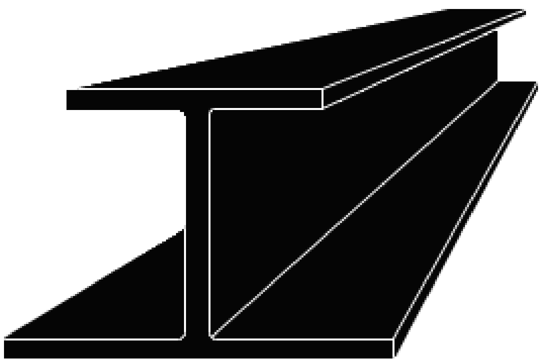
NAUTICUS HULL

USER MANUAL

3D BEAM



DET NORSKE VERITAS

NAUTICUS HULL

USER MANUAL

3D BEAM

JULY 2012

Valid from program version 15.4

Developed and marketed by
DET NORSKE VERITAS

Based on DNV Report No.: 98-0513

Published by:

Det Norske Veritas
Veritasveien 1
N-1322 Høvik
NORWAY

Telephone:	+47 67 57 76 50
Fax:	+47 67 57 72 72
E-mail, sales:	dnv.software@dnv.com
E-mail, support:	software.support@dnv.com
Website:	www.dnv.com/software

Contents

1 INTRODUCTION....5

1.1 Summary....5

1.2 Program updates....5

1.3 About this manual....6

1.4 Quick look guide....6

2 FEATURES OF 3D BEAM....7

2.1 Graphical user interface....7

3 A GUIDED TOUR OF 3D BEAM....9

3.1 A step by step example....9

3.2 General settings....9
3.2.1 Program settings....9
3.2.2 Model properties....10

3.3 Geometry modelling....10
3.3.1 Example model....10
3.3.2 Creating a model....12
3.3.3 Insert rigid ends....14
3.3.4 Insert nodes....15
3.3.5 Rotating beams (Local axis)....16
3.3.6 Creating beams by numerical input (Beam Wizard)....17
3.3.7 Moving nodes....18
3.3.8 Deleting beams....19
3.3.9 Copy and mirror beams....19
3.3.10 Working in 3D....20

3.4 Entering input properties....22
3.4.1 Defining materials....22
3.4.2 Defining beam profiles....23
3.4.3 Import/Export of profiles....24
3.4.4 Applying profiles to beams....24

3.4.5 Apply boundary conditions 26
3.4.6 Loads and load cases 28

3.5 Analysing the model 33

3.6 Result presentation 34
3.6.1 Presentation alternatives 34
3.6.2 Display the responses on the model 34
3.6.3 Tabulated result values 35
3.6.4 Response Plot 38
3.6.5 Responses on individual nodes and beams 38

3.7 Reporting 39

4 COMMAND REFERENCE 43

4.1 Introduction 43

4.2 Menu bar 43
4.2.1 File menu 43
4.2.2 Edit menu 44
4.2.3 View menu 46
4.2.4 Symbols Menu 49
4.2.5 Model menu 49
4.2.6 Analysis menu 53
4.2.7 Tools menu 53
4.2.8 Window menu 62
4.2.9 Help menu 63

4.3 Toolbars 63
4.3.1 Show or hide toolbars 63
4.3.2 File toolbar 64
4.3.3 View toolbar 64
4.3.4 Loads toolbar 65
4.3.5 Named Selection toolbar 65
4.3.6 Geometry toolbar 66
4.3.7 Display toolbar 66
4.3.8 Response toolbar 67

4.4 Input property window 67
4.4.1 Input property window overview 67
4.4.2 Node properties 68
4.4.3 Beam properties 70
4.4.4 Selection properties 76
4.4.5 Model properties 77

4.5 Response property window 78
4.5.1 Response property window overview 78
4.5.2 Response properties - single node 78
4.5.3 Response properties - single beam 80

4.6 Output window 82
4.6.1 Output window overview 82
4.6.2 Structure and loads tables 83
4.6.3 Result tables 86

4.7 Model window 90

4.8 Shortcut keys 91

4.9 Mouse operations 92
4.9.1 Select modes 92
4.9.2 MS IntelliMouse: 92

5 3D BEAM AS PART OF NAUTICUS PROJECT MANAGER 95

5.1 Creating and importing 3D Beam models 95

5.2 Saving 3D Beam models .... Error! Bookmark not defined.

6 APPENDIX A: THEORY 97

6.1 Calculation method 97

6.2 Non linear model 97

6.3 Shear area 98

6.4 Shear centre offset 98

6.5 Profile sectional properties 100
6.5.1 Notations 100
6.5.2 General profile 101
6.5.3 I-profile 101
6.5.4 Double skin profile 102
6.5.5 Box profile 102
6.5.6 Pipe profile 103

7 APPENDIX B: USING SPREADSHEETS WITH 3D BEAM 105

7.1 Purpose 105

7.2 Importing geometry 105

7.3 Exporting results 107

8 APPENDIX C: WINDOW CONFIGURATION 109

8.1 Window configuration 109

9 APPENDIX D: 3D BEAM AS COM SERVER 111

9.1 Purpose 111

9.2 The 3D Beam COM interface 111

1 Introduction

1.1 Summary

3D Beam is a state-of-the-art application for linear static analysis of 2D and 3D frame structures. It is designed with ship and offshore structures and -equipment in mind but it is also very well suited for analysis of other types of typical frame and truss structures.

3D Beam is developed by Det Norske Veritas and the application is distributed as a part of the NAUTICUS program suite. The program may be started as a stand-alone program or from within certain NAUTICUS Jobs.

1.2 Program updates

New in version 10.5:

• Combined stresses according to von Mises are presented for basic profiles, i.e. pipe-, box-, I-, channel-, and L-sections.

• The dialog for specifying profiles has been modified.

New features in version 9.0:

• The range of available profile types has been extended.

• A load wizard for generating distributed loads due to inertia forces is included. Specifying -1g for acceleration in global Z-direction will generate a load corresponding to gravity.

• A load wizard is included for generating distributed loads in the local coordinate system due to loads defined in the global coordinate system.

New in version 10.9:

• Incremental rotation of the copied selection is implemented.

• The Load case Names are now reported in the Output window for the analysis.

New in version 15.2:

• The feature enabling estimation of effective flange has been modified. Now this feature is available only for relevant cross sections and the calculated effective flange is transferred back to the cross section.

1.3 About this manual

In 1.4 "Quick look guide" you will find a figure showing an overview of where to find information about main issues related to the user interface of the program.

When using the program for the first time it is recommended to follow the step-by-step example in 3 "A guided tour of 3D Beam". This includes creating a structure, applying beam and node properties, presenting the results and reporting.

Chapter 4 "Command Reference" gives an overview of the operations of 3D Beam, a summary of the shortcut keys and a description of the menu- and toolbars available.

In "Appendix A: Theory" you will find information about the Timoshenko beam theory used by the program including basic assumptions and definitions made, such as implications of the definitions of shear area and shear centre of profiles.

"Appendix B: Using spreadsheets with 3D Beam" describes features related to the use of spreadsheets in connection with the program. This may ease the work when creating the model and presenting the results.

"Appendix C: Window configuration" shows how to customise the window layout of the program.

1.4 Quick look guide

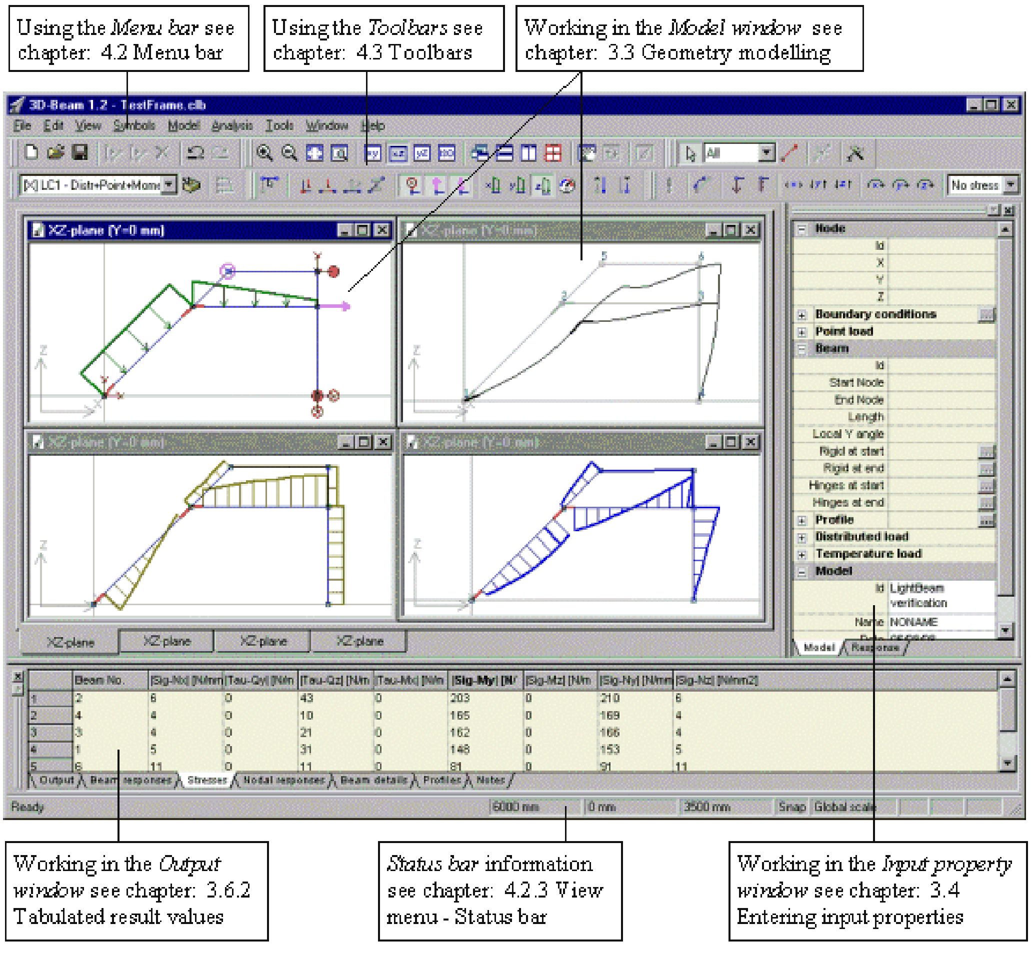


Figure 1.4-1 Quick Look Guide

2 Features of 3D Beam

2.1 Graphical user interface

The figure below gives an overview of the graphical user interface of 3D Beam. It consists of five different interfaces: i) *Menu bar* ii) *Toolbars* iii) *Model window* iv) *Input property window* v) *Output window*. Each of the interfaces provides you with various features and functionality. This will be described in the following sections.

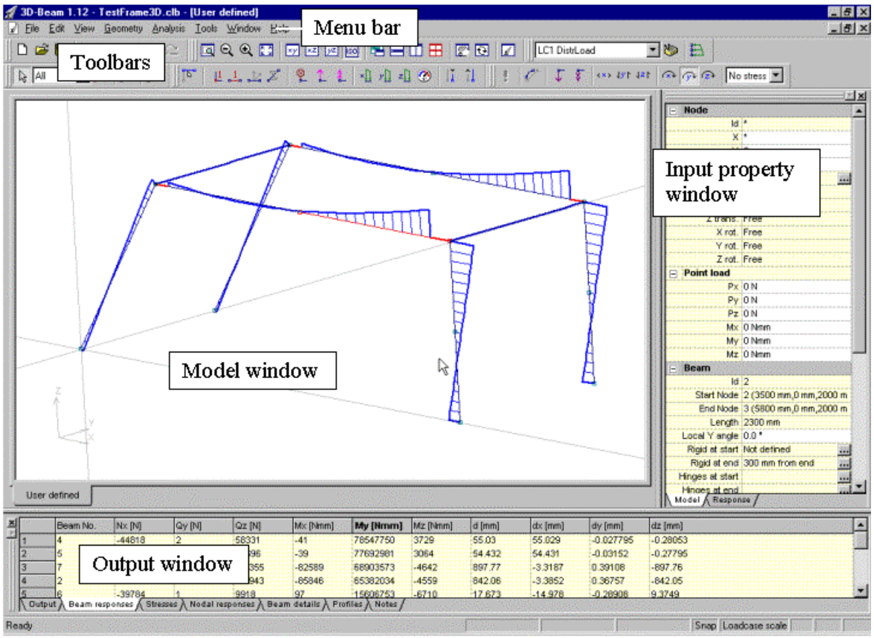


Figure 2.1-1 Graphical User Interface of 3D Beam

Menu bar

The Menu bar provides you with access to all functions and features of 3D Beam.

For more details see 4.2 "Menu bar"

Toolbars

The Toolbars provides easy access to the most used features of 3D Beam.

For more details see 4.3 "Toolbars"

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* you define the boundary conditions, loads, profile properties including material and orientation of the local beam axes, hinged ends and rigid ends.

For more details see 4.4 "Input property window"

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 and results from the analysis is made available in the *Output window* when you run the analysis.

For more details see 0 "At position.: Position of the stress from beam start along local x-axis

Output window"

Model window

The *Model window* is used when creating a model. You may view as many model windows as you like.

For more details see 4.7 "Model window"

3 A guided tour of 3D Beam

3.1 A step by step example

This chapter describes a simple frame analysis step by step to introduce you to the operations and techniques you will use in 3D Beam. The example will guide you through all the main operations including geometry modelling, input of beam and node properties, analysis, result presentation and reporting.

3.2 General settings

3.2.1 Program settings

Before you start with the modelling it is advisable to define appropriate program settings. Adjust the grid size and define the units in the Options dialog box from the Tools | Options menu.

Check and apply display options, colour settings and directories as appropriate before you commence. See 4.2.7 "Options…" for more details.

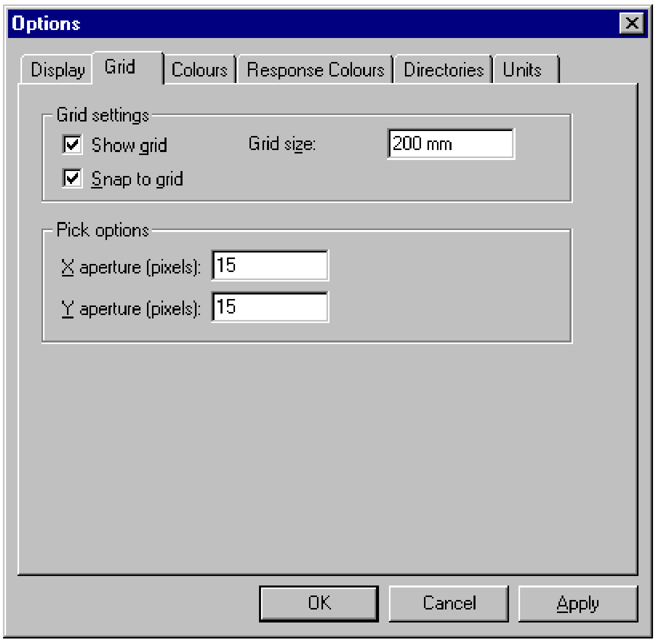


Figure 3.2-1 Define grid size and units

3.2.2 Model properties

Enter the model properties in the input property grid.

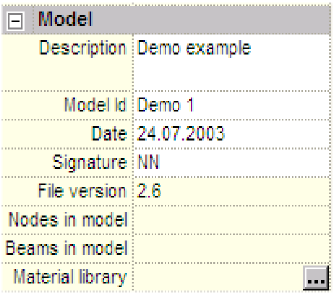


Figure 3.2-2 Model properties

Description: Give a description of the model

Mode Id: The model id, of maximum 6 characters.

Date: The default date is the date when the model was created.

Signature: Your id, e.g. signature, name etc.

File version: The version of the file format used by 3D Beam at the time when the file was last saved.

3.3 Geometry modelling

3.3.1 Example model

The structure to be analysed in this example is shown below. This is a typical midship frame of a Type B tanker with double bottom, double side and vertically corrugated centre line bulkhead (and transverse bulkheads with stools).

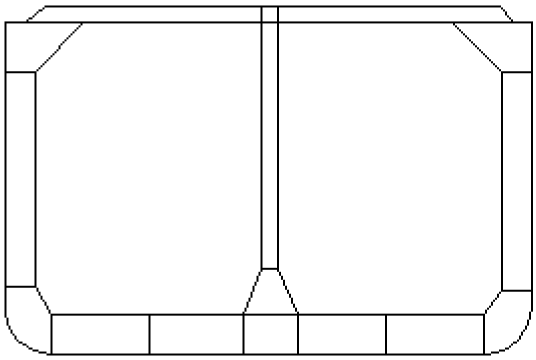


Figure 3.3-1 Midship frame of a tanker

First of all the structure in question must be idealised into a beam model. The double bottom, the double sides and the corrugated bulkhead will be modelled as single beams with the double skin profile. The neutral axis of the structural members and the extension of rigid ends should be defined. The node positions are also determined by the position of point loads, changes in member cross sections and the position of boundary conditions.

The next two figures show the geometry of the idealised beam model with rigid ends and boundary conditions and the load case no. 1. In the following we will explain step by step how to make this model.

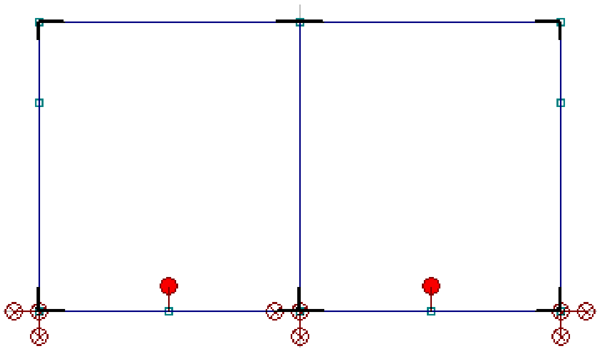


Figure 3.3-2 Idealised beam model

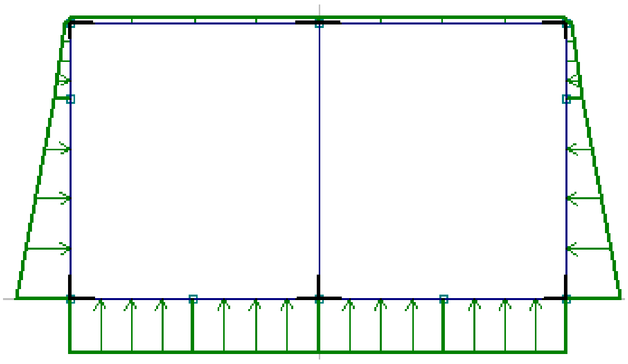
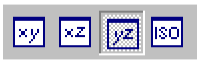
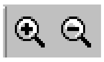


Figure 3.3-3 Load case 1 - Sea pressure

Preparing the Model window

• Apply the YZ view in the *Model window* 

• Set the grid size, e.g. grid size = 200 mm, see 3.2.1 "Program settings"

• Define the workspace: *Zoom*  and move  the *Model window* to fit your purpose. Check the breadth and height of the *Model window* in the coordinate display at bottom of the *Model window* when you move the cursor to the boundaries of the window. (X,Y and Z coordinate of cursor: 

).

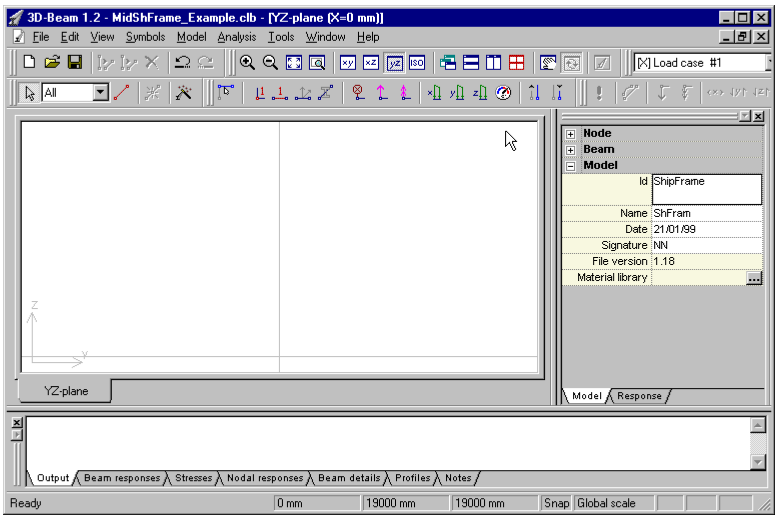


Figure 3.3-4 Preparing the Model window

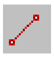
3.3.2 Creating a model

Drawing or entering coordinates

In 3D Beam you can create the model geometry by drawing the beams one by one in the *Model window*, you may define the geometry by entering the node coordinates (numerically) in the Beam Wizard, see 3.3.6 "Creating beams by numerical input (Beam Wizard)", or by utilizing a spreadsheet, see App. B.

The actions described in the following may be cancelled by clicking the *Undo* button  in the toolbar.

Drawing a model

• To draw the geometry select the *Create beams* tool  from the toolbar.

• Position the pointer at the XYZ coordinate for the first node (0,-16200,0) in the YZ view. Click the left mouse button to create the start node of the first beam.

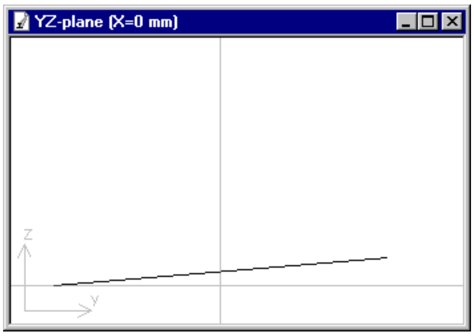
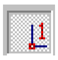


Figure 3.3-5 Draw a beam

• Move the pointer to the position for the next node (0,16200,0) and click the left mouse button to fix the end node and create the beam.

By default 3D Beam provides snap to node. If you draw the next beam from the previous node the beams is connected by default.

• Define the remaining external beams by drawing beams between following coordinates: from (0,16200,0) to (0,16200,18000) to (0,-16200,18000) to (0,-16200,0).

• Click the *Node numbers* button  in the toolbar to display the node numbers.

The geometry should then look like this:

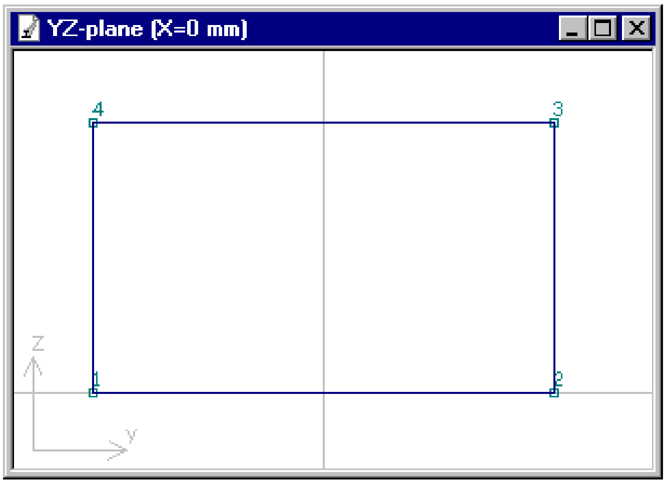
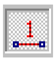


Figure 3.3-6 Draw four continuous beams

• To define the centre line bulkhead beam (between bottom and deck) along the global Z-axis, you may draw a beam from coordinate (0,0,0) to (0,0,18000). The bottom and deck beams will automatically split and the centre line bulkhead beam is automatically connected to the bottom and deck beams.

Note If you inadvertently begin to draw a beam, you may interrupt the action by clicking the right mouse button.

• Click the *Beam numbers* button  in the toolbar to display the beam numbers.

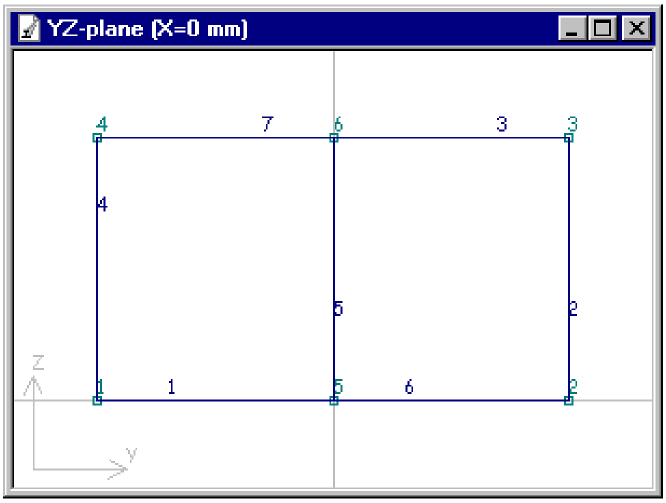


Figure 3.3-7 Insert a beam in-between

• Save the model.

3D Beam stand-alone: Open the Save As dialog from the File-menu and select or create the file catalogue where you want to store your model. Use a descriptive file name (max. 256 characters) to ease the retrieval.

3D Beam in a NAUTICUS Job: Use the Save option in the File-menu to save the job in the current project. You may update the Name-field to give a more descriptive name on the model. The location of the database (.clb-file) is shown in the line below. The default location is the subfolder WFDepot beneath the project folder.

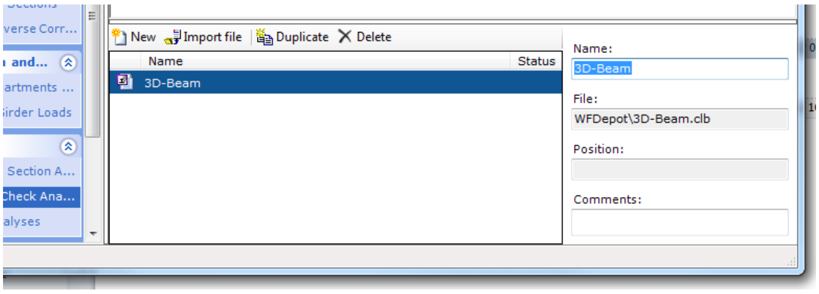


Figure 3.3-8 3D Beam model name and file location

3.3.3 Insert rigid ends

Rigid ends should be applied to members to take into account the effect of rigid parts of the structure in question, e.g. girder connections.

Apply rigid ends at the start of beams in our example.

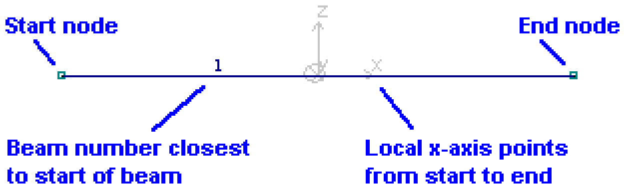


Figure 3.3-9 How to identify the start and end of a beam

Activate the *Select* tool  from the toolbar. Press the Ctrl-key and click on the beams to select them. The selected beams change colour from blue to red (if you use default colour settings).

• Select beams: 1,2,3,5,6,7, by clicking

• Activate the *Rigid at start* dialog box by clicking the open button  in the input property grid.

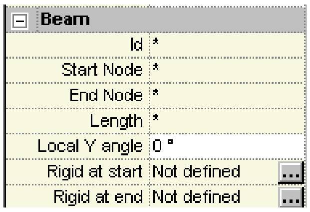


Figure 3.3-10 Input property grid - Rigid ends

• Enter: 1500 mm in the *Distance* field and click OK.

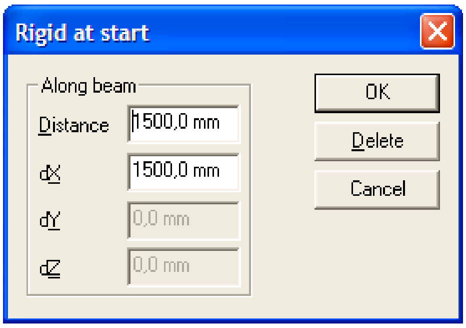


Figure 3.3-11 Rigid end input box

You will now see that the rigid end symbol is displayed at the start of the beams in question.

Apply more rigid ends:

- Select beams: 1,3,4,6,7
- Carry out the same operations as described above with the *Rigid at end* dialog box.
- Select beam no 2 and apply 1000 mm as rigid end of the beam.
- Select beam no 4 and apply 1000 mm as rigid start of the beam.

All rigid ends are now defined in our example model.

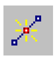
3.3.4 Insert nodes

Nodes need to be inserted at relevant positions where there are changes in beam cross section properties, changes in loads, e.g. at the waterline, and at positions where we need to define boundary conditions.

Insert the following nodes:

On beam no. 2 and beam no. 4: At Z = 13000 mm

On beam no. 1 and beam no. 6: 50% from start

- Select beam no 2 and 4.
- Activate the insert node dialog box by clicking the *Insert node* button  in the toolbar.

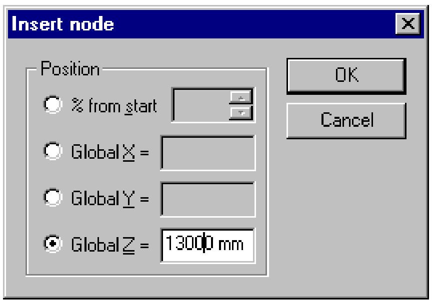


Figure 3.3-12 Input node dialog box

- Select the *Global Z* option and enter 13000 in the input field. Click the OK button.

The beams are split and a new node is inserted at Z = 13000 mm.

- Select beam no. 1 and 6. Activate the insert node dialog box. Select the *% from start* option.
- Specify 50% in the input field.

• Click the OK button.

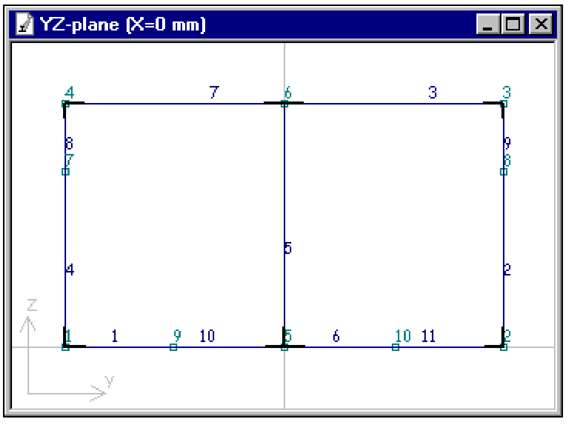


Figure 3.3-13 Insert nodes

3.3.5 Rotating beams (Local axis)

Definition of beam's local axis

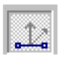
The local positive x-axis is defined along the direction of which the beam is created. The local z-axis is the local "up-axis" and the "dominating" axis with respect to rotation of the local coordinate system.

In cases where the beam is parallel to the global Z-axis, i.e. vertical, the local z-axis is parallel to the global X-axis, both in same direction. Otherwise, the positive local z-axis is orientated such that it gets a positive global Z-projection, i.e. the angle between them is less than 90 degrees.

Rotating beams

You may rotate a beam about its local x-axis. This is relevant when you need to align the local y- and z-axis in the right directions with respect to the orientation of the profile's cross section (i.e. the web direction is defined along the local z-direction) and the direction of distributed loads.

In our example we are soon to define and apply the profiles and the loads. For specification of profiles, see 3.4.2 "Defining beam profiles". In this case we need to align the local z-axis (direction of the I-profile web) in way of our working plane, i.e. in the global YZ-plane. In addition we would like to apply the outer distributed load (sea pressure) along <u>positive</u> local z-axis.

• Click the *Local axes* button  in the toolbar to display the local axes.

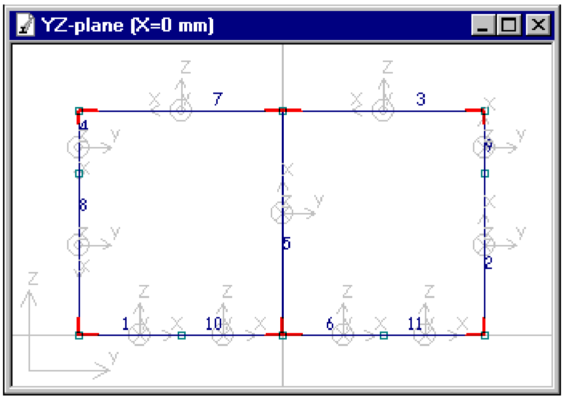


Figure 3.3-14 Local axis

• Select the beams to be rotated 90 degrees: Beams: 2,5,9

• Enter 90 in the *Local rotation* field and press enter. When defining a *Local rotation* it describes the angle of rotation of the beam clockwise about positive local x-axis.

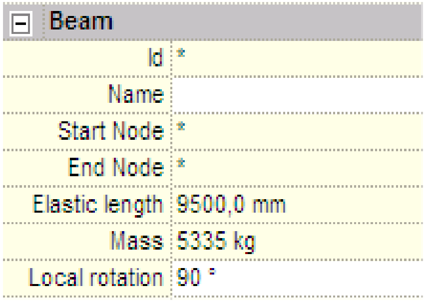


Figure 3.3-15 Rotating the beams

• Select the beams to be rotated 180 degrees: Beams: 3,7

• Enter 180 in the *Local rotation* field and press enter.

• Select the beams to be rotated 270 degrees: Beams: 4,8

• Enter 270 in the *Local rotation* field and press enter.

The local axes should now look like this:

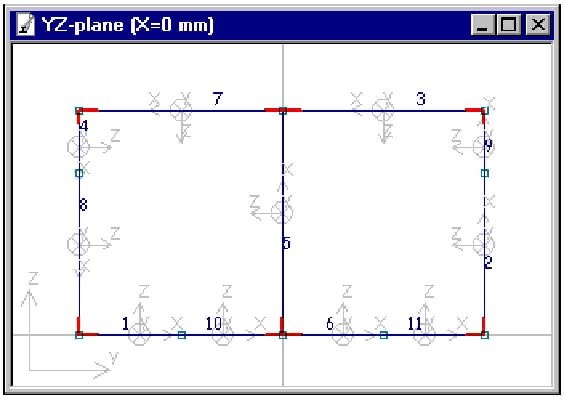


Figure 3.3-16 Rotated local axis

• Save the model.

To continue with our example go to 3.4.2 "Defining beam profiles".

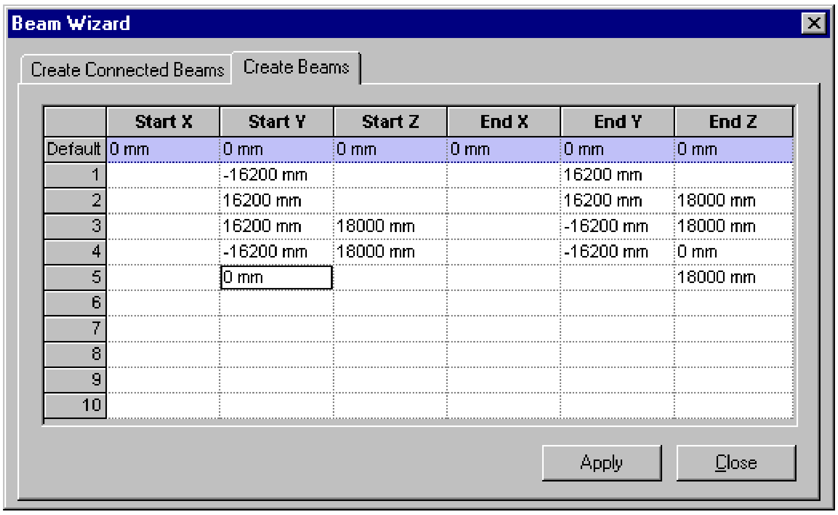
3.3.6 Creating beams by numerical input (Beam Wizard)

It is possible to define the geometry by entering the node coordinates of the beams numerically.

To do this you use the Beam Wizard.

• For the purpose of practice open a new, blank project. Open the Beam Wizard dialog box by clicking the *Beam Wizard* button in the toolbar .

• Enter the node coordinates of the beams in our example:



	Start X	Start Y	Start Z	End X	End Y	End Z
Default	0 mm	0 mm	0 mm	0 mm	0 mm	0 mm
1		-16200 mm			16200 mm	
2		16200 mm			16200 mm	18000 mm
3		16200 mm	18000 mm		-16200 mm	18000 mm
4		-16200 mm	18000 mm		-16200 mm	0 mm
5		0 mm				18000 mm
6						
7						
8						
9						
10						

Figure 3.3-17 Input node coordinates (Beam Wizard)

• Click Apply to generate the beams.

If you are going to create a continuous chain of beams connected to each other you may enter the node coordinates in the *Create Connected Beams* tab card in the Beam Wizard.

If no value is entered in a cell in the Beam Wizard the *Default* value in the top row (blue cells) is used.

You may copy (Ctrl+C) and paste (Ctrl+V) values within the Beam Wizard or copy values from an external source such as MS Excel or MS Word. See Appendix B: Using spreadsheets with 3D Beam" for more details.

3.3.7 Moving nodes

To move/ edit node positions in your model you may do this graphically by drag and drop or alternatively edit the node coordinate value.

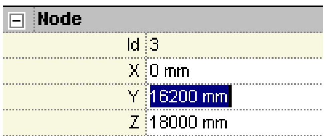
Drag and drop nodes

To move/ edit a node positions in your model graphically by drag and drop you select the node in question by pressing the left mouse button at the node and dragging it to the new position. Observe the running node coordinates of the cursor position in the status bar, at the bottom of the 3D Beam window, while dragging.

Your Tools | Options | Grid settings | Snap to grid, *Grid size* and *Pick options* will influence your possibilities to move the nodes graphically. The smaller grid size and pick options being defined, the more exact you will be able to move the nodes graphically. If *Snap to grid* is active you will not be able to drop the node between grid points.

Edit node coordinates

To move/ edit node positions in your model by editing the node coordinate values you should first select the node(s) to be moved then edit the node coordinate value in the *Input property window*.



Id	3
X	0 mm
Y	16200 mm
Z	18000 mm

Figure 3.3-18 Edit the node coordinates

Several nodes may be selected and edited in one operation.

If you need to merge more nodes into one node then select the nodes to be merged before you enter the node coordinate as described above.

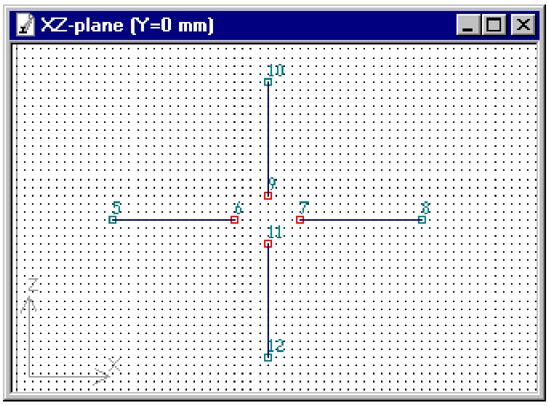


Figure 3.3-19 Merge nodes

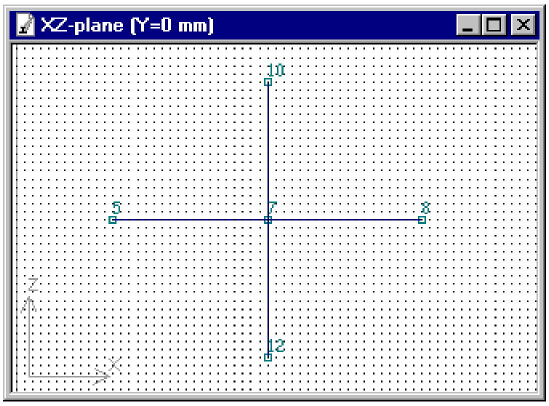


Figure 3.3-20 Merged node

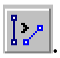
3.3.8 Deleting beams

To delete beam(s) you first select the beam(s) and then click the *Delete* button  in the toolbar.

Note. It is not possible to delete nodes only. A node is defined as the end of a beam and will automatically be deleted when the beam(s) connected to it is removed.

3.3.9 Copy and mirror beams

You may copy beams in any directions by defining the relative distance between the copies (dX, dY, and dZ) and the number of copies.

- Select the beams to copy.
- Click the *Copy and Transform (Mirror, scale)* button in the toolbar .

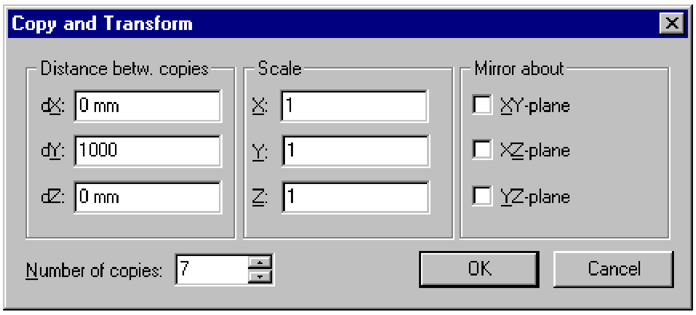
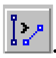


Figure 3.3-21 Copy and Transform

• Enter the relative distance between the copies along one (or more) of the global axes and the number of copies to be made. Click OK.

You may <u>mirror</u> beams about any of the 3 default planes (XY, XZ and YZ).

• Select the beams you want to mirror.

• Click the *Copy and Transform (Mirror, scale)* button in the toolbar .

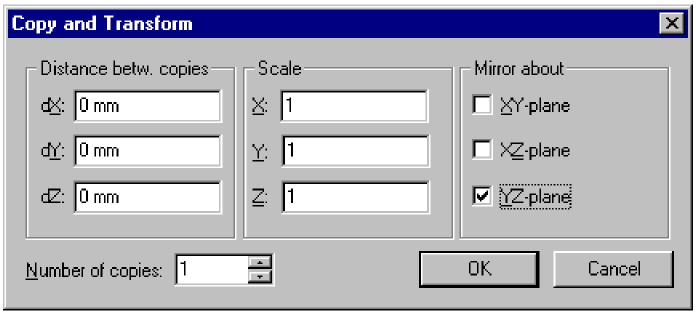


Figure 3.3-22 Copy and Transform

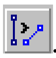
• Select the relevant plane(s) to mirror about. Click OK.

All properties (profiles, loads, boundary conditions, rigid ends etc.) connected to the beams and nodes are transferred to the copies. It is therefore an advantage to apply the properties to the beams before you make the copy.

3.3.10 Working in 3D

Now, create a 3D model by copying the example model along the global X-axis:

• Select all beams (Ctrl+A)

• Click the *Copy and Transform (Mirror, scale)* button in the toolbar .

• Enter the relative distance between the beams to be copied: dX = 2000 mm

• Enter number of copies: 7

• Click OK

• Click the *User defined view* button in the toolbar .

You should now see the following picture in the *Model window*:

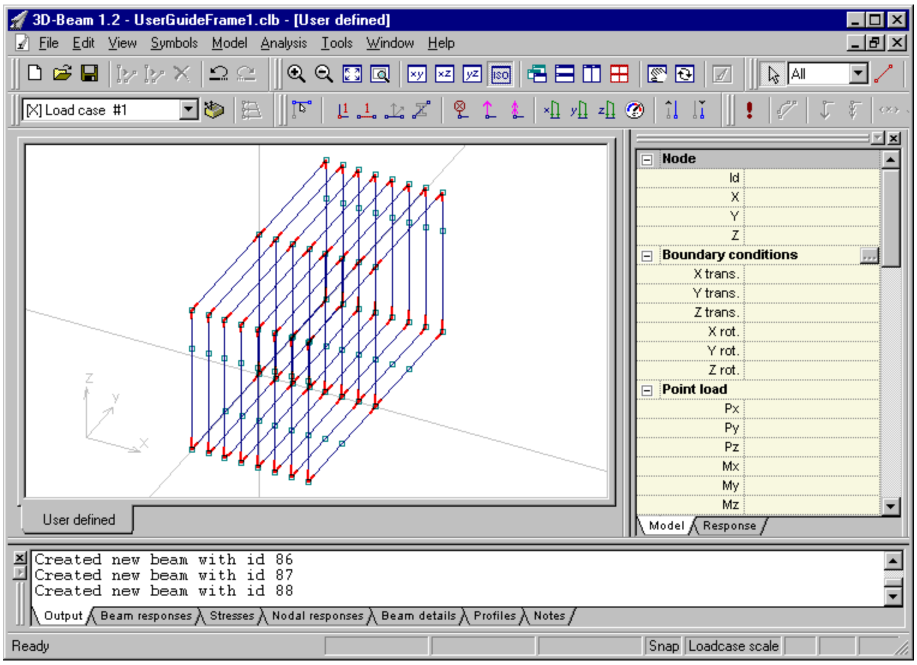
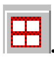


Figure 3.3-23 3D model

Creating beams in the longitudinal direction of a model

To create beams in the longitudinal direction of a model you may use the Beam Wizard, see 3.3.6 "Creating beams by numerical input (Beam Wizard)", to enter the node coordinates numerically.

Alternatively you may draw the beams with the *Create beam* tool:

- Display the model in four views by clicking the *Four views* button in the toolbar .
- Click the right mouse button inside the YZ-window to activate the working plane pop-up menu.

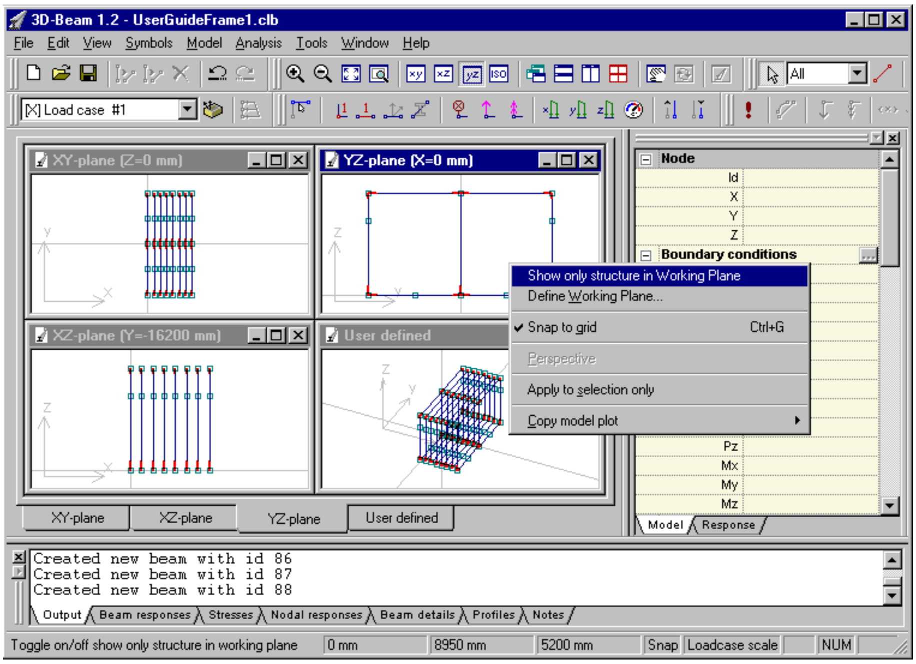


Figure 3.3-24 Four views

• Select the *Show only structure in working plane* option to display only the nodes and beams in the active plane (i.e. in the X=0 plane).

Notice that the beams and nodes outside the active working plane are disabled (grey colour) in the *User defined* view.

• Display the node numbers in the YZ-plane by clicking the *Node number* button in the toolbar .

• Select the *Create beams* tool from the toolbar .

• Attach the first node of the new beam in node number 2 (in the lower right corner of the model) by clicking at the node with the left mouse button and release the button.

• Press the tab key on the keyboard (7 times) to move the working plane in positive X-direction until the plane X=14000 mm is activated. (You may simultaneously press the Shift-key and the Tab-key on the keyboard to move in negative X-direction.) Observe the movement of the active working plane in the *User defined view*.

• Attach the second node of the new beam in node number 73 (in the lower, right corner of the model) by clicking at the node with the left mouse button and release the button.

You have now created a line of longitudinal beams (in the global X-direction) in the lower right corner between the outermost frames.

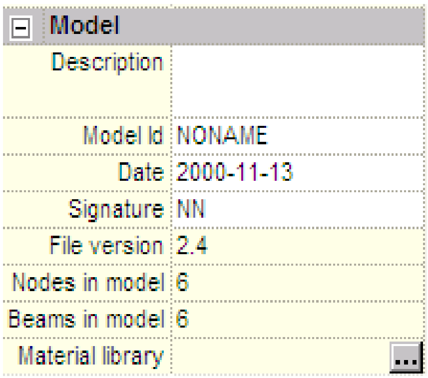
Repeat the above procedure to create lines of longitudinal beams in the centreline and in the lower left corner.

You may apply this technique in any of the main planes (XY, XZ or YZ).

3.4 Entering input properties

3.4.1 Defining materials

3D Beam provides aluminium and steel as default materials. To define your own materials or edit the properties of the default materials you should open the material library. Click the *Material library* open button in the *Input property window* in the *Model* input.



Model	
Description	
Model Id	NONAME
Date	2000-11-13
Signature	NN
File version	2.4
Nodes in model	6
Beams in model	6
Material library	...

Figure 3.4-1 Open the Material library

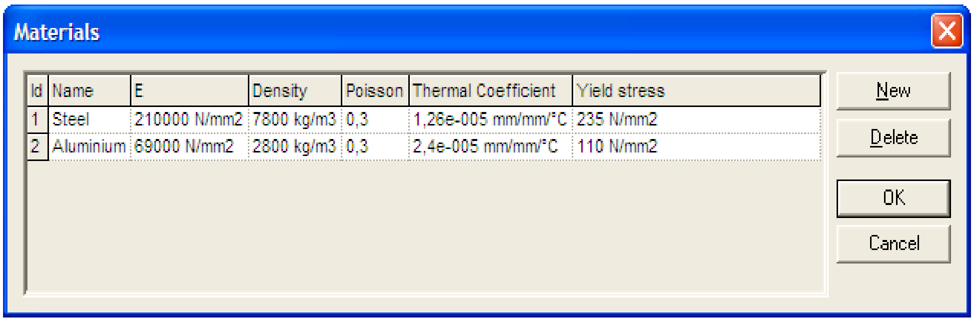


Figure 3.4-2 Properties of materials

To define a new material, click the new button and enter the material name and the properties.

Any material defined in the library may be applied to a profile in the profile dialog box.

3.4.2 Defining beam profiles

The next step in our frame example is to define and apply profiles to the beams.

• Open the profile dialog box by clicking the open button next to the *Profile* heading in the *Input property window*.

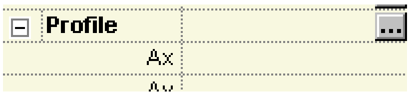


Figure 3.4-3 Open profile dialog

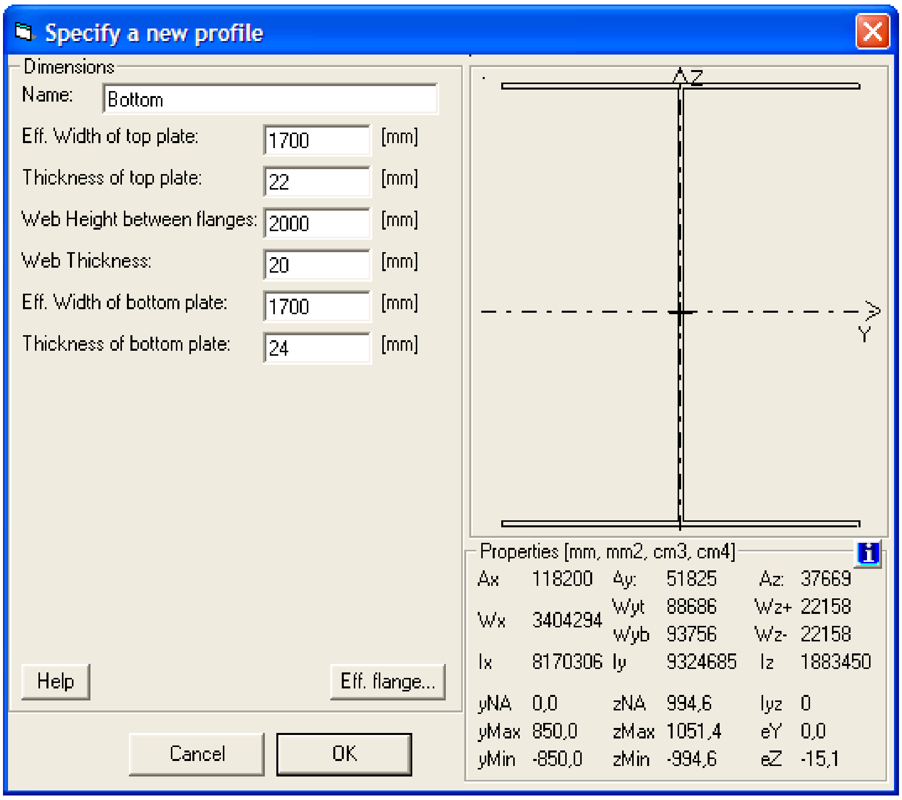


Figure 3.4-4 Profile dialog

To define a new profile click the *New profile* button and choose one of the profile types.

- Choose the *Double skin profile*
- Enter the profile data:

  Name: Bottom

  Effective width of top plate = 1700 mm

  Thickness of top plate = 22 mm

  Web height between flanges = 2000 mm

  Web thickness = 20

  Effective width of bottom plate = 1700 mm

  Thickness of bottom plate = 24 mm

  .

- Define another new Double skin profile with following data:

  Name: Side, B = 1700 mm, Ttop = 16 mm, Hw = 1400 mm, Tw = 16 mm, Tbottom = 16 mm.

- Define an I-profile with following data:

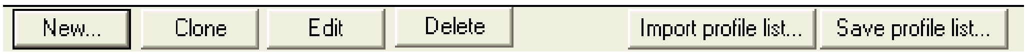
  Name: Deck, Upper flange width = 300 mm, Upper flange thickness = 25 mm, Web height between flanges = 1000 mm, Web thickness = 18 mm, Lower flange width = 1700 mm, Lower flange thickness = 20 mm

- Click OK

Note If one or more beams are selected when the OK button is clicked, then the current profile is applied to the selected beams.

3.4.3 Import/Export of profiles

Specified profiles may be re-used between models. Click Import profile list … or Save profile list… in the toolbar to read/write the profiles from/to files.



3.4.4 Applying profiles to beams

- Select beams: 1,6,10,11 using the *Select* tool in the *Model window*
- Open the *Profile* dialog box
- Select the *Double skin profile* named Bottom and specify:

  Shear factor in local y-direction, fy = 1.0

  Shear factor in local z-direction, fz = 1.0

  Material: Steel

  In this example we will ignore the shear centre offset, thus tick the *Ignore Shear Centre Offset* option 

- Click OK
- Select beams: 2,4,5,8,9 using the *Select* tool in the *Model window*

• Open the Profile dialog box

• Select the *Double skin profile* named Side

• Specify the same material and parameters as for Bottom

• Click OK

• Select beams no. 3 and 7 using the *Select* tool in the *Model window*

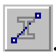
• Open the Profile dialog box

• Select the I-profile named Deck

• Specify the same material and parameters as for Bottom

• Click OK

• Display the model in 3D by clicking the *User defined view* button in the toolbar .

• Display the profiles' cross sections on the beams by clicking the *Cross section* button in the toolbar .

You can now do a visual verification of the profiles that are applied to the beams.

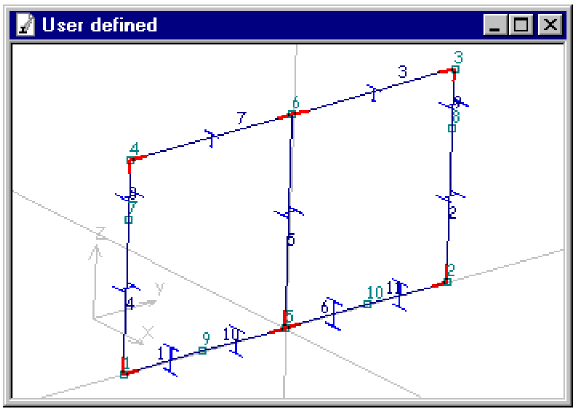
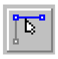


Figure 3.4-5 Display of profiles

Another alternative to verify the profile types and profile orientation is to display the solid view of the frame.

Open a window for viewing the model in solid mode by clicking the *Solid View* button . Any of the projections and the ISO view may be selected. Rigid ends and general cross sections are displayed as lines in the *Solid View*.

Note To show only <u>selected</u> beams in *Solid View* press the *Apply to Selection only* button 

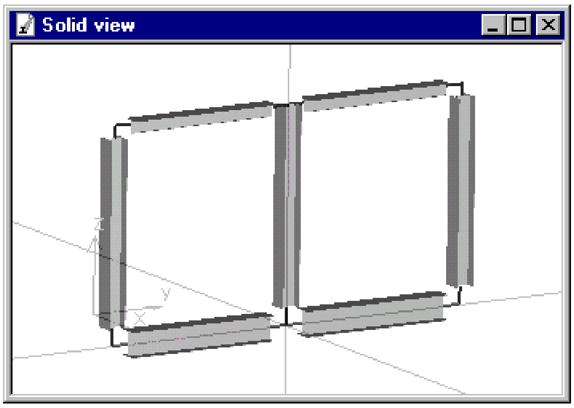


Figure 3.4-6 Solid view

The *Solid View* is automatically updated by changes to the model.

3.4.5 Apply boundary conditions

Evaluate the structure in question to find out what kinds of boundary conditions are relevant. The types and positions of the boundary conditions will have a significant impact on the results and should therefore be carefully evaluated.

Note The boundary conditions need to be applied such that the model in no case is free to rotate or move, i.e.to avoid a singular stiffness system.

Note You may also apply forced displacement or forced rotation of nodes as boundary condition.

In our example we shall apply the following boundary conditions:

Node no.: 1,2,5 - fixed in all degrees of freedom

Node no.: 9,10 - spring supported in the global Z-direction.

• Select node no. 1,2 and 5

• Open the boundary condition dialog box by clicking the open button  next to the *Boundary conditions* heading in the *Input property window*.

• Select all degrees of freedom in the list by pressing the Shift-key on the keyboard when you click with the left mouse button at the bottom of the list (at the Z rot.).

• Select the *Fixed* option below the list.

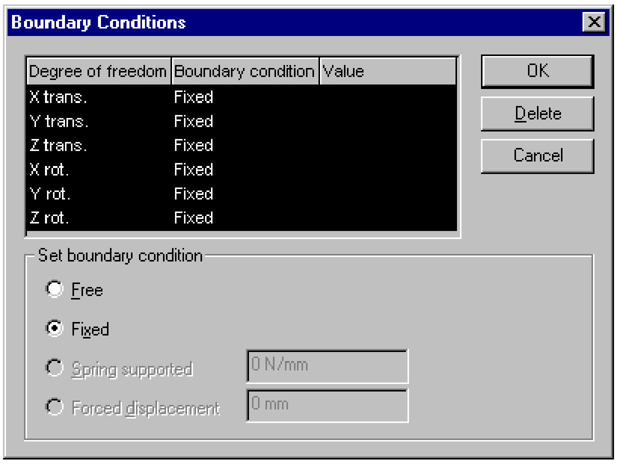
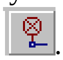


Figure 3.4-7 Boundary conditions

- Click OK
- Select node no. 9 and 10
- Open the *Boundary conditions* dialog box
- Select the Z trans. degree of freedom in the list
- Select the *Spring supported* option below the list
- Enter 100000 N/mm in the related input field
- Click OK
- Display the boundary condition symbols on the model by clicking the *Boundary conditions* button in the toolbar .

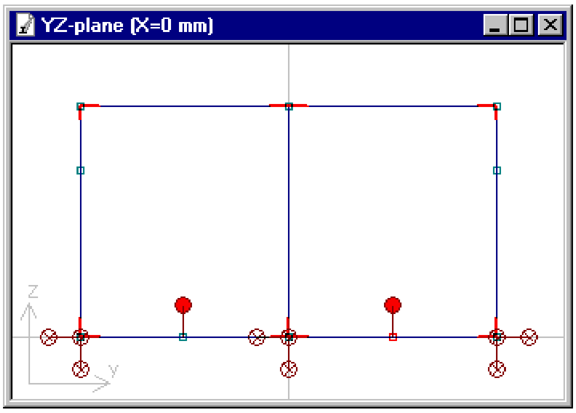
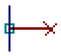
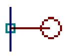
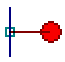


Figure 3.4-8 Display of boundary conditions

Boundary condition symbols:

 Fixed in translation along global axis

 Fixed in rotation about global axis

 Spring supported or forced displacement along/about global axis

3.4.6 Loads and load cases

In 3D Beam you may apply the following load types:

Node loads

Node moments

Distributed line loads on beams

Temperature loads

Define a load case

• Click the *Load Case Manager* button  in the toolbar to define the load case name of load condition 1 (default name = "Load case #1").

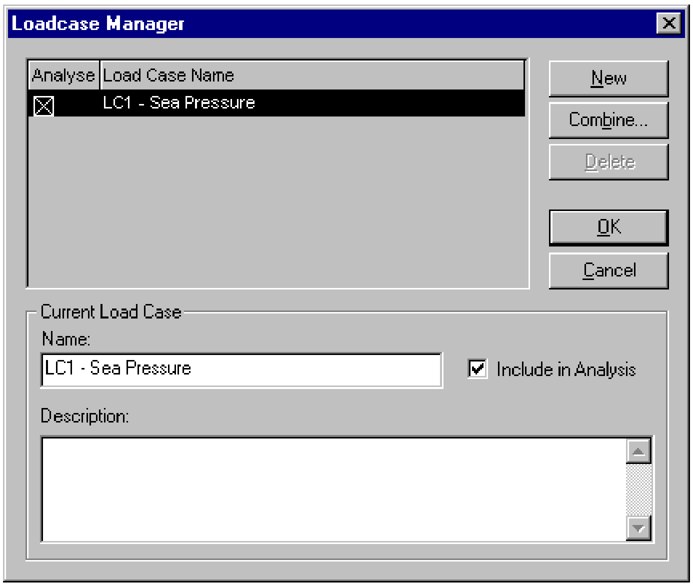


Figure 3.4-9 Load case manager

See 4.2.4 "Manage Load Cases…" for more details about the Load Case Manager.

• Enter a descriptive load case name in the "Name:" field, e.g.: LC1 - Sea Pressure

• Click OK

Apply following loads as load case no. 1 in our example:

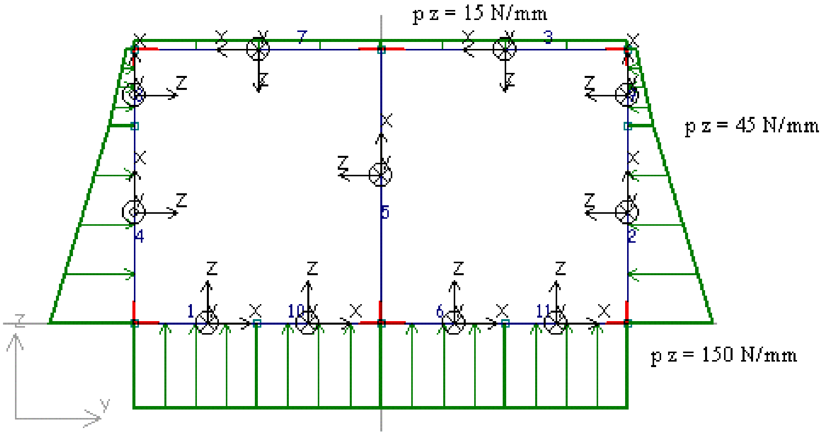
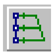


Figure 3.4-10 Distributed loads applied

• Select beams: 1,2,4,6,10,11

• Click the *Create loads over selection* button  in the toolbar.

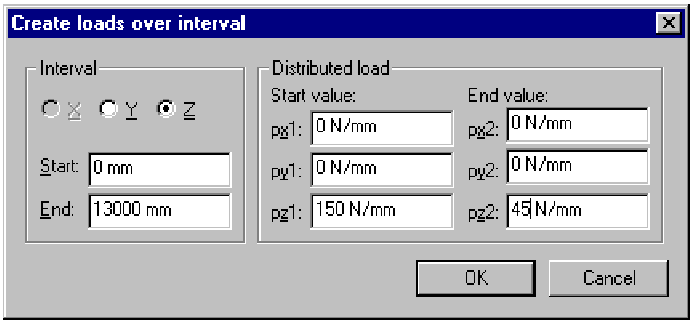


Figure 3.4-11 Create loads over selection

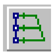
In the *Create loads over selection* input box you can define a linear varying (or constant) distributed load on a selection of beams. The load is applied along the local beam axis (px, py, pz). When defining a linear varying load, select the interval (X,Y,Z) in which global direction the load should be varied.

• Verify that the interval is defined along the global Z-axis with start value = 0 mm and end value = 13000 mm.

• Enter the distributed load start value, pz1 = 150 N/mm, and the distributed load end value, pz2 = 45 N/mm.

• Click OK

• Select beams: 3,7,8,9

• Click the *Create loads over selection* button  in the toolbar.

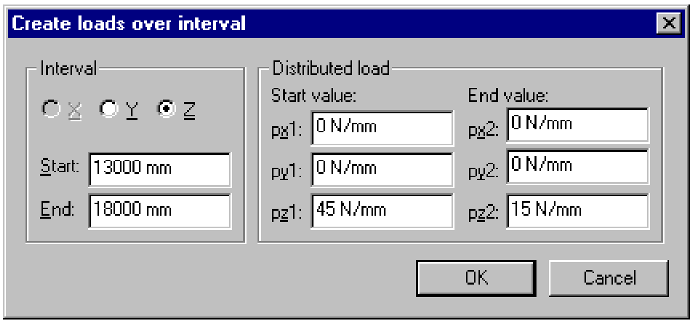
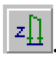


Figure 3.4-12 Create loads over selection

• Verify that the interval is defined along the global Z-axis with start value = 13000 mm and end value = 18000 mm.

• Enter the distributed load start value, pz1 = 45 N/mm, and the distributed load end value, pz2 = 15 N/mm.

• Click OK

• Display the distributed load in local z-direction on the model by clicking the distributed load z-component button in the toolbar .

You have now defined load case no. 1.

New load cases

• To define a new load case, open the *Load Case Manager* .

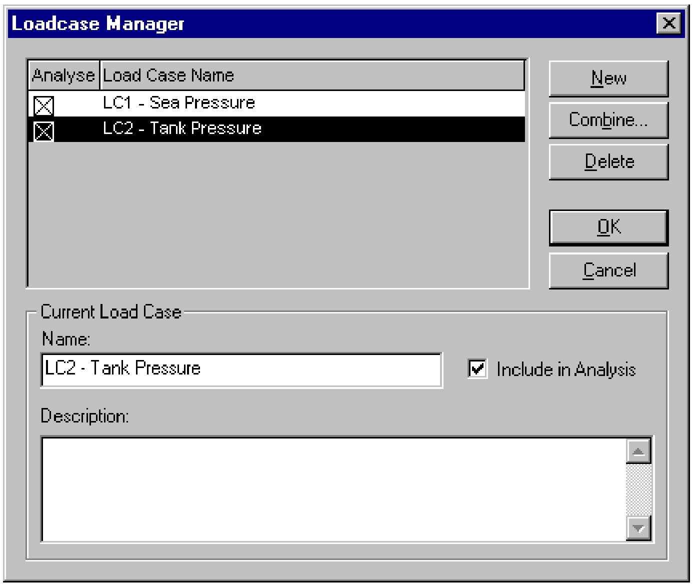


Figure 3.4-13 Load case manager - New load case

• Click the *New* button

• Enter a descriptive load case name in the "Name:" field, e.g.: LC2 - Tank Pressure

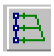
• Click OK

The new load case "LC2 - Tank Pressure" is now activated and it is displayed in the *Active load case* drop down list in the toolbar 

.

Define the loads in the new load case:

• Select beams: 1,2,4,6,8,9,10,11

• Click the *Create loads over selection* button  in the toolbar.

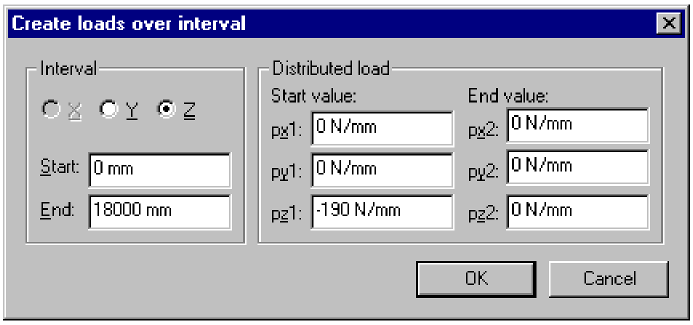
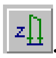


Figure 3.4-14 Create loads over selection - Tank Pressure

• Verify that the interval is defined along the global Z-axis with start value = 0 mm and end value = 18000 mm.

• Enter the distributed load start value, pz1 = -190 N/mm, and the distributed load end value, pz2 = 0 N/mm.

• Click OK

• Display the distributed load in local z-direction on the model by clicking the distributed load z-component button in the toolbar .

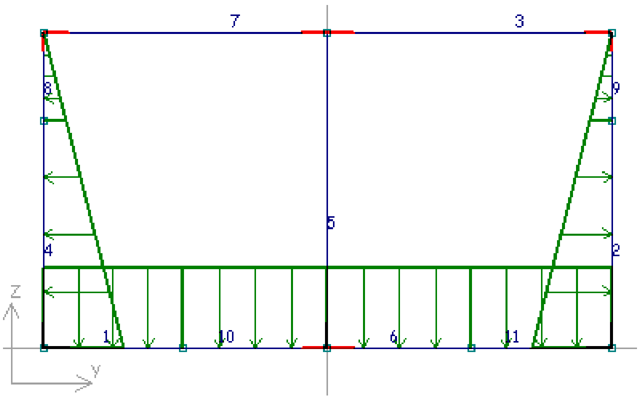


Figure 3.4-15 Load case no. 2 - Tank Pressure

You have now defined load case no. 2.

Combine load cases

• To make a load case combination, open the *Load Case Manager* .

• Click the *Combine…* button

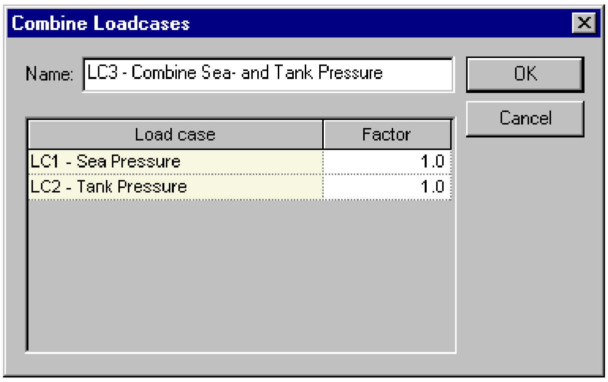


Figure 3.4-16 Combine load cases

• Enter a descriptive load case name in the "Name:" field, e.g.: LC3 - Combine Sea- and Tank Pressure

• Enter the multiplication factor in the *Factor* fields. A factor =1.0 means that 100% of the LC is included in the combined load case.

See 4.2.4 "Manage Load Cases…" for more details

• Click OK

The combined load case is a combination of 1.0 x LC1 + 1.0 x LC2

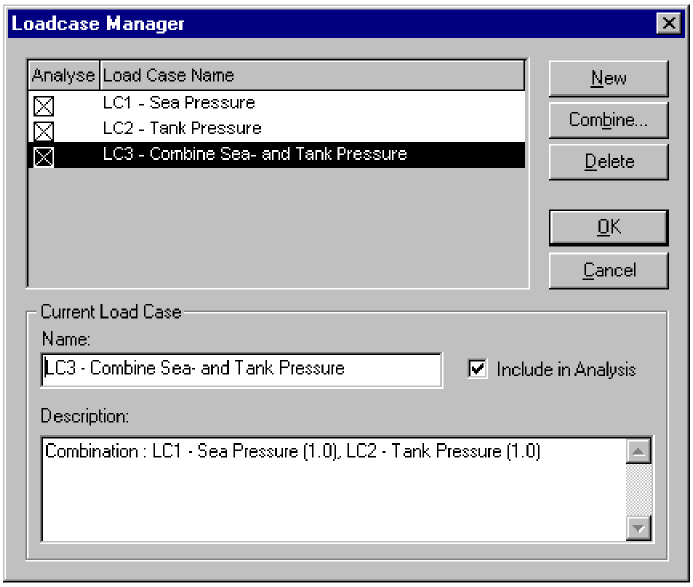


Figure 3.4-17 Load case manager - Combined load case

• Click OK

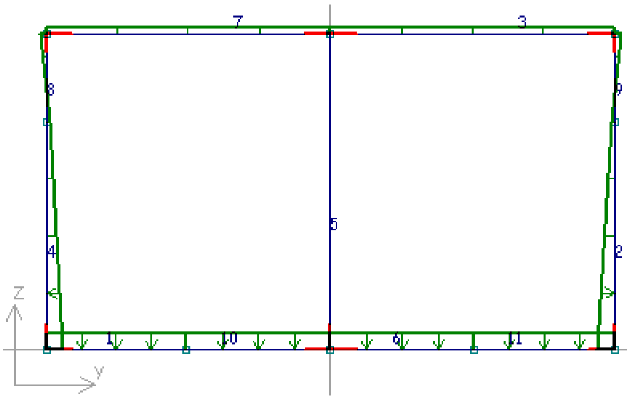


Figure 3.4-18 Load case no. 3 - LC3 - Combine Sea- and Tank Pressure

You have now defined the combined load case no. 3.

3.5 Analysing the model

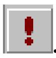
The model may be analysed if:

All beams have profiles.

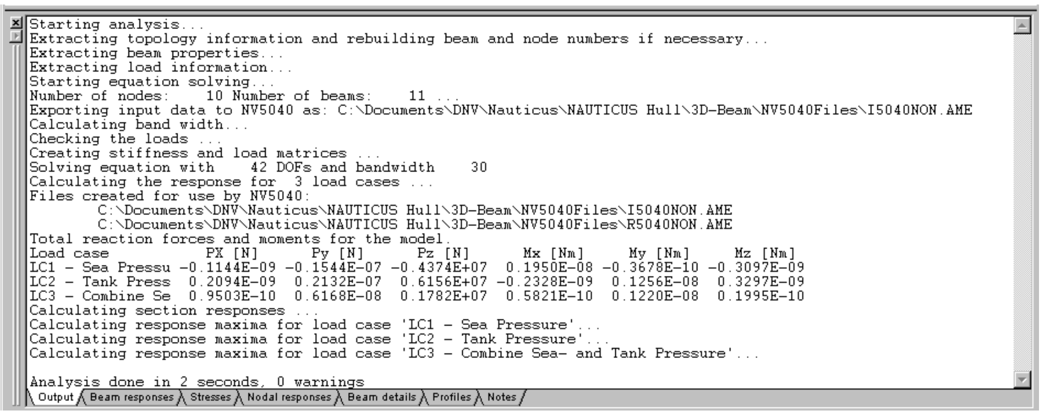
Sufficient boundary conditions are applied to prevent rigid body movements.

All parts are connected.

There is at least one active load case.

Run the analysis of the example by pressing the *Analyse model* button .

You should then receive the following output messages in the *Output window*:



```
Starting analysis...
Extracting topology information and rebuilding beam and node numbers if necessary...
Extracting beam properties...
Extracting load information...
Starting equation solving...
Number of nodes:    10 Number of beams:    11 ...
Exporting input data to NV5040 as: C:\Documents\DNV\Nauticus\NAUTICUS Hull\3D-Beam\NV5040Files\I5040NON.AME
Calculating band width...
Checking the loads ...
Creating stiffness and load matrices ...
Solving equation with    42 DOFs and bandwidth    30
Calculating the response for  3 load cases ...
Files created for use by NV5040:
        C:\Documents\DNV\Nauticus\NAUTICUS Hull\3D-Beam\NV5040Files\I5040NON.AME
        C:\Documents\DNV\Nauticus\NAUTICUS Hull\3D-Beam\NV5040Files\R5040NON.AME
Total reaction forces and moments for the model.
Load case            PX [N]      Py [N]      Pz [N]      Mx [Nm]     My [Nm]     Mz [Nm]
LC1 - Sea Pressu -0.1144E-09 -0.1544E-07 -0.4374E+07  0.1950E-08 -0.3678E-10 -0.3097E-09
LC2 - Tank Press  0.2094E-09  0.2132E-07  0.6156E+07 -0.2328E-09  0.1256E-08  0.3297E-09
LC3 - Combine Se  0.9503E-10  0.6168E-08  0.1782E+07  0.5821E-10  0.1220E-08  0.1995E-10
Calculating section responses ...
Calculating response maxima for load case 'LC1 - Sea Pressure'...
Calculating response maxima for load case 'LC2 - Tank Pressure'...
Calculating response maxima for load case 'LC3 - Combine Sea- and Tank Pressure'...

Analysis done in 2 seconds, 0 warnings
```


Figure 3.5-1 Output messages from analysis

Any error messages or warnings given during the analysis are printed at the bottom of the output message list.

Note. If the error message or warning is related to a node or a beam you may highlight the object by double clicking the message line.

3.6 Result presentation

3.6.1 Presentation alternatives

3D Beam provides several ways of presenting the results. These are described in the following chapters. For more details about the result presentation reference is made to 4.5 "Response property window", 0 "At position.: Position of the stress from beam start along local x-axis

Output window", and 4.7 "Model window".

3.6.2 Display the responses on the model

To display the various result diagrams and responses on the model, click the buttons on the *Response toolbar*. You may display as many responses as you like simultaneously.

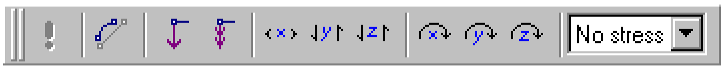


Figure: Response toolbar

• Select: LC1 - Sea Pressure from the *Active load case* drop down in the toolbar.

• Apply the YZ-plane and click the *Bending moment about local y-axis* and the *Displacements* buttons.

The bending moment diagrams and the deformed shape are displayed on the model.

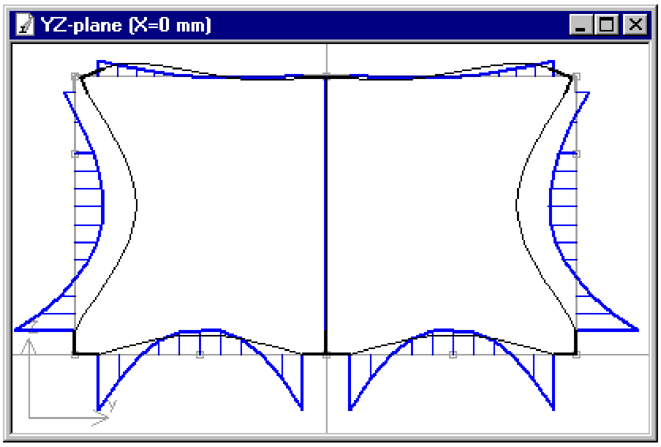
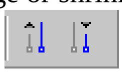


Figure 3.6-1 Display of deflections and bending moments for LC1

In the Tools | Options… | Display you may set the scaling of the response display on the model to apply globally (i.e. based on the highest value across all load cases) or per load case (i.e. based on the highest value within each load case).

This may help you to detect possible errors in the model. It makes it also easier when comparing the responses for each load case to find the most critical one.

You may enlarge or shrink the loads and response symbols' size by clicking the *Enlarge symbols* or *Shrink symbols*  on the toolbar.

• Select: LC2 - Tank Pressure from the *Active load case* drop down in the toolbar.

The response display in the *Model window* is updated instantly.

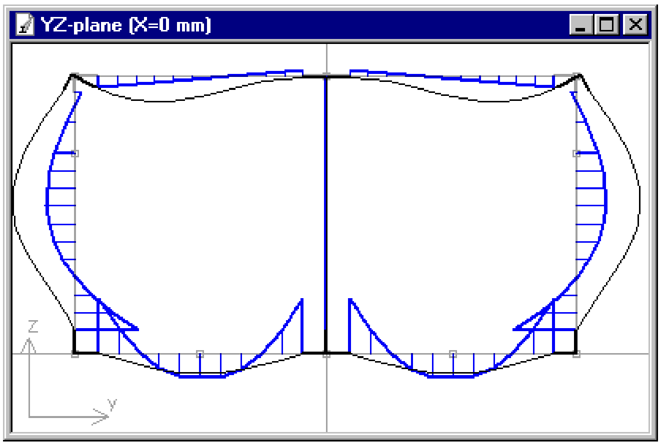


Figure 3.6-2 Display deflections and bending moments of LC2

• When displaying the reaction forces and reaction moments you should apply the *User defined view* to see the directions of the reactions moments more easily.

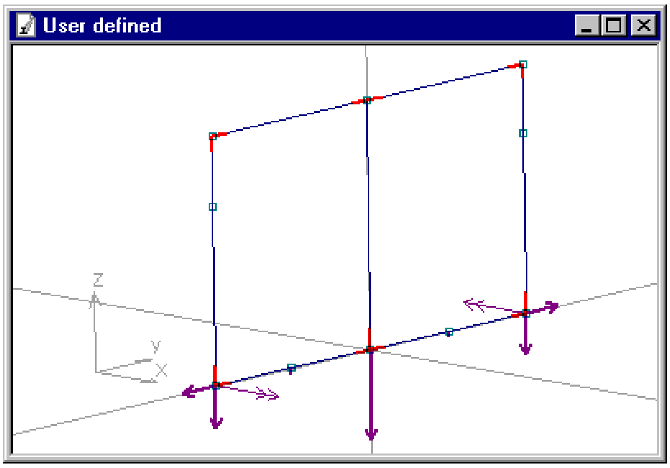


Figure 3.6-3 Display reaction forces and reaction moments of LC1

3.6.3 Tabulated result values

In the *Output window* the maximum values of the response components for all beams and nodes are presented.

The tabulated results are grouped under the following tab cards in the *Output window*:

Beam responses: Maximum beam forces, - beam moments and - beam deflections.

Stresses: Maximum stress components: Axial-, shear-, bending-, torsion- and normal-stresses.

Node responses: Deflections, rotations, reaction forces, and moments.

Carry out the following operations in our example:

• Select: LC1 - Sea Pressure from the *Active load case* drop down in the toolbar.

• Activate the *Stresses* tab card in the *Output window*

• Drag the *splitter bar* between the *Model window* and the *Output window* to enlarge the *Output window* (see the symbol of the cursor in the figure below).

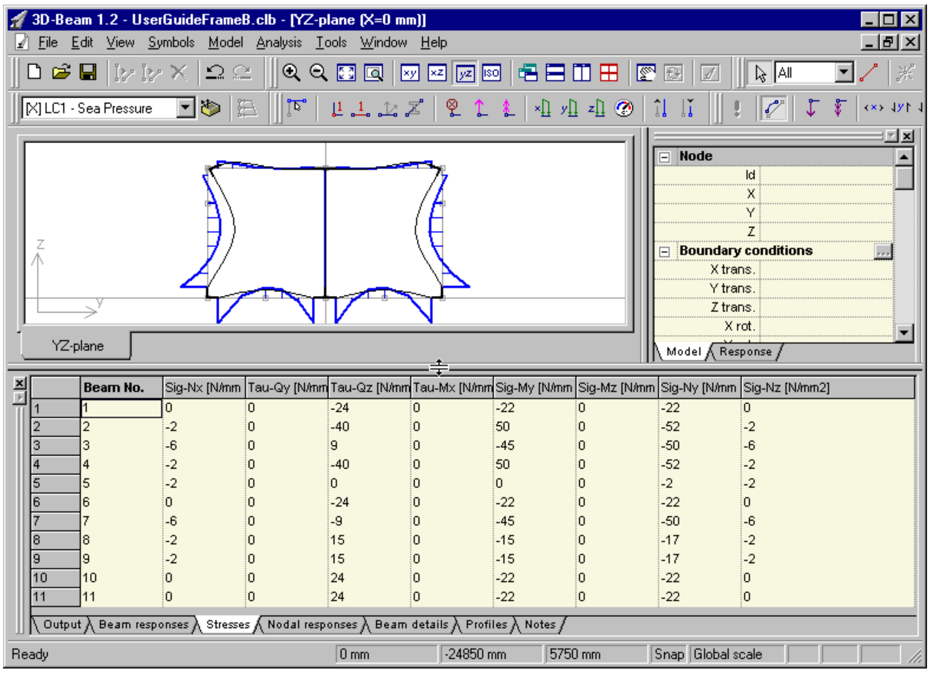


Figure 3.6-4 Result stresses - enlarge the Output window

By clicking the right mouse button in the table, a pop-up menu appears and you will get access to the sort, format and report options for the table.

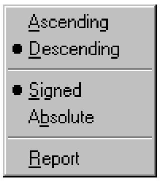


Figure 3.6-5 Result table options

Ascending: Sorts the table in ascending order with respect to selected column

Descending: Sorts the table in descending order with respect to selected column

Signed: Applies signed values

Absolute: Applies absolute values

Report: Creates an MS Word report of the table

- Activate one of the cells in the Sig-My column (click one of the cells)
- Activate the pop-up menu (click right mouse button)
- Select the descending option

Alternatively, the table may be sorted according to any column by double-clicking the desired column heading.

The two beams at the top of the table should now be beam no. 2 and beam no. 4, both with Sig-My = 50 [N/mm2]

- Select the two upper rows in the table by pressing the Ctrl key on the keyboard and click the left mouse button at the row number heading no. 1 and at row number heading no. 2. See below figure. The rows become highlighted with black colour and the beams in the *Model window* are highlighted with red colour (as default).

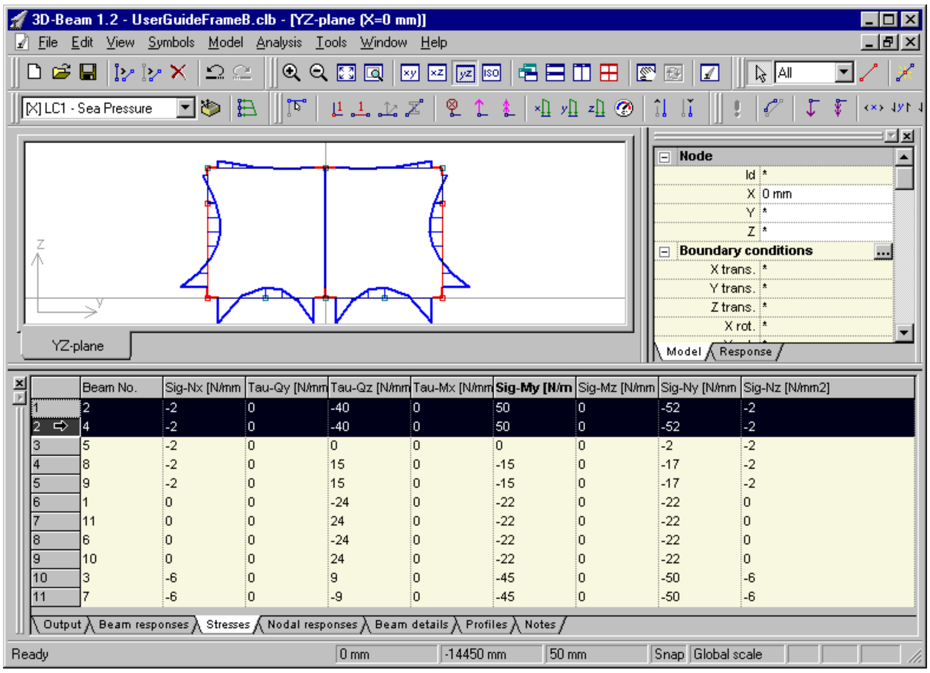
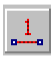
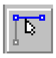


Figure 3.6-6 Select beams in the result table

- Click the *Beam numbers* button  in the toolbar to display the beam numbers
- Click the *Apply to selection only* button in the toolbar  to reduce your result set.

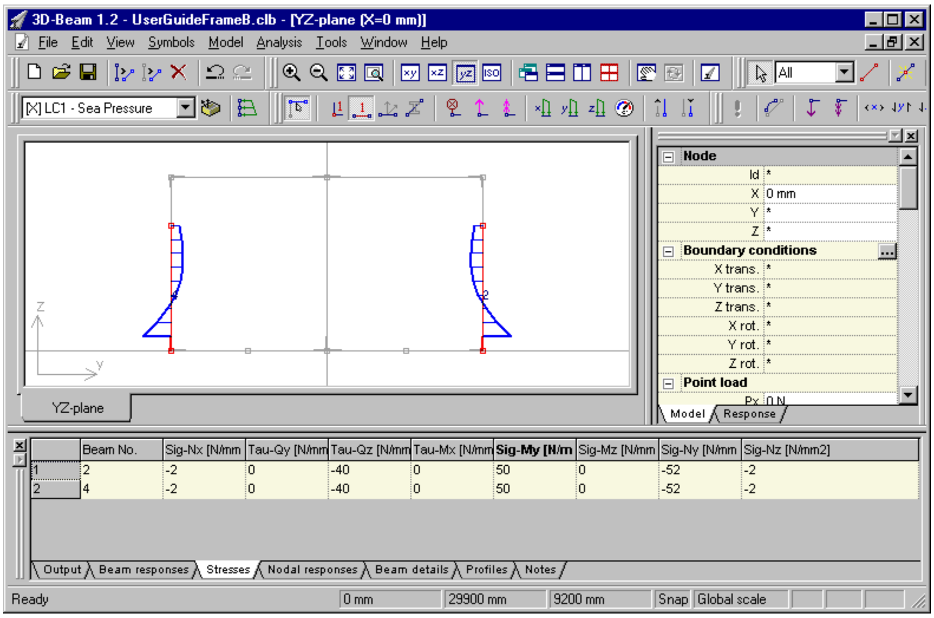


Figure 3.6-7 Apply to selection only

By activating the *Apply to selection only* button, the result set is reduced to the selected beams/ nodes only. This gives you better overview of the results and makes it possible to display results above certain values only. In above example we have chosen to show results only for beams having a bending stress of at least 50 [N/mm2]. The results should be symmetrical, and the plot shows that they are.

3.6.4 Response Plot

If you need to carry out a more refined result analysis of individual beams you should enter the *Response plot* tab card in the *Output window*. You may enlarge the window as appropriate by dragging the splitter bar between the windows.

• Select beam number 2 in the model (see below figure)

• Select Sig-My from the *Response* drop down list in the *Response plot* view

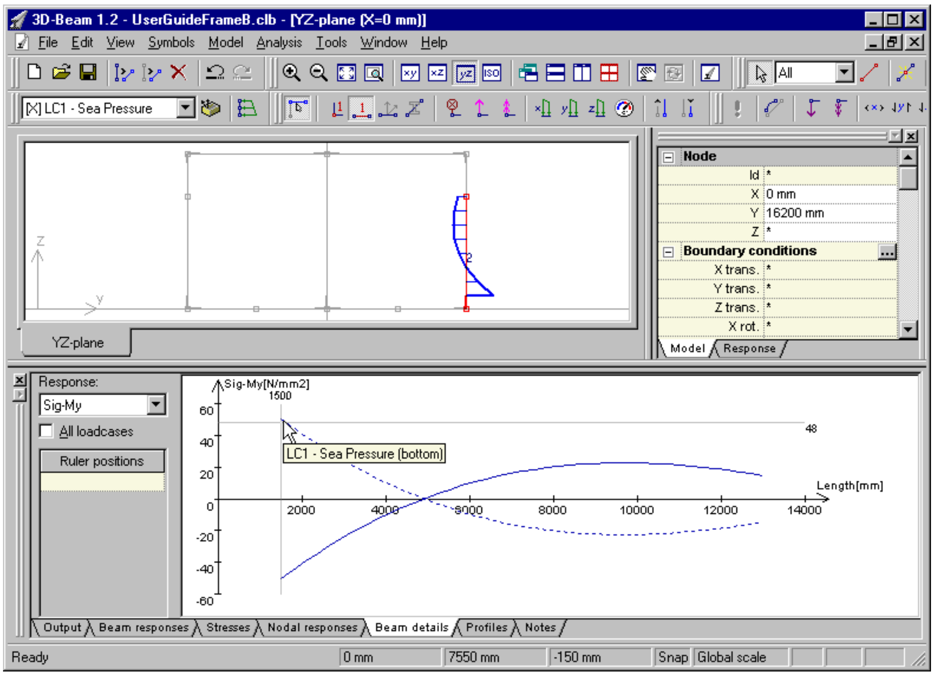


Figure 3.6-8 Response plot

The diagrams in the *Response plot* view apply to the <u>selected</u> beam(s). The start of the beam is always at the left side of the diagram. The beam length from the start node is displayed along the horizontal axis and the response value is displayed along the vertical axis. The diagrams are displayed along the flexible part of the beam, i.e. no responses are calculated along rigid ends.

• Hold the cursor steady above the diagram as shown in the figure above to display the load case name and the flange identification (top/bottom) in the tool-tip that appears.

The response value and the position along the beam are displayed at the end of the cross hairs. By pointing at the one of the curves you will find the exact response value at the position considered.

• Select the *All load cases* option in the *Response plot* view to display responses for all load cases.

3.6.5 Responses on individual nodes and beams

In the *Response* tab card in the <u>*Input*</u> *property window* the responses for selected individual beams and nodes are displayed.

• Activate the *Response* tab card in the *Input property window* (see below figure)

• Select beam number 2 in the model

You may now read all maximum responses and end response components for beam number 2 from the list.

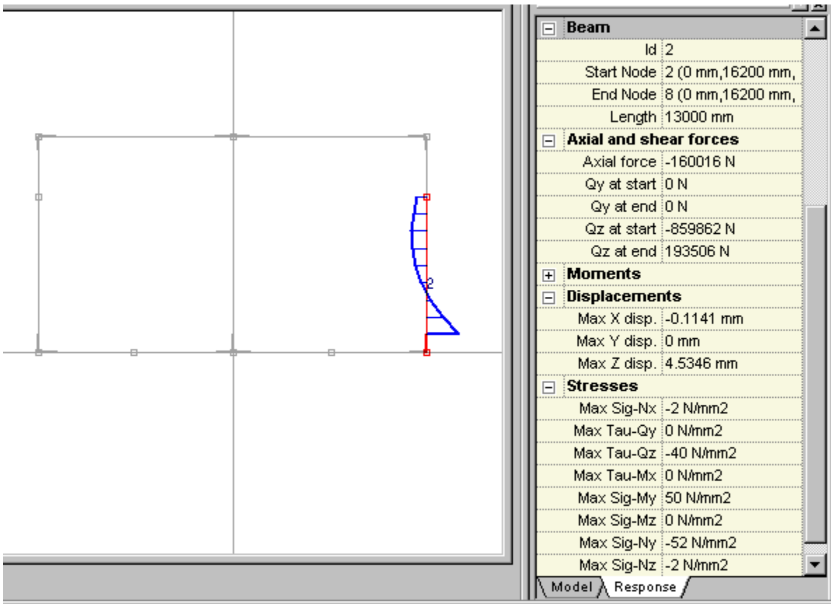


Figure 3.6-9 Responses in individual beams and nodes

You may scroll the list up and down by dragging the vertical scroll bar on the right hand side of the list. The headings (in bold letters) may be collapsed or expanded by clicking the [+] or [-] sign next to the headings.

For more information about the responses in the *Response* tab card see 4.5 "Response property window".

3.7 Reporting

3D Beam provides a flexible reporting function using MS Word97, or higher, to generate the reports. The reports are based on pre-defined templates for *Beams*, *Nodes*, *Profiles*, *Beam loads*, *Node loads*, *Beam responses Stresses*, *Node responses, and Response plot.* You may copy and past any image of the model and the *Response plot* diagrams, with input and response properties, from 3D Beam to MS Word.

• In the *Stresses* table in the *Output window* click the right mouse button and select the Report option from the pop-up menu.

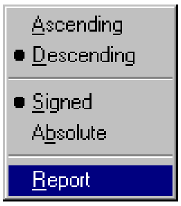


Figure 3.7-1 Select the Report option

MS Word is started and the report is created.

• Apply beam numbers and bending moments about local y-axis on all elements in the *Model window*.

• Click the right mouse button in the *Model window* and select the *Copy model plot* | *To Clipboard* option from the pop-up menu. (Alternatively you may activate the *Model window* and then press the Ctrl+C key combination on the keyboard.)

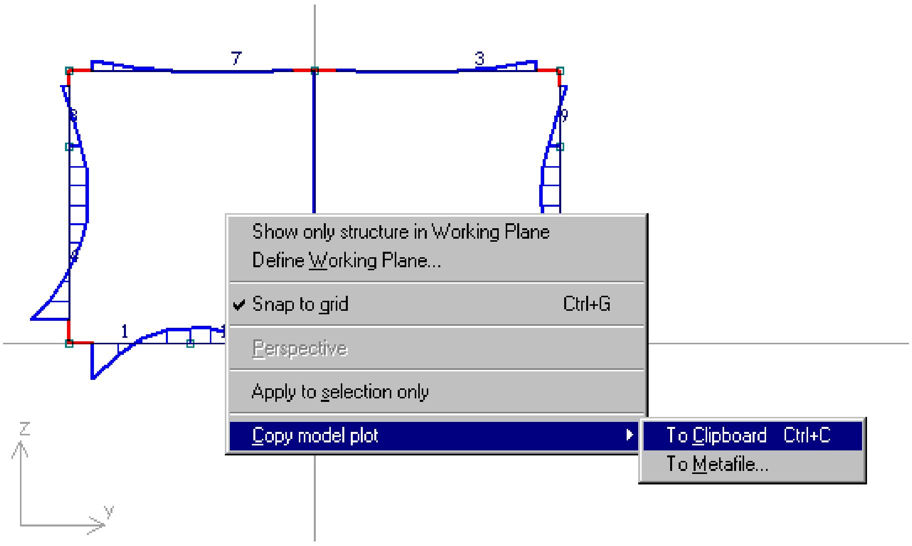


Figure 3.7-2 Create an image-copy of the model

• Enter the *Beam Stresses* report in MS Word and paste the image by pressing the Ctrl+V key combination on the keyboard.

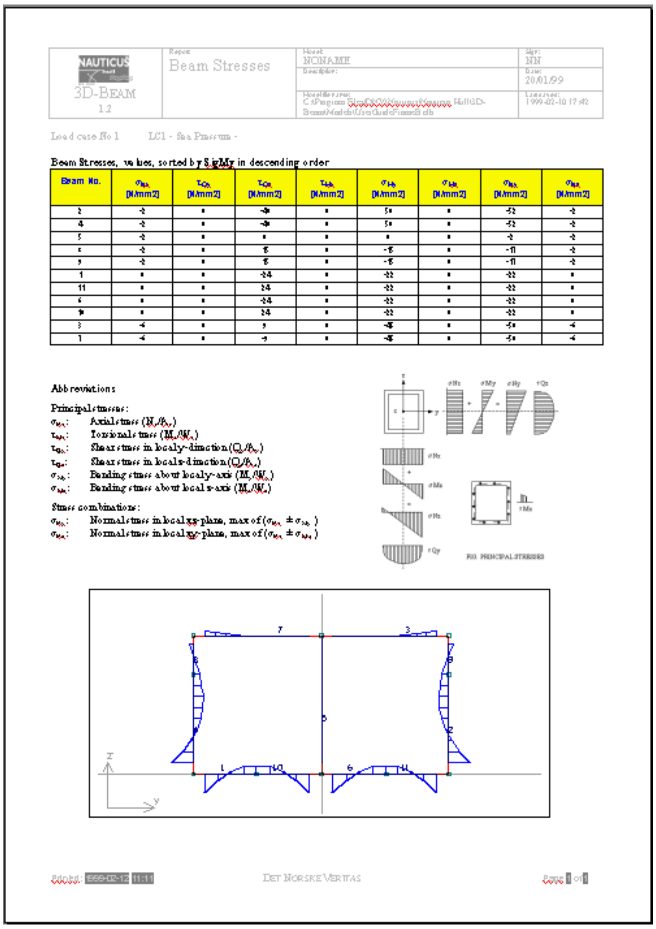


Figure 3.7-3 Stresses report

You may use the same procedure as described above to create a *Beam responses* report, a *Node responses* report and refined *Response plot* reports that includes <u>all responses</u> at user defined positions along the beam:

• Enter the Response plot view in the *Output window*

• Select beam number 2 in the model

• Select Sig-My from the *Response* drop down list in the *Response plot* view

• Click the left mouse button at following positions along the beam to define at which sections the responses are to be reported: 1500 mm, 2000 mm, 12500 mm and 13000 mm. (Alternatively the values may be entered in the *Ruler positions* list.)

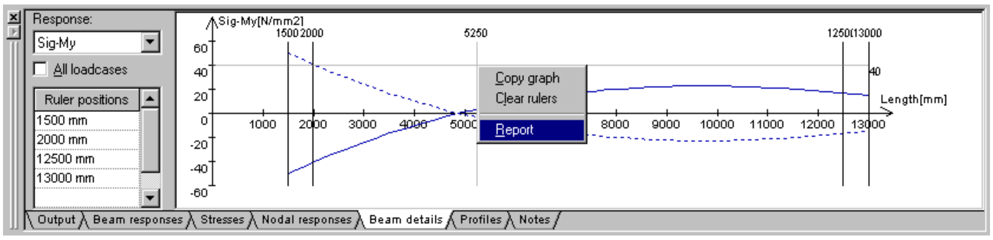


Figure 3.7-4 Create a Response plot report

• Click the right mouse button and select the *Report* option to create a *Response plot* report.

• Click the right mouse button and select the *Copy graph* option to make a copy of the graph, which you may paste into your report.

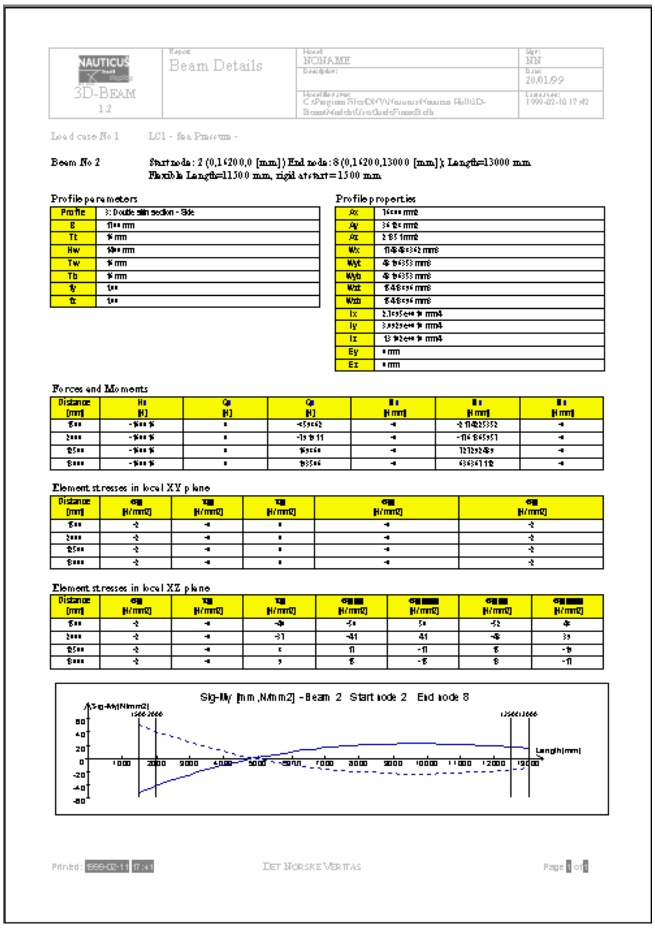


Figure 3.7-5 Response plot report

Note You should print or save the report and then <u>close</u> it before you create a new one.

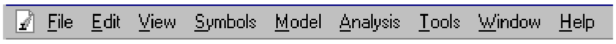
4 Command Reference

4.1 Introduction

This chapter summarises the available functions and features of the program and it gives an overview of the operations of 3D Beam. This includes the graphical user interface, a summary of the shortcut keys and a description of the menu- and toolbars, that are available.

4.2 Menu bar

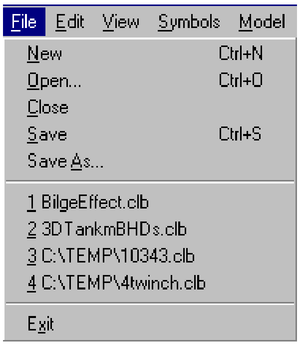
The 3D Beam menu bar:



In the following sections each of the items in the menu bar are described in detail.

4.2.1 File menu

The *File* menu described here is valid for a stand-alone version of 3D Beam only. See also 5 "3D Beam as part of NAUTICUS Project Manager"



New (Ctrl+N)

Opens a new blank project. You will be prompted to save the current model before starting a new.

Open… (Ctrl+O)

Open an existing file By selecting in the *Files of type* drop-down list you may open either 3D Beam files (.clb) or NV5040 files (I5040*.*).

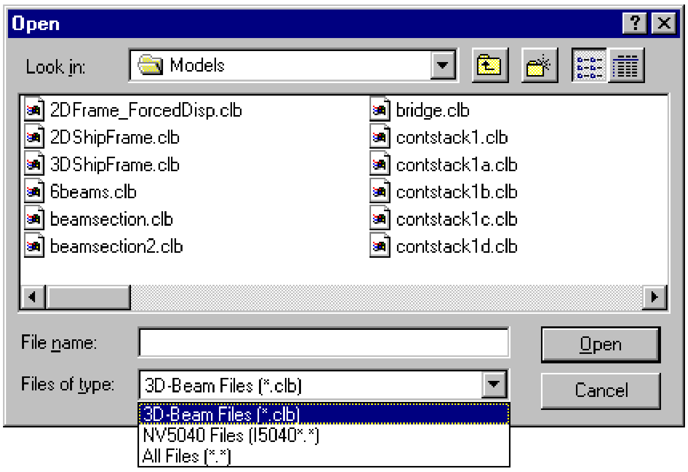


Figure 4.2-1 File | Open

For more information about NV5040 files, see 4.2.7 "Options / Directories"

Close

Closes the active model. If any changes have been made to the model file 3D Beam will prompt you to save the file.

Save (Ctrl+S)

Save active model with the same name to the active directory.

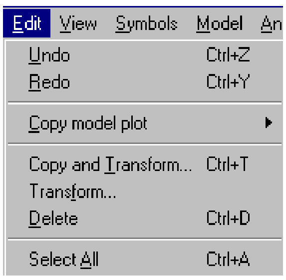
Save as…

Save model in a file with a new name or to a new directory.

Exit

Exit 3D Beam. 3D Beam will prompt you to save any changes to the model.

4.2.2 Edit menu



Undo (Ctrl+Z)

Undo the last action you carried out. The undo feature in 3D Beam has roll back functionality. This means that you can repeatedly undo an unlimited number of last actions.

Redo (Ctrl+Y)

Return to the last action you carried out before selecting undo, i.e. redo the undo.

Copy model plot

When you select: *Copy model plot* - *To Clipboard* (Ctrl + C) a picture of what you see in the *Model window* is copied to the clipboard. You may paste (Ctrl+V) the picture into any document, for example the 3D Beam report.

When you select: *Copy model plot* - *To Metafile…* a picture of what you see in the *Model window* will be created in an enhanced metafile (.emf).

Copy and Transform… (Ctrl+T)

Select the beams to be copied, mirrored or scaled before you apply this command.

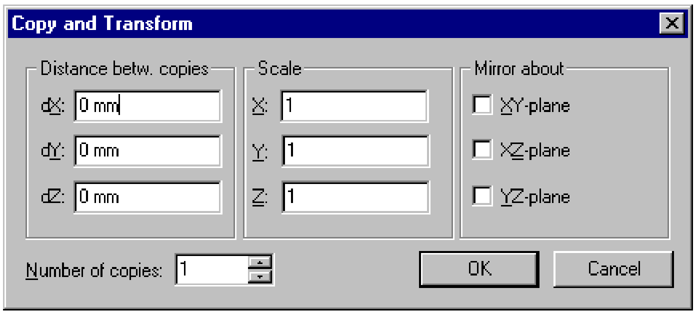


Figure 4.2-2 Copy and Transform dialog

In the *Distance betw. copies* fields you may specify the relative distance between the copied beams along one or more of the three main axes (X,Y,Z).

In the *Scale* fields you may specify a multiplication factor to scale the length and/or the relative distance of the copied beams.

In the *Number of copies* field you should specify the number of copies to be created.

You may mirror the selected beams about one or more of the three main planes (XY, XZ and YZ) by ticking off in the "Mirror about" check boxes.

Transform…

Select the beams to be moved, scaled or flipped (about one of the main planes) before you apply this command.

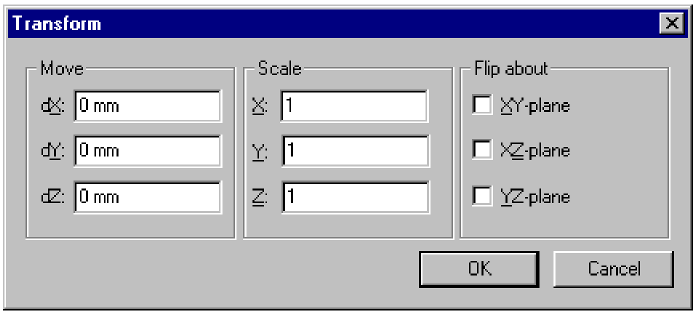


Figure 4.2-3 Transform dialog

In the *Move* field you may specify the relative movement of the selected beams in one or more of the three main directions (X,Y,Z).

In the *Scale* fields you may specify a multiplication factor to scale the length and/or the relative movement of the selected beams.

You may flip the selected beams about one or more of the three main planes (XY, XZ and YZ) by ticking off in the *Flip about* check boxes.

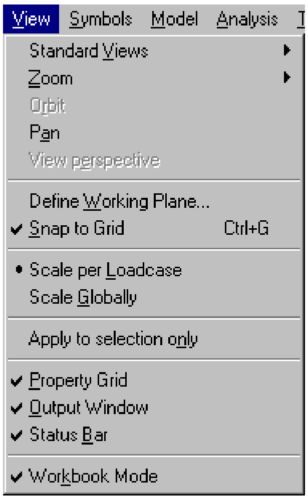
Delete (Ctrl+D)

Deletes the selected beam(s) with their nodes unless the nodes are shared with remaining beams.

Select All (Ctrl+A)

Selects all beams in the model.

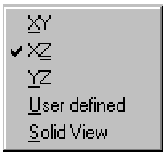
4.2.3 View menu



Standard Views

Activates one of the predefined views.

XY = top view, XZ = side view, YZ = aft view, User defined = ISO view / 3D view. Select beams before you apply solid view.



Zoom

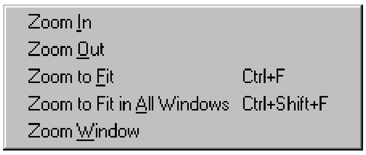
Activates one of the zoom functions.

The *Zoom In* and *Zoom Out* functions provide a stepwise zoom of the active window.

The *Zoom to Fit* function zooms the model to fit in the active window.

If you want the zoom the model to fit in all windows select the *Zoom to Fit in All Windows* function.

The *Zoom Window* function provides zooming of any part of the current display. Press the left mouse button to activate the rubber band. Drag the rubber band to surround the area of interest. Release button to apply the zoom.



Shortcut key to zoom active window to fit: Ctrl+F

Shortcut key to zoom all windows to fit: Ctrl+Shift+F

Orbit

Rotate the model by selecting the Orbit function. Press the left mouse button at the model in the User defined (ISO) view and rotate it by moving the mouse. The model will rotate around the centre of the current display.

The Orbit function does only work in the User defined (ISO) view.

Pan

Moves the model in the current view. Press the left mouse button at the model in any view and move it by moving the mouse.

View perspective

Displays the model in perspective in the ISO view.

Define working plane…

When working in 3 dimensions this function is useful. In the *Working plane at …=* field you may define a specific plane along one of the three main axes in which you will work. Only the working plane will be displayed.

When you check the "*Show only structure in working plane*" check box, only nodes and beams in the working plane are enabled.

Tab: Jump along global axis in positive direction, to the next plane where nodes are defined.

Shift+Tab: Jump along global axis in negative direction.

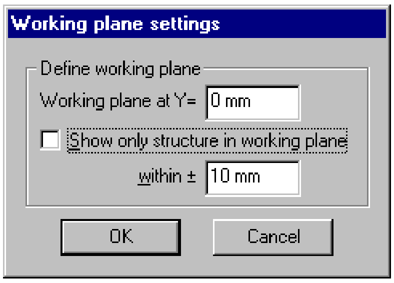


Figure 4.2-4 Specification of Working plane

In the "*within +/-*" field you may define the depth of the active "working-plane". For most models "within +/- 10 mm" is suitable.

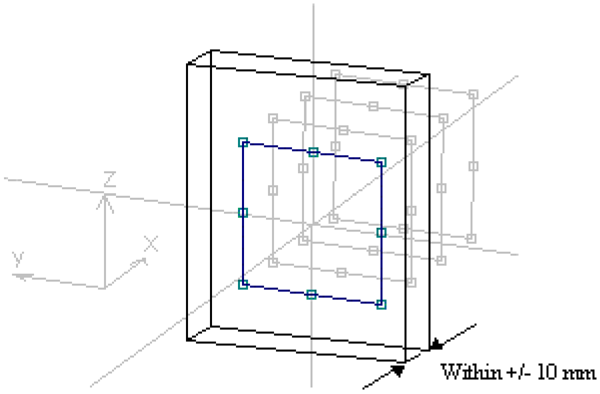


Figure 4.2-5 Depth of working plane

Snap to Grid (Ctrl+G)

Toggles the Snap to Grid on/off. The grid is available in all views except the User defined (ISO) view. The grid makes it easier to draw your model with the *Create Beams* tool.

Scale per Loadcase

Scales the response display symbols, such as moment- and shear force diagrams, deflection curves, stress colours etc., and the input loads against the maximum value within each load case.

Scale Globally

Scales the response display symbols, such as moment- and shear force diagrams, deflection curves, stress colours etc., and the input loads against the maximum value of all load cases.

Apply to selection only

When *Apply to selection only* is active then information is shown only for <u>selected</u> beams and nodes. This includes all properties for the selection as well as loads and responses.

Note If it suddenly looks as if all loads and responses have "disappeared", check that the *Apply to selection only* has not inadvertently been activated.

Property Grid

Show/ hide the *Input property window*.

Output window

Show/ hide the *Output window*.

Status Bar

Show/ hide the status bar at the bottom of the 3D Beam window.

In the status bar you will find the following information fields from the left:

1. Context sensitive information: Displays information of toolbar button functions

2. Global X-coordinate: Displays the global X coordinate of the position of the cursor

3. Global Y-coordinate: Displays the global Y coordinate of the position of the cursor

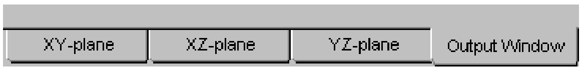
4. Global Z-coordinate: Displays the global Z coordinate of the position of the cursor

5. Snap to grid on /off. If *Snap* is displayed the snap to grid function is turned on. You may toggle the snap to grid by pressing the Ctrl+G combination on the keyboard.

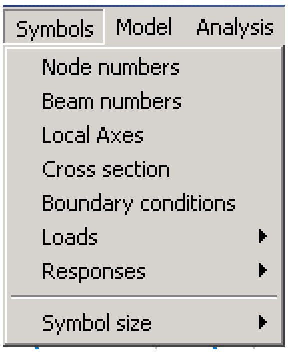
6. Global scaling on / off. If *Global scale* is displayed the global scaling of input and response symbols is scaled based on the maximum value across all load cases. If *Loadcase scale* is displayed the input and response symbols is scaled based on the maximum value within active load case.

Workbook Mode

Turns the *Workbook Mode* on/off in 3D Beam. In workbook mode the windows are arranged as a workbook with a tab card for each window.

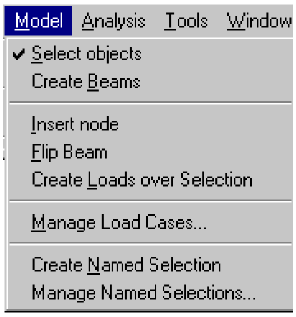


4.2.4 Symbols Menu



Click on any of the items in the list to toggle the display of the corresponding item on the model.

4.2.5 Model menu



Select objects

To select nodes and beams in your model, activate the *Select objects*.

Create Beams

To created new beams by drawing them, activate the *Create Beams*. The cursor becomes a cross and you may draw new beams.

Insert node

Insert a new node on selected beams.

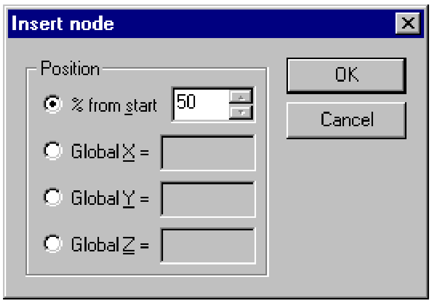


Figure 4.2-6 Insert node

To insert a node on one or more beams, first select the beam(s). When clicking the insert node tool button in the geometry toolbar the insert node dialog box appears. You may specify the insert position on the beam(s) either as a percent of the beam length from the start node or by an absolute reference along one of the global axes.

Flip Beam

The local X-axis for a beam is automatically set to point from the start node to the end node. In some cases, e.g. for display of the response plot, it is desirable to swap the start and end node of a beam, and hence flip the direction of the local X-axis. Clicking this option will flip all selected beams. Note that the direction of the local Y- and Z-axes may automatically change too, and the direction of local loads may have to be changed.

Create Loads over Selection

With the Create Loads over Selection tool you may generate a varying (or constant) local distributed load over a selection of beams.

See below figure for the operations of this tool.

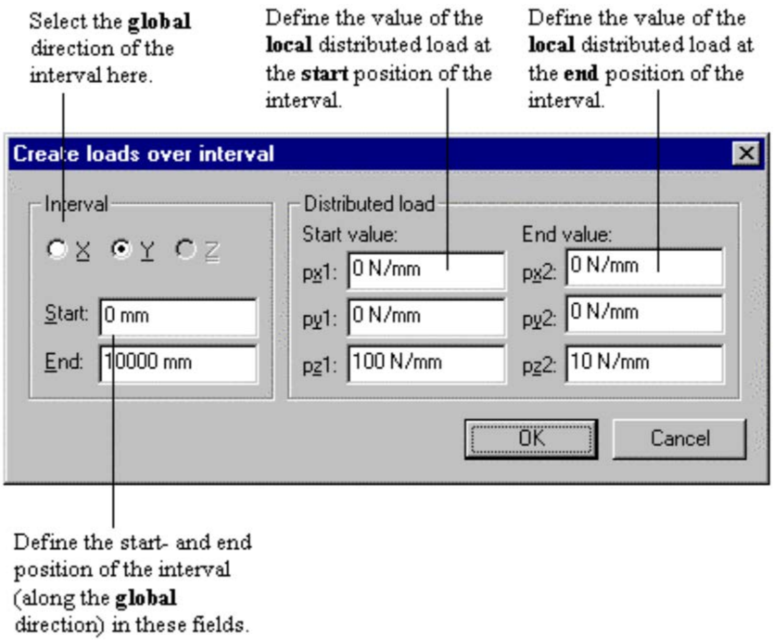


Figure 4.2-7 Create loads over interval

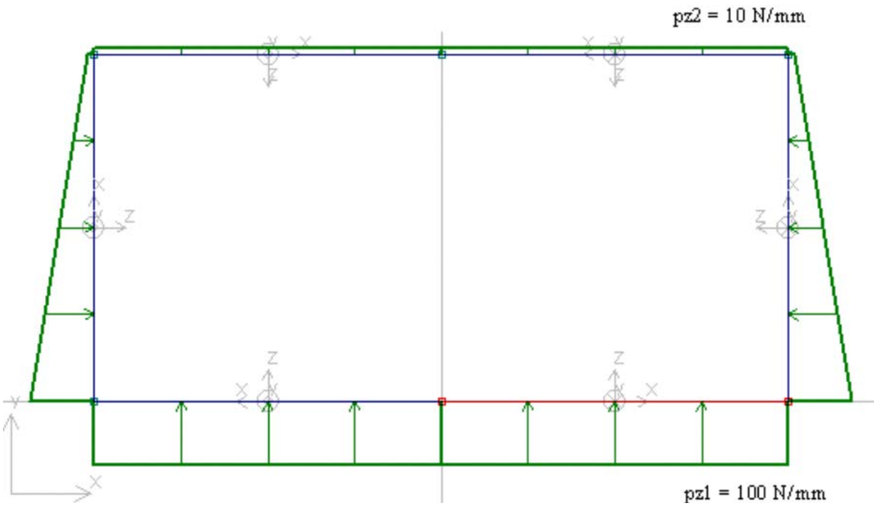


Figure 4.2-8 Distributed load over interval

Manage Load Cases…

In the Load Case Manager you can define new load cases, combine existing load cases, delete load cases or rename load cases.

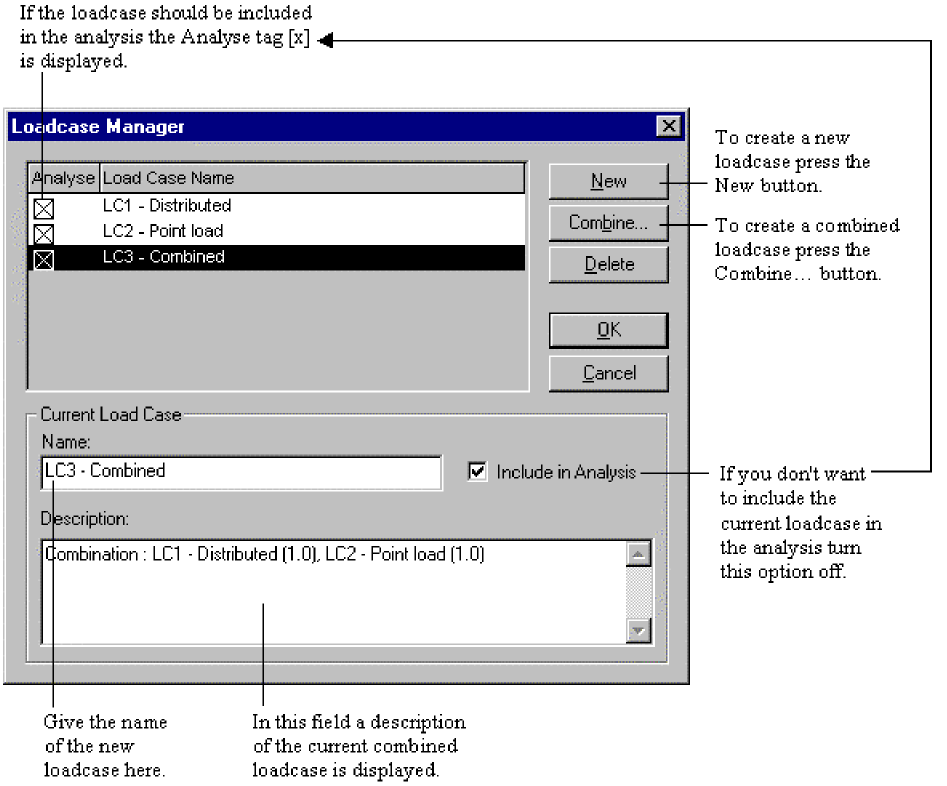


Figure 4.2-9 Load case manager

When clicking the Combine… button in the Load case Manager the Combine Load cases dialog box appears.

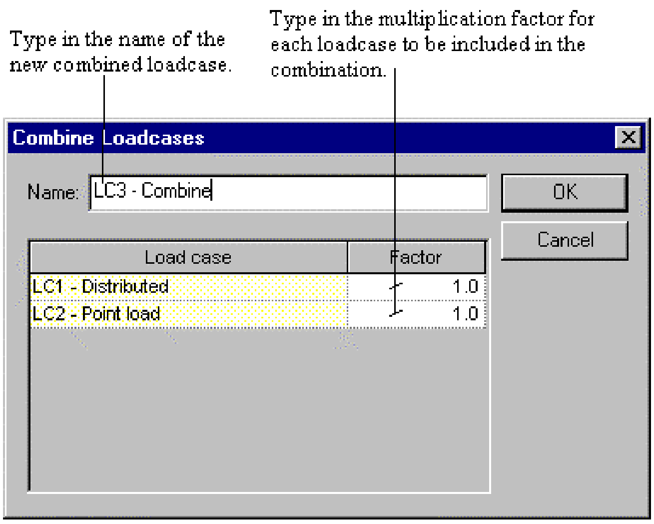


Figure 4.2-10 Combine Load cases

The multiplication factor scales all the load components, also the temperature loads, with the given factor.

Note The combined load case is not linked to the basis load cases. If you make a change in the load components in one of the basis load cases this is not reflected in the combined load case, and you should make a new combination.

Create named selection

In some cases, especially for large models, it may be convenient to identify parts of the structure with a name, e.g. "Outer shell", "Main supports", "New profiles". This allows quick access to these parts for input specification and result analyses. To create a *Named Selection* simply select the desired beams and/or nodes and click *Create Named Selection*. The default name given to the selection is "Set#*n*", where *n* is automatically incremented. To specify a more descriptive name to selection, select the option *Manage Named Selections*.

Manage named selections…

In the Named Selection Manager you may specify names to, give descriptions of, and delete, *Named Selections*.

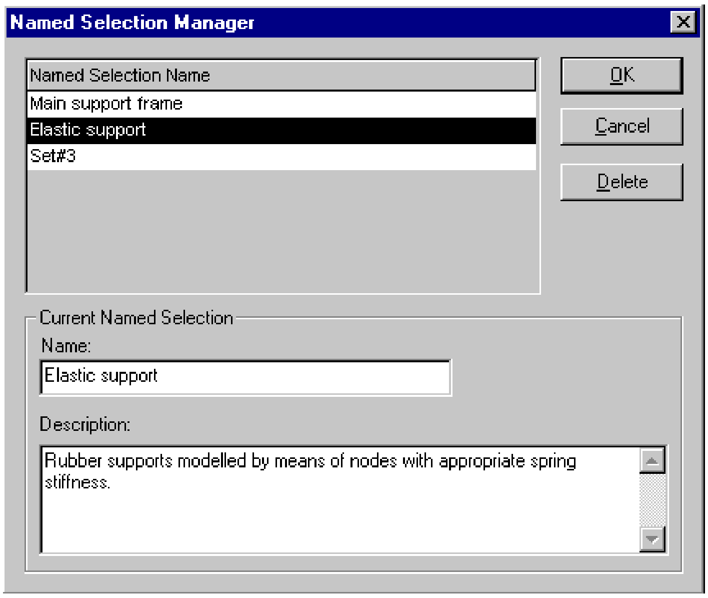


Figure 4.2-11 Named Selection Manager

The new names will not appear until the OK button is clicked.

4.2.6 Analysis menu

Analyse (F9)

Run the analysis.

4.2.7 Tools menu

Beam Wizard (Ctrl+W)

With the *Beam Wizard* you may generate beams. On the *Create Connected Beams* tab card you may generate a continuous chain of new beams by defining each node coordinate in the X,Y and Z column. When you click the Apply button the beams are created in the *Model window*.

In the *Default* row you may give a default value which is used unless another value is specified for the node. In below figure all Z-values are defined by the default value as 0 mm.

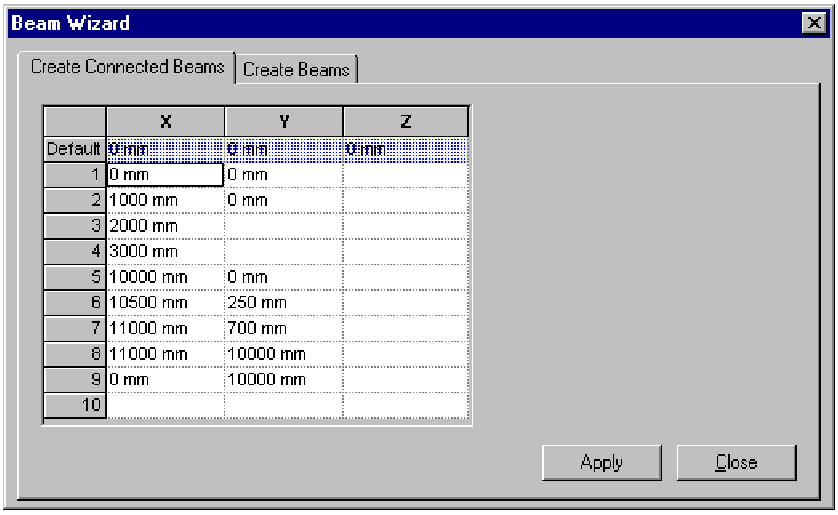


Figure 4.2-12 Beam wizard-Create Connected Beams

On the *Create Beams* tab card you may generate a set of separate beams by defining the coordinate at each end of the beam in the X,Y and Z columns. Coordinates may also be pasted from a spreadsheet. When you click the Apply button the beams are created in the *Model window*.

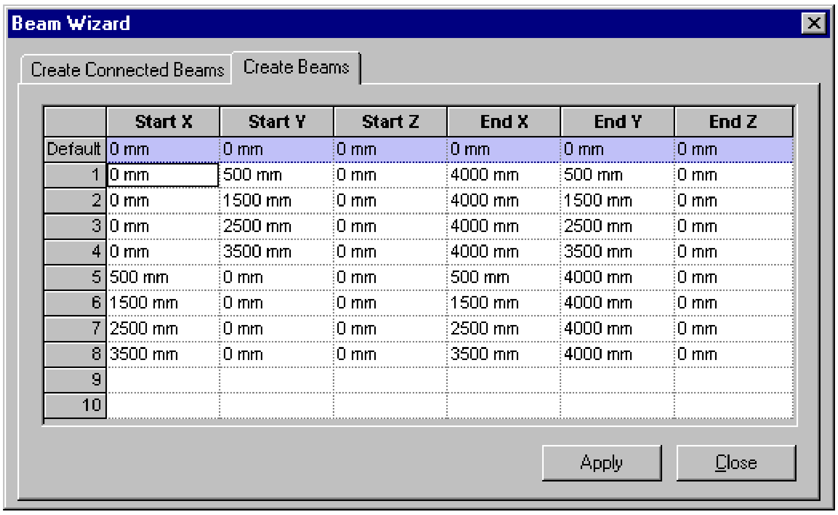


Figure 4.2-13 Beam wizard - Create Beams

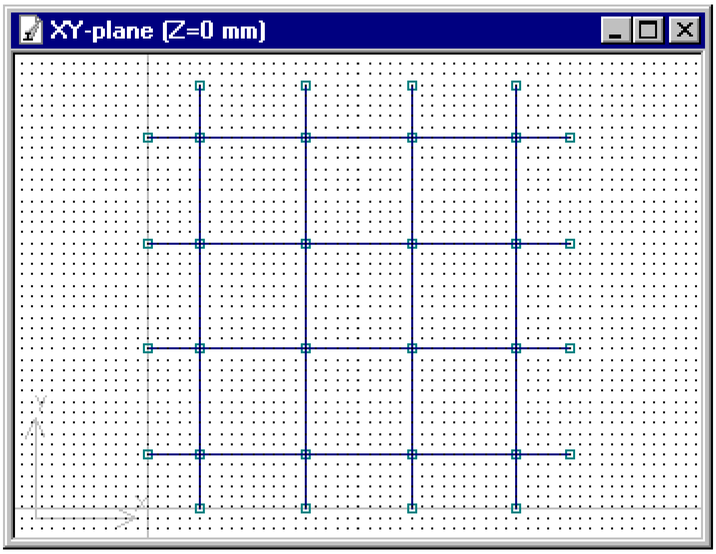


Figure 4.2-14 Grillage structure generated with the Beam Wizard

You may copy (Ctrl+C) and paste (Ctrl+V) the numbers within the beam wizard or from and external source as for example MS Excel. This makes it possible to tailor make simple tools to calculate node coordinates of general geometric shapes. In the NAUTICUS Hull package you will find a spreadsheet which calculates node coordinates of various geometries.

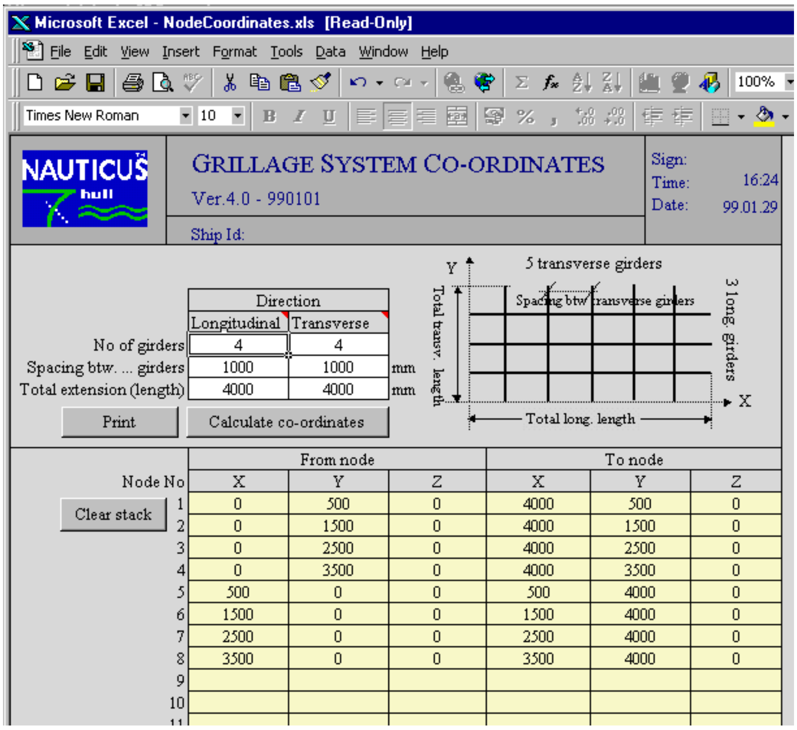


Figure 4.2-15 Generate node coordinates with spreadsheet

See "Appendix B: Using spreadsheets with 3D Beam" for more details about how to use spreadsheets to calculate node coordinates.

Options…

Options / Display

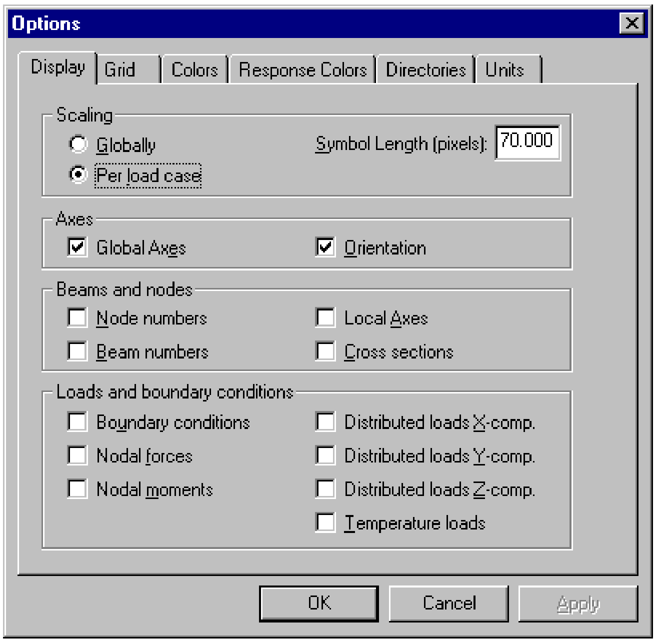


Figure 4.2-16 Options / Display

Scaling

Globally scaling of symbols in the *Model window* means that response symbols (e.g. moment diagrams) and the input loads are scaled (globally) across all load cases. Each response / input load is scaled with respect to the maximum value of the response / input load across all load cases.

Example: If the moments on the beam in question are 10 times smaller than the maximum moment value of any beam across all load cases, the moment diagram symbol becomes 10 times smaller than the maximum value.

Scaling *Per load case*: means that response symbols (e.g. moment diagrams) and the input loads are scaled per load case. Each response / input load is scaled with respect to the maximum value of the response / input load in the active load case.

Example: If the moments on the beam in question are 10 times smaller than the maximum moment value of any beam across the active load case, the moment diagram symbol becomes 10 times smaller than the maximum value.

Symbol length (pixels): Defines the extension of the symbols in pixels.

Axes

Global axes: Turn on/off the global axes symbol in the *Model window*.

Orientation: Turn on/off XYZ-axes orientation symbol (i.e. the arrow cross).

Beams and nodes

Turn on/off the: *Node numbers, Beam numbers, Local Axes symbols, Cross section* symbols.

Loads and boundary conditions

Turn on/off the *Loads and boundary conditions* symbols.

Options / Grid

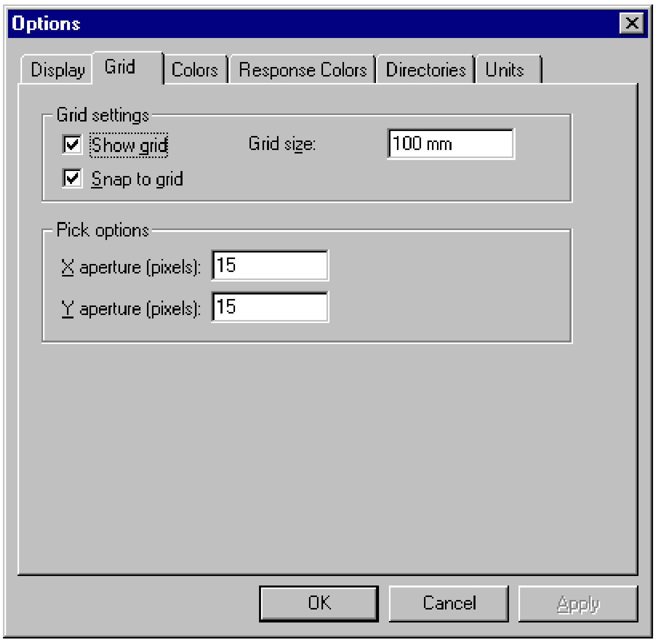


Figure 4.2-17 Options / Grid

Grid settings

Show grid: Turn on/off the grid point symbols in the 2D views in the *Model window*.

Snap to grid (Ctrl+G): Turn on/off the Snap-to-grid function. If you draw a model in the *Model window* with Snap-to-grid activated the nodes are being snapped to the nearest grid point.

Grid size: Defines the size between each grid point.

Pick options

X aperture (pixels): Defines the pick accuracy of the cursor in the horizontal direction. The smaller value the smaller pick range (but more accurate).

Y aperture (pixels): Defines the pick accuracy of the cursors in the vertical direction. The smaller value the smaller pick range (but more accurate).

Options / Colours

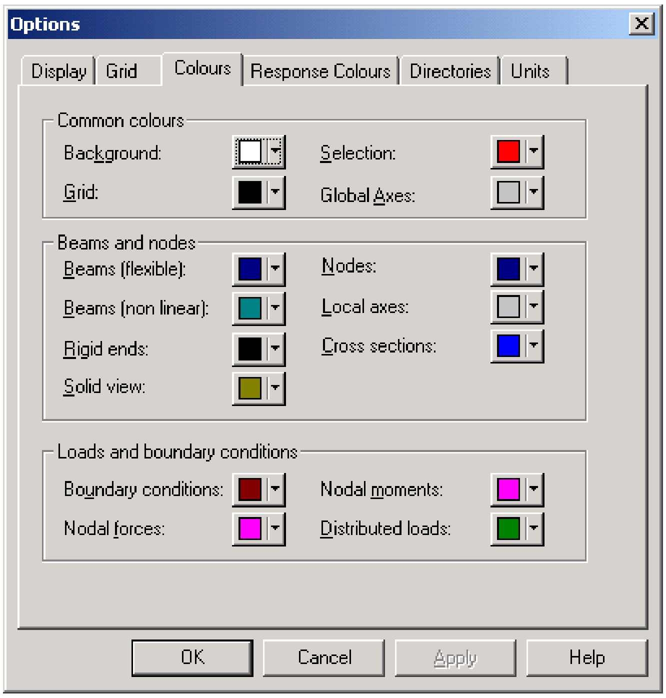


Figure 4.2-18 Options / Colours

Common colours

Defines the colour of the background of the *Model window*, grid points, and beams when selected (Selection) and global axis symbols.

Beams and nodes

Defines the colour of the flexible part of the beams, non linear beams, rigid ends, the solid view, node symbols, local axis symbols and cross section symbols.

Loads and boundary conditions

Defines the colour of the load symbols and the boundary condition symbols.

Options / Response Colours

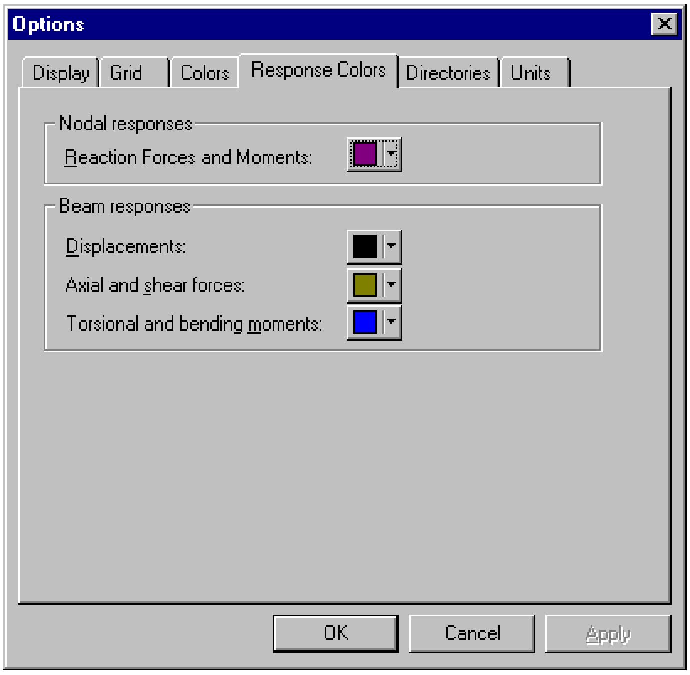


Figure 4.2-19 Options / Response Colours

Node responses

Defines the colour of the node reaction forces and moment symbols.

Beam responses

Defines the colour of the beam response symbols.

Options / Directories

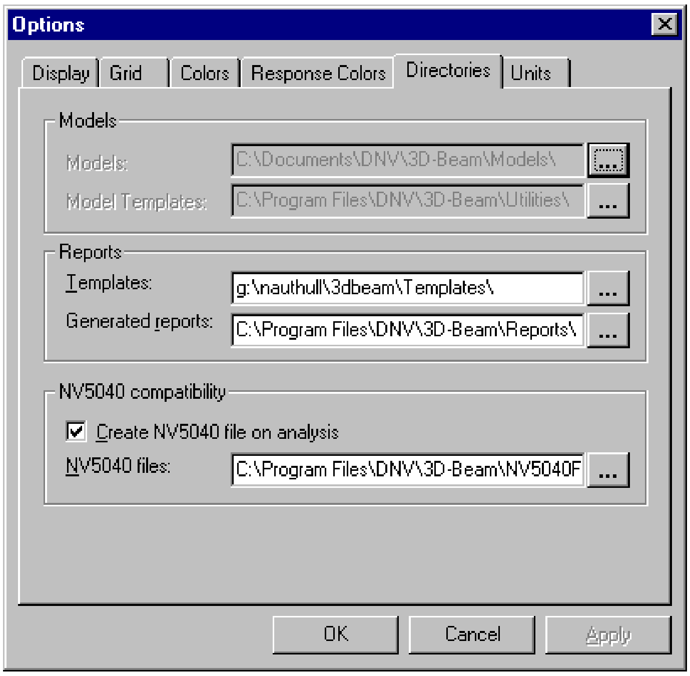


Figure 4.2-20 Options / Directories

Models

Not implemented.

Reports

Templates: Defines the path to the directory where the MS Word report templates are located. These templates are needed to create the reports. The directory is usually defined during the installation as a sub directory to the 3D Beam directory.

Generated reports: Defines the path to your default directory of generated reports.

NV5040 compatibility

Create NV5040 file on analysis: If this option is activated 3D Beam will create one NV5040 input file and one NV5040 result file during the analysis. These files may be read by the PILOT program NV5040, which is DNV's previous beam analysis program. The NV5040 input file may also serves as a backup file in case the 3D Beam model file (.clb) becomes corrupt.

NV5040 files: Defines the directory where the NV5040 files are stored.

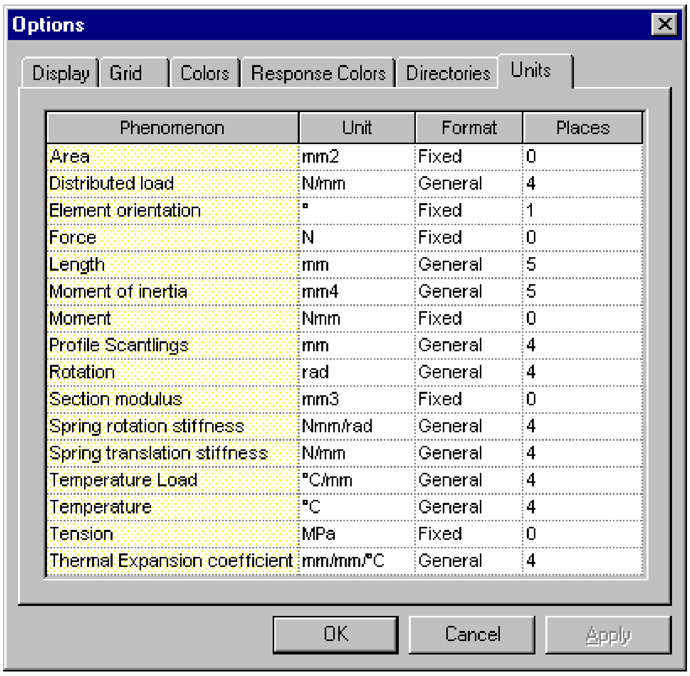
Note The directory for the NV5040 files <u>must exist</u>.

<u>Options / Units</u>

From the Units dialog box you may define the default display units. In the *Format* and *Places* fields you may change the number format and the number places of the phenomenon.

Example:

Format:	Fixed	Fixed	General	Scientific
Places:	5 places	0 places	6 places	5 places
	17927.58220 mm	17928 mm	17927.6 mm	1.79276e+004 mm



Phenomenon	Unit	Format	Places
Area	mm2	Fixed	0
Distributed load	N/mm	General	4
Element orientation	°	Fixed	1
Force	N	Fixed	0
Length	mm	General	5
Moment of inertia	mm4	General	5
Moment	Nmm	Fixed	0
Profile Scantlings	mm	General	4
Rotation	rad	General	4
Section modulus	mm3	Fixed	0
Spring rotation stiffness	Nmm/rad	General	4
Spring translation stiffness	N/mm	General	4
Temperature Load	°C/mm	General	4
Temperature	°C	General	4
Tension	MPa	Fixed	0
Thermal Expansion coefficient	mm/mm/°C	General	4

Figure 4.2-21 Options / Units

Phenomenon

The list of phenomenon shows the quantities in 3D Beam that may be formatted by the user.

For each phenomenon you may change the unit, the format and the number of places.

The *Length* applies to beam length, rigid-end length, deflections and the unit of the coordinate system.

Unit

From the drop down list in the *Units* fields you may select one of the predefined units in 3D Beam.

Format

From the drop down list in the *Format* fields you may select one of the predefined formats in 3D Beam.

Places

From the drop down list in the *Places* fields you may select the number of number places to be used by 3D Beam.

Customize…

Activates the Customize-toolbar dialog box.

Customize / Toolbars

In the Customize dialog box you may customize your toolbar configuration. By selecting / un-selecting toolbars in the toolbars list you turn on / off the toolbars.

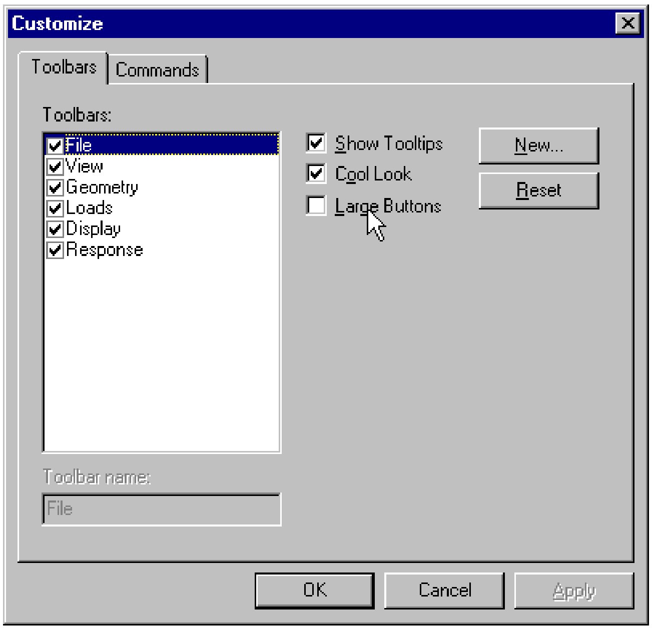


Figure 4.2-22 Customize / Toolbars

You may create a new customized toolbar by clicking the **New…** button. Give the new toolbar a name and add new tool buttons to the new toolbar from the Commands tab card.

Customize / Commands

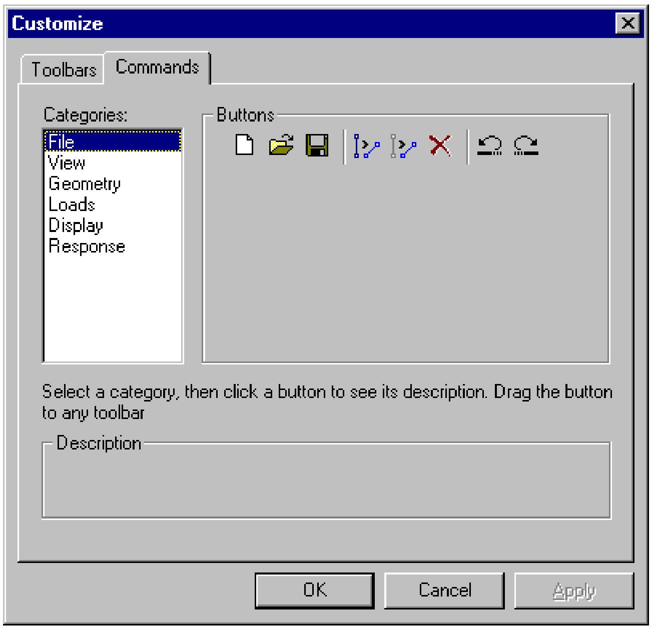
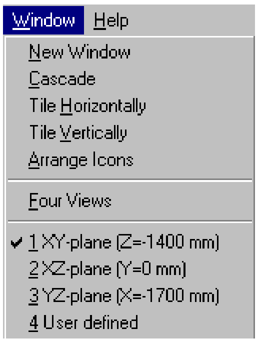


Figure 4.2-23 Customize / Commands

To add a tool button to a toolbar first select a category, then click a button and drag the button to the toolbar. When you click a tool button you may see the description of the tool button in the description field.

4.2.8 Window menu



New window

Creates a new *Model window*. All current windows remain unchanged.

Cascade

Cascades all windows in the workbook.

Tile Horizontally

Tile all windows in workbook horizontally.

Tile Vertically

Tile all windows in workbook vertically.

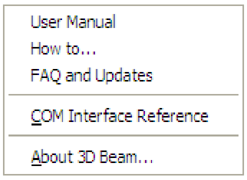
Arrange Icons

Arrange the icons of minimised windows at the bottom.

Four Views

Regenerates all *Model windows*. Creates the four default views: XY-plane, XZ-plane, YZ-plane and ISO view. The model is zoomed to fit in all windows.

4.2.9 Help menu



User Manual

Opens this User Manual

How to…

Opens the 3D Beam How to… manual with quick references to how to accomplish various tasks.

FAQ and updates

Connects to the Frequently asked questions and updates on DNV Software web page.

COM Interface Reference

Opens the manual describing the COM interface to 3D Beam.

About 3D Beam…

Tells which version of 3D Beam you are using.

4.3 Toolbars

4.3.1 Show or hide toolbars

The toolbars provides easy access (short cuts) to the most used functions and features of 3D Beam. All functions and features are described in detail in 4.2 "Menu bar".

From the Tools | Customise menu you may show or hide the various toolbars. For more details about customising the toolbars see 4.2.7 "Tools menu". The toolbars may be rearranged by dragging them to a new location on the screen.

The various toolbars may be toggled by clicking the right mouse button somewhere in the toolbar area (at the top of the 3D Beam window). See below figure.

Figure 4.3-1 Toggle toolbars

Default Settings

Regenerates the default window- and toolbar configuration of 3D Beam. When you apply the Default Settings the following message appears. To activate the default window configuration you should close 3D Beam and restart.

4.3.2 File toolbar

The File toolbar provides easy access to the save, open and new file functions. You will also find Copy and Transform, Apply Transform, Delete Beams and the Undo / Redo features here.

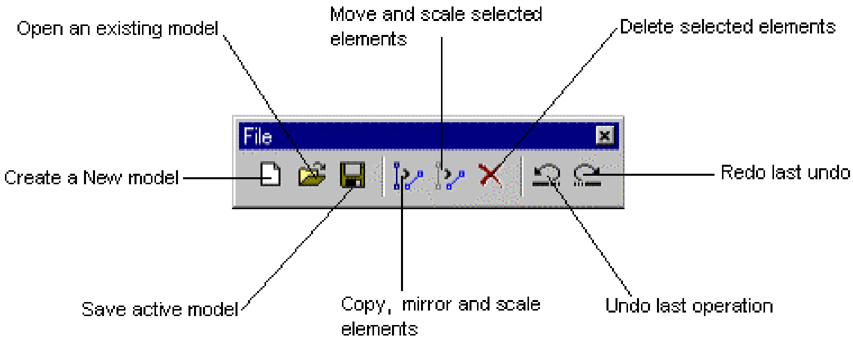


Figure 4.3-2 File toolbar

4.3.3 View toolbar

The View toolbar provides easy access to the various view features in 3D Beam. It includes zoom functions, view default working planes, arrange windows, pan, rotate, and solid view.

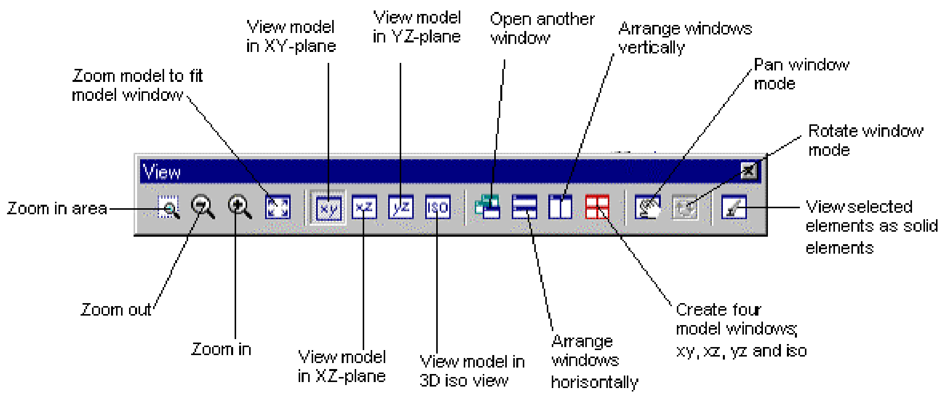


Figure 4.3-3 View toolbar

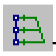
4.3.4 Loads toolbar

The Loads toolbar provides easy access to the load case manager and the distributed-loads-over-a-selection-of-beams function. With the select-load case drop down you can select among the defined load cases. Results and inputs relates to the current load case selected in the drop down.

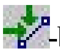
Figure 4.3-4 Loads toolbar

To build a load case select the relevant load case from the load case drop down list and define the loads in the input property grid or use the Load wizards.

Click the -button to open the Load Case Manager.

Click the -button to open the Create loads over interval dialog.

Click the -button to open the Create loads due to inertia forces dialog.

Click the -button to open the Apply loads in global directions dialog.

4.3.5 Named Selection toolbar

The Named Selection toolbar

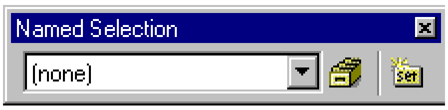


Figure 4.3-5 Named Selection toolbar

allows you to save a selection of beams and/or nodes for easy retrieval. To create a Named Selection, simply select the desired beams and/or nodes and click the -button.

The selections are automatically given the default names Set#1, Set#2, etc. To specify a descriptive name, press the -button to bring up the Named Selection Manager where the selection may be given a name and a description.

To retrieve a selection press the drop-down button and click the desired selection.

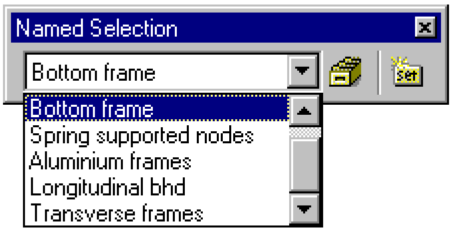


Figure 4.3-6 Retrieval of a Named Selection

4.3.6 Geometry toolbar

The Geometry toolbar provides easy access to the select tool with a node / beam filter, create beam tool, insert node tool and the beam wizard.

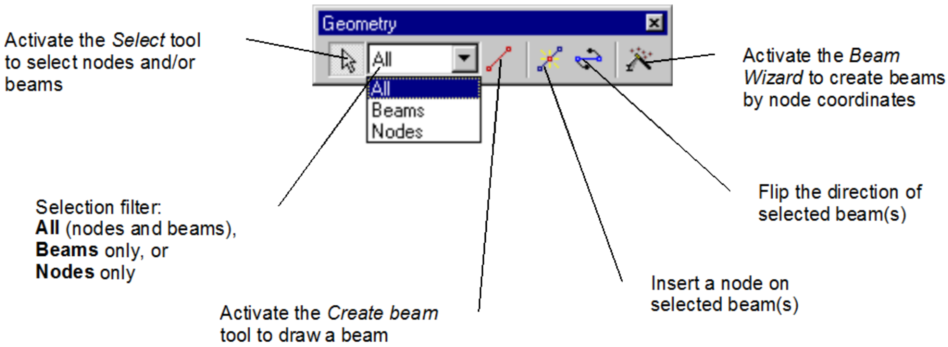


Figure 4.3-7 Geometry toolbar

4.3.7 Display toolbar

The Display toolbar provides easy access to display of symbols. In addition to turning on and off the various symbols, you have access to the Apply-to-selection-only function and the symbol scaling. The Apply-to-selection-only function makes you able to select a subset of the model on which you can display input and result information.

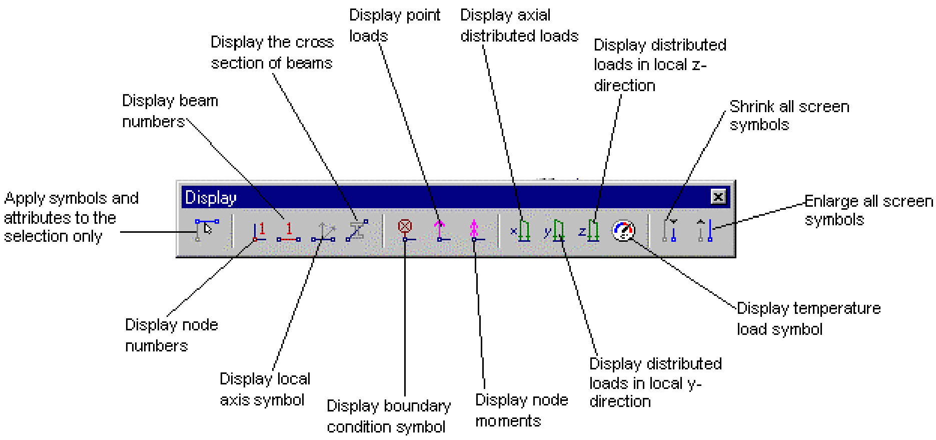


Figure 4.3-8 Display toolbar

4.3.8 Response toolbar

The Response toolbar provides easy access to the response symbols and the analysis tool. The analysis commences when clicking the analysis tool button. When selecting a stress component in the Stress type drop down a colour plot of the stress distribution is shown on the model.

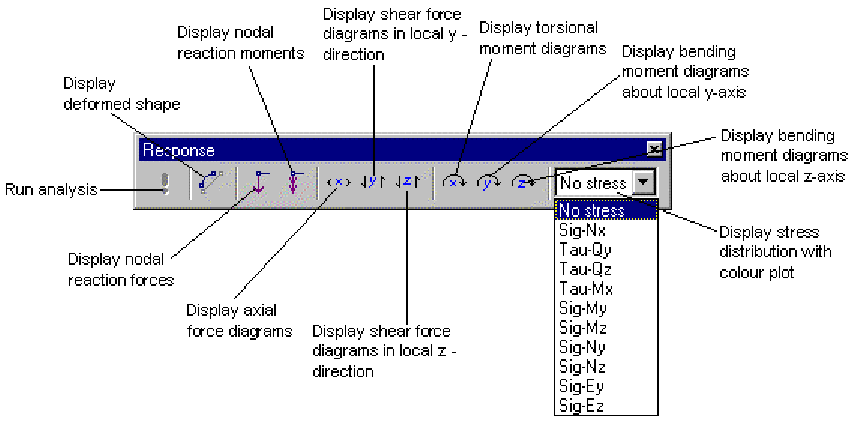


Figure 4.3-9 Response toolbar

4.4 Input property window

4.4.1 Input property window overview

This chapter summarise the functions and features of the *Input property window*. The *Input property window* is located on the right hand side in 3D Beam. See below figure.

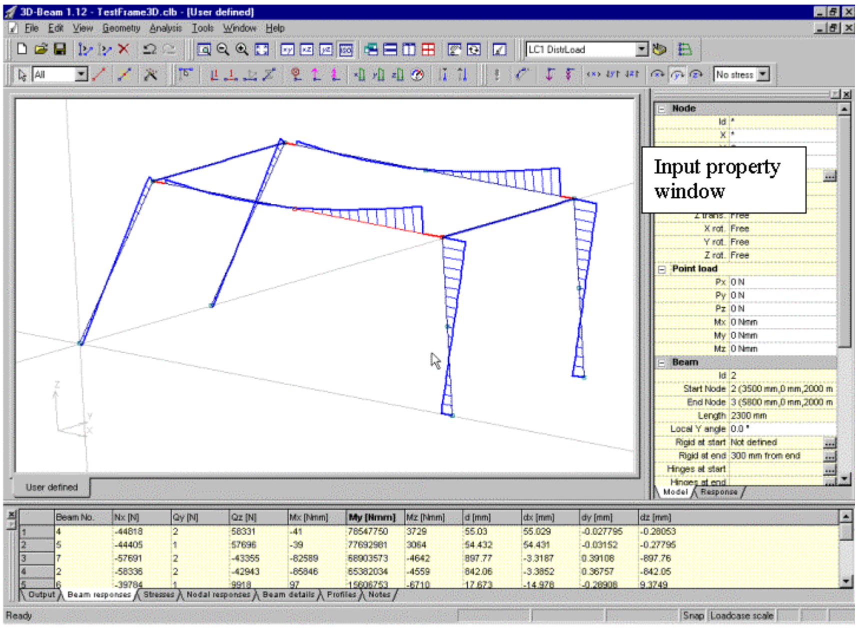


Figure 4.4-1 Input property window

The *Input property window* is the user interface for applying properties to nodes and beams. In the *Input property window* you define the boundary conditions, loads, profile properties including material and orientation of the local beam axes, hinged ends and rigid ends.

Below each heading in the *Input property window* input- and output data fields are available. Fields with white background are read- and write fields. Fields with yellow background are read-only fields. The fields are activated when a node/beam selection is made. The content of the fields reflects the properties of your node/beam selection. If the asterisk (*) is shown in a field the values are not the same for all nodes/beams in the selection.

Edit features

For an efficient model specification the following editing features are available:

Edit a selection: The value in the input field is applicable to the currently selected item or items. E.g.: two nodes are selected and the field for Z:

- shows a value, then the two nodes have that same Z position,
- shows an asterisk "*", then the two nodes have different Z positions,
- is given a value, then both nodes get this Z position.

Specific unit: Input may be specified in any valid unit. If a value without unit is entered, the specified display unit is used, see Tools | Options | Units. If a value and unit is entered, the value is converted to the specified display unit. E.g.: the unit for length is specified to metre, fixed 3 decimals. Entering 12' (twelve feet) then results in 3.658 m being displayed.

Delta modification: Input may be specified as an increment, e.g.:

+= 4.5 adds 4.5 to the value(s) in the field,

-= 5 in subtracts 5 inches to the value(s) in the field.

If the field shows an asterisk, i.e. the selected items are different, the increment is applied to the actual values in the selection.

4.4.2 Node properties

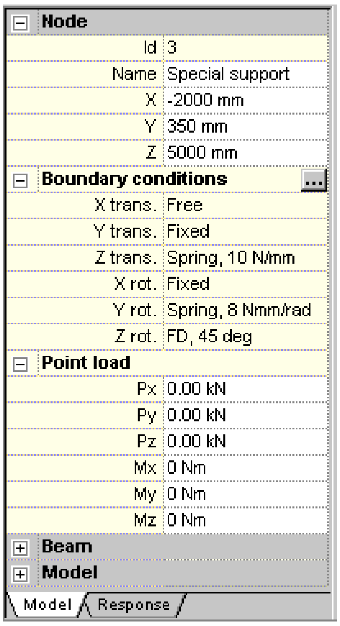


Figure 4.4-2 Input property window - Node properties

Node

Id:	Echo of the Node Id (node number) of selected node. Generated automatically by the program.
Name:	User's identification of selected node(s)
X:	Global X coordinate of selected node
Y:	Global Y coordinate of selected node
Z:	Global Z coordinate of selected node

Several nodes can be selected and edited simultaneously.

Boundary conditions

Following boundary conditions may be specified:

Free:	Free to move in specified direction. This is default.
Fixed:	Fixed in specified direction
Spring supported:	Spring supported in specified direction
Forced displacement:	Forced displacement in specified direction

To enter boundary conditions click the open button at the right hand side of the Boundary conditions heading.

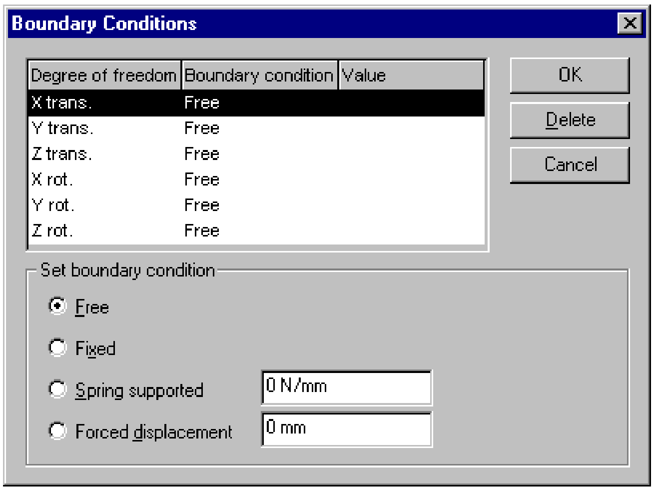


Figure 4.4-3 Input property window- Boundary conditions

Select the relevant degree of freedoms in the list. Press the Ctrl-key on the keyboard when selecting to make a multiple selection. Press the shift-key when selecting to select more/all items in one operation. If Spring support or Forced displacement should be defined by giving the value in the belonging input box.

Point load

Point loads may be applied to nodes only.

Px, Py, Pz:	Node load in global X-, Y-, and Z- direction
Mx, My, Mz:	Node moment about global X-, Y-, and Z- axis (positive for right-handed screw)

4.4.3 Beam properties

Following beam properties are available:

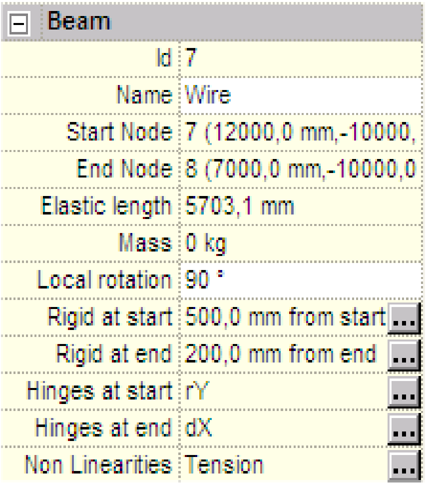


Figure 4.4-4 Input property window - Beam properties

Id:	Beam Id (beam number) of selected beam. Generated automatically by the program.
Name:	User's identification of selected beam
Start node:	The start node Id and coordinate
End node:	The end node Id and coordinate
Elastic length:	The length of the beam excluding possible rigid ends.
Mass:	The mass of the elastic length of the beam.
Local rotation:	Input and display of the angle of the local z-axis of the beam relatively to the default angle defined by 3D Beam. Input of a positive angle rotates the beam clockwise about the positive x-axis (right-hand rule). To rotate the beam relatively (clockwise) by adding an angle, type += in front of the angle (e.g. +=90 deg). To rotate the beam relatively (counter clockwise) by subtracting an angle, type -= in front of the angle (e.g. -=90 deg).

Rigid ends

A rigid end reduces the length of the flexible part of the beam. Lateral distributed load over the rigid part is transferred directly into the node. No forces, moments or deflections are calculated along the rigid part.

The rigid ends need to be defined at each beam end separately. You may define rigid end for several beams in one operation.

To enter the rigid ends dialog box click the open button ... at the right hand side of the rigid ends fields.

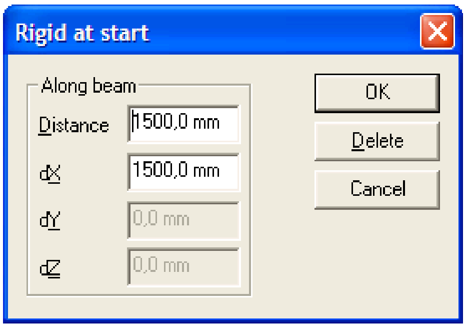


Figure 4.4-5 Input property window - Rigid end properties

Note It is not allowed to create a rigid end over the full beam length.

Hinges

A hinge is a property that eliminates the transfer of a force (translation) and/or a moment (rotation) between the beam-end in question and the adjacent beam-ends.

The hinges have to be defined at each beam-end separately. You may define hinges for several beams in one operation. One or several of the global degrees of freedom may be hinged.

To enter the hinged end dialog box click the open button at the right hand side of the hinged end fields.

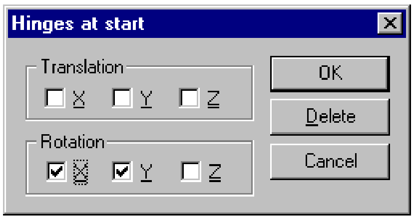


Figure 4.4-6 Input property window - Hinged end properties

Example of hinge end about global Z-axis:

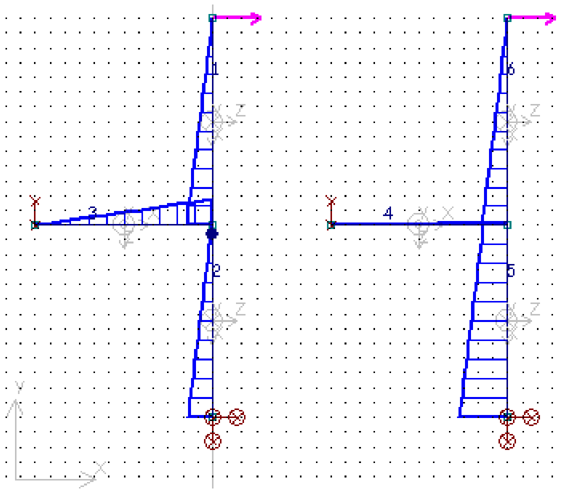


Figure 4.4-7 Hinge - No hinge

In above figure a rotation hinge is defined about global Z-axis at the top (start) of beam no. 2. (lower beam in the left hand figure). The hinge symbol is the small, filled circle. The bending moment is therefore not transferred from beam no. 1 to beam no. 2 in the beam intersection.

In the right hand figure no hinge is defined and the moment is transferred from the upper to the lower beam. (Note that in above example the bending stiffness of the horizontal beams is smaller than the vertical beams.)

Note The deformations of beams with hinges may be incorrect, especially if the hinge is at the start node of the beam. The moments and shear forces are however correct.

To alleviate the problem you may make the beams with the hinges very short. The easiest way to do this is to select the beam with the hinge, click the *Insert node* tool and insert a node 1%, or 99% from the start of the beam. Use 1% if the hinge is at the start of the beam, and 99% if the hinge is at the end of the beam. The effect of this is illustrated in the figures below, where a vertical translation hinge, dz., is defined.

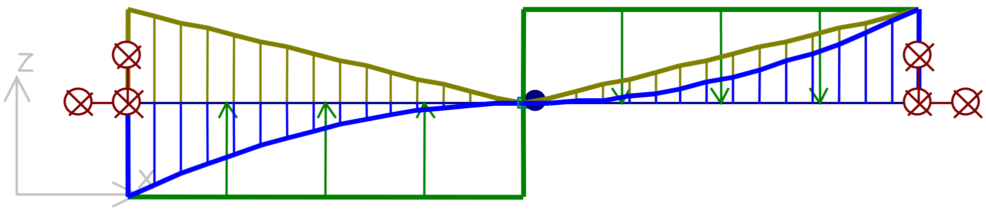


Figure 4.4-8 Two beams joined with a dZ hinge. Evenly distributed loads from below and above. Moment and shear distributions are correct

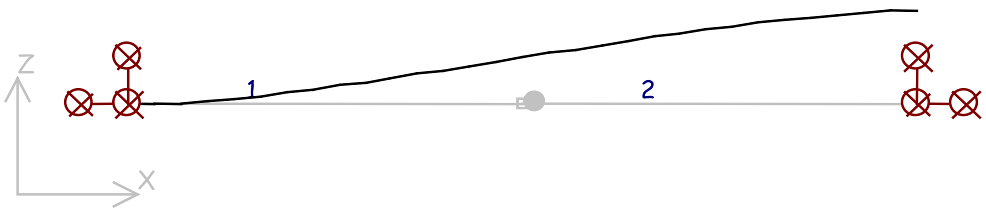


Figure 4.4-9 Beam no. 2 is hinged at the start node. Deformation incorrect

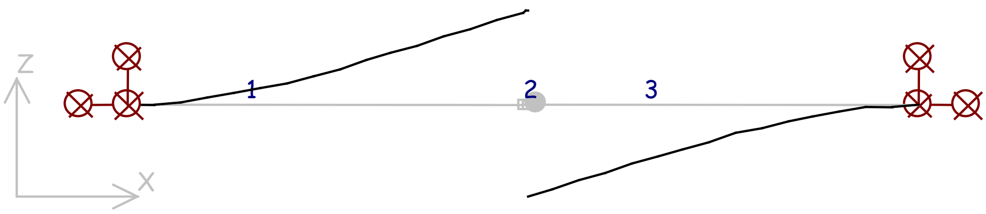


Figure 4.4-10 Beam no. 2 is split 1% of its length from its start node. Deformation corrected

Non Linearities

It is possible to define beams which are not linearly elastic, such as wires which cannot take compression. The possibility to model play in the structure is also included by means of specifying a gap. This means that a beam may be stretched, or compressed, a specified distance before it takes load. Hence, the non-linear properties can be defined according to the figure below.

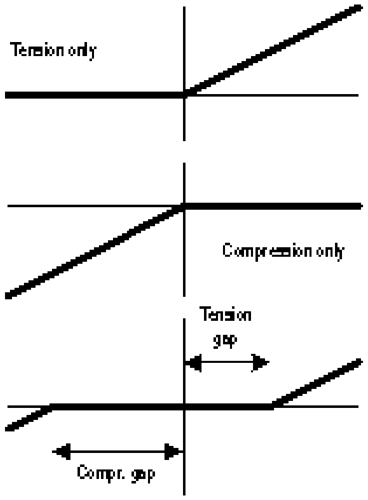


Figure 4.4-11 Definition of non-linear beam properties

A model containing beams with gaps may lead to "Domino" effects involving several beams depending on the loads. The solution will therefore be based on iterations.

To specify non-linear properties click the open button at the right hand side of the Non Linearities field.

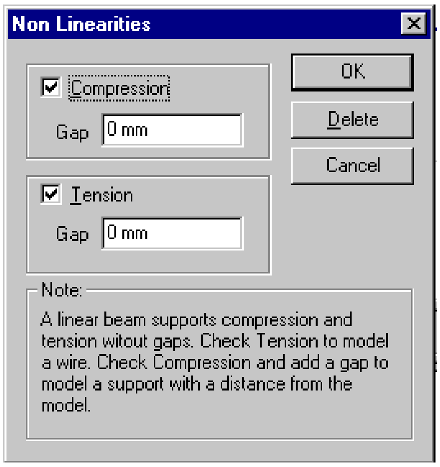


Figure 4.4-12 Specification of non-linear beam properties

In order to distinguish between non-linear beams and normal beams, a separate colour may be specified in Tools | Options… | Colours.

Profile

By expanding the profile node in the *Input property window* you may view the profile properties of the selected beam.

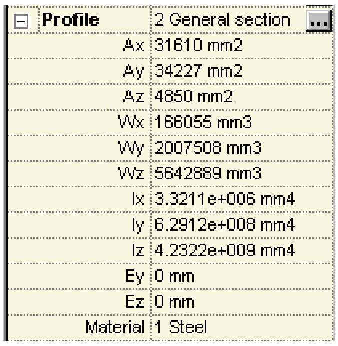


Figure 4.4-13 Profile properties of selected beam

By holding the cursor above the Profile-type field in the input property grid you will see the profile tool tip which shows the profile scantlings defined by the user.

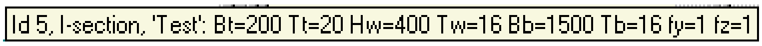


Figure 4.4-14 Profile tool tip

To define a new profile, or edit an existing profile, open the profile dialog. To enter the profile dialog box click the open button ... at the right hand side of the profile heading.

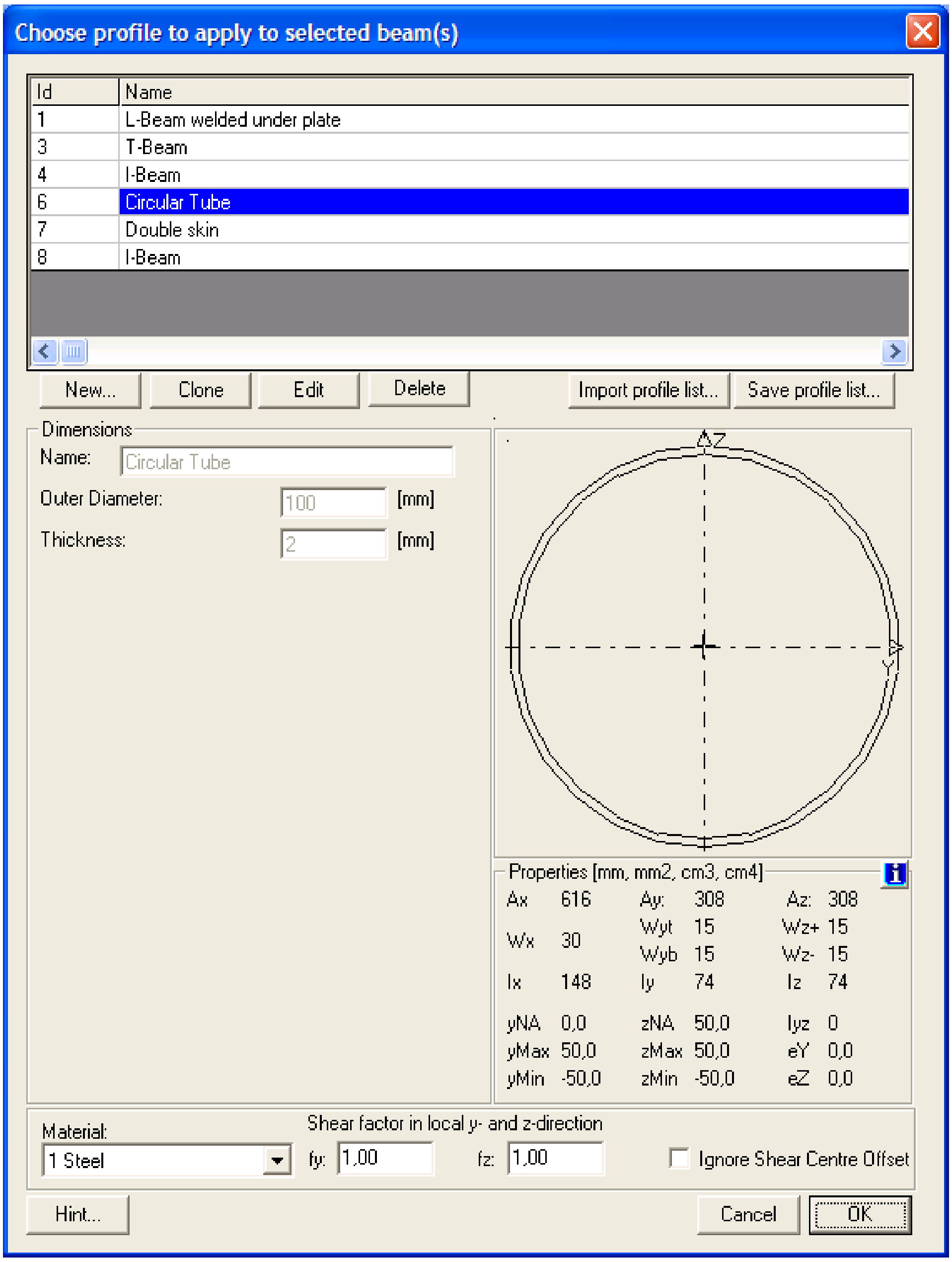
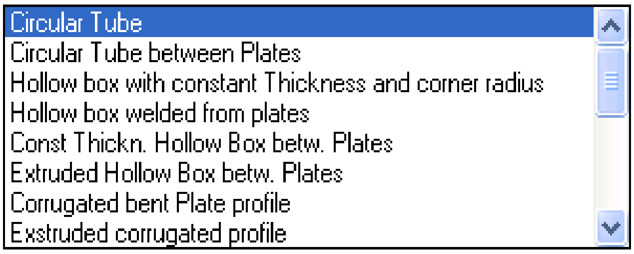


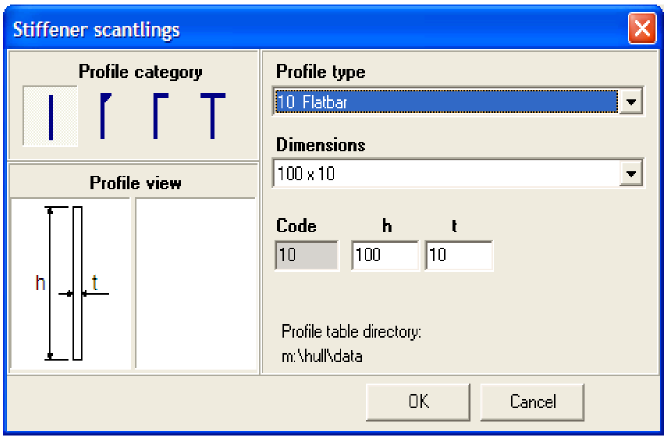
Figure 4.4-15 Input property window - Profile properties

Available profile types

When clicking the *New profile* button you may either select among the predefined profile types:



or chose from tables of standard profiles:



Note The shear factors fy and fz are used for the shear stiffness of the beam only, and not for calculation of shear stress.

Material:

Select in the material drop down list to connect a material to the profile. The default material is steel. Other materials than steel or aluminium should be defined in the material library, see 4.4.5 "Material library".

Ignore shear centre offset:

You may ignore the shear centre offset of the profile by ticking this option. The shear centre of the profile is then placed in the centre of gravity of the profile and no "secondary" shear- and bending stress components is introduced due to rotation of the profile. This may be relevant if one of the flanges of the beam is a part of a continuous plate field. See 6.3 "Shear area"

Distributed load

px1: Distributed load along local x-axis at end 1 of selected beam(s)

py1: Distributed load along local y-axis at end 1 of selected beam(s)

pz1: Distributed load along local z-axis at end 1 of selected beam(s)

px2: Distributed load along local x-axis at end 2 of selected beam(s)

py2: Distributed load along local y-axis at end 2 of selected beam(s)

pz2: Distributed load along local z-axis at end 2 of selected beam(s)

Temperature load

Gy: Temperature gradient along local y-axis

Gz: Temperature gradient along local z-axis

Temperature: Mean relative temperature of beam. A non-zero value, in centigrades, indicates a temperature load.

4.4.4 Selection properties

If one or more beams are selected, the following read-only information is available:

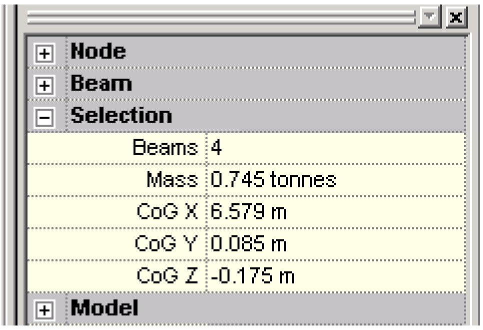


Figure 4.4-16 Input property window – Properties for the current selection

Selection

Beams: The number of beams in the current selection

Mass: Total mass for the selected beams.

CoG X, Y, Z: The centre of gravity for the selection in the global coordinate system.

Note that beams without specified profiles will have mass=0 and CoG=0.

4.4.5 Model properties

Following model properties are available:

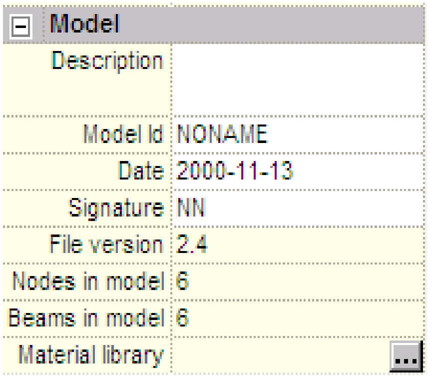


Figure 4.4-17 Input property window - Model properties

Model

Description: Give a description of the model

Mode Id: The model id, of maximum 6 characters, is used as the identification of the PILOT NV5040 input- and result files if these are set to be created during the analysis. The file name conventions for NV5040 are I5040nnn.nnn, and R5040nnn.nnn, where "nnn.nnn" is the *Name*. The file I5040nnn.nnn may serve as backup and may be retrieved both by PILOT NV5040 and 3D Beam. See 4.2.7 "Options / Directories" for more details.

Date: The default date is the date when the model was created.

Signature: Your Id, signature or Name.

Nodes in model: Total number of nodes in the model

Beams in model: Total number of beams in the model

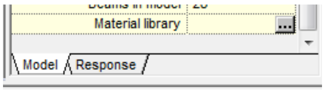
File version: The version of the file format used by 3D Beam at the time when the file was last saved.

Material library

In the material library you may enter new materials or edit existing materials. New materials are stored in the model file. Steel and aluminium are defined as default materials in 3D Beam.

Material is connected to the relevant profile in the beam profile dialog box. See 4.4.3 "Profile".

To enter the material library dialog box click the open button at the right hand side of the material library field.



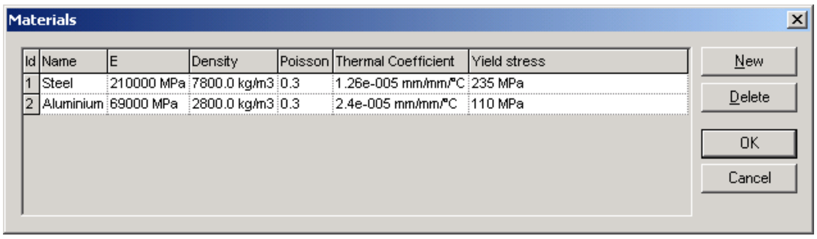


Figure 4.4-18 Input property window - Material properties

Edit any of the material properties by entering directly into the relevant fields in the dialog box.

The density property is used for calculating the mass of selected beams, whereas the value for yield stress is available for information only and is currently not used by the program.

To create a new material, click the *New* button.

4.5 Response property window

4.5.1 Response property window overview

This chapter summarises the functions and features of the *Response property window*. The response properties are located in the second tab card in the same window as the input properties.

In the *Response property window* you will find the response values of single nodes and single beams.

The contents of the fields reflect the response properties of your node/beam selection. If the asterisk (*) is shown in a field no unique value is available. The fields are activated when a node/beam selection is made.

4.5.2 Response properties - single node

Following node response properties are available:

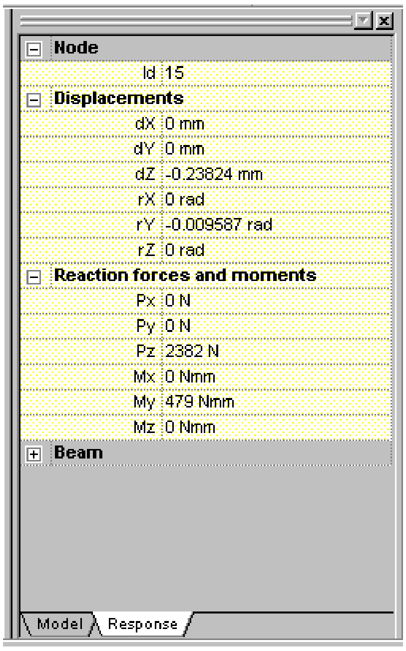


Figure 4.5-1 Response property window - Node properties

Node

Id: Echo of the Node Id (node number) of selected node

Displacements

dX: Absolute translation in global X-direction

dY: Absolute translation in global Y-direction

dZ: Absolute translation in global Z-direction

rX: Absolute rotation about global X-direction

rY: Absolute rotation about global Y-direction

rZ: Absolute rotation about global Z-direction

Positive translation along positive global axes and positive rotation is clockwise about positive global axes.

Reaction forces and moments

Px: Reaction force in global X-direction

Py: Reaction force in global Y-direction

Pz: Reaction force in global Z-direction

Mx: Reaction moment about global X-direction

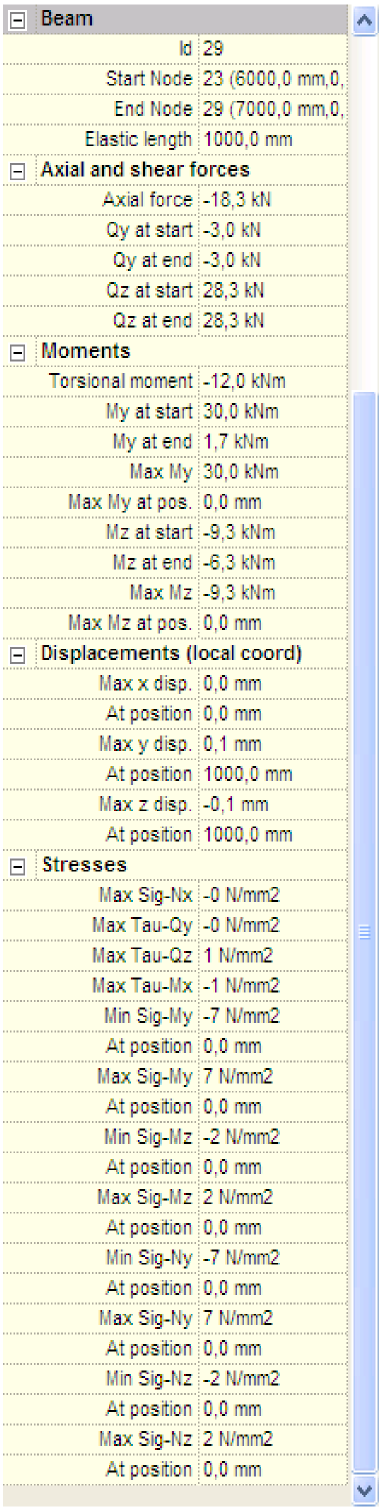
My: Reaction moment about global X-direction

Mz: Reaction moment about global Y-direction

Positive force along positive global axes and positive moment is clockwise about positive global axes.

4.5.3 Response properties - single beam

Following beam response properties are available:



Beam	
Id	29
Start Node	23 (6000,0 mm,0,
End Node	29 (7000,0 mm,0,
Elastic length	1000,0 mm
Axial and shear forces	
Axial force	-18,3 kN
Qy at start	-3,0 kN
Qy at end	-3,0 kN
Qz at start	28,3 kN
Qz at end	28,3 kN
Moments	
Torsional moment	-12,0 kNm
My at start	30,0 kNm
My at end	1,7 kNm
Max My	30,0 kNm
Max My at pos.	0,0 mm
Mz at start	-9,3 kNm
Mz at end	-6,3 kNm
Max Mz	-9,3 kNm
Max Mz at pos.	0,0 mm
Displacements (local coord)	
Max x disp.	0,0 mm
At position	0,0 mm
Max y disp.	0,1 mm
At position	1000,0 mm
Max z disp.	-0,1 mm
At position	1000,0 mm
Stresses	
Max Sig-Nx	-0 N/mm2
Max Tau-Qy	-0 N/mm2
Max Tau-Qz	1 N/mm2
Max Tau-Mx	-1 N/mm2
Min Sig-My	-7 N/mm2
At position	0,0 mm
Max Sig-My	7 N/mm2
At position	0,0 mm
Min Sig-Mz	-2 N/mm2
At position	0,0 mm
Max Sig-Mz	2 N/mm2
At position	0,0 mm
Min Sig-Ny	-7 N/mm2
At position	0,0 mm
Max Sig-Ny	7 N/mm2
At position	0,0 mm
Min Sig-Nz	-2 N/mm2
At position	0,0 mm
Max Sig-Nz	2 N/mm2
At position	0,0 mm

Figure 4.5-2 Response property window - Beam properties

Beam particulars

Id:	Beam Id (beam number) of selected beam
Start Node:	The start node Id and coordinate
End Node:	The end node Id and coordinate
Dist. betw. Nodes:	Distance between the Start Node and the End Node

Axial and shear forces

Axial force:	Axial force in selected beam along local x-axis
Qy at start:	Shear force at start of selected beam in local y-direction
Qy at end:	Shear force at end of selected beam in local y-direction
Qz at start:	Shear force at start of selected beam in local z-direction
Qz at end:	Shear force at end of selected beam in local z-direction

Moments

Torsional moment:	Torsional moment in selected beam about local x-axis
My at start:	Bending moment at start of selected beam about local y-axis
My at end:	Bending moment at end of selected beam about local y-axis
Max My:	Max. bending moment in selected beam about local y-axis
Max My at pos.:	Position of max My from beam start along local x-axis
Mz at start:	Bending moment at start of selected beam about local z-axis
Mz at end:	Bending moment at end of selected beam about local z-axis
Max Mz:	Max. bending moment in selected beam about local z-axis
Max Mz at pos.:	Position of max Mz from beam start along local x-axis

Displacements (local coordinate system)

Max x disp.:	Max. absolute translation of selected beam along local x-axis
Max y disp.:	Max. absolute translation of selected beam along local y-axis
Max z disp.:	Max. absolute translation of selected beam along local z-axis
At position.:	Position of max displacement from beam start along local x-axis

Stresses

Max Sig-Nx:	Max. axial stress (Nx/Ax) in selected beam along local x-axis
Max Tau-Qy:	Max. shear stress (Qy/Ay) in selected beam in local y-direction
Max Tau-Qz:	Max. shear stress (Qz/Az) in selected beam in local z-direction
Max Tau-Mx:	Max. torsional stress (Mx/Wx) in selected beam about local x-axis
Min/Max Sig-My:	Min./max. bending stress (My/Wy) in selected beam about local y-axis
Min/Max Sig-My:	Min./max. bending stress (Mz/Wz) in selected beam about local y-axis

Min/Max Sig-Ny:	Min./max. normal stress (Sig-Nx + Sig-My) in selected beam along local x-axis based on bending about local y-axis
Min/Max Sig-Nz:	Min./max. normal stress (Sig-Nx + Sig-Mz) in selected beam along local x-axis based on bending about local z-axis
At position.:	Position of the stress from beam start along local x-axis

4.6 Output window

4.6.1 Output window overview

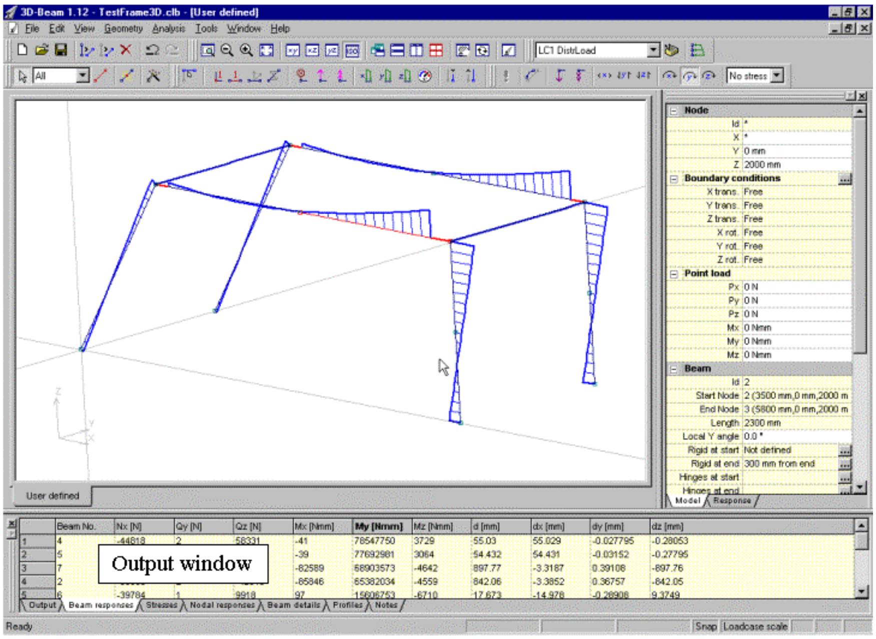


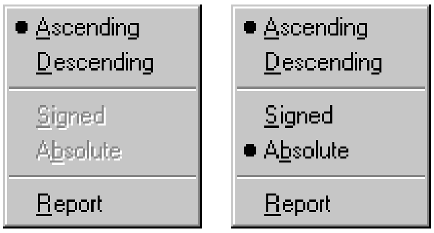
Figure 4.6-1 Output window

The *Output window* consists of several views, which may be activated by clicking the tab cards at the bottom of the window. Information and results from the analysis are made available in the *Output window* when the analysis is completed.

Output Table features

Most of the information in the *Output window* views are presented in tables. Please note the following features and options for the tables.

- The table may be sorted on any column by double-clicking the column heading.
- The table contains only selected nodes and beams if the Apply to selection only option is active.
- You may select nodes or beams in a table by clicking the left mouse button in the row column. More than one row may be selected according to standard Windows procedures, i.e. press Shift to select a range, and Ctrl to select individual rows. The items you select are highlighted in the *Model window*.
    - A right-click on a table displays the following pop-up menus for tables with input and results respectively:



• You may sort the values in ascending or descending order. Right-Click on the desired column heading and select Ascending or Descending from the pop-up menu.

• The values in the tables with results are by default presented as absolute values. To view the signed values you select the Signed option in the pop-up menu.

• Select the Report option to generate a report of the contents in the table.

Output messages

The *Output* view shows messages related to the modelling and analysis. You may scroll the window (up / down) to see all messages.

During modelling, messages about failed operations will be displayed. A beam copied on to an existing beam is an example of such an operation.

During analysis, messages such as shown in the figure below are displayed. By double clicking on error messages related to a node or beam the relevant node/beam is highlighted in the *Model window*.

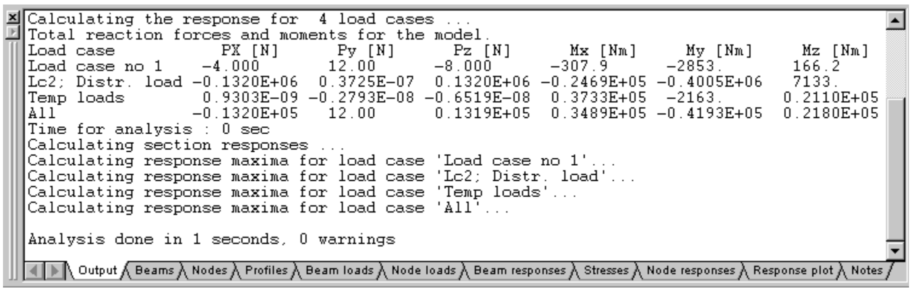


Figure 4.6-2 Output window - Analysis messages

Information provided by 3D Beam in the Output view during the analysis:

• General analysis status messages.

• Number of nodes and beams.

• Directory and filename of exported NV5040 files, if selected.

• Number of load cases calculated.

• Total reaction forces and moments for the model for each load case.

• Relevant error messages and warnings

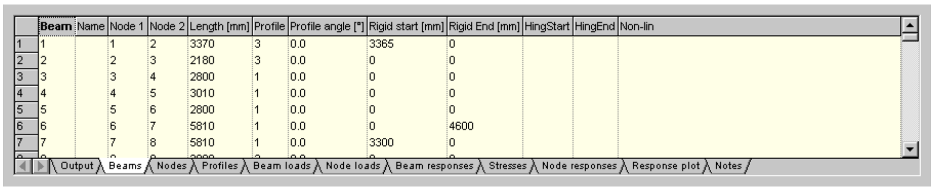
• Analysis time

• Number of error messages and warnings

4.6.2 Structure and loads tables

Beams

The *Beams* view shows the beams in the model.



	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	3370	3	0.0	3365	0			
2	2		2	3	2180	3	0.0	0	0			
3	3		3	4	2800	1	0.0	0	0			
4	4		4	5	3010	1	0.0	0	0			
5	5		5	6	2800	1	0.0	0	0			
6	6		6	7	5810	1	0.0	0	4600			
7	7		7	8	5810	1	0.0	3300	0			

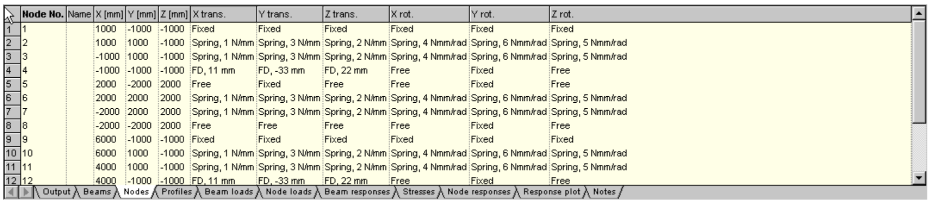
Figure 4.6-3 Output window - Beams in the model

Listed beam properties:

Beam:	Beam identification number
Beam Name:	User's beam identification
Start/End Node:	Node numbers for the start and end nodes respectively
Length:	Elastic length of beam, excluding possible rigid ends
Mass:	The mass of the elastic length of the beam
Profile:	Profile identification number
Angle:	Angle between the profile's z-axis and the plane through the beam and the global Z-axis. Positive for clockwise rotation when seen in positive local x-direction.
Rigid Start/End:	Length of possible rigid part of the beam at the start and end ends respectively
Hinged at Start/End:	Possibly defined hinge at the start and end nodes respectively, where hinges are defined as:
dX, dY, dZ:	Hinged with respect to translation in the global X-, Y-, and Z-direction respectively
rX, rY, rZ:	Hinged with respect to rotation about the global X-, Y-, and Z-axis respectively
Non Linearities:	Possibly specified non-linear properties for the beam. For definition, see 4.4.3 Non Linearities.

Nodes

The *Nodes* view shows the nodes in the model.



	Node No.	Name	X [mm]	Y [mm]	Z [mm]	X trans.	Y trans.	Z trans.	X rot.	Y rot.	Z rot.
1	1		1000	-1000	-1000	Fixed	Fixed	Fixed	Fixed	Fixed	Fixed
2	2		1000	1000	-1000	Spring, 1 N/mm	Spring, 3 N/mm	Spring, 2 N/mm	Spring, 4 Nmm/rad	Spring, 6 Nmm/rad	Spring, 5 Nmm/rad
3	3		-1000	1000	-1000	Spring, 1 N/mm	Spring, 3 N/mm	Spring, 2 N/mm	Spring, 4 Nmm/rad	Spring, 6 Nmm/rad	Spring, 5 Nmm/rad
4	4		-1000	-1000	-1000	FD, 11 mm	FD, -33 mm	FD, 22 mm	Free	Fixed	Free
5	5		2000	-2000	2000	Free	Fixed	Free	Free	Fixed	Free
6	6		2000	2000	2000	Spring, 1 N/mm	Spring, 3 N/mm	Spring, 2 N/mm	Spring, 4 Nmm/rad	Spring, 6 Nmm/rad	Spring, 5 Nmm/rad
7	7		-2000	2000	2000	Spring, 1 N/mm	Spring, 3 N/mm	Spring, 2 N/mm	Spring, 4 Nmm/rad	Spring, 6 Nmm/rad	Spring, 5 Nmm/rad
8	8		-2000	-2000	2000	Free	Free	Free	Free	Fixed	Free
9	9		6000	-1000	-1000	Fixed	Fixed	Fixed	Fixed	Fixed	Fixed
10	10		6000	1000	-1000	Spring, 1 N/mm	Spring, 3 N/mm	Spring, 2 N/mm	Spring, 4 Nmm/rad	Spring, 6 Nmm/rad	Spring, 5 Nmm/rad
11	11		4000	1000	-1000	Spring, 1 N/mm	Spring, 3 N/mm	Spring, 2 N/mm	Spring, 4 Nmm/rad	Spring, 6 Nmm/rad	Spring, 5 Nmm/rad
12	12		4000	-1000	-1000	FD, 11 mm	FD, -33 mm	FD, 22 mm	Free	Fixed	Free

Figure 4.6-4 Output window - Nodes in the model

Listed node properties:

Node No:	Node identification number
Name:	User's node identification
X, Y, Z:	Node coordinates in the global coordinate system
X trans, Y trans, Z trans:	Boundary conditions w.r.t. translation along the global axes

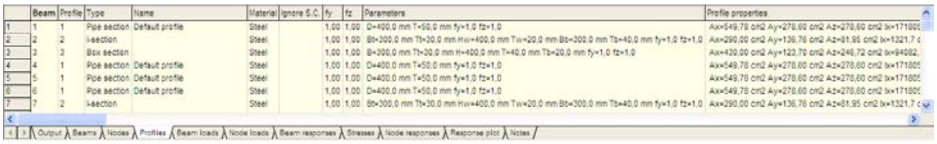
X rot, Y rot, Z rot:	Boundary conditions w.r.t. rotation about the global axes

Where:

Free:	The node is free
Fixed:	The node is fixed
FD:	The node has a prescribed displacement or rotation
Spring:	The node is supported by a spring

Profiles

The *Profiles* view shows the profiles in the model.



	Beam	Profile	Type	Name	Material	Ignore S.C.	fy	fz	Parameters	Profile properties
1	1	1	Pipe section	Default profile	Steel		1,00	1,00	D=400,0 mm T=50,0 mm fy=1,0 fz=1,0	Ax=549,78 cm2 Ay=278,60 cm2 Az=278,60 cm2 Ix=171805
2	2	2	I-section		Steel		1,00	1,00	Bt=300,0 mm Tt=30,0 mm Hw=400,0 mm Tw=20,0 mm Bb=300,0 mm Tb=40,0 mm fy=1,0 fz=1,0	Ax=290,00 cm2 Ay=136,78 cm2 Az=81,95 cm2 Ix=1321,7 c
3	3	3	Box section		Steel		1,00	1,00	B=300,0 mm Tt=30,0 mm H=400,0 mm T=40,0 mm Tb=20,0 mm fy=1,0 fz=1,0	Ax=420,00 cm2 Ay=123,70 cm2 Az=246,72 cm2 Ix=94082,
4	4	1	Pipe section	Default profile	Steel		1,00	1,00	D=400,0 mm T=50,0 mm fy=1,0 fz=1,0	Ax=549,78 cm2 Ay=278,60 cm2 Az=278,60 cm2 Ix=171805
5	5	1	Pipe section	Default profile	Steel		1,00	1,00	D=400,0 mm T=50,0 mm fy=1,0 fz=1,0	Ax=549,78 cm2 Ay=278,60 cm2 Az=278,60 cm2 Ix=171805
6	6	1	Pipe section	Default profile	Steel		1,00	1,00	D=400,0 mm T=50,0 mm fy=1,0 fz=1,0	Ax=549,78 cm2 Ay=278,60 cm2 Az=278,60 cm2 Ix=171805
7	7	2	I-section		Steel		1,00	1,00	Bt=300,0 mm Tt=30,0 mm Hw=400,0 mm Tw=20,0 mm Bb=300,0 mm Tb=40,0 mm fy=1,0 fz=1,0	Ax=290,00 cm2 Ay=136,78 cm2 Az=81,95 cm2 Ix=1321,7 c

Figure 4.6-5 Output window - Profiles in the model

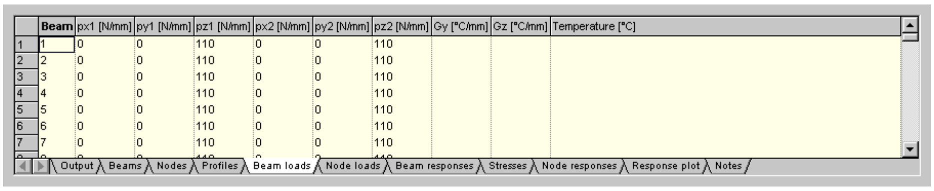
Listed profile information:

Beam:	Beam identification number
Profile:	Profile identification number
Type:	Profile type
Name:	User's profile identification
Material:	Material name
Ignore S.C.:	Displays a X if the Shear Centre should be ignored
fy/fz:	Shear factor in the local y and z-direction
Parameters:	Input parameters defining the profile

Profile properties: The main properties of the profile

Beam Loads

The *Beam loads* view shows the beams on which loads have been applied.



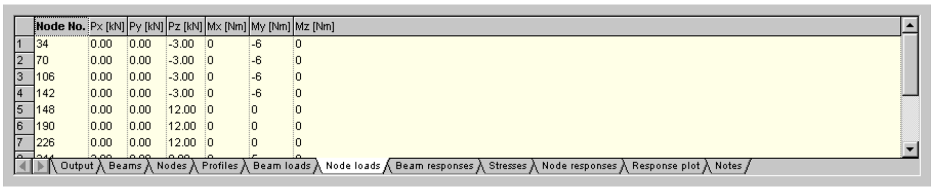
	Beam	px1 [N/mm]	py1 [N/mm]	pz1 [N/mm]	px2 [N/mm]	py2 [N/mm]	pz2 [N/mm]	Gy [°C/mm]	Gz [°C/mm]	Temperature [°C]
1	1	0	0	110	0	0	110			
2	2	0	0	110	0	0	110			
3	3	0	0	110	0	0	110			
4	4	0	0	110	0	0	110			
5	5	0	0	110	0	0	110			
6	6	0	0	110	0	0	110			
7	7	0	0	110	0	0	110			

Figure 4.6-6 Output window - Beam loads

For definitions, see 4.4.3 Distributed load and Temperature load.

Node Loads

The *Node loads* view shows the nodes subject to loads.



	Node No.	Px [kN]	Py [kN]	Pz [kN]	Mx [Nm]	My [Nm]	Mz [Nm]
1	34	0.00	0.00	-3.00	0	-6	0
2	70	0.00	0.00	-3.00	0	-6	0
3	106	0.00	0.00	-3.00	0	-6	0
4	142	0.00	0.00	-3.00	0	-6	0
5	148	0.00	0.00	12.00	0	0	0
6	190	0.00	0.00	12.00	0	0	0
7	226	0.00	0.00	12.00	0	0	0

Figure 4.6-7 Output window - Beam responses

For definitions, see 4.4.2 Point load.

Notes

In the *Notes* view you may write notes, comments, assumptions, conclusions, summary etc. about the analysis. This may be useful for documentation purposes or just as your personal scratch pad.

The notes are saved with the model.

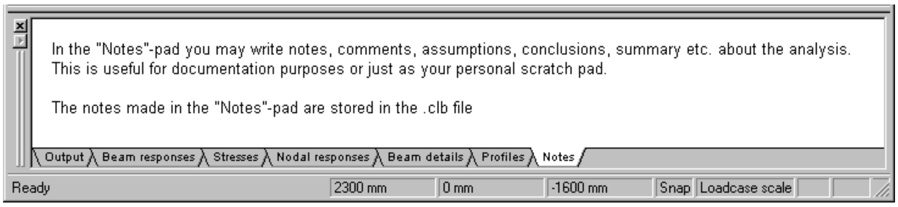
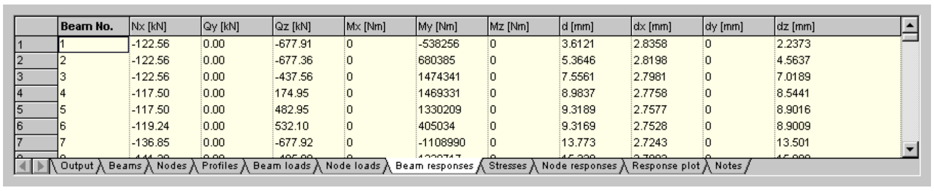


Figure 4.6-8 Output window - Notes

4.6.3 Result tables

Beam responses

The *Beam responses* view shows the maximum values (negative or positive) of forces, moments and deflections along the beams.



	Beam No.	Nx [kN]	Qy [kN]	Qz [kN]	Mx [Nm]	My [Nm]	Mz [Nm]	d [mm]	dx [mm]	dy [mm]	dz [mm]
1	1	-122.56	0.00	-677.91	0	-538256	0	3.6121	2.8358	0	2.2373
2	2	-122.56	0.00	-677.36	0	680385	0	5.3646	2.8198	0	4.5637
3	3	-122.56	0.00	-437.56	0	1474341	0	7.5561	2.7981	0	7.0189
4	4	-117.50	0.00	174.95	0	1469331	0	8.9837	2.7758	0	8.5441
5	5	-117.50	0.00	482.95	0	1330209	0	9.3189	2.7577	0	8.9016
6	6	-119.24	0.00	532.10	0	405034	0	9.3169	2.7528	0	8.9009
7	7	-136.85	0.00	-677.92	0	-1108990	0	13.773	2.7243	0	13.501

Figure 4.6-9 Output window - Beam responses

Following beam responses are provided:

Nx: Max. axial force in the beam

Qy: Max. shear force in local y-direction

Qz: Max. shear force in local z-direction

Mx: Max. torsional moment in the beam

My: Max. bending moment about local y-direction

Mz: Max. bending moment about local z-direction

d: Max. total deflection of the beam ($\sqrt{(\delta_x^2 + \delta_y^2 + \delta_z^2}$)

dx: Max. deflection of the beam in global X-direction

dy: Max. deflection of the beam in global Y-direction dz: Max. deflection of the beam in global Z-direction

Node responses

The *Node responses* view shows the maximum values (negative or positive) of deflections, rotations, forces and bending moments of the nodes.

	Node No.	|dX| [mm]	|dY| [mm]	|dZ| [mm]	|rX| [deg]	|rY| [deg]	|rZ| [deg]	|Px| [kN]	**|Py| [kN]**	|Pz| [kN]	|Mx| [Nm]	|My| [Nm]
1	115	2.8105	0	8.2612	0.000824	0.02057	0.0006853	0.00	27015.35	0.00	0	0
2	123	2.2973	0	8.2612	0.0007944	0.02056	0.0007669	0.00	26600.11	0.00	0	0
3	193	3.125	0	5.5764	0.0007172	0.04379	0.001562	0.00	23799.32	0.00	0	0
4	201	2.0352	0	5.5766	0.0006716	0.04374	0.001421	0.00	22796.92	0.00	0	0
5	79	2.7691	0	9.4486	0.0005906	0.01967	0.0003605	0.00	19327.19	0.00	0	0
6	87	2.2782	0	9.4486	0.0005698	0.01967	0.0003342	0.00	19074.29	0.00	0	0

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

Figure 4.6-10 Output window - Node responses

The following node responses are provided:

dX, dY, dZ: Translation in global X-, Y-, and Z- direction

rX, rY, rZ: Rotation about global X-, Y-, and Z- axis (positive for right-handed screw)

Px, Py, Pz: Reaction force in global X-, Y-, and Z- direction

Mx, My, Mz: Reaction moment about global X-, Y-, and Z- axis (positive for right-handed screw)

Stresses

The *Stresses* view shows the maximum values (negative or positive) of axial stresses, shear stresses, bending stresses and normal stresses along the beams.

	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	1	-0	-1	-2	7	4	-6	8	-4	5
2	2	0	0	0	0	0	0	0	0	0	0
3	3	0	0	0	0	0	0	0	0	0	0
4	4	0	0	0	0	0	0	0	0	0	0
5	5	-0	-0	1	-1	7	2	-7	7	-3	2
6	6	-1	-0	1	1	7	4	-8	6	-5	3
7	7	1	-0	-1	-0	-1	2	-0	1	-1	2
8	8	0	-0	-1	1	7	4	-7	7	-4	5

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

Figure 4.6-11 Output window - Stresses

Following stress components are calculated:

Principal stresses:

Sig-Nx: Axial stress (N_x/A_x)

Tau-Qy: Max shear stress in local y-direction (Q_y/A_y), in the neutral axis.

Tau-Qz: Max shear stress in local z-direction (Q_z/A_z), in the neutral axis.

Tau-Mx: Torsional stress (M_x/W_x)

Sig-My: Maximum bending stress about local y-axis (M_y/W_y)

Sig-Mz: Maximum bending stress about local z-axis (M_z/W_z)

Stress combinations:

Min/max Sig-Ny: Min/max of the normal stress in local xz-plane, max of (Sig-Nx ± Sig-My)

Min/max Sig-Nz: Min/max of the normal stress in local xy-plane, max of (Sig-Nx ± Sig-Mz)

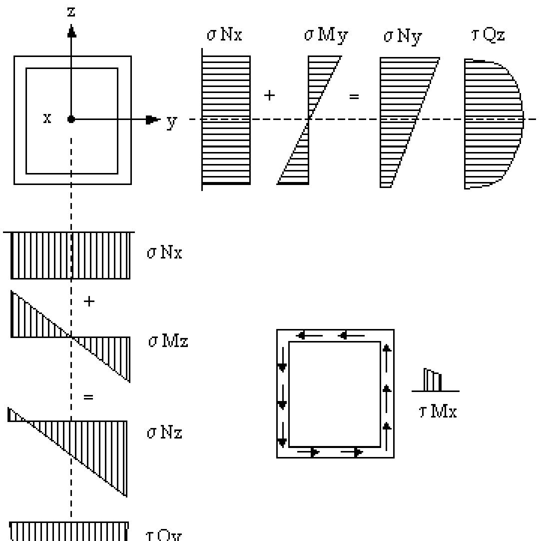
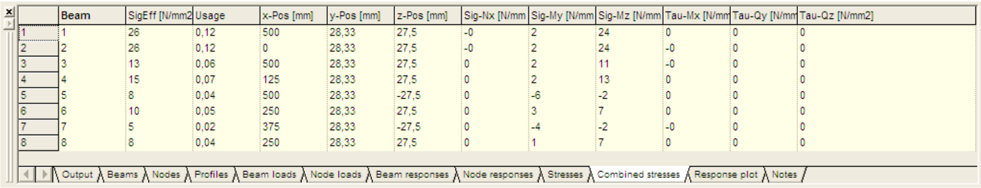


FIG. PRINCIPAL STRESSES

Combined stresses

The *Combined stresses* view shows the combined stresses according to von Mises for some types of profiles. The following profiles are currently handled:

pipe-, box-, I-, channel-, and L-sections.



	Beam	SigEff [N/mm2	Usage	x-Pos [mm]	y-Pos [mm]	z-Pos [mm]	Sig-Nx [N/mm	Sig-My [N/mm	Sig-Mz [N/mm	Tau-Mx [N/mm	Tau-Qy [N/mm	Tau-Qz [N/mm2]
1	1	26	0,12	500	28,33	27,5	-0	2	24	0	0	0
2	2	26	0,12	0	28,33	27,5	-0	2	24	-0	0	0
3	3	13	0,06	500	28,33	27,5	0	2	11	-0	0	0
4	4	15	0,07	125	28,33	27,5	0	2	13	0	0	0
5	5	8	0,04	500	28,33	-27,5	0	-6	-2	0	0	0
6	6	10	0,05	250	28,33	27,5	0	3	7	0	0	0
7	7	5	0,02	375	28,33	-27,5	0	-4	-2	-0	0	0
8	8	8	0,04	250	28,33	27,5	0	1	7	0	0	0

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

Figure 4.6-12 Output window - Combined stresses

The following results are provided:

SigEff: Effective stress according to von Mises,

$$\sigma_{eff} = \sqrt{(\sigma_{Nx} + \sigma_{My} + \sigma_{Mz})^2 + 3(|\tau_{Mx}| + |\tau_{Qy} + \tau_{Qz}|)^2}$$

Usage: Usage factor = $\sigma_{eff} / (\sigma_{yield} / \gamma_M)$

where: σ_{yield} = specified yield stress

γ_M = material factor = 1.0 unless otherwise specified

Position of stress point where σ_{eff} is computed:

x-pos: Distance from start of beam

y-pos: y-coordinate on profile

z-pos: z-coordinate on profile

Stresses at the stress point:

σNx:	Axial stress
σMy:	Bending stress about local y-axis
σMz:	Bending stress about local z-axis
τMx:	Torsional stress
τQy:	Shear stress in local y-direction
τQz:	Shear stress in local z-direction

Response plot

In the *Response plot* view you can display any response type at any position along the selected beam(s). Select the response type in the drop down list in the upper left corner of the view.

You may view the diagrams of all load cases simultaneously by checking the All load cases check box (below the Response type drop down).

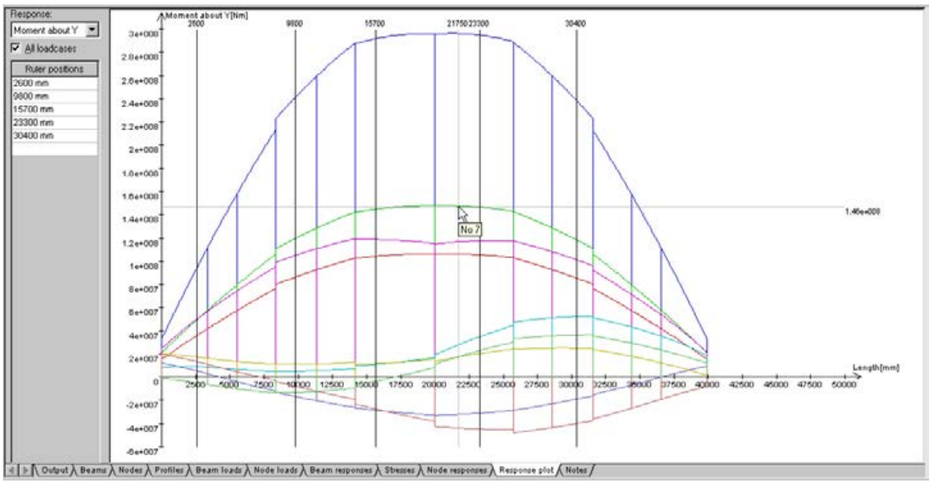


Figure 4.6-13 Output window - Response plot

You may create a report with all cross section response details at desired positions. Enter the positions (from the start node of the beam(s) = left side of diagram) in the Ruler positions list, or mark the position by clicking the left mouse button in the diagram.

To clear the defined ruler positions select Clear rulers from the pop-up menu (click the right mouse button in the diagram).

By holding the pointer (steady) above a response line in the diagram a label showing the load case name (= active load case) and the flange id (top- or bottom flange) will appear.

At the left end of the horizontal ruler line you will see the response value at the position considered.

At the upper end of the vertical ruler line you will see the x-value from the start node of the first beam.

Please note: When more than one beam is selected the order of the beams in the selection and the direction of the beams are imported. The responses are plotted from the start to end of each beam in the selection. Hence, avoid rubber-band selections as the order of the beams in the selection is unknown. Instead, while pressing Shift, select one beam at a time in the order you wish the plot to appear.

In order to produce a correct plot it may be necessary to change the direction of one or more beams. This may be achieved by means of the Flip Beam feature.

An image of the plot may be copied by selecting the Copy graph option from the pop-up menu (right-click in the diagram). The image may be pasted (Ctrl+V) into e.g. the 3D Beam report in MS Word.

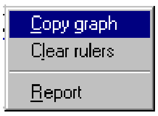


Figure 4.6-14 Pop-up menu in the Response plot view

4.7 Model window

The beam model with selected properties (input and/or results) is displayed in the *Model window*. You may view as many model windows as you like, one by one, or several windows simultaneously. You may define the working plane in each window. The basic working planes are: XY-plane (Z=0), XZ-plane (Y=0) and YZ-plane (X=0). In addition you may display the ISO-view (3D-view) and the solid view.

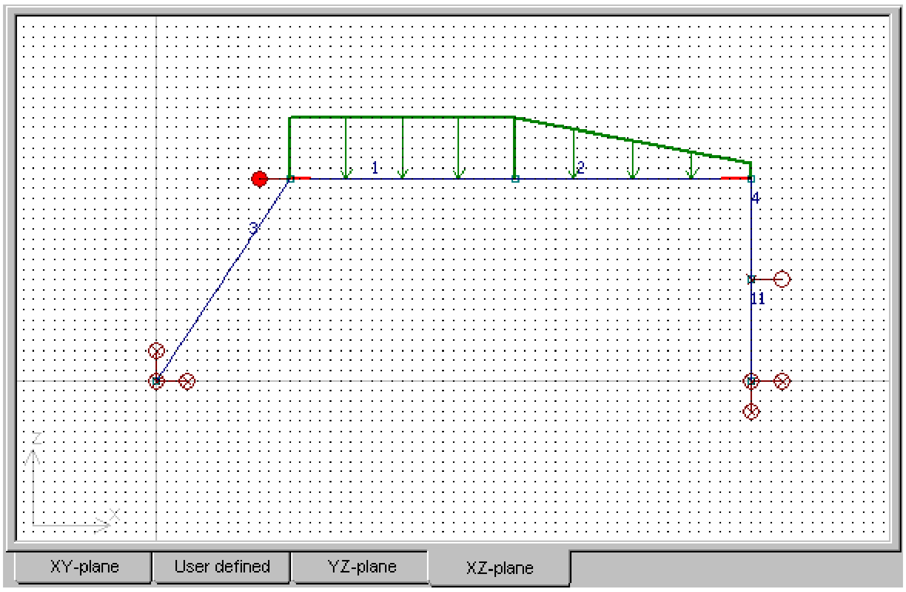


Figure 4.7-1 Model window - XZ plane

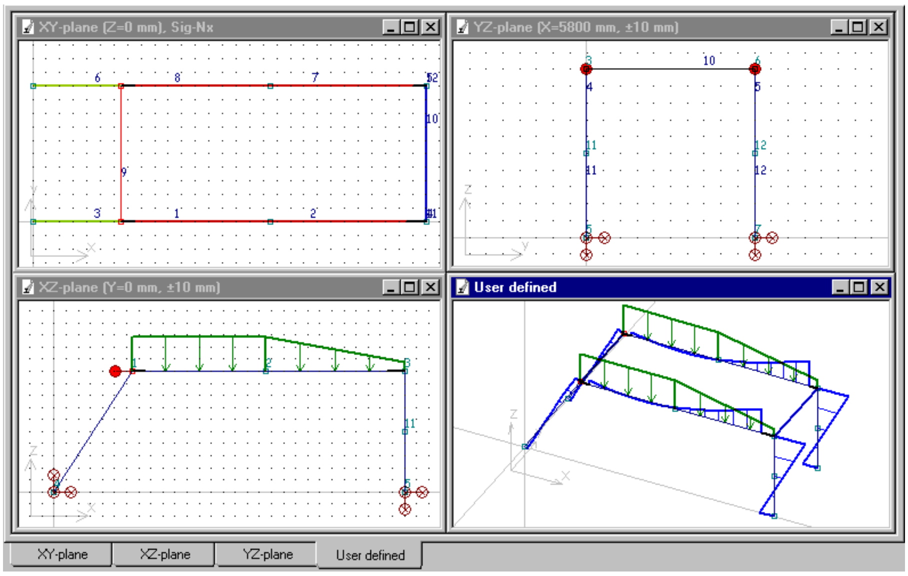


Figure 4.7-2 Model window - Four views

Click the right mouse button in the *Model window*

When you click the right mouse button in the *Model window* a pop-up menu with the most used features appears.

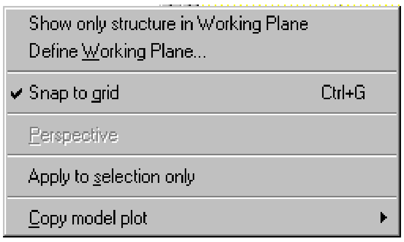


Figure 4.7-3 Pop-up menu in the Model window

The features are described in 4.2 "Menu bar".

You may create a screen dump (an image of the model from the *Model window*) by selecting Copy model plot => To Clipboard Ctrl+C. The image may be pasted (Ctrl+V) into, for example, the 3D Beam report in MS Word.

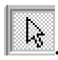
4.8 Shortcut keys

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; E.g. 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
Alt + Up Arrow	Enlarge symbols displayed on the model
Alt + Down Arrow	Shrink symbols displayed on the model

4.9 Mouse operations

4.9.1 Select modes

To make a selection of nodes or beams in 3D Beam you should activated the *Select* tool .

Use the selection filter to specify if only nodes, only beams or both nodes and beams should be included in the selection.

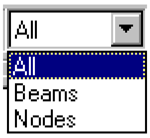


Figure 4.9-1 Selection filter

Select nodes and/or beams enclosed by the rubber band: Drag rubber band while pressing the left mouse button.

Select nodes and/or beams touched by the rubber band: Press Ctrl + drag rubber band while pressing the left mouse button.

To add or remove new nodes or beams to the current selection, press Shift while selecting.

4.9.2 MS IntelliMouse:

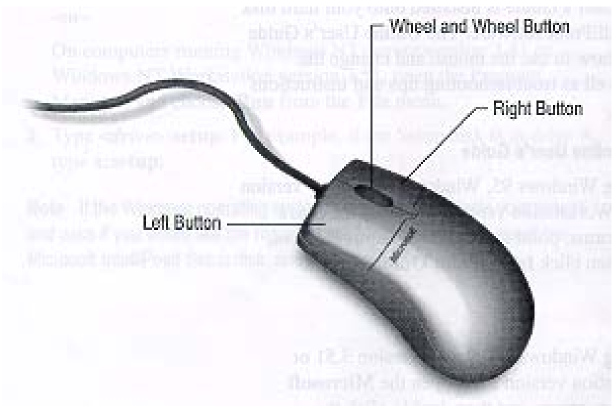


Figure 4.9-2 MS IntelliMouse

By using the MS IntelliMouse following additional 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 property window*:

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

In the *Result property window*:

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

5 3D Beam as part of NAUTICUS Explorer

5.1 Creating and importing 3D Beam models

To create or import a 3D Beam model in Nauticus Explorer, you select the task *Rule Check Analyses*.

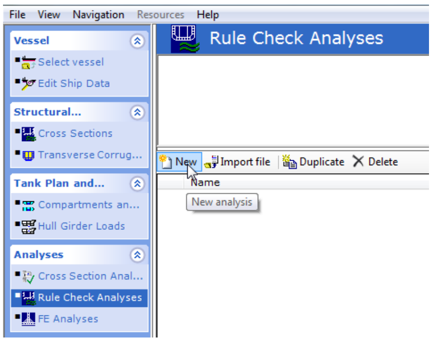


Figure 5.1-1 New / Import 3D Beam model in NAUTICUS Explorer

Press *New* to open the list of available analysis modules in the program, and select 3D Beam from the list. To import an existing 3D Beam model (.clb file) select the *Import file* option. Then double-click to open 3D-Beam.

You may update the Name-field to give a more descriptive name on the model. The location of the database (.clb-file) is shown in the line below. The default location is the subfolder WFDepot beneath the project folder.

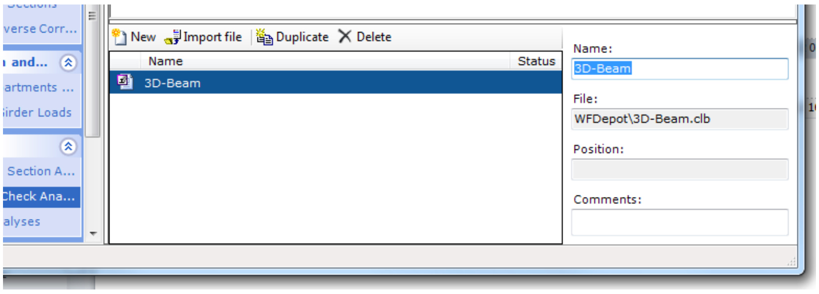


Figure 5.1-2 3D Beam model name and file location

See the NAUTICUS Explorer User Manual for further details related to job templates and operations of NAUTICUS Explorer.

6 Appendix A: Theory

6.1 Calculation method

The program is based on the matrix displacement method and the structural frame is idealised by nodes, beams and supports.

The elastic beams are analysed as "Timoshenko beams" which implies the angle between the beam line and the cross section normal is proportional to the shear force. This is a modification of Bernoulli's theorem saying that plane cross sections remain normal to the beam line.

The Timoshenko beam theory allows the program to handle short tall beams as well as long, slim Bernoulli-Euler beams.

For each beam a stiffness matrix **k** is formed. The matrix gives the relationship between forces, **S,** and displacement, **v**, at the beam ends:

S = k * v

The stiffness contribution from each beam is added into the final stiffness matrix, **K**. The given loads, **R**, shall be in equilibrium with the beam forces, **S**. The relationship between forces and displacements, **r**, for the entire system may be written as

R = K * r

This system of equations is finally solved for the desired displacements corresponding to a set of external forces.

6.2 Non linear model

A model containing beams with an updated E-modulus is analysed by means of an iterative method. In the first step all beams are treated as normal beams. The results are examined and new equivalent E-moduli are applied to the non-linear beams if appropriate, i.e.:

- beams in the "wrong mode", e.g. a wire in compression, are given an E-modulus close to zero,
- for beams with a defined gap, the equivalent E-modulus, E*, is found according to the principle shown in the figure below.

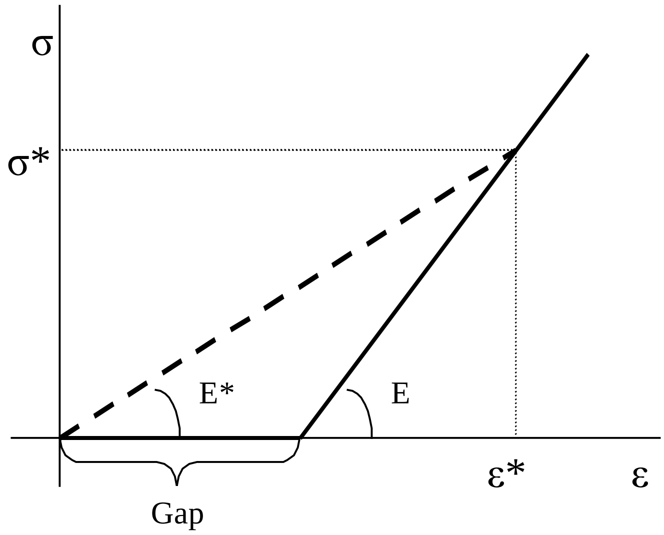


Figure 6.2-1 Finding an equivalent E-modulus, E, for a beam with gap.*

The analysis is repeated with the new E-moduli and the new results examined. Because of possible re-distribution of loads and "domino" effects, new equivalent E-moduli may have to be applied. The iteration criterion is satisfied when the difference between the E* between two iterations is less than 10%.

6.3 Shear area

The shear area in local z-direction is defined as:

$$Az = \frac{Iy \cdot tp}{Sy}$$

where Iy is the moment of inertia about local y-axis, tp is the profile thickness and Sy the 1st area moment about the local y-axis.

The shear area is used to calculate the shear stiffness of the beam and the maximum shear stress over the profile height.

Max shear stress is found in the neutral axis of the profile and is defined as:

$$\tau_{Qz} = \frac{Qz \cdot Sy}{Iy \cdot tp} = \frac{Qz}{Az}$$

where Qz is the shear force.

6.4 Shear centre offset

The shear centre offset of profiles may be included or excluded in the analysis. This is defined in the profile dialog box. Select the Ignore Shear Centre Offset option in the profile dialog box to exclude the shear centre offset.

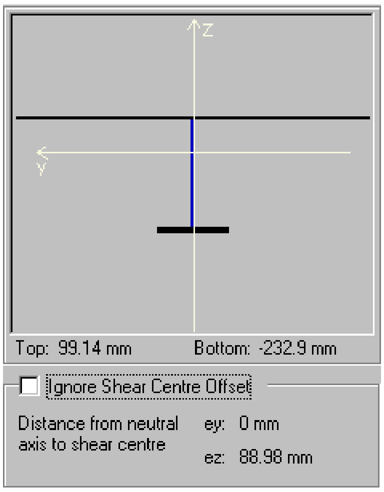


Figure 6.4-1 Shear centre offset

By excluding the shear centre offset, i.e. locating the shear centre (CS) in the centre of gravity (CoG) of the profile cross section, no node rotation or node moment is introduced when applying node loads (forces) or distributed loads. Excluding the shear centre offset is relevant when the profile is part of a continuous plate field. A continuous plate-field will restrict the profile against torsional rotation.

Following applies for a beam with shear centre offset:

Ex 1: If the load is a node load (F), then:

The load is applied to the node which is located at the centre of gravity of the profile.

Node translations, $dz \neq 0$ and $dy \neq 0$,

Node rotation, $rx \neq 0$

Torsional reaction moment, $Mx = 0$

Ex 2: If the load is a distributed load (q) on the beam, then:

The load is applied to the shear centre of the profile.

Node translation, $dz \neq 0$

Node rotation, $rx = 0$

Torsional reaction moment, $Mx \neq 0$

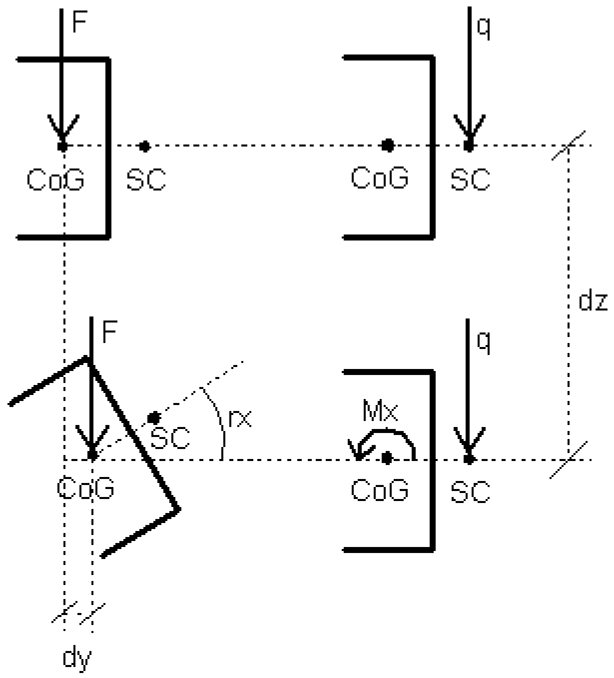


Figure 6.4-2 Ex. 1 Node load Ex. 2 Distributed load

6.5 Profile sectional properties

6.5.1 Notations

The following notations for the section properties are used:

Ax: Axial area (total profile area)

Ay: Shear area in local y-direction

Az: Shear area in local z-direction

Sy: 1st area moment about local y-axis

Sz: 1st area moment about local z-axis

Ix: Torsional moment of inertia

Iy: Moment of inertia about local y-axis

Iz: Moment of inertia about local z-axis

Wx: Torsion section modulus

Wyt: Section modulus about local y-axis at top of profile

Wyb: Section modulus about local y-axis at bottom of profile

Wzt: Section modulus about local z-axis at top of profile

Wzb: Section modulus about local z-axis at bottom of profile

Note: Wzt = Wzb = Wz min for all profile types except I - profiles

ey: Shear centre distance from vertical neutral axis

ez: Shear centre distance from horizontal neutral axis fy: Shear factor in local y-direction

fz: Shear factor in local z-direction

Note: The shear factor is used for shear stiffness of beam, but not for calculation of shear stress

Naz: Vertical height of neutral axis from base line

Note Only profiles which are symmetrical about the local z-axis can be handled.

6.5.2 General profile

The user explicitly specifies all section properties.

6.5.3 I-profile

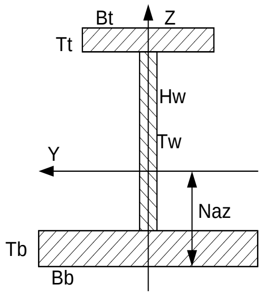


Figure 6.5-1 I-profile parameters

Ax = Bt • Tt+Hw • Tw+Bb • Tb
Ay = Iz • (Tb+Tt)/Sz
Az = Iy • Tw/Sy
Sy = (Bb • Tb • (Naz-Tb/2) + (Naz-Tb)^2 •Tw/2 +Bt • Tt • (H-Tt/2-Naz) + (Hw+Tb-Naz)^2 • Tw/2)/2
Sz = (Tt • Bt^2 + Tb • Bb^2 + Hw • Tw^2)/8
if Tt = Tw and Tw = Tb then
Ix = Tw^3 • (Hw+Bb+Bt-1.2 • Tw) / 3
else
Ix = 1.30/3 • (Bb • Tb^3+Hw • Tw^3+Bt • Tt^3)
Iy = Bb • Tb • (Naz-Tb/2)^2+Hw • Tw • (Hw/2+Tb-Naz)^2+Bt • Tt • (H-t/2-Naz)^2 +(Bt • Tt^3+Tw • Hw^3+Bb • Tb^3)/12
Iz = (Tb • Bb^3+Tt • Bt^3+Hw • Tw^3)/12
Wx = Ix / max(Tt, max(Tw,Tb))
Wzb = Iz / (Bb/2)
Wzt = Iz / (Bt/2)
Wyb = Iy / Naz
Wyt = Iy / (H-Naz)
ey = 0
ez = ((H-Tt/2) • Tt • Bt^3 + Tb^2 • Bb^3/2) / (Tt • Bt^3 +Tb • Bb^3)-Naz
Naz = (Bt • Tt • (H-Tt/2)+Hw • Tw • (Hw/2+Tb)+Bb • Tb^2/2) / Ax
H = Hw+Tb+Tt

6.5.4 Double skin profile

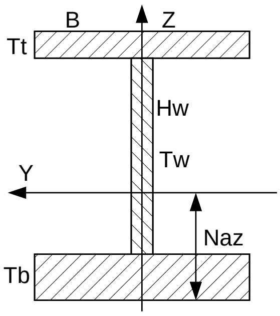


Figure 6.5-2 Double skin profile parameters

Ax = B • Tt+Hw • Tw+B • Tb
Ay = Iz • (Tb+Tt)/Sz
Az = Iy • Tw/Sy
Sy = (B • Tb • (Naz-Tb/2) + (Naz-Tb)^2 •Tw/2 +B • Tt • (H-Tt/2-Naz) + (Hw+Tb-Naz)^2 • Tw/2) / 2
Sz = (Tt • B^2 + Tb • B^2 + Hw • Tw^2)/8
if Tt = Tw and Tw = Tb then
Ix = Tw^3 • (Hw+B+B-1.2 • Tw) / 3
else
Ix = 1.30 / 3 • (B • Tb^3+Hw • Tw^3+B • Tt^3)
Ix = Ix + B • H^2 / (1/Tb+1/Tt)
Iy = B • Tb • (Naz-Tb/2)^2 + Hw • Tw • (Hw/2+Tb-Naz)^2+B • Tt • (H-t/2-Naz)^2 +(B • Tt^3+Tw • Hw^3+B • Tb^3)/12
Iz = (Tb • B^3+Tt • B^3+Hw • Tw^3)/12
Wx = Ix / max(Tt, max(Tw,Tb))
Wzb = Wzt = Iz / (B/2)
Wyb = Iy / Naz
Wyt = Iy / (H-Naz)
ey = 0
ez = ((H-Tt/2) • Tt • B^3 + Tb^2 • B^3/2) / (Tt • B^3 +Tb • B^3)-Naz
Naz = (B • Tt • (H-Tt/2)+Hw • Tw • (Hw/2+Tb)+B • Tb^2/2) / Ax
H = Hw+Tb+Tt

6.5.5 Box profile

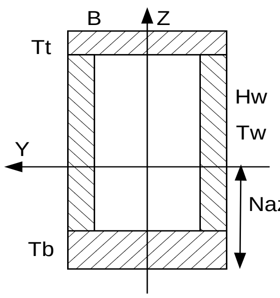


Figure 6.5-3 Box profile parameters

Ax = B • Tt+2 • Hw • Tw+B • Tb
Ay = (Iz/Sz) • (Tb+Tt)
Az = (Iy/Sy) • 2 • Tw
Sy = B • Tb • (Naz-Tb/2) + Tw • (Naz-Tb)^2
Sz = (Tb+Tt) • Bb^2/8 + Hw • Tw • (B-Tw)/2

$Ix \quad = 4 \bullet ((B\text{-}Tw) \bullet (H\text{-}Tb/2\text{-}Tt/2))^2/((B\text{-}Tw)/Tb+(B\text{-}Tw)/Tt+2 \bullet (H\text{-}Tb/2\text{-}Tt/2)/Tw)$
$Iy \quad = B \bullet Tb \bullet (Naz\text{-}Tb/2)^2 + 2 \bullet Hw \bullet Tw \bullet (Hw/2+Tb\text{-}Naz)^2 + B \bullet Tt \bullet (H\text{-}Tt/2\text{-}Naz)^2$
$\quad + (B \bullet Tt^3+2 \bullet Tw \bullet Hw^3+B \bullet Tb^3) / 12$
$Iz \quad = (Tb \bullet Bb^3+Tt \bullet Bt^3+2 \bullet Hw \bullet T^3) /12 + 2 \bullet Hw \bullet Tw \bullet (B/2\text{-}Tw/2)^2$
$Wx \quad = Ix \bullet (H\text{-}(Tt+Tb)/2+ Bt\text{-}Tw)/((H\text{-}(Tt+Tb)/2) \bullet (Bt\text{-}Tw))$
$Wzt \quad = Wzb = Iz / (Bt/2)$
$Wyb \quad = Iy / Naz$
$Wyt \quad = Iy / (H\text{-}Naz)$
$ey \quad = 0$
$ez \quad = H\text{-}Tt/2\text{-}Naz\text{-}(Tb \bullet (H\text{-}(Tb+Tt)/2)/(Tb+Tt))$
$Naz \quad = (B \bullet Tt \bullet (H\text{-}Tt/2)+2 \bullet Hw \bullet Tw \bullet (Hw/2+Tb)+B \bullet Tb^2/2)/Ax$
$H \quad = Hw+Tb+Tt$

6.5.6 Pipe profile

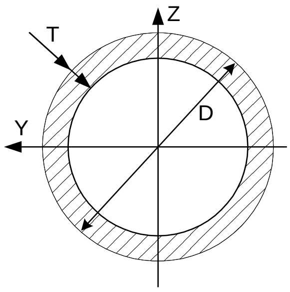


Figure 6.5-4 Pipe profile parameters

$Ax \quad = \pi \bullet ((D/2)^2 - (Di/2)^2)$
$Ay \quad = Az = Iy \bullet 2 \bullet T/Sy$
$Sy \quad = Sz = (D^3 - Di^3) / 12$
$Ix \quad = \pi /32 \bullet (D^4 - Di^4)$
$Iy \quad = Iz = \pi /64 \bullet (D^4 - Di^4)$
$Wx \quad = Ix / (D/2)$
$Wy \quad = Wz = Iy / (D/2)$
$ey \quad = ez = 0$
$Di \quad = D - 2 \bullet T$

7 Appendix B: Using spreadsheets with 3D Beam

7.1 Purpose

This appendix explains how you can simplify geometry modelling and result interpretation by use of a spreadsheet. These features are based on standard MS Windows functionality such as copy (Ctrl+C), cut (Ctrl+X) and paste (Ctrl+V).

7.2 Importing geometry

Carry out following steps to import geometry from a spreadsheet to 3D Beam:

1. Use a spreadsheet to generate a set of node coordinates. In this example is shown a spreadsheet developed by DNV for this purpose, *NodeCoordinates.xls*. This spreadsheet is included in a standard Nauticus Hull installation.

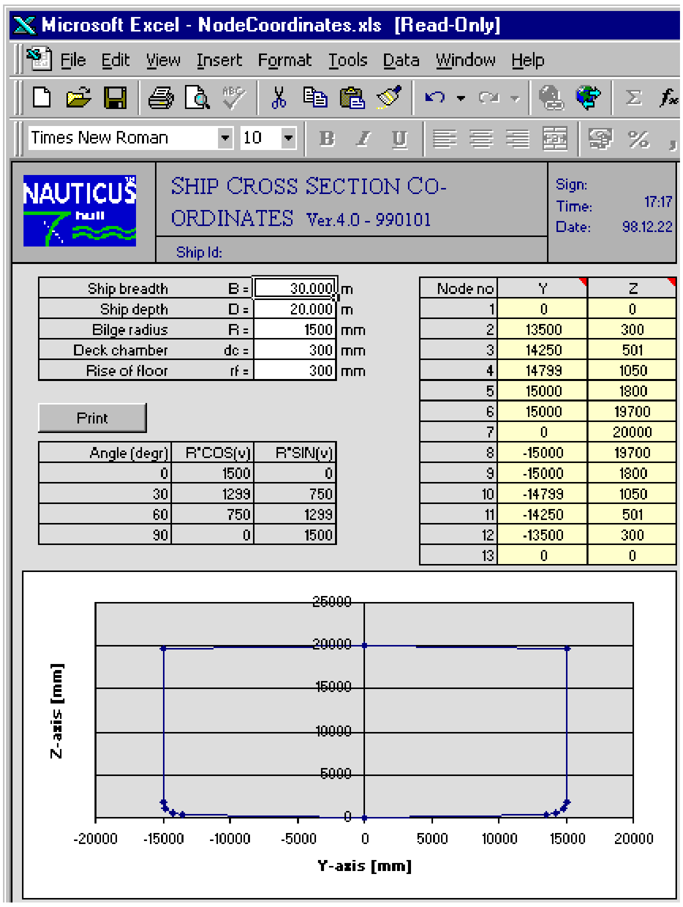
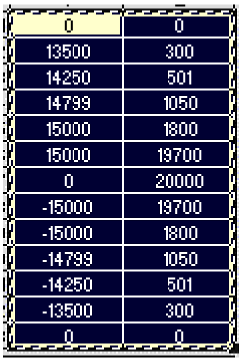


Figure 7.2-1 Node coordinates generated in a spreadsheet

2. Copy relevant values from the spreadsheet.



0	0
13500	300
14250	501
14799	1050
15000	1800
15000	19700
0	20000
-15000	19700
-15000	1800
-14799	1050
-14250	501
-13500	300
0	0

Figure 7.2-2 Node coordinates selected

3. Paste (Ctrl+V) the values into the Beam Wizard in 3D Beam.

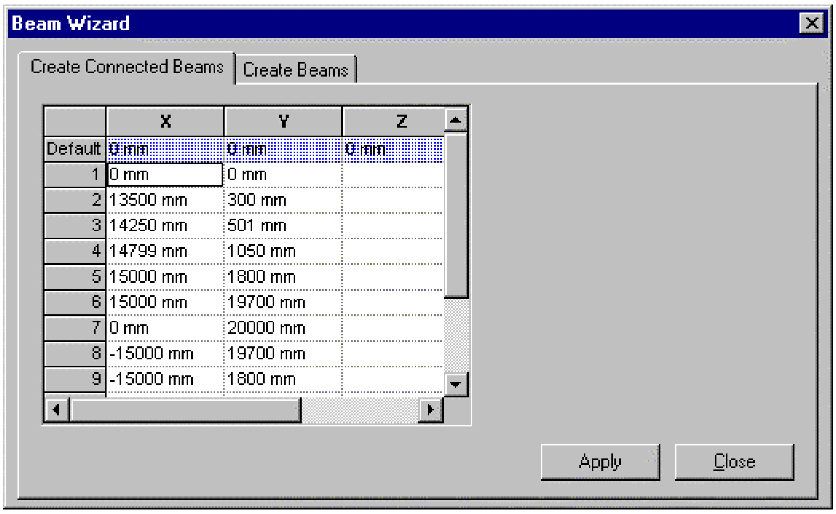


Figure 7.2-3 Node coordinates pasted into the Beam Wizard

4. Click the apply button.

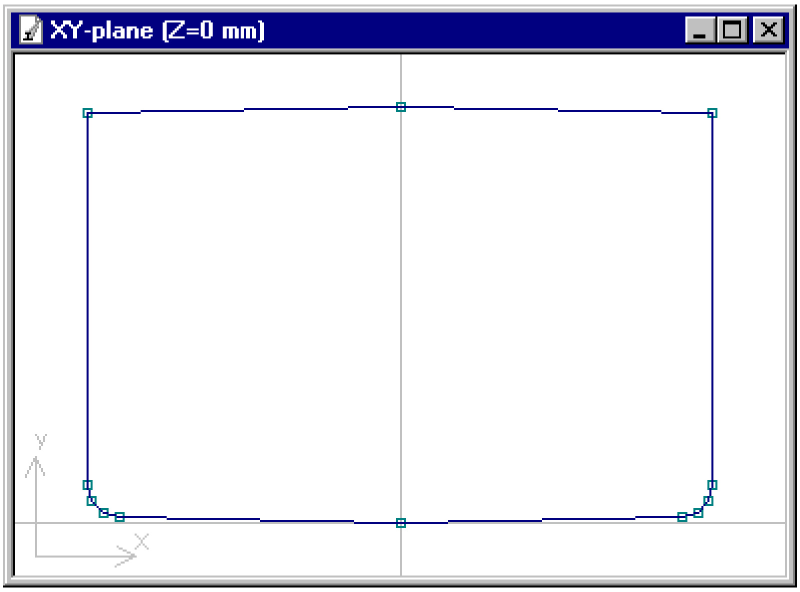


Figure 7.2-4 Beams generated by the Beam Wizard

7.3 Exporting results

To export result from 3D Beam you should select and copy (Ctrl+C) a range of cells in the result table. Paste (Ctrl+V) the selection into MS Word or MS Excel.

	Beam No.	Nx [N]	Qy [N]	Qz [N]	Mx [Nmm]	My [Nmm]
1	7	-621175	5551	389028	-2260377	-928795687
2	6	-682996	5551	306995	1396545	660018703
3	5	-723289	5551	226205	2584964	905196355
4	4	-742717	5550	-150624	2616272	905196354
5	3	-662041	5550	-356140	307955	763022460
6	2	-575883	5550	-442326	-3367227	-583375814
7	1	-555359	5550	-395673	-4809535	-1330609993
8	107	-681538	5486	306445	1383894	664625405
9	108	-721762	5486	225895	2553328	909439696
10	111	-619846	5486	388215	-2224637	-920896345
11	109	-741229	5486	-150159	2568861	909439695

Output | Beam responses | Stresses | Nodal responses | Beam details | Profiles | Notes

Figure 7.3-1 Copy a selection from result table

Microsoft Excel - Book1

File Edit View Insert Format Tools Data Window Help

Arial 10 B I U

	A	B	C	D	E
1	-723289	5551	226205	2584964	
2	-742717	5550	-150624	2616272	
3	-662041	5550	-356140	307955	
4	-575883	5550	-442326	-3367227	
5	-555359	5550	-395673	-4809535	
6	-681538	5486	306445	1383894	
7					
8					
9					

Figure 7.3-2 Export (paste) results into MS Excel

8 Appendix C: Window configuration

8.1 Window configuration

3D Beam provides features to customise your window configuration. Note that the default window configuration is normally the most optimal for interplay between the *Model window,* the *Input property window*, and the *Result property window*.

Float in main window:

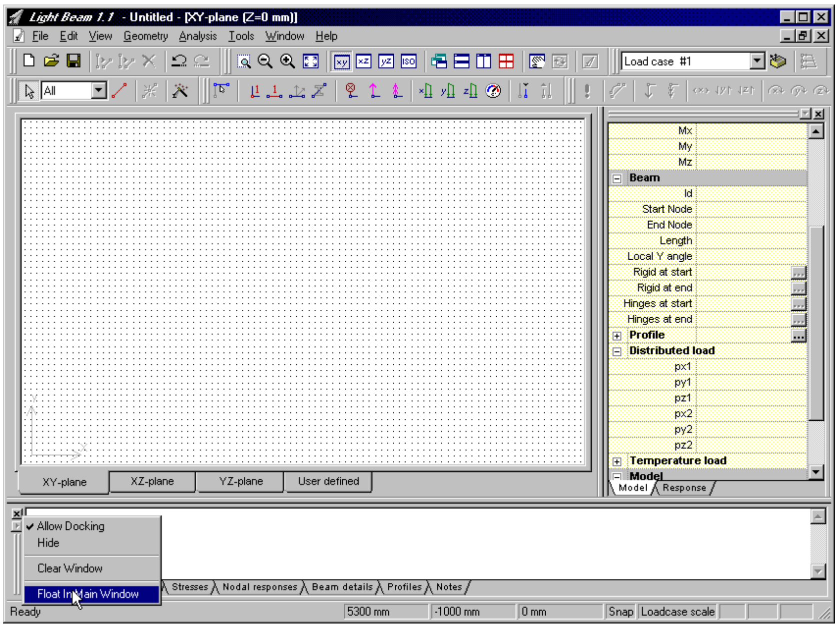


Figure 8.1-1 Selecting "Float in main window"

By clicking the right mouse button on the border frame of the *Input property window or Output window* and selecting the "Float in Main Window" option, the window will appear together with the *Model windows* in the workbook tab card row and it may be organised (tiled, cascaded etc.) in the same way as the *Model windows*.

By floating the *Input property window* and *Output window* in the main window you obtain more space for each window but loose the overview.

Undocking the windows:

If you un-select the "Allow docking" option in above pop-up menu the window un-docks from the application container and becomes free floating.

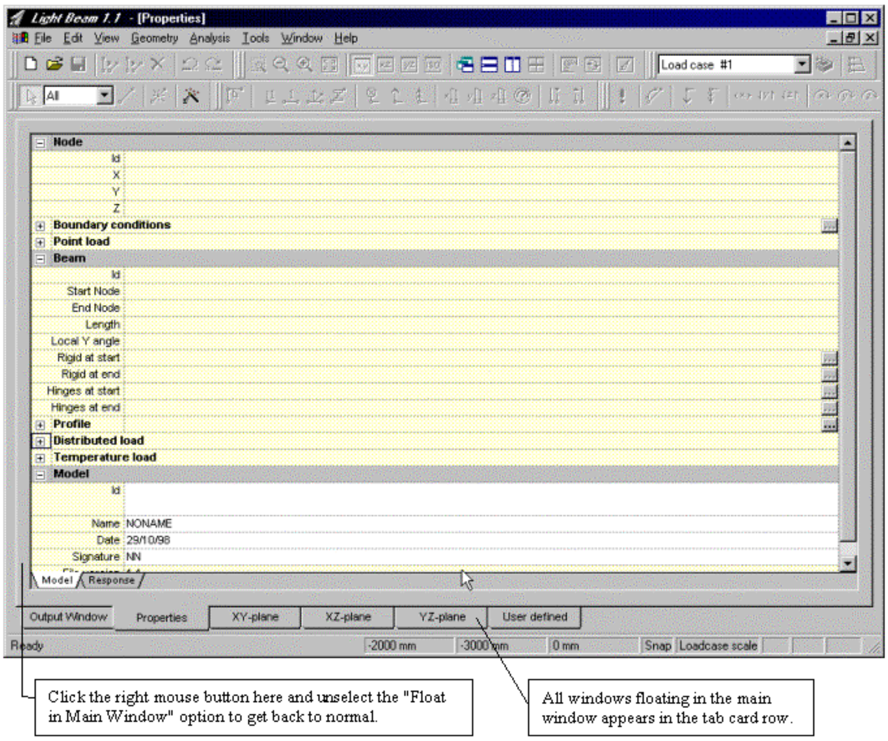


Figure 8.1-2 Floating in main window

If you click the right mouse button on the border frame of the *Input property window* or *Output window* and un-select the "Float in main window" it will return to the undocked (normal) position as before.

If you have maximised the active window in 3D Beam you will see the following buttons in the upper right corner of the 3D Beam window:

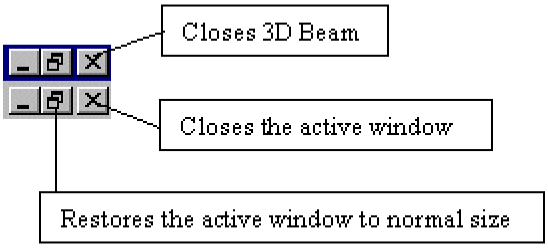


Figure 8.1-3 Close and restore window functions

9 Appendix D: 3D Beam as COM server

9.1 Purpose

The features of 3D Beam can be utilized from other applications by means of a COM interface suite. This means that applications for special purposes may be developed in say VB or Excel, and the analysis power of 3D Beam utilized. The following are possible examples:

- Generation of models and loads based on specific rules and parameters.
- Specialized result analysis such as buckling analyses or combined stress analyses.
- Tailor-made report design.

9.2 The 3D Beam COM interface

Please, see the separate document "3D Beam API" with description of the Na3DBeam library, which offers a set of COM interfaces to 3D-Beam modelling functionality. Most of the modelling available in 3D- Beam is available through this library, including analysis and result inspection.