

SESAM TUTORIAL

# GeniE

## Topside with Detailed Modelling of Joint

Valid from program version 8.2

---



## Sesam Tutorial

GeniE – Topside with Detailed Modelling of Joint

Date: June 2021

Valid from GeniE version 8.2

Prepared by: Digital Solutions at DNV

E-mail support: [software.support@dnv.com](mailto:software.support@dnv.com)

E-mail sales: [digital@dnv.com](mailto:digital@dnv.com)

© DNV AS. All rights reserved

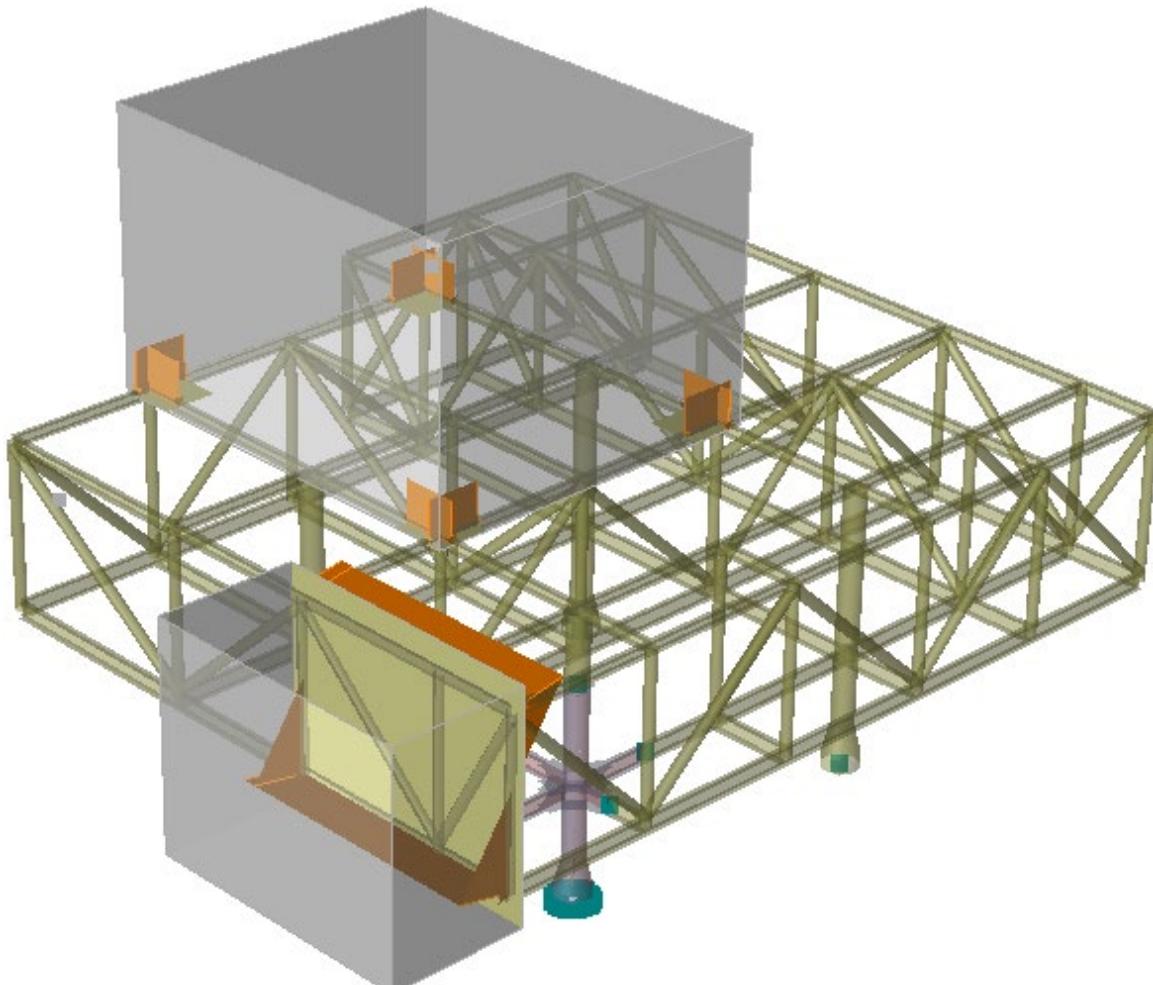
This publication or parts thereof may not be reproduced or transmitted in any form or by any means, including copying or recording, without the prior written consent of DNV AS.

## TABLE OF CONTENTS

1. Introduction	Page 4
2. Open New Workspace	Page 5
3. Material and Beam Cross Sections	Page 6
4. Meshing Rules	Page 9
5. Guiding Geometry	Page 10
6. Cellar Deck	Page 11
7. Main Deck	Page 12
8. Columns	Page 14
9. Deck Rows 1 to 6 in XZ-Plane	Page 18
10. Deck Rows A to E in YZ-Plane	Page 23
11. Loads	Page 26
12. Boundary Conditions	Page 39
13. Static Analysis	Page 40
14. Present Results	Page 41
15. Save the Workspace	Page 42
16. Detailed Modelling of a Joint	Page 43
17. Create FE Mesh	Page 58
18. Static Analysis of Combined Beam and Plate/Shell Model	Page 59

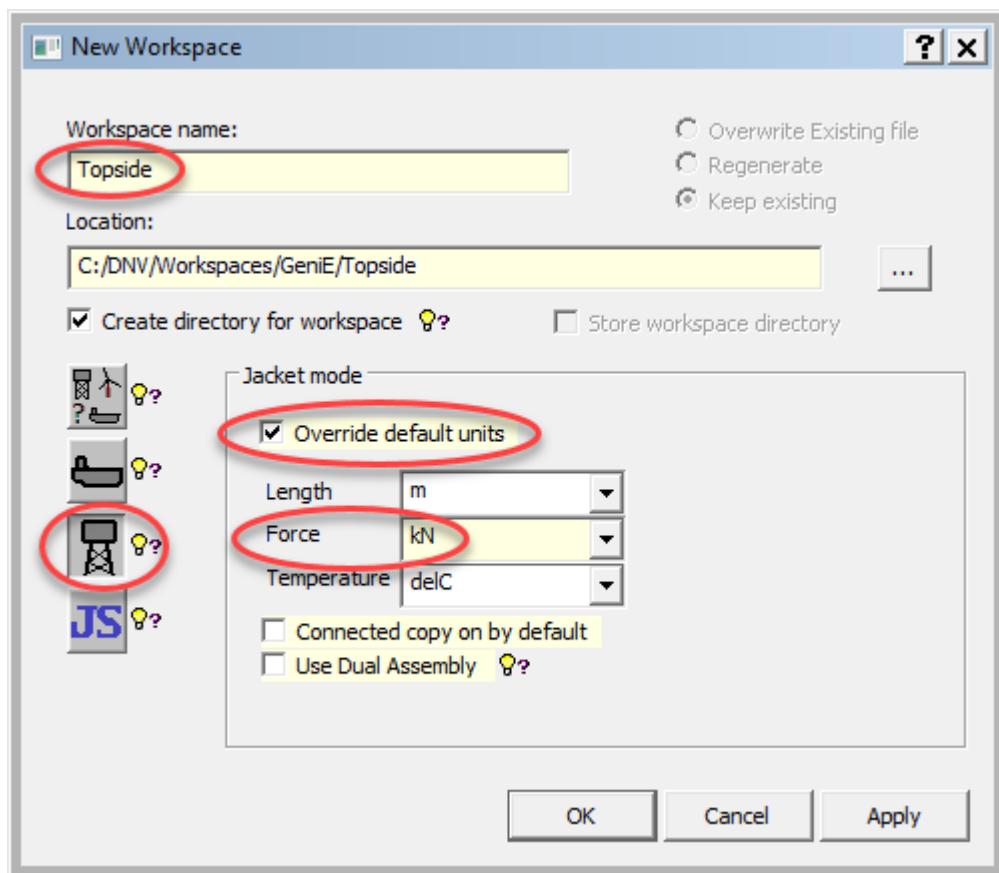
## 1 INTRODUCTION

- In this tutorial a beam model is created for a topside structure and analysed for self weight and equipment loads. Thereafter detailed modelling of a joint using plate/shell elements is performed and the revised model is analysed for the same loads. The tutorial contains the following steps:
  - Define materials and beam cross sections.
  - Define guiding geometry to facilitate the modelling.
  - Model the cellar deck, main deck, columns and deck rows.
  - Add loading conditions, specify boundary conditions and run linear analysis.
  - Convert a joint from beam model to plate/shell model.
  - Create a FE mesh for the joint.
  - Run linear analysis for the combined beam and plate/shell model.
- A GeniE input file for creating the complete model is provided.
- The model with equipments is displayed below.



## 2 OPEN NEW WORKSPACE

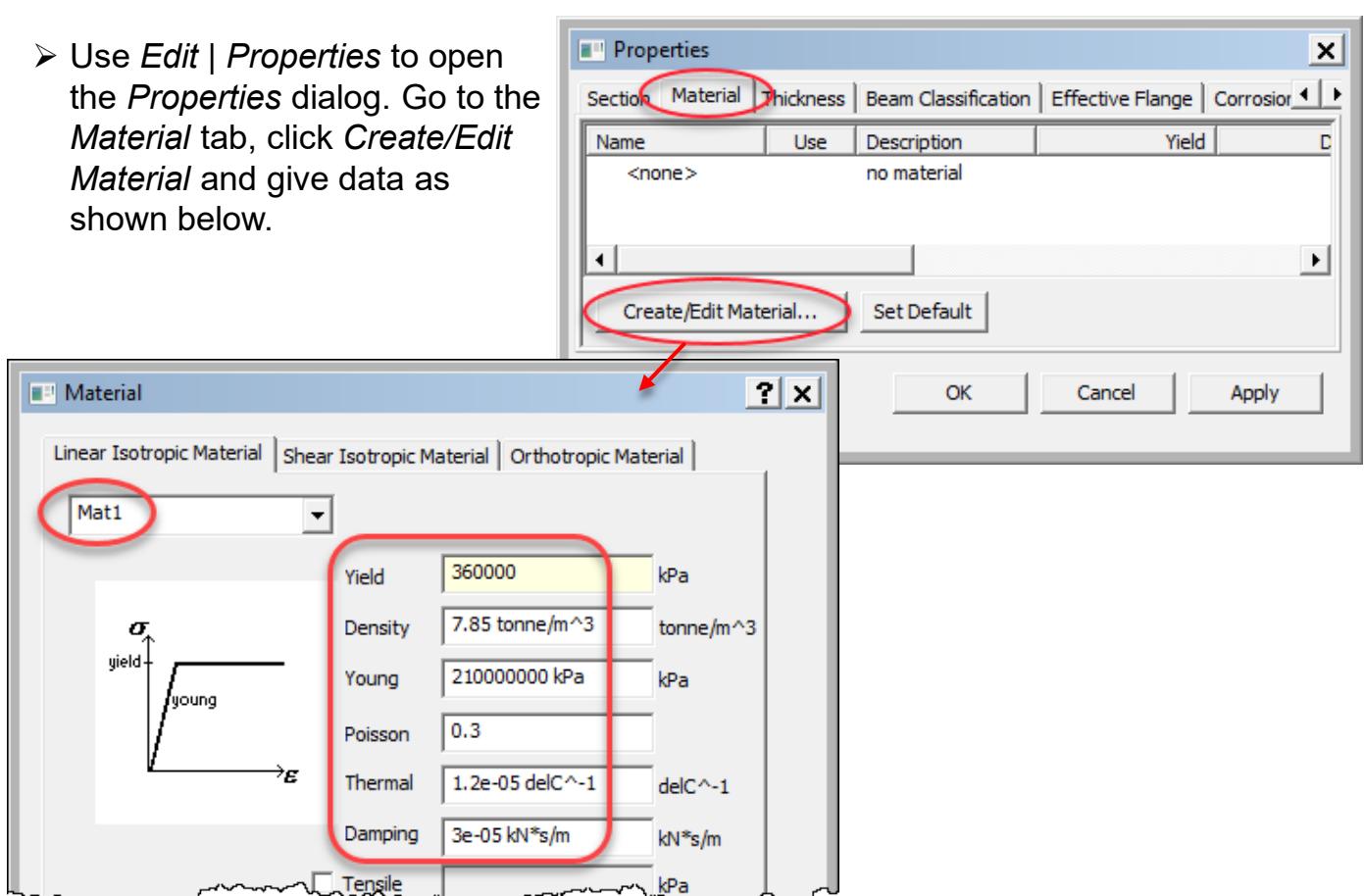
- Start GeniE and open a new workspace.
- Give a *Workspace name*.
  - Click the *Jacket mode* button to customise for jacket (frame) modelling, i.e. limit menus and buttons to those relevant for frame modelling.
  - Check *Override default units* and change *Force* unit to kN and click *OK*.



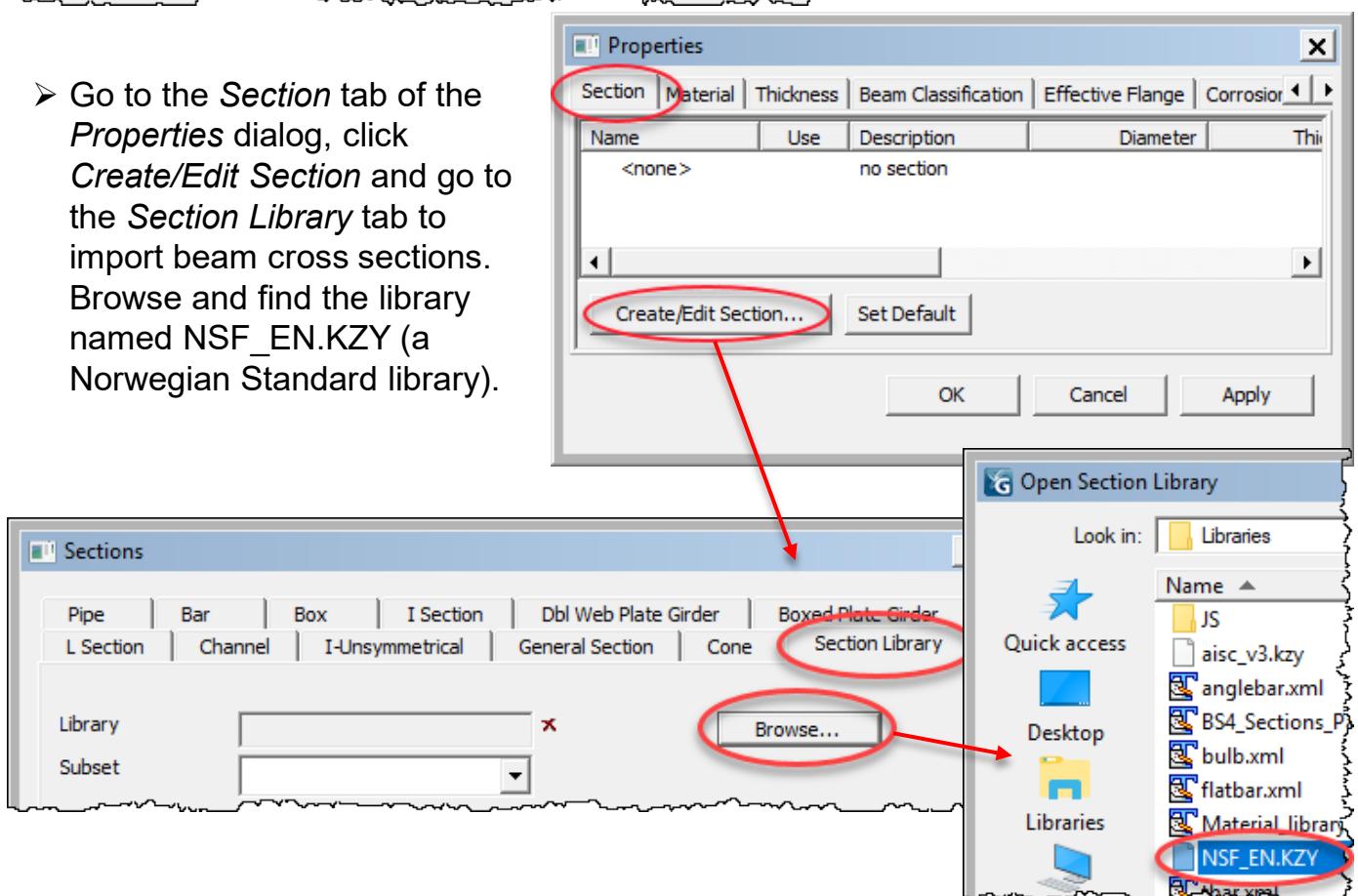
- If the workspace name exists select *Overwrite Existing file* or give another name.

### 3 MATERIAL AND BEAM CROSS SECTIONS

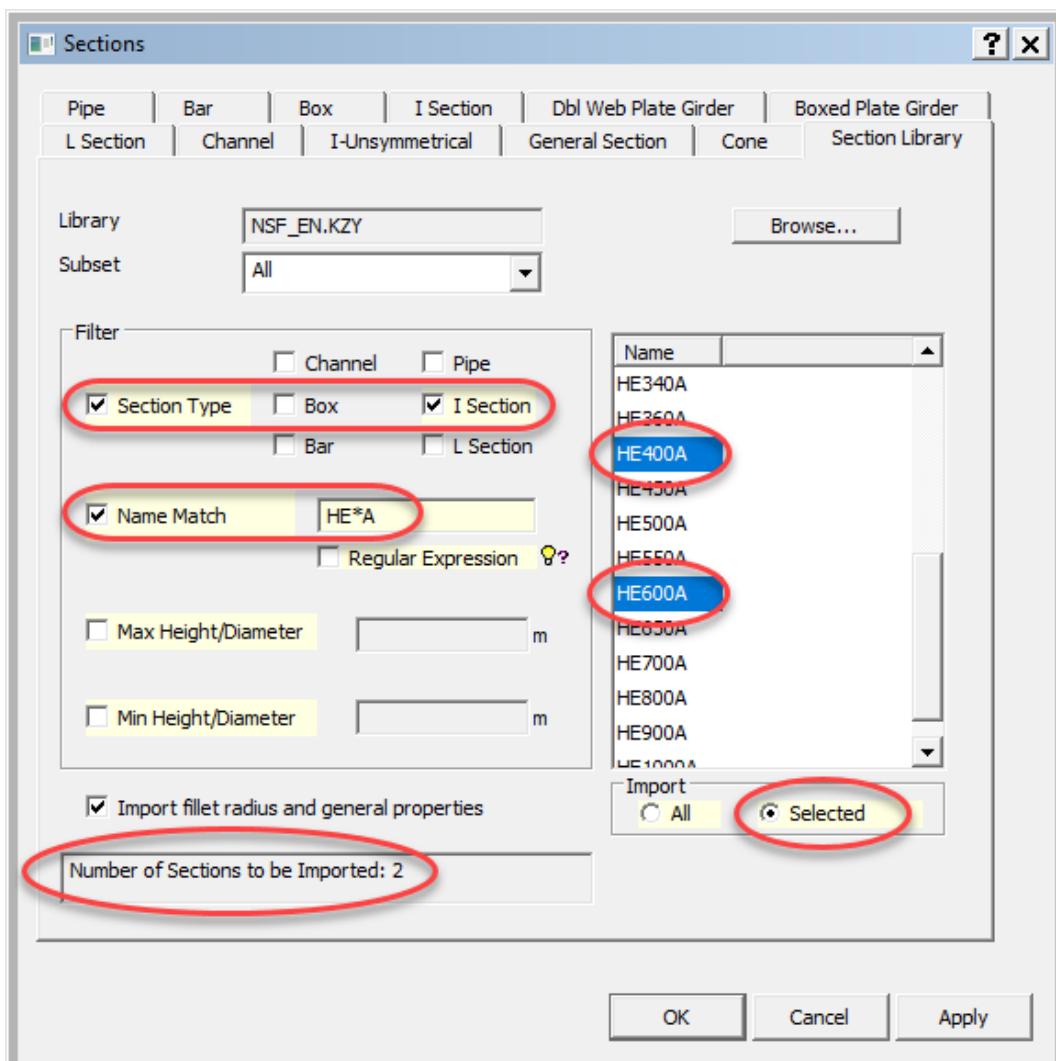
- Use *Edit | Properties* to open the *Properties* dialog. Go to the *Material* tab, click *Create/Edit Material* and give data as shown below.



- Go to the *Section* tab of the *Properties* dialog, click *Create/Edit Section* and go to the *Section Library* tab to import beam cross sections. Browse and find the library named **NSF\_EN.KZY** (a Norwegian Standard library).



- Having selected the library NSF\_EN.KZY see that several beam section names are listed.
- Check *Section Type* and *I Section* to filter the list of sections.
  - Check *Name Match* and enter HE\*A to filter the list further.
  - Select the section names (Click and Ctrl+Click):
    - HE400A
    - HE600A
  - Check *Selected* below the list to import only the two selected sections.
  - Make sure number of sections to be imported is 2 and click *OK*.

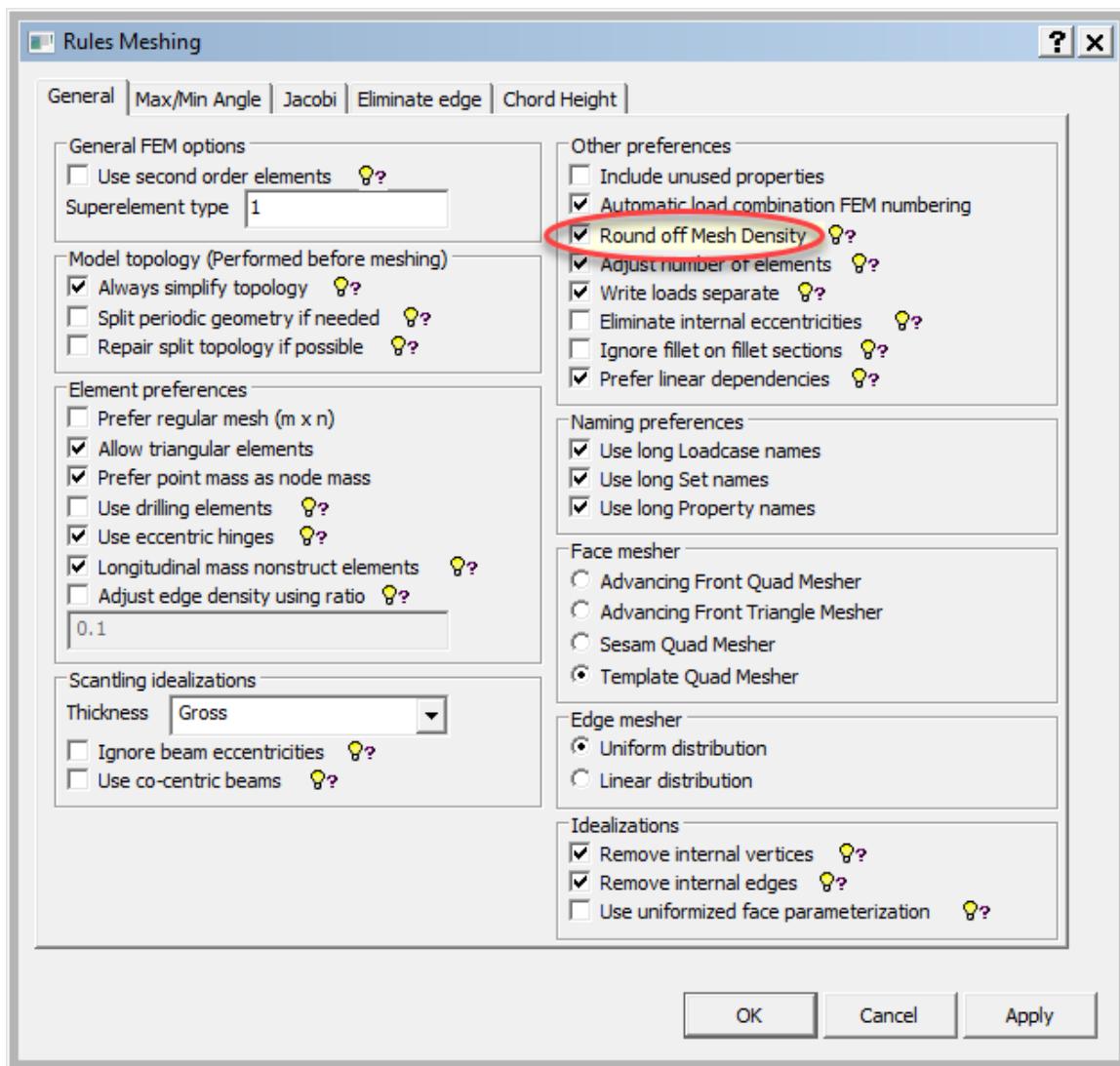


- Once more, in the *Section* tab of the *Properties* dialog, click *Create/Edit Section* and go to the *Pipe Section* tab to create four pipe (tube) beam cross sections as shown. Notice the use of mixed units: the unit specified to the right of the field is valid unless a unit is specified together with the value in the field.



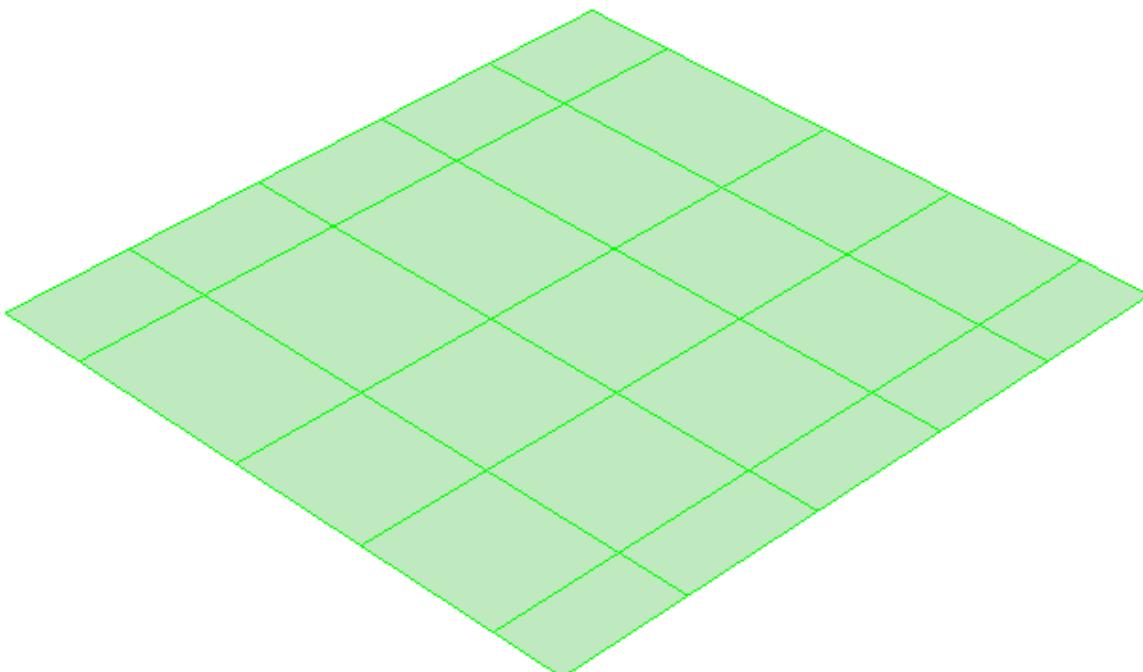
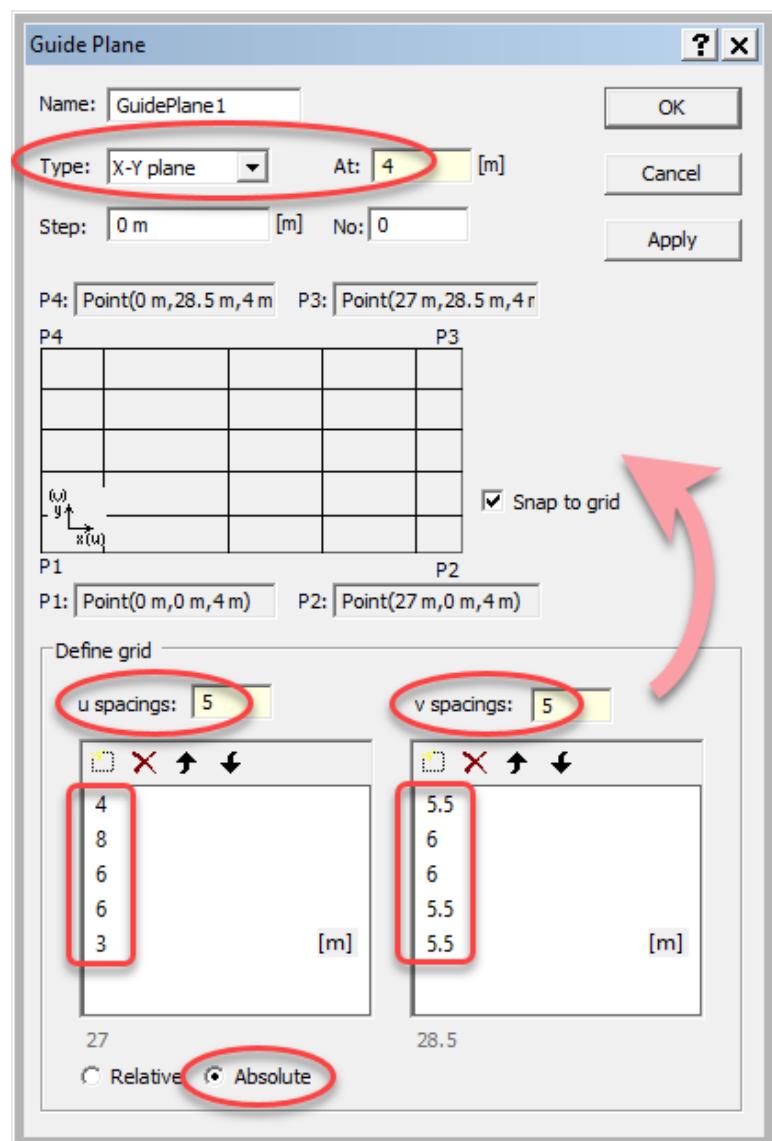
## 4 MESHING RULES

- Use *Edit | Rules | Meshing Rules* to open the dialog below and check the *Round off Mesh Density* check box which involves that element edges/lengths are allowed to be slightly larger than the given mesh density.
- Keep the default for other options.



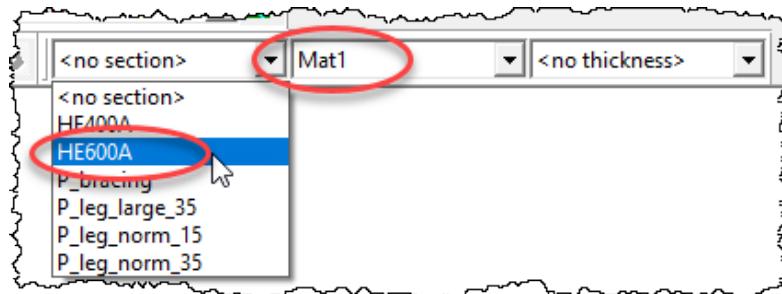
## 5 GUIDING GEOMETRY

- Use *Guiding Geometry | Planes | Guide Plane Dialog* to open the dialog to the right.
- Alternatively to filling in the coordinates for points  $P1$  to  $P4$  directly, do as follows:
  - Select *Type X-Y plane At 4*.
  - Switch from *Relative* to *Absolute* spacing at the bottom of the dialog.
  - Set both number of *u spacings* and *v spacings* to 5 and fill in the spacing values as shown.
  - See that the coordinates for points  $P1$  to  $P4$  are filled in automatically reflecting the given data.
  - Click *OK* and see that guide plane shown below appears.

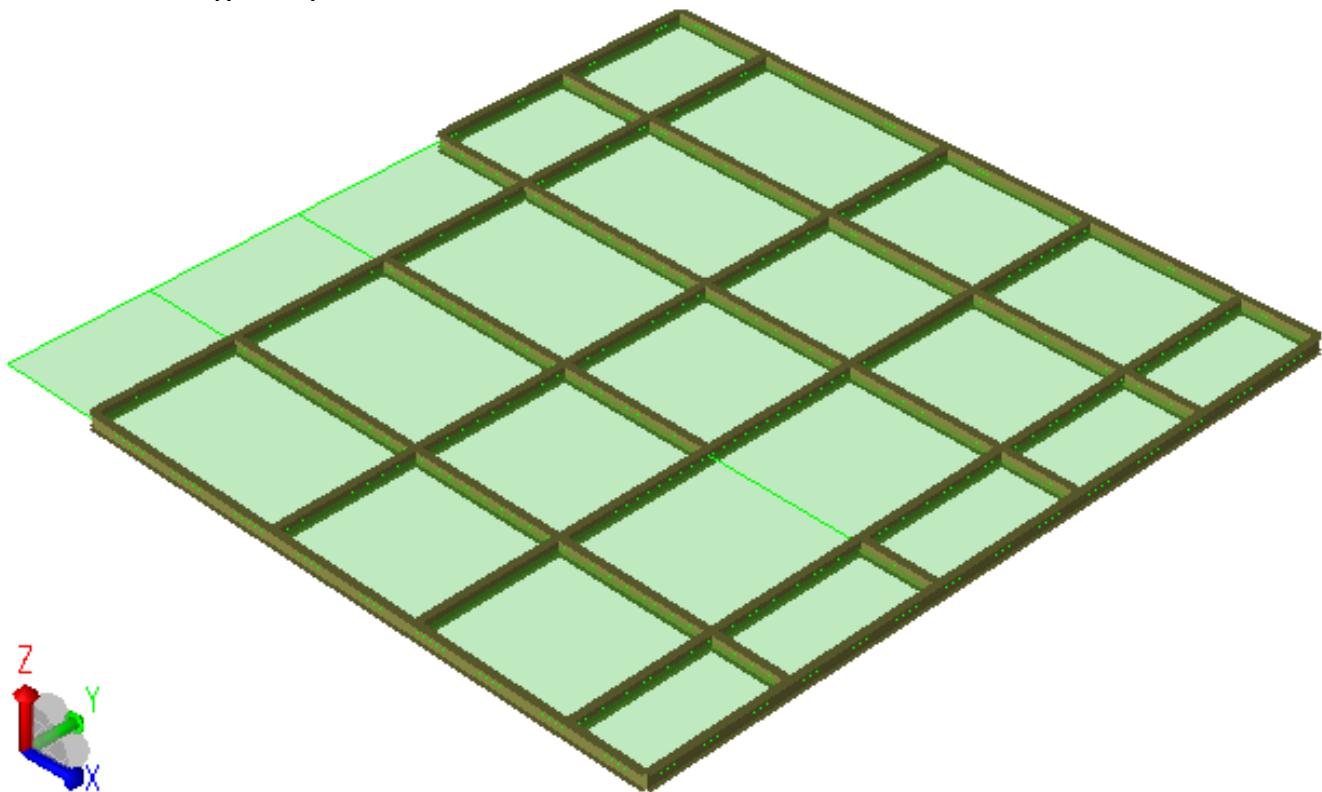


## 6 CELLAR DECK

- Start by setting default material and beam cross section to Mat1 and HE600A, respectively:

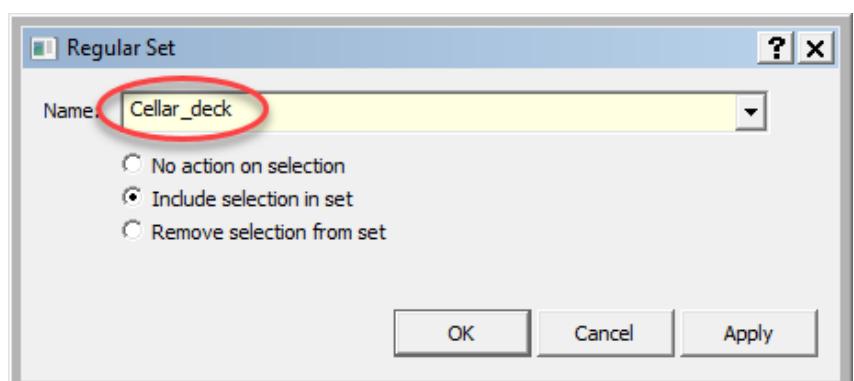


- Create beams by *Structure | Beams and Piles | Straight Beam* (or press ). Click in the guide plane to insert the beams shown below.



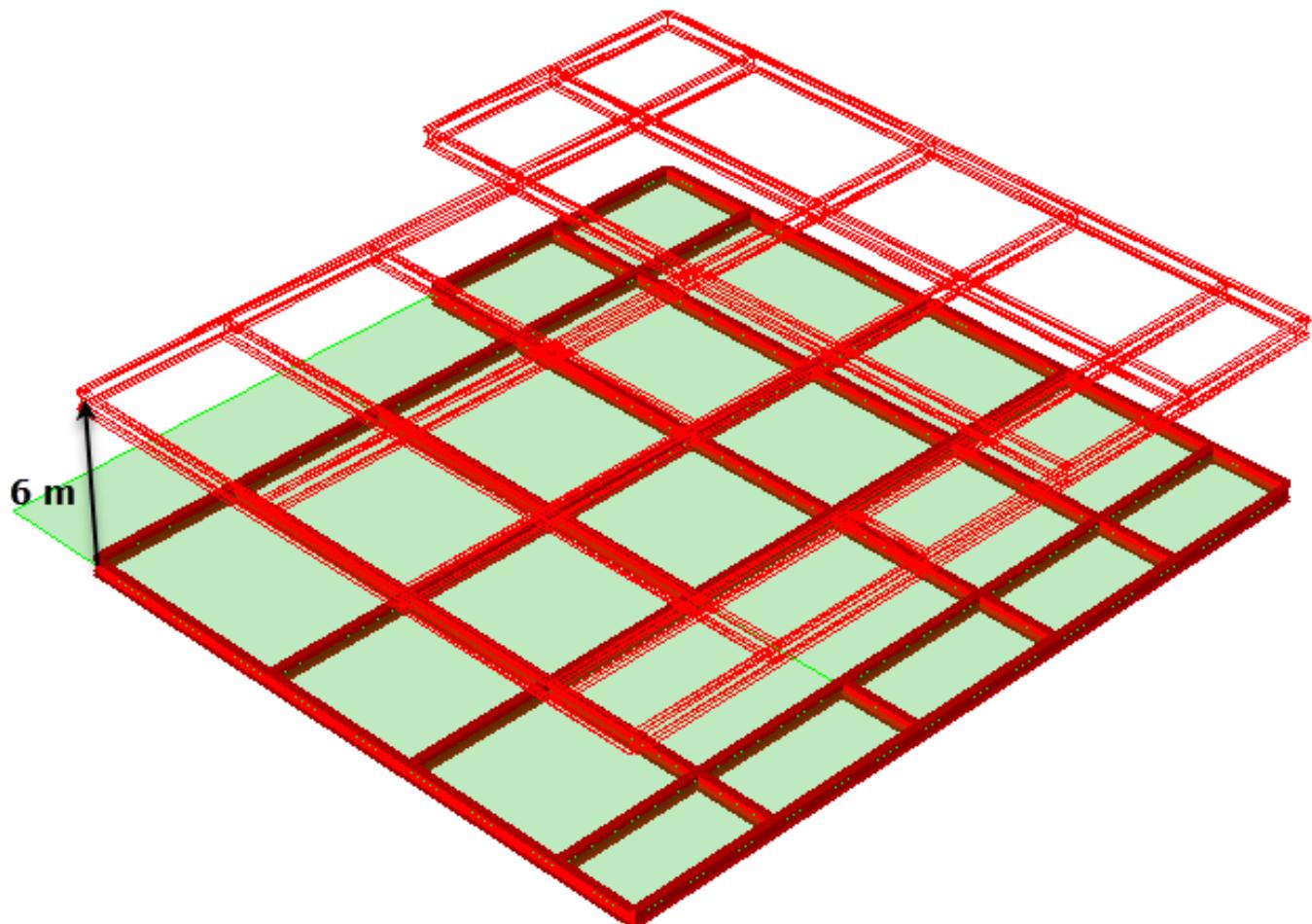
- Press the Esc key to leave insert/create mode and return to select mode.

- Create a set containing the beams of the cellar deck by dragging a rubberband around the beams (avoid selecting the guide plane), right-clicking to select *Named set* and give the name *Cellar\_deck* in the dialog to the right.

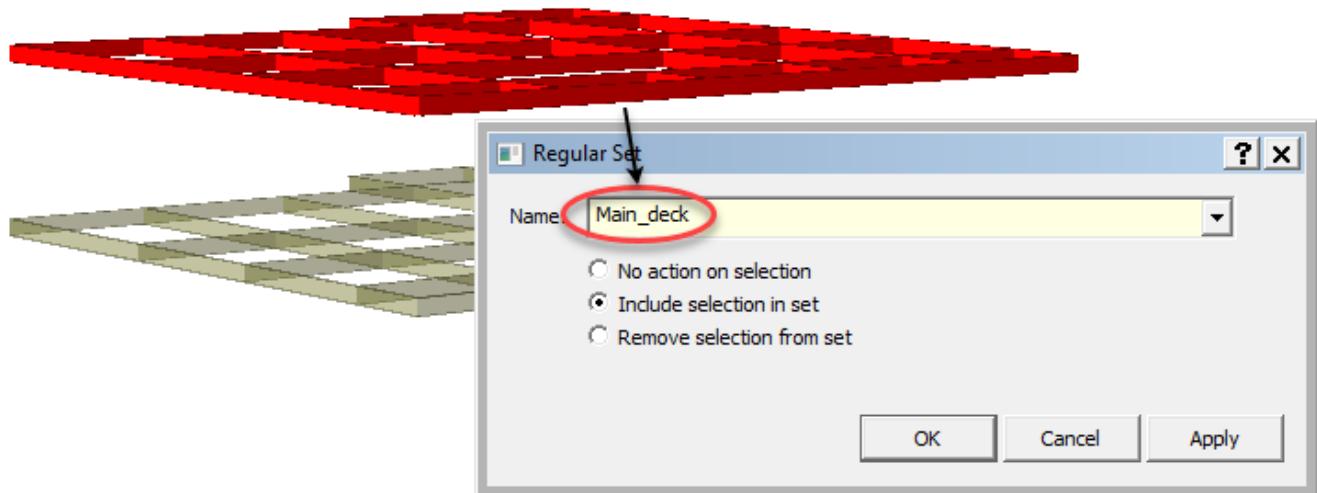


## 7 MAIN DECK

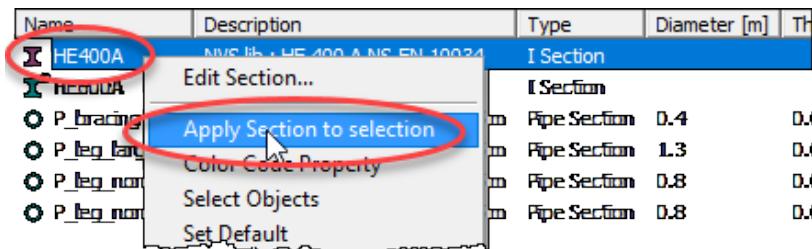
- Copy the cellar deck 6 m upwards to create the main deck.



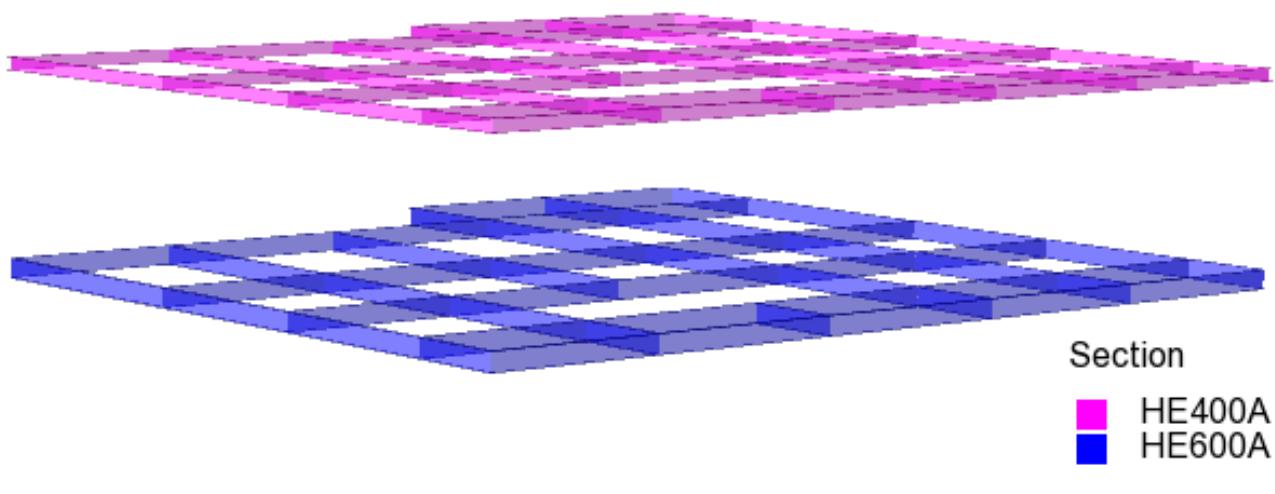
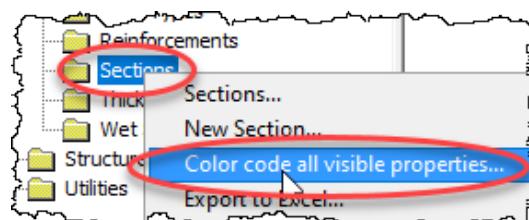
- Switch to display configuration *Modelling - Transparent*.  
➤ Deselect the cellar deck and select the main deck to create a set named `Main_deck`.



- Change the beam cross section in the main deck to HE400A. Ensure all beams of the main deck are selected and:



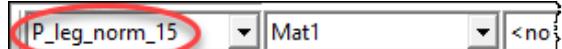
- Verify the beam cross sections by right-clicking *Properties | Sections* in the browser and selecting *Color code all visible properties*:



- Switch off colour coding by lifting the *Property color coding of all visible properties* (paintbrush) button:



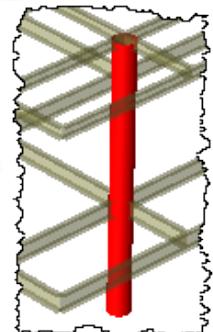
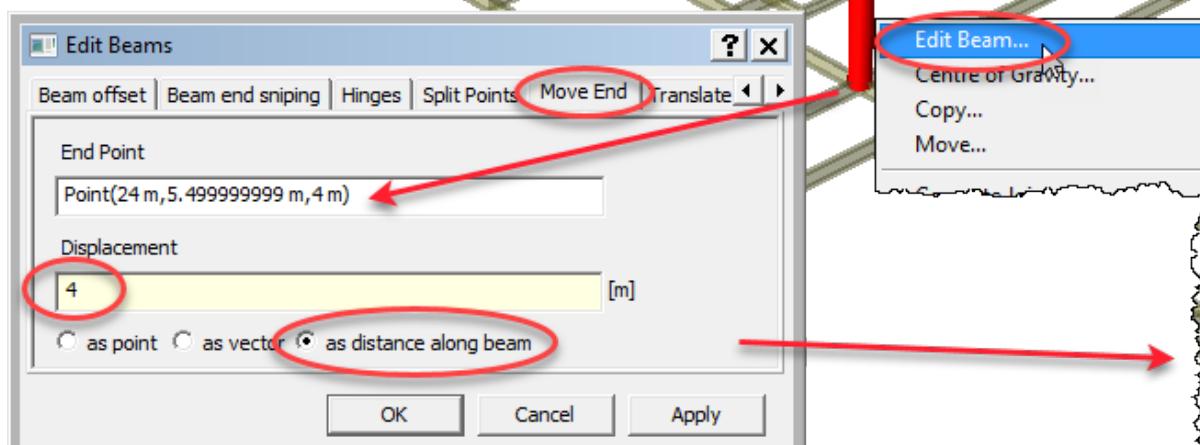
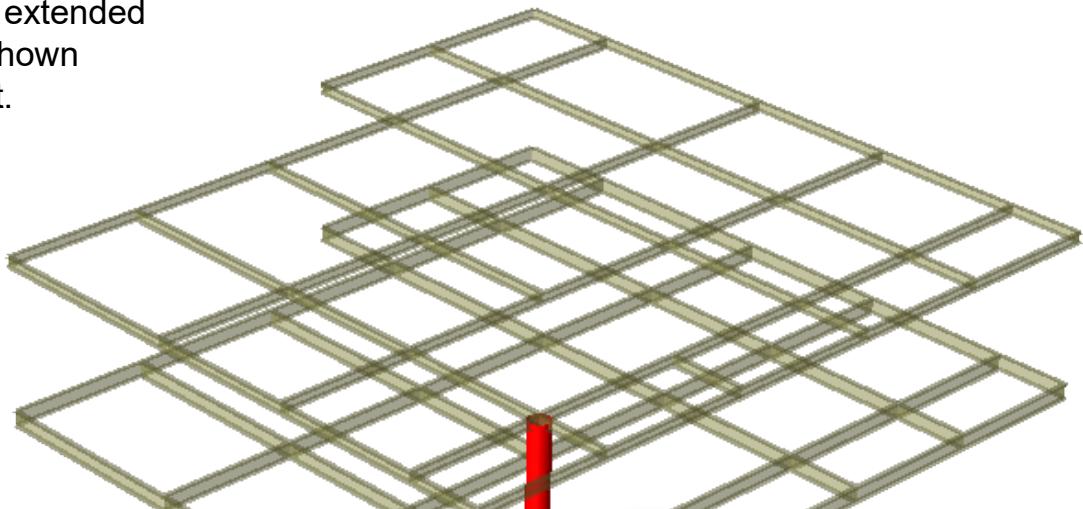
## 8 COLUMNS

➤ Switch default section to P\_leg\_norm\_15: 

- Note that this can also be done by right-clicking in the *Properties | Sections* folder in the browser and that in either case a small check mark in the browser indicates which section is currently default:

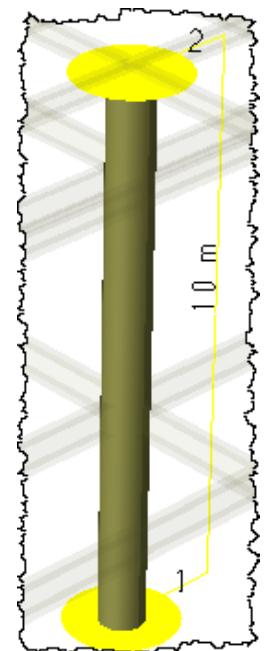
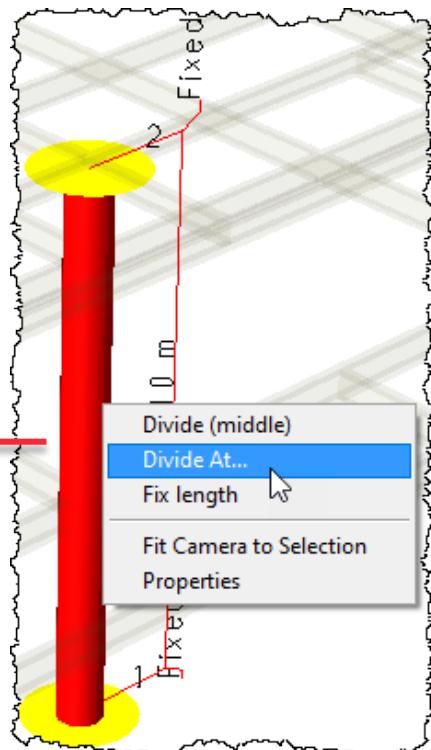
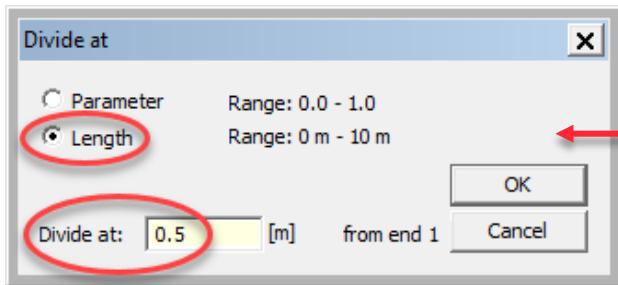
Name	Description	Type	Diam
HE400A	NVS lib : HE 400 A NS-EN 10034	I Section	
HE600A	NVS lib : HE 600 A NS-EN 10034	I Section	
P_bracing	Pipe Section, d=0.4 m, t=0.015 m	Pipe Section	0.4
P_leg_large_35	Pipe Section, d=1.3 m, t=0.035 m	Pipe Section	1.3
<b>P_leg_norm_15</b>	Pipe Section, d=0.8 m, t=0.015 m	Pipe Section	0.8
P_leg_norm_35	Pipe Section, d=0.8 m, t=0.035 m	Pipe Section	0.8

➤ Create a column as shown. Thereafter extend it downwards as follows. Select it, right-click and select *Edit Beam*. In the *Edit Beams* dialog go to the *Move End* tab. Click the lower end of the column to fill the *End Point* field with the lower end coordinates. Check *as distance along beam* and fill in 4 m. Click *OK* and the beam is extended downwards as shown to the lower right.

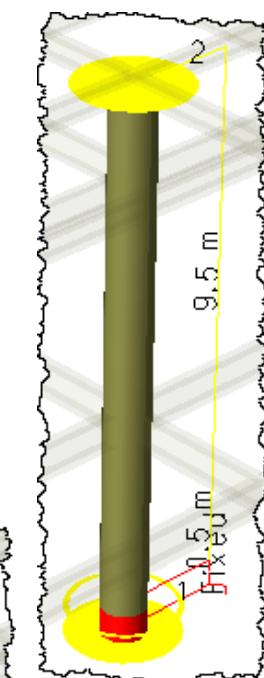
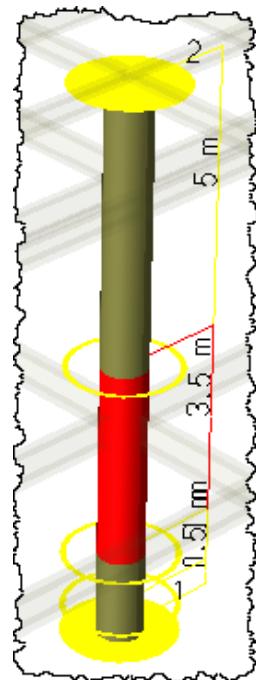
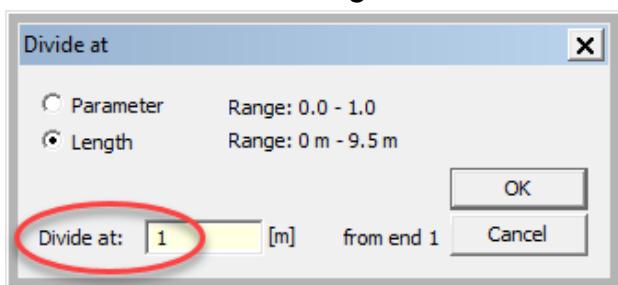


- Currently the column has pipe section P\_leg\_norm\_15. However, it shall have three different pipe sections plus a cone transition between two sections along its length. This is achieved by dividing the column into segments. This is done as follows.

- Double-click the column to enter segmenting mode as shown to the right.
- Select it, right-click and click *Divide At* as shown to the right.
- In the *Divide at* dialog shown below select *Length* and give the value 0.5.

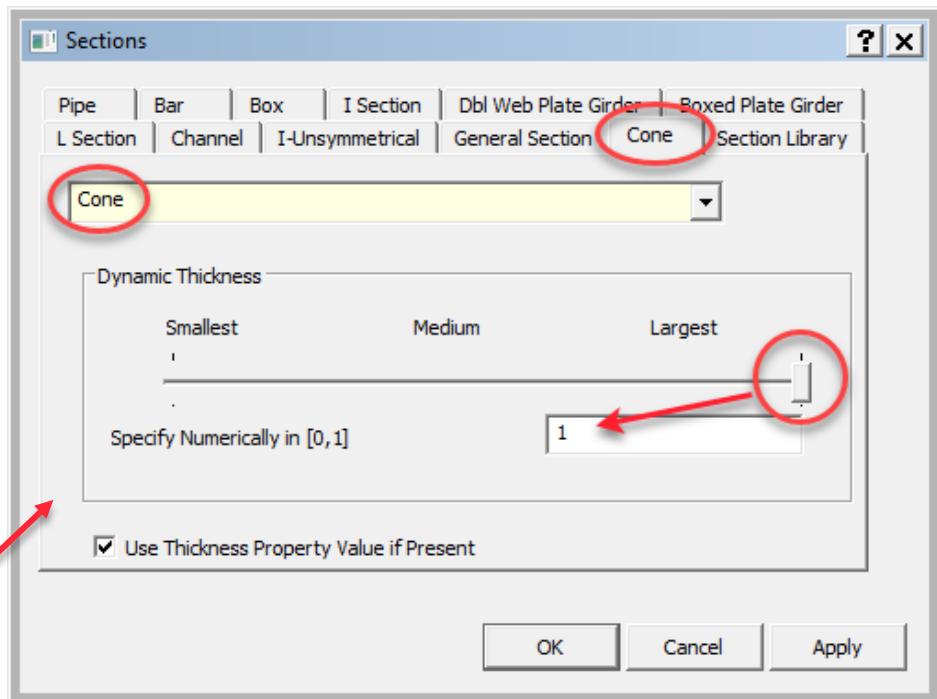
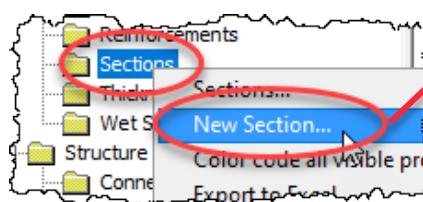


- The 10 m long column has now been divided into two segments of lengths 0.5 m and 9.5 m as shown to the right.
- Select the 9.5 m long segment and divide this further into two segments. Give *Divide at* length of 1 m which is measured from the lower end (provided that the column when initially created was created clicking the lower end first).



- Finally, divide the remaining upper 8.5 m long segment into two: 3.5 m and 5 m long segments.. The end result is shown to the right.

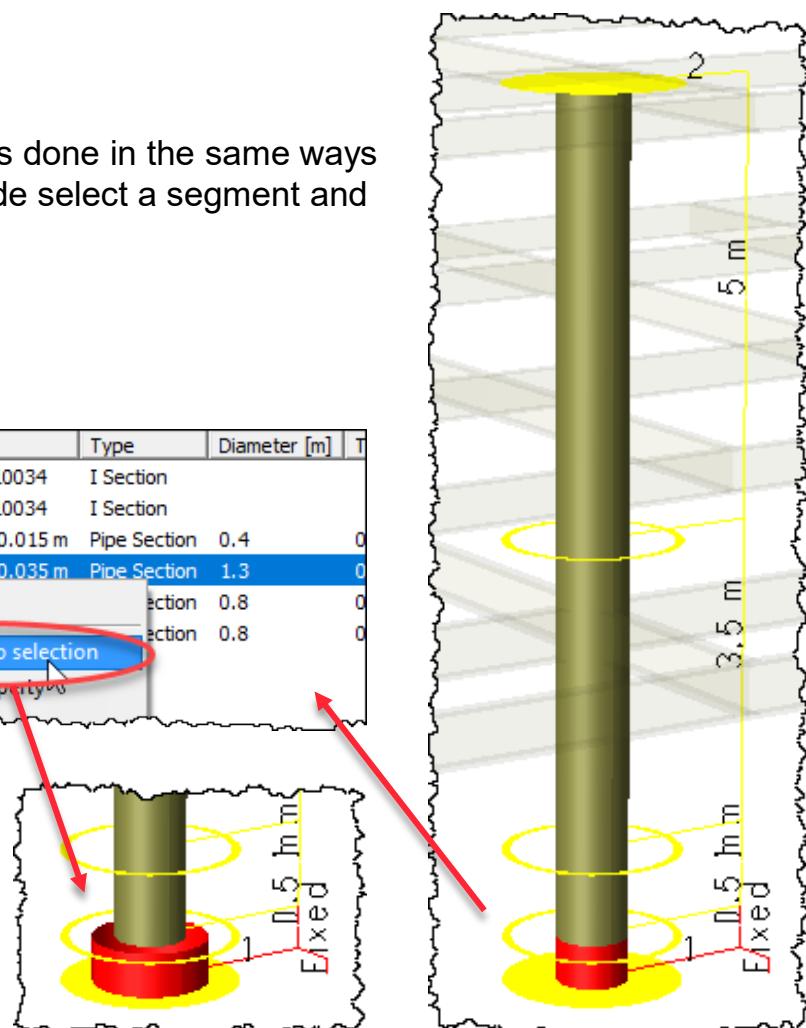
- The purpose of dividing the column into segments is to assign various beam cross sections to it. Three different pipe sections to be used have already been defined. One of the pipe sections (P\_leg\_large\_35) have larger diameter and need to be connected with the other pipe sections by a cone section. This must be defined.



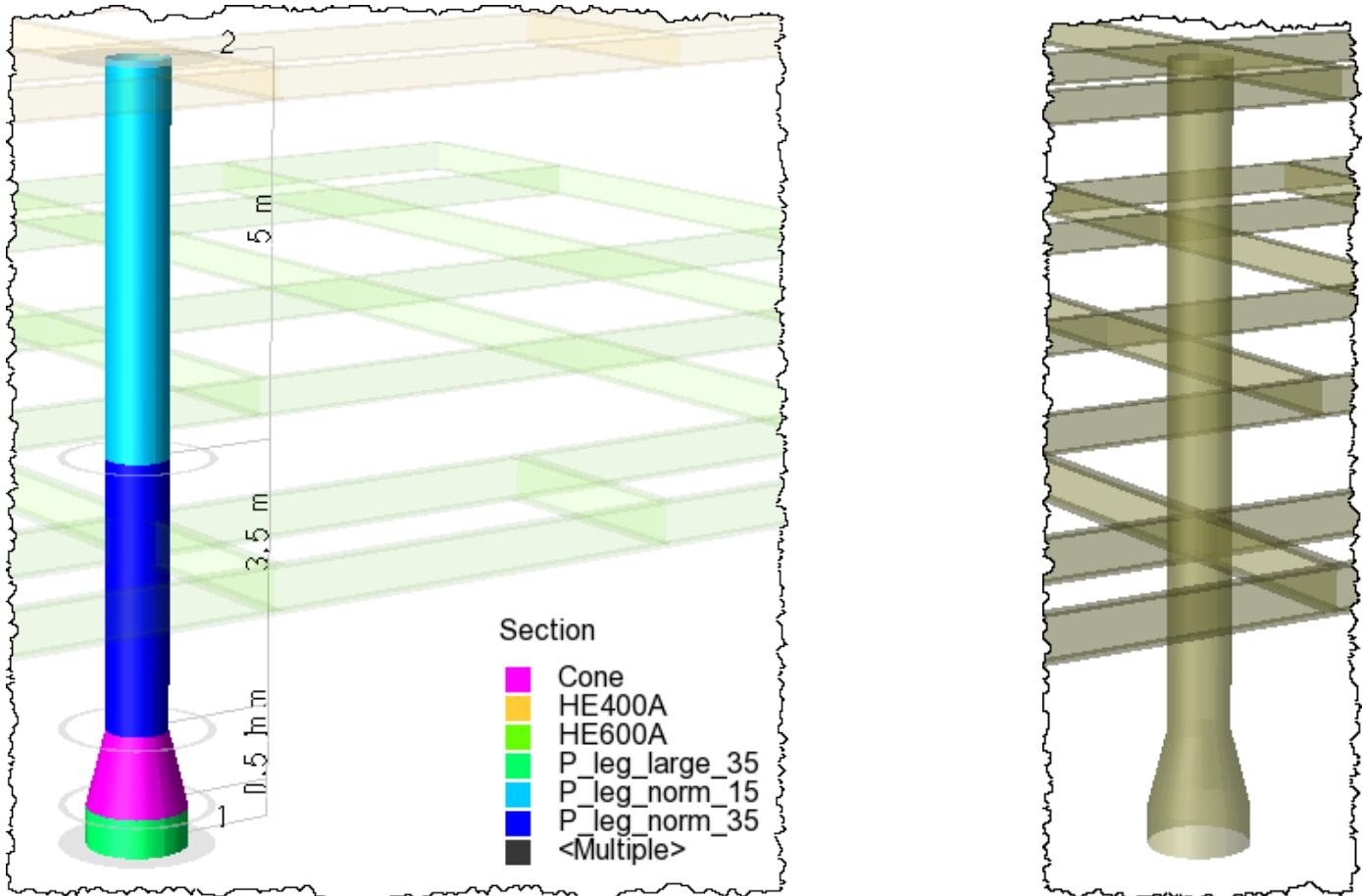
- Assigning sections to segments is done in the same ways as for beams. In segmenting mode select a segment and apply the proper section to it.

Name	Description	Type	Diameter [m]	T
HE400A	NVS lib : HE 400 A NS-EN 10034	I Section		
HE600A	NVS lib : HE 600 A NS-EN 10034	I Section		
P_brauing	Pipe Section, d=0.4 m, t=0.015 m	Pipe Section	0.4	0
P_leg_large_35	Pipe Section, d=1.3 m, t=0.035 m	Pipe Section	1.3	0
P_leg_norm_15	Pipe	Edit Section...	0.8	0
P_leg_norm_35	Pipe	Edit Section...	0.8	0

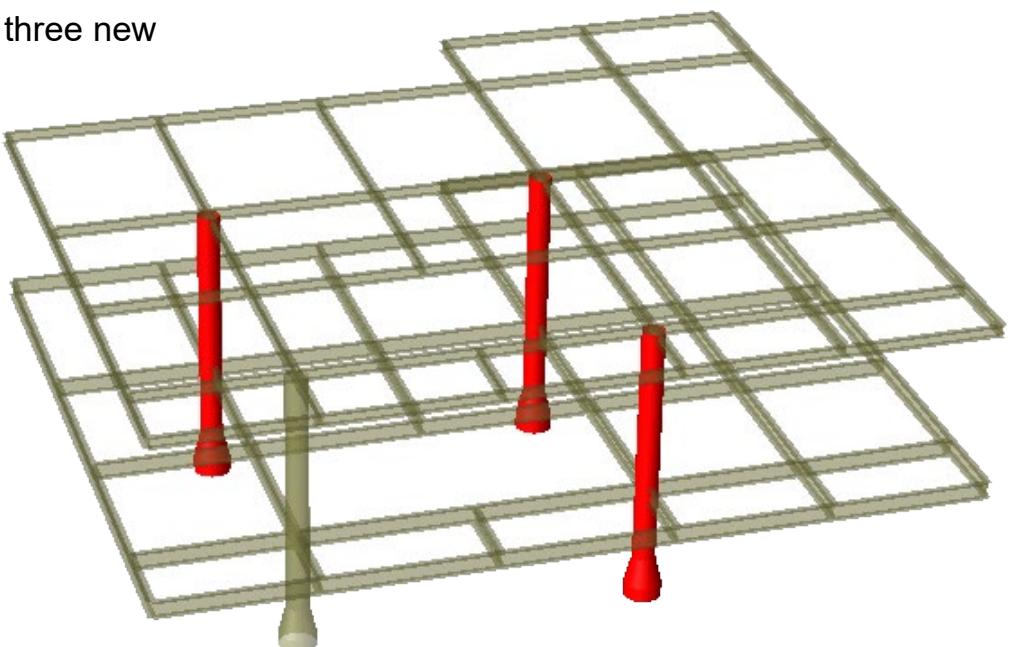
Apply Section to selection



- Assign beam cross sections to achieve the result shown below. That is, from bottom, the sections are P\_leg\_large\_35, Cone, P\_leg\_norm\_35, P\_leg\_norm\_15.
- Double-click anywhere to leave segmenting mode and switch off colouring.



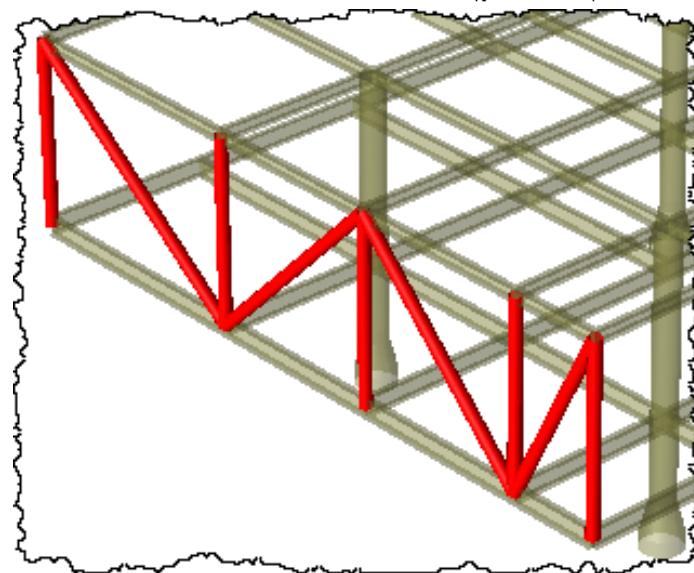
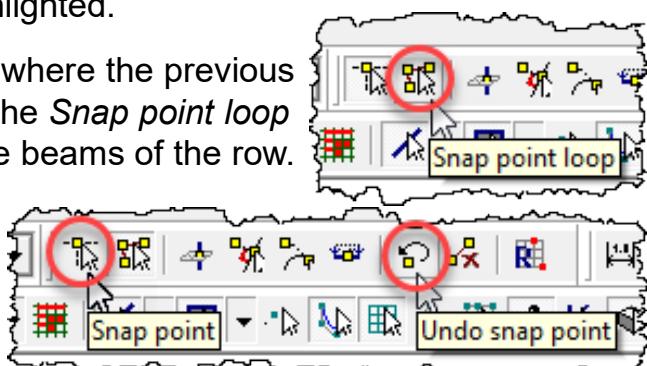
- Copy the column to three new positions as shown.  
The three copies are highlighted.



## 9 DECK ROWS 1 TO 6 IN XZ-PLANE

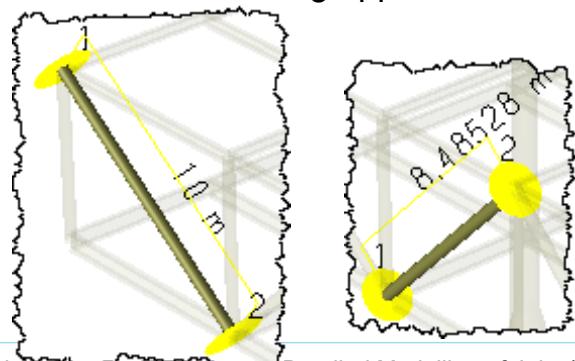
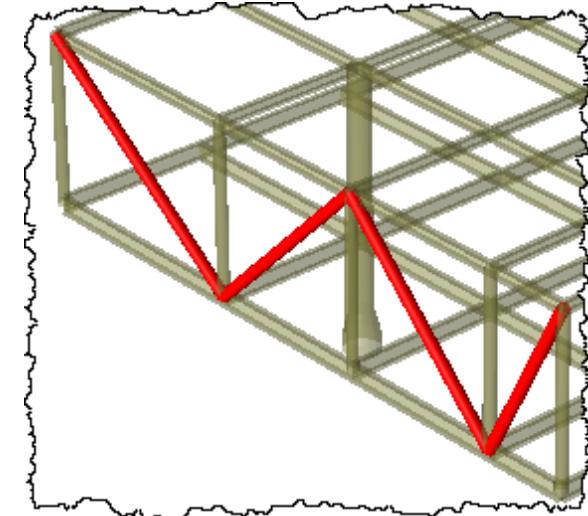
- Set default beam cross section to P\_bracing and create the beams of deck row 1 shown below. The beams to create are highlighted.

- Note that a row of beams (a beam starts where the previous ends) is most efficiently created clicking the *Snap point loop* button. Use this procedure for most of the beams of the row.
- To return to normal creation mode, press the *Undo snap point* button (to allow clicking the first end of a beam) and the *Snap point* button.

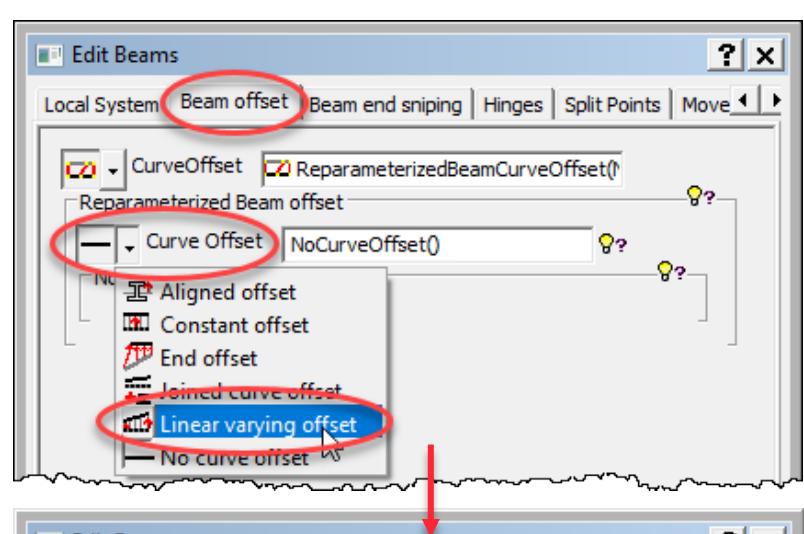
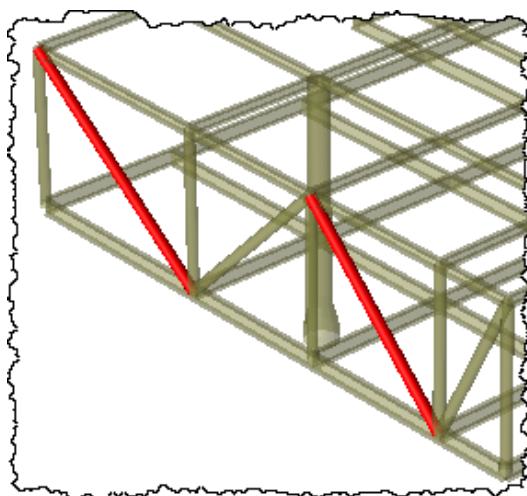


- The diagonal bracings highlighted to the right shall be given vertical offsets (eccentricities) in their lower and upper ends.

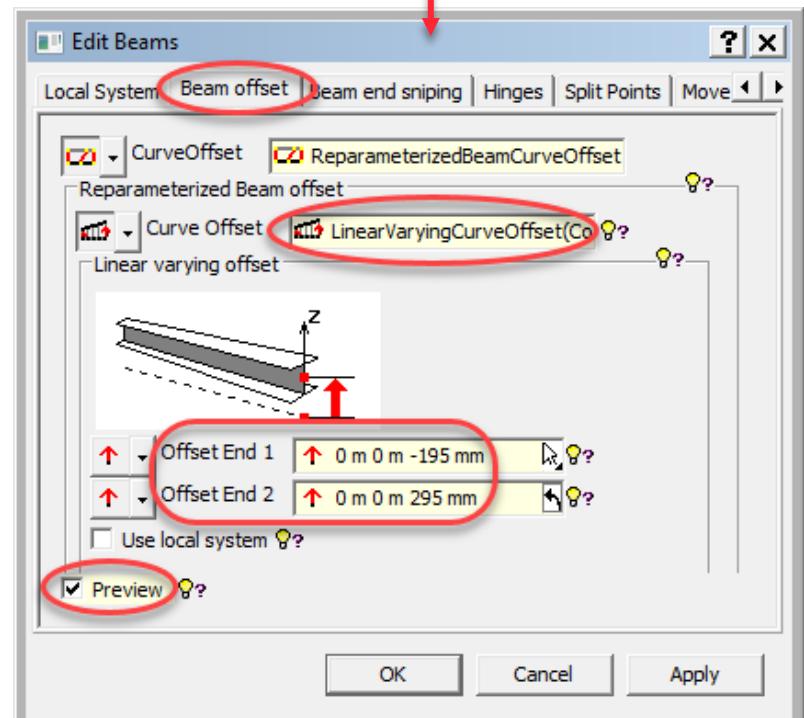
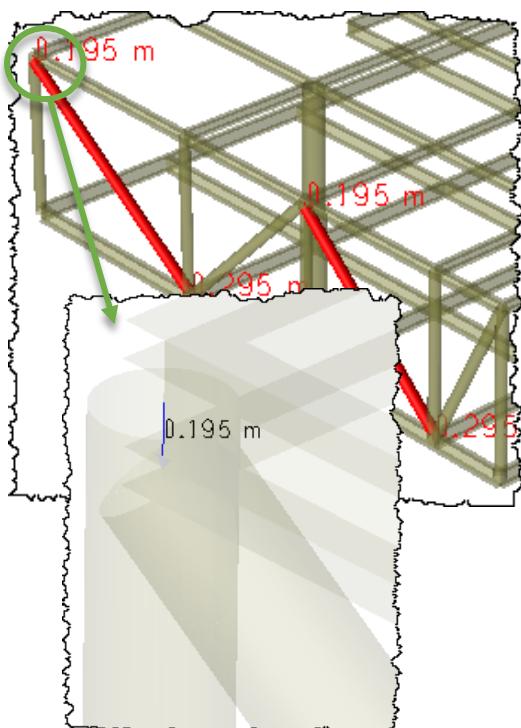
- However, depending on the order of clicking when creating the diagonal bracings their ends 1 and 2 (to which offsets are assigned) may both be lower and upper ends. This is seen when double-clicking a couple of the diagonal bracings. Notice that end 1 alternates between being upper and lower.



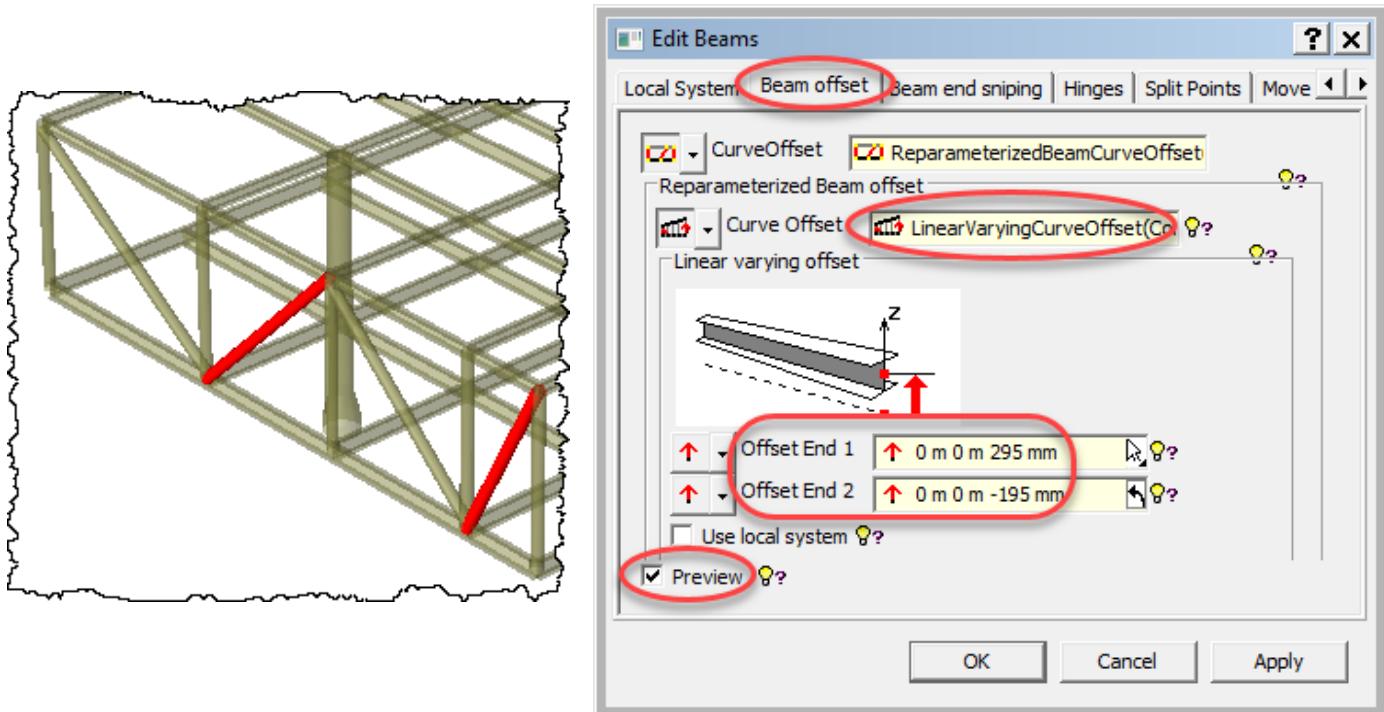
- Therefore, the assignment of offsets must be done in two operations.
- Select the diagonal bracings with their end 1 being the upper end, right-click and select *Edit Beam*.
- In the *Edit Beams* dialog go to the *Beam offset* tab and set *Curve Offset* to *Linear varying offset*.
- The lower vertical offset shall be 295 mm (half the height of the I section of the cellar deck, HE600A) and the upper vertical offset shall be -195 mm (half the height of the I section of the main deck, HE400A).



- Label the eccentricities to verify the result. Zoom in to verify further.

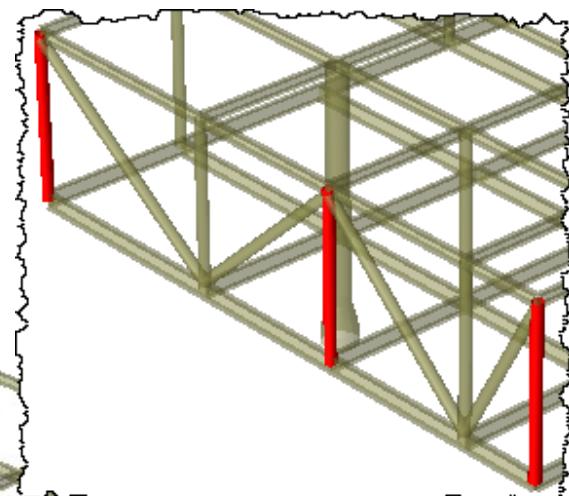
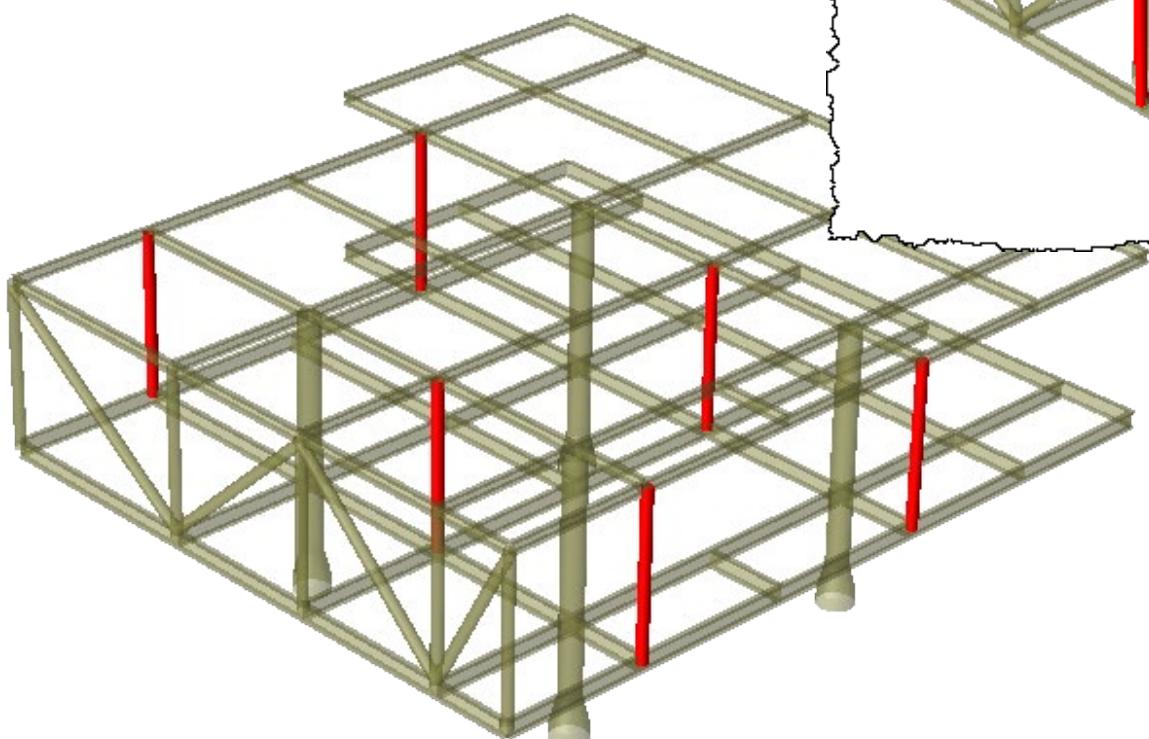


- Repeat the process for the diagonal bracings with their end 1 being the lower end.



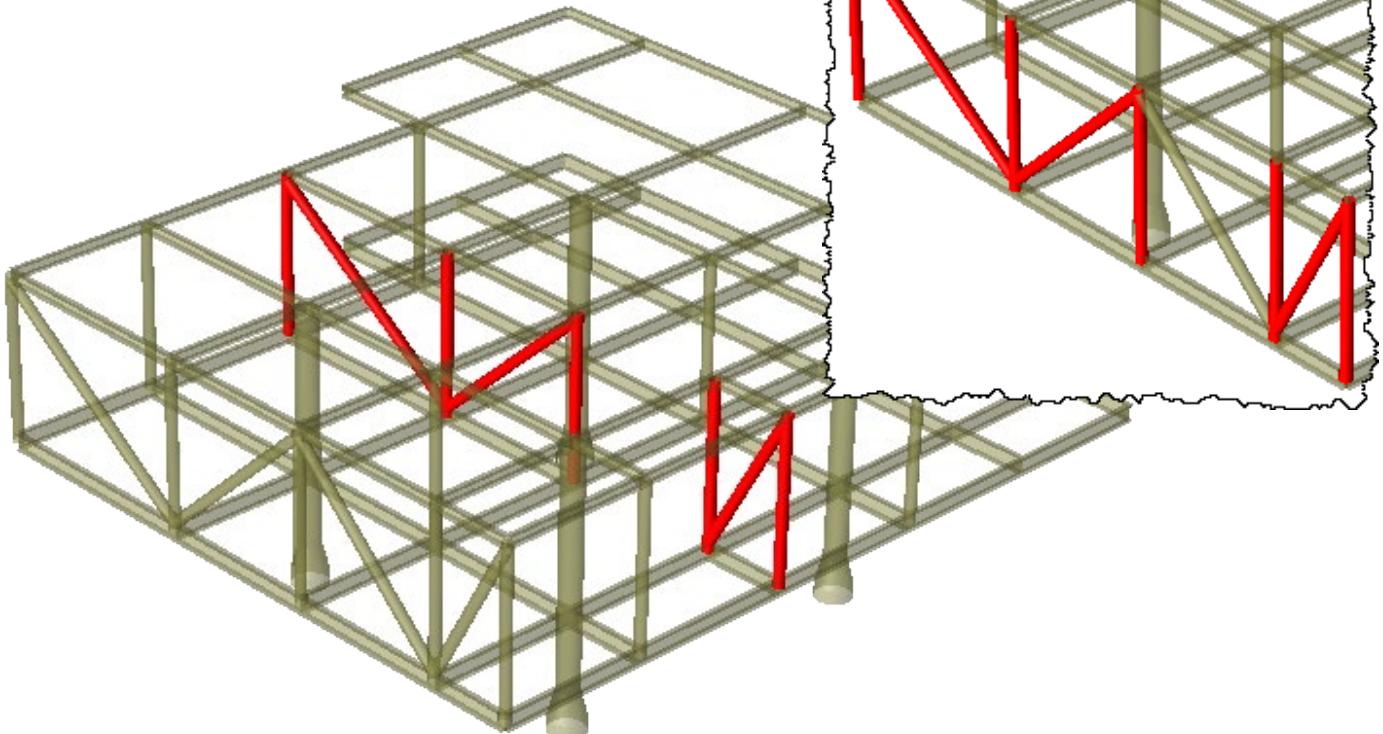
- Copy selected vertical bracings in Y-direction to rows 2 and 4.

- The beams to copy are shown to the right and the copies are highlighted below.

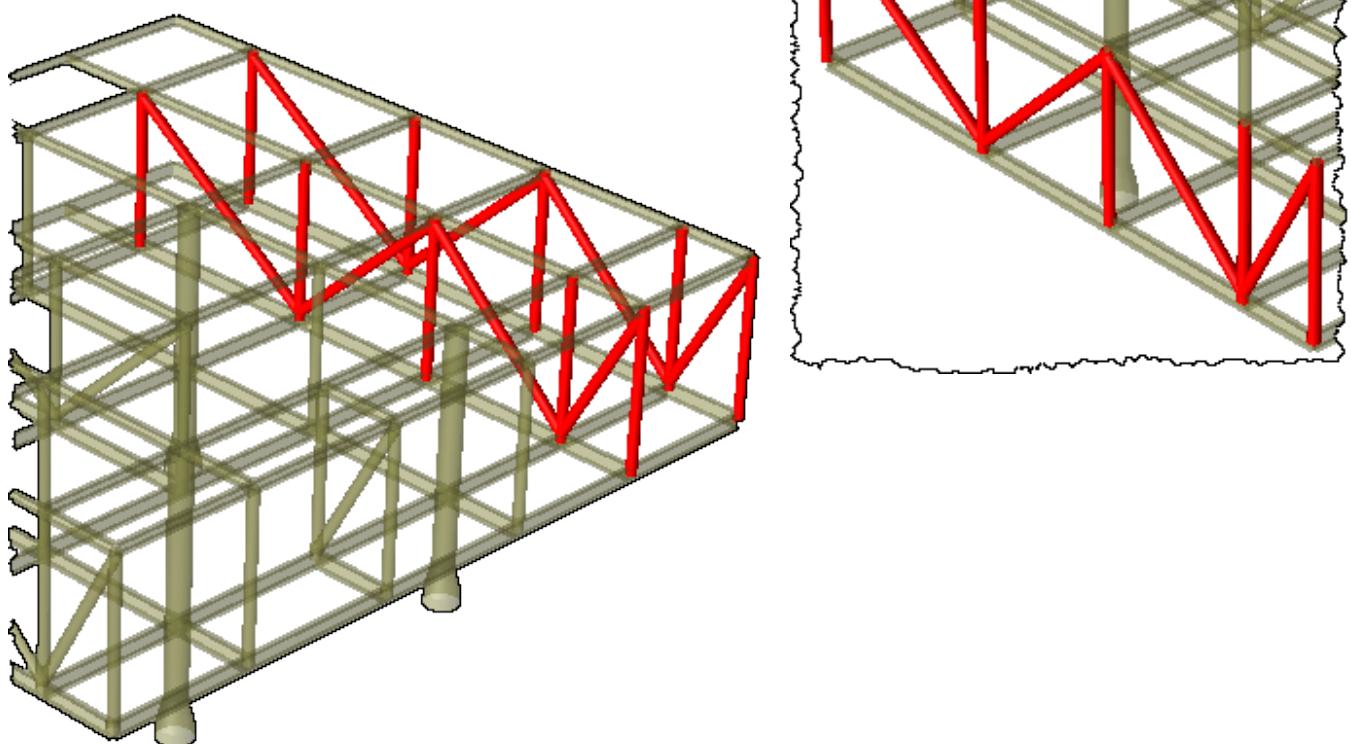


- Copy selected vertical and diagonal bracings in Y-direction to row 3.

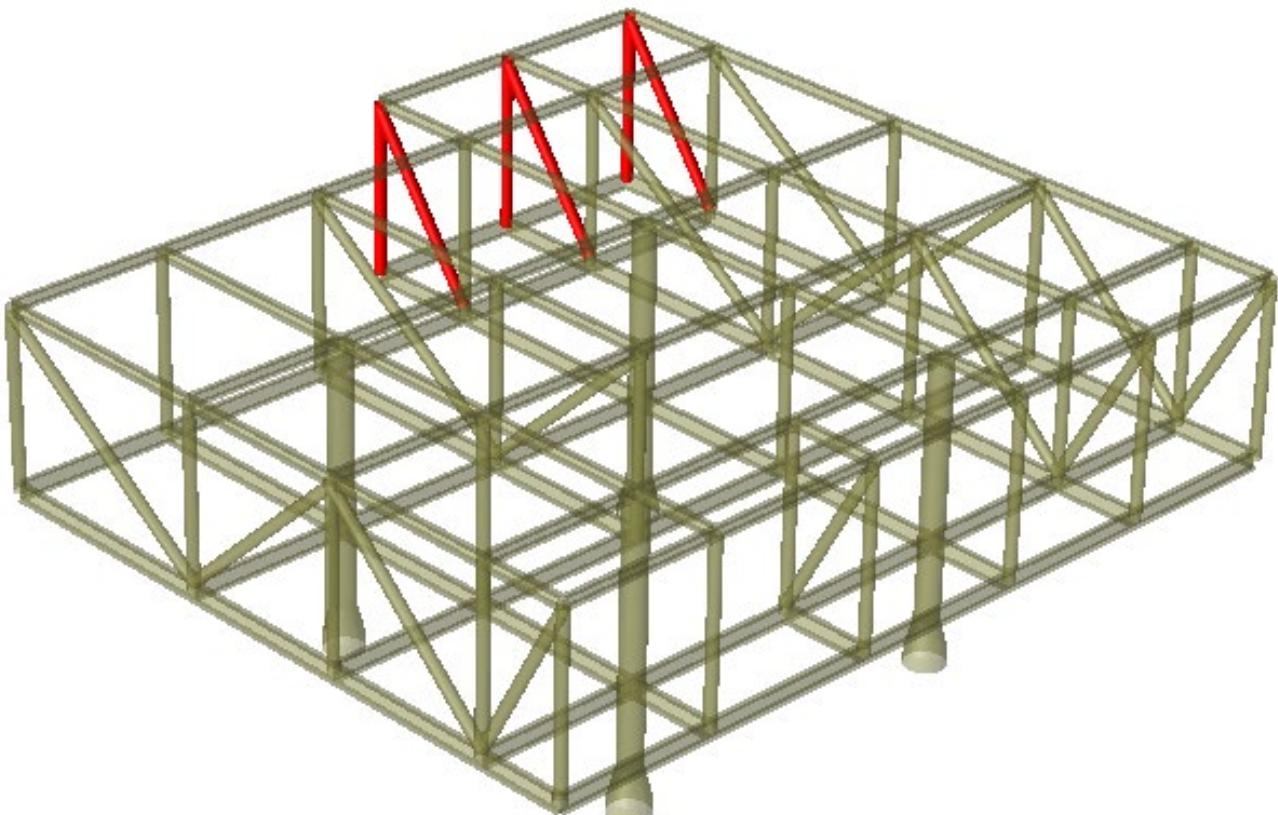
- The beams to copy are shown to the right and the copies are highlighted below.



- Copy beams shown to the right to rows 5 and 6. The copies are highlighted below.

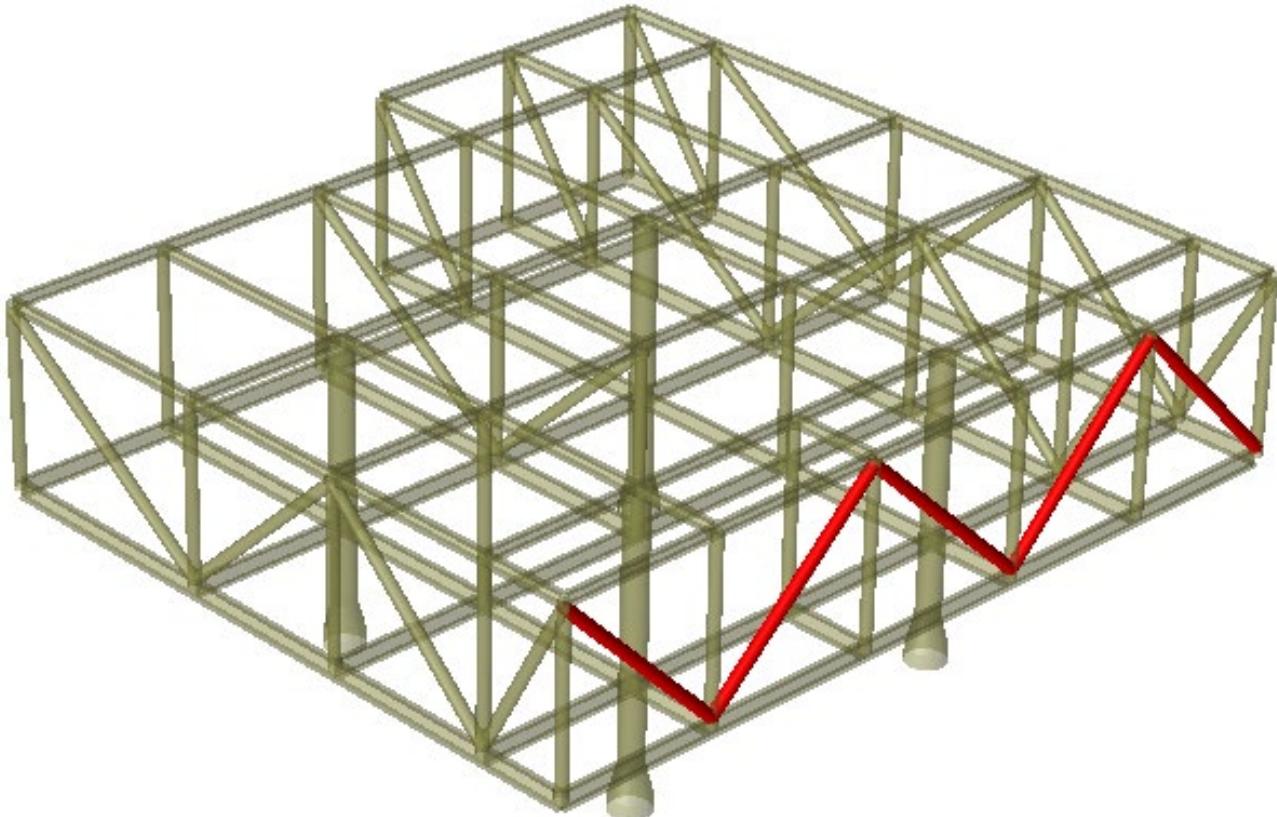


- Add vertical and diagonal bracings to the protruding part of rows 5 and 6.
  - The diagonal bracings shall, as before, have lower vertical offset of 295 mm and upper vertical offset of -195 mm.

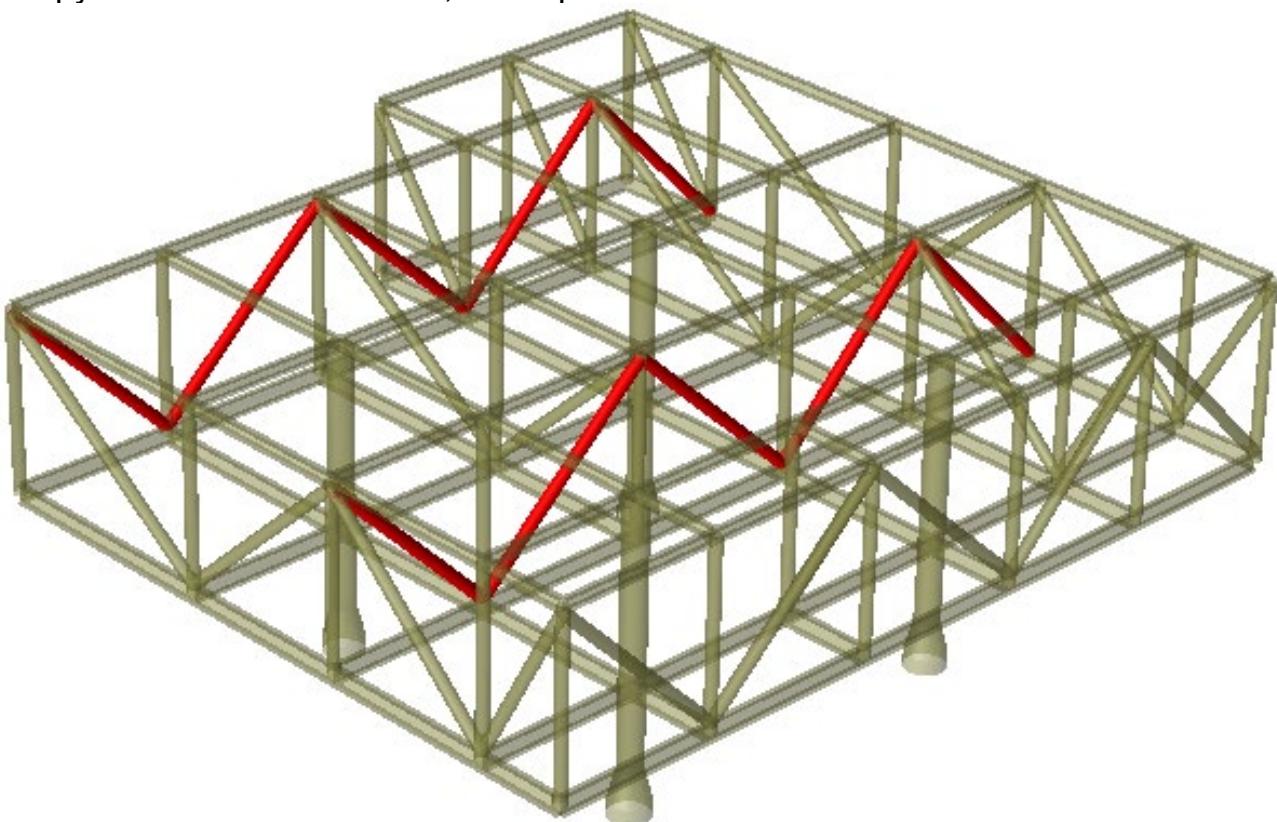


## 10 DECK ROWS A TO E IN YZ-PLANE

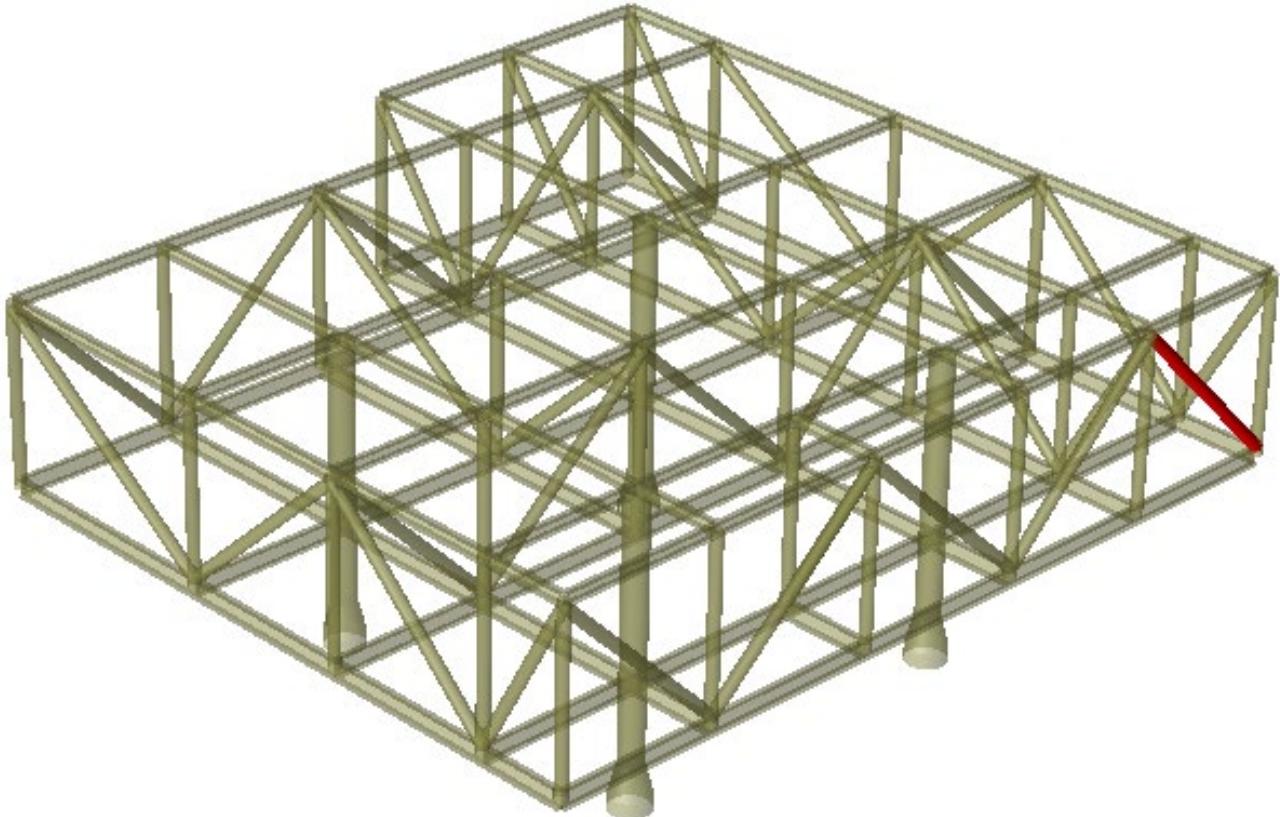
- Create the highlighted beams of deck row A with the same beam cross sections and offsets as for row 1.



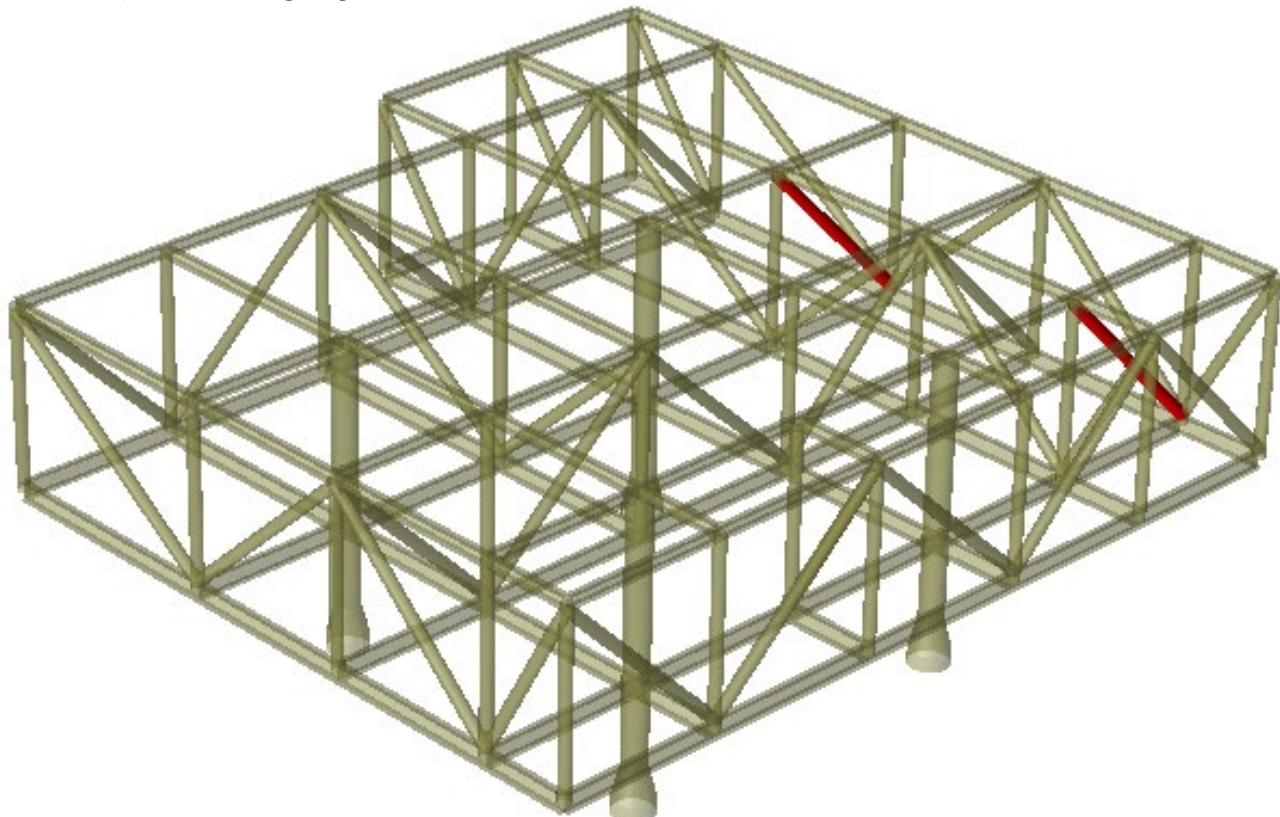
- Copy these to rows C and E, the copies shown below.



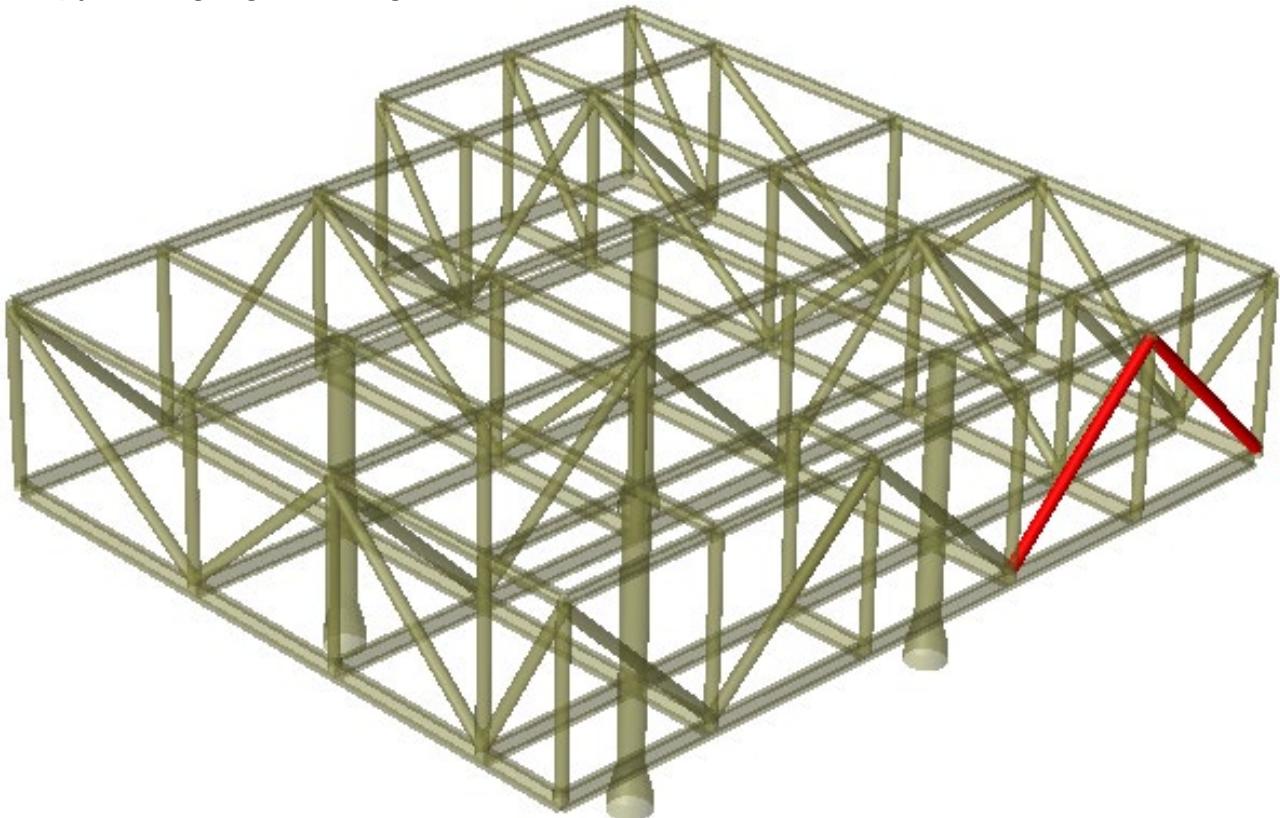
- Copy the highlighted diagonal to rows B and D.



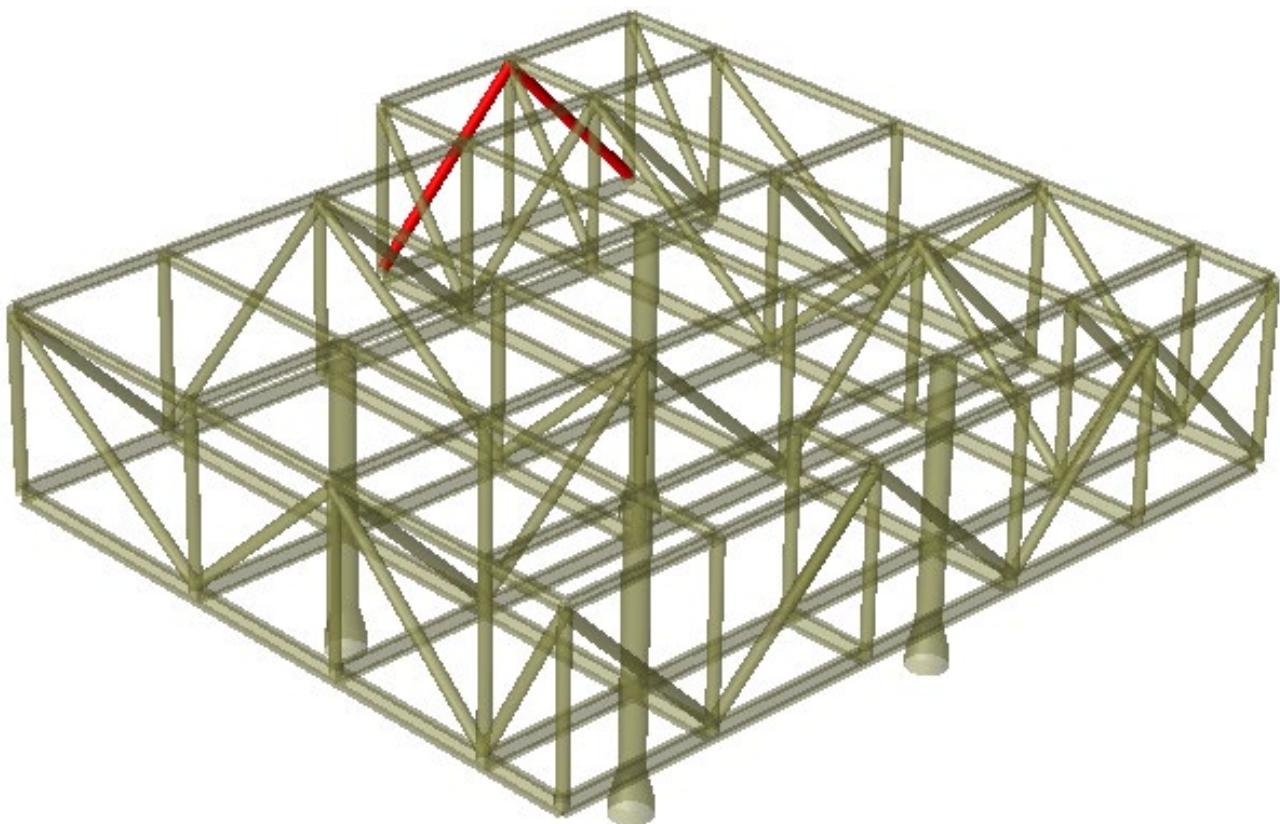
- The copies are highlighted below.



- Copy the highlighted diagonals to row F.



- The copies are highlighted below.



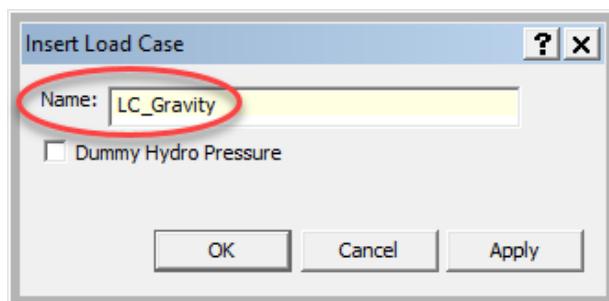
## 11 LOADS

➤ The deck is subjected to three loads:

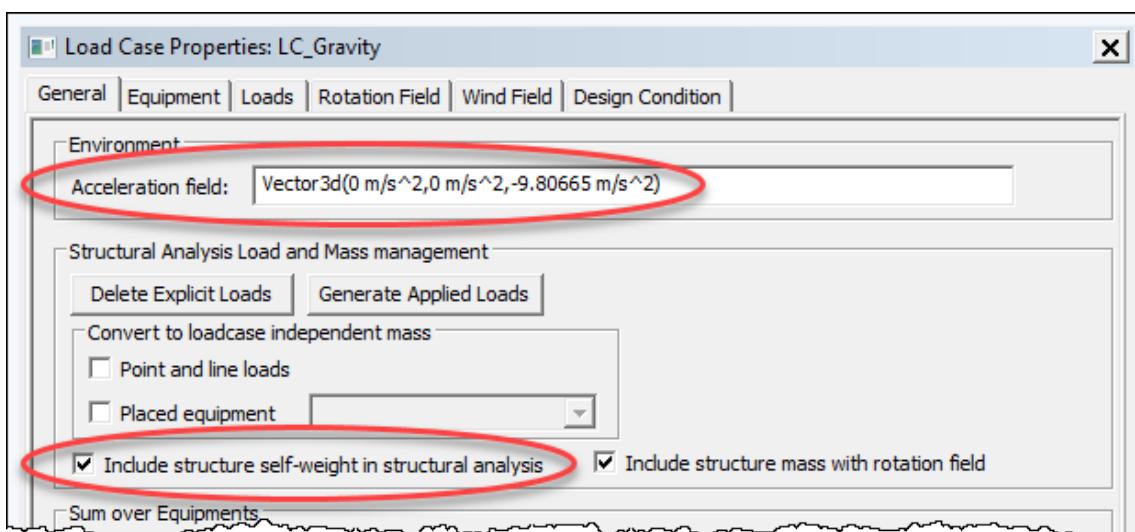
1. Gravity (self weight)
2. Helideck modelled as an equipment
3. Flare tower modelled as an equipment

➤ Gravity load case

- Use *Loads | Load Case*, or right-click the *Analysis | Load Cases* folder in the browser and click *New Loadcase*, to open the *Insert Load Case* dialog. Give the name LC\_Gravity.

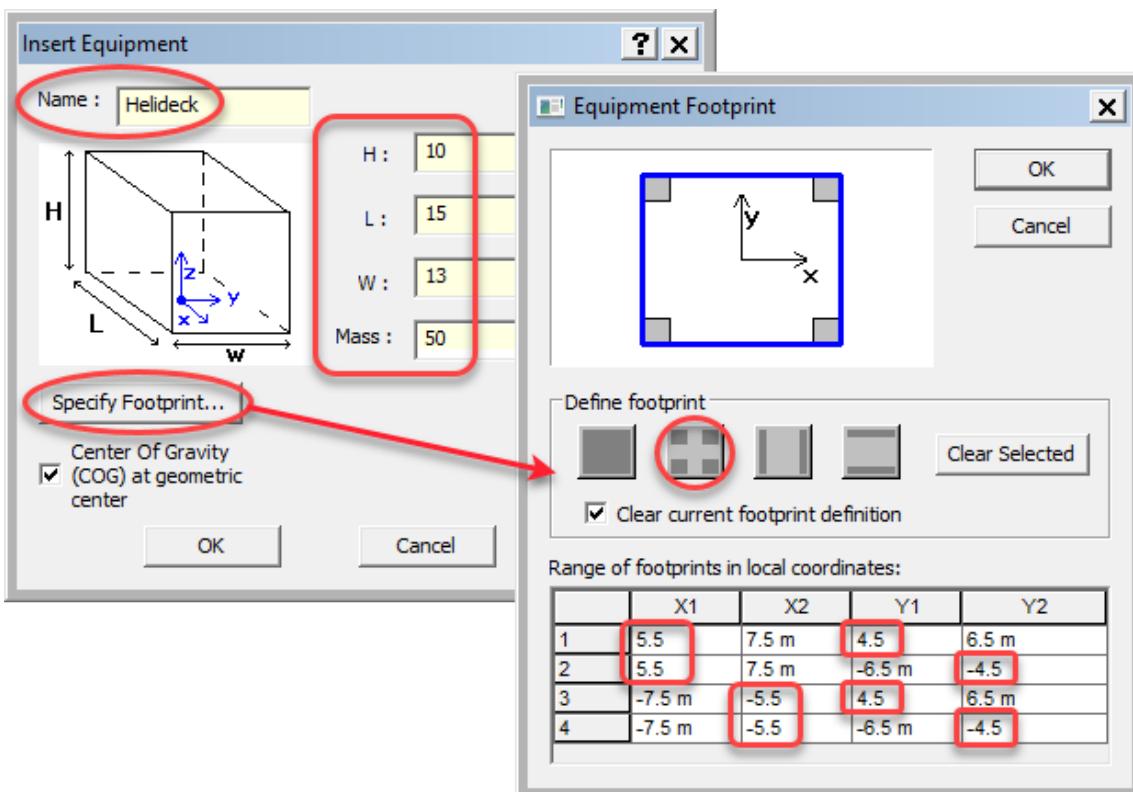


- Select the load case in the browser, right-click and select *Properties*. In the dialog check *Include structure self-weight in structural analysis* and ensure the *Acceleration field* specifies the acceleration of gravity.

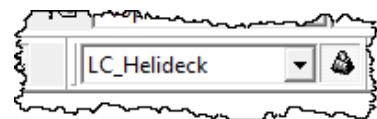


➤ Helideck equipment load case

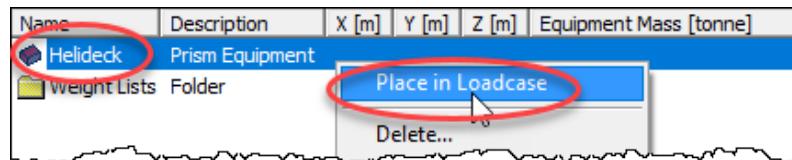
- Use *Loads | Prismatic Equipment*, or right-click the *Equipment* folder in the browser and click *New Equipment*, to open the *Insert Equipment* dialog. Fill in the dimensions for and mass of the equipment as shown below.
- Click the *Specify Footprint* button to change the footprint (load application area) from being the whole bottom surface of the equipment to being four small areas in the corners. Click the button with four shaded corner areas and see that four rows appear in the table, one row for each shaded area. An efficient way of filling in the proper *X1* to *Y2* values for the four rows is to start in the upper left cell, fill in the proper value and use the Tab key to move to the next cell. The cells with values deviating from the default values are encircled.



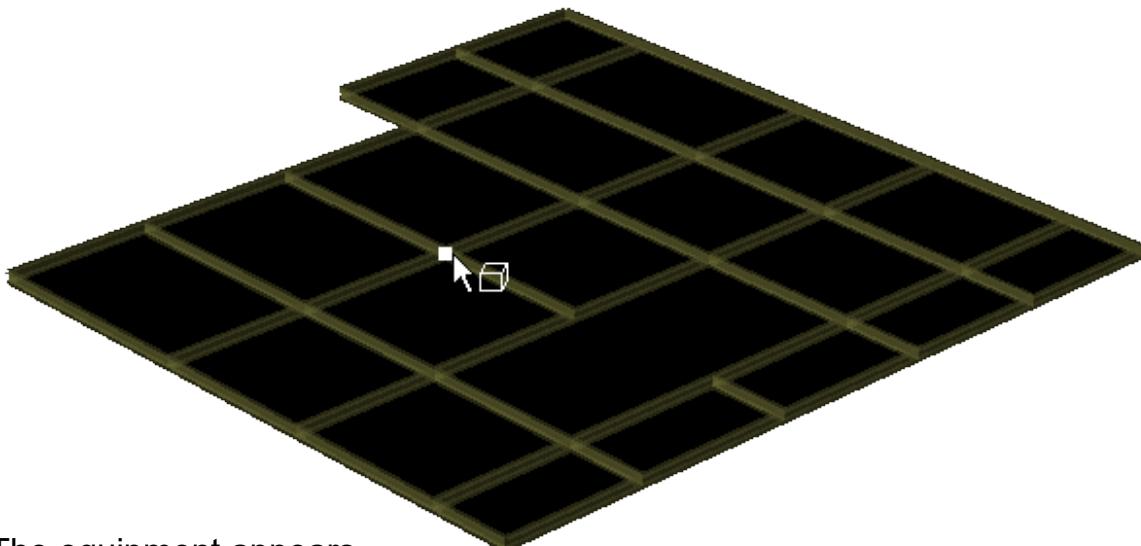
- Click *OK* to close the dialogs and find the equipment listed in the *Equipment* folder in the browser.
- Create a new load case named LC\_Helideck. Creating a new load case makes it the currently selected one as shown in the load case selector.
- To ease placing the equipment in the main deck, display only this deck. Select the set Main\_deck (*Utilities | Sets | Regular Sets* folder) and make only this visible by Alt+S.



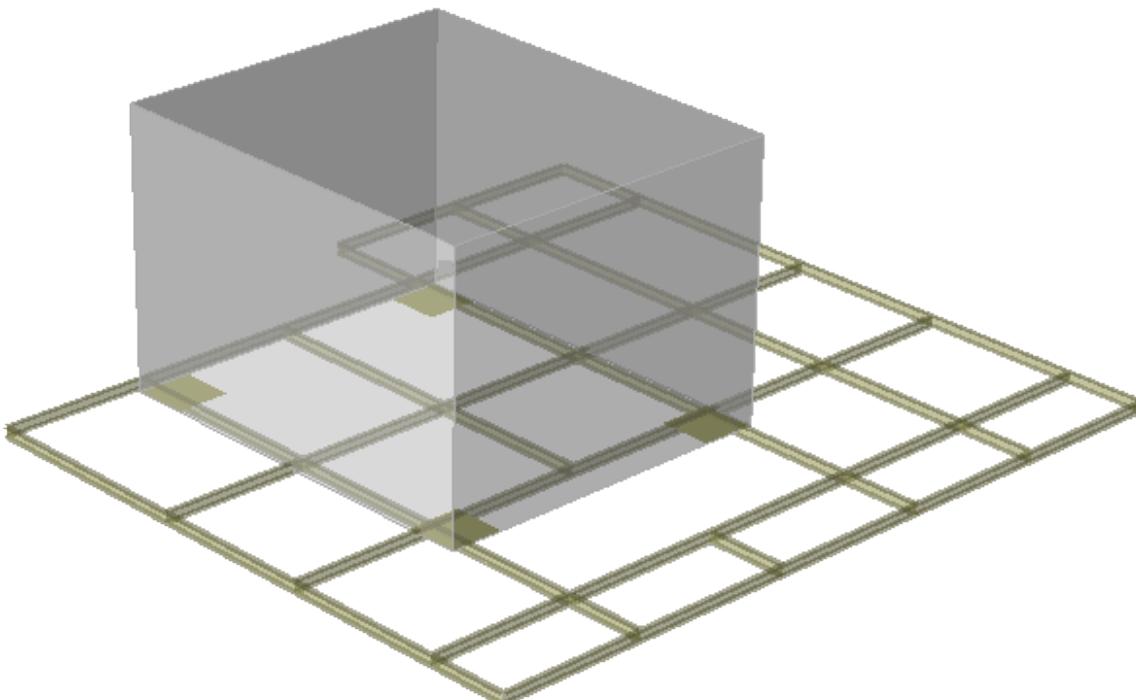
- The equipment Helideck shall now be placed in the model and the current load case. Do so by selecting the equipment in the *Equipment* folder, right-clicking and clicking *Place in Loadcase*.



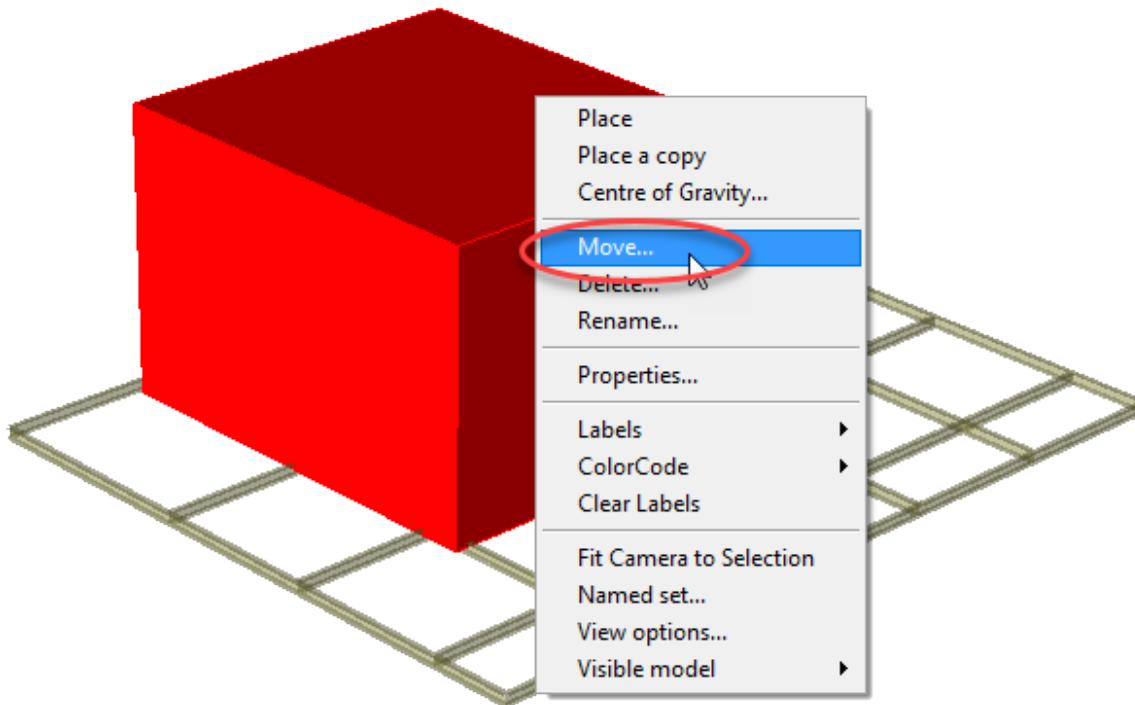
- When moving the pointer over the model notice the small cube indicating that an equipment is in the process of being placed. Click the point as shown below (with black background to make the cube more visible).



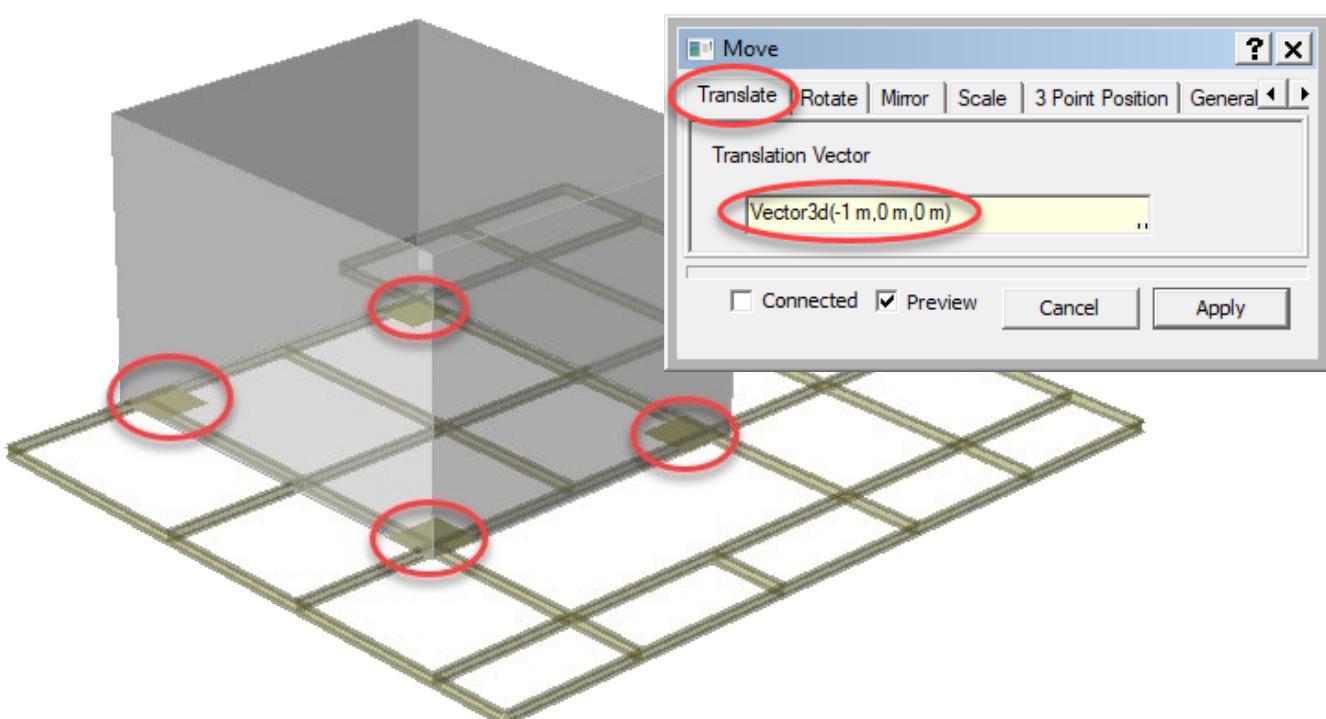
- The equipment appears.



- However, as placing an equipment is restricted to clickable points in the model its horizontal position may not be accurate. Adjust the horizontal position by right-clicking the equipment and clicking *Move*.



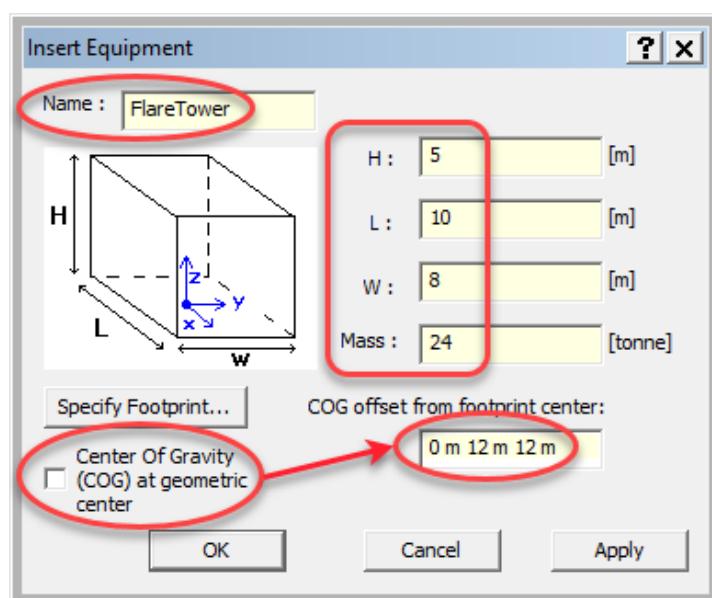
- Move the equipment 1 m in negative X-direction. Notice that the four corner footprint areas now coincide with joints in the main deck.



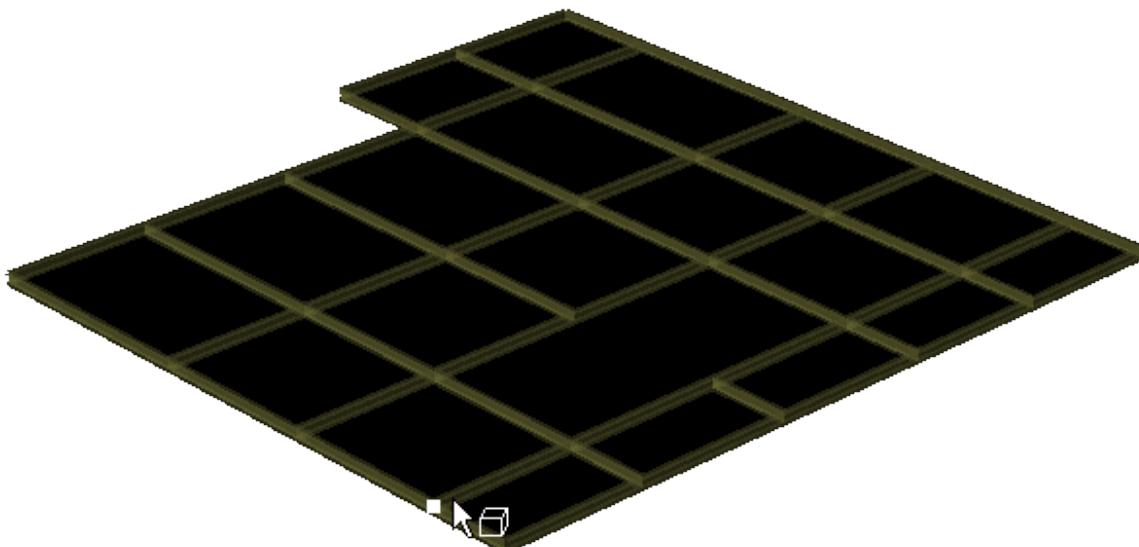
- Equipment loads are automatically computed when doing an analysis.

➤ Flare tower equipment load case

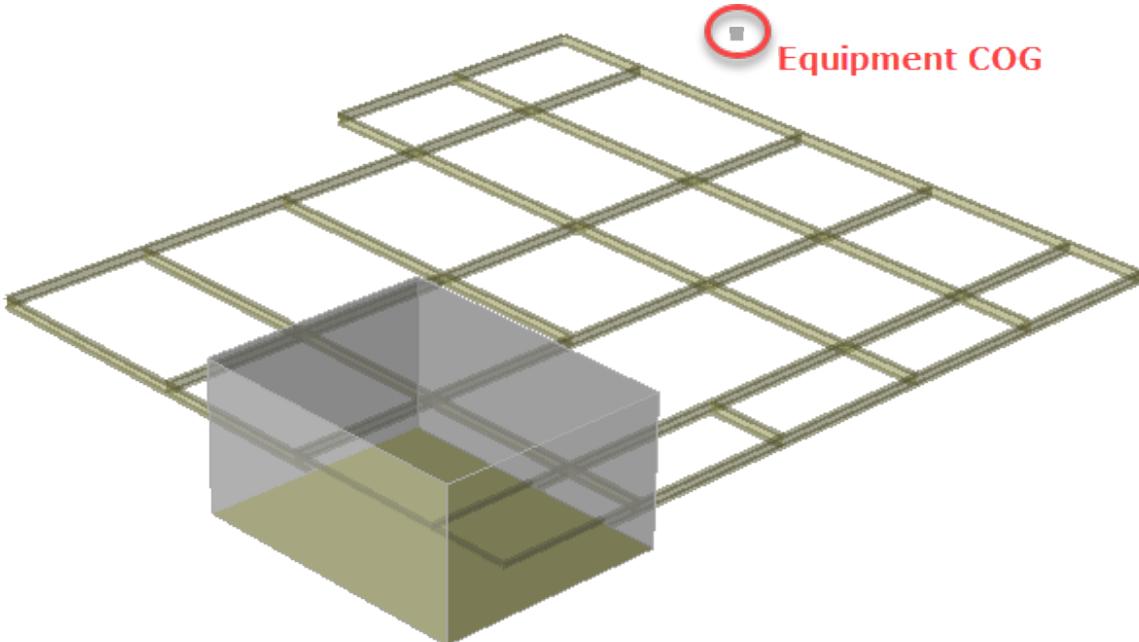
- Create a new equipment named FlareTower with dimensions and mass as shown below.
- Uncheck *Center of Gravity (COG) at geometric center* to allow specifying the COG. Specify (0 m, 12 m, 12 m) which is a point outside the bounds of the equipment dimensions. (Since both footprint for load application and COG may be adjusted the equipment dimensions reduce to a mere graphic illustration of the equipment.)



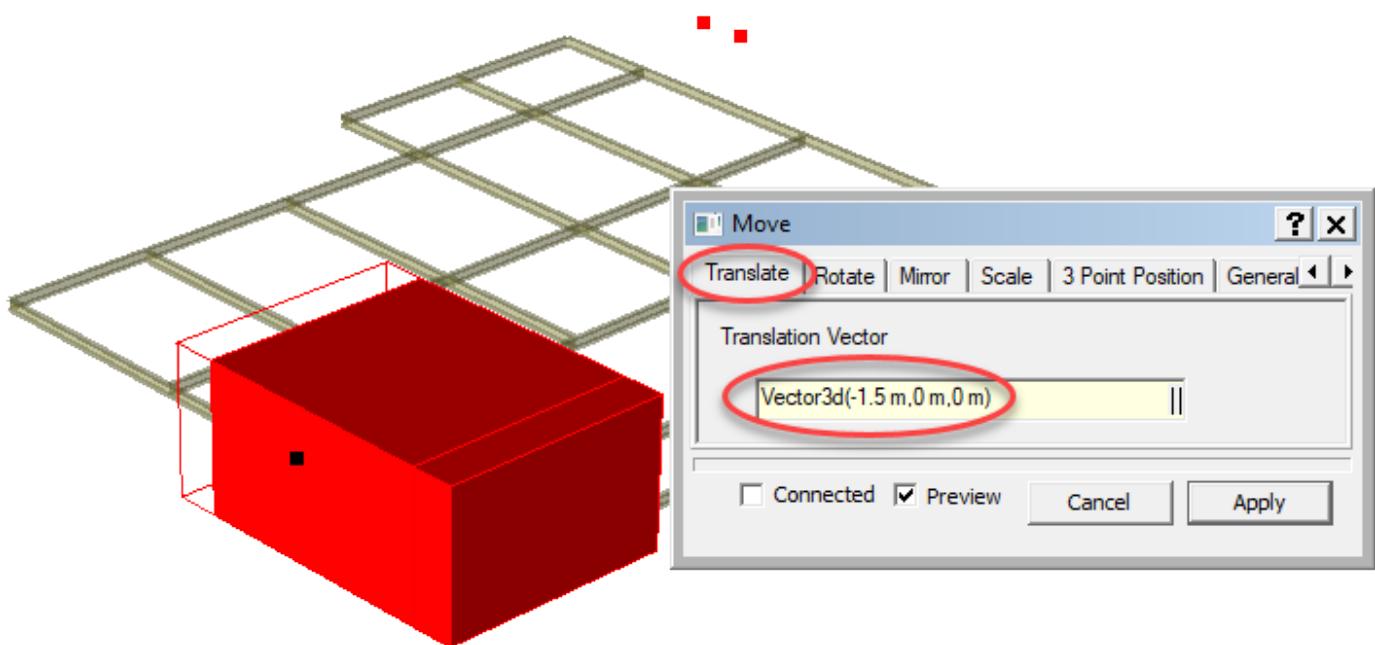
- Create a new load case named LC\_FlareTower.
- Place the equipment as shown below.



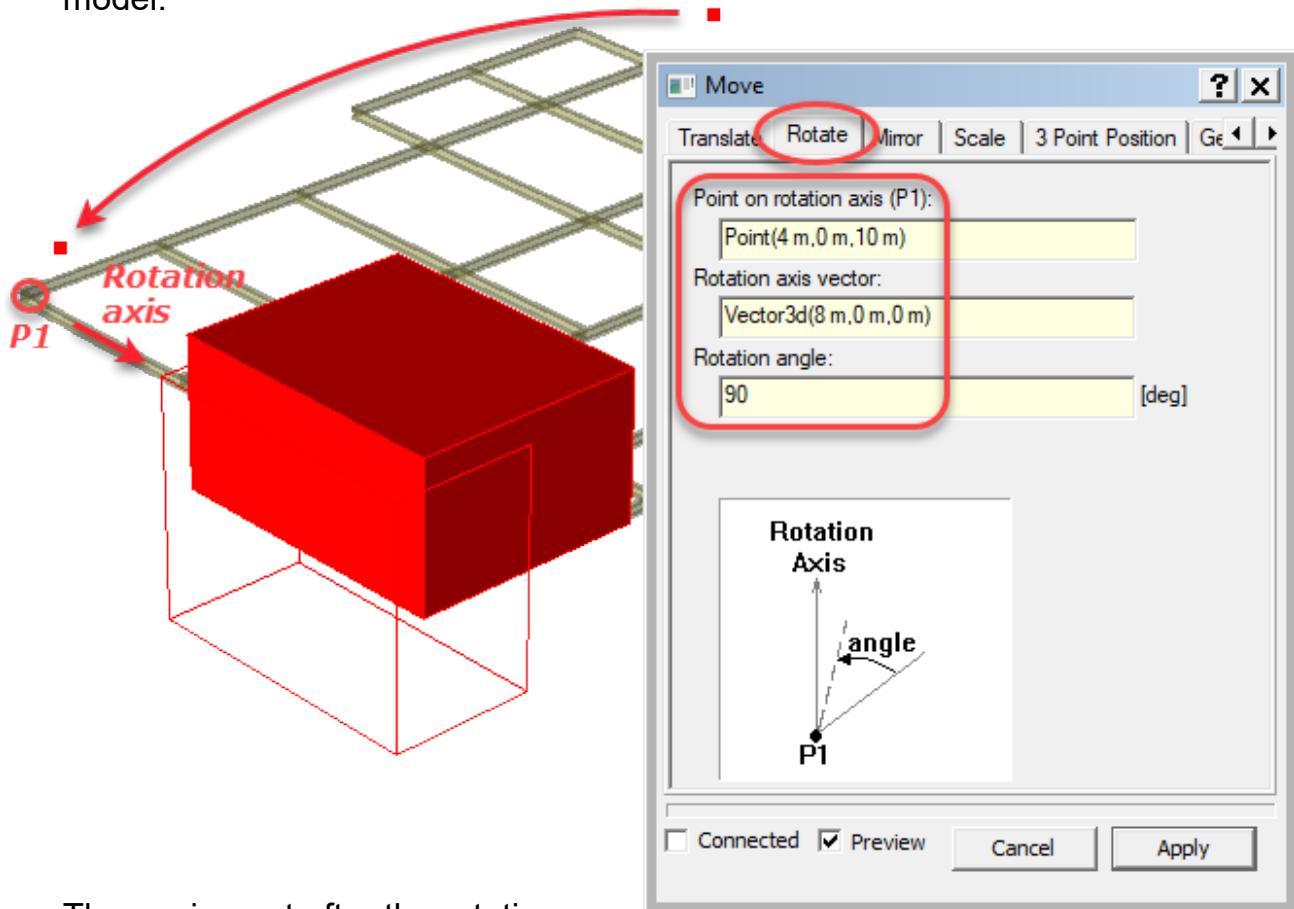
- The equipment appears. Notice the grey dot marking the equipment COG.



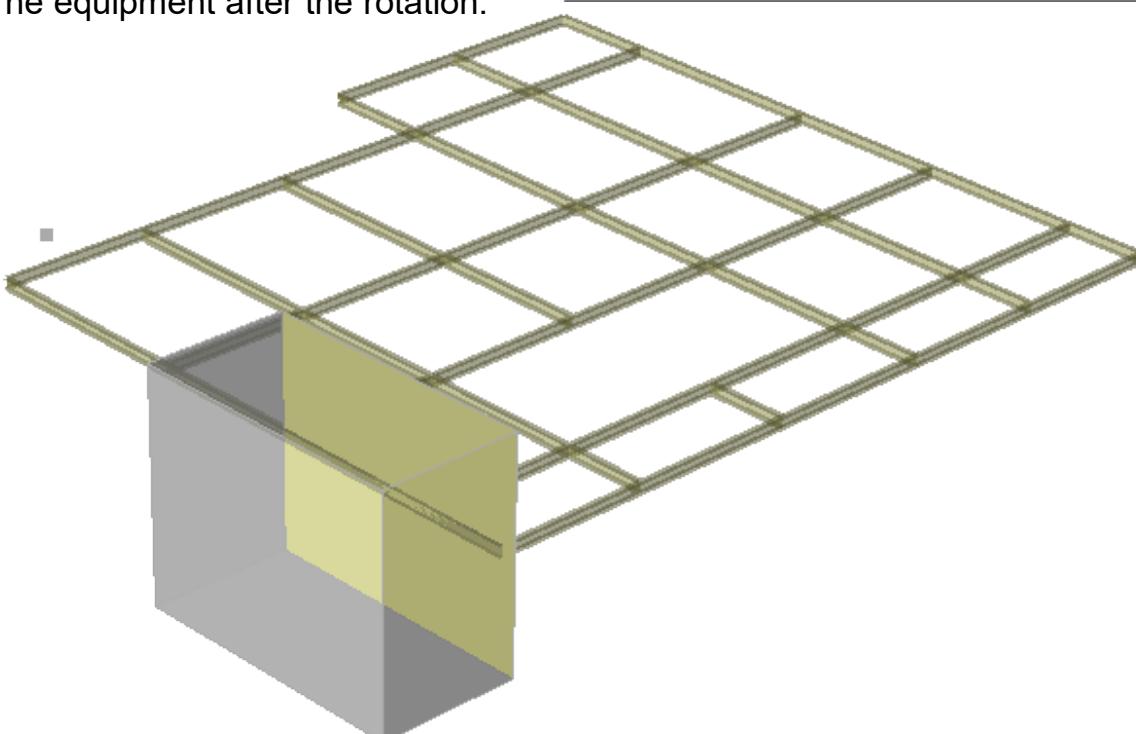
- Move the equipment 1.5 m in negative X-direction.



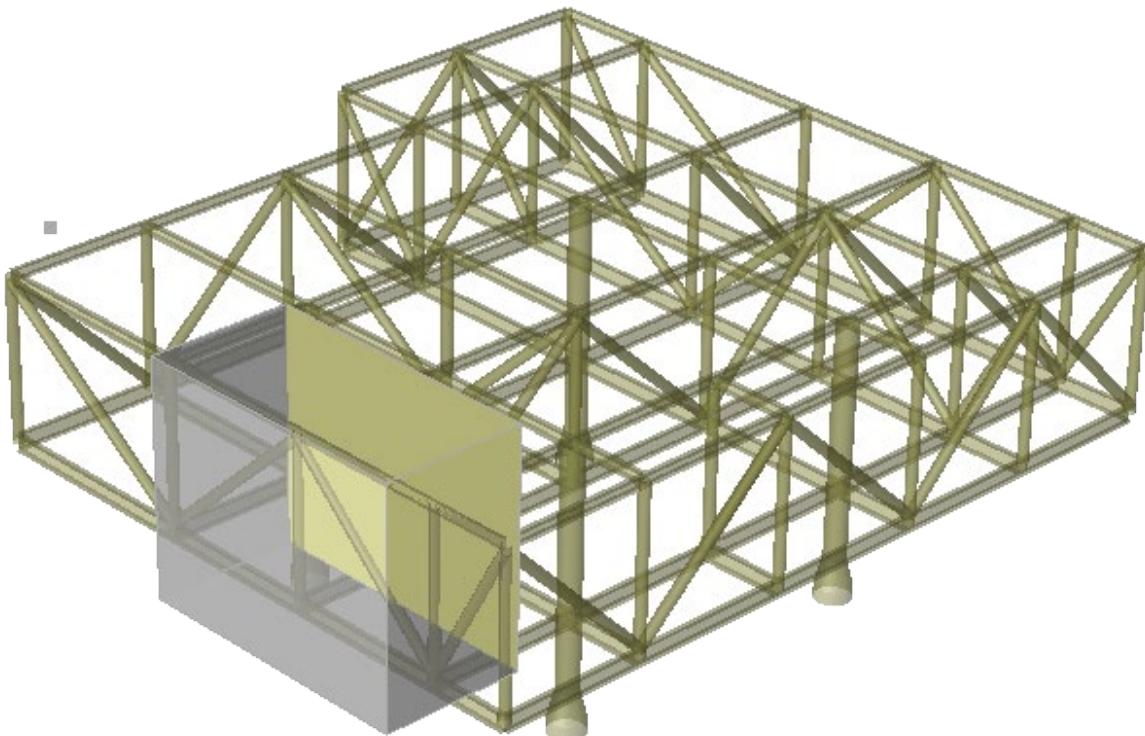
- The equipment shall also be rotated so as to be hanging on the side of the deck at row 1 rather than sitting on the main deck. The *Point on rotation axis (P1)* and the *Rotation axis vector* may be entered manually or be fetched by clicking in the model.



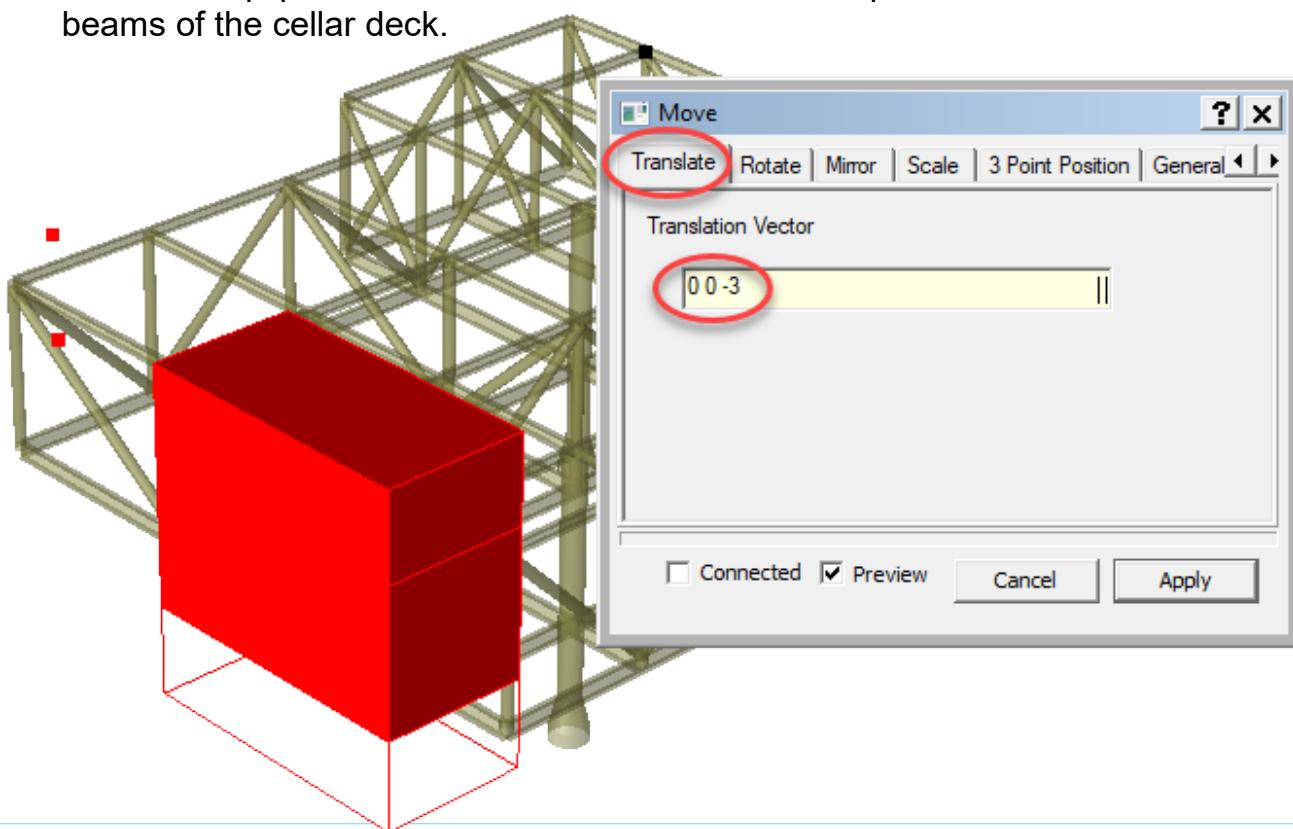
- The equipment after the rotation.



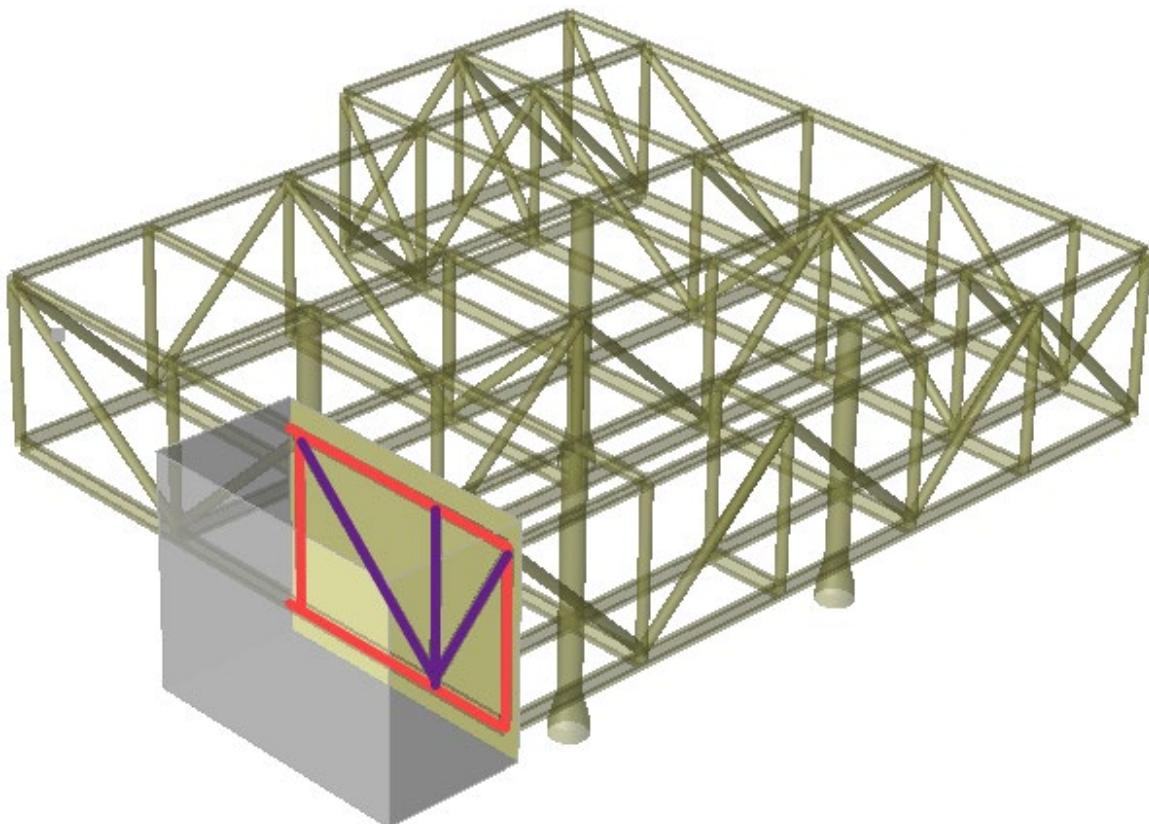
- Use Alt+A to display the complete model. Notice that the olive coloured equipment footprint now is vertical. This is required for the equipment loads to be applied to the side of the deck. (Line loads on beams are only computed where the footprint intersects the beam system lines – prior to any offset.)



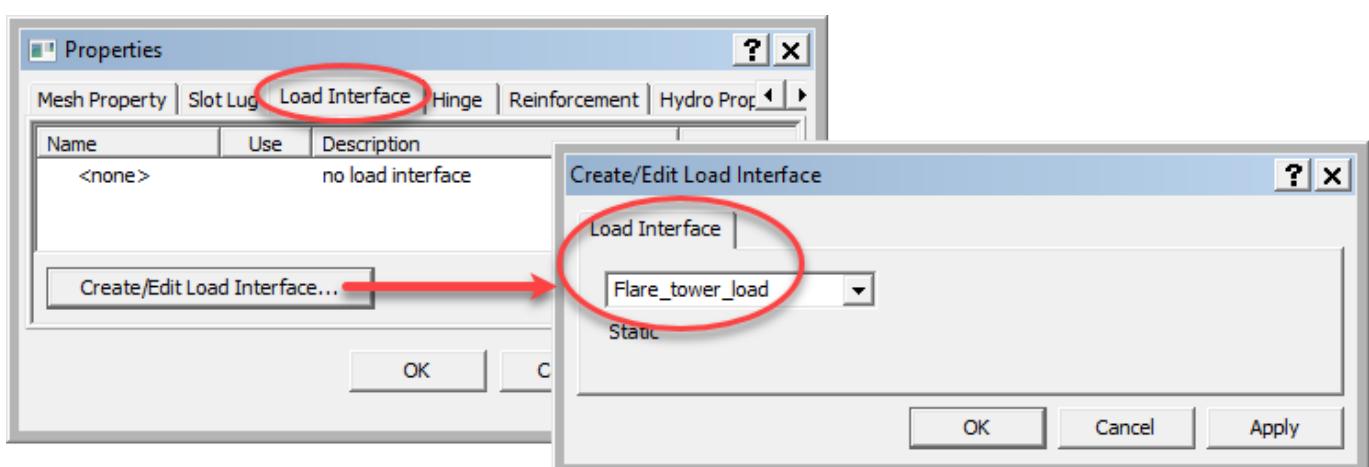
- Move the equipment downwards 3 m so that the footprint intersects also the beams of the cellar deck.



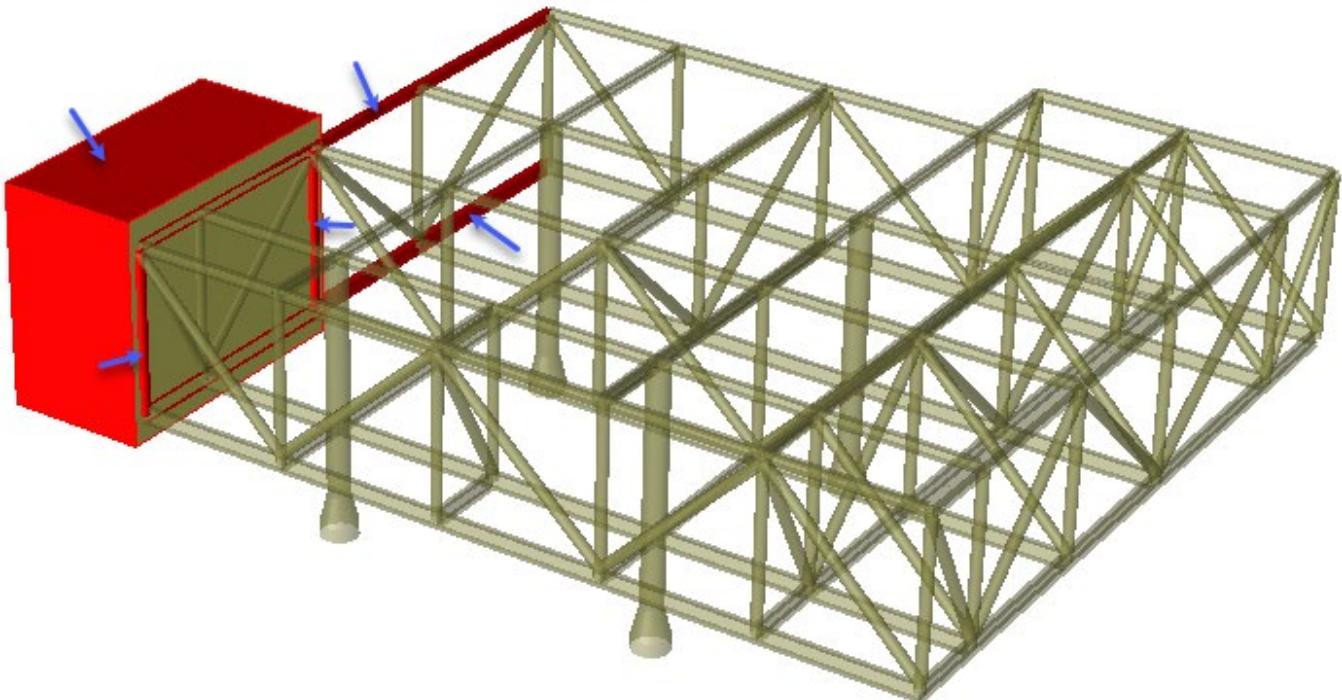
- In its current position and with its whole bottom surface (now vertical) as footprint, the equipment will be loading the beams highlighted in red and purple below.
- In this case, we only want the four red coloured beams to carry the equipment load. This can be achieved by creating four narrow strips of footprint matching the beams. This is, however, awkward. Moreover, no matter how narrow the strips are the purple coloured beams will be subjected to very short line loads.



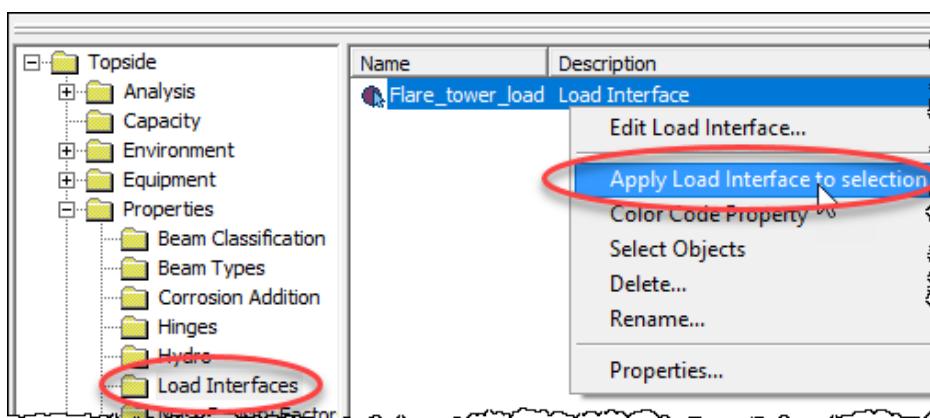
- The solution is to define a so-called load interface. This is done as follows.
  - Use *Edit | Properties* to open the *Properties* dialog, go to the *Load Interface* tab and click *Create/Edit Load Interface*. In the *Create/Edit Load Interface* dialog enter a name, e.g. Flare\_tower\_load, there is no more data to enter.



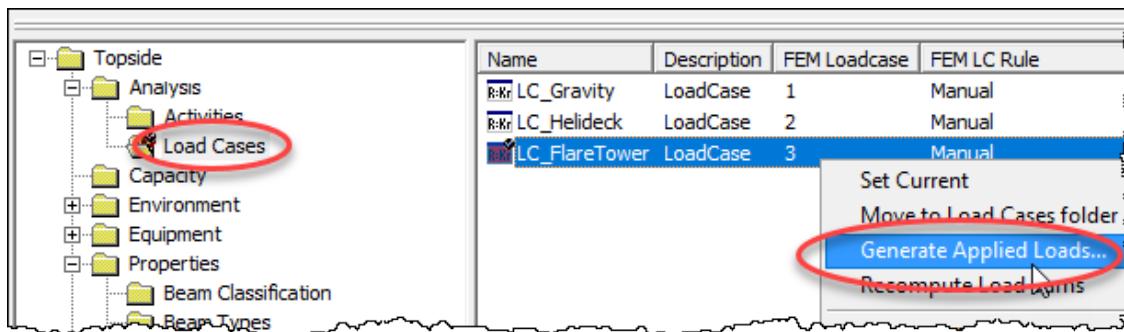
- Select the four red coloured beams and the equipment. In the figure below, blue arrows point to these five objects.



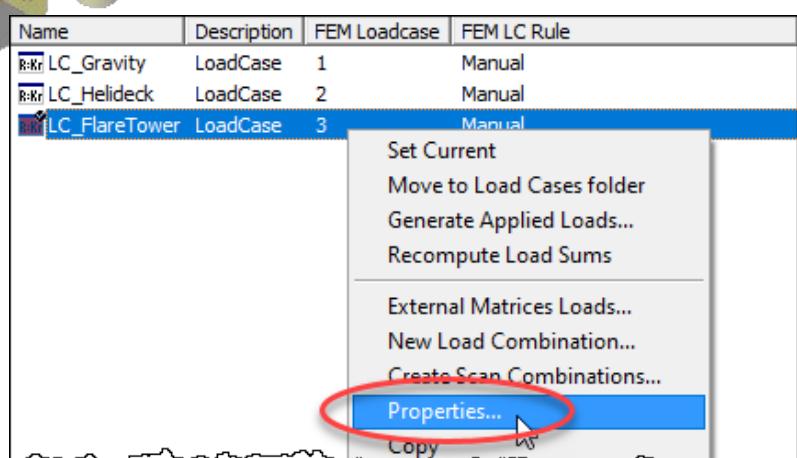
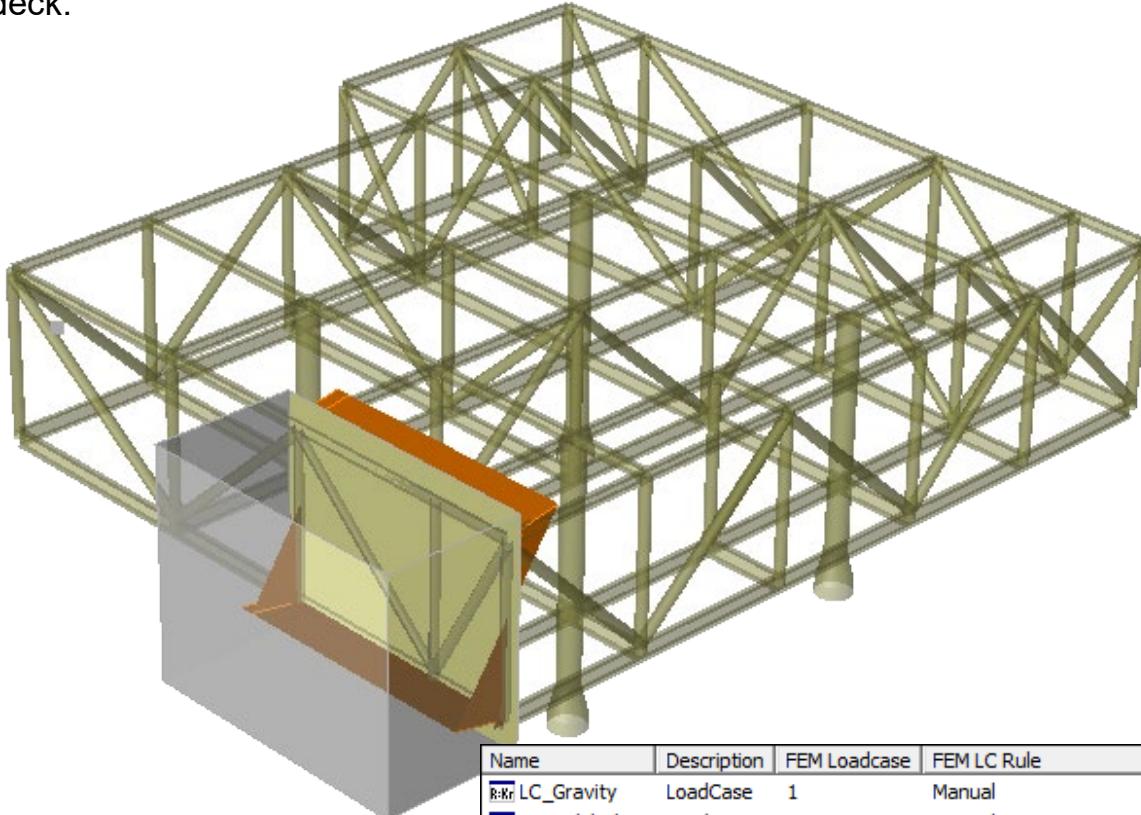
- Go to the *Properties | Load Interfaces* folder in the browser, select the load interface, right-click and click *Apply Load Interface to selection*. The load interface is thus a means of linking the equipment with selected beams thereby limiting load application to the selection.



- The beam line loads will be computed when an analysis is performed. To see the loads prior to an analysis, go to the *Analysis | Load Cases* folder in the browser, right-click the relevant load case and click *Generate Applied Loads*.

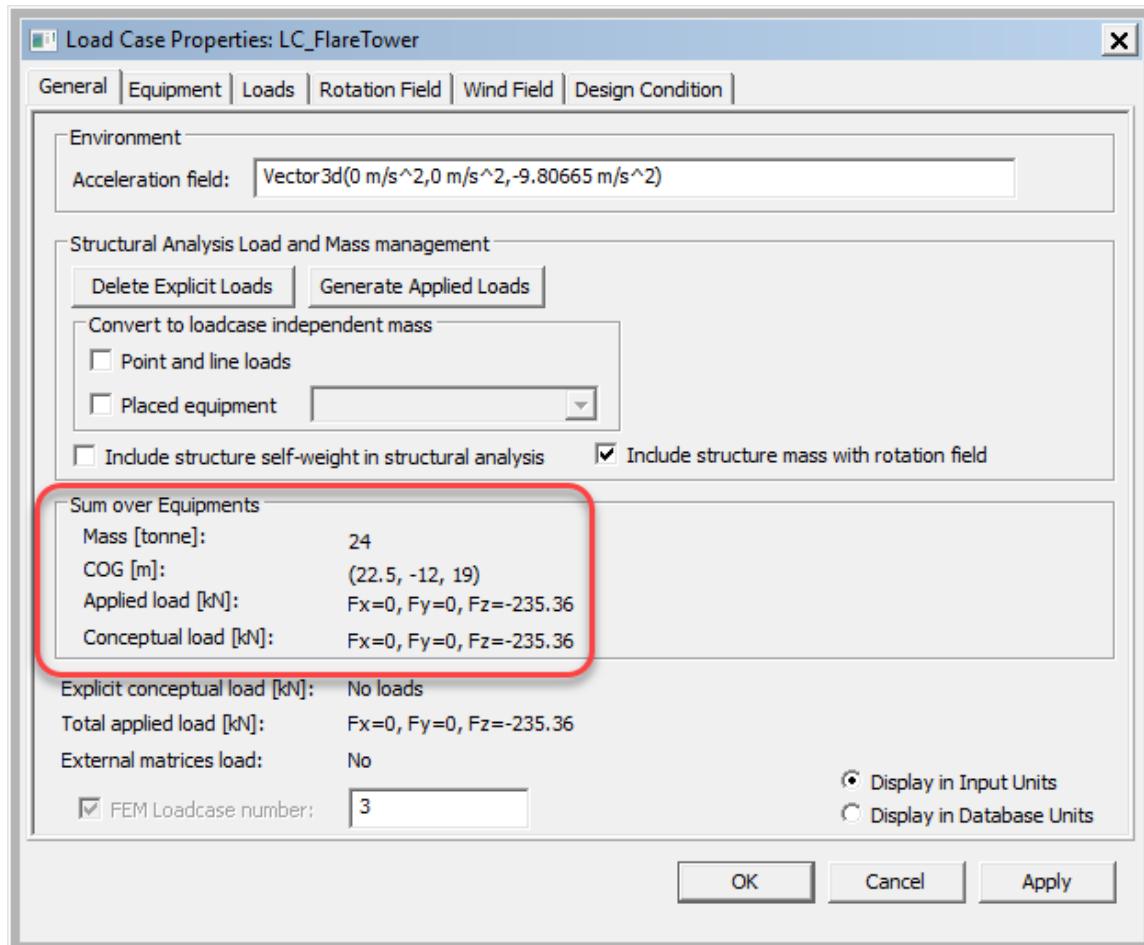


- The orange coloured line loads show the moment effect of the equipment on the deck.



- To see more details about the loads, right-click the load case and click *Properties*.

- In the *Load Case Properties* dialog see that *Applied load* (computed beam line loads) matches the *Conceptual load* (computed from the equipment alone). This ensures that the weight of the equipment is properly transferred to the deck.

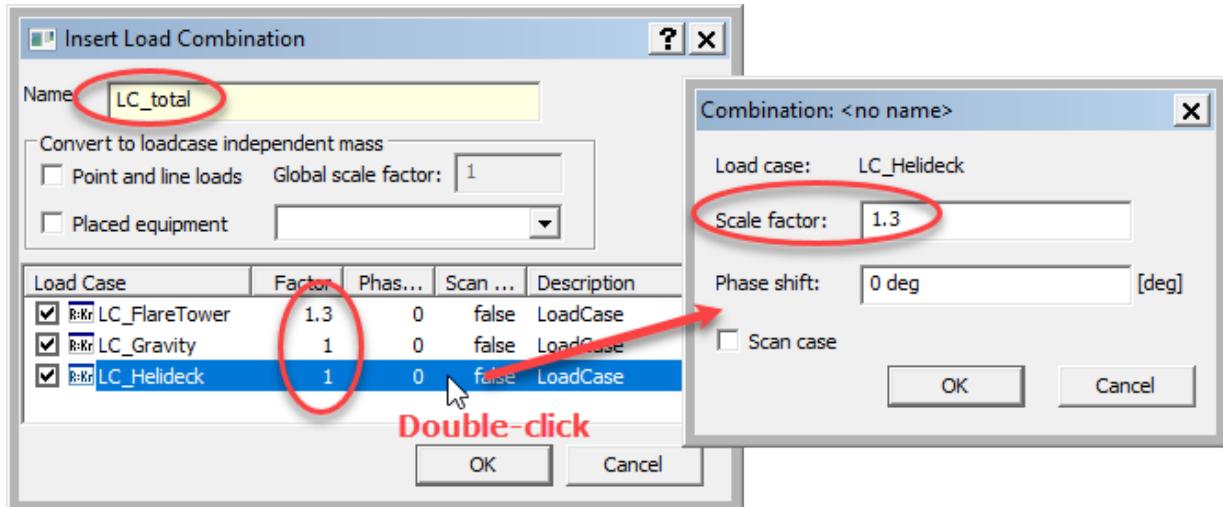


- Go to the *Loads* tab of the *Load Case Properties* dialog to find a table with details on line loads on the individual beams.

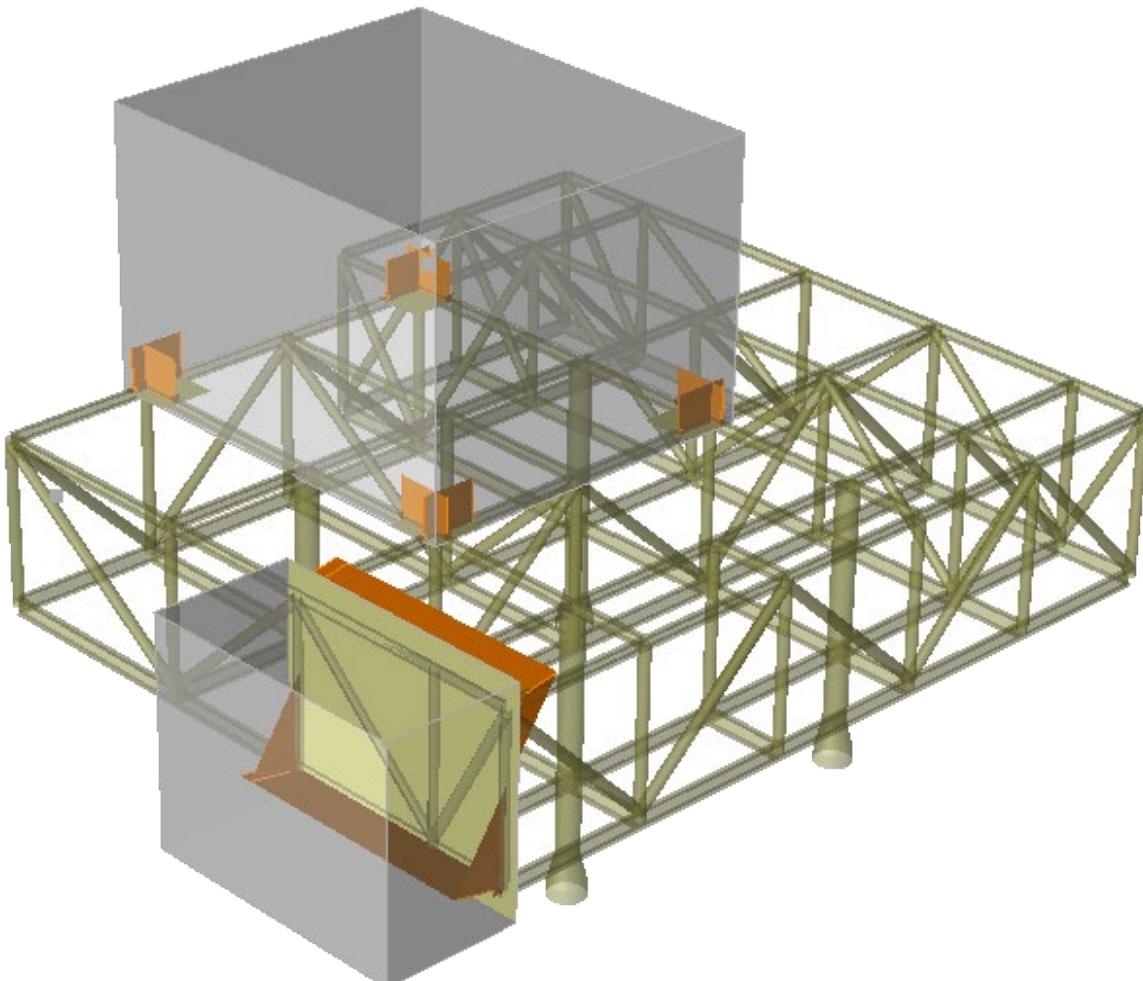
Load Generator	Structure	Description	x-coord	y-coord	z-coord	fx	fz	mx	my	mz	
LC_FlareTower.equipment(FlareTower)	Bm6	Applied Line Line Load.pos1	17.5	0	4	0	-7.13984	-40.9321			
LC_FlareTower.equipment(FlareTower)	Bm6	Applied Line Line Load.pos2	27	0	4	0	-8.02659	-40.9321			
LC_FlareTower.equipment(FlareTower)	Bm19	Applied Line Line Load.pos1	17.5	0	10	0	-7.13984	40.9321			
LC_FlareTower.equipment(FlareTower)	Bm19	Applied Line Line Load.pos2	27	0	10	0	-8.02659	40.9321			
LC_FlareTower.equipment(FlareTower)	Bm36	Applied Line Line Load.pos1	27	0	10	0	-8.02659	40.9321			
LC_FlareTower.equipment(FlareTower)	Bm36	Applied Line Line Load.pos2	27	0	4	0	-8.02659	-40.9321			
LC_FlareTower.equipment(FlareTower)	Bm38	Applied Line Line Load.pos1	18	0	4	0	-7.18651	-40.9321			
LC_FlareTower.equipment(FlareTower)	Bm38	Applied Line Line Load.pos2	18	0	10	0	-7.18651	40.9321			

At the bottom, there are three radio buttons: 'Explicit Loads' (unchecked), 'Applied Loads' (checked and highlighted with a red circle), and 'From Equipments'. There are also two checkboxes: 'From Explicit Loads' (checked) and 'Display Unit Notations' (unchecked).

- Create a combination of the three load cases by *Loads | Load Combination*. In the *Insert Load Combination* dialog give a name of the combination. Double-click a load case to change its scaling factor. The equipment loads shall be scaled to 1.3.

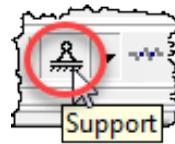


- Right-click the load combination in the browser and click *Generate Applied Loads* to see the beam line loads caused by the two equipments.

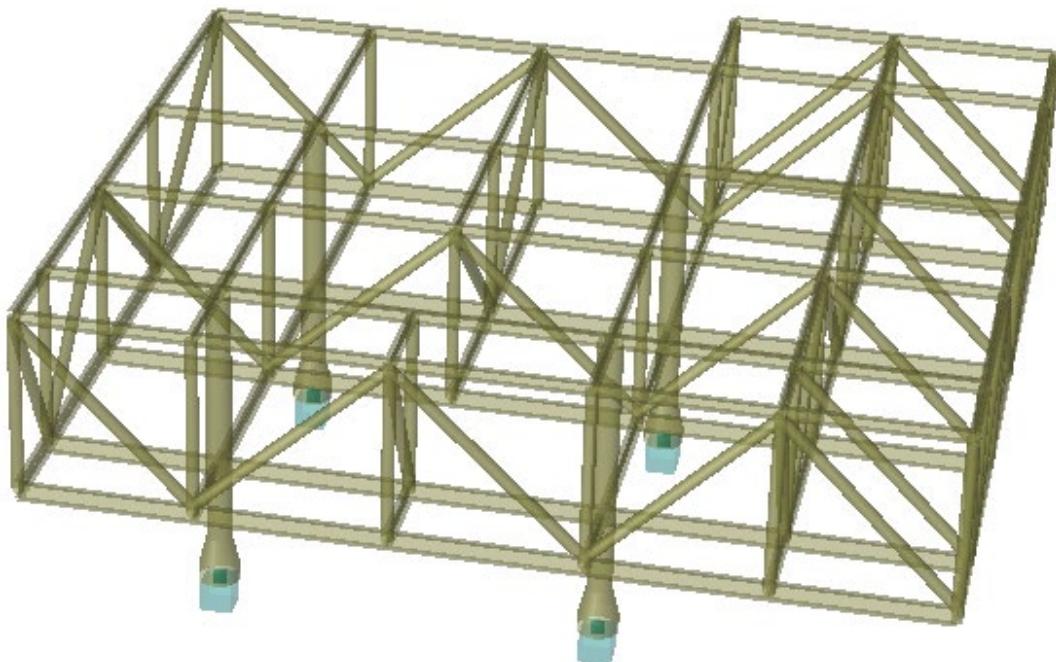
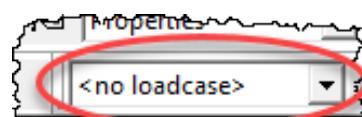


## 12 BOUNDARY CONDITIONS

- The bottom of the four columns shall be fixed. Click the *Support* button to create the four fixations. By default, all six degrees of freedom are fixed which is what we want.

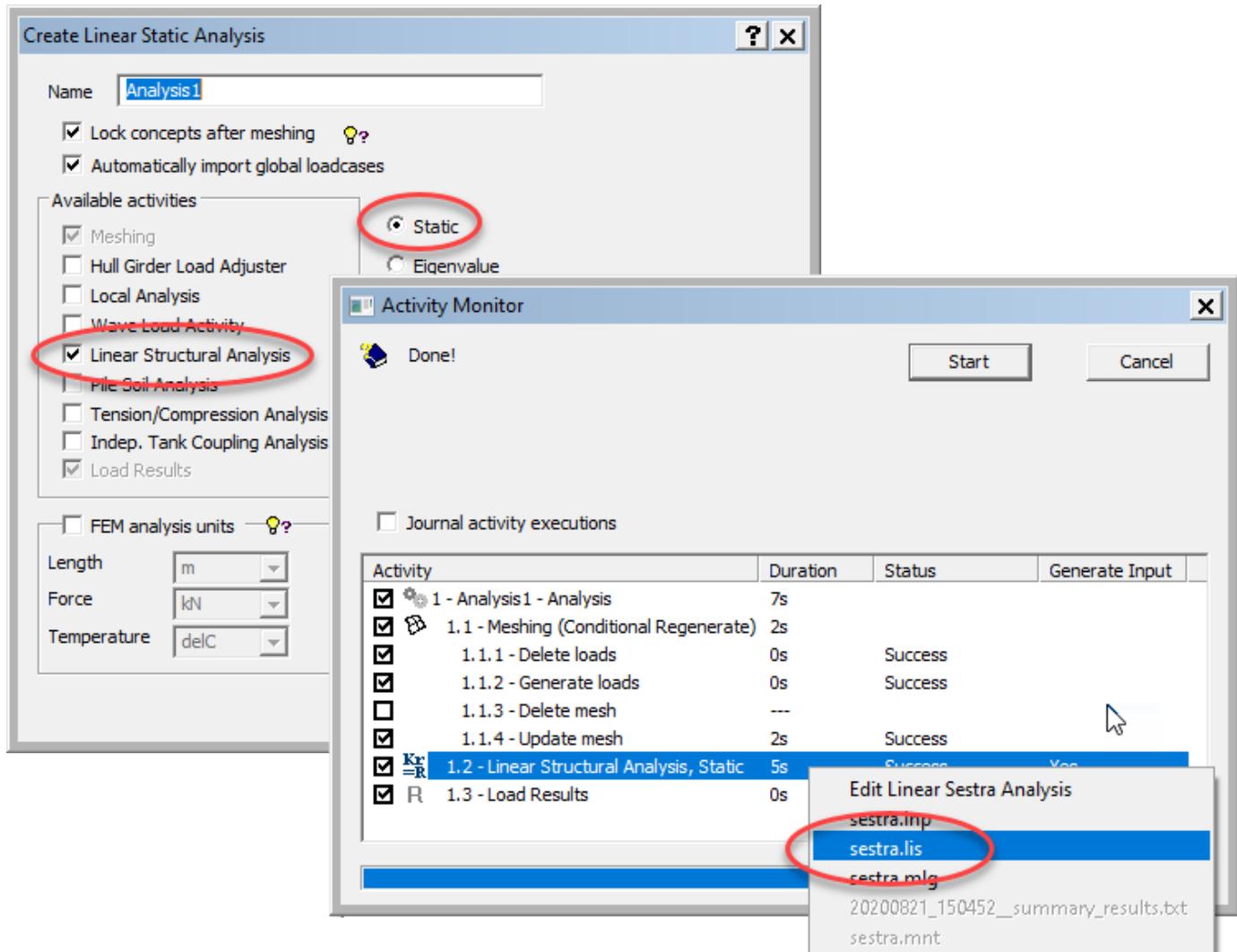


- To display the model without the equipments, set to <no loadcase> in the load case selector.



## 13 STATIC ANALYSIS

- Create a linear static analysis by *Mesh & Analysis | Activity Monitor* (or Alt+D). Click *OK* to create the analysis and *Start* in the *Activity Monitor* to run the analysis. Right-click *Linear Structural Analysis, Static* and click *sestra.lis* to see sum of loads, sum of reactions and the sum of the two that should be zero. Notice that the load combination (*FEM Loadcase 4*) is not listed, this is because *Smart load combination* is by default switched on. Read about this in the user manual.



```

Load sum for all result cases (internal number):
result case;          tx;          ty;          tz;          rx;          ry;          rz
1; -1.865175e-14; 4.440892e-15; -1.735255e+03; -2.618041e+04; 2.643786e+04; 3.654854e-13
2; 0.000000e+00; 0.000000e+00; -4.903325e+02; -5.638824e+03; 5.393658e+03; 0.000000e+00
3; 0.000000e+00; -9.947598e-14; -2.353596e+02; 2.824312e+03; 5.295591e+03; -1.364242e-12

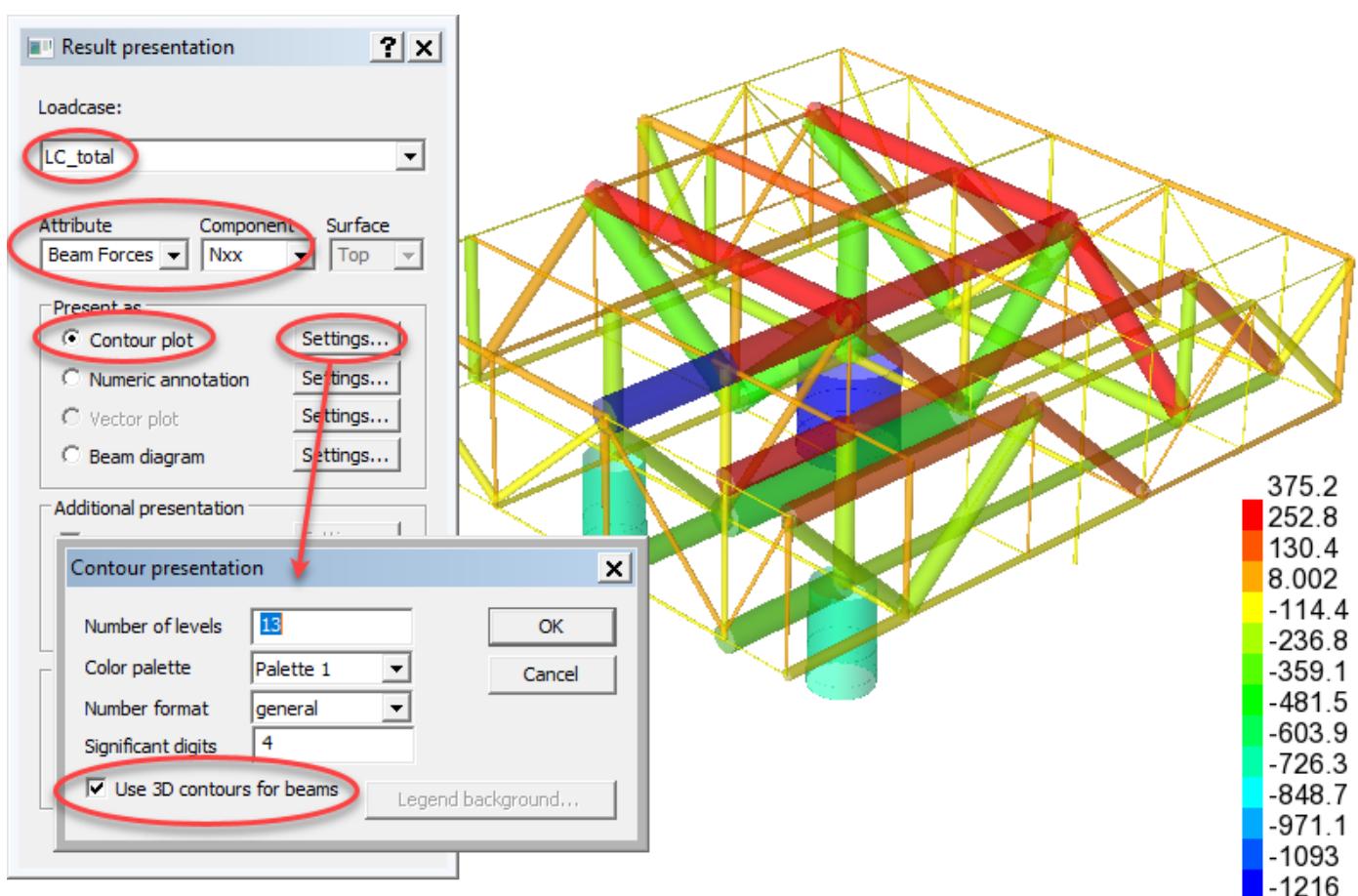
Reaction force sum for all result cases (internal number):
result case;          tx;          ty;          tz;          rx;          ry;          rz
1; -8.007983e-12; 1.708500e-11; 1.735255e+03; 2.618041e+04; -2.643786e+04; 2.252420e-10
2; -8.139267e-12; -2.775558e-13; 4.903325e+02; 5.638824e+03; -5.393658e+03; 7.041479e-11
3; 5.034195e-12; -6.991741e-12; 2.353596e+02; -2.824312e+03; -5.295591e+03; -1.737135e-10

Sum of load sum and reaction force sum for all result cases (internal number):
result case;          tx;          ty;          tz;          rx;          ry;          rz;      magnitude
1; -8.026635e-12; 1.708944e-11; -6.525624e-11; -1.778972e-09; 6.548362e-10; 2.256075e-10; 1.910252e-09
2; -8.139267e-12; -2.775558e-13; -1.443823e-11; -2.073648e-10; 1.637090e-11; 7.041479e-11; 2.202299e-10
3; 5.034195e-12; -7.091217e-12; 1.693934e-11; 5.998118e-10; -3.428795e-10; -1.750777e-10; 7.129904e-10

```

## 14 PRESENT RESULTS

- Switch to display configuration *Results - All* and use *Results | Presentation* (or Alt+P) to open the *Result presentation* dialog. In the dialog select *Loadcase*, e.g. the load combination, *Attribute* and *Component*, e.g. *Beam Forces* and *Nxx*, and how to present the result, e.g. *Contour plot*.
  - When selecting beam forces the *Settings* button may be clicked and *Use 3D contours for beams* checked to see beams presented as tubes (whichever cross section they actually have) with diameters proportional to the absolute values the forces/moment.



## 15 SAVE THE WORKSPACE

- Save the model – which is recommended to be done from time to time during modelling.
  - Find in the workspace folder a file with name of the workspace and extension js, in this case Topside.js.
    - This file contains commands corresponding to all actions performed (including any mistakes, deletions, recreations, etc). Sometimes, it is therefore messy.
    - It can be used to recreate the model.
    - It can also be edited and used as input to a new session.

```

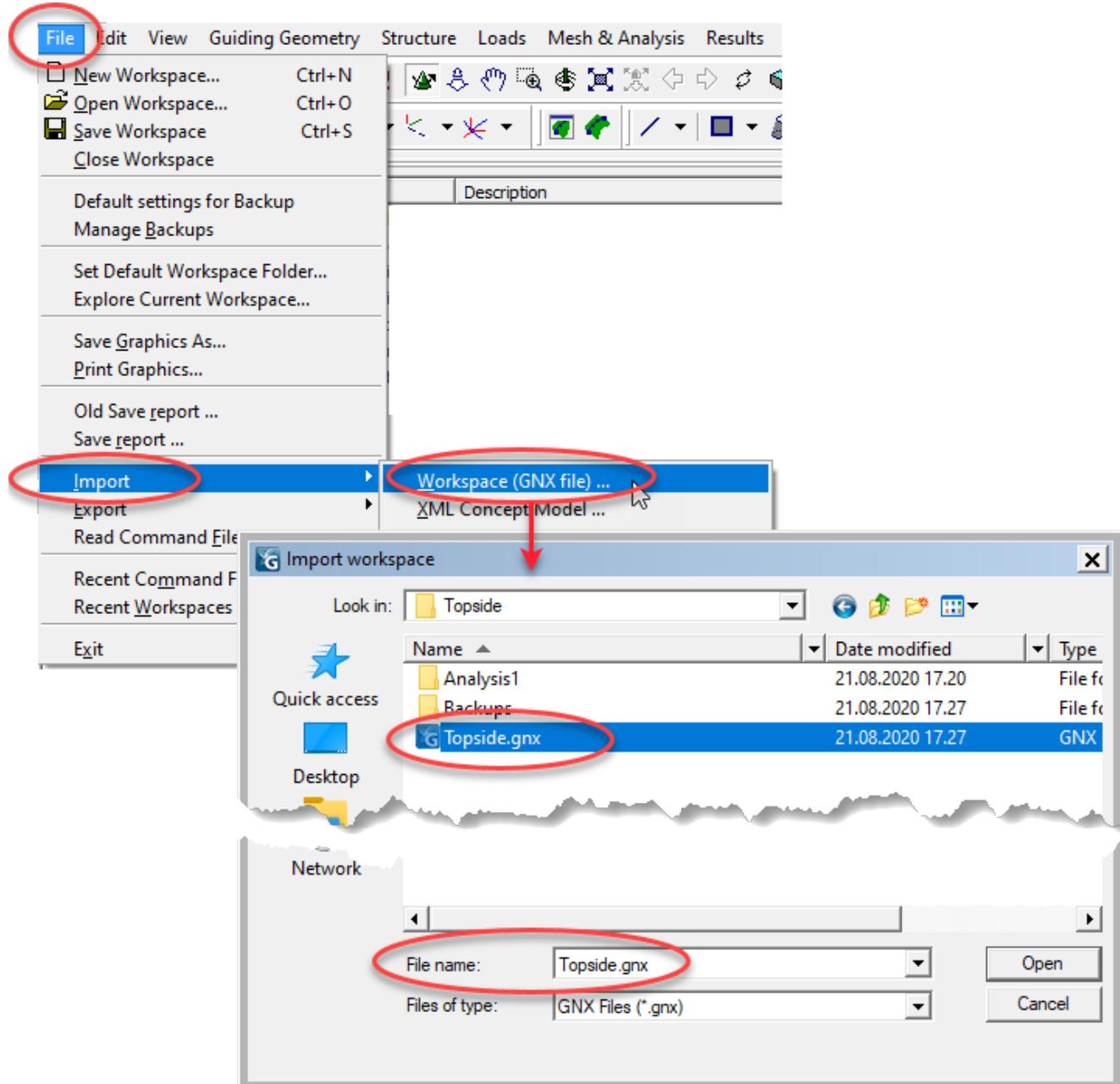
1 // .GenieE.V7.14-05.64bit.started.21-Aug-2020.15:55:32]
2 GenieRules.Compatibility.version = "V7.14-05";
3 GenieRules.Tolerances.useTolerantModelling = true;
4 GenieRules.Tolerances.angleTolerance = 2.deg;
5
6 GenieRules.Meshing.autoSimplifyTopology = true;
7 GenieRules.Meshing.eliminateInternalEdges = true;
8 GenieRules.BeamCreation.DefaultCurveOffset = ReparameterizedBeamCurve;
9 GenieRules.Transformation.DefaultConnectedCopy = false;
10 GenieRules.Units.setOutputUnits("m", "kN", "deg");
11 GenieRules.Units.setInputUnit(Length, "m");
12 GenieRules.Units.setInputUnit(Force, "kN");
13 GenieRules.Units.setInputUnit(TempDiff, "deg");
14 // .Material.and.beam.cross.sections
15 Mat1 = MaterialLinear(360000, 7.85.tonne/m^3, 210000000.kPa, 0.3, 1.2);
16 HE400A = ISection(0.39.m, 0.3.m, 0.011.m, 0.019.m, 0.027.m);
17 HE400A.description = "NVS.lib::HE.400.A.NS-EN.10034";
18 HE400A.libraryGeneralSection = GeneralSection(0.0159.m^2, 1.9e-06.m^4);
19 HE600A = ISection(0.59.m, 0.3.m, 0.013.m, 0.025.m, 0.027.m);
20 HE600A.description = "NVS.lib::HE.600.A.NS-EN.10034";
21 HE600A.libraryGeneralSection = GeneralSection(0.0226.m^2, 3.99e-06.m^4);
22 P_bracing = PipeSection(0.4, 15.mm);
23 P_leg_norm_15 = PipeSection(80.cm, 15.mm);
24 P_leg_norm_35 = PipeSection(0.8, 0.035);
25 P_leg_large_35 = PipeSection(1.3, 0.035);
26 // .Guide.plane
27 GenieRules.Meshing.meshDensityRounded = true;
28 GuidePlane1 = GuidePlane(Point(0.m, 0.m, 4.m), Point(27.m, 0.m, 4.m), Point(27.m, 28.5.m, 4.m));
29 GuidePlane1.snapmode = true;
30 // .Create.beams.of.cellar.deck
31 Mat1.setDefault();
32 HE600A.setDefault();
33 Bm2 = StraightBeam(Point(0.m, 28.5.m, 4.m), Point(27.m, 28.5.m, 4.m));

```

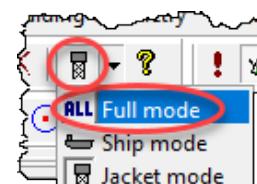
- Also find a file with name of the workspace and extension gnx, in this case Topside.gnx.
  - This is the Genie database and can be opened directly or imported into a new Genie session.

## 16 DETAILED MODELLING OF A JOINT

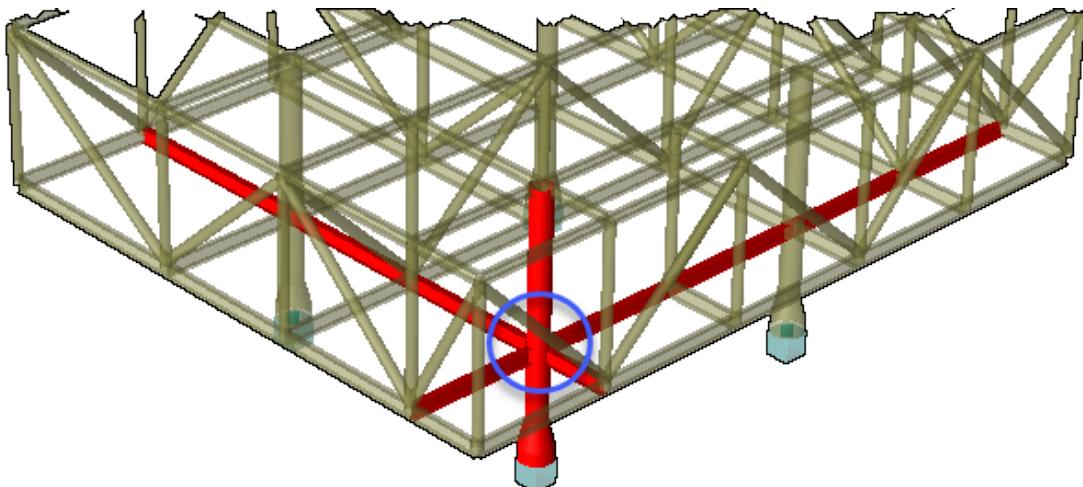
- Next step is to create a detailed plate/shell model of a joint, insert this into the beam model and do a new static analysis.
- Create a new workspace named Detailed\_Modelling\_Joint and with units m and kN.
- Import the database of the previous session, i.e. the file named Topside.gnx.



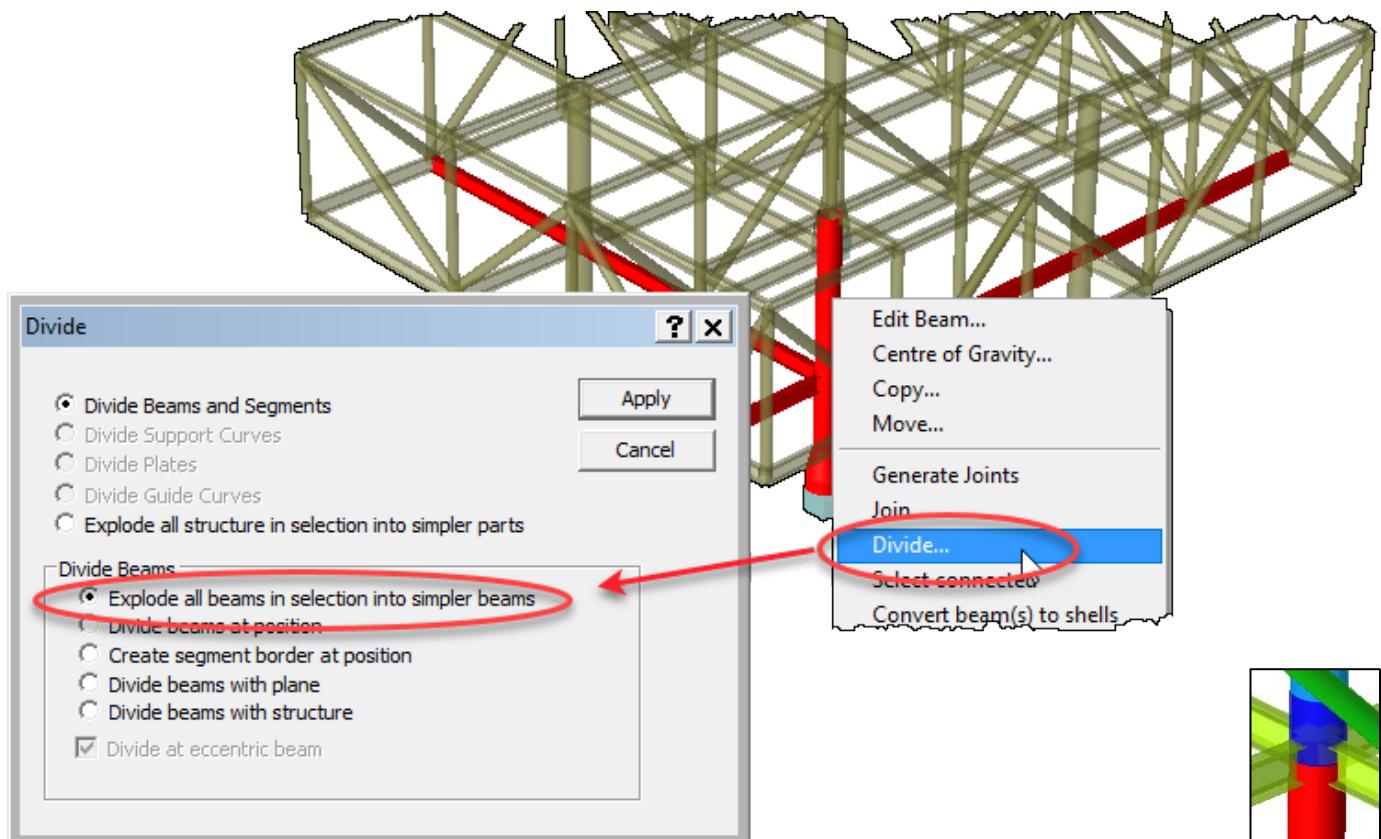
- Switch from *Jacket mode* to *Full mode* to get access to all curved shell modelling features.



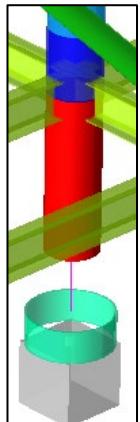
- Having imported the gnx file, the beam model of the topside appears (press F5).
- The joint to be converted into a plate/shell model is the one encircled below.



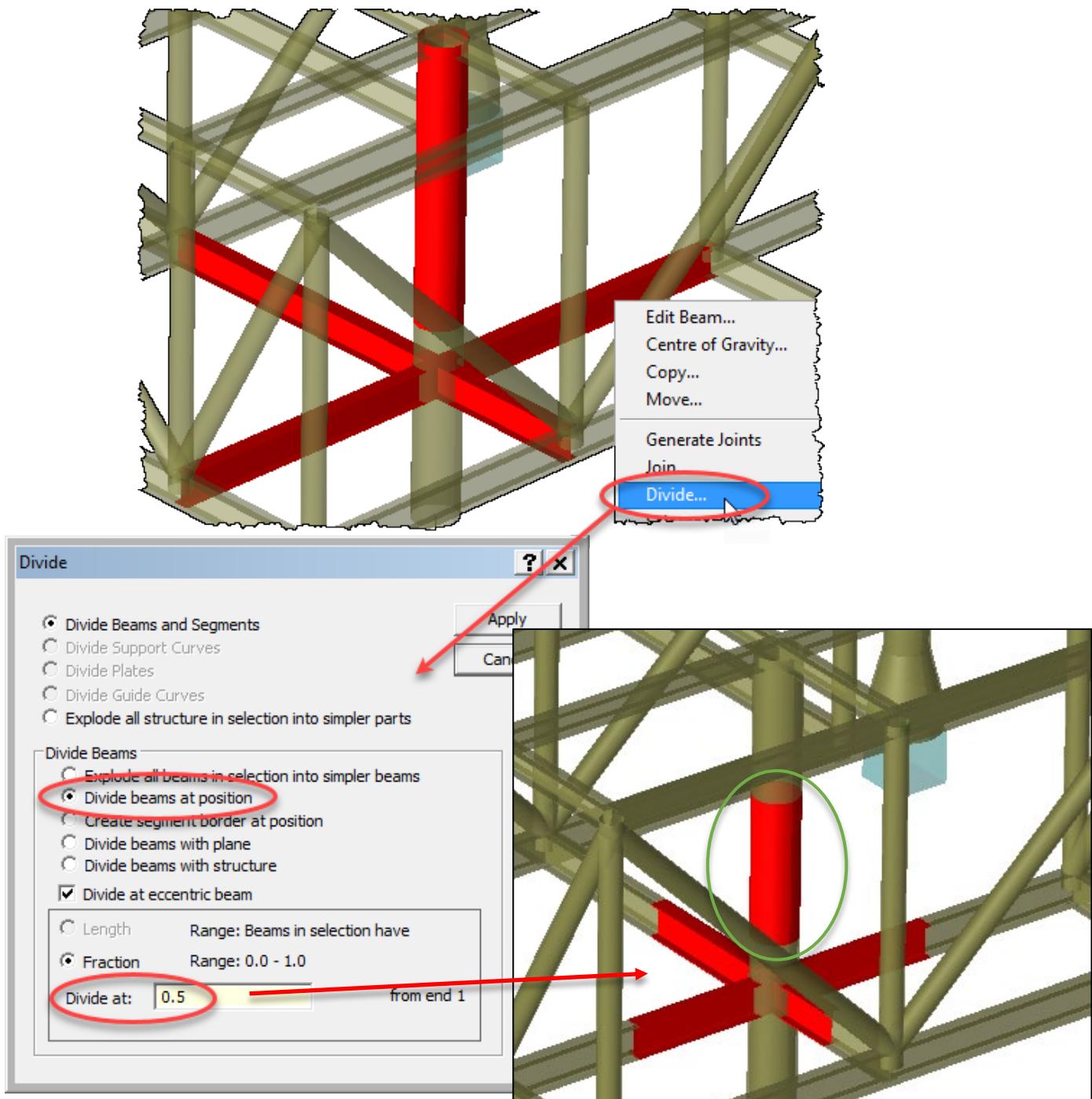
- Start by splitting the three intersecting beams at the joints.



- Note that the vertical tube being segmented is split so that each segment becomes an individual beam as seen to the right.
- Also note that the cone segment becomes a beam without cross section. This is because a cone section is defined by its adjoined segments. I.e. a cone is required to be a segment in between two other segments.



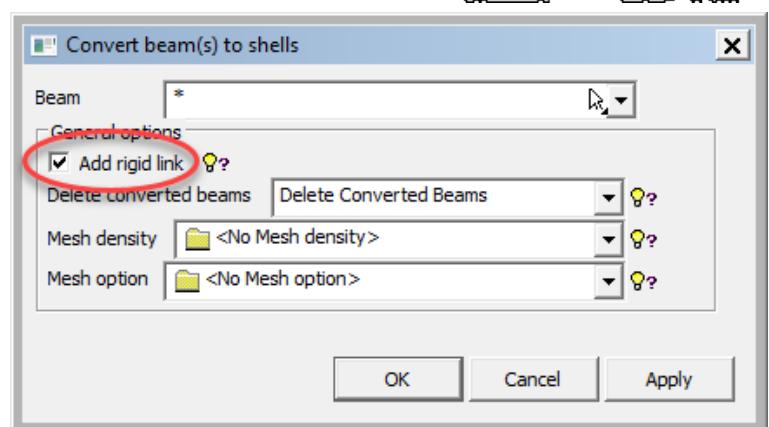
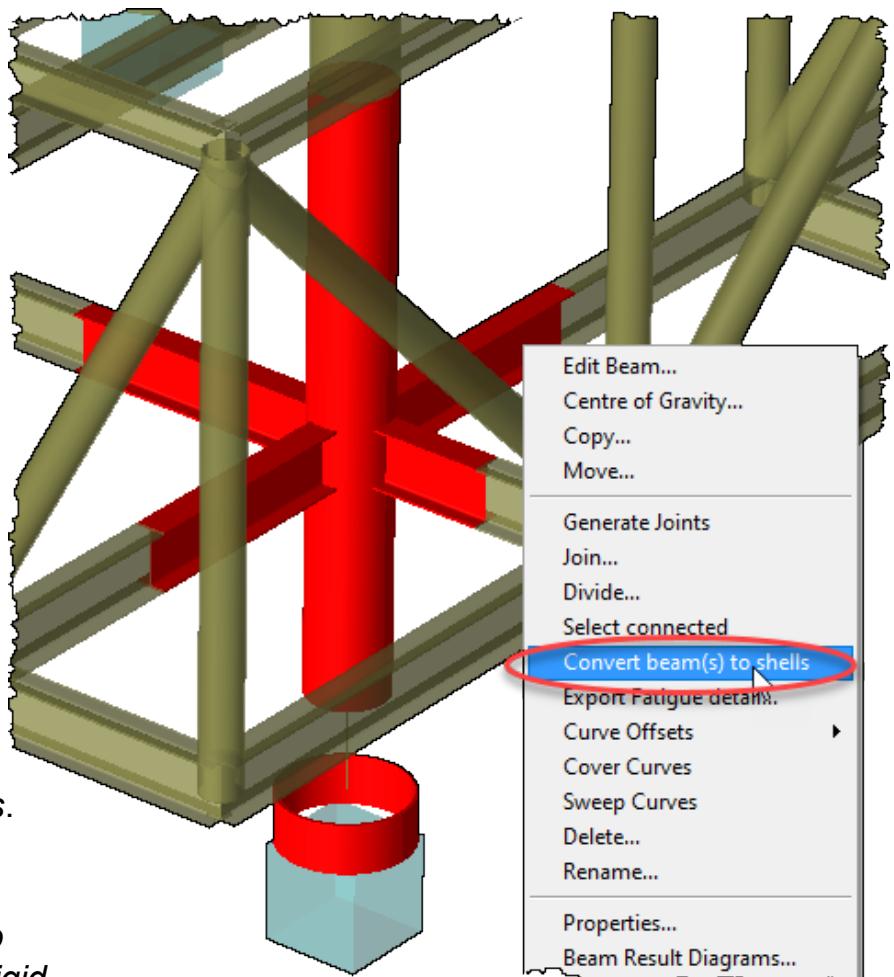
- Divide the beams further at their midpoints. These midpoints will define the extent of the conversion to plate/shell. Therefore, select the four horizontal I-beams extending out from the joint and the vertical pipe above the joint. All highlighted below. The two pipes connected to the joint, the beam without section (the cone) and the supported pipe at bottom shall not be divided.



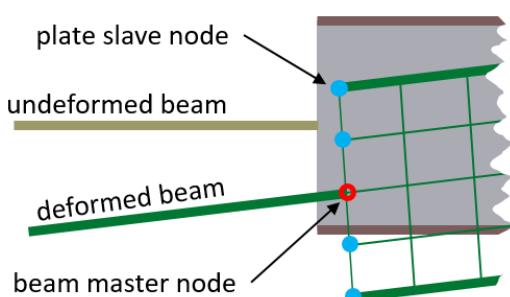
- Change the cross section for the beam encircled above to P\_leg\_norm\_35, i.e. to the same as for the beams underneath. This is required because converting beams with different sections to shells would result in cylinders with different diameters, i.e. unconnected cylinders.

- Convert I-beams and pipes into plate/shell models.  
First select beams:

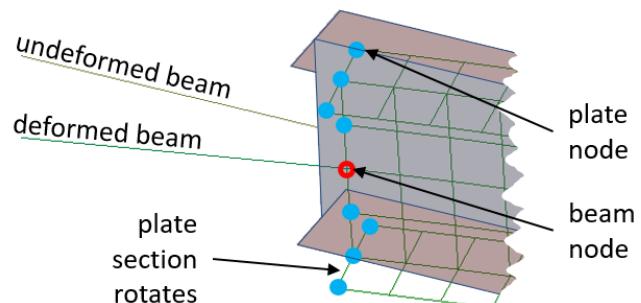
- The four horizontal I-beams
- The lower part of the divided pipe above the joint
- The two pipes connected to the joint
- The pipe at bottom with larger diameter
- Not the beam without section (the cone)
- Right-click and select *Convert beam(s) to shells*.
- In the *Convert beam(s) to shells* dialog check *Add rigid link*. This adds so-called 'support rigid links' at the ends of all converted beams.
- The 'support rigid links' connects the plate/shell model to the beam model by use of linear couplings (master-slave).
- The effect of a 'support rigid link' is illustrated below.



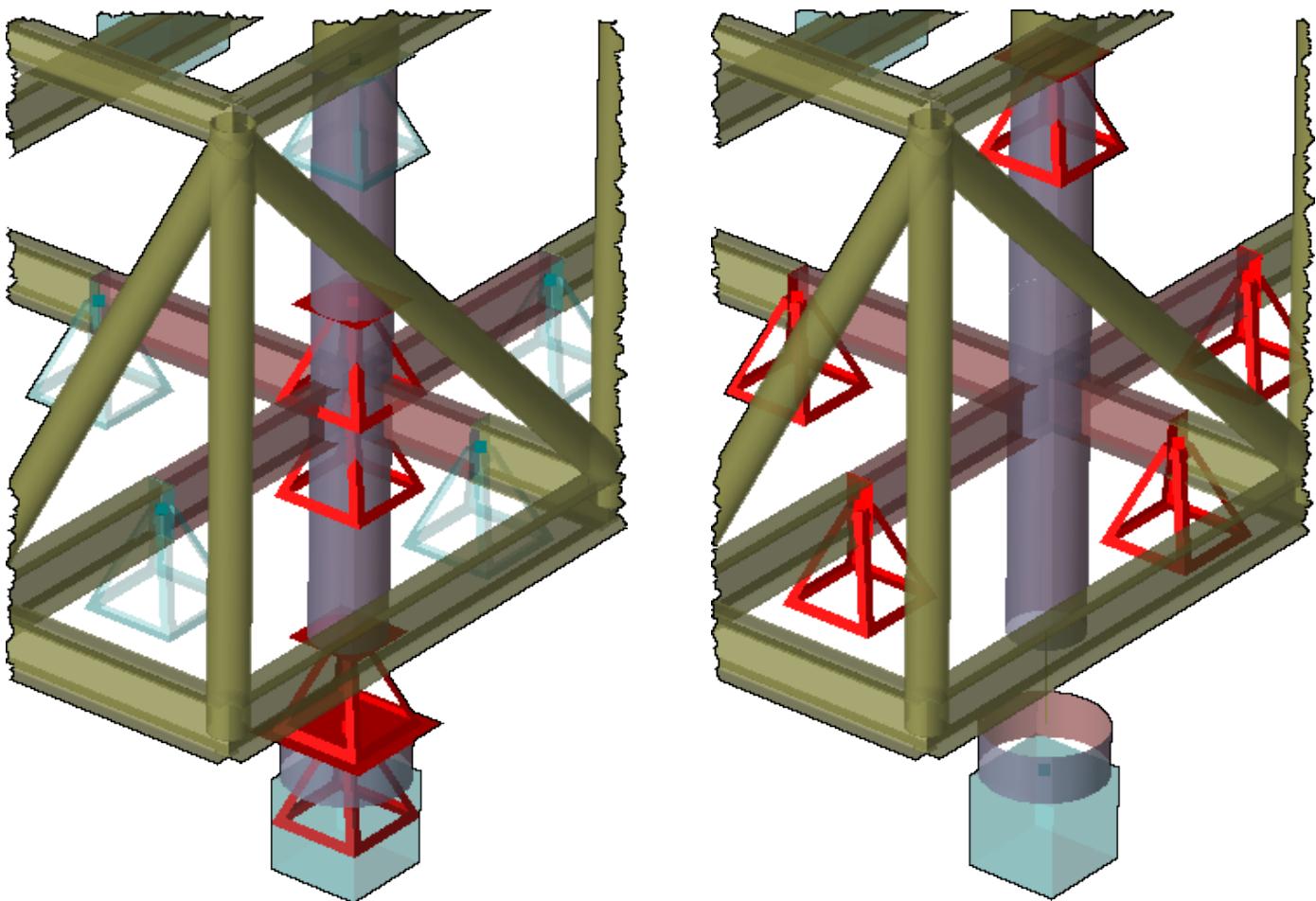
#### 2D illustration



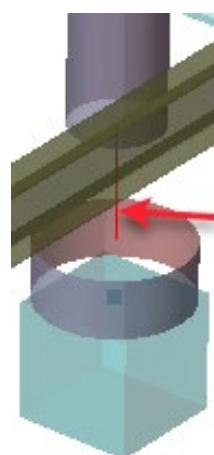
#### 3D illustration



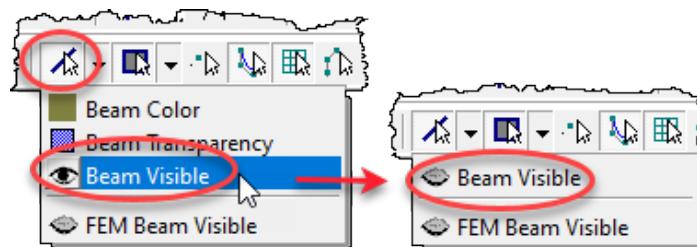
- However, the conversion from beam to plate/shell adds ‘support rigid links’ also at in between plate/shell parts. Delete these four superfluous ‘support rigid links’ shown to the left below.
- Only the five ‘support rigid links’ shown to the right below shall remain.



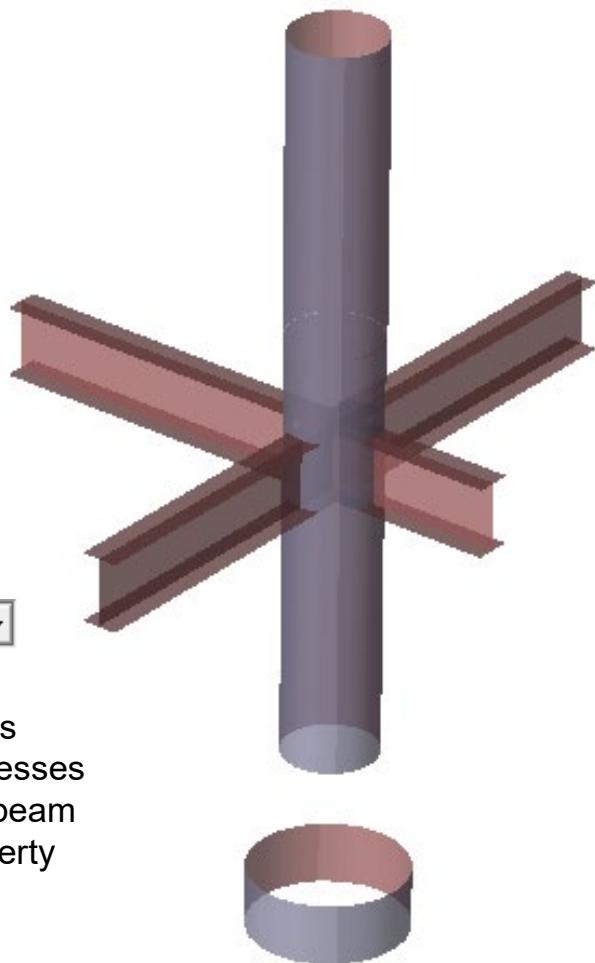
- Also delete the beam without section. The cone part must be created manually.



- For convenience, display only the plate/shell part of the model by right-clicking the *Beam selection* button and clicking *Beam Visible* (closing the eye).
- Note that this change of the current display configuration is stored in the registry and is thus persistent. It must therefore be set back (open eye) for the current display configuration to show beams.



- Do the same (close the eye) for supports.
- Only the plate/shell part is displayed.

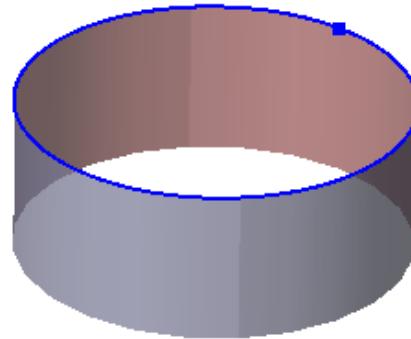
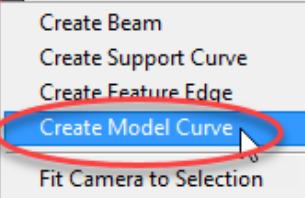
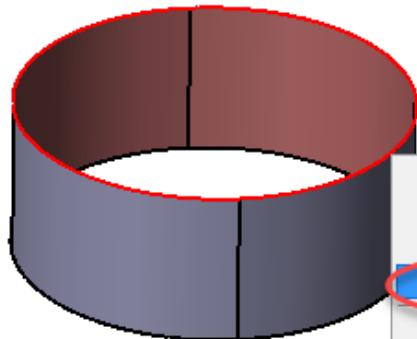
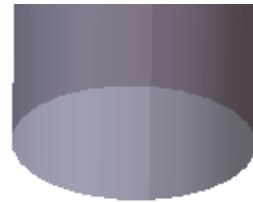
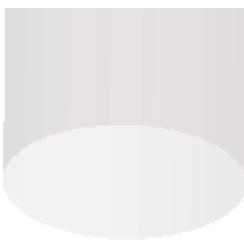
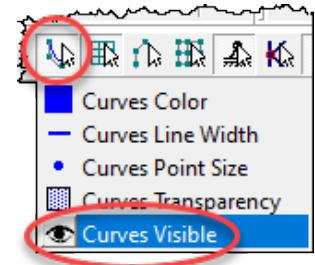


- The cone must be created. Set the proper thickness 35 mm as default.
- Note that the conversion from beam to plate/shell automatically created thicknesses corresponding to the web and flange thicknesses as well as tube thicknesses for all relevant beam cross sections. Find that the thickness property named Tck3 is the one with 35 mm.

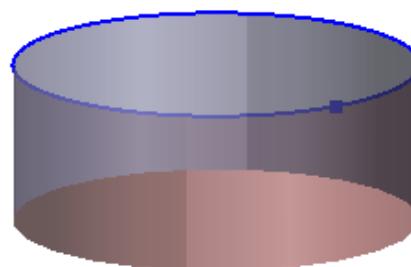
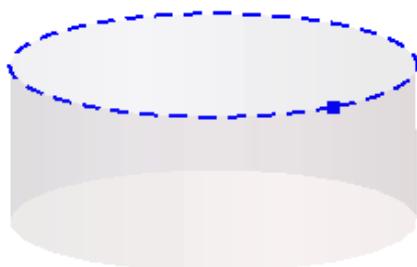
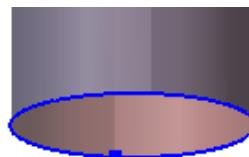
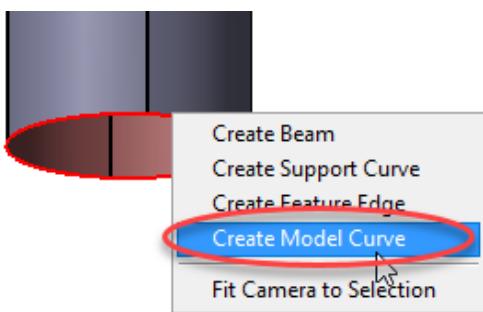
Name	Description	Thickness [m]
● Tck1	Thickness, t=0.025 m	0.025
● Tck2	Thickness, t=0.013 m	0.013
● Tck3	Thickness, t=0.035 m	0.035
● Tck4	Thickness, t=0.015 m	0.015

- Double-click the large diameter pipe below the cone, select the shell edges constituting the top edge of the cylinder and press *Create Model Curve* to create a guide curve. Double-click anywhere to return to normal display mode.

- To see the guide curve created ensure that the eye is open for *Guide curve selection*.

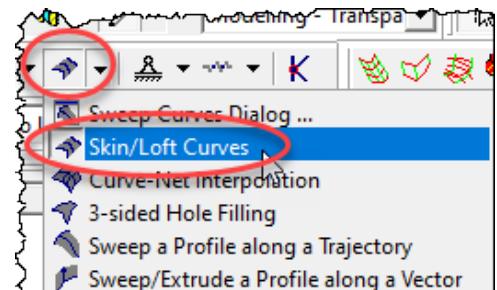
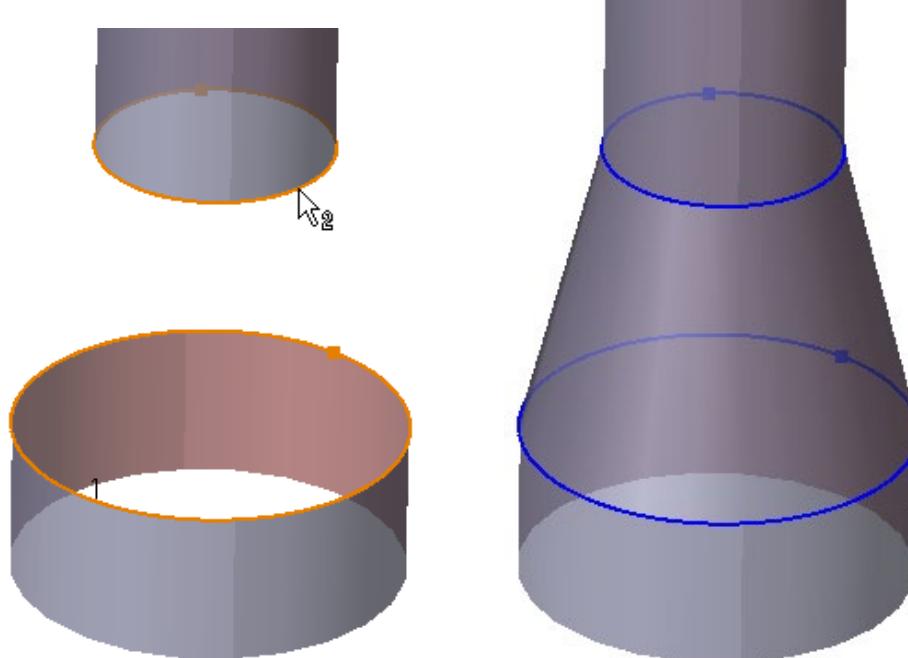


- Create a guide curve for the bottom edge of the pipe above the cone in the same way.



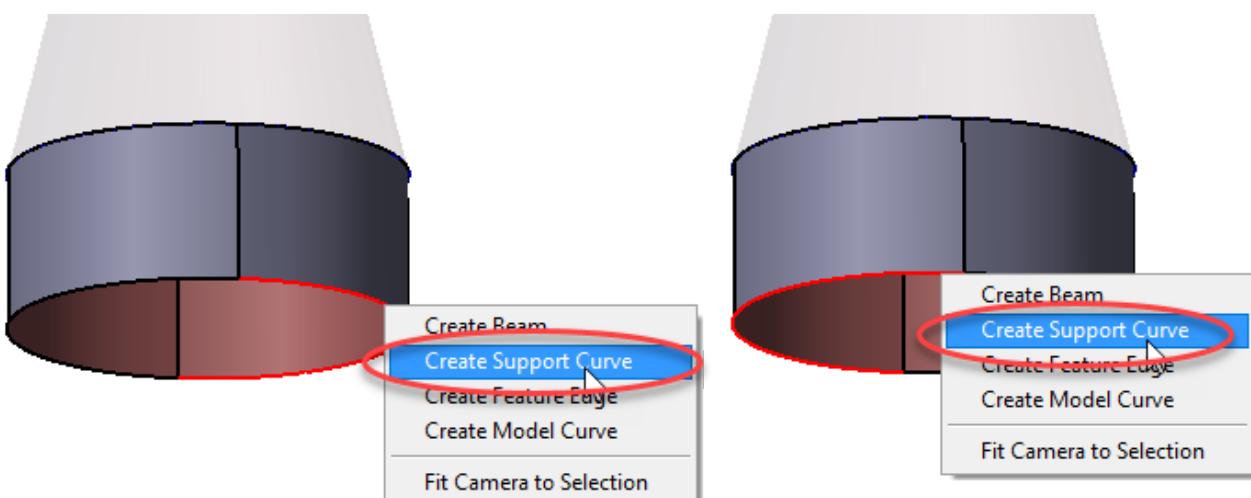
➤ Create a cone between the two guide curves by *Structure | Free Form Shells | Skin/Loft Curves* or by clicking the button shown to the right.

- Click the edge of the lower cylinder followed by double-clicking the edge of the upper cylinder as shown below. The double-clicking closes the creation – several guide curves may be selected.

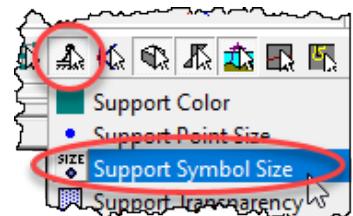


➤ Replace the support point at bottom of the lower cylinder by a support curve.

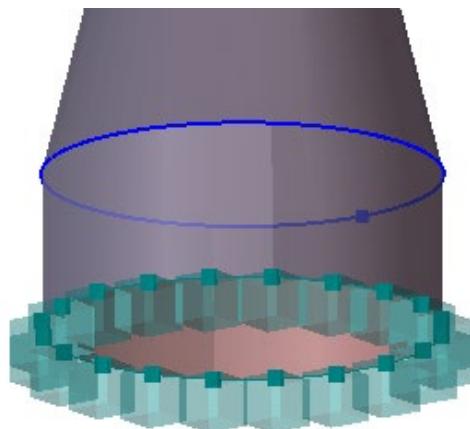
- Double-click the cylinder to select the bottom edge.
- A support curve is not allowed to be a full circle so do this in two operations.



- Double-click anywhere to return to normal display mode.
- Right-click the *Support selection* button to make supports visible again (open the eye).
- Possibly also reduce the *Support Symbol Size*.

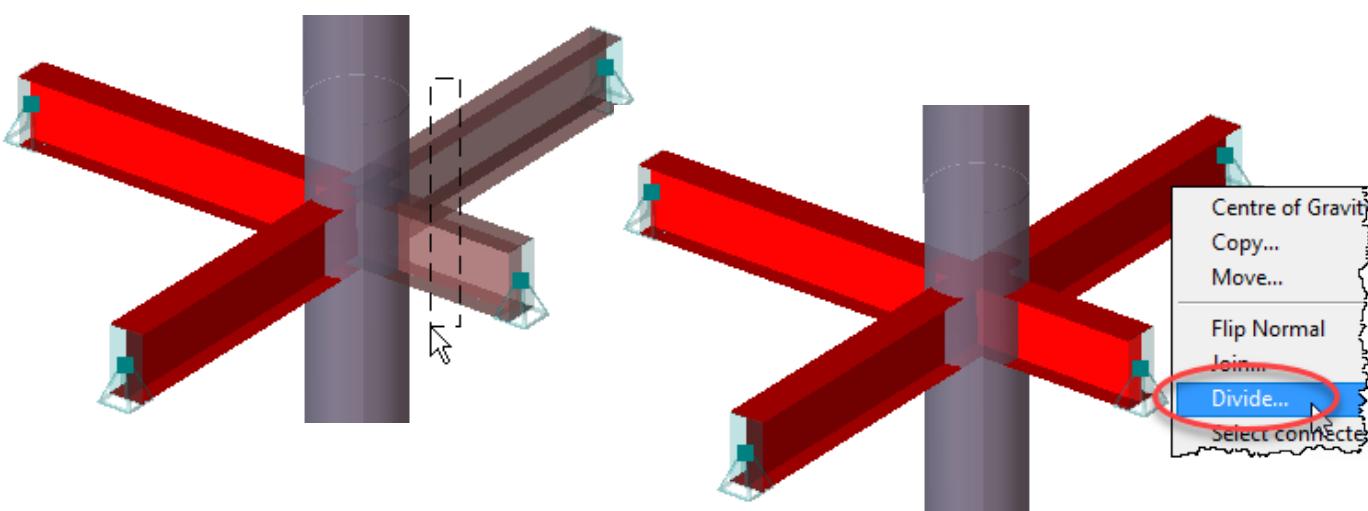


- Having done so, click the *Refresh graphics* button to make the change take effect.
- Delete the support point.

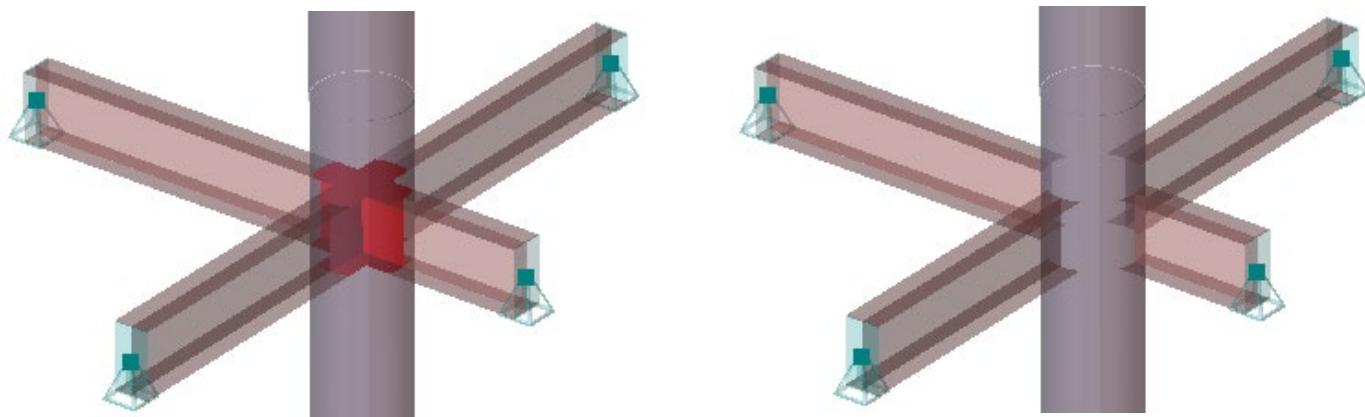


➤ Delete the parts of the horizontal I-beams extending into the vertical cylinder.

- To select the I-beams only, drag a rubberband from right to left touching only the I-beams to the left in the figure below followed by pressing the Shift button and dragging a rubberband from right to left touching only the I-beams to the right.
- Right-click and select *Divide* to *Explode all structure in selection into simpler parts*.



- Delete the parts of the I-beams inside the cylinder.
- To better see the parts inside the cylinder plates have been made 50% transparent by right-clicking the *Plate selection* button.



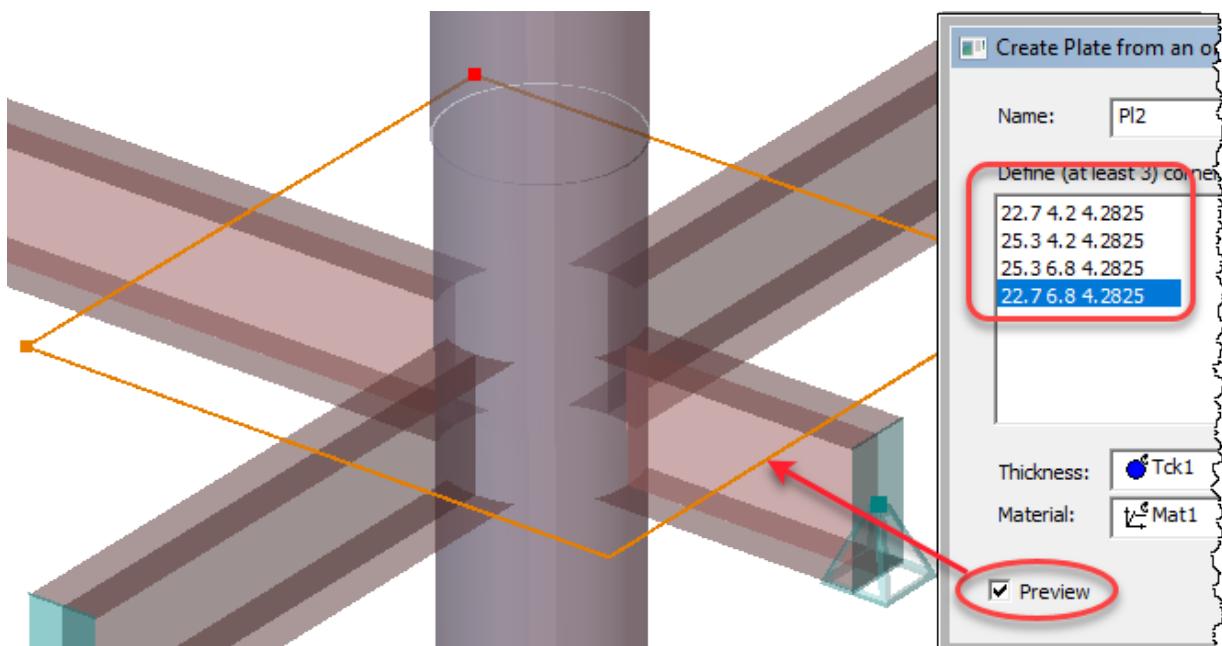
- Add plates as shown below to strengthen the joint. The plate thickness shall be 25 mm which is the same as the flange thickness of the beam cross section HE600A.

- Find that the thickness property named Tck1 is the one with 25 mm.
- Set Tck1 as default plate thickness.

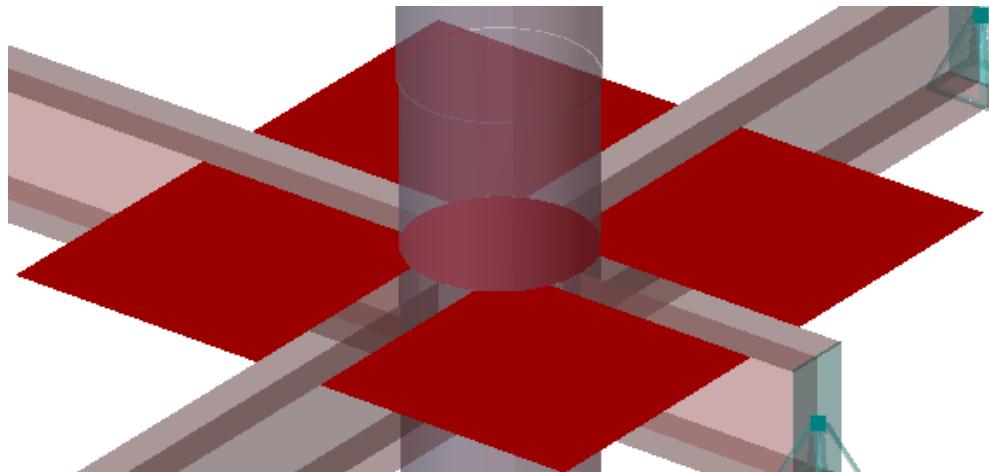
▾

Name	Description	Thickness [m]
Tck1	Thickness, t=0.025 m	0.025
Tck2	Thickness, t=0.013 m	0.013
Tck3	Thickness, t=0.035 m	0.035
Tck4	Thickness, t=0.015 m	0.015

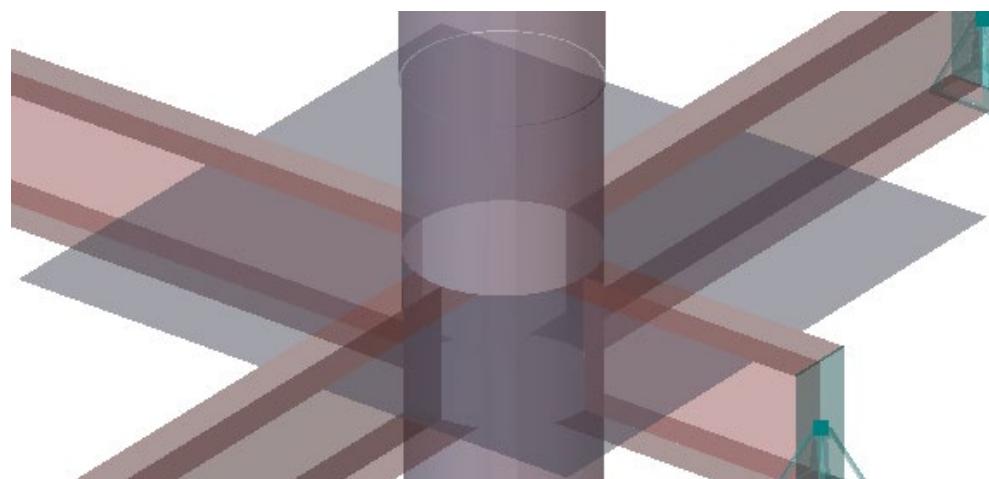
- First add a 2.6 m by 2.6 m plate at elevation Z = 4.2825 m extending in X-direction from 22.7 m to 25.3 and in Y-direction from 4.2 m to 6.8 m.



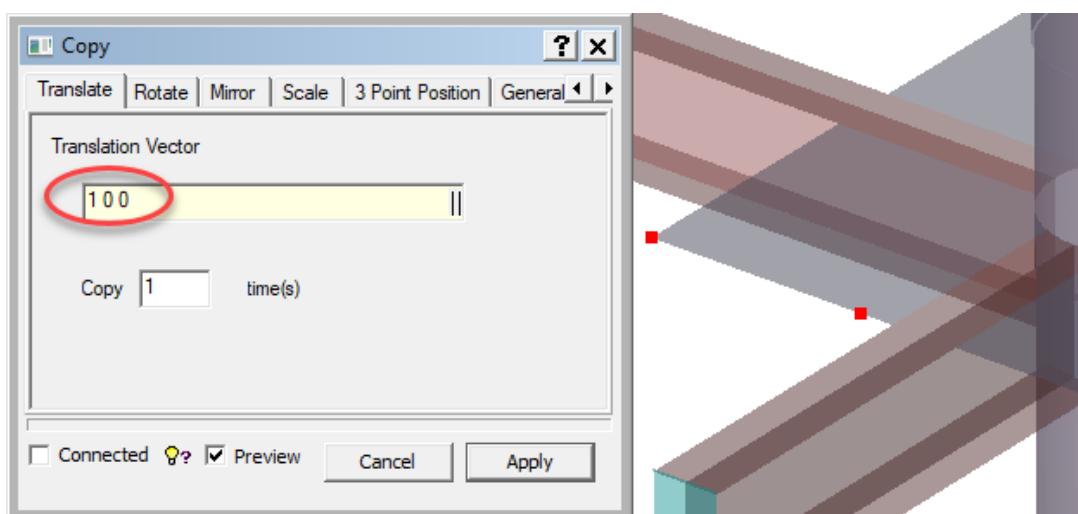
- Notice that where the new plate overlaps the existing top flanges of the I-beams no plates will be created thereby avoiding overlapping plates.



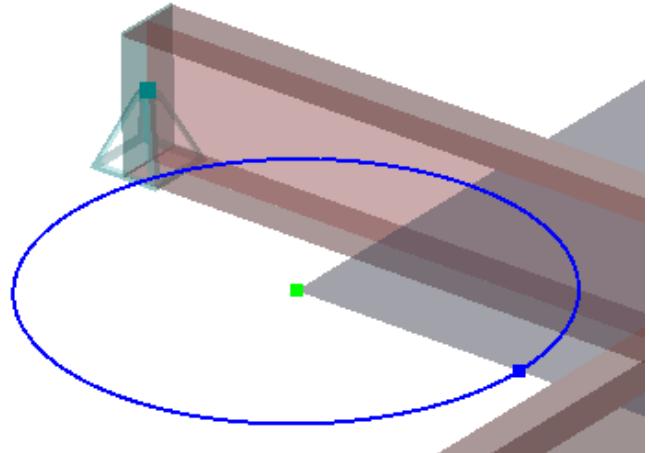
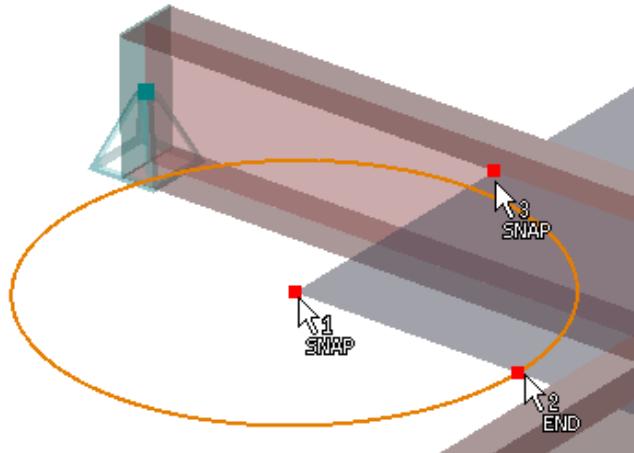
- Explode the new plate into simpler parts and delete the disk inside the column.



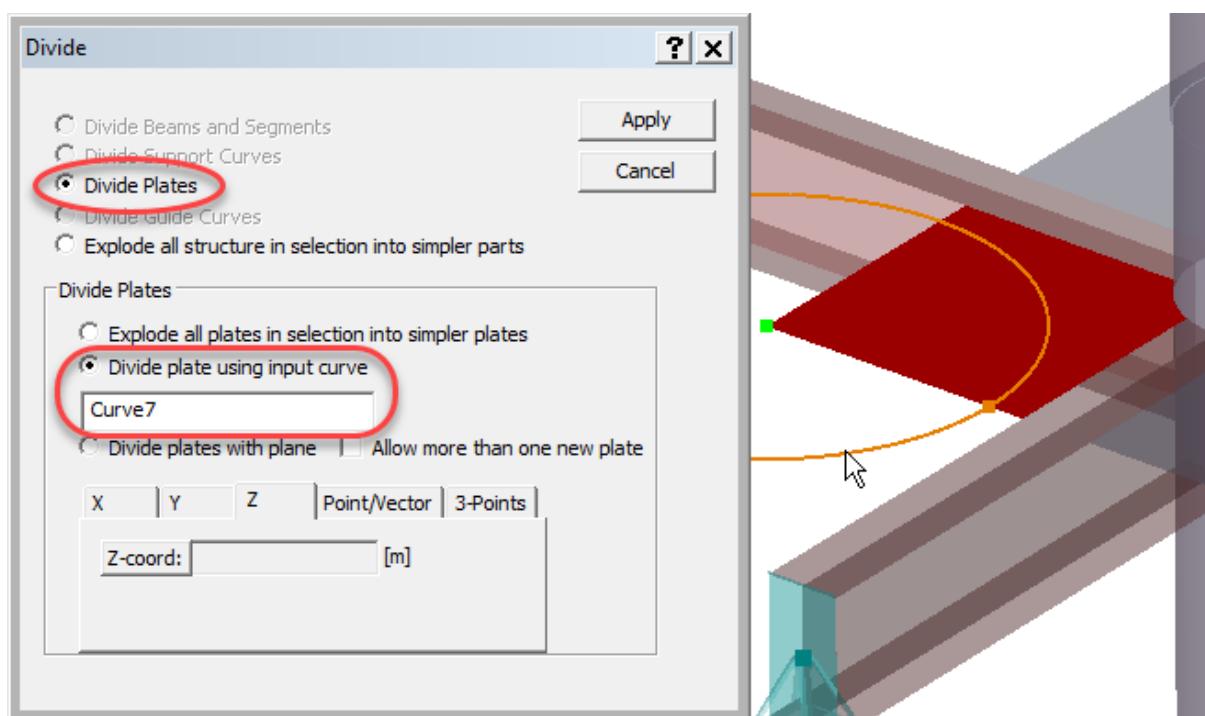
- Create a guide point at the corner (22.7, 4.2, 4.2825) of the new plate and copy this 1 m in X-direction.



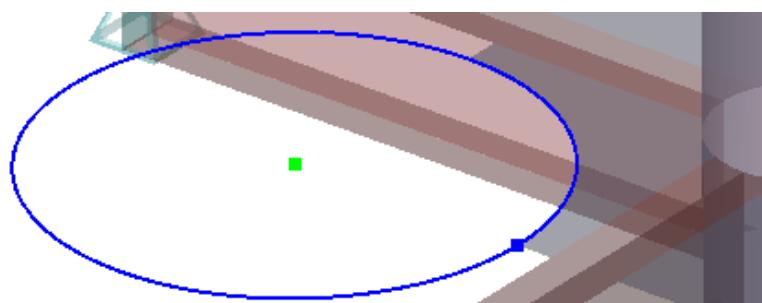
- Use the two guide points to create a guide circle by *Guiding Geometry | Conic Sections | Circle from Center and Radius* or click .



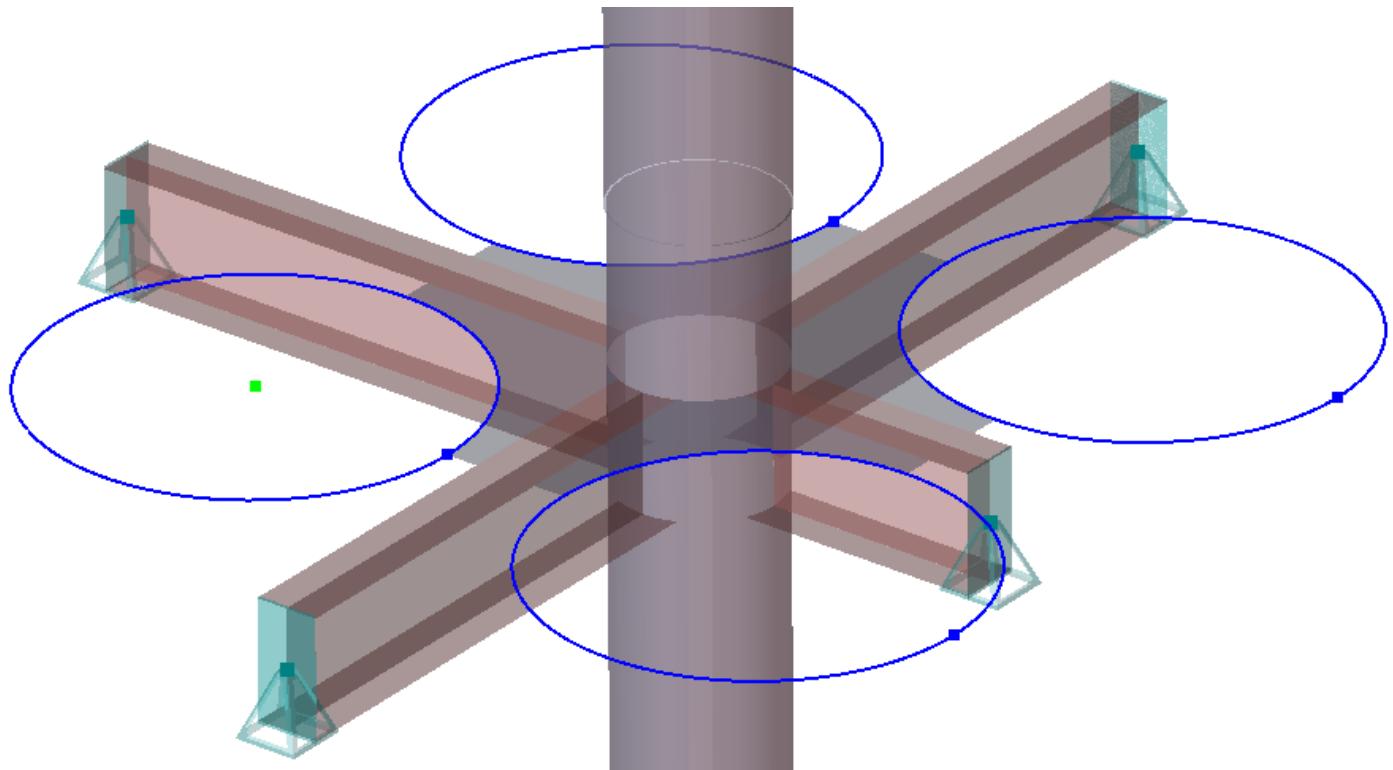
- Divide the plate using the guide circle.



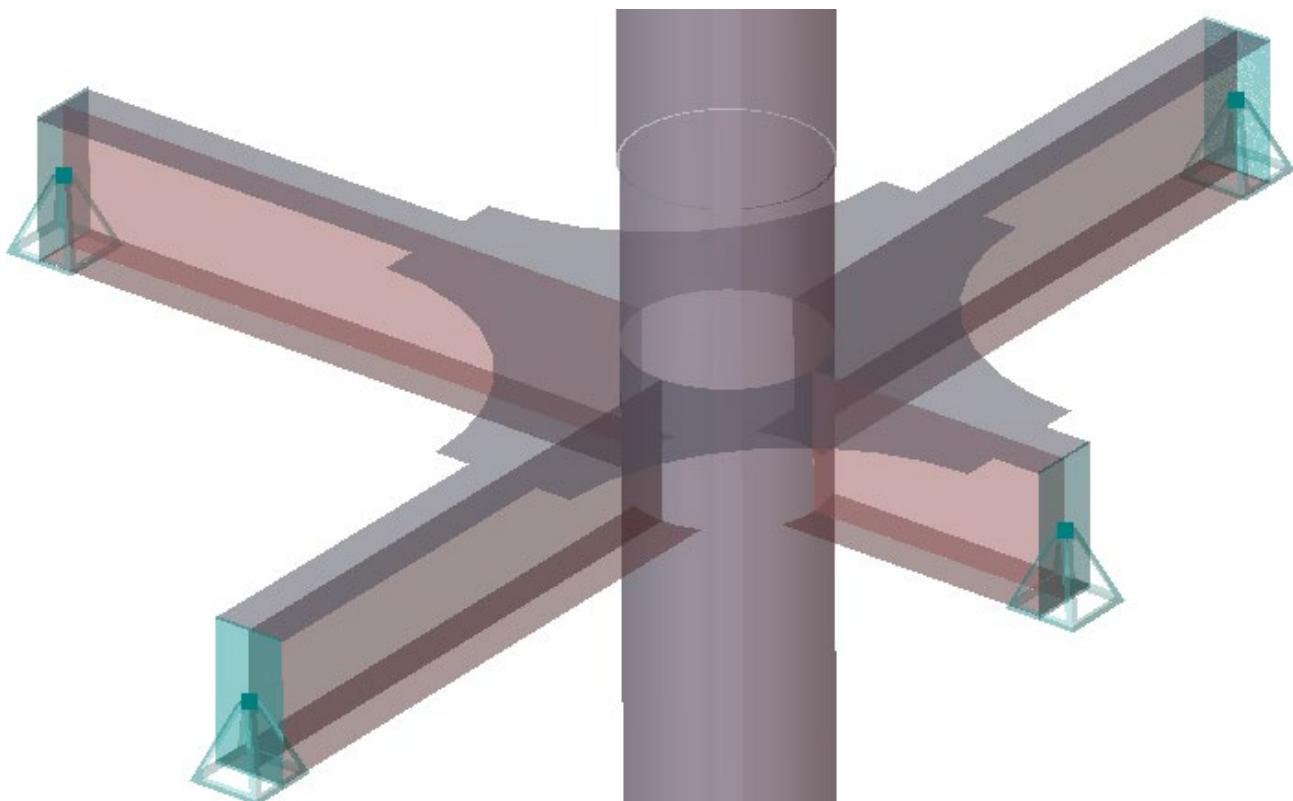
- Delete the part inside the guide circle.



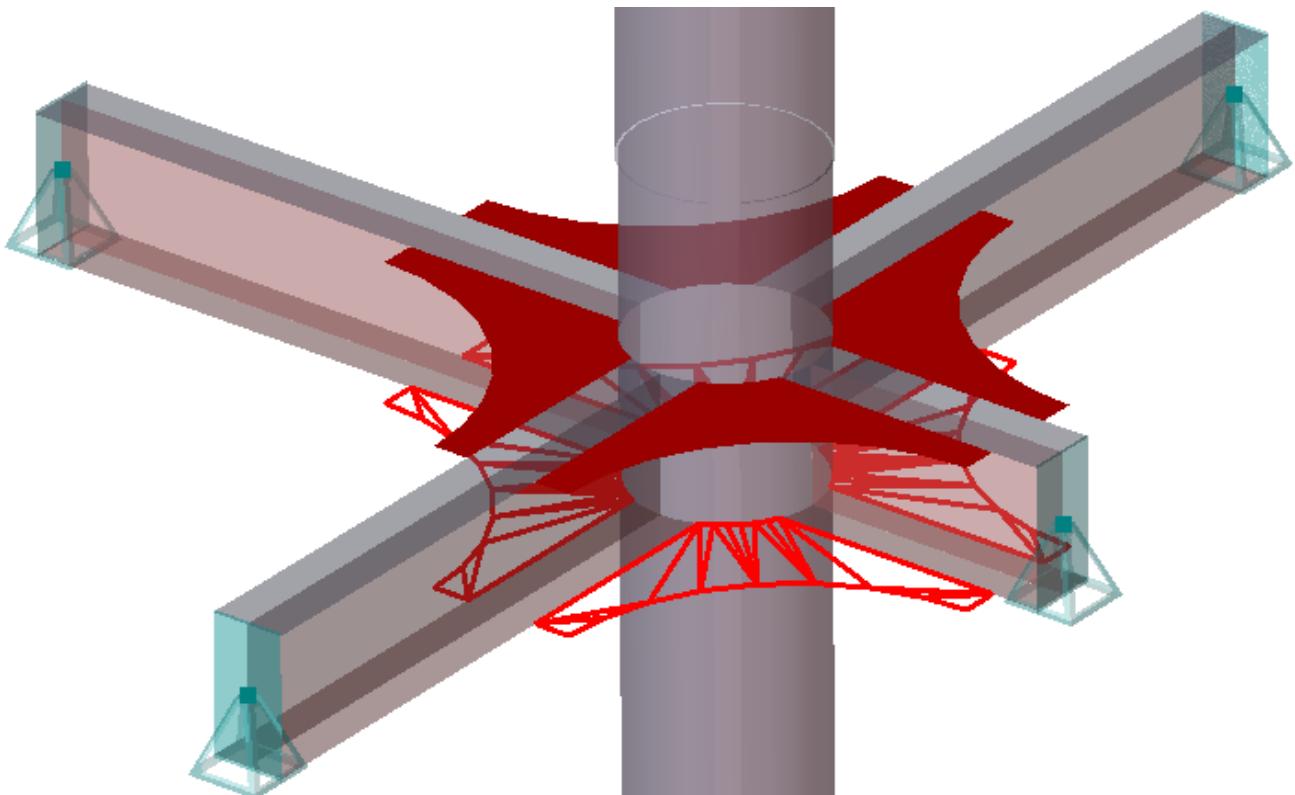
- Copy the guide circle and use this to make cutouts in the three other corners of the plate.



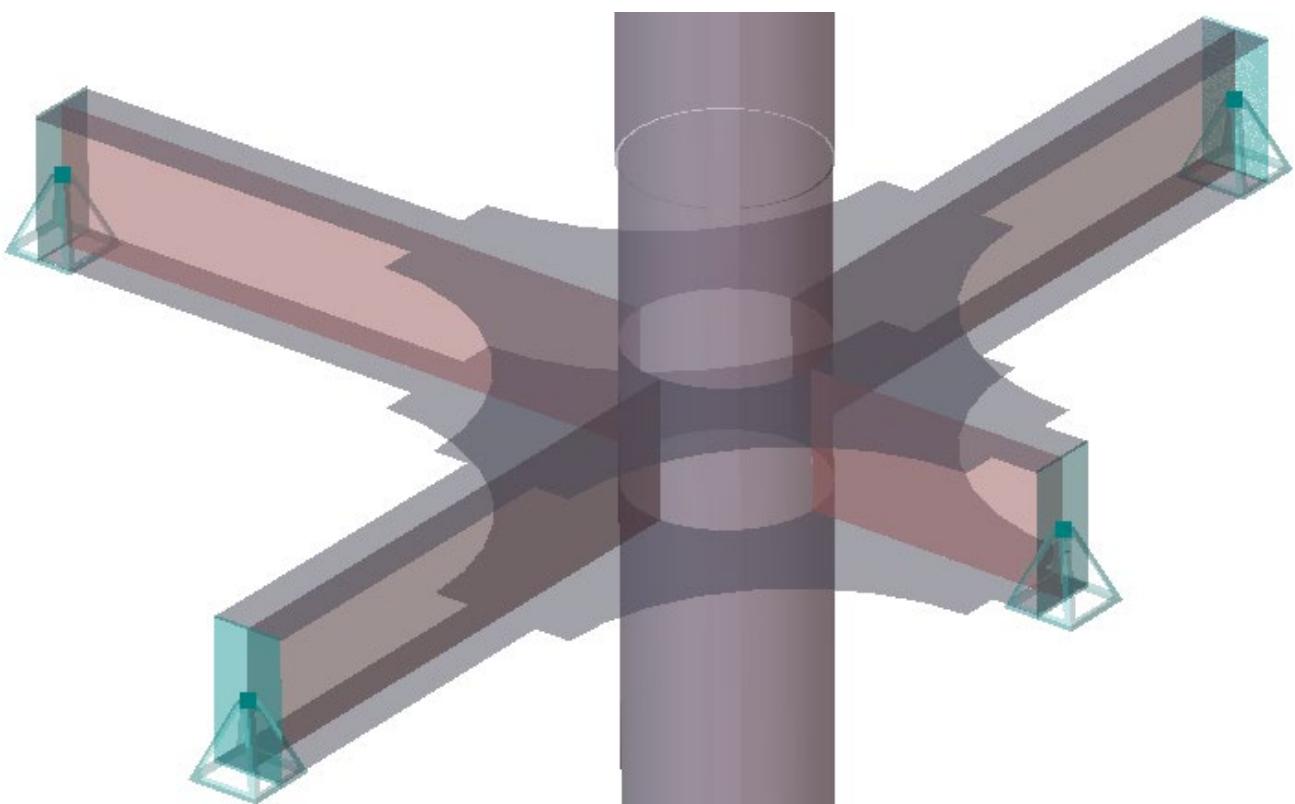
- As seen by the bluish and reddish colours the surface normals are inconsistent. Rectify this by flipping the normals of the top flanges.



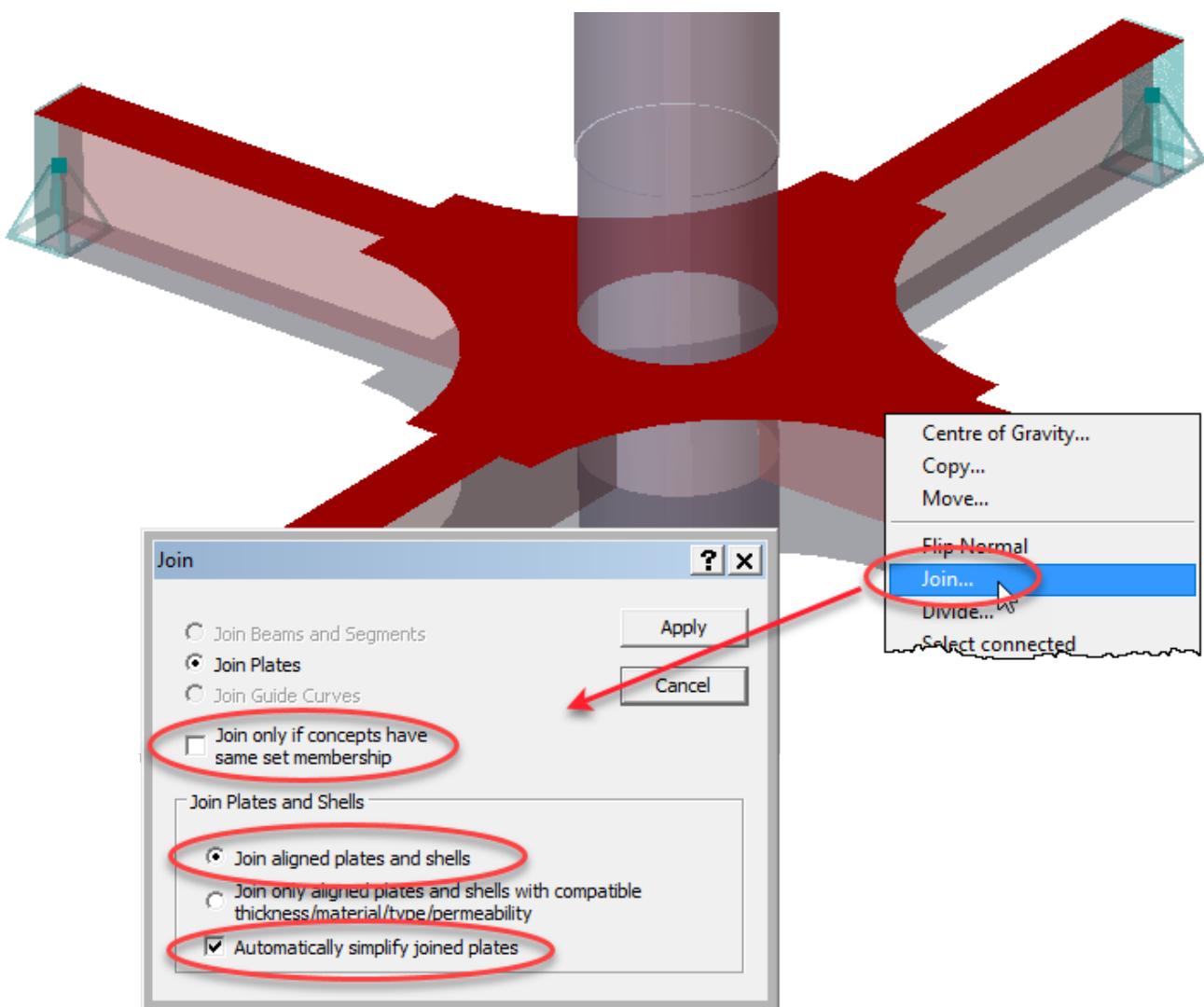
- Copy the four new strengthening plates from the top flange to the bottom flange.



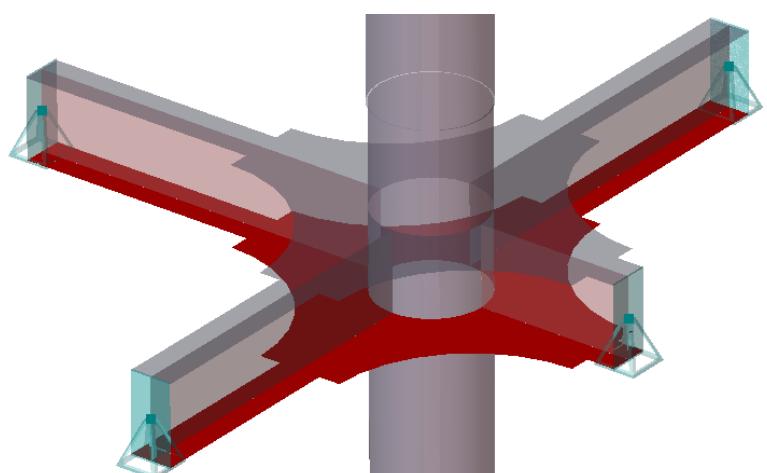
- Again, flip the surface normals of the flanges.



- Select the top flanges and the coplanar plates and join them into a single surface.
- In the *Join* dialog uncheck *Join only if concepts have same set membership*. This is required because the flanges, originating from beams of the cellar deck, are members of the set *Cellar\_deck* while the new plates are not.



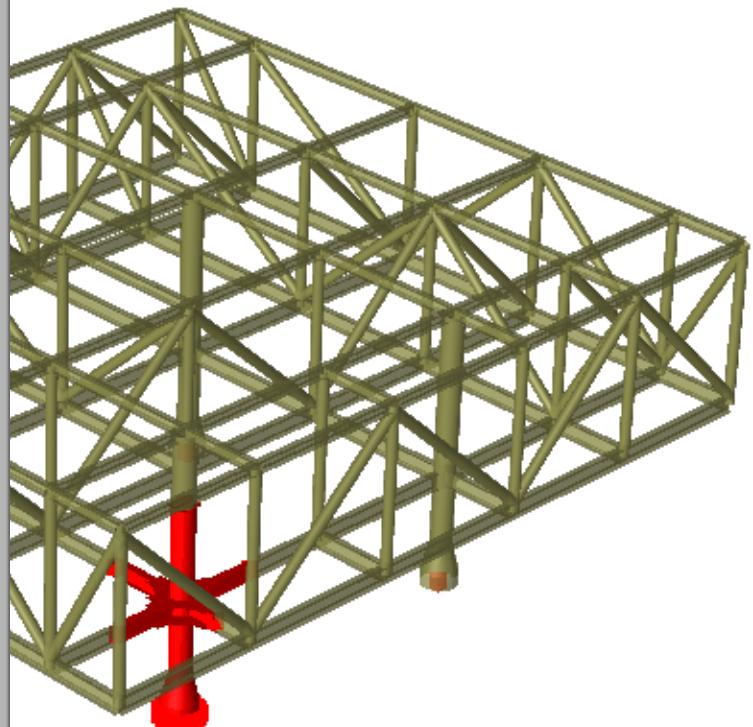
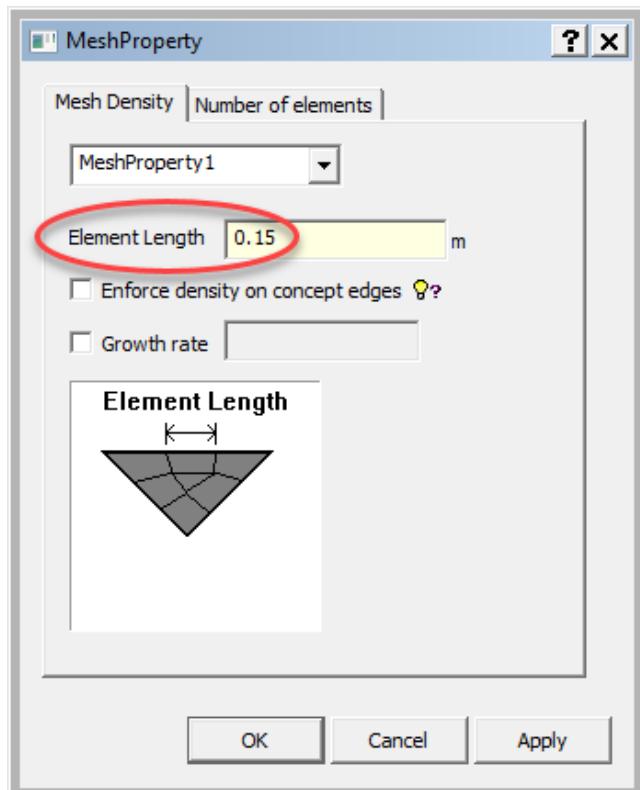
- Do the same for the bottom flange and coplanar plates.
- The modelling is at this point complete.
- Use *Structure | Topology | Verify Model* to verify the model and if there are any problems, correct them.



## 17 CREATE FE MESH

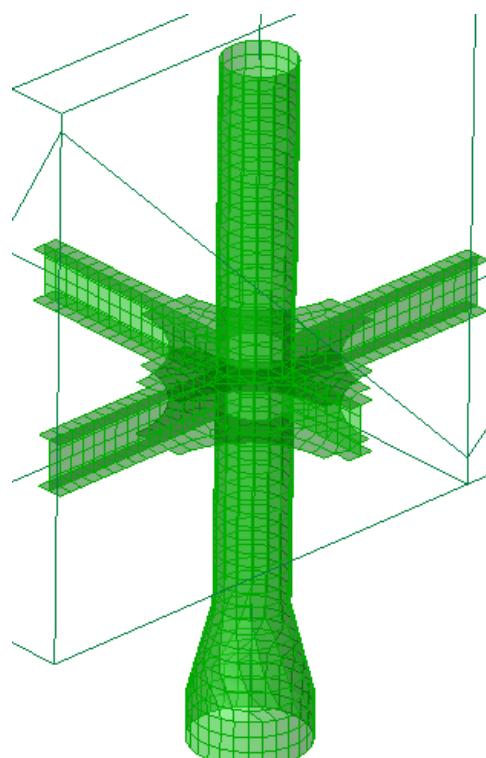
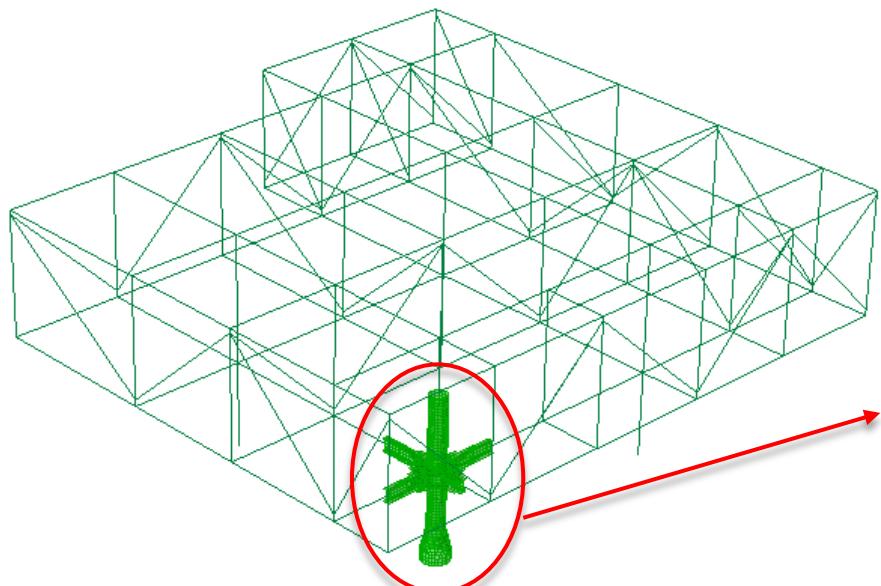
- Right-click *Properties | Mesh* and click *New Mesh Property* to create a mesh property and assign this to the plates and shell.

- Select only plates and shells by lifting the *beam selection button* .



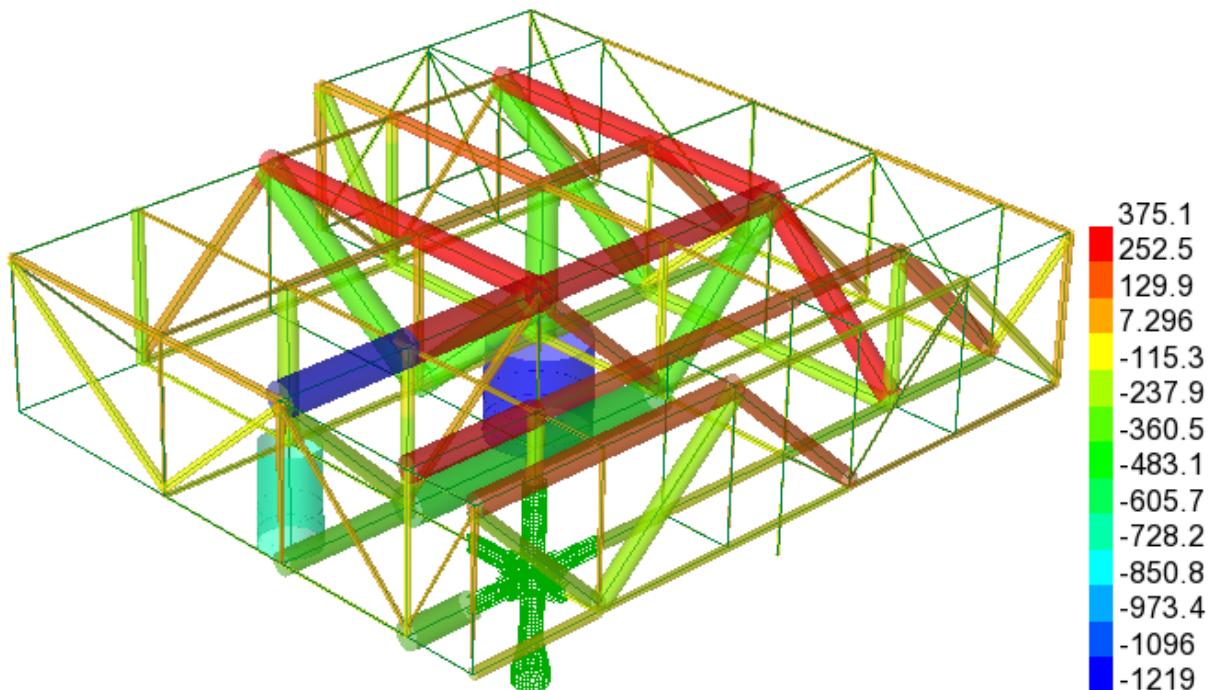
- Use Alt+M to create a meshing activity and run it.

- Normally an effort will be put in to control the FE mesh of the joint but in this case we accept it as is.

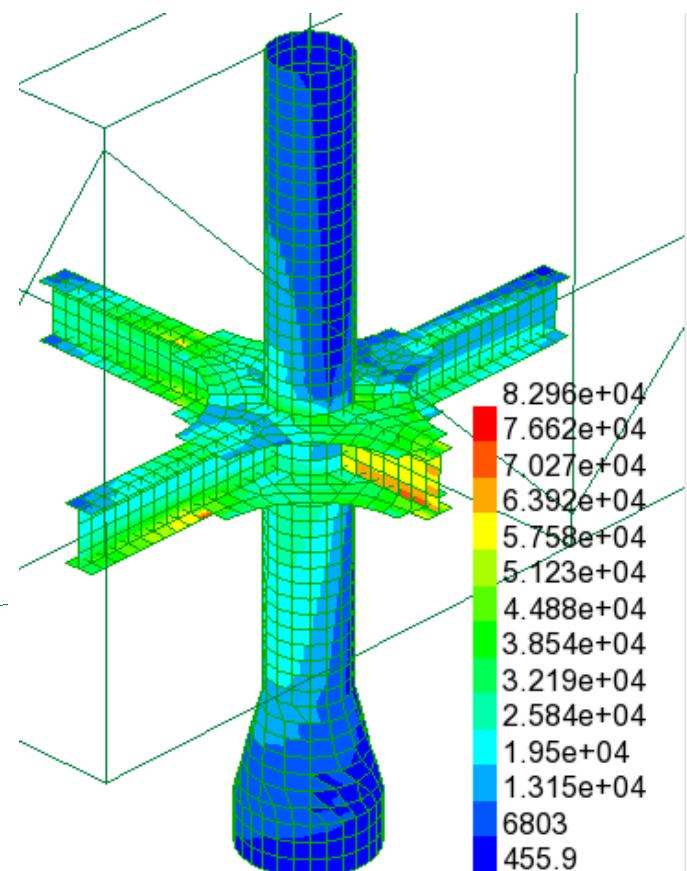


## 18 STATIC ANALYSIS OF COMBINED BEAM AND PLATE/SHELL MODEL

- Use Alt+D to create an analysis activity and run it.
- Switch to *Results - with Mesh* display configuration and present results.
  - For example *Beam Forces* and *Nxx* as shown below.



- And von Mises stress in the middle surface as shown to the right for the joint.
- The deformed model below show that the coupling between the beams and plate/shell model provided by the 'support rigid links' properly transfers moments from one to the other.



## About DNV

We are the independent expert in risk management and quality assurance. Driven by our purpose, to safeguard life, property and the environment, we empower our customers and their stakeholders with facts and reliable insights so that critical decisions can be made with confidence. As a trusted voice for many of the world's most successful organizations, we use our knowledge to advance safety and performance, set industry benchmarks, and inspire and invent solutions to tackle global transformations.

## Digital Solutions

DNV is a world-leading provider of digital solutions and software applications with focus on the energy, maritime and healthcare markets. Our solutions are used worldwide to manage risk and performance for wind turbines, electric grids, pipelines, processing plants, offshore structures, ships, and more. Supported by our domain knowledge and Veracity assurance platform, we enable companies to digitize and manage business critical activities in a sustainable, cost-efficient, safe and secure way.