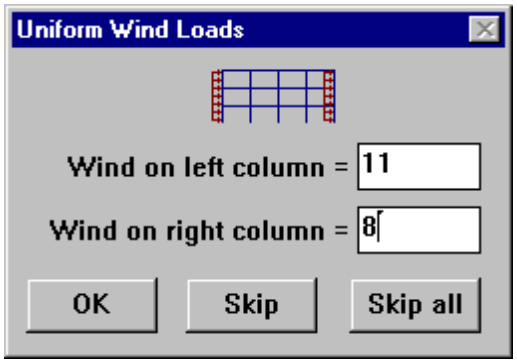
← *Type in the dead load value*

← *Type in the live load value*

← *Type in the self weight factor*

← *Click the* OK *button*

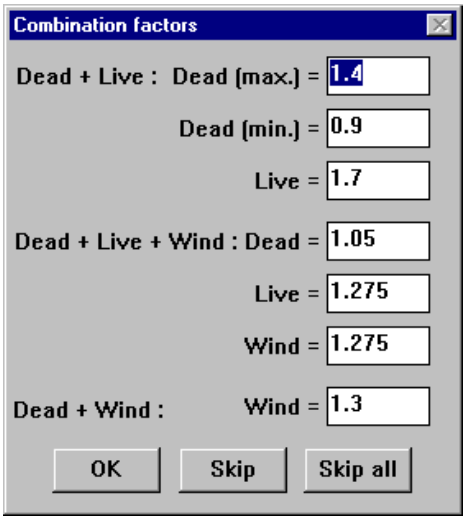
- Define Wind loads:



← *Type in the wind loads*

← *Click the* OK *button*

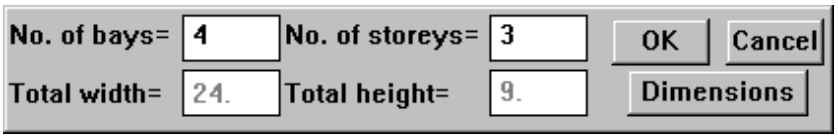
- Define combination factors:



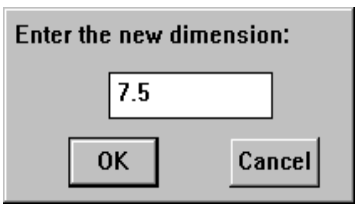
← *Enter the load factors*

← *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:

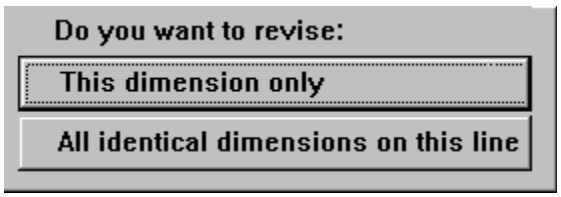


- To revise the dimension of the third bay to 7.5 m, click the Dimension button.
- Move the to the horizontal dimension line so that the dimension of the third bay is highlighted with the rectangular blip; click the mouse.



← *Enter the new dimension*

← *Click the* OK *button*



← *Select* **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.

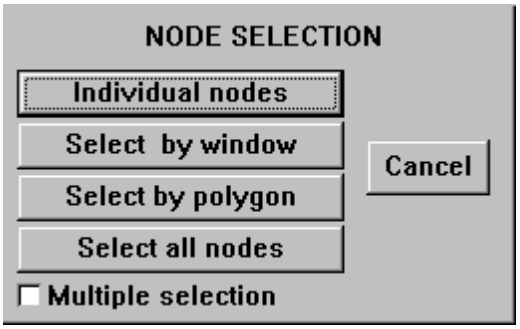
• **Geometry**

- click  in the menu bar to display the node numbers
- To define the bracing, click the  icon in the Main menu
- Click the  icon in the Beam options menu at the side of the screen
- move the  into the Dialog box at the bottom of the screen and set **Prop.= 3**; click **Screen** .
- move the  adjacent to node **3** so that the node is highlighted with a ■ ; click the mouse
- move the adjacent to node **9** so that the node is highlighted with a ■ ; click the mouse

  the program creates a beam linking nodes 3 and 9.
- In a similar manner, create a beam linking nodes 4 and 8.

Copy the bracing to all floors:

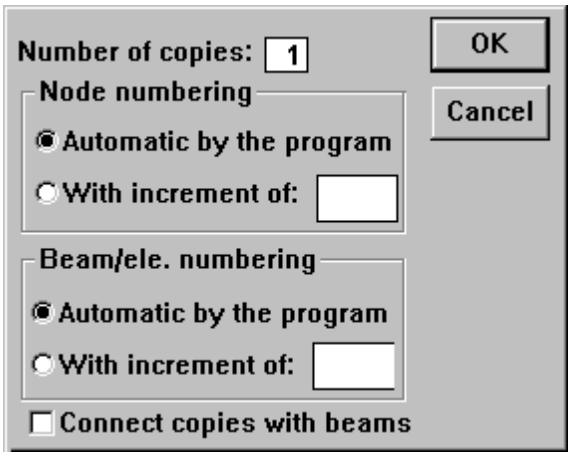
- Click the  icon
- Click the  icon
- Click the  icon in the Copy options menu at the side of the screen



← Click **Individual nodes**

- move the  adjacent to node **3** so that the node is highlighted with a ■; click the mouse
- Similarly select nodes 4, 8 and 9
- after the node 9 has been selected, click the mouse again without moving the crosshair (or click the **End selection** button in the Dialog box at the bottom of the screen).
- Select the reference node: move the  adjacent to node **3** so that the node is highlighted with a ■ ; click the mouse

- Select the new location of the reference node: click the [at an existing node] button and select node 8.



← *Set* **Number of copies** *to* **2**

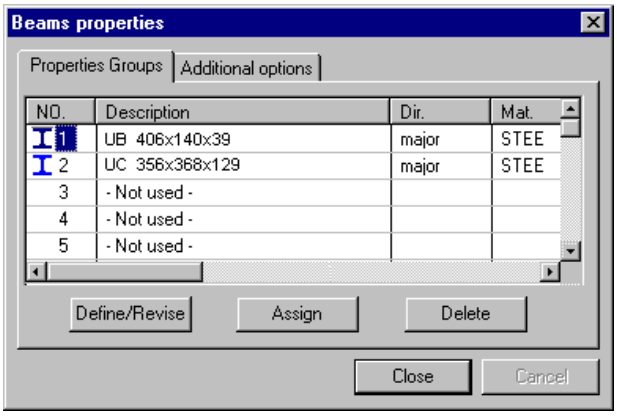
← *Select* **Automatic numbering**

← *Select* **Automatic numbering**

- click [OK]; the program creates and draws the new bracing

Define the section properties of the bracing:

- Click the main menu icon.
- Click the beam icon.
- Click the prop. icon.



← *Select* **3 – Undefined**.

← *Click* **Define/revise**

- click the steel table icon and select **2L** and **150x150x10**; click [OK]
- Click the [Close] button to complete the property definition.
- Click the main menu icon

You have now completed the definition of the geometry and the loads. To select the next Demo model:
- click **File** in the top menu bar
- select **STRAP model list** in the pull-down menu.

2.2 Demo 2 - Plane Grid

Define the grid of finite elements shown in Figure D.2.

Demo 2 demonstrates *STRAP*'s powerful features of automatically generating a mesh of elements. The user can define an area of any shape (which may also include holes in it) and the size of the elements that he wants. The program identifies existing supports (e.g. columns) and includes them in the mesh nodes.

When defining a mesh area adjacent to an already defined mesh, the program will take care of a smooth connection between the elements of the two areas.

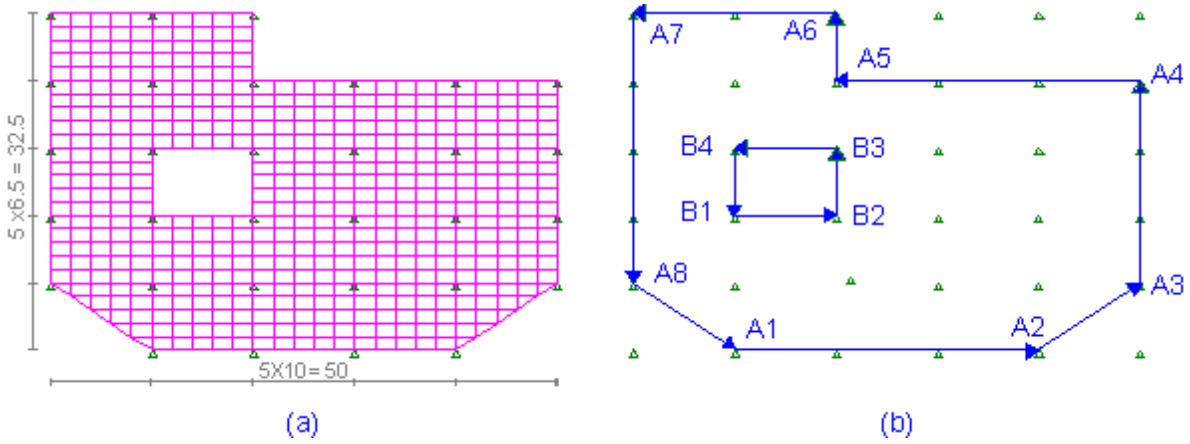


Figure D.2

- **Main Menu**
  double-click **DEMO - Empty model** in the models list. This is an empty demo model that allows you to define any size model up to the program capacity.

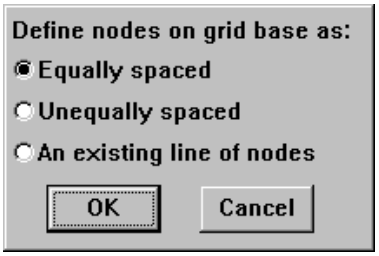
- Preliminary Menu

- set the default **Units** to **meter** and **kN** , the **Display width** range from **-5** to **60** and the **Display height** range from **-5** to **40** (if necessary).
- set the **Model type** to **Grid**
- click the user defined button

- **Geometry**

Define the grid of support nodes:

- click the node icon
- click the grid icon



← *Select* **Equally spaced**

← *and click the* OK *button*

- The base line starts at coordinates (0,0); type **0** (the cursor automatically enters the **X1=** box) and press [Enter]. The cursor moves to the **X2=** box; type **0** and press [Enter]. Press [Enter] or click the OK button.

  (Alternatively -
  - move the on the screen so that the coordinate in the Dialog box at the bottom of the screen are **X1=0.** and **X2=0.**; click the mouse.)
- The base line ends at coordinates (50,0); type **50** and press [Enter]. The cursor moves to the **X2=** box; type **0** and press [Enter]. Press [Enter] three times or click the OK button.

  (Alternatively -
  - move the on the screen so that the coordinate in the Dialog box at the bottom of the screen are **X1=50** and **X2=0.**; click the mouse.)
- Specify **5** segments and click the OK button
- Define the nodes on the grid height line as **Equally spaced** and click the OK button
- The height line ends at coordinates (50,32.5); type **50** and press [Enter]. The cursor moves to the **X2=** box; type **32.5** and press [Enter]. Press [Enter] three times or click the OK button.

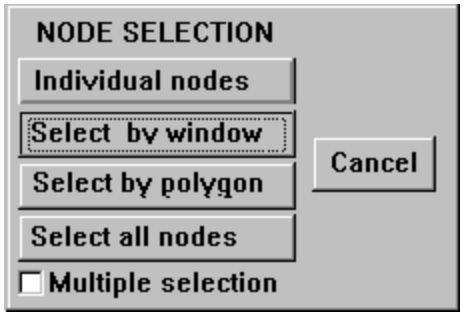
  (Alternatively -
  - move the on the screen so that the coordinates in the Dialog box at the bottom of the screen are **X1=50** and **X2=32.5** (or **dX1=0** and **dX2=32.5**); click the mouse.)
- Specify **5** segments and click the OK button; the program creates a grid of nodes and displays them on the screen.

Note that nodes not attached to elements are ignored by the program.

- Click the main menu icon.

Define the supports:

- Click the rest-raints icon.
- Click the pin. icon in the Restraints option menu at the side of the screen.



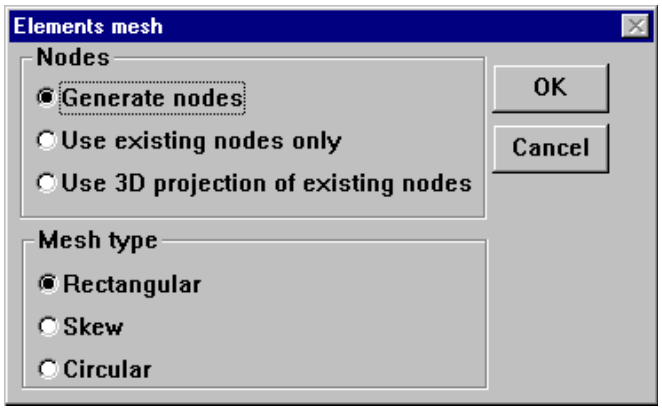
← *Click* **Select all nodes**

the program creates the pinned supports ; click in the icon bar at the to p of the screen to draw them on the nodes.

- Click the  icon.

Define the mesh of elements:

- click the  icon
- click the  icon

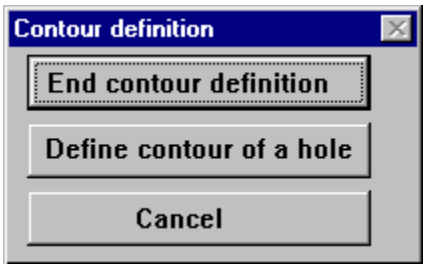


← *Select* **Generate nodes**

← *Select* **Rectangular**

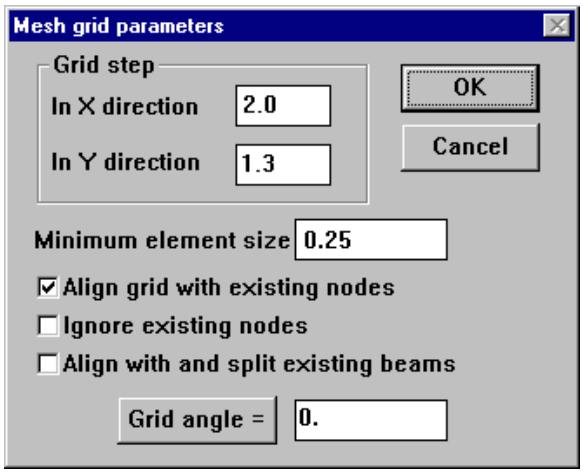
and click the OK *button*

- Define the contour line enclosing the floor area:
  move the adjacent to node A1 (Figure D.2b) so that the node is highlighted with a ■; click the mouse. Similarly select nodes A2 to A8 and A1 (again) to close the contour.



← *Click* **Define contour of a hole**

- move the adjacent to node B1 so that the node is highlighted with a ■; click the mouse. Similarly select nodes B2, B3, B4 and B1 (again) to close the contour of the opening.
- Select **End contour definition**
- Arrange the mesh parameters menu as follows:



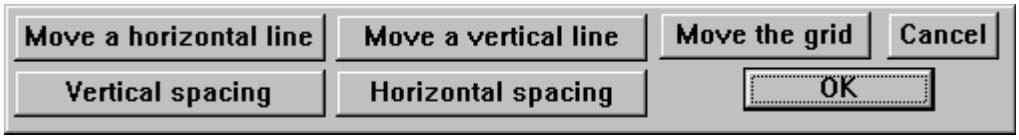
← *Set the* **Grid step** *to* **2.0**

← *Set the* **Grid step** *to* **1.3**

← *Set* **Align grid with existing nodes**

Click the OK *button*

- The program superimposes the preliminary mesh (6.0 x 4.0) on the area defined by the contour.



- Click the OK button.

  The elements as displayed in Figure D.2b 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?**

Define the element thickness:

- click the prop. icon.
- double-click the **1 - Undefined** line (all elements were assigned to Property Group no.1 by default)
- Set the **Thickness = 25 (cm.)**
- click the OK button
- Click the **End** button to complete the property definition.
- Click the main menu icon

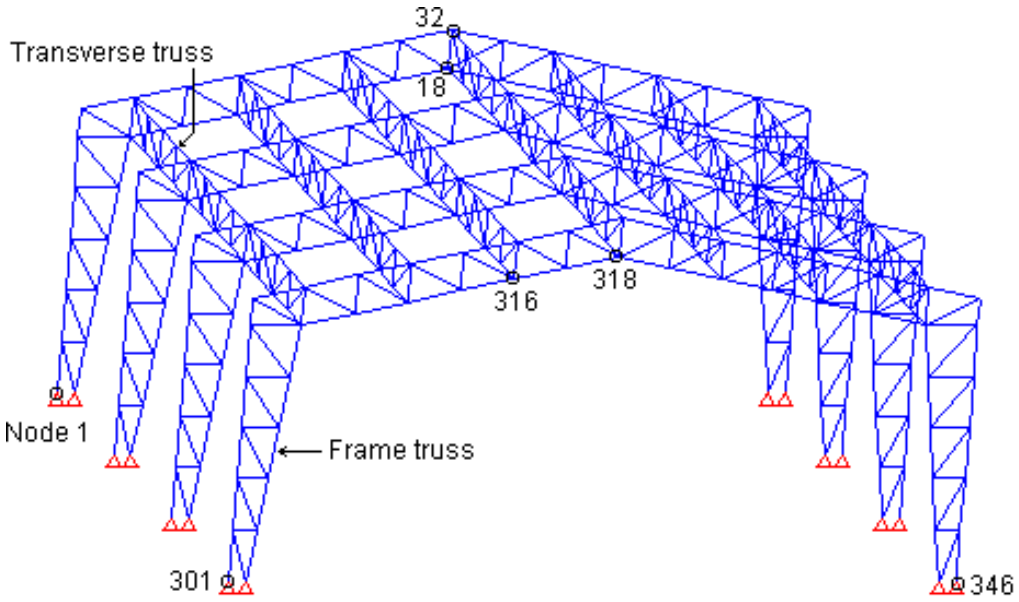
You have now completed the definition of the geometry.

To select the next Demo model -

- click **File** in the top menu bar
- select **STRAP model list** in the pull-down menu.

2.3 Demo 3 - Space Frame

In Demo 3 we will define a space structure.



In this Demo you will learn how to use *STRAP*'s more advanced features to quickly and easily define this complex space frame.

Note: You are advised to run this DEMO after trying Demo 1.

The demo demonstrates the following features of *STRAP*:

- Definition of a single frame by using a Model Wizard (similar to Demo 1).
- Duplication of the single frame to form the space model.
- Definition of the transverse truss as a submodel, also using a model wizard.
  The submodel feature enables to define part of the structure as a separate model and then add it to the main model at specified locations. Similar to the Copy option, the program does not generate double members.
- Working in two windows (split screen). The whole model is displayed in one window. In the other one, the transverse truss is displayed and selected for duplication.
- The Mirror option for copying the transverse truss.

To define the model:

- **Main Menu**

- Double-click **DEMO-Empty model** in the model list.

- **Preliminary Menu**

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

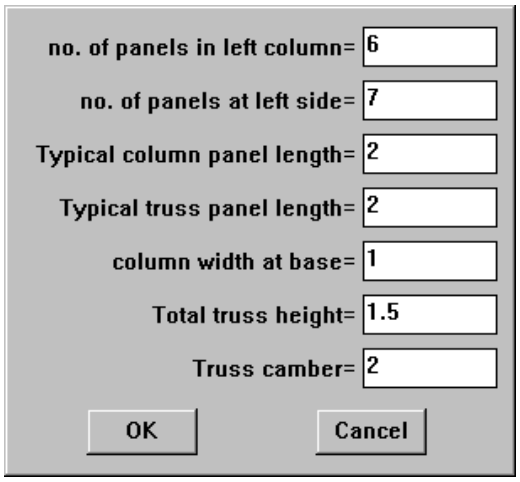
- Click the 

icon

- **Models wizard**

- Click the **Frame truss** button

- Specify the parameters:

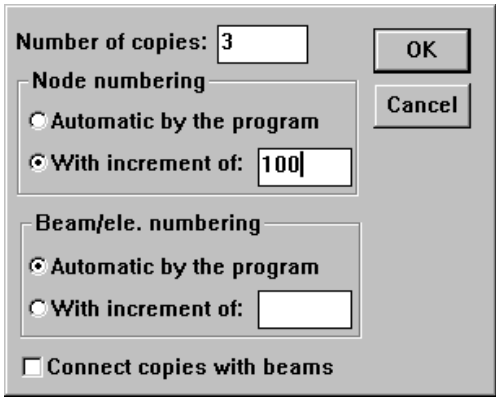


← *Click the* OK *button*

- click **Skip all** twice as we will not define section properties and loads in the Wizard.
- The program displays the model, click OK to continue

• **Copy the frame truss**

- click  in the menu bar to display the node numbers (note that it will be very convenient to switch the node numbers on/off many times during the course of building this model; these instructions will not mention it every time)
- Click the  icon in the main menu.
- Click the  icon in the copy options menu
- Click the **Select all nodes** button
- Select a reference mode: move the  adjacent to node 1 so that the node is highlighted with a ■; click the mouse.
- Define the new location of the reference node:
- Click the **by coordinates** button
- Type the coordinates **X1=0**, **X2=0**, **X3=8** in the box at the bottom of the screen.
- Click OK to continue



← Set **Number of copies = 3**

← *For convenience in identifying nodes in parallel trusses, set* **increment = 100**

click the OK *button*

- The program creates the new trusses. Click the +Y button in the icon bar twice and the +X button once to rotate the model to the isometric view.

- Click the main menu icon.

- **Restraints**

- Click in the icon bar; note that restraints we not copied. To define them on the copied trusses -

- Click the rest-raints icon.

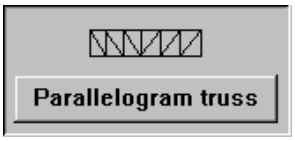
- Click the pin. icon in the Restraints option menu at the side of the screen.

- Click the **Individual nodes** button and select the 12 new base nodes.

- **Definition of the transverse truss:**

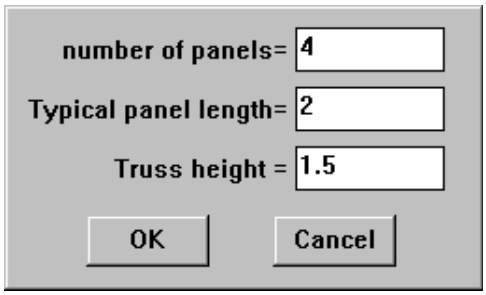
- Click the sub-model icon.

- Click the **Model wizard** button in the submodel menu

- Click the 

icon

- Define one bay only (4 panels):



← *Click the* OK *button*

- the program displays the model, click OK to continue

The Wizard defines by default two support restraints (Click in the icon to display them). As our structure is a submodel, we have to delete them.

- Click the rest-raints icon.

- Click the Del icon.

- Click the **Select all nodes** button

- Click the main menu icon.

Copy the truss to create 3 bays:

- Click the  icon in the main menu
- Click the  icon in the copy options menu
- Click the **Select all nodes** button
- Select a reference node: move the adjacent to node 1 (lower left corner of the submodel) so that the node is highlighted a ■; click the mouse.
- Define the new location of the reference node: click the **at an existing node** button
- move the adjacent to node 5 (lower right corner of the submodel) so that the node is highlighted with a ■; click the mouse.
- Set **Number of copies = 2** and click the OK button; the program creates and draws the truss.
- Click the full drawing icon in the icon bar at the top of the screen.

Add the submodel to the main model:

- Click the  icon.
- Click the  icon.
- Select reference nodes: move the adjacent to node 22 (at lower right end) so that the node is highlighted with a ■; click the mouse
- Select nodes 1 and 6
- Define the new location of the reference nodes:
- Select nodes 318 (the ridge of the bottom chord of the first frame), 18 (the ridge of the bottom chord of the last frame) and 32 (the ridge of the top chord of the last frame) or type the node numbers in the Dialog box at the bottom of the screen.

- **Duplicate the transverse truss**

  We will use the Zoom option - **Number of windows** - to simplify the node selection procedure.

- Select **Zoom** in the menu bar at the top of the screen.
- move the adjacent to - **Number of windows** - and then click **Display 2 geometry windows**
- move the adjacent to the right window and double-click the mouse or click the title bar; the "focus" will now be on this right window (note that the title bar is shaded!)
- Select **Zoom** in the menu bar.
- Create a window about the transverse truss.
- Select **Remove** in the menu bar at the top of the screen then select **Limit display to a plane**
- Select any 3 nodes in the truss plane (nodes 318, 18, 32 for example)

Copy the transverse truss:

- Click the  icon.

- Click the  icon in the copy menu.
- Click the **Select all nodes** button
- Select a reference node: select node 318 (the bottom node of the transverse truss, at the near end).

Define the new location of the reference node:

- Click the **at an existing node** button
- Move the adjacent to the left window and double click the mouse (to shift the focus to the left Window) then select node 316 (two nodes to the left from node 318).
- Set **Number of copies = 3** and click the OK button

• **Duplicate the transverse trusses to the other side**

Complete the definition of the secondary trusses by creating a mirror image *of the entire model*!

- Click the  icon.
- Click the **Select all nodes** button
- Select a reference node: select node 301 (the bottom left node of the left leg)
- Define the new location of the reference node:
- Click the **at an existing node** button and select node 346 (the bottom right node of the right leg).
- Click the OK button to copy the trusses

Please note that although we selected the entire model, the program copied the transverse trusses only because it automatically eliminates double members generated by the command.

To select another model:

- Click the **File** in the top menu bar.
- Select **STRAP models list** in the pull-down menu.

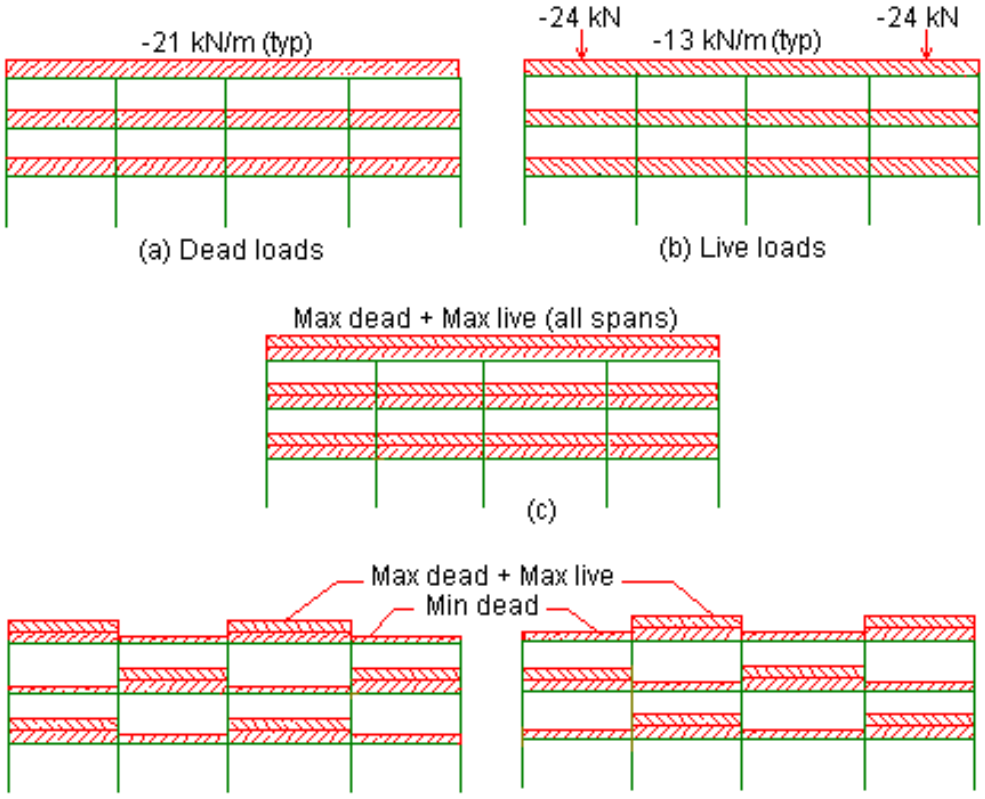
2.4 Demo 4 - Chess Loads

In this Demo we will create the following load cases for the frame defined in DEMO1:

This Demo demonstrates *STRAP*'s unique "Chess load" option that automatically generates the critical loading cases from basic Dead load and Live load cases (a & b in the drawings).

The program by itself identifies the continuous beams and applies the dead load and live load to the appropriate spans and with the correct load factors. The program is smart enough to identify intermediate nodes if defined and ignore them when applying the load.

- define the dead loads shown in Figure (a) in load case 1
- define all live loads shown in Figure (b) in load case 2
- use the program **Chess loads** option to automatically create load cases (c), (d) and (e).



- **Main Menu**

- highlight **DEMO 4** - **plane frame** - **LOADS** in the model list
- Click the loads icon in the side menu.

- **Load case 1 - dead loads**

- Click the new icon.
- enter the load case tile; type **Dead loads** and click the OK button
- specify **Beam loads** and click the define icon
- Click the unif. icon.
- Specify **Select only beams parallel to a beam** and click the **Select all beams** button
- Select the "parallel" beam (the load will be applied only to beams parallel to this one); move the adjacent to any horizontal beam so that the beam is highlighted with a ■; click the mouse.
- Type in the uniform **load =-21** (check that **Direction: FX2**) and click OK to continue
- The program creates the loads and draws them on the screen. Click the end load case icon.

- **Load case 2 - live loads**

- Click the new icon.
- enter the load case tile; type **Live loads** and click the OK button
- proceed as described for Load case 1, except enter a uniform **load = -13**
- to define the point loads, click the define icon
- Click the conc. icon.
- Click the **Individual beams** button
- Select one of the exterior roof beams; move the adjacent to any horizontal beam so that the beam is highlighted with a ■; click the mouse. Repeat for the other exterior roof beam.
- Click the **End selection** button
- Type in the point **load =-24** (check that **Direction: FX2** and that **Fraction = 0.5**) and click **End** to continue
- The program creates the loads and draws them on the screen. Click the end load case icon.

- **Chess loads**

- Click the 

icon in the main menu.
- The program correctly assumes that load case 1 contains all of the dead loads and that load case 2 contains all of the live loads. Correct the maximum/minimum load factors if necessary; click 

to continue.
- The program displays the series of generated load cases; click 

to continue after each case.

Check the loads in the generated load cases:

- Click the 

icon in the main menu and select the load case to be displayed, or
- select **Output** in the top menu bar, then **Display load commands** and select the load case to be displayed.

You have now completed the definition of the loads. To select the next Demo model -

- click **File** in the top menu bar
- select **STRAP model list** in the pull-down menu.

2.5 Demo 5 - Finite element results

For the plane grid defined in Demo 2, display graphic results for finite elements.

The graphic results for elements can be displayed in any one of three ways:

- **Results at element centres**
  display the model geometry with the numerical value of the result written at the centre of each element.
- **Contour map**
  display the model geometry with a contour map of the results superimposed. Each line of the contour map gives the location of a specified value of the result.
- **Results along a line**
  display the results plotted along a section through the model.

For example:

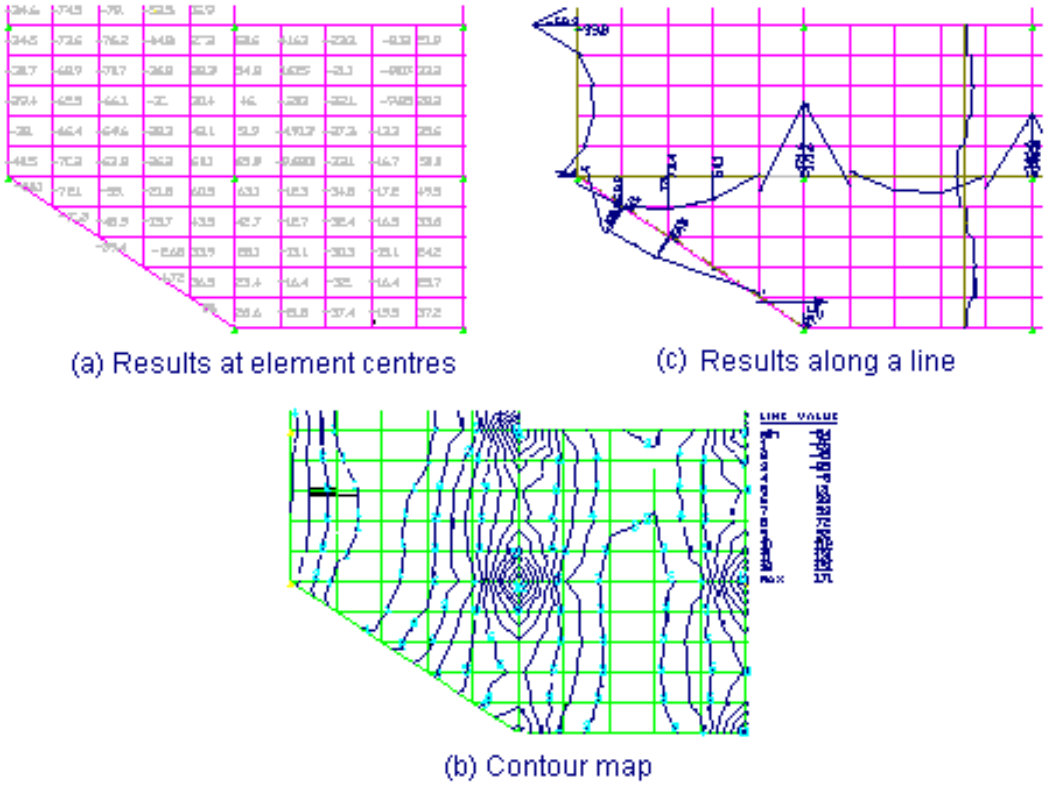


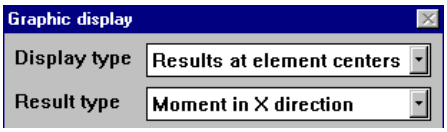
Figure D5 - Finite Element Results

- **Main Menu**

- highlight **Demo 5 - grid - mesh - RESULTS** in the model list
- click **Results** in the menu bar at the top of the screen

- **Results at element centres**

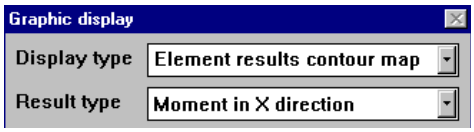
- select **Draw** and click the screen icon
- Specify:



- click the OK button; the program displays the results as shown in Figure D5a.
- Click in the icon bar to zoom in on any part of the model (Figure D5a shows the lower-left corner).

- **Contour map**

- Click the screen icon.
- Specify:

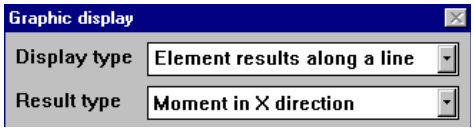


and **Fill contour regions with colour**

- click the OK button; the program displays the results as shown in Figure D.5b.

- **Results along a line**

- Click the screen icon.
- Specify:



- click the **Define a section line** button; define the horizontal section line shown in Figure D.5c.
- click the **Parallel to X1** button
- Move the to any point on the first interior horizontal line, i.e. so that **X2=7** is displayed in the bottom dialog box. Click the mouse.
- Define additional section lines or click the **End** button
- click the OK button; the program draws moment diagrams along the sections as shown in Figure D.5c.

To select the next Demo model –

- click **File** in the top menu bar
- select **STRAP model list** in the pull-down menu.

2.6 Demo 6 - Result Combinations

For the plane frame defined in Demo 1 and Demo 3, display results for beams.

The example will demonstrate the use of the Combinations options to create a series of combinations 1.2D+1.2W+1.2L (BS8110), each combining the wind load case with **one** of the staggered load cases.

Please note the following:

- In *STRAP*, unlike some other programs, you do not have to define the combinations along with the basic loading cases. You can define them after the solution and change them as you like without solving the model again.
- *STRAP* also has an option to define a "group" of load cases. If a group is added to a combination definition, the program will either:
  - automatically generate a different combination for each load in the group.
  - add the sum of the load cases in the group to the combination.

Using this feature when applicable can reduce the work required to create large numbers of loading combinations.

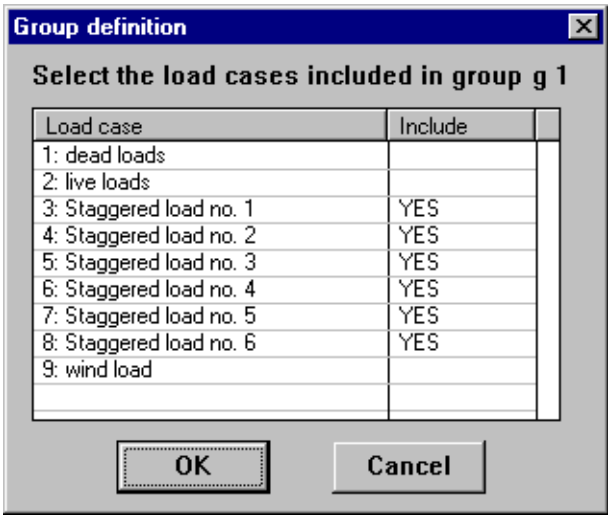
- **Main Menu**

Define the group:

- highlight **DEMO 6 - plane frame - Results** in the model list
- click **Results** in the menu bar

- **Combinations**

- click **Combinations** in the menu bar
- click **Define/revise groups** in the pull-down menu
- click the **Add/revise a group** button
- Click the first -**UNDEFINED**- group in the list, type in a group name and click the OK button



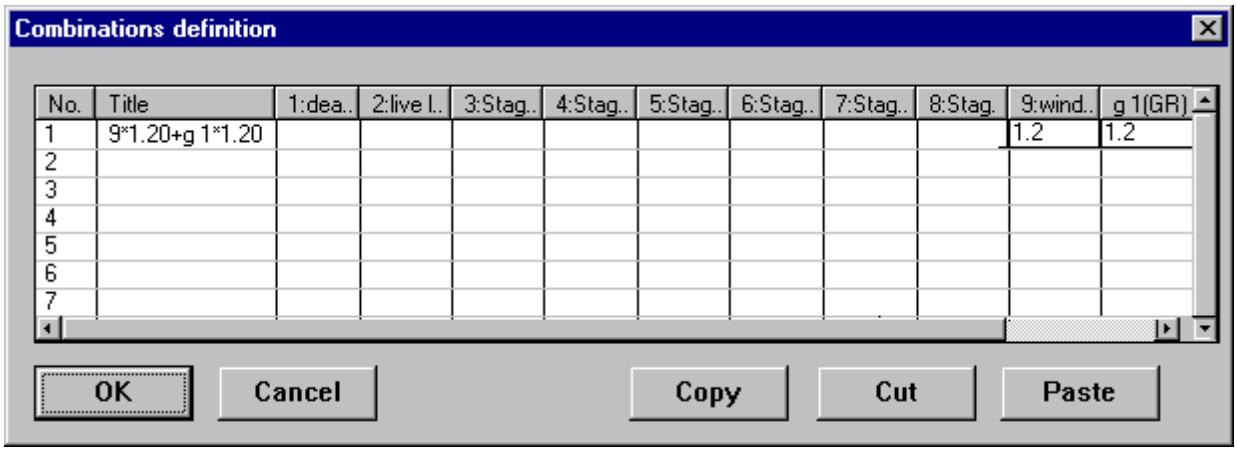
← *Click each of the* **Staggered load no. n** *so that* **YES** *is displayed on the right-hand side of the table opposite these cases.*

← *click the* OK *button*

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

Define the combinations:

- click **Combinations** in the menu bar
- click **Define/Revise combinations** in the pull-down menu
- in the following menu: move the to column "**9:wind**" for combination no.1 (first row), click the mouse and type **1.2**. Press [Enter]; the crosshair moves into the adjacent column "**group 1**"; type **1.2**.



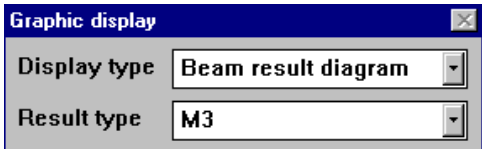
- click anywhere else in the menu; the program creates the title "**9*1.2+g1 *1.2**" in the title column.
- click the OK button. The program generates 'n' load combinations, where 'n' = the number of staggered load cases defined.

• Tabular results

- Specify **Tables** and click the screen icon
- Specify **Sort results by elements/nodes**
  Beam results: End and max. in span
- click the OK button; the program displays a table showing the moment, shear and axial force at both beam ends as well as the maximum span result.
- Click **Exit** in the menu bar of the table window to return to the main menu

• **Graphic results**

- Specify **Draw** and click the screen icon
- Specify:

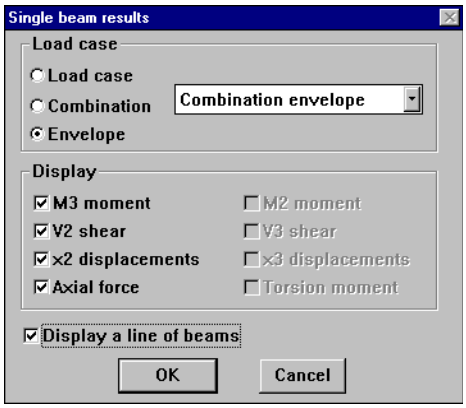


  and **Combination**; select any combination from the pull-down list box
- click the OK button; the program superimposes the result diagram on the model geometry.

• **Graphic results - Single beam**

- Display all envelope results for the beams on the first floor:

- Specify **Single beam** and click the screen icon
- Specify:



← *Select result types*

← *Select* **Display a line of beams**

← *click the* OK *button*.

- move the adjacent to beam 5 (start of line) so that it is highlighted with a ■; click the mouse. Similarly select beam 8 (end of line); the program draws envelope diagrams for all result types for the selected line of beam.

To select the next Demo model –

- click **File** in the top menu bar
- select **STRAP model list** in the pull-down menu.

2.7 Demo 7 - Structural steel design

The model geometry as defined in *STRAP* does not provide sufficient information for the postprocessor to carry out an accurate design.

For example, Figure D.7b shows a typical steel frame elevation. Figure D.7a shows the same frame as analyzed in *STRAP*. It is obvious that the postprocessor is unable to determine which STRAP members form continuous beams (chains of members must be designed as a single unit by the Postprocessor), the location of intermediate supports for lateral-torsional buckling and axial buckling as well as design constraints.

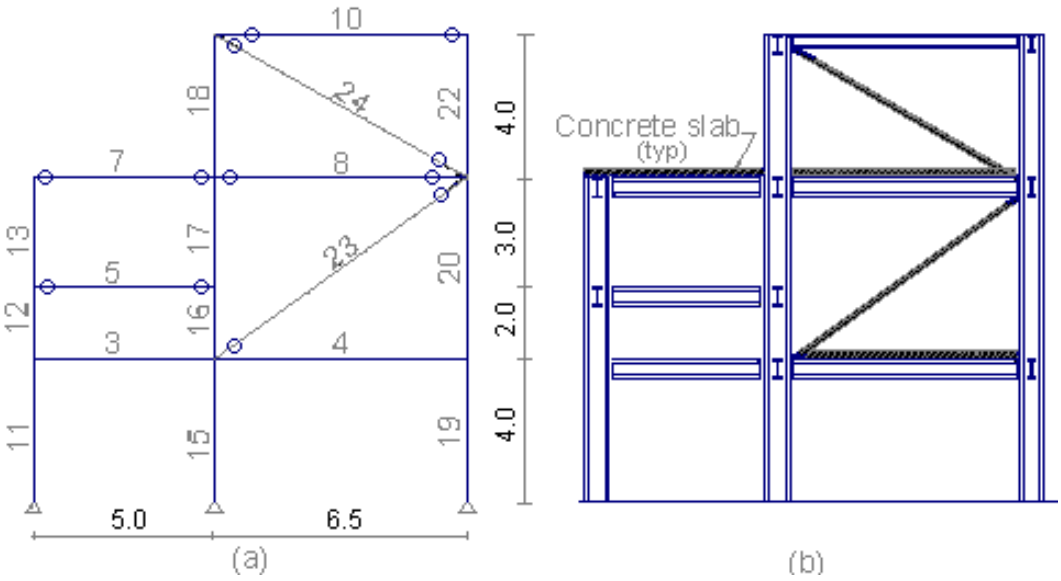


Figure D7 - Steel Postprocessor - Design Example

Define the following design data:

- Sections: Limit selection to:
  - beams - UB shapes
  - columns - UC shapes
  - bracing - double angles

- Identical beams:
  - Column 11-12-13 specified as an identical group
  - Column 15-16-17-18 specified as an identical group
  - Column 19-20-22 specified as an identical group
  - Bracing 23 and 24 specified as an identical group

- Intermediate supports:
  Members 4,7,8: "Continuous" support for +z major axis bending, major and minor axis buckling. The support is provided by the floor slab.

- Combined beams:
  Members 11,12: Beam 3 provides buckling support only for the major axis of this column, i.e. members 11 and 12 act as a single unit for minor axis buckling. Define major axis buckling and -z bending supports at the common node.

Define the model:

- **Main Menu**

- highlight **Demo 7 - steel postprocessor** in the model list
- click **Design** in the menu bar at the top of the screen
- click **Steel postprocessor** in the pull-down menu

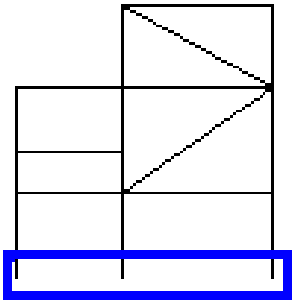
- **General**

- specify the ◉ **British section table** (the first time that the postprocessor is run on the model)
- Note the default parameters listed at the bottom of the screen. They automatically apply to all members in the model.
- To change the default parameters, click the defau·lts icon.
- To define different parameters for **a specific** member, click the param icon.

- **Section types**

The program can select the lightest section from a list (type or group) or check a specified section.

- Click the secti·ons icon in the side menu
- Specify ◉ **Limit section to section type** and select **UB**; click the OK button
- Click **Select all beams**
- Select **Display** in the menu bar and **Section type/group/check** in the pulldown menu; **UB** is displayed adjacent to every beam.
- Click the secti·ons icon in the side menu
- Specify ◉ **Limit section to section type** and select **UC**; click the OK button
- Set ☑ **Select only beams parallel to a beam**, click **Select all beams** and select any column member
- Click the secti·ons icon and select **2L** in the listbox; Click OK
- Click the **Individual beams** button
- Select the two diagonal bracing members.



- **Identical**

- Click the same sect. icon in the side menu
- Click the **Define lines of identical beams** button
- Specify ◉ **One node in the window/polygon** and click the **Select by window** button
- Create a window about the lower end of all the columns. Note that each column forms a separate “identical” group

- Click the 

icon
- Click the **Select a series of identical beams** button
- Click the **Individual beams** button and select the two diagonal members.

- **Supports**

- Click the 

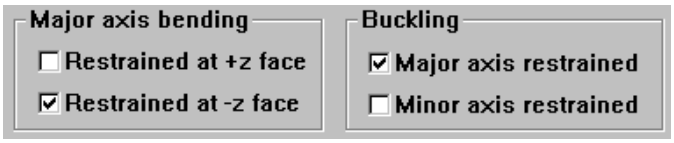
icon in the side menu
- Specify the following supports:

- Click the **Continuous support** button
- Click the **Individual beams** button and select beams 4, 7, and 8.
- Click the **End** button

- **Combined beams**

- Combine members 11 and 12 to form a single design unit and specify the major axis support provided by beam 3.
- Click the 

icon in the side menu
- Specify the following supports at the common nodes:



- Click the **Select each beams individually**
- Select beams 11 and 12: click on beam 11, click on beam 12 and then click on beam 12 **again** to end the selection.

- **Compute**

- click the 

icon and click the 

button; the program now selects the lightest section according to the user defined parameters. All load combination are checked for axial, bending, shear, LTB and combined stress capacity and for slenderness and deflection limitations.
- The program displays a table showing for each member the section selected, the critical combination, the slenderness, deflection and the capacity ratios (actual/capacity) for each of the design checks.
- click **Exit** in the menu bar.

- **Graphic results**

- click **Results** in the menu bar.
- click **Display selected sections** in the menu bar. The program superimposes the names of the selected sections on the model geometry.
- click **Results** in the menu bar.
- click **Display capacity** in the menu bar.
- specify ☑ **Colour by capacity** and ☑ **Display % of capacity**
- click the **Result type** box and select **Axial force+moment** from the list displayed (or any other type).
- click the OK button; the program superimposes the capacity percentage on the model geometry. Note the colour coding of the members and the text.

To select the next Demo model –

- click **File** in the top menu bar
- select **STRAP model list** in the pull-down menu.

2.8 Demo 8 - Reinforced concrete design

Design the beams and columns of the frame shown in Figure D.8 and create a column schedule.

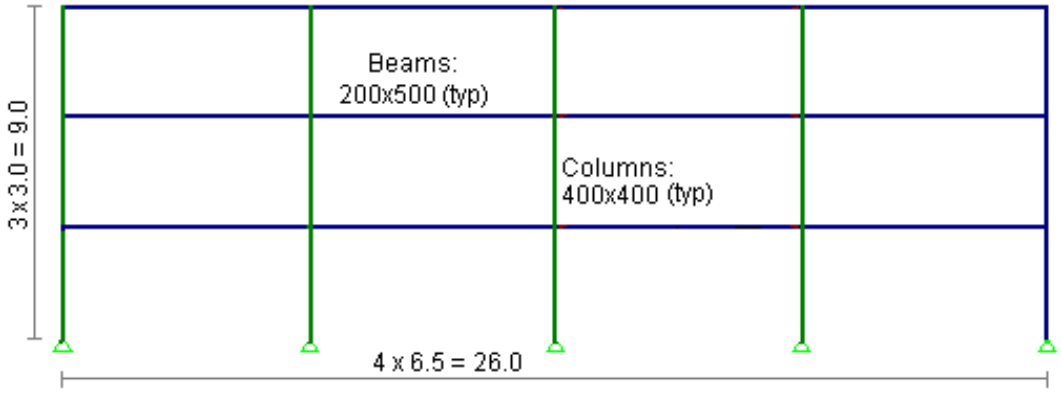


Figure D.8 - Concrete Postprocessor - Design Example

Beams and columns are designed separately; each has its own default parameters. The continuous beams and columns must be defined by the user

- **Main Menu**
- highlight **Demo 8 - concrete postprocessor** in the model list
- click **Design** in the menu bar at the top of the screen
- click **Concrete postprocessor** in the pulldown menu

Note:

- Note the **Height axis = X2** parameter. The program assumes by default that all members parallel to this axis are columns and that all members perpendicular to this axis are beams.
- beam default parameters are listed at the bottom of the screen. They automatically apply to all beams in the model. If you specify **columns** in the side menu, different parameters will be displayed.

  To change the default parameters, click the defaults icon.

  To define different parameters for **a specific** member, click the param icon.

Beams

- Specify **Beams** in the side menu

- Parameters

- click the defaults button and the Shear tab
- Specify **Shear reinforcement** as:
  Links only
  Diameter: Min=8 Max=8
  Shear reduction
- click the OK button

- **Define**

  Define the continuous beams in the model. The program assumes that beams are perpendicular to the "**Height axis**" and can create them automatically.

- Click the define icon
- click the **Automatic definition of all beams** button.
- click the **Display/revise beams** button.
- move the adjacent to any beam so that the beam is highlighted with a ■; click the mouse. The program displays the continuous beam schematically.
- click the **End** button.
- display more beams or click the **End** button.

- **Compute**

- Click the comp-ute icon
- Specify **For all defined beams** and click the OK button. The program calculates moment and shear reinforcement for all of the beams and displays a summary table listing the results.
- click **Exit** in the table menu bar

- **Results**

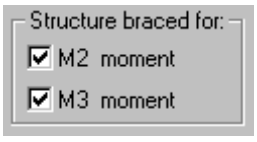
- select **Results** in the menu bar at the top of the screen
- select **Display detailed results** in the pull-down menu
- move the adjacent to any beam so that the beam is highlighted with a ■; click the mouse.
- Specify **Display envelope only** and click the OK button.
- The program displays the detailed results for the beam. Scroll though them.
- click **Exit** in the table menu bar

Columns

- Specify **Columns** in the side menu

- **Default parameters**

- Click the defau-lts icon
- Specify



- Click the Detailing tab

- Specify the column lap and link details
- click the OK button.

- **Define**

  Define the continuous columns in the model. The program assumes that columns are parallel to the "**Height axis**" and can create them automatically.

- Click the 

icon
- click the **Automatic definition of all columns** button.
- click the **Display/revise columns** button.
- move the adjacent to any column so that the column is highlighted with a ■; click the mouse. The program displays the continuous column schematically.
- click the **End** button.
- display more columns or click the **End** button.

- **Compute**

- Click the 

icon
- Specify **For all defined columns** and click the OK button. The program calculates reinforcement for all of the columns and displays a summary table listing the results.
- click **Exit** in the table menu bar

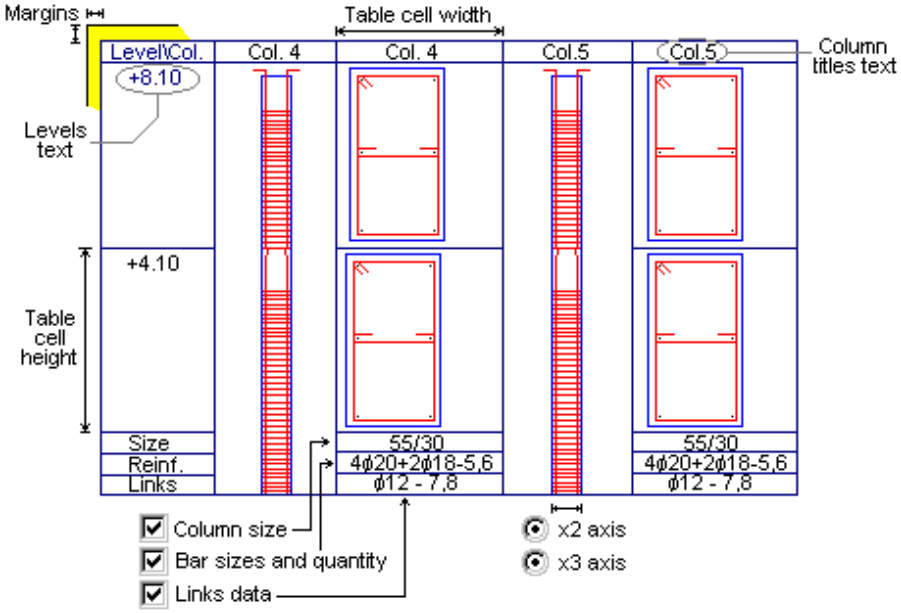
- **Results**
- select **Results** in the menu bar at the top of the screen
- select **Display detailed results** in the pull-down menu
- move the adjacent to any column so that the beam is highlighted with a ■; click the mouse.
- Specify **Design combination only** and click the OK button.
- The program displays the detailed results for the column. Scroll though them.
- click **Exit** in the table menu bar

- **Column schedule**

- Click the 

icon
- click the **New table** button.
- type in the table name, select the table size and orientation.
- Click **Edit**
- click **Select all beams** to include all columns in the table

- specify the schedule format parameters:



- click the OK button to display the schedule.

  The schedule is displayed without the elevations and with diamond-shaped additional links. Let's add the elevation for column 4 and revise the links to rectangular ones.

- Click the edit icon.
- Click and highlight **COL 4** in the table and click the **Add elevation** button on the right of the dialog box; a new line is added to the table.
- click the OK button to display the schedule again with the elevation for column 4.
- Click the main menu icon.
- Click the detail param icon.
- Click the **Link types** button and select **Use rectangular links where possible**
- click **Select all beams** to revise all the columns in the model
- click the **End** button and click the draw icon
- Click **Edit** To display the revised schedule.

To select the next Demo model –

- click **File** in the top menu bar
- select **STRAP model list** in the pull-down menu.

2.9 Demo 9 - Light gauge steel design

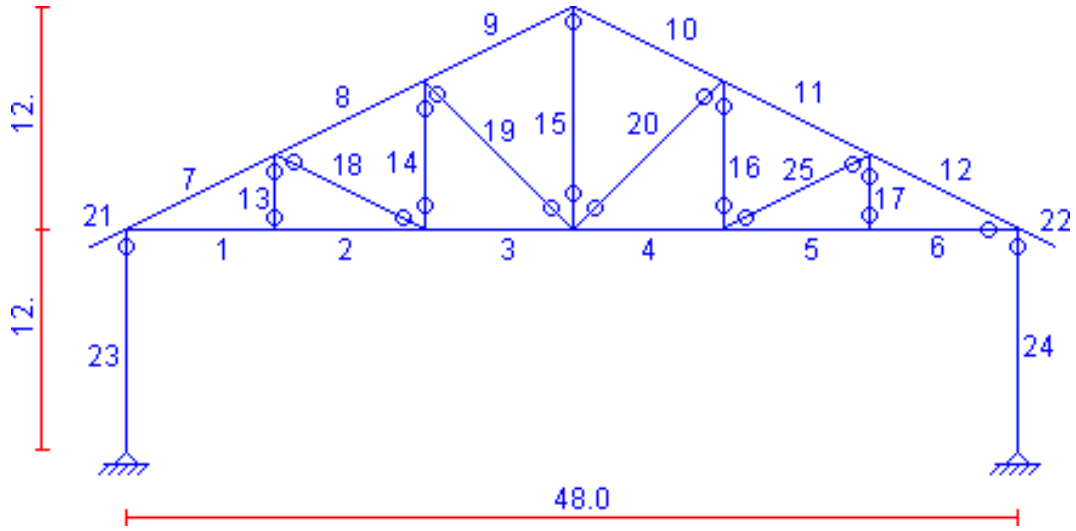


Figure D9 - Light Gauge Steel Design

This demo demonstrates the definition and design of a structure fabricated from Cold-formed (light gauge) steel sections.

The program designs cold formed sections according to one of the following codes:

- AISI ASD or LRFD specification - 1996.
- AISI ASD specification - 1986 with the 1989 addendum.

The user may use the standard section tables included in the program or define his own sections.

Note that structures with both cold-formed and hot-rolled sections may be designed by the program.

The following design data will be defined in the demo:

Sections:
- truss members : limit sections to C+Lips
- Columns : limit sections to CDEE

Identical beams:
- Members 23-24 specified as an identical group.
- Members 7-8-9-10-11-12-21-22 specified as an identical group.
- Members 1-2-3-4-5-6 specified as identical group.

Intermediate supports:

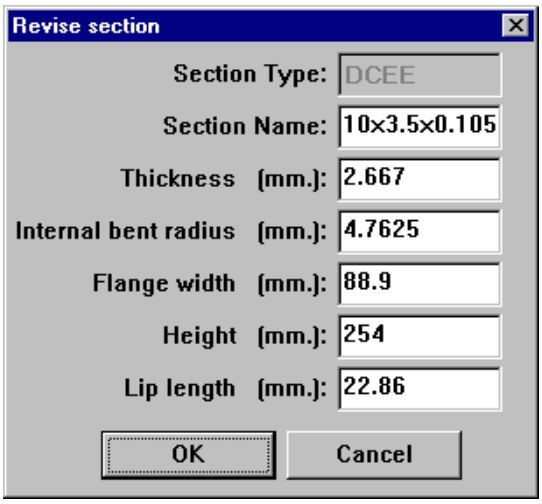
Members 7-8-9-10-11-12:	support for +Z major axis bending , minor axis buckling.
Members 23-24:	support for +Z major axis bending. minor axis buckling.

Combined beams:

Members 1-2-3; 4-5-6:	Define major axis buckling at the common nodes.

Define the model:

- **Sections**
- Select **Files** in the menu bar of the *STRAP* models list.
- Move the to **Utilities** and select **Create/Edit a steel section table.**
- Select **File** in the menu at the top of the screen.
- Select **Edit cold form table file.**
- Select **Edit** in the menu bar at the top of the screen.
- Select **Add section type** in the pull down menu.
- Click the button to open the section shape pull-down-menu
- Select: **2C sections with lips front to front**
- Type the **Section type name** as - **DCEE** (notice that DCEE in the section type menu is highlighted).
- Click the OK button
- Select **Edit** in the menu bar and Select **Add section to current type.**



← *Enter the dimensions*

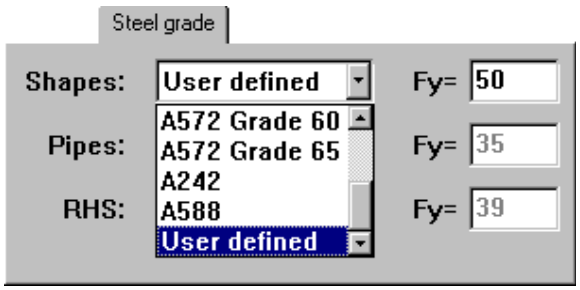
← *Click the* OK *button*

- Click OK in the section data table
- Click **File** in the menu bar
- Click **Exit** in the file menu.

- **Main Menu**
- Highlight **Demo 9 - Cold Formed** in the model list.
- Click **Design** in the menu bar at the top of the screen.
- Click **Steel postprocessor** in the pull-down menu.

- **Default parameters**
- Note the current default parameters listed at the bottom of the screen. To change a parameter, click the defau lts icon in the side menu

- specify a new value for Fy:



← *Click the* **Steel grade** *tab*

← *select "***User defined***" in the "***Steel grade***" list box and type "***50***" in the adjacent "***Fy=***" text box.*

- Click OK

• **Sections**

- Click the sections icon in the side menu
- Specify **Limit section to section type** and select **C+Lips**; click the OK button
- Specify **both nodes in the Window/polygon** and click **Select by window**
- Create a window about the truss.
- Click the sections icon and select **DCEE** section type; Click OK
- Click the **Individual beams** button
- Select the two column members.

To display and check the last definition:

- Click **Display** in the menu bar at the top of the screen
- Click **Section Type/Group/Check** in the pull-down menu
- (To Delete the section type from the display use the same option again)

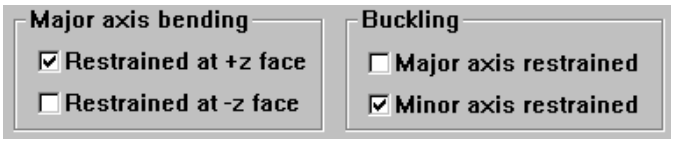
• **Identical**

- Click the same sect. icon in the side menu
- Click the **Select a series of identical beams** button
- Click the **Individual beams** button and select the two columns (members 23,24)
- Click the same sect. icon
- Click the **Define lines of identical beams** button
- Click the **Individual beams** button and select the beam members 21,22 and 1.

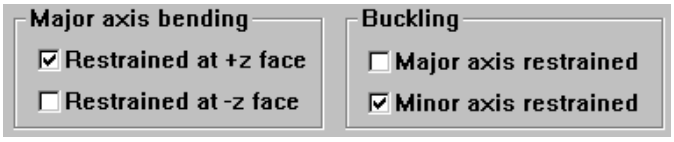
Note that each beam forms a separate identical group. (use "Display" option)

- **Supports**

- Click the  icon in the side menu
- Specify the following supports:



- Click the **Intermediate supports** button
- Click the **Individual beams** button and select the beams 7-8-9-10-11-12.
- Specify **Distance from beam start = 4.5**
- Click the **End** button
- Click the  icon
- Specify the following supports:

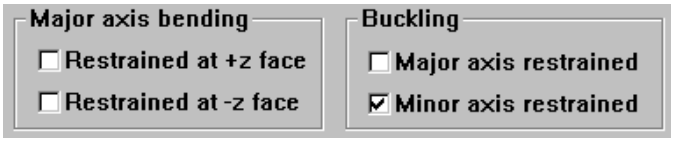


- Click the **Intermediate supports** button
- Click the **Individual beams** button and select the columns 23, 24.
- Specify **Distance from beam start = 3.0**
- Click the **Next support** button
- Specify **Distance from beam start = 6.0**
- Click the **Next support** button
- Specify **Distance from beam start = 9.0**
- Click the **Next support** button
- Click the **End** button
- Use the “Display” option to check your definition

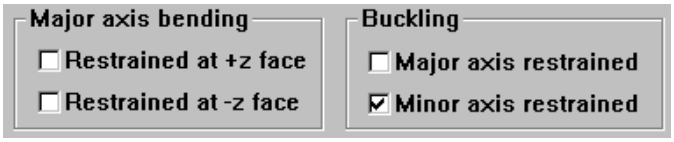
- **Combined beams**

Combine members 1-2-3 and 4-5-6 to form a single design unit:

- Click the  icon in the side menu
- Specify the following supports at the common nodes:



- Click the **Select start and end beams in a line**
- Select beams 1 and 3
- Click the comb. beam icon in the side menu
- Specify the following supports at the common nodes:



- Click the **Select start and end beams in a line**
- Select beams 4 and 6.
- Use the “Display” option to check your definition.

• **Compute**

- Click the I? comp- ute icon in the side menu
- Click the OK button; the program now selects the lightest section according to the user defined parameters and slenderness and deflection limitations.
- The program displays a table showing for each member the section selected, the critical combination, the slenderness, deflection and the capacity ratios (actual/capacity) for each of the design checks.
- Click **Exit** in the table menu bar.

• **Results**

- Select **Results** in the menu bar
- Select **Display detailed results** in the pull-down menu.
- Move the adjacent to column 23 so that the column is highlighted with a ■; click the mouse.
- Specify **Design combination only** and click the OK button
- The program displays the detailed results for the column. Scroll through them.
- click **Exit** in the menu bar.

To select the next Demo model -

- Click **File** in the top menu bar.
- Select **STRAP model list** in the pull-down menu.

2.10 Demo 10 - Bridge loading analysis

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

Analyze the bridge shown in Figure D10.1

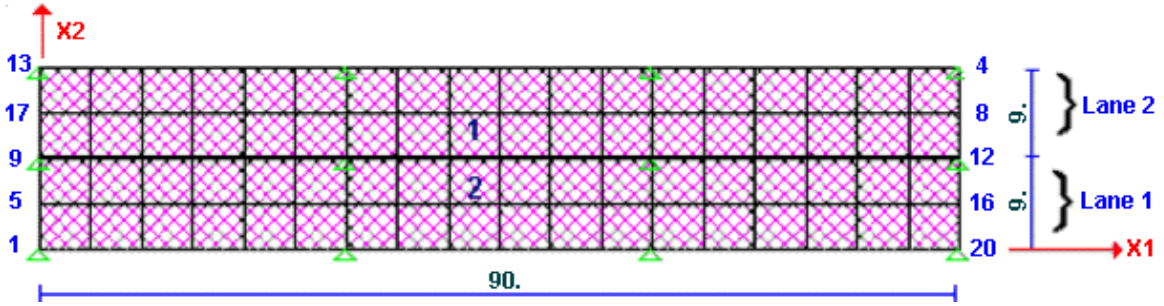


Figure D10.1 - Bridge example, Lanes

- **General**

- 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.

- **Main Menu**

- Highlight **Demo10 – Two lanes bridge** in the model list.
- Click **Design** in the menu bar and **Bridge module** in the pull-down menu.

- **Lanes**

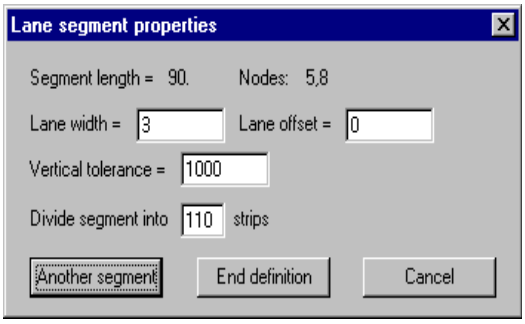
- The bridge consists of 2 parallel lanes, each 9 feet wide. They are defined by specifying the center line axis and width.

← *Select* **Lanes**

- Click the 

icon to define lane 1.
- Click Node 5 to specify the start of lane.
- Click Node 8 to specify the end of lane.

- Define the lane parameters:



← *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*

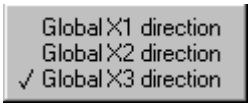
← *Click* **End definition**

- Similarly define lane 2 between nodes 13 and 16.

- **Load distribution**

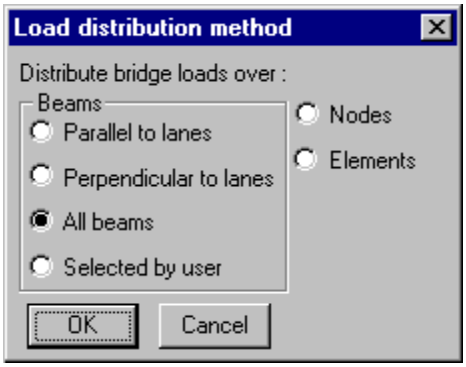
The generated loads may be applied to nodes, elements or selected beams.

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



← *Select Global X3 direction*

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



← *Click* **All beams**

← *Click the* OK button

- **Solve the model**

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

  - Demo version:
    The large number of load cases exceeds the limit of the Demo version. To continue, please go to **Demo 11** which is identical to this model but includes the result files.
  - Regular version:
    Select **File** in the menu bar and **Solve** in the pulldown menu and then continue according to the instructions in **Demo 11**.

2.11 Demo 11 - Bridge analysis – Results

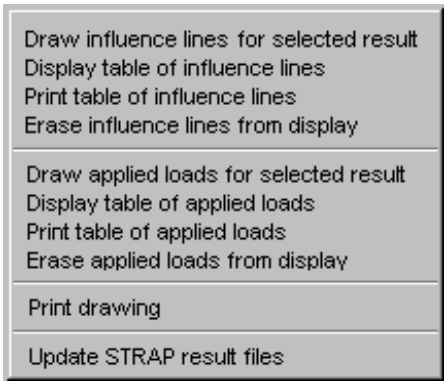
- **Main Menu**

- Highlight **Demo 11 – Two lanes bridge – Results.**
- Click **Design** in the menu bar and **Bridge model** in the pull down menu.

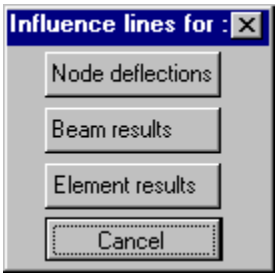
- **Display influence lines**

  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.

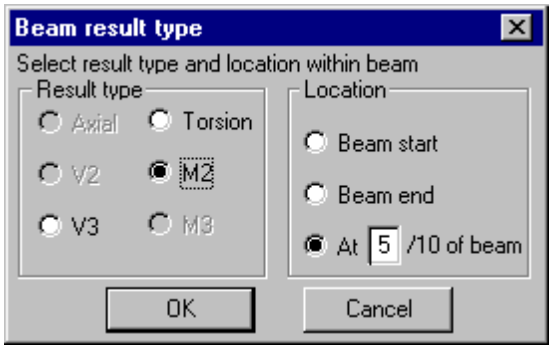
- Select **Results** in the menu bar:



← *Select* **Draw influence lines**



← *Select* **Beam results**

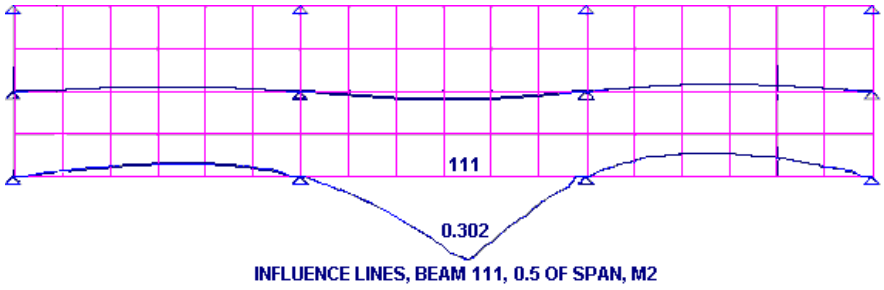


← *Select* **M2**

← *At the midpoint of the span*

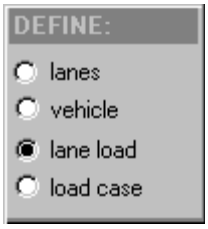
← *Click the* OK *button*

- To select beam 111, highlight it with the ■ and click the left button of the mouse.
- The program displays the following influence line:



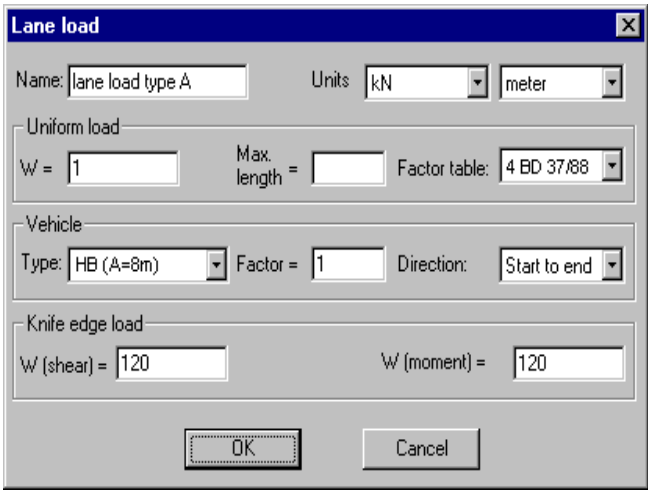
- **Define lane loads**

- 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**

← *Select a load reduction (vs. length) factor table*

← *Select a* **Vehicle** *group*

← *Type in* **Knife edge load** *value(applied by the program as a uniform load to a strip)*

← *Click the* OK *button*

- **Define load cases**

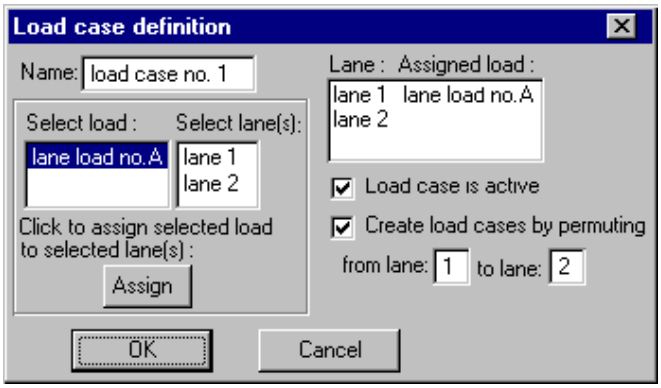
Assign a lane load to each lane to define a load case. The program applies the load to each of the strips along the length of the lanes; only those loads that contribute to the requested maximum/minimum result are used.

← *Select* **load cases**

- Click the define icon
- Define three load cases:
  1 - apply lane load A to lane 1 only.
  2 - apply lane load A to lane 2 only.
  3 - apply lane load A to both lane 1 and lane 2.

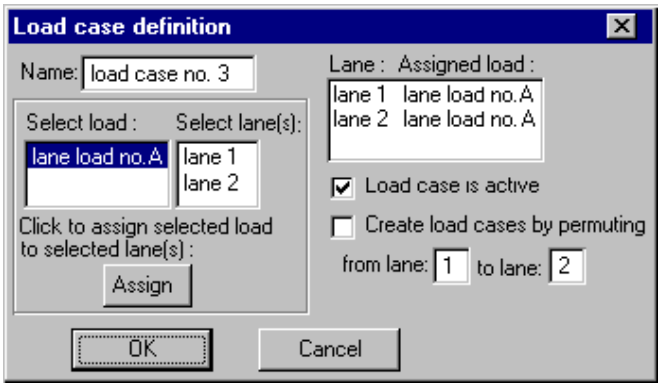
- Define load case 1 and 2:

  Create the load cases by specifying 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* **Assign** *button*
← *Click the* OK *button*

- Define load case no. 3.

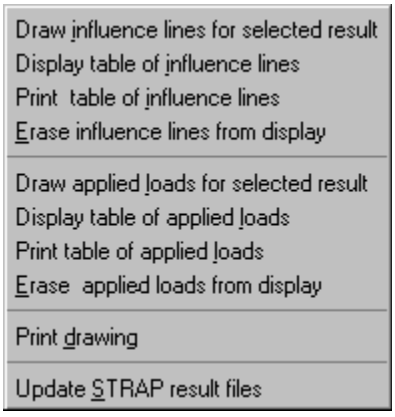


← *Select lane load A*
← *Select lane 1 and 2*
← *Click the* **Assign** *button*
← *Click the* OK *button*

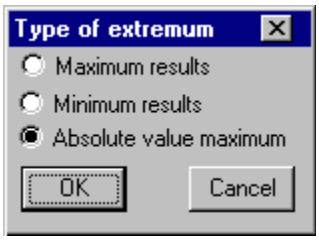
- **Display selected results**

  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.

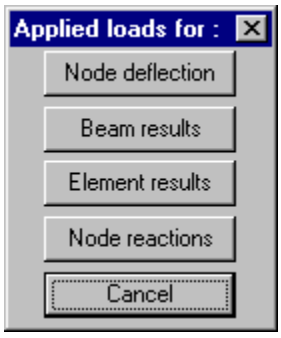


← *Select* **Draw applied loads for selected results**

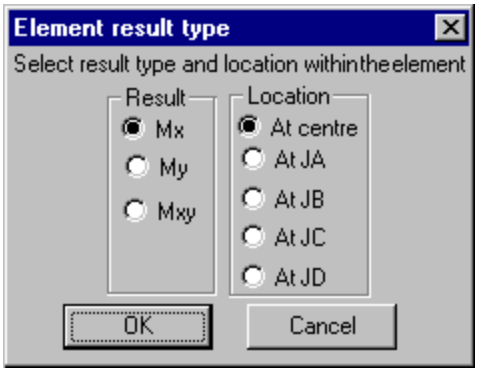


← *Select* **Absolute value maximum**

← *Click the* OK *button*



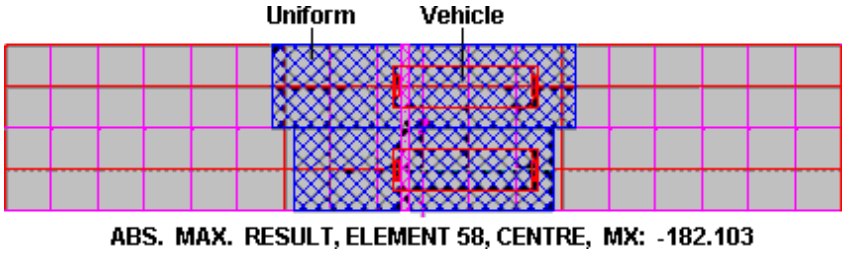
← Select **Element results**



← *Select* **Result type** *and* **Location**

← *Click the* OK *button*

- To select element 58 click the mouse.
- The program displays the location of the loads applied to achieve the max results.

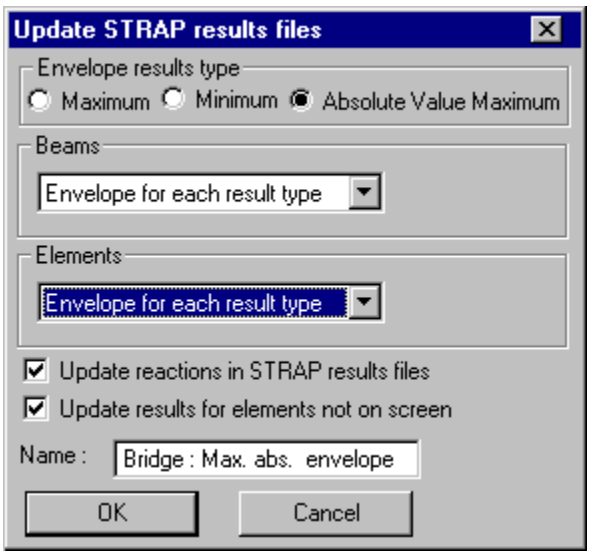


- **Transfer results to STRAP**

  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.

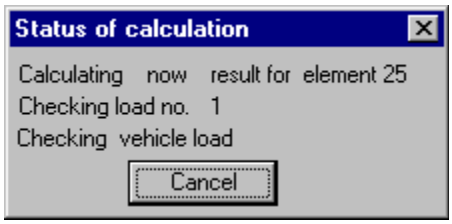


Select envelope result type

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.

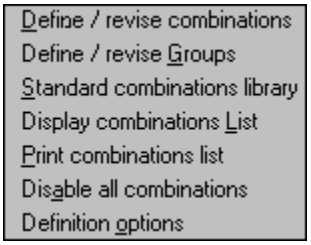


- Select **File** in the menu bar.
- Select **STRAP results** to return to the STRAP results screen.

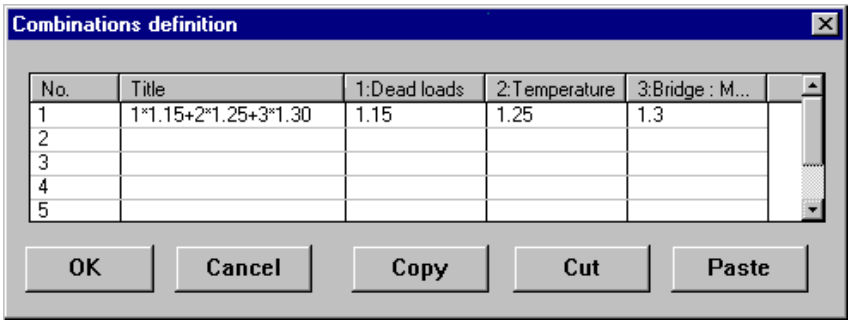
- **Combinations**

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

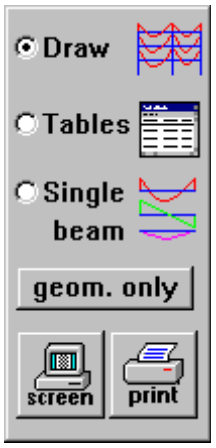
- Select **Combinations** in the menu bar



Define/revise combinations

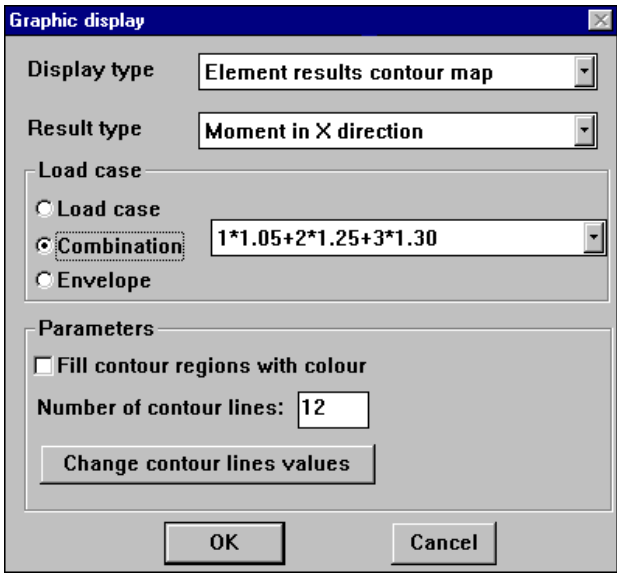


- Click the OK button to exit from table.
- To display results:



← *click* **Draw**

← *click the* screen *icon*



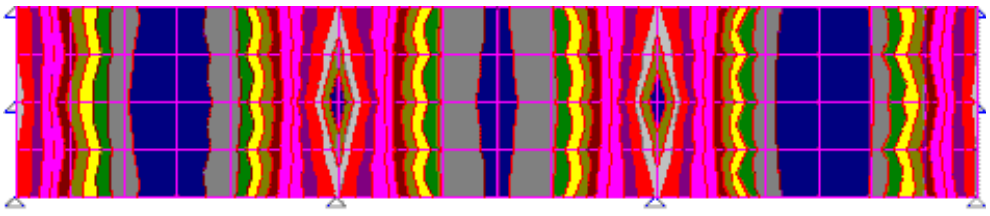
← *Select* **Display type**

← *Select* **Result type**

← *Select* **Load combination**

← *Select* **Fill contour regions with color**

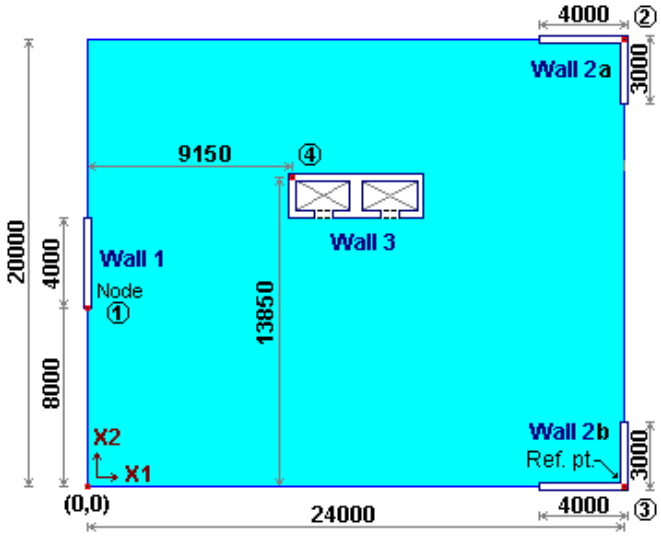
← *Click the* OK *button*

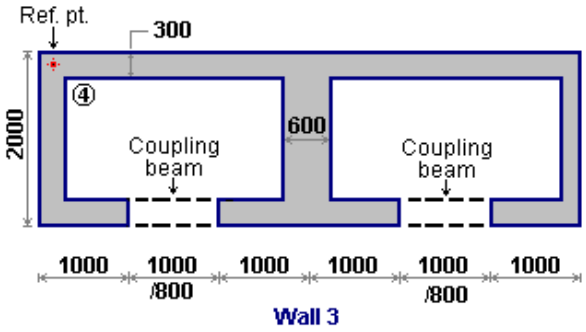


MX CONTOUR LINES COMB. NO. 1 1* 1.15 + 2*1.25 + 3*1.30

2.12 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 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**

- Double-click **DEMO-Empty model** in the model list.

- **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):

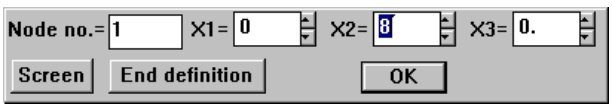
- click the 

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

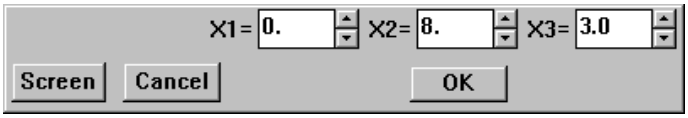
Copy the nodes to all 10 floors:

- 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 

- click the Isometric View icon  in the icon bar to display the entire model.

Define the wall sections:

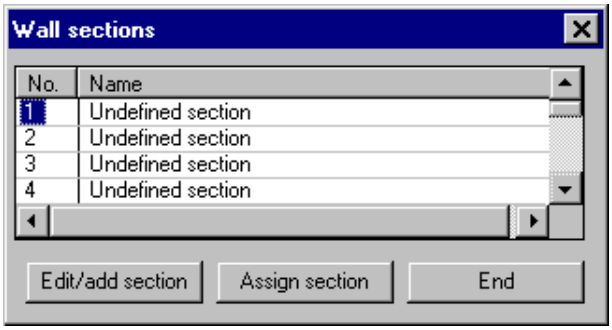
- Click the 

icon
- Click the 

icon
- Click the 

icon in the menu at the side of the screen

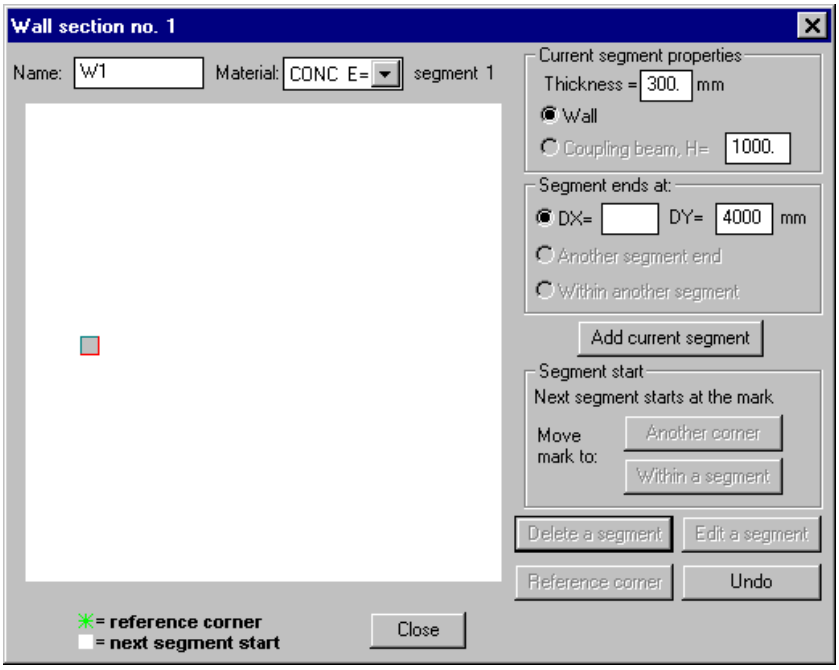
- Select the section:



← *Select section* **1**

← *Click* **Edit/add section**

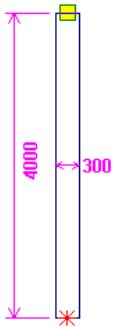
- Define the properties and dimensions of Wall 1; the wall consists of one segment only and we will define the segment end at DX=0, DY=400:



← *Specify thickness* = **300**

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

The segment is drawn in the center box as



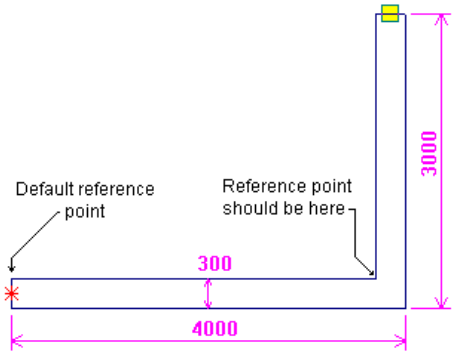
← *Click* **Close**

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:



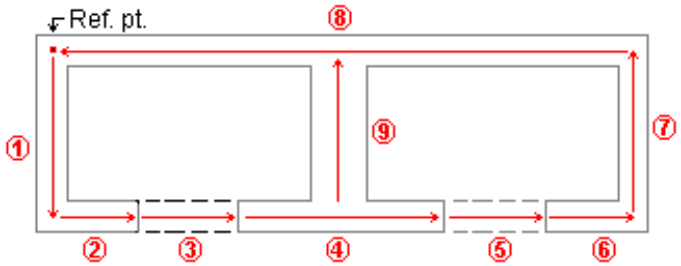
Note the "reference point" at the first corner denoted by the ✳. This is the point where the wall section will be attached to nodes in the structural model. Referring to the floor plan at the beginning of this example, it is apparent that Walls 2a/2b will be attached at the corner joining the two segments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

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

- Click **Close**

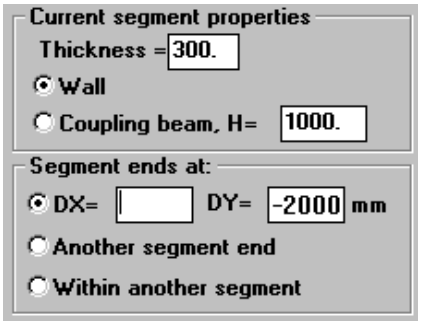
Define Wall 3 section:

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



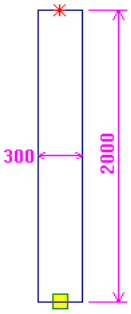
Note:
- segments 3 and 5 are coupling beams
- segment 9 bisects segments 4 and 8 and divides them each into two new segments.

- Click the section icon in the menu at the side of the screen
- Highlight section no . **3** and click **Edit/add section**
- Segment 1:

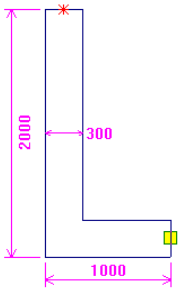


← *Specify thickness* = **300**

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



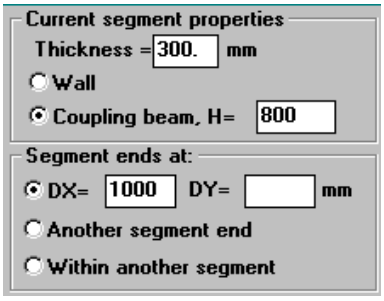
The segment is drawn in the center box:



- Segment 2:

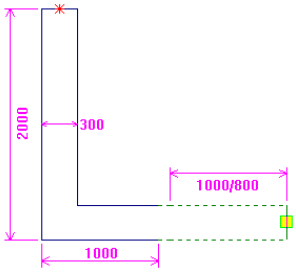
  Specify **DX = 1000**mm and press [Enter]. The segment is drawn:

- Segment 3 (coupling beam):

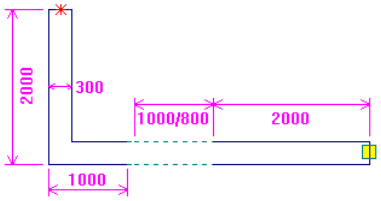


← *Specify* **Coupling beam H= 800** mm

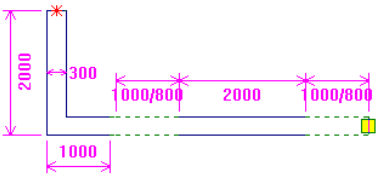
← *Specify* **DX = 1000**mm and press [Enter]



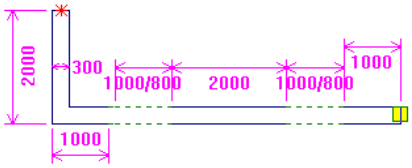
- Segment 4:
  Specify **Wall** and **DX = 2000**



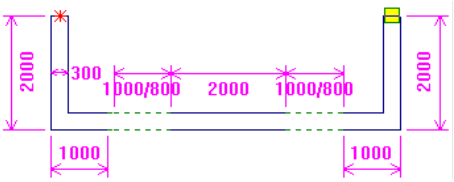
- Segment 5:
  Specify **Coupling beam** and **DX = 1000**



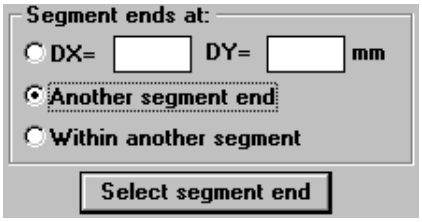
- Segment 6:
  Specify **Wall** and **DX = 1000**



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



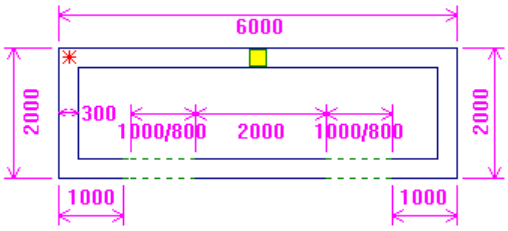
- Segment 8:
  This segment ends at the start of segment 1



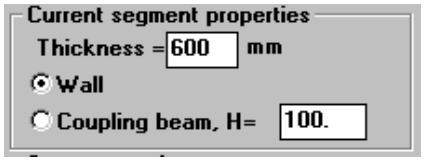
← *Specify* **Another segment end**

← *Click* **Select segment end**

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

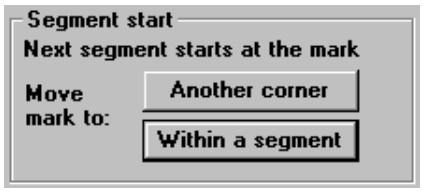


- Segment 9:

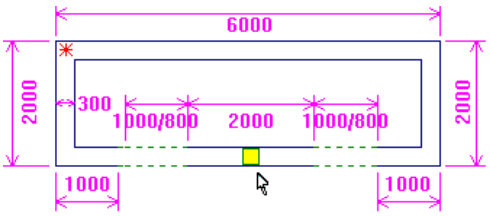


← *Specify* **Thickness = 600 mm**

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

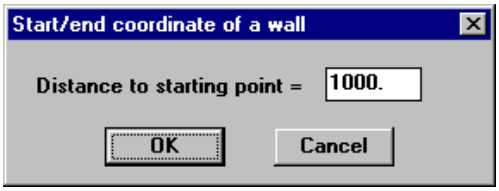


← *Click* **Within a segment**



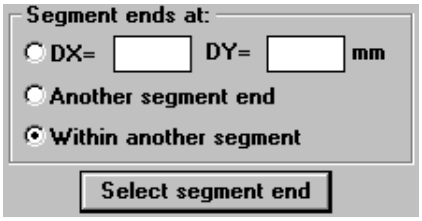
← *move the adjacent to segment 4 so that it is highlighted with the ■; click the mouse.*

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 

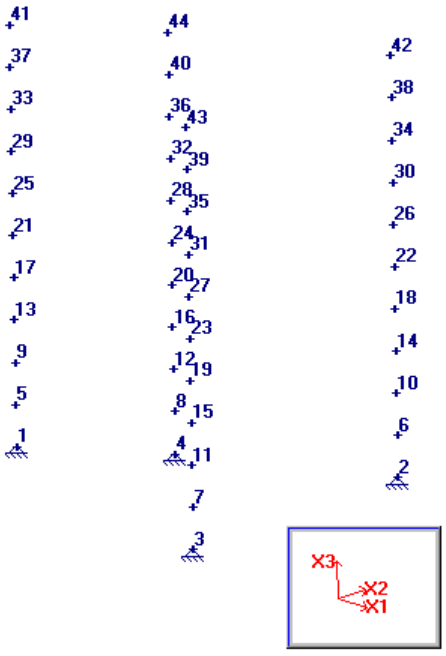
- select the wall section:

- 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;

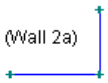
  The program adds the wall section to the line 1-41.

- 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.

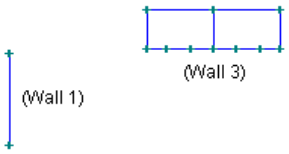


- attach section 2 to line 3-43:
  Specify **Sect =2** and select nodes 3 and 43
- attach section 3 to line 4-44:
  Specify **Sect =3** and select nodes 4 and 44
- click the node number icon, the Isometric View icon and the Restraints icon . The model is displayed on theX1-X2 without numbering or restraints:

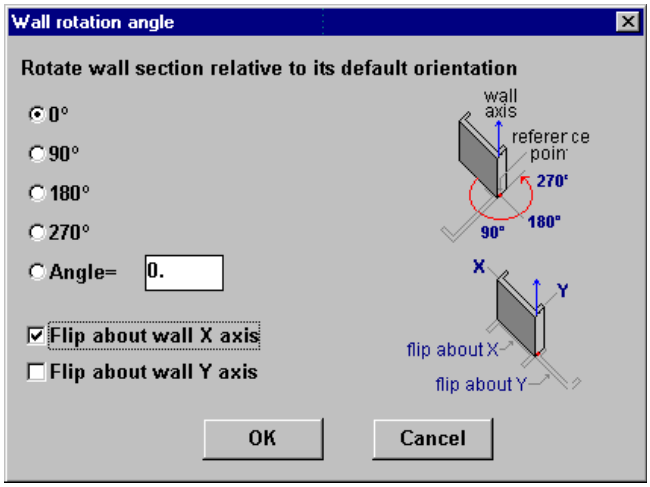
  All of the walls are oriented correctly except **Wall2a** (check the coordinates of the wall corners by clicking on the data Data icon)

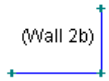


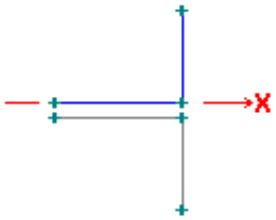
- Rotate Wall 2a:
  click the Isometric View icon
- click rotate icon



- 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 

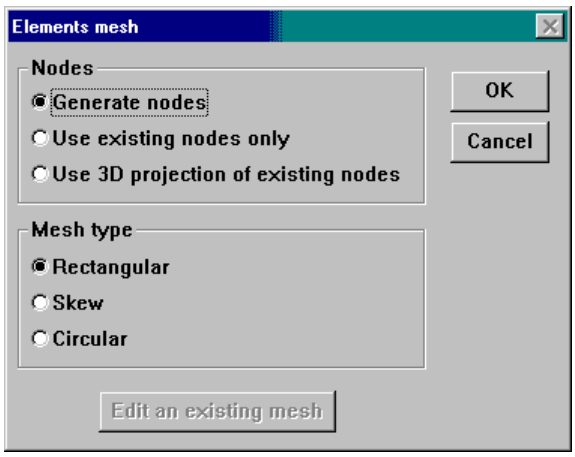
icon
- Type in the coordinates **X1 = 0.0, X2 = 0.0, X3 = 3.0** in the dialog box at the bottom of the screen
- Type in the coordinates **X1 = 0.0, X2 = 20.0, X3 = 3.0** in the dialog box

Define the element mesh:

- click the 

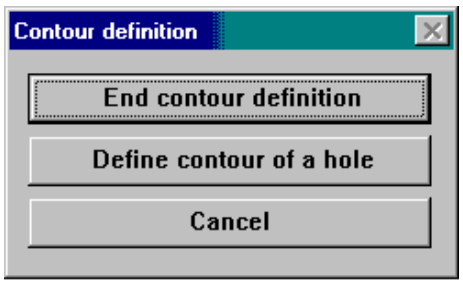
icon
- click the 

icon

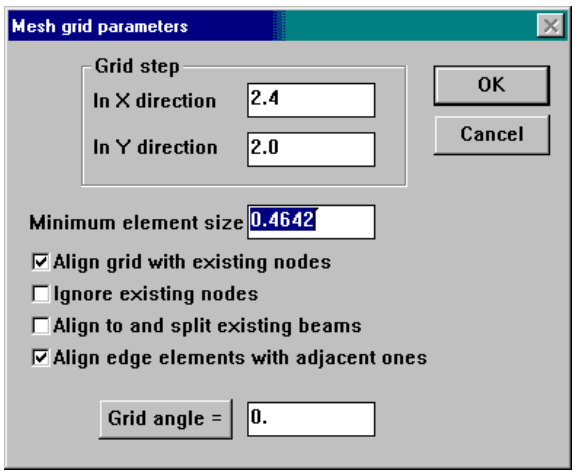


← *Click* **OK**

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



← *Select* **End contour definition**

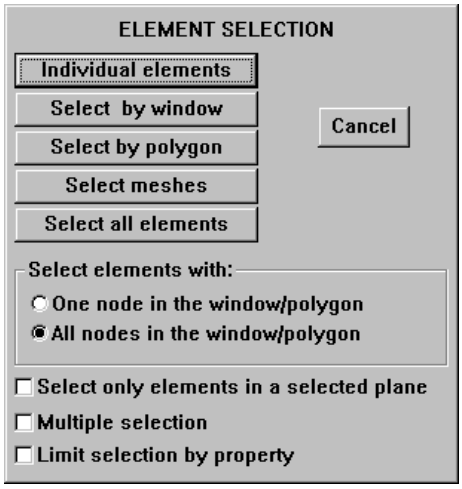


← *Specify* Grid steps: X = 2.4, Y=2.0 *and click* OK

- click 

in the Elements menu

- click **Dummy element**

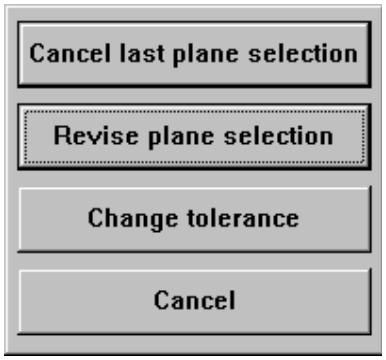


← **Select meshes**

- 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 

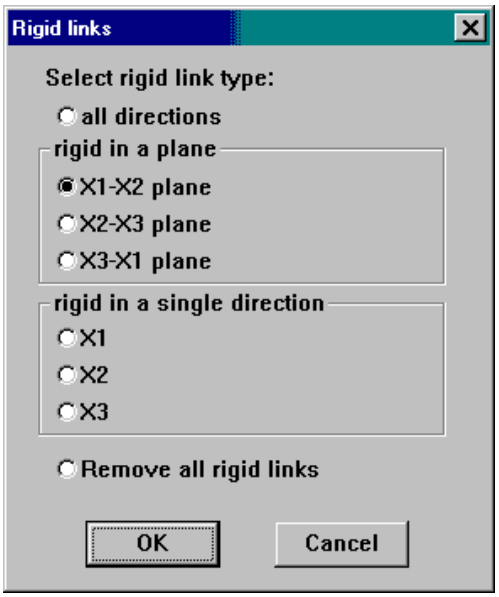
button
- Select level X3 as the height axis; select X3 = +3.00 and click 

- 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.

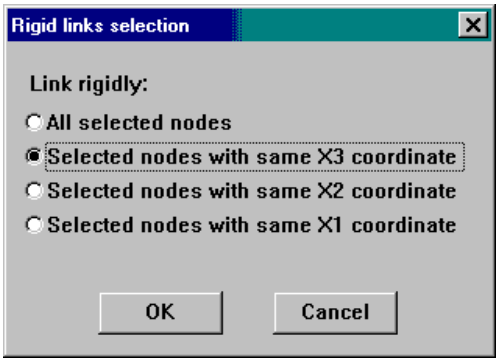
- Define the new location of the reference node:
- Click the **by coordinates** button
- Click and hold the left mouse button and drag the cursor into the dialog box at the bottom of the screen - the coordinate values in the box will not change as the cursor moves. Type the coordinates **X3=6.0**.
- 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, Y = 0, Z = 0**
- Click the rest-raints icon in the main menu.
- Click the rigid links 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. Refer to the next example for the dynamic analysis of this model.

2.13 Wall elements - dynamic analysis

This example is a continuation of Example 2.12. The files for this model contain the geometry as defined as well as the following static loads applied to all floor slabs on all levels:

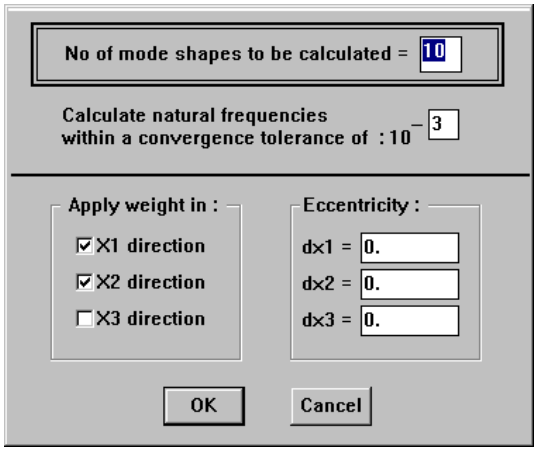
- Dead = 8.0 kN/m2
- Live = 1.5 kN/m2

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

Main Menu:

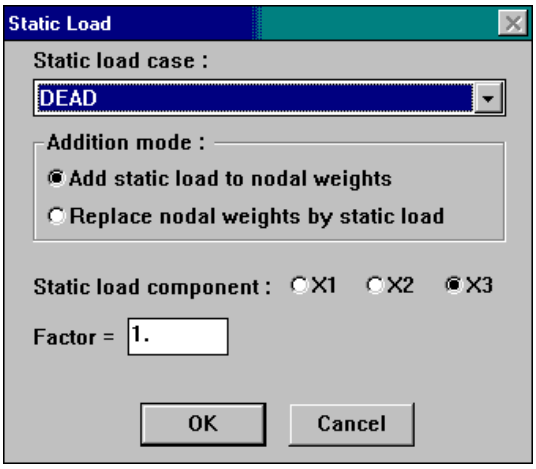
- highlight **DEMO 13 - WALLS - DYNAMIC** in the model list
- click on **Dynamics** in the toolbar and **Weight data** in the pulldown menu.
- click the 

icon

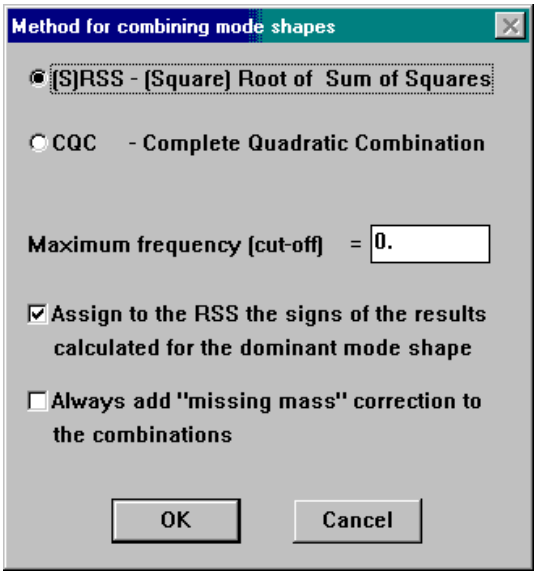


- click the 

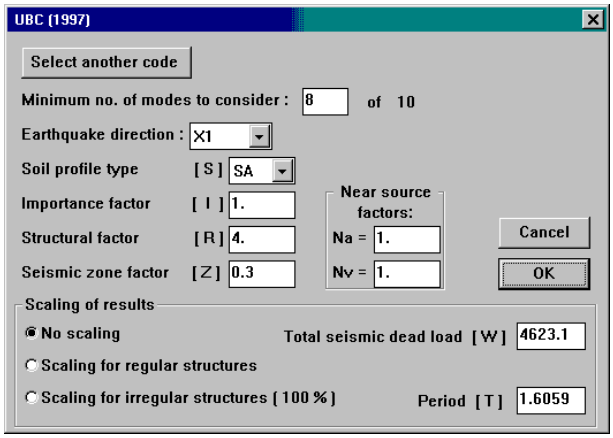
icon



- 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

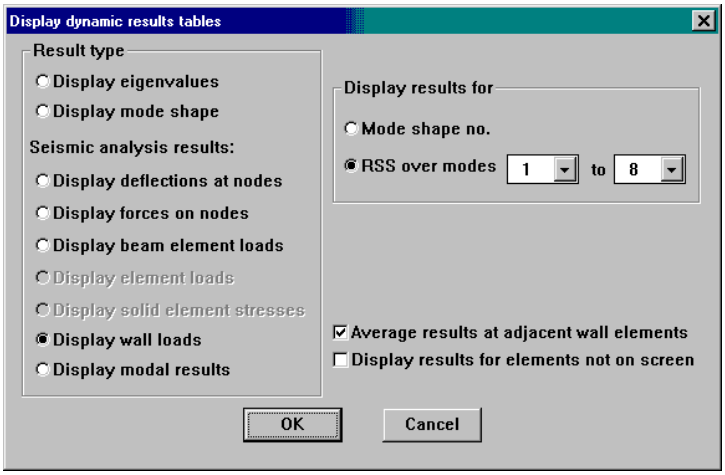


- Select **Seismic analysis** in the tool bar and **Parameters** in the pulldown menu



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

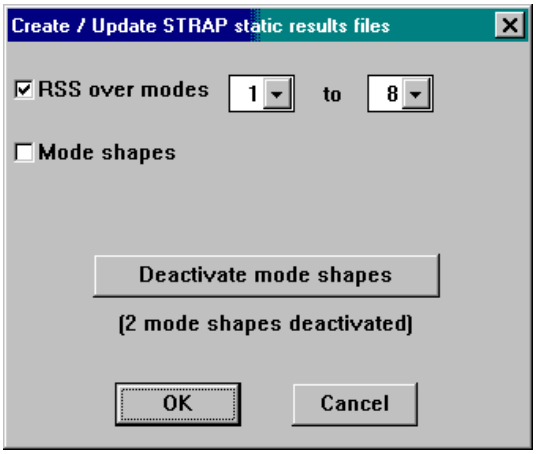
- click the **Tables** -  icon



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

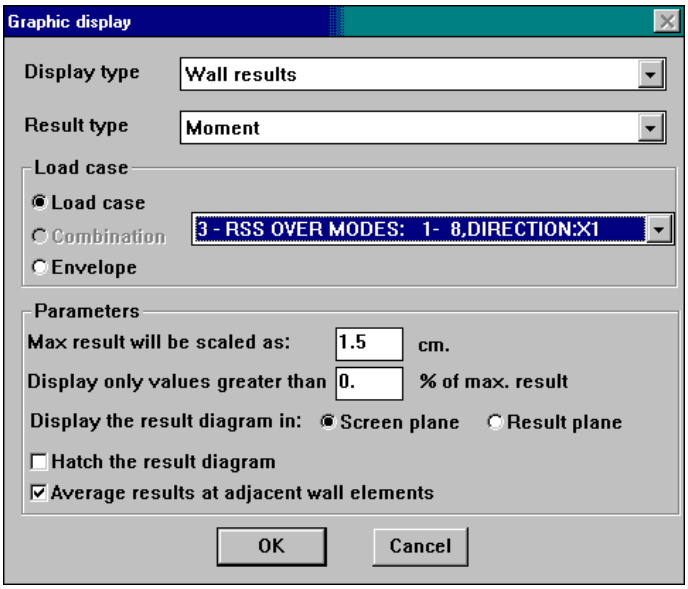
Add the dynamic results to the static results file:

- 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



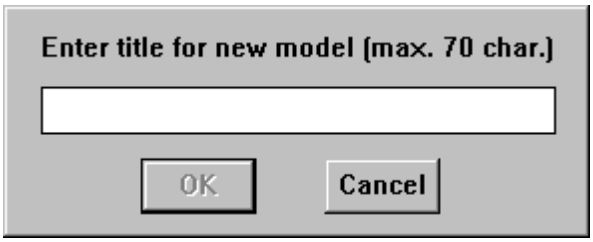
The moments in the walls are displayed.

3 Miscellaneous

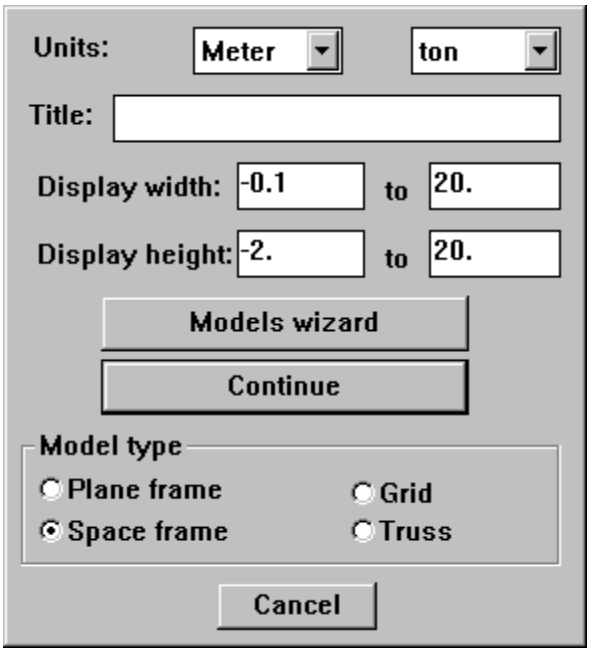
3.1 Defining new models

New models with up to 12 nodes may be defined, saved and solved.

- select **Files** in the menu bar
- select **New model** in the pull-down menu.
- Revise model title if necessary:



- arrange the preliminary geometry menu:



← *Select the default units*

← *Enter the model title*

← *click* **Continue** *to proceed to geometry*

← *Select the model type*

3.2 How to Use HELP

The windows version of *STRAP* provides the user with extensive on-line Help; the **entire User's Manual** is an integral part of the program and may be displayed at any time.

STRAP Help uses the standard Windows Help format; a window containing the Help is displayed on the right half of the screen.

There are two ways to call Help while running the program:

- press the  key: Help for the current option will be displayed.
- click the **Help** option in the menu bar at the top of the screen and select **Contents** in the pull-down menu: The Table of Contents of the full *STRAP* User's Manual is then displayed; select any topic in the list.