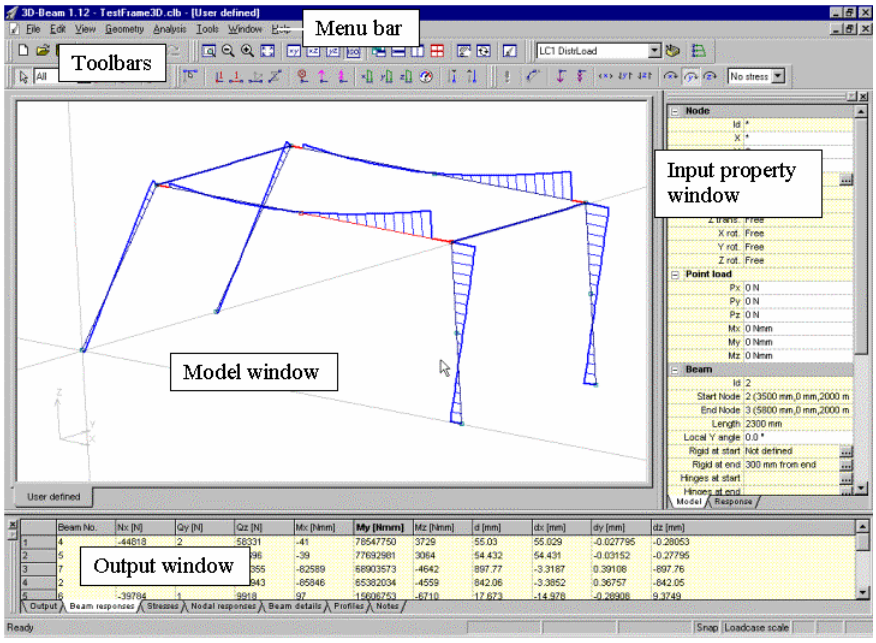
NAUTICUS 3D BEAM

HOW TO…



DET NORSKE VERITAS

1. Introduction

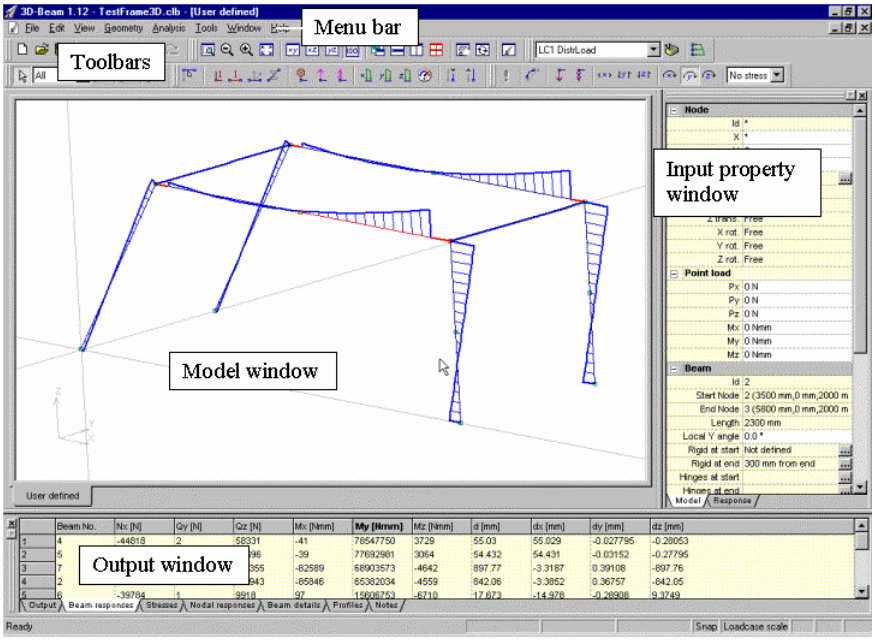
1.1. About this manual

The purpose of this manual is to provide a quick reference on how to accomplish certain tasks or to access various features of 3D Beam. For more detailed explanations of 3D Beam it is referred to the 3D Beam User Manual.

It is assumed that the user knows the basic laws and theories of static beam analysis.

1.2. Overview of 3D Beam

The figure below shows an overview of the graphical user interface of 3D Beam. It consists of five basically different interfaces: i) **Menu bar**, ii) **Toolbars**, iii) **Model window**, iv) **Input property window**, v) **Output window**. Each of the interfaces provides various features and functionality.



Graphical User Interface of 3D-Beam

1.2.1. Menu bar

The **Menu bar** gives access to all features and functions of 3D Beam.

1.2.2. Toolbars

The **Toolbars** gives easy access to all the most used features of 3D Beam.

1.2.3. Input property window

The **Input property window** is the user interface for applying properties to the nodes and beams in the model. In the **Input property window** the boundary conditions, loads, profile properties including material and orientation of the local beam axis, hinges and rigid ends may be defined.

The window also has a tab for *Response* which shows the results on the model after an analysis.

1.2.4. Output window

The **Output window** consists of more views which may be activated by clicking the tab cards at the bottom of the window. Information about the model and results from the analysis is made available in the **Output window**.

1.2.5. Model window

The **Model window** is used when creating a model. Any number of views may be opened.

2. Make a beam element analysis in 9 steps

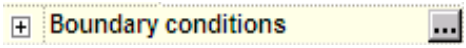
2.1. Specify model properties

Use the **Input property window** to enter the identification data for the **Model**.

2.2. Describe the geometry

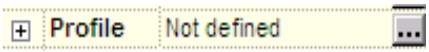
Either click to use the cursor to draw the model in the **Model window** or use the *Beam Wizard* to specify the geometry numerically.

2.3. Specify the boundary conditions

Select the desired node(s) and click 

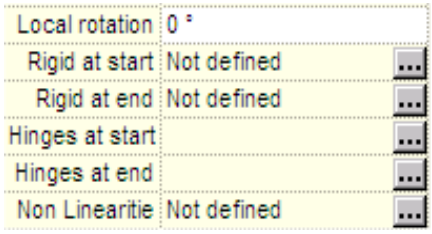
in the **Input property window** to specify the boundary condition(s). Click to toggle the display of boundary conditions on and off.

2.4. Specify the beam profile(s) and material(s)

Select the desired beam(s) and click 

in the **Input property window** to select a profile or create a new profile, with desired material, for the beam(s).

2.5. Specify special properties

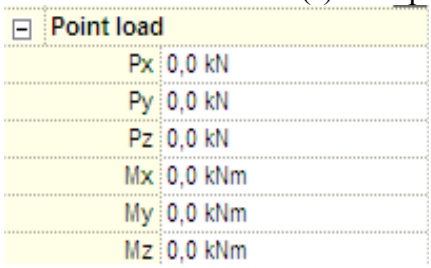


Click the appropriate button in the **Input property window** to specify possible features such as Beam rotation, Rigid ends, Hinges, or Non-linearities.

2.6. Specify the loads

2.6.1. Node loads

Select the desired node(s) and specify node loads in the **Input property window**



Click and to toggle the display of node loads.

2.6.2. Beam loads

Select the desired beam(s) and specify distributed loads or temperature loads in the **Input property window**

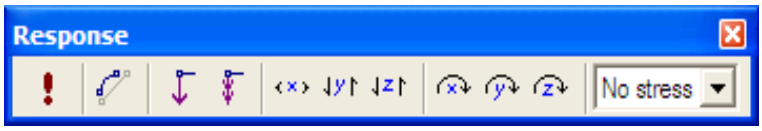
Distributed load	
px1	0,0 kN/m
py1	0,0 kN/m
pz1	0,0 kN/m
px2	0,0 kN/m
py2	0,0 kN/m
pz2	0,0 kN/m
Temperature load	
Gy	0 °C/mm
Gz	0 °C/mm
Temperature	0,0 °C

Click , , and to toggle the display of distributed loads and to toggle the display of temperature loads.

2.7. Run the analysis

Click the button to run the analysis. Watch the messages in the **Output window.**

2.8. Examine the results

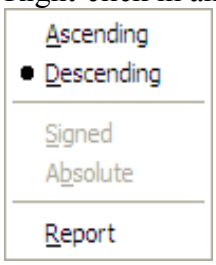
Click the buttons in the *Response toolbar* 

to toggle the display of the various responses.

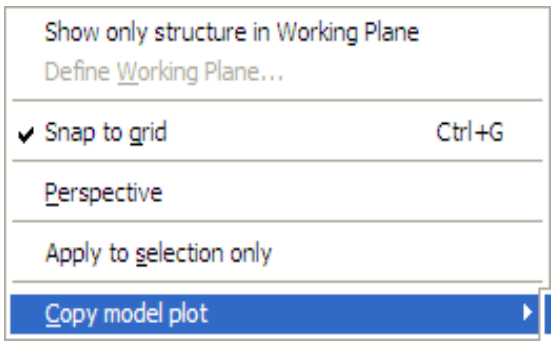
Click the various tabs in the **Output window** to examine the results numerically.

2.9. Create reports

Right-click in any of the tables in the **Output window** and select *Report* from the pop-up window.



The report is generated as an MS Word document. If desired, one or more copies of the model in the **Model window** may be included in the report. Right-click in the **Model window** and select



Copy model plot to the clipboard, and then paste this into the Word document.

3. Create a model

3.1. Draw a model

Click the *Create beams* button  in the toolbar. The cursor changes to a cross.

The cursor position is shown on the status bar at the bottom of the screen. Keep the left mouse button depressed and drag to draw the beam. Release and click at the end position of each beam.

Nodes are created at the end of each beam and at the intersection of beams. To prevent creation of nodes at intersection of beams, keep the Ctrl-key depressed while dragging.

3.1.1. Activate/deactivate snap to grid

Menu: View | Snap to Grid

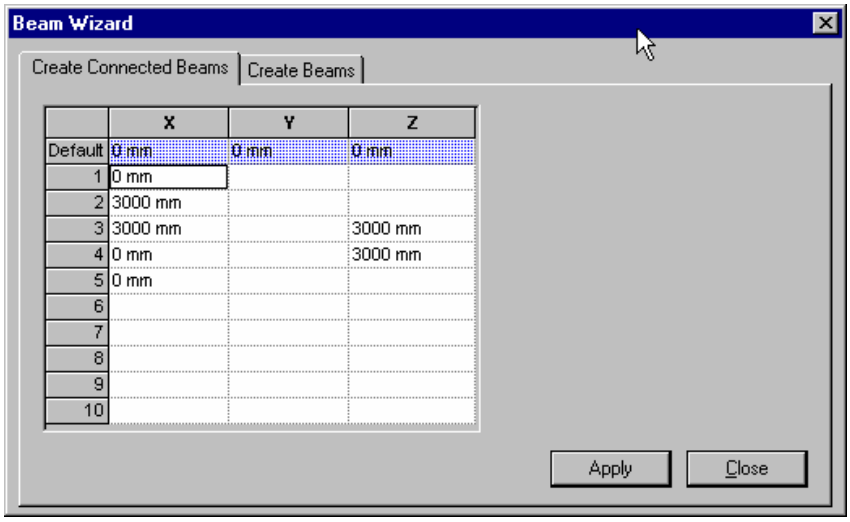
Shortcut Ctrl+G

3.2. Create a model by specifying node coordinates with the Beam Wizard

Click the *Beam Wizard* button  on the toolbar to bring up the *Beam Wizard* dialog.

3.2.1. Specify a set of connected beams

Select the tab *Create Connected Beams* and specify the coordinates for each node forming a continuous set of beams. **Note:** if a field is left blank, the value above in the same column is used.

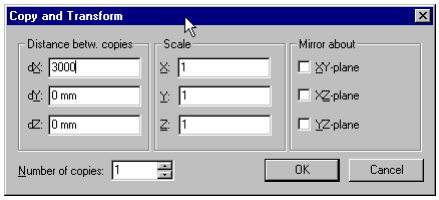


3.2.2. Specify individual beams

Select the tab *Create Beams* and specify the start and end coordinates for each beam. **Note:** if a field is left blank, the value above in the same column is used.

3.3. Copy beams

Select the desired beam(s) and click the *Copy and Transform* button  in the toolbar to bring up the *Copy and Transform* dialog:



to specify the copy parameters.

3.4. Move/Mirror beams

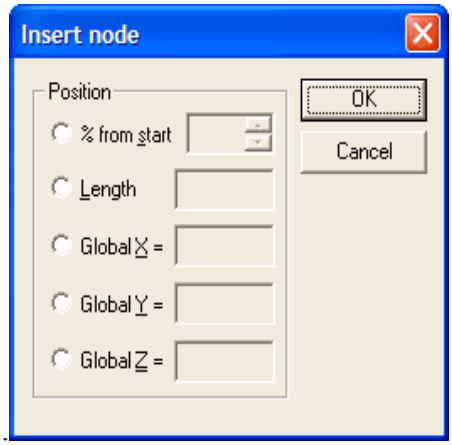
Select the desired beam(s) and click the *Apply transform* button  in the toolbar to specify the transfer parameters.

3.5. Delete beams

Select the desired beam(s) and click the *Delete* button  in the toolbar.

3.6. Insert node/Split beam

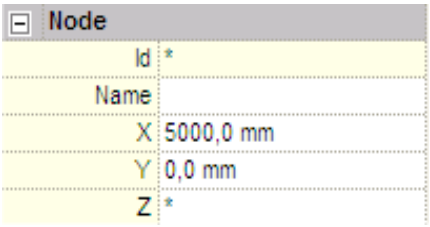
Select the desired beam(s) and click the *Insert node* button  in the toolbar.



3.7. Edit node coordinates

Select the desired node(s).

In the Input Property Window, below the **Node** heading, enter the desired X, Y, and Z coordinates for the node(s).



Note: in the **Input property window** properties common to the selected items are shown. If there are different values an asterix is shown.

4. Make selections of nodes and beams

4.1. Select nodes and beams in the Model window with the cursor

By default, beams and nodes turn red when selected. Most default colours in the **Model window** may be changed by the user in the menu Tools | Options…

4.1.1. Select all beams in the model

Menu: Edit | Select All,

or the shortcut: Ctrl+A,

or enclose the whole model by a "rubber-band".

4.1.2. Un-select all beams

Click the left mouse button anywhere in the **Model window**, away from the model.

4.1.3. Adding/removing beams to/from a selection (One by one)

Click on the beam with the left mouse button while keeping the Shift-key depressed.

4.1.4. Select/un-select beams within an area

Press the left mouse button and drag the rubber band around the beams.

4.1.5. Select/un-select beams touched by the rubber band (Ctrl-key)

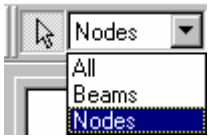
Drag the rubber band across the beams, while keeping Ctrl-key depressed.

4.1.6. Select/un-select beams with the "touch"-method.

Press both the Ctrl-key and the Shift-key while touching the beams to be selected with the rubber band.

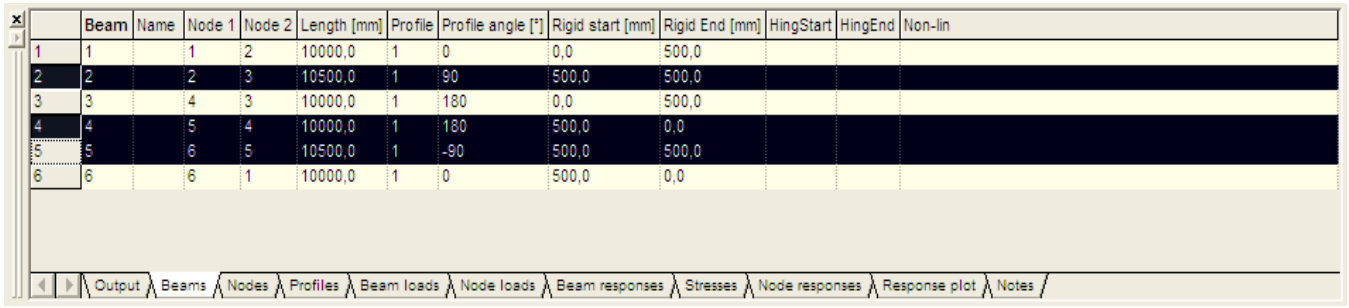
4.1.7. Select nodes or beams only

Use the selection filter, which is located next to the *Select* tool in the toolbar.



4.2. Select nodes and beams in the Output window

Click on the line number to the left of the first column to select the beam or node. Using the Shift- and/or Ctrl-key while clicking to extend the selection:



	Beam	Name	Node 1	Node 2	Length [mm]	Profile	Profile angle [°]	Rigid start [mm]	Rigid End [mm]	HingStart	HingEnd	Non-lin
1	1		1	2	10000,0	1	0	0,0	500,0			
2	2		2	3	10500,0	1	90	500,0	500,0			
3	3		4	3	10000,0	1	180	0,0	500,0			
4	4		5	4	10000,0	1	180	500,0	0,0			
5	5		6	5	10500,0	1	-90	500,0	500,0			
6	6		6	1	10000,0	1	0	500,0	0,0			

Output | Beams | Nodes | Profiles | Beam loads | Node loads | Beam responses | Stresses | Node responses | Response plot | Notes

4.3. Use Named Selections

In large models it can be a tedious task to select desired beams, and it may be useful to save a selection for later use.

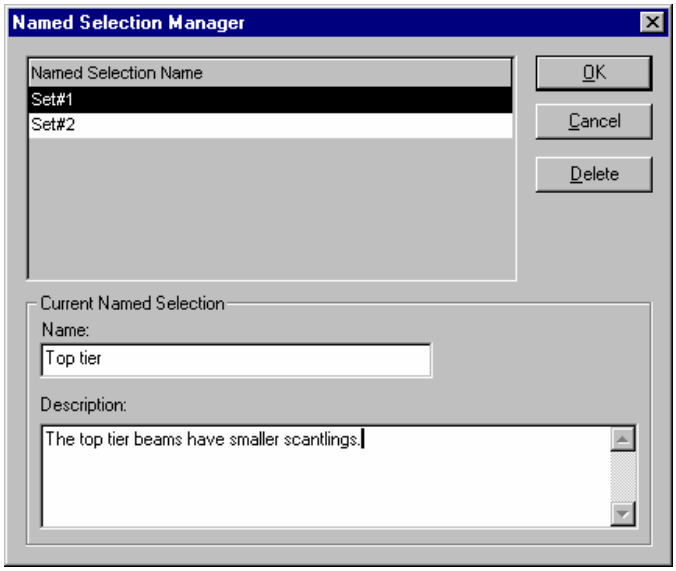
4.3.1. Save a selection

Select desired node(s) and/or beam(s) and Click the button *Create new named selection* on the *Named Selection* toolbar 

. The selection is given a default name.

4.3.2. Change the default names of selections

Click the button  to bring up the *Named Selection Manager*:



Where the selections can be given more informative names

4.3.3. Activate a selection

Choose selection from the drop-down list.

4.4. Show features on selected beams only

A large model can be clobbered and difficult to see. Click  to see features of selected beams only.

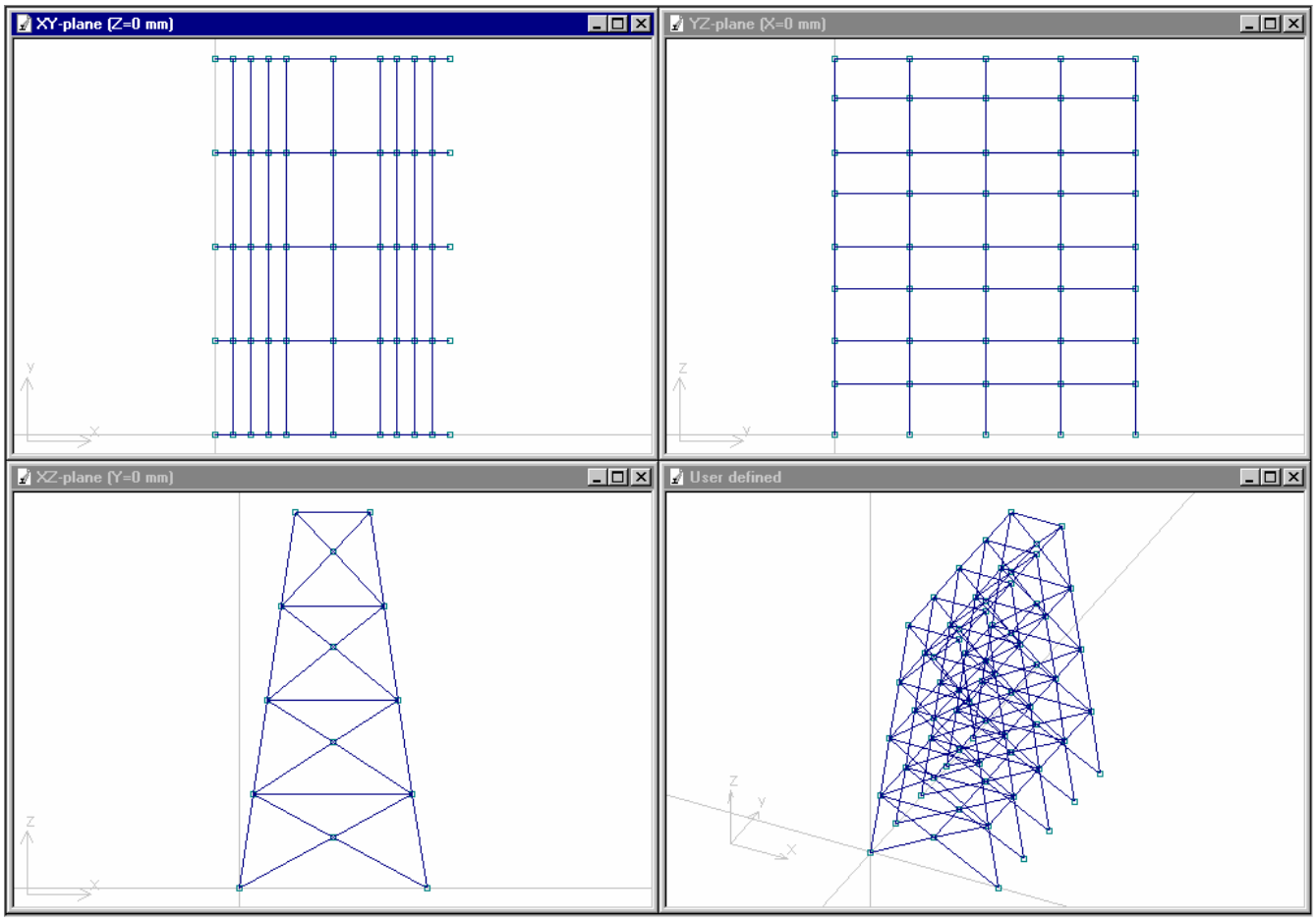
5. Work in 3D

5.1. Use Views in the Model Window

By default a new model is displayed in the XZ-plane. To change view click on one of the three projections XY, XZ, YZ , or the ISO view:

5.2. See 3 projections and an ISO view

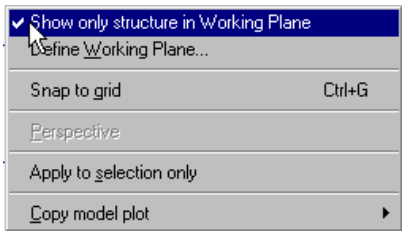
Click the button 



Click in a window to select it. The frame of an active window is blue.

5.3. Use Working Planes

By default the projection of the model is "indefinitely deep". To see only the part of the model within the Working Plane, right-click in the desired window to bring up the pop-up menu:

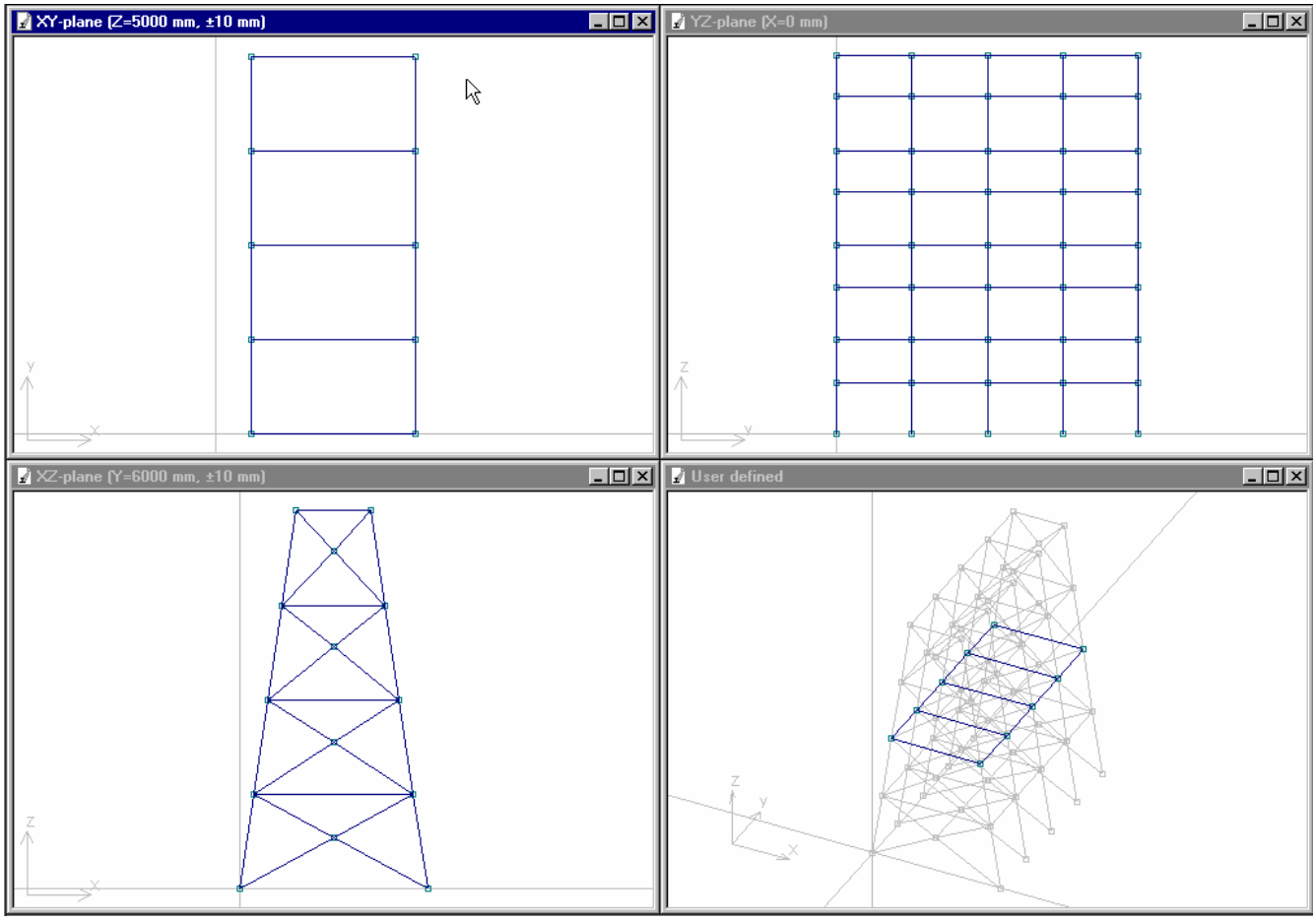


Click on the first item *Show only structure in Working Plane*.

The projection now contains only the structure within the Working Plane.

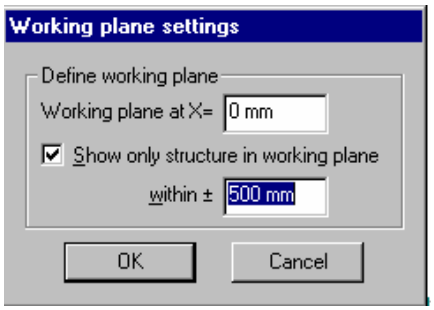
To move to the next working plane, press the Tab-key.

- This feature is best seen if 3 projections and an ISO view are selected, see the figure below.



5.3.1. Set the depth of the Working Plane

Right-click in a projection window and select *Define Working Plane…*, which brings up the dialog:



where the depth of the Working Plane may be set.

5.4. Zoom the model

The buttons  are available for zooming in any one window.

5.5. Move the model

To move the model click the button  and move the model with the cursor.

5.6. Rotate the model.

Click the button  to rotate the model in the *User defined* view and the *Solid view* only.

5.7. Add additional views

To open a new window click the button  and select the projection/view. Arrange the windows in standard Windows manner by clicking the buttons 

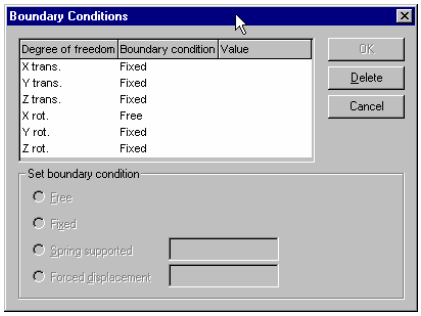
5.8. Open a solid view

To view the model in solid mode click the button 

Note: The model can be viewed in solid mode in one window only. Selection of beams or nodes is not possible.

6. Define boundary conditions

Select the desired node(s) and click the *Boundary conditions* open button  , located in the **Input property window** in the **Boundary Conditions** field, in the **Nodes** section:



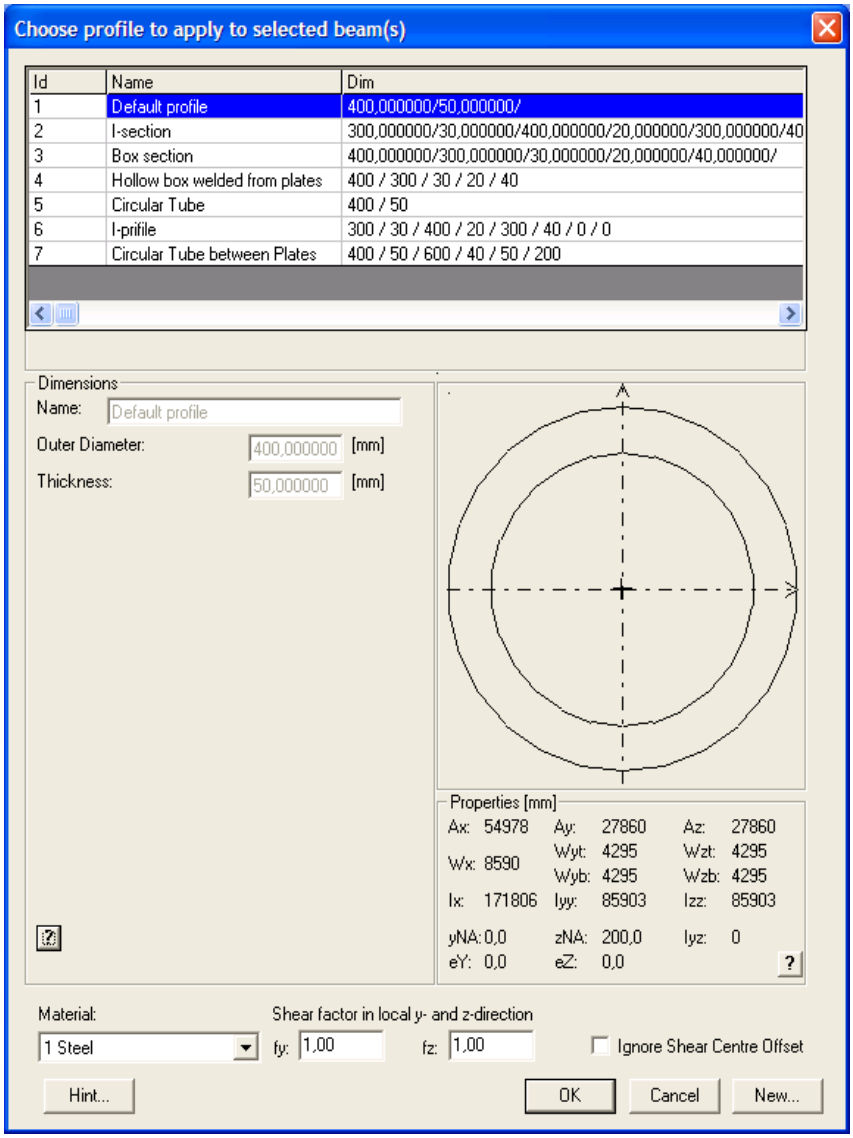
Specify the relevant boundary conditions in the dialog box.

Click the button  in the toolbar to verify the boundary conditions visually.

7. Define and apply profiles and material

7.1. Apply a profile to beams

Select the desired beam(s) and click the *Profile open* button located in the **Input property window** in the **Profile** field, in the **Beam** section to bring up the *Profiles* dialog:



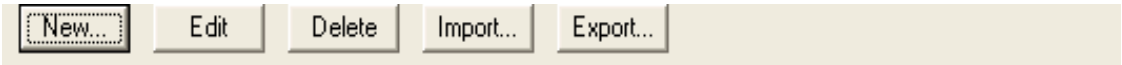
Click on the desired profile in the list.

Select material from the drop-down list and specify the additional parameters.

Note: Click the *Hint...* button for instructions.

7.2. Create a new profile

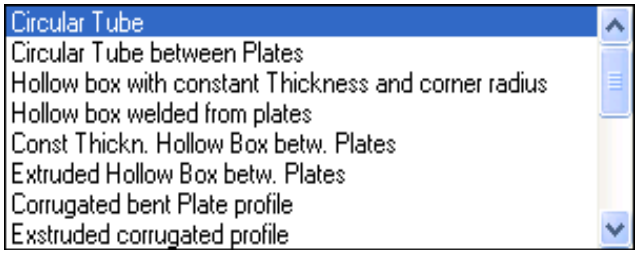
In the toolbar:



Click *New...* to enable the selection from a list of profile types or from tables of manufactured profiles:

7.2.1. Specify a profile from a list of profile types

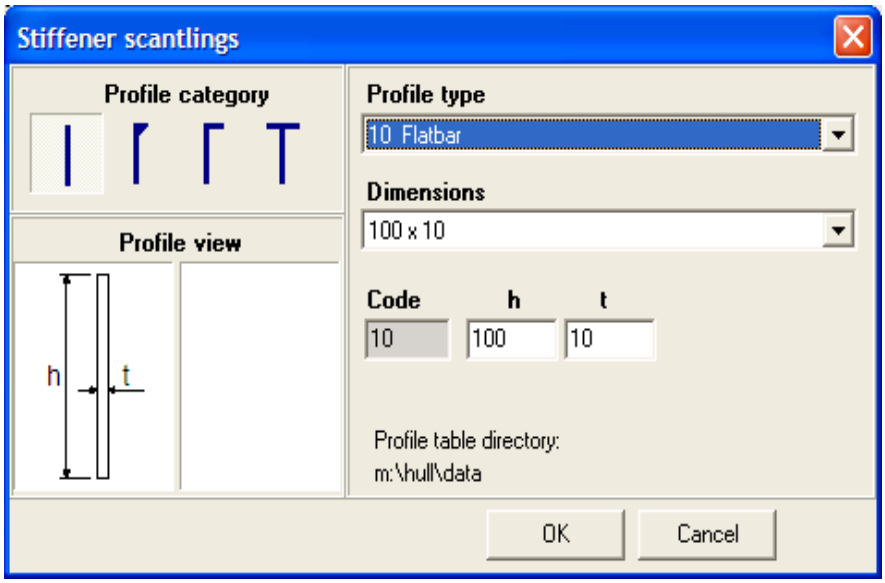
Click the drop-down button and click on the desired profile type:



Enter the dimensions of the profile and click *Add* to add the new profile to the list of selectable profiles.

7.2.2. Select a profile from tables of standard profiles

Click the *Select Standard Profile ...* to bring up the selection dialog:

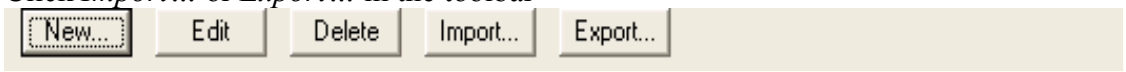


Select the profile type and dimension.

Edit the dimensions of the profile if necessary and click *Add* to add the new profile to the list of selectable profiles.

7.2.3. Import/Export profiles

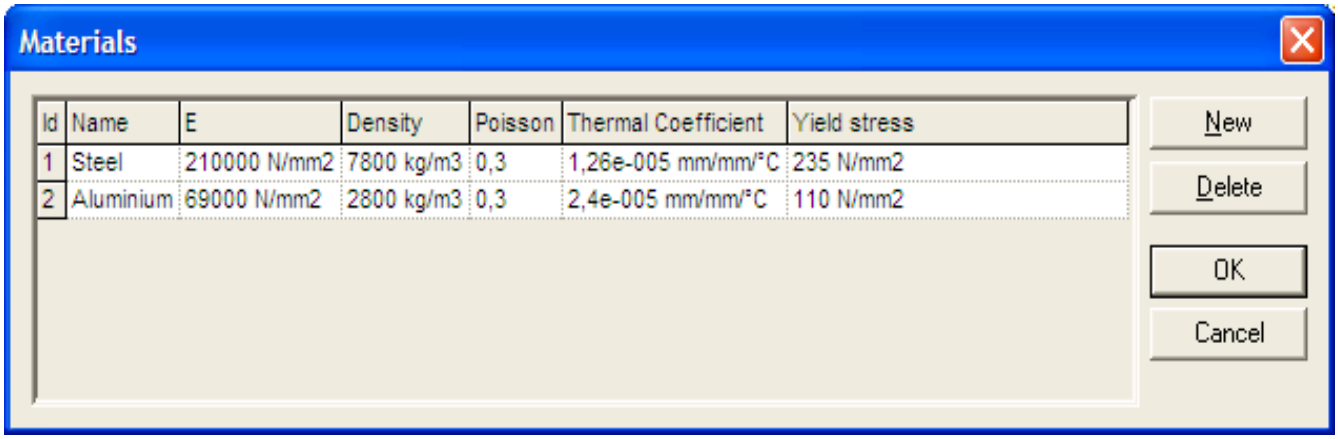
Click *Import…* or *Export…* in the toolbar



to re-use profiles between models.

8. Define new materials

Click the *Material library* open button ... located in the **Input property window** at the bottom of the **Model** section to bring up the *Materials* dialog:

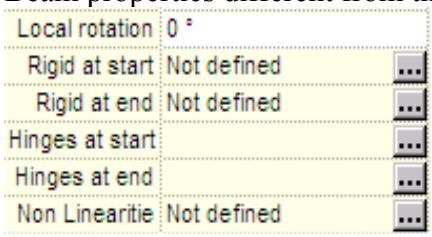


Steel and aluminium are available by default. Click *New* to specify new materials.

The materials will be available in a drop-down list when a profile is selected, see **Apply a profile to beams.**

9. Specify extra beam properties

Beam properties different from the default values may be specified in the **Input property window:**



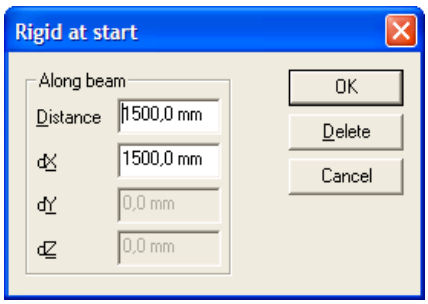
Click the appropriate button to specify Beam rotation, Rigid ends, Hinges, and Non-linearities.

9.1. Define the beam rotation

By default the beam is rotated such that the beam's local z-axis is as parallel to the global Z-axis as possible. This corresponds to a beam rotation equal to zero. Enter a value for *Local rotation* to change this. Click  to toggle the display of the beams' local coordinate axes.

9.2. Define rigid ends

Select the desired beam(s) and click the *Rigid at start/end* open button  to bring up the *Rigid at start/end* dialog:



Specify the necessary parameters.

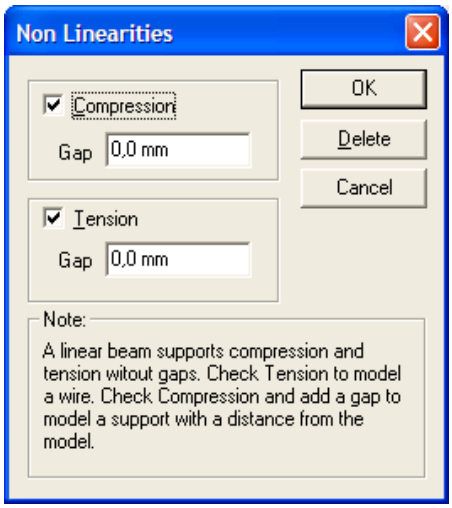
9.3. Define hinged ends

Select the desired beam(s) and click the *Hinges at start/end* open button  to bring up the *Hinges at start/end* dialog:

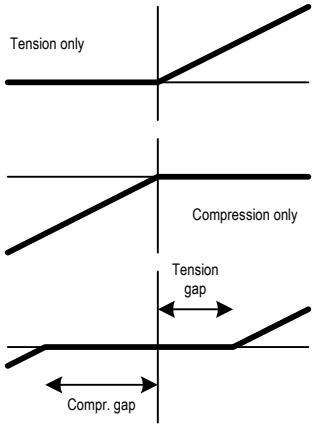
Specify the necessary parameters.

9.4. Define non-linear beams

Select the desired beam(s) and click the *Non Linearities* open button to bring up the *Non Linearities* dialog:



The effects of these parameters are illustrated in the tension-strain diagrams below:



10. Define and apply loads

10.1. Apply point loads

Select the desired node(s) and specify node loads in the **Input property window**

Point load	
Px	0,0 kN
Py	0,0 kN
Pz	0,0 kN
Mx	0,0 kNm
My	0,0 kNm
Mz	0,0 kNm

Click  and  to toggle the display of node loads.

10.2. Apply distributed loads

Select the desired beam(s) and specify distributed loads according to the alternatives below.

Click  ,  , and  to toggle the display of distributed loads and  to toggle the display of temperature loads.

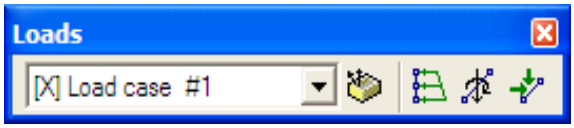
10.2.1. Specify by direct input

Specify distributed loads or temperature loads in the **Input property window**

Distributed load	
px1	0,0 kN/m
py1	0,0 kN/m
pz1	0,0 kN/m
px2	0,0 kN/m
py2	0,0 kN/m
pz2	0,0 kN/m
Temperature load	
Gy	0 °C/mm
Gz	0 °C/mm
Temperature	0,0 °C

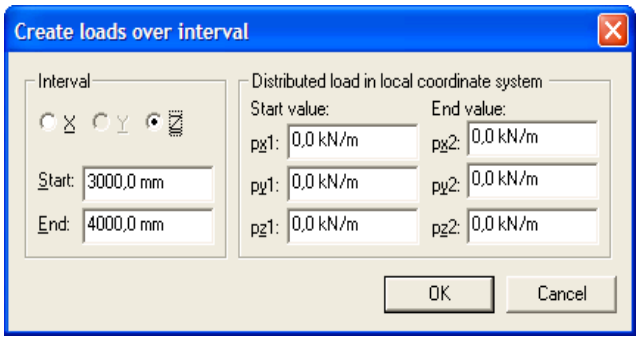
10.2.2. Using Load Wizards

Activate the load wizards from the Loads toolbar



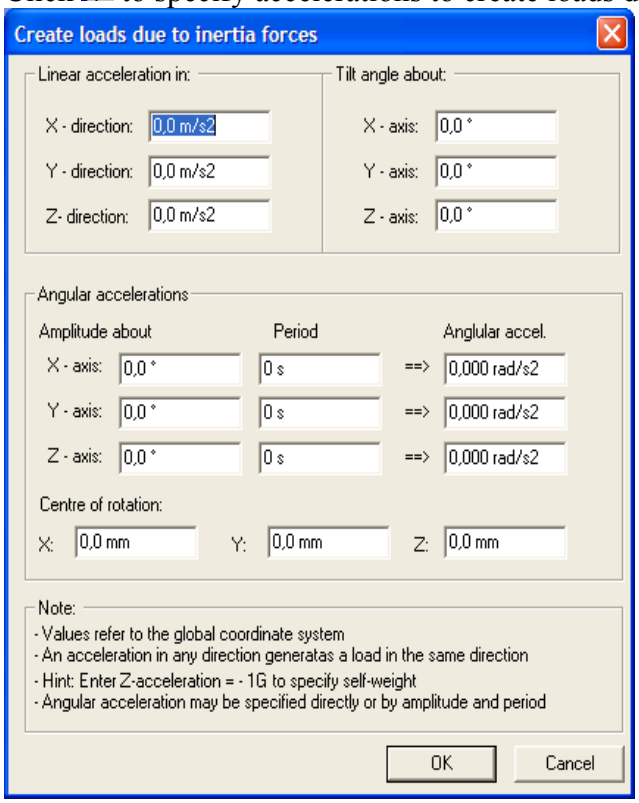
10.2.2.1. Create loads over selection

Click  to specify the load intensity at each end of the selected beams in the local coordinate system:



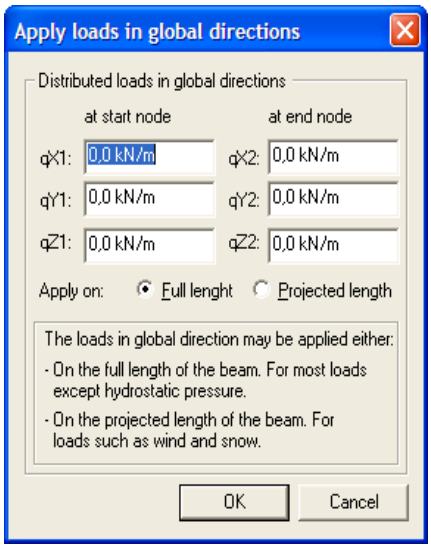
10.2.2.2. Create inertia loads

Click to specify accelerations to create loads due to inertia:



10.2.2.3. Apply global loads

Click to specify the load intensity at each end of the selected beams in the global coordinate system:



10.3. Apply temperature loads

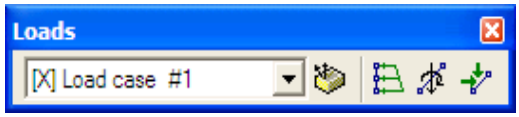
Enter temperature loads in the table

Distributed load	
px1	0,0 kN/m
py1	0,0 kN/m
pz1	0,0 kN/m
px2	0,0 kN/m
py2	0,0 kN/m
pz2	0,0 kN/m
Temperature load	
Gy	0 °C/mm
Gz	0 °C/mm
Temperature	0,0 °C

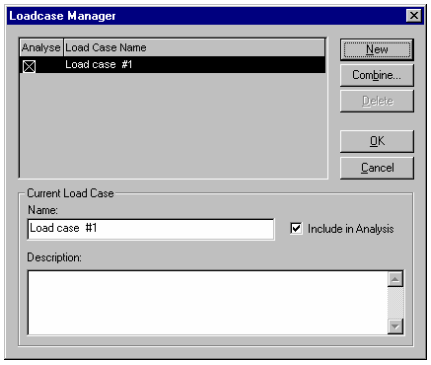
A Temperature different from 0 results in an axial load.

Gy and Gz are temperature gradients in the local coordinate system and values different from 0 results in a bending load.

11. Manage Loadcases

Click the *Manage Load Cases* button  in the *Loads* toolbar 

to bring up the *Loadcase manager*:



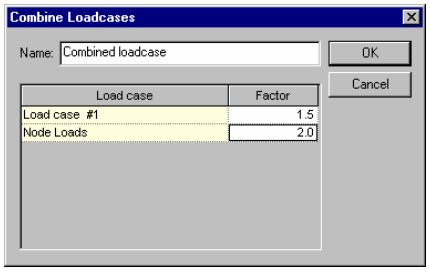
11.1. Specify a new loadcase

Click *New* and specify a name for the new load case and optionally a description.

Click Ok. This will be the current empty loadcase.

11.2. Combine loadcases

Click *Combine…* to bring up the *Combine Loadcases* dialog with a list of the available loadcases:

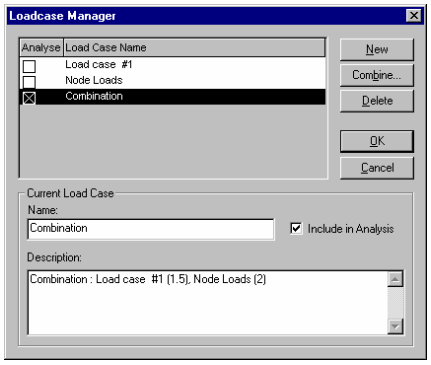


Specify a name for the loadcase.

Specify a multiplication factor for each loadcase to be part in the combination.

Click Ok.

The combined loadcase is created and a description is generated automatically:



Note: A combined loadcase does not keep a link to the underlying loadcases. This means that a change to an underlying loadcase does **not** affect the combined loadcase.

11.3. Exclude a loadcase from the analysis

The "basic" loadcases, on which the combined loadcases are based, may be excluded from the analysis by un-ticking the *Include in Analysis* flag.

11.4. Activate a loadcase

Click the arrow button in the *Loads* toolbar to select from the defined loadcases.

12. Analyse the model

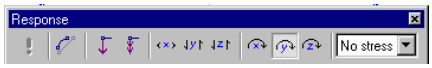
Select *Analysis* in the **Menu bar**, or click the *Analyse model* button  in the *Response* toolbar, or press F9.

Observe the messages in the **Output window**.

13. Examine the results

13.1. Display results in the Model Window

Click the various buttons in the *Response* toolbar to toggle the display of the various responses.



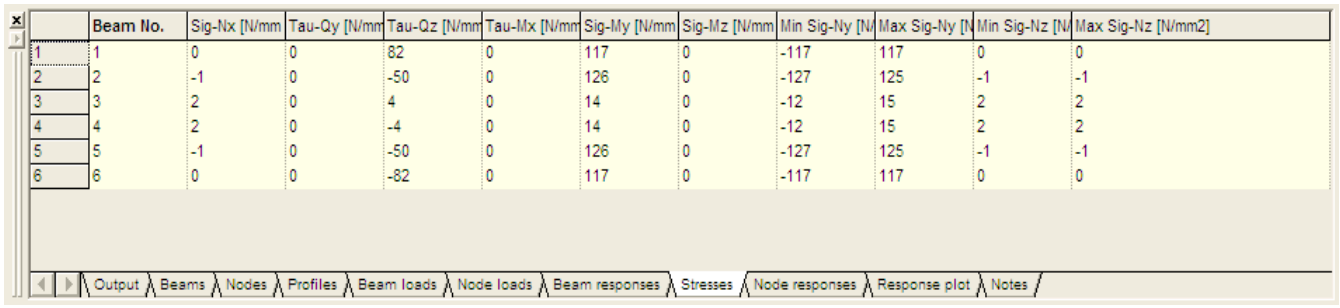
13.1.1. Adjust the display scaling of responses

The largest amplitude of the displayed responses may be scaled either by Loadcase or relative to all loadcases. Specify desired scaling in the *Menu bar View Scale per Loadcase* or |*Scale Globally*.

To adjust the plotted amplitudes click the buttons: 

13.2. View the results in tabulated form

The **Output window** holds grids for Beam responses, Stresses, Node responses, and an area for Response plot.



	Beam No.	Sig-Nx [N/mm	Tau-Qy [N/mm	Tau-Qz [N/mm	Tau-Mx [N/mm	Sig-My [N/mm	Sig-Mz [N/mm	Min Sig-Ny [N/	Max Sig-Ny [N	Min Sig-Nz [N/	Max Sig-Nz [N/mm2]
1	1	0	0	82	0	117	0	-117	117	0	0
2	2	-1	0	-50	0	126	0	-127	125	-1	-1
3	3	2	0	4	0	14	0	-12	15	2	2
4	4	2	0	-4	0	14	0	-12	15	2	2
5	5	-1	0	-50	0	126	0	-127	125	-1	-1
6	6	0	0	-82	0	117	0	-117	117	0	0

13.2.1. Sort the results in the grid

Click the right mouse button in the column heading of a response to bring up the pop-up menu:

Specify sorting preferences.

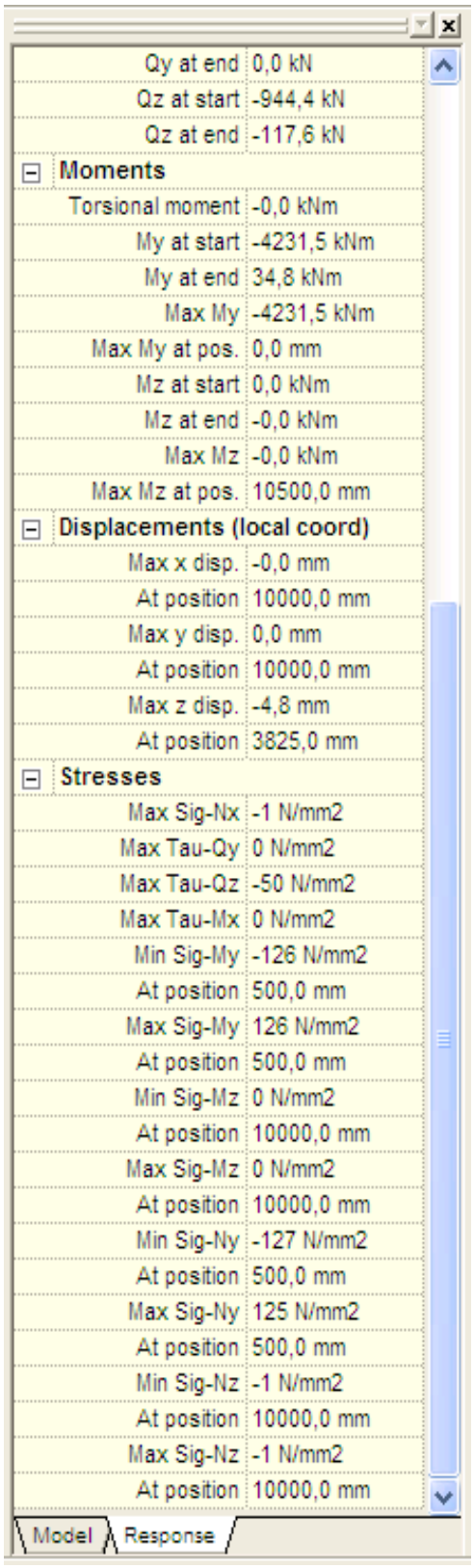
Double-click on another column heading to sort on that column according to the same criteria.

13.2.2. Display the response for selected beams only

Select the desired beam(s) and click the *Apply to selection only* button  in the toolbar. Properties and results are now available only for the selected beams.

13.3. View results in the Response Property Window

To view more detailed results for a single beam or node, select a node or a beam and click the *Response* tab in the **Input property window**.



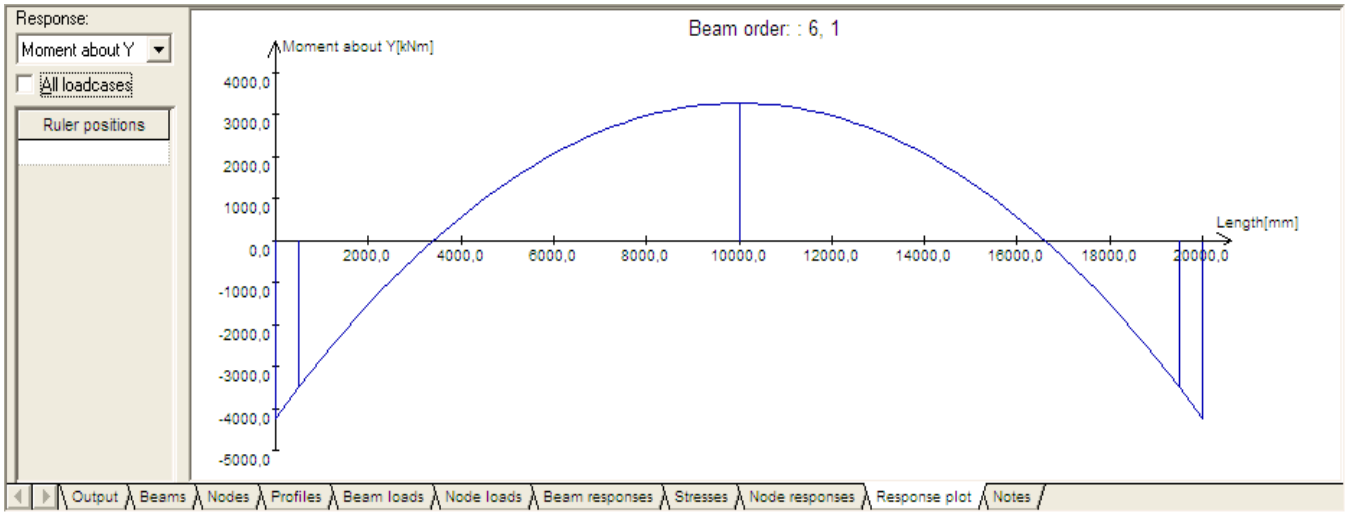
Note: the displacements for a beam are presented in the beam's local coordinate system, whereas in the **Output window** it is in the global coordinate system.

13.4. View detailed response plots

Click the *Response plot* tab in the **Output window**.

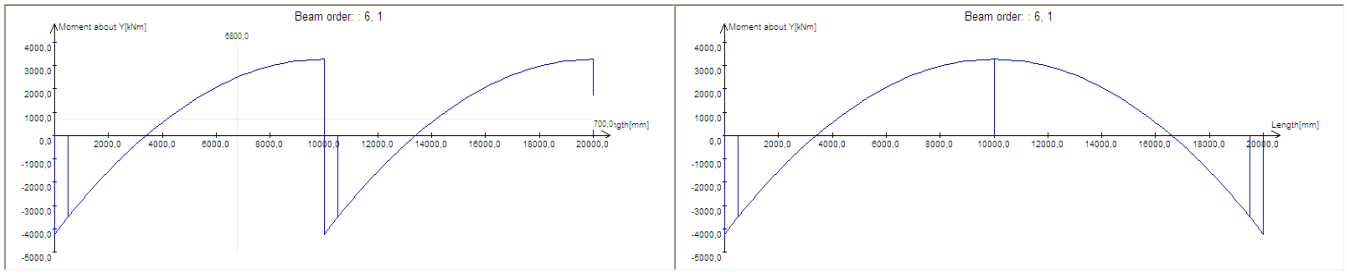
Select the desired beam(s).

Select the desired response from the drop-down list.



13.4.1. Flip direction of beams

Each beam is plotted from its start node to its end node. This means that plots of connected beams may appear confusing. To correct this, the direction of one or more beam(s) can be changed. Select the desired beam(s) and click the *Flip Beam* button  on the *Geometry* tool bar and run the analysis again. Below figure shows the plot before and after flipping the direction of beam no 1.



Note: The order of the plotted beams may be important for the appearance of the plot. Hence, avoid selecting beams by the "rubber-band" as this creates an arbitrary order of beams. Instead select the beams in desired order with the Shift+Click method, see **Select nodes and beams in the Model window with the cursor**.

14. Create reports

Report may be generated from the tabs in the **Output window**.

14.1. Create reports from the tables

Right-click in the table and select *Report* in the menu.

MS Word is started and a report is generated with bases in the table.

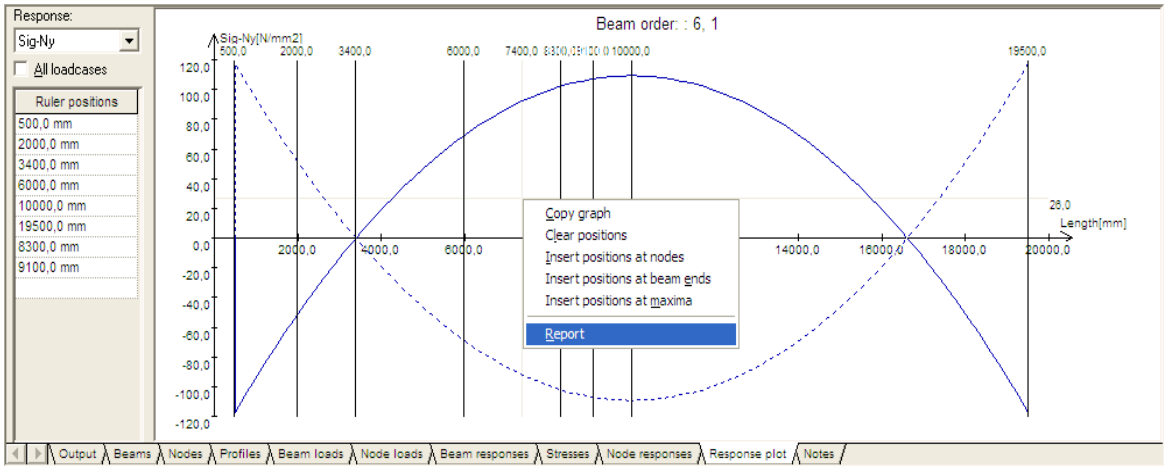
To generate a report for selected items only, click the *Apply to selection only* button  .

Delete or save the report before generating a new report.

14.2. Create reports from the Response plot

Reports with detailed results for the beam(s) may be generated at desired positions.

Right-click and select *Report* from the menu.



Note: The *Report* option is not available if the *Ruler positions* table is empty.

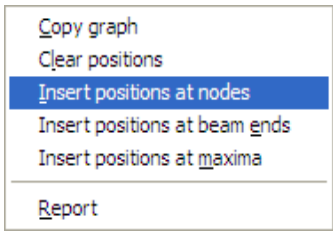
14.2.1. Set Ruler positions by clicking

Click in the plot to set a Ruler position as desired. The positions will be sorted automatically before the report is generated.

To clear the Ruler positions: right-click and select *Clear positions*.

14.2.2. Set Ruler positions automatically

Right-click in the plot and select the desired *Insert position at …*

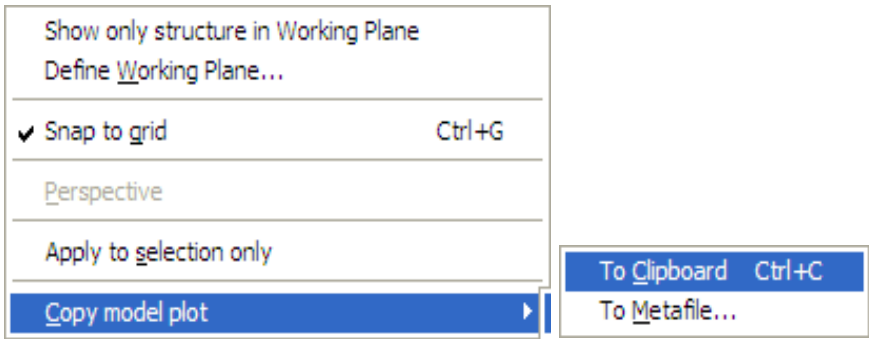


14.3. Include plots in the report

14.3.1. Include plots of the model

Generate a desired report. Bring 3D Beam to the front (Alt+Tab)

Right-click in the *Model Window* and select *Copy model plot* | *To Clipboard*



Return to the report (Alt+Tab) and paste the plot to desired location.

Any number of plots may be added to the report.

14.3.2. Include the Response plot

Generate a desired report. Bring 3D Beam to the front (Alt+Tab)

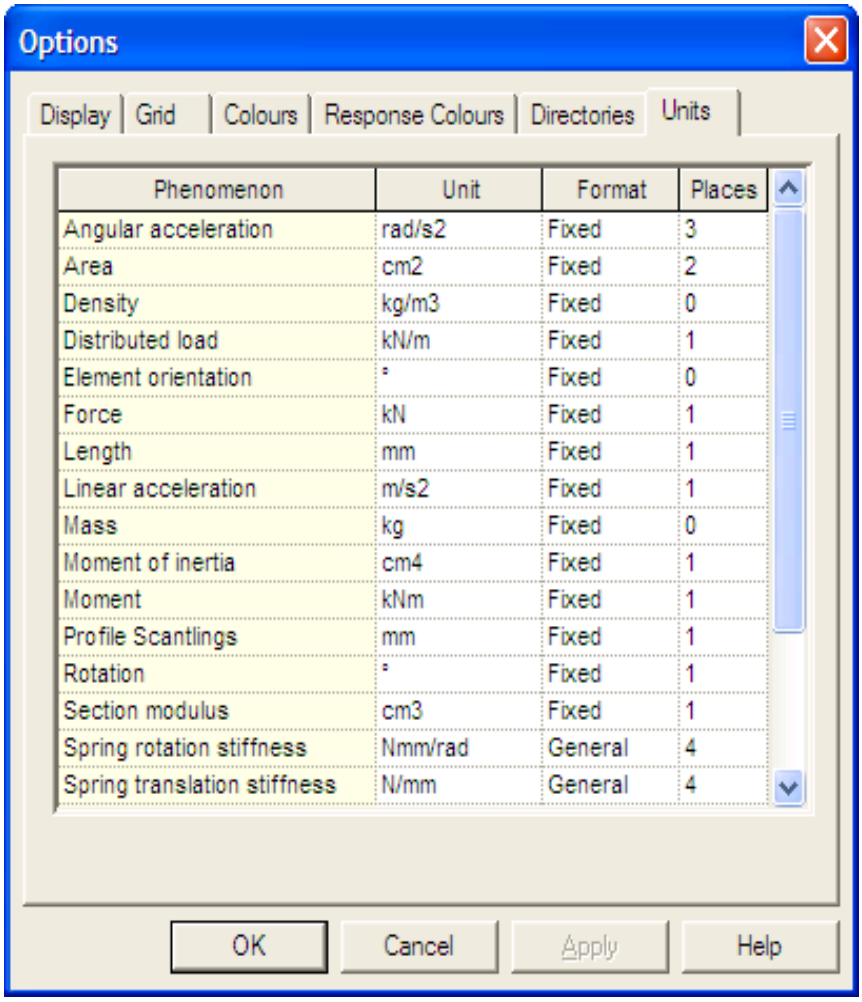
Right-click and select *Copy graph* from the menu.

Return to the report (Alt+Tab) and paste to desired location.

Repeat for as many plots as desired.

15. Specify display units and formats

The display units and formats may be set according to personal preferences. Select *Tools*|*Options*|*Units* from the **Menu bar** to bring up the Units dialog:



16. Use input fields

16.1. Use units in input

When entering a value in an input field, the user specified unit is assumed. If a different unit is entered, an on-the-fly conversion will take place to the desired display unit.

E.g. entering 1 in a field with mm as display unit, will result in 1,0 mm being displayed, assuming 1 decimal is specified. However, entering 1 ft in the field will result in 304,8 mm being displayed.

16.2. Use incremental input

Input fields accept incremental input in the format += *value[unit]* and -= *value[unit]*.

Example: A field is displaying 1,0 mm.

Entering +=1ft will result in 305,8 mm being displayed.

Entering -=1m will result in -694,2 mm being displayed.

17. Use shortcut keys

Pressing these keys	Performs this action
Ctrl + A:	Select All; Nodes and beams
Ctrl + C:	Copy; Selected beams, input values
Ctrl + D:	Delete; Selected beams
Ctrl + F:	Zoom Active Window to Fit; Maximise model within active window
Ctrl + G:	Snap to Grid (on/off); Turn on and off the snap to grid feature
Ctrl + N:	New; Create a new, blank project. Existing projects are terminated
Ctrl + O:	Open; Open a model file
Ctrl + P:	Print; Plot active window
Ctrl + S:	Save; Save project to file
Ctrl + T:	Copy and Transform; Copy, mirror and/or transform selected beams
Ctrl + V:	Paste; I.e. copied values
Ctrl + W:	Beam Wizard; Create beams by input of node coordinates
Ctrl + Y:	Redo; Redo last action
Ctrl + Z:	Undo; Undo last redo
Ctrl + F6:	Next Window; Activate next window if more windows
Ctrl + F4:	Close Window; Close active window
Ctrl + Shift + F:	Zoom All Windows to Fit; Maximise model in all windows
F9:	Analyse; Run analysis
F10:	Activate menu bar
Alt + A:	Activate Analyse menu
Alt + E:	Activate Edit menu
Alt + F:	Activate File menu
Alt + G:	Activate Geometry menu
Alt + H:	Activate Help menu
Alt + T:	Activate Tools menu
Alt + V:	Activate View menu
Alt + W:	Activate Window menu

18. Use MS Excel with 3D Beam

18.1. Post-process tables in MS Excel

Select desired cells in a table. Press Ctrl+C. Paste into MS Excel.

To include the column headings, select the whole table by clicking at the upper left corner of the table, i.e. above the row numbers.

18.2. Generate geometries from MS Excel

The file NodeCoordinates.xls which is shipped together with 3D Beam contains spreadsheets with examples of geometries generated by formulae. The generated coordinates may be selected and copied (Ctrl+C) and pasted into the appropriate tab of the *Beam Wizard* in 3D Beam, see **Create a model by specifying node coordinates with the Beam Wizard**

19. Use 3D Beam in a Nauticus Job

Start Nauticus Project Manager

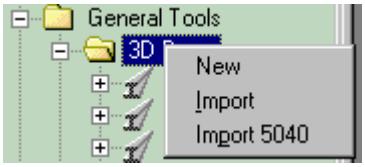
Open a new or existing "Extended" job, i.e. RuleCheckExt… or HSLCExtended… Job.

Locate the 3D Beam tool in the folder:

- General Tools for a RuleCheck Job
- Structural Response for a HSLC Job

Right-click on the 3D Beam node and select:

- *New* to create a new model
- *Import* to create a model from an external 3D Beam file (.clb)
- *Import 5040* to create a model from a FILE CREATED BY nv5040 (I5040*)
-



To rename a 3D Beam model in a Job, click on the text, e.g. "3D-Beam0" to select and modify the text.

To open a model in 3D Beam, select a 3D-Beam node in the Job

Click the button  to open 3D Beam

Please note the following differences to the stand-alone version:

- The model is embedded in, and saved with, the Job.
- Use the menu: *File* | *Save Copy As*... to export a copy of the model to a .clb file for use elsewhere.

20. Use a mouse with a wheel

3D Beam is compliant with MS IntelliMouse actions.

The following features are provided:

In the **Model window**:

This operation	Performs this action
Turn Wheel:	Zoom (In/Out)
Ctrl + Wheel:	Rotate around the vertical axis
Shift + Wheel:	Rotate around the horizontal axis
Click Wheel button:	Pan left/right/up/down

In the input grid window:

This operation	Performs this action
Shift + Wheel:	Scroll up/down
Ctrl + Wheel:	Zoom (In/Out)

In the result grid window:

This operation	Performs this action
Turn Wheel:	Scroll up/down
Ctrl + Wheel:	Zoom (In/Out)