

NAUTICUS 3D BEAM How to...



DET NORSKE VERITAS

Ноw то…

1.	IN	TRO	DUCTION	.1
	1.1.	Авоі	JT THIS MANUAL	. 1
	1.2.	Over	RVIEW OF 3D BEAM	. 1
	1.2	.1.	Menu bar	.1
	1.2	.2.	Toolbars	.1
	1.2	.3.	Input property window	.1
	1.2	.4.	Output window	.2
	1.2	.5.	Model window	.2
2.	MA	AKE	A BEAM ELEMENT ANALYSIS IN 9 STEPS	.3
	2.1	SPEC	IFY MODEL PROPERTIES	3
	2.2.	Desc	RIBE THE GEOMETRY	.3
	2.3.	SPEC	IFY THE BOUNDARY CONDITIONS	.3
	2.4.	SPEC	IFY THE BEAM PROFILE(S) AND MATERIAL(S)	.3
	2.5.	SPEC	IFY SPECIAL PROPERTIES	.3
	2.6.	SPEC	IFY THE LOADS	.3
	2.6	.1.	Node loads	.3
	2.6	.2.	Beam loads	.3
	2.7.	Run '	THE ANALYSIS	.4
	2.8.	Exan	AINE THE RESULTS	.4
	2.9.	Crea	ITE REPORTS	.4
3.	CR	REAT	TE A MODEL	.5
	3.1.	Drav	W A MODEL	.5
	3.1	.1.	Activate/deactivate snap to grid	. 5
	3.2.	Crea	TE A MODEL BY SPECIFYING NODE COORDINATES WITH THE BEAM WIZARD	.5
	3.2	.1.	Specify a set of connected beams	.5
	3.2	.2.	Specify individual beams	. 5
	3.3.	Сору	2 BEAMS	.5
	3.4.	Mov	e/Mirror beams	.6
	3.5.	Dele	TE BEAMS	.6
	3.6.	Insef	RT NODE/SPLIT BEAM	.6
	3.7.	Edit	NODE COORDINATES	.6
4.	MA	AKE	SELECTIONS OF NODES AND BEAMS	.7
	4.1.	Sele	CT NODES AND BEAMS IN THE MODEL WINDOW WITH THE CURSOR	.7
	4.1	.1.	Select all beams in the model	.7
	4.1	.2.	Un-select all beams	.7
	4.1	.3.	Adding/removing beams to/from a selection (One by one)	. 7
	4.1	.4.	Select/un-select beams within an area	.7
	4.1	.5.	Select/un-select beams touched by the rubber band (Ctrl-key)	.7

Hov	V ТО		
	4.1.6.	Select/un-select beams with the "touch"-method	7
	4.1.7.	Select nodes or beams only	7
4.	2. Selec	CT NODES AND BEAMS IN THE OUTPUT WINDOW	7
4.	3. USE N	JAMED SELECTIONS	
	4.3.1.	Save a selection	
	4.3.2.	Change the default names of selections	
	4.3.3.	Activate a selection	8
4.4	4. Show	/ FEATURES ON SELECTED BEAMS ONLY	
5.	WORK	IN 3D	9
5.	1. USE V	/iews in the Model Window	9
5.	2. See 3	PROJECTIONS AND AN ISO VIEW	9
5.	3. USE V	WORKING PLANES	9
	5.3.1.	Set the depth of the Working Plane	
5.4	4. Zoom	1 THE MODEL	
5.:	5. Move	E THE MODEL	
5.	6. Rota	TE THE MODEL	
5.	7. ADD A	ADDITIONAL VIEWS	
5.	8. Open	A SOLID VIEW	
6.	DEFINE	E BOUNDARY CONDITIONS	
7.	DEFINE	E AND APPLY PROFILES AND MATERIAL	
7	1 Арріл	Y A PROFILE TO BEAMS	13
7.	2 Crea	TE A NEW PROFILE	13
,	7.2.1	Specify a profile from a list of profile types	14
	7.2.2.	Select a profile from tables of standard profiles	
	7.2.3.	Import/Export profiles	
8.	DEFINE	E NEW MATERIALS	
0	SPECIE	V FYTDA REAM PROPERTIES	16
<i>.</i>	SIECH	I LATRA DLAM I KOI EKTILS	10
9.	I. DEFIN	JE THE BEAM ROTATION	
9.	2. DEFIN	JE RIGID ENDS	
9	3. DEFIN	↓E HINGED ENDS	
9.4	4. DEFIN	JE NON-LINEAR BEAMS	
10.	DEFI	INE AND APPLY LOADS	
10).1. Apply	Y POINT LOADS	
10).2. Apply	Y DISTRIBUTED LOADS	
	10.2.1.	Specify by direct input	
	10.2.2.	Using Load Wizards	
10).3. Apply	Y TEMPERATURE LOADS	
11.	MAN	AGE LOADCASES	

11.1	1. Specify a new loadcase	
11.2	2. COMBINE LOADCASES	21
11.3	3. EXCLUDE A LOADCASE FROM THE ANALYSIS	22
11.4	4. Activate a loadcase	22
12.	ANALYSE THE MODEL	23
13.	EXAMINE THE RESULTS	24
13.1	I. DISPLAY RESULTS IN THE MODEL WINDOW	24
1	3.1.1. Adjust the display scaling of responses	24
13.2	2. VIEW THE RESULTS IN TABULATED FORM	24
1	3.2.1. Sort the results in the grid	24
1	<i>13.2.2.</i> Display the response for selected beams only	24
13.3	3. VIEW RESULTS IN THE RESPONSE PROPERTY WINDOW	25
13.4	4. VIEW DETAILED RESPONSE PLOTS	
1	13.4.1. Flip direction of beams	
14.	CREATE REPORTS	27
14.1	1. CREATE REPORTS FROM THE TABLES	27
14.2	2. CREATE REPORTS FROM THE RESPONSE PLOT	27
1	14.2.1. Set Ruler positions by clicking	27
1	14.2.2. Set Ruler positions automatically	27
14.3	3. INCLUDE PLOTS IN THE REPORT	
1	14.3.1. Include plots of the model	
1	14.3.2. Include the Response plot	
15.	SPECIFY DISPLAY UNITS AND FORMATS	29
16.	USE INPUT FIELDS	30
16.1	1. USE UNITS IN INPUT	
16.2	2. Use incremental input	
17.	USE SHORTCUT KEYS	31
18.	USE MS EXCEL WITH 3D BEAM	32
18 1	POST-PROCESS TABLES IN MS EXCEL	32
18.2	2. Generate geometries from MS Excel	
19.	USE 3D BEAM IN A NAUTICUS JOB	
20.	USE A MOUSE WITH A WHEEL	34

1. Introduction

1.1. About this manual

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

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

1.2. Overview of 3D Beam

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



Graphical User Interface of 3D-Beam

1.2.1. Menu bar

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

1.2.2. Toolbars

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

1.2.3. Input property window

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

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

1.2.4. Output window

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

1.2.5. Model window

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

2. Make a beam element analysis in 9 steps

2.1. Specify model properties

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

2.2. Describe the geometry

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

2.3. Specify the boundary conditions

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

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

Select the desired beam(s) and click **Profile** Not defined in the **Input property window** to select a profile or create a new profile, with desired material, for the beam(s).

2.5. Specify special properties

Local rotation	0 °
Rigid at start	Not defined
Rigid at end	Not defined 🔜
Hinges at start	
Hinges at end	
Non Linearitie	Not defined

Click the appropriate button Non Linearitie Not defined in the **Input property window** to specify possible features such as Beam rotation, Rigid ends, Hinges, or Non-linearities.

2.6. Specify the loads

2.6.1. Node loads

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

E Po	oint load	
	Px	0,0 kN
	Py	0,0 kN
	Pz	0,0 kN
	Mx	0,0 kNm
	My	0,0 kNm
	Mz	0,0 kNm

Click 1 and 1 to toggle the display of node loads.

2.6.2. Beam loads

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

Ноw то...

Distribute	ed load
px1	0,0 kN/m
py1	0,0 kN/m
pz1	0,0 kN/m
px2	0,0 kN/m
py2	0,0 kN/m
pz2	0,0 kN/m
Temperation	ture load
Gy	0 °C/mm
Gz	0 °C/mm
Temperature	0,0 °C

Click 1, 1, and 2 to toggle the display of distributed loads and 2 to toggle the display of temperature loads.

2.7. Run the analysis

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

2.8. Examine the results



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

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

2.9. Create reports

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



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

Show only structure in Working Plane Define Working Plane		
✓ Snap to <u>g</u> rid	Ctrl+G	
Perspective		
Apply to selection only		
<u>C</u> opy model plot	•	Copy model plot to the clipboard and then paste this in
Word document		copy model plot to the enpotentia, and then puble this h

Word document.

3. Create a model

3.1. Draw a model

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

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

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

3.1.1. Activate/deactivate snap to grid

Menu: View | Snap to Grid

Shortcut Ctrl+G

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

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

3.2.1. Specify a set of connected beams

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

	X	Y	Z	Τ	
Default	0 mm	0 mm	0 mm		
1	0 mm				
2	3000 mm				
3	3000 mm		3000 mm		
4	0 mm		3000 mm		
5	0 mm				
6					
7					
8					
9					
10					

3.2.2. Specify individual beams

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

3.3. Copy beams

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

Ноw то...

Copy and Transform		X	1
Distance betw. copies-	Scale	Mirror about	
d⊻: 3000	≚: 1	□ ⊻Y-plane	
d <u>Y</u> : 0 mm	Y: 1	□ XZ-plane	
dZ: 0 mm	<u>∠</u> : 1	□ <u>Y</u> Z-plane	
Number of copies: 1		OK Cancel	to specify the copy parameters

3.4. Move/Mirror beams

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

3.5. Delete beams

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

3.6. Insert node/Split beam

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

Insert node	
Position ○ % from start ○ Length ○ Global X = ○ Global Y = ○ Global Z =	Cancel

3.7. Edit node coordinates

Select the desired node(s).

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

Node				
ld	*			
Name				
Х	5000,0 mm			
Y	0,0 mm			
Z	*			

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

4. Make selections of nodes and beams

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

By default, beams and nodes turn red when selected. Most default colours in the **Model window** may be changed by the user in the menu <u>T</u>ools | <u>O</u>ptions...

4.1.1. Select all beams in the model

Menu: <u>E</u>dit | Select A<u>l</u>l, or the shortcut: Ctrl+A,

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

4.1.2. Un-select all beams

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

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

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

4.1.4. Select/un-select beams within an area

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

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

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

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

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

4.1.7. Select nodes or beams only

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

5	Nodes 💌
	All Beams Nodes

4.2. Select nodes and beams in the Output window

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

Ноw то...

×		Beam	Name	Node 1	Node 2	Length [mm]	Profile	Profile angle [°]	Rigid start [mm]	Rigid End [mm]	HingStart	HingEnd	Non-lin
쉐	1	1		1	2	10000,0	1	0	0,0	500,0			
	2	2		2	3	10500,0	1	90	500,0	500,0			
	3	3		4	3	10000,0	1	180	0,0	500,0			
	4	4		5	4	10000,0	1	180	500,0	0,0			
	5	5		6	5	10500,0	1	-90	500,0	500,0			
	6	6		6	1	10000,0	1	0	500,0	0,0			
			_	_				<u>,</u>					
	- €	Outpu	t ∧ Bea	ams (N	odes 👌 F	Profiles A Bea	m loads	Node loads	Beam response	s λ Stresses λ №	lode respor	ises À Re	sponse plot Notes /

4.3. Use Named Selections

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

4.3.1. Save a selection

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

Selection toolbar

4.3.2. Change the default names of selections

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

Named Selection Manager	×
Named Selection Name	<u>0</u> K
Set#1	
Set#2	<u>C</u> ancel
	<u>D</u> elete
, Current Named Selection	
Name:	
Top tier	
Description:	
The top tier beams have smaller scantlings.	*

Where the selections can be given more informative names

4.3.3. Activate a selection

Choose selection from the drop-down list.

4.4. Show features on selected beams only

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

5. Work in 3D

5.1. Use Views in the Model Window

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



5.2. See 3 projections and an ISO view



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

5.3. Use Working Planes

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

Ноw то...

 Show only structure in Working Plane Werline Working Plane 							
Snap to grid	Ctrl+G						
<u>P</u> erspective							
Apply to selection only							
<u>C</u> opy model plot	•						

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

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

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

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



5.3.1. Set the depth of the Working Plane

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

Working plane settings								
Define working plane								
Working plane at X= 0 mm								
Show only structure in w	orking plane							
within ± 500 mm								
	Several 1							
	wh							

where the depth of the Working Plane may be set.

5.4. Zoom the model

5.5. Move the model

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

5.6. Rotate the model.

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

5.7. Add additional views

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

5.8. Open a solid view

To view the model in solid mode click the button \blacksquare

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

6. Define boundary conditions

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

Boundary Condition	ns	×
Degree of freedom	Boundary condition Value	0K.
X trans.	Fixed	
Y trans.	Fixed	Delete
Z trans.	Fixed	Cancel
X rot.	Free	
Y rot.	Fixed	
Z rot.	Fixed	
Set boundary cond	lition	
C Eree		
C Figed		
C Spring suppor	ted	
C Forced displat	sement	

Specify the relevant boundary conditions in the dialog box.

Click the button $\stackrel{\textcircled{P}}{\cong}$ in the toolbar to verify the boundary conditions visually.

7. Define and apply profiles and material

7.1. Apply a profile to beams

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

Choose profile to apply to selected beam(s)								
Id 1 2 3 4 5 6 7 ()	Name Default profile I-section Box section Hollow box welded from plates Circular Tube I-prifile Circular Tube between Plates	Dim 400.000000/50.000000/ 300.000000/30.000000/400.000000/20.000000/300.000000/40 400.000000/300.000000/30.000000/20.000000/40.000000/ 400 / 300 / 30 / 20 / 40 400 / 50 300 / 30 / 400 / 20 / 300 / 40 / 0 / 0 400 / 50 / 600 / 40 / 50 / 200						
- Dimensio Name: Outer Dia Thicknes	ns	(mm) (mm)						
2 Material:	Shear fac	tor in local y-a	Properties [mm] Ax: 54978 Ay: 27860 Az: 27860 Wyt: 4295 Wzt: 4295 Wx: 8590 Wyb: 4295 Wzb: 4295 Ix: 171806 Iyy: 85903 Izz: 85903 yNA: 0.0 zNA: 200.0 Iyz: 0 eY: 0.0 eZ: 0.0 ?					
1 Steel Hint.		fz	z: 1,00 Ignore Shear Centre Offset					

Click on the desired profile in the list.

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

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

7.2. Create a new profile

In the toolbar:

|--|--|

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

Ноw то...

Circular Tube

Select Standard Profile ...

7.2.1. Specify a profile from a list of profile types

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

Circular Tube	~
Circular Tube between Plates	H
Hollow box with constant Thickness and corner radius	
Hollow box welded from plates	
Const Thickn. Hollow Box betw. Plates	
Extruded Hollow Box betw. Plates	
Corrugated bent Plate profile	
Exstruded corrugated profile	~

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

•

7.2.2. Select a profile from tables of standard profiles

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

Stiffener scantlings	
Profile category	Profile type
	10 Flatbar
	Dimensions
Profile view	100 x 10
h <u>t</u>	Code h t 10 100 10 Profile table directory: m:\hull\data
	OK Cancel

Select the profile type and dimension.

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

7.2.3. Import/Export profiles

Click Import... or Export... in the toolbar

Export	Import	Delete	Edit	(New)
--------	--------	--------	------	-------

to re-use profiles between models.

8. Define new materials

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

N	Materials										
	ld 1 2	Name Steel Aluminium	E 210000 N/mm2 69000 N/mm2	Density 7800 kg/m3 2800 kg/m3	Poisson 0,3 0,3	Thermal Coefficient 1,26e-005 mm/mm/°C 2,4e-005 mm/mm/°C	Yield stress 235 N/mm2 110 N/mm2	<u>N</u> ew Delete			
								OK Cancel			

Steel and aluminium are available by default. Click <u>New</u> to specify new materials.

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

9. Specify extra beam properties

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

Non Linearitie	Not defined 🔜
Hinges at end	
Hinges at start	
Rigid at end	Not defined
Rigid at start	Not defined
LUCATIVIATION	•

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

9.1. Define the beam rotation

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

9.2. Define rigid ends

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

Along beam OK Distance 1500,0 mm dX 1500,0 mm dY 0,0 mm dZ 0,0 mm	Rigid at start				
	- Along bea Distance d⊻ d⊻ d⊻	m 1500,0 mm 1500,0 mm 0,0 mm 0,0 mm	OK Delete Cancel		

Specify the necessary parameters.

9.3. Define hinged ends

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

Hinges at start 🛛 🔀				
Translation	ОК			
	<u>D</u> elete			
Rotation	Cancel			
Note: Hinged beams should be short in order to obtain accurate displacements.				

Specify the necessary parameters.

9.4. Define non-linear beams

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

Non Linearities				
	ОК			
Gap 0,0 mm	<u>D</u> elete			
	Cancel			
✓ <u>T</u> ension				
Gap 0,0 mm				
Note:				
A linear beam supports compression and tension witout gaps. Check Tension to model a wire. Check Compression and add a gap to model a support with a distance from the model.				

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



10. Define and apply loads

10.1. Apply point loads

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

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

Click 1 and 1 to toggle the display of node loads.

10.2. Apply distributed loads

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

Click \mathbf{M} , \mathbf{M} , and \mathbf{M} to toggle the display of distributed loads and \mathbf{O} to toggle the display of temperature loads.

10.2.1. Specify by direct input

Specify distributed loads or temperature loads in the Input property window

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

10.2.2. Using Load Wizards

Activate the load wizards from the Loads toolbar



10.2.2.1. Create loads over selection

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

Create loads over interval					
Interval C⊻C⊻€Z	Distributed load in local Start value: p <u>x</u> 1: 0,0 kN/m	coordinate system End value: p <u>x</u> 2: 0,0 kN/m			
<u>Start:</u> 3000,0 mm	py1: 0,0 kN/m	ру2: 0,0 kN/m			
<u>E</u> nd: 4000,0 mm	p <u>z</u> 1: 0,0 kN/m	p <u>z</u> 2: 0,0 kN/m			
		OK Cancel			

10.2.2.2. Create inertia loads

Click 🕸 to specify accelerations to create loads due to inertia:

Create loads due to inertia forces					
Linear acceleration in:					
X - direction: 0.0 m/s2 X - axis: 0,0 *					
Y - direction: 0,0 m/s2 Y - axis: 0,0 *					
Z- direction: 0,0 m/s2 Z - axis: 0,0 *					
Angular accelerations					
Amplitude about Period Anglular accel.					
X · axis: 0,0 * 0 s ==> 0,000 rad/s2					
Y - axis: 0,0 ° 0 s ==> 0,000 rad/s2					
Z - axis: 0,0 * 0 s ==> 0,000 rad/s2					
Centre of rotation:					
X: 0,0 mm Y: 0,0 mm Z: 0,0 mm					
Note:					
 Values refer to the global coordinate system An acceleration in any direction generatas a load in the same direction Hint: Enter Z-acceleration = - 1G to specify self-weight Angular acceleration may be specified directly or by amplitude and period 					
OK Cancel					

10.2.2.3. Apply global loads

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

Ноw то...

A	ply loads in global directions					
	Distributed loads in global directions					
	at start node at end node					
	qX1: 0.0 kN/m qX2: 0.0 kN/m					
	qY1: 0,0 kN/m qY2: 0,0 kN/m					
	qZ1: 0,0 kN/m qZ2: 0,0 kN/m					
Apply on: Eull lenght <u>Projected length</u>						
	The loads in global direction may be applied either: - On the full length of the beam. For most loads except hydrostatic pressure.					
 On the projected length of the beam. For loads such as wind and snow. 						
OK Cancel						

10.3. Apply temperature loads

Enter temperature loads in the table

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

A Temperature different from 0 results in an axial load.

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

11. Manage Loadcases

	Loads	X
Click the Manage Load Cases button in the Loads to	olbar	
to bring up the <i>Loadcase manager</i> :		
Loadcase Manager		
Analyse Load Case Name Load case #1 Combine Delete <u>D</u> K <u>C</u> ancel		
Current Load Case Name: Load case #1 ✓ Include in Analysis		
Description:		

11.1. Specify a new loadcase

Click <u>New</u> and specify a name for the new load case and optionally a description.

Click Ok. This will be the current empty loadcase.

11.2. Combine loadcases

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

Combine Loadcases		×
Name: Combined loadcase		OK
Load case	Factor	Cancel
Load case #1	1.5	
Node Loads	2.0	
,		

Specify a name for the loadcase.

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

Click Ok.

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

Loadcase Manager	×
Analyse Load Case Name Load case #1 Node Loads Consideration	New Combine Delete
Current Load Case	<u>Q</u> K <u>C</u> ancel
Combination	Include in Analysis
Combination : Load case #1 (1.5), Node Loads (2)	K

Ноw то...

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

11.3. Exclude a loadcase from the analysis

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

11.4. Activate a loadcase

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

12. Analyse the model

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

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

13. Examine the results

13.1. Display results in the Model Window

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

1 🖉 🍹 🛊 🚥 4/1 4/21 🐼 🖓 🖓 No stress 💽

13.1.1. Adjust the display scaling of responses

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

To adjust the plotted amplitudes click the buttons:

13.2. View the results in tabulated form

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

×		Beam No.	Sig-Nx [N/mm	Tau-Qy [N/mm	Tau-Qz [N/mm	Tau-Mx [N/mm	Sig-My [N/mm	Sig-Mz [N/mm	Min Sig-Ny [N/	Max Sig-Ny [N	Min Sig-Nz [N/	Max Sig-Nz [N/mm2]
눼	1	1	0	0	82	0	117	0	-117	117	0	0
	2	2	-1	0	-50	0	126	0	-127	125	-1	-1
	3	3	2	0	4	0	14	0	-12	15	2	2
	4	4	2	0	-4	0	14	0	-12	15	2	2
	5	5	-1	0	-50	0	126	0	-127	125	-1	-1
	6	6	0	0	-82	0	117	0	-117	117	0	0
	▲)	Output A Bea	ms ∖ Nodes ∖	Profiles A Bea	im loads 👌 Nod	le loads 👌 Bea	m responses 👌	Stresses No	de responses 👌	Response plot	Notes /	

13.2.1. Sort the results in the grid

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



Specify sorting preferences.

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

13.2.2. Display the response for selected beams only

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

13.3. View results in the Response Property Window

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

Qy at end 0,0 kN Qz at start -944,4 kN Qz at end -117,6 kN Image: Torsional moment -0,0 kNm My at start -4231,5 kNm My at end 34,8 kNm Max My at pos 0,0 mm Max My at pos 0,0 kNm Max My at pos 0,0 kNm Max Mz at end -0,0 kNm Max MZ at pos 10500,0 mm Max Mz at pos 10500,0 mm Max X disp. -0,0 mm At position 10000,0 mm Max y disp. 0,0 mm At position 10000,0 mm Max Z disp. -4,8 mm At position 3825,0 mm Stresses Max Sig-Nx Max Tau-Qy 0 N/mm2 Max Sig-My 126 N/mm2 At position 500,0 mm Max Sig-My		<u> </u>	×		
Qz at start -944,4 kN Qz at end -117,6 kN □ Moments Torsional moment -0,0 kNm My at start -4231,5 kNm Max My at end 34,8 kNm Max My at pos. 0,0 mm Max My at pos. 0,0 kNm Max My at pos. 0,0 kNm Max Mz at start 0,0 kNm Max Mz at pos. 10500,0 mm Max Mz at pos. 10500,0 mm Max Mz at pos. 10000,0 mm Max X disp. -0,0 mm At position 10000,0 mm Max z disp. -4,8 mm At position 3825,0 mm □ Stresses Max Tau-Qy N/mm2 Max Sig-My 126 N/mm2 At position 500,0 mm Max Sig	Qy at end	0,0 kN	^		
Qz at end -117,6 kN □ Moments Torsional moment -0,0 kNm My at start -4231,5 kNm My at end 34,8 kNm Max My at pos 0,0 mm Max My at pos 0,0 kNm Max My at pos 0,0 kNm Mz at start 0,0 kNm Max Mz at pos 10500,0 mm □ Displacements (local coord) Max X disp. -0,0 mm At position 10000,0 mm Max y disp. 0,0 mm At position 3825,0 mm □ Stresses Max Sig-Nx -1 N/mm2 Max Tau-Qy 0 N/mm2 Max Tau-Qy 0 N/mm2 Max Tau-Qy 0 N/mm2 Max Tau-Mx 0 N/mm2 Max Sig-My 126 N/mm2 Min Sig-Mz 0 N/mm2 Max Sig-My 126 N/mm2 At position 500,0 mm Max Sig-My 126 N/mm2 At position 10000,0 mm Max Sig-My 125 N/mm2 At position 500,0 mm	Qz at start	-944,4 kN			
Image: model of the matrix of thematrix of thematrix of the matrix of thematrix of the matrix of	Qz at end	-117,6 kN			
Torsional moment -0,0 kNm My at start -4231,5 kNm Max My -4231,5 kNm Max My at pos. 0,0 mm Max My at pos. 0,0 mm Mz at start 0,0 kNm Mz at start 0,0 kNm Max Mz at pos. 10500,0 mm Max Mz at pos. 10500,0 mm Max X disp. -0,0 mm At position 10000,0 mm Max z disp. -0,0 mm At position 10000,0 mm Max z disp. -4,8 mm At position 3825,0 mm Stresses Max Tau-Qy Max Tau-Qy 0 N/mm2 Max Tau-Qy 0 N/mm2 Max Tau-Qy 0 N/mm2 Max Tau-Mx 0 N/mm2 Max Sig-My 126 N/mm2 At position 500,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Ny 125 N/mm2	Moments				
My at start -4231,5 kNm My at end 34,8 kNm Max My -4231,5 kNm Max My at pos. 0,0 mm Mz at start 0,0 kNm Mz at end -0,0 kNm Max Mz at pos. 10500,0 mm Max Mz at pos. 10500,0 mm Max Mz at pos. 10000,0 mm Max y disp. -0,0 mm At position 10000,0 mm Max z disp. -4,8 mm At position 10000,0 mm Max Z disp. -4,8 mm At position 3825,0 mm Stresses Max Sig-Nx Max Tau-Qy 0 N/mm2 Max Tau-Qy 0 N/mm2 Max Tau-Qy 0 N/mm2 Max Tau-Qy 0 N/mm2 Max Sig-My 126 N/mm2 Max Sig-My 126 N/mm2 At position 500,0 mm Max Sig-My 10000,0 mm Max Sig-My 126 N/mm2 At position 10000,0 mm Max Sig-My 126 N/mm2 At position 500,0 mm Max Sig-My 125 N/mm2 <	Torsional moment	-0,0 kNm			
My at end 34,8 kNm Max My at pos. 0,0 mm Max My at pos. 0,0 kNm Mz at start 0,0 kNm Mz at start 0,0 kNm Max Mz -0,0 kNm Max Mz at pos. 10500,0 mm □ Displacements (local coord) Max x disp. -0,0 mm At position 10000,0 mm Max y disp. 0,0 mm At position 10000,0 mm Max Z disp. -4,8 mm At position 3825,0 mm Stresses Max Tau-Qy Max Tau-Qy 0 N/mm2 Max Tau-Qy 0 N/mm2 Max Tau-Mx 0 N/mm2 Max Sig-My 126 N/mm2 Max Sig-My 126 N/mm2 At position 500,0 mm Max Sig-My 126 N/mm2 At position 10000,0 mm Max Sig-My 126 N/mm2 At position 500,0 mm Max Sig-My 126 N/mm2 At position 10000,0 mm Max Sig-My 125 N/mm2 At position 500,0 mm <td>My at start</td> <td>-4231,5 kNm</td> <td></td>	My at start	-4231,5 kNm			
Max My -4231,5 kNm Max My at pos. 0,0 mm Mz at start 0,0 kNm Mz at end -0,0 kNm Max Mz -0,0 kNm Max Mz at pos. 10500,0 mm Displacements (local coord) Max x disp. Max y disp. 0,0 mm At position 10000,0 mm Max z disp. -4,8 mm At position 3825,0 mm Stresses Max Tau-Qy Max Tau-Qy 0 N/mm2 Max Tau-Qy 0 N/mm2 Max Tau-Mx 0 N/mm2 Max Tau-Mx 0 N/mm2 Max Sig-My 126 N/mm2 Min Sig-My 126 N/mm2 At position 500,0 mm Max Sig-My 126 N/mm2 At position 500,0 mm Min Sig-My 126 N/mm2 At position 500,0 mm Min Sig-My 127 N/mm2 At position 10000,0 mm Max Sig-Ny 127 N/mm2 At position 500,0 mm Min Sig-Nz 1 N/mm2 At position 500,0 mm <td>My at end</td> <td>34,8 kNm</td> <td></td>	My at end	34,8 kNm			
Max My at pos. 0,0 mm Mz at start 0,0 kNm Mz at end -0,0 kNm Max Mz -0,0 kNm Max Mz at pos. 10500,0 mm Displacements (local coord) Max x disp. Max y disp. 0,0 mm At position 10000,0 mm Max z disp. -4,8 mm At position 3825,0 mm Stresses Max Tau-Qy Max Tau-Qy 0 N/mm2 Max Tau-Qy 0 N/mm2 Max Tau-Mx 0 N/mm2 Max Sig-My -126 N/mm2 Min Sig-My -126 N/mm2 Max Sig-My 126 N/mm2 At position 500,0 mm Min Sig-My 126 N/mm2 At position 500,0 mm Min Sig-Mz 0 N/mm2 At position 500,0 mm Min Sig-Mz 0 N/mm2 At position 10000,0 mm Min Sig-Ny -127 N/mm2 At position 500,0 mm Min Sig-Ny 125 N/mm2 At position 500,0 mm Min Sig-Nz -1 N/mm2	Max My	-4231,5 kNm			
Mz at start 0,0 kNm Mz at end -0,0 kNm Max Mz -0,0 kNm Max Mz at pos. 10500,0 mm Displacements (local coord) Max x disp. Max y disp. 0,0 mm At position 10000,0 mm Max z disp. -4,8 mm At position 10000,0 mm Max Z disp. -4,8 mm At position 3825,0 mm Stresses Max Tau-Qy Max Tau-Qy 0 N/mm2 Max Tau-Qy 0 N/mm2 Max Tau-Qy 0 N/mm2 Max Tau-Qy 0 N/mm2 Max Sig-My -126 N/mm2 At position 500,0 mm Max Sig-My 126 N/mm2 At position 500,0 mm Min Sig-Mz 0 N/mm2 At position 10000,0 mm Min Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Ny -127 N/mm2 At position 500,0 mm Min Sig-Ny 125 N/mm2 At position 500,0 mm Min Sig-Nz -1 N/mm2	Max My at pos.	0,0 mm			
Mz at end -0,0 kNm Max Mz at pos. 10500,0 mm Displacements (Iocal coord) Max x disp. -0,0 mm At position 10000,0 mm Max y disp. 0,0 mm At position 10000,0 mm Max z disp. -4,8 mm At position 3825,0 mm Stresses Max Sig-Nx Max Tau-Qy 0 N/mm2 Max Tau-Qy 0 N/mm2 Max Tau-Mx 0 N/mm2 Max Sig-My -126 N/mm2 Max Sig-My 126 N/mm2 At position 500,0 mm Max Sig-My 126 N/mm2 At position 500,0 mm Max Sig-My 126 N/mm2 At position 500,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Ny -127 N/mm2 At position 500,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Max Sig-Nz -1 N/mm2	Mz at start	0,0 kNm			
Max Mz -0,0 kNm Max Mz at pos. 10500,0 mm □ Displacements (local coord) Max x disp. -0,0 mm At position 10000,0 mm Max y disp. 0,0 mm At position 10000,0 mm Max z disp. -4,8 mm At position 3825,0 mm Stresses Max Sig-Nx Max Tau-Qy 0 N/mm2 Max Tau-Qz -50 N/mm2 Max Tau-Qz -50 N/mm2 Max Tau-Qz -50 N/mm2 Max Tau-Mx 0 N/mm2 Max Sig-My -126 N/mm2 At position 500,0 mm Max Sig-My 126 N/mm2 At position 500,0 mm Max Sig-My 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Ny -127 N/mm2 At position 500,0 mm Max Sig-Nz -1 N/mm2 At position 500,0 mm Max Sig-Nz -1 N/mm2	Mz at end	-0,0 kNm			
Max Mz at pos. 10500,0 mm □ Displacements (local coord) Max x disp. -0,0 mm At position 10000,0 mm Max y disp. 0,0 mm At position 10000,0 mm Max z disp. -4,8 mm At position 3825,0 mm □ Stresses Max Sig-Nx Max Tau-Qy 0 N/mm2 Max Tau-Qz -50 N/mm2 Max Tau-Mx 0 N/mm2 Max Sig-My 126 N/mm2 At position 500,0 mm Min Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Min Sig-Ny -127 N/mm2 At position 500,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Min Sig-Nz -1 N/mm2 At position 500,0 mm Min Sig-Nz -1 N/mm2 <tr< td=""><td>Max Mz</td><td>-0,0 kNm</td><td></td></tr<>	Max Mz	-0,0 kNm			
□ Displacements (local coord) Max x disp. -0,0 mm At position 10000,0 mm Max y disp. 0,0 mm At position 10000,0 mm Max z disp. -4,8 mm At position 3825,0 mm Imax Sig-Nx -1 N/mm2 Max Sig-Nx -1 N/mm2 Max Tau-Qy 0 N/mm2 Max Tau-Qy 0 N/mm2 Max Tau-Qy 0 N/mm2 Max Tau-Mx 0 N/mm2 Max Sig-My -126 N/mm2 At position 500,0 mm Max Sig-My 126 N/mm2 At position 500,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Ny -127 N/mm2 At position 500,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm	Max Mz at pos.	10500,0 mm			
Max x disp. -0,0 mm At position 10000,0 mm Max y disp. 0,0 mm At position 10000,0 mm Max z disp. -4,8 mm At position 3825,0 mm Imax Sig-Nx -1 N/mm2 Max Tau-Qy 0 N/mm2 Max Tau-Qz -50 N/mm2 Max Tau-Qz 0.00 mm Max Sig-My 126 N/mm2 At position 500,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Max Sig-Nz -1 N/mm2 At position 500,0 mm Min Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 <tr< td=""><td>Displacements (I</td><td>ocal coord)</td><td></td></tr<>	Displacements (I	ocal coord)			
At position 10000,0 mm Max y disp. 0,0 mm At position 10000,0 mm Max z disp. -4,8 mm At position 3825,0 mm Stresses Max Sig-Nx Max Tau-Qy 0 N/mm2 Max Tau-Qz -50 N/mm2 Max Tau-Qz -50 N/mm2 Max Tau-Mx 0 N/mm2 Max Sig-My -126 N/mm2 At position 500,0 mm Max Sig-My 126 N/mm2 At position 500,0 mm Max Sig-My 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Min Sig-Mz 0 N/mm2 At position 10000,0 mm Min Sig-Ny -127 N/mm2 At position 500,0 mm Max Sig-Nz -1 N/mm2 At position 500,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm	Max x disp.	-0,0 mm			
Max y disp. 0,0 mm At position 10000,0 mm Max z disp. -4,8 mm At position 3825,0 mm Imax Sig-Nx -1 N/mm2 Max Sig-Nx -1 N/mm2 Max Tau-Qy 0 N/mm2 Max Tau-Qz -50 N/mm2 Max Tau-Qz -50 N/mm2 Max Tau-Qz -50 N/mm2 Max Tau-Mx 0 N/mm2 Min Sig-My -126 N/mm2 At position 500,0 mm Max Sig-My 126 N/mm2 At position 500,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Max Sig-Nz -1 N/mm2 At position 500,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm <t< td=""><td>At position</td><td>10000,0 mm</td><td>_</td></t<>	At position	10000,0 mm	_		
At position 10000,0 mm Max z disp. -4,8 mm At position 3825,0 mm Stresses Max Sig-Nx Max Tau-Qy 0 N/mm2 Max Tau-Qz -50 N/mm2 Max Tau-Mx 0 N/mm2 Max Sig-My 126 N/mm2 At position 500,0 mm Max Sig-My 126 N/mm2 At position 500,0 mm Min Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm <	Max y disp.	0,0 mm			
Max z disp. -4,8 mm At position 3825,0 mm Stresses Max Sig-Nx Max Tau-Qy 0 N/mm2 Max Tau-Qz -50 N/mm2 Max Tau-Mx 0 N/mm2 Min Sig-My -126 N/mm2 At position 500,0 mm Min Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Min Sig-Mz 0 N/mm2 At position 10000,0 mm Min Sig-Ny -127 N/mm2 At position 500,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Min Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2	At position	10000,0 mm			
At position 3825,0 mm Stresses Max Sig-Nx -1 N/mm2 Max Tau-Qy 0 N/mm2 Max Tau-Qz -50 N/mm2 Max Tau-Qz -50 N/mm2 Max Tau-Qz -50 N/mm2 Max Tau-Mx 0 N/mm2 Max Sig-My -126 N/mm2 At position 500,0 mm Max Sig-My 126 N/mm2 At position 500,0 mm Min Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Ny -127 N/mm2 At position 500,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 100	Max z disp.	-4,8 mm			
Stresses Max Sig-Nx -1 N/mm2 Max Tau-Qy 0 N/mm2 Max Tau-Qz -50 N/mm2 Max Tau-Mx 0 N/mm2 Max Tau-Mx 0 N/mm2 Min Sig-My -126 N/mm2 At position 500,0 mm Max Sig-My 126 N/mm2 At position 500,0 mm Min Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Ny -127 N/mm2 At position 500,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm	At position	3825,0 mm			
Max Sig-Nx -1 N/mm2 Max Tau-Qy 0 N/mm2 Max Tau-Qz -50 N/mm2 Max Tau-Mx 0 N/mm2 Max Tau-Mx 0 N/mm2 Min Sig-My -126 N/mm2 At position 500,0 mm Max Sig-My 126 N/mm2 At position 500,0 mm Min Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Min Sig-Ny -127 N/mm2 At position 500,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm	Stresses				
Max Tau-Qy 0 N/mm2 Max Tau-Qz -50 N/mm2 Max Tau-Mx 0 N/mm2 Min Sig-My -126 N/mm2 At position 500,0 mm Max Sig-My 126 N/mm2 At position 500,0 mm Max Sig-My 126 N/mm2 At position 500,0 mm Min Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Ny -127 N/mm2 At position 500,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm	Max Sig-Nx	-1 N/mm2			
Max Tau-Qz -50 N/mm2 Max Tau-Mx 0 N/mm2 Min Sig-My -126 N/mm2 At position 500,0 mm Max Sig-My 126 N/mm2 At position 500,0 mm Max Sig-My 126 N/mm2 At position 500,0 mm Min Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Ny -127 N/mm2 At position 500,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Min Sig-Nz -1 N/mm2 At position 10000,0 mm Min Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm	Max Tau-Qy	0 N/mm2			
Max Tau-Mx 0 N/mm2 Min Sig-My -126 N/mm2 At position 500,0 mm Max Sig-My 126 N/mm2 At position 500,0 mm Min Sig-Mz 0 N/mm2 At position 10000,0 mm Min Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Min Sig-My -127 N/mm2 At position 500,0 mm Min Sig-Ny -127 N/mm2 At position 500,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Min Sig-Nz -1 N/mm2 At position 10000,0 mm Min Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm	Max Tau-Qz	-50 N/mm2			
Min Sig-My -126 N/mm2 At position 500,0 mm Max Sig-My 126 N/mm2 At position 500,0 mm Min Sig-Mz 0 N/mm2 At position 10000,0 mm Min Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Min Sig-Ny -127 N/mm2 At position 500,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm	Max Tau-Mx	0 N/mm2			
At position 500,0 mm Max Sig-My 126 N/mm2 At position 500,0 mm Min Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Min Sig-Ny -127 N/mm2 At position 500,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Min Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm	Min Sig-My	-126 N/mm2			
Max Sig-My 126 N/mm2 At position 500,0 mm Min Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Min Sig-Ny -127 N/mm2 At position 500,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Min Sig-Nz -1 N/mm2 At position 10000,0 mm Min Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm	At position	500,0 mm			
At position 500,0 mm Min Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Min Sig-Ny -127 N/mm2 At position 500,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Min Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm	Max Sig-My	126 N/mm2			
Min Sig-Mz 0 N/mm2 At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Min Sig-Ny -127 N/mm2 At position 500,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Min Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm	At position	500,0 mm			
At position 10000,0 mm Max Sig-Mz 0 N/mm2 At position 10000,0 mm Min Sig-Ny -127 N/mm2 At position 500,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Min Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm	Min Sig-Mz	0 N/mm2			
Max Sig-Mz 0 N/mm2 At position 10000,0 mm Min Sig-Ny -127 N/mm2 At position 500,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Min Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm	At position	10000,0 mm			
At position 10000,0 mm Min Sig-Ny -127 N/mm2 At position 500,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Min Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm	Max Sig-Mz	0 N/mm2			
Min Sig-Ny -127 N/mm2 At position 500,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Min Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm	At position	10000,0 mm			
At position 500,0 mm Max Sig-Ny 125 N/mm2 At position 500,0 mm Min Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm	Min Sig-Ny	-127 N/mm2			
Max Sig-Ny 125 N/mm2 At position 500,0 mm Min Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm	At position	500,0 mm			
At position 500,0 mm Min Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm	Max Sig-Ny	125 N/mm2			
Min Sig-Nz -1 N/mm2 At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm	At position	500,0 mm			
At position 10000,0 mm Max Sig-Nz -1 N/mm2 At position 10000,0 mm	Min Sig-Nz	-1 N/mm2			
Max Sig-Nz -1 N/mm2 At position 10000,0 mm	At position	10000,0 mm			
At position 10000,0 mm	Max Sig-Nz	-1 N/mm2			
Model Response /	At position	10000,0 mm	¥		
	Model Response /				

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

Ноw то...

13.4. View detailed response plots

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

Select the desired beam(s).

Select the desired response from the drop-down list.



13.4.1. Flip direction of beams

Each beam is plotted from its start node to its end node. This means that plots of connected beams may appear confusing. To correct this, the direction of one or more beam(s) can be changed. Select the

desired beam(s) and click the *Flip Beam* button on the *Geometry* tool bar and run the analysis again. Below figure shows the plot before and after flipping the direction of beam no 1.



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

14. Create reports

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

14.1. Create reports from the tables

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



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

[D

To generate a report for selected items only, click the *Apply to selection only* button Delete or save the report before generating a new report.

14.2. Create reports from the Response plot

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

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



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

14.2.1. Set Ruler positions by clicking

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

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

14.2.2. Set Ruler positions automatically

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

Ноw то...

<u>C</u> opy graph Clear positions
Insert positions at nodes
Insert positions at beam ends
Insert positions at <u>m</u> axima
<u>R</u> eport

14.3. Include plots in the report

14.3.1. Include plots of the model

Generate a desired report. Bring 3D Beam to the front (Alt+Tab) Right-click in the *Model Window* and select *Copy model plot* | *To Clipboard*



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

Any number of plots may be added to the report.

14.3.2. Include the Response plot

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

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

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

Repeat for as many plots as desired.

15. Specify display units and formats

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

Phenomenon	Unit	Format	Places	^
Angular acceleration	rad/s2	Fixed	3	
Area	cm2	Fixed	2	
Density	kg/m3	Fixed	0	
Distributed load	kN/m	Fixed	1	
Element orientation	•	Fixed	0	
Force	kN	Fixed	1	
Length	mm	Fixed	1	
Linear acceleration	m/s2	Fixed	1	
Mass	kg	Fixed	0	
Moment of inertia	cm4	Fixed	1	
Moment	kNm	Fixed	1	
Profile Scantlings	mm	Fixed	1	-
Rotation	•	Fixed	1	
Section modulus	cm3	Fixed	1	
Spring rotation stiffness	Nmm/rad	General	4	
Spring translation stiffness	N/mm	General	4	~

16. Use input fields

16.1. Use units in input

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

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

16.2. Use incremental input

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

Example: A field is displaying 1,0 mm.

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

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

17. Use shortcut keys

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

18. Use MS Excel with 3D Beam

18.1. Post-process tables in MS Excel

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

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

18.2. Generate geometries from MS Excel

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

19. Use 3D Beam in a Nauticus Job

Start Nauticus Project Manager

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

Locate the 3D Beam tool in the folder:

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

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

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



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

Click the button 🖆 to open 3D Beam



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

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

20. Use a mouse with a wheel

3D Beam is compliant with MS IntelliMouse actions.



The following features are provided:

In the **Model window**:

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