

SESAM TUTORIAL

# GeniE

## Curved Structure Modelling – Crane Pedestal

Valid from program version 8.2

---



## Sesam Tutorial

GeniE – Curved Structure Modelling – Crane Pedestal

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

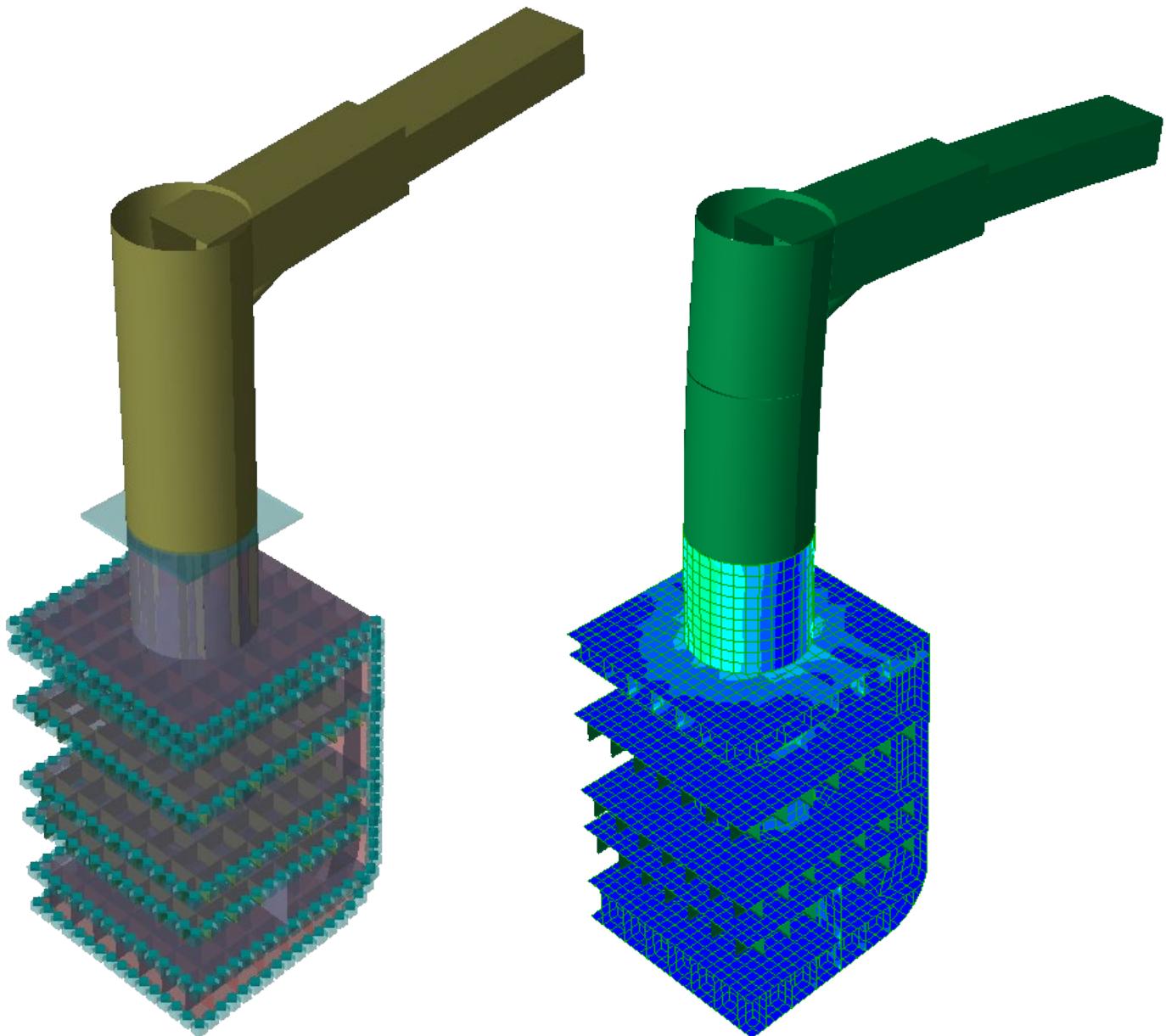
This publication or parts thereof may not be reproduced or transmitted in any form or by any means, including copying or recording, without the prior written consent of DNV AS.

## TABLE OF CONTENTS

1. Introduction	Page 4
2. Units, Material, Cross Sections, Plate Thicknesses	Page 5
3. Guiding Geometry	Page 12
4. Create Outer Hull	Page 16
5. Create Deck Plates	Page 18
6. Create Vertical Stiffener Plates	Page 20
7. Create Stiffener Beams for Decks	Page 25
8. Create Column	Page 28
9. Create Web Frames	Page 34
10. Create Column Stiffeners	Page 36
11. Create Crane	Page 40
12. Boundary Conditions	Page 46
13. Analysis Activity and Loads	Page 49
14. Verify Model	Page 53
15. Run Analysis with Coarse Mesh	Page 54
16. Run Analysis with Finer Mesh	Page 55
17. Create Sets and Run Analysis with even Finer Mesh	Page 56

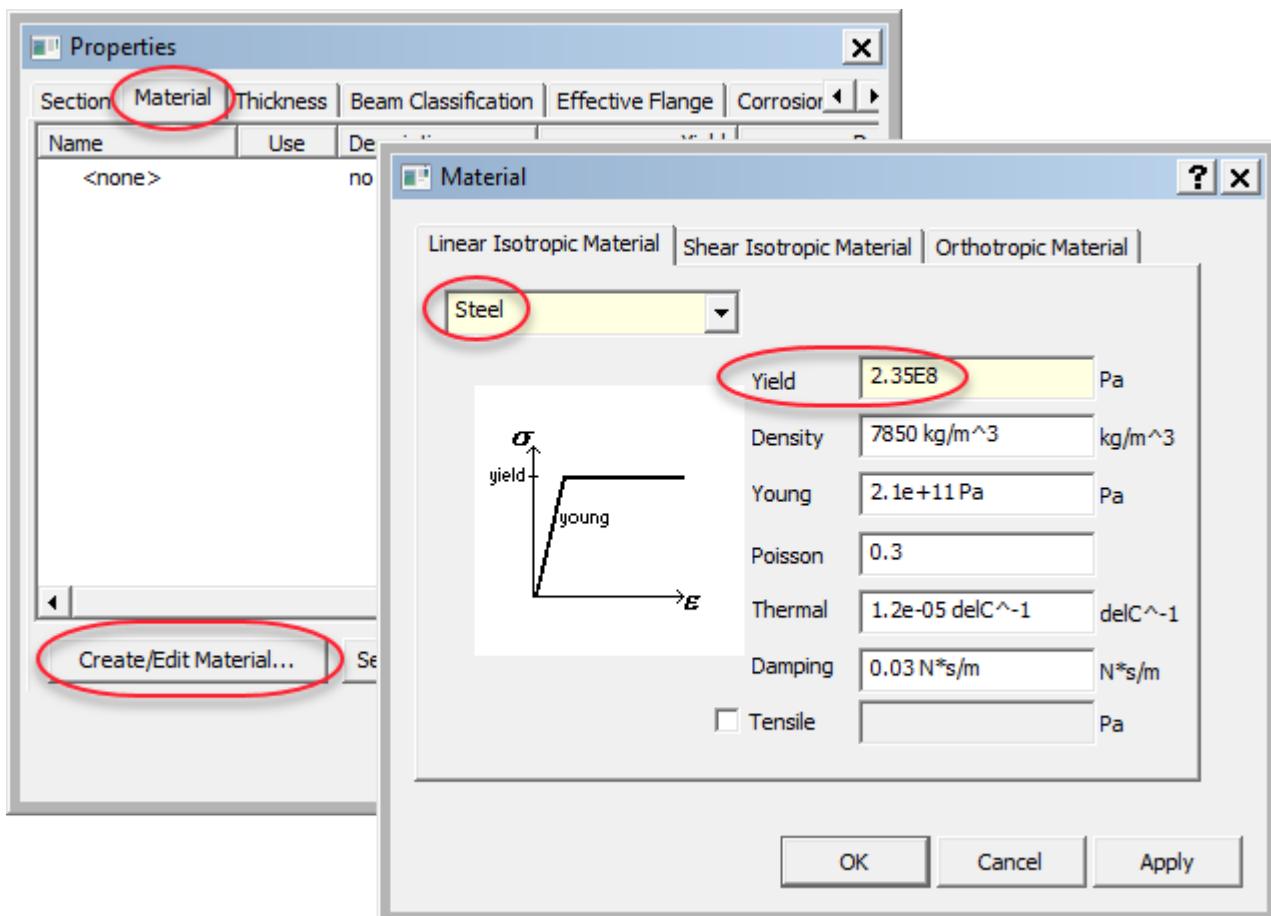
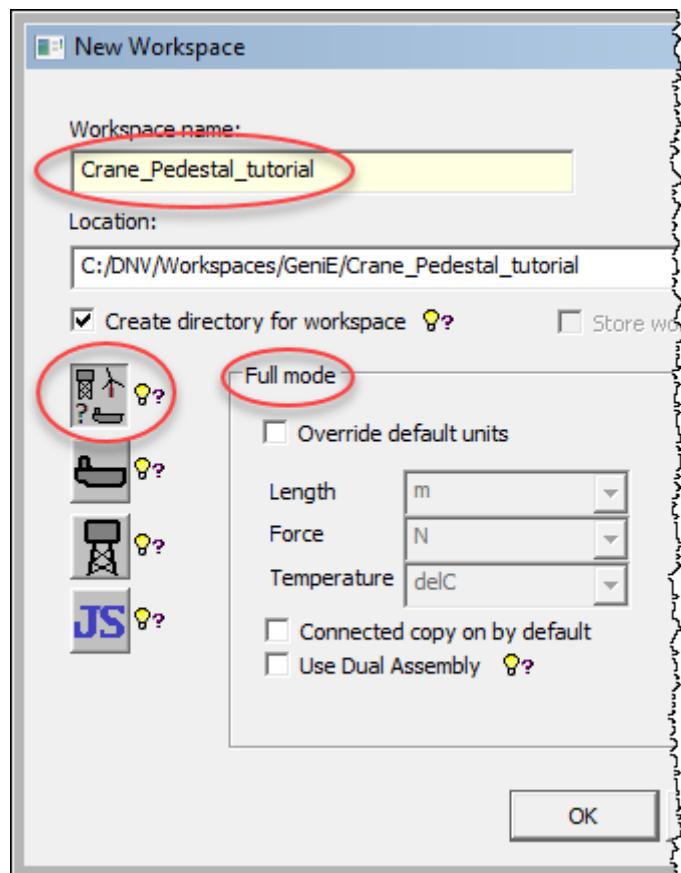
## 1 INTRODUCTION

- In this tutorial a crane pedestal installed on a floating vessel is modelled for stress analysis.
- Focus is on creating the geometry rather than on loads and detailed FE mesh control. There are two load cases; self weight and crane load. In addition, there is a load combination accounting for dynamic amplification of the crane load.
- The tutorial presuppose some basic knowledge in GeniE's GUI.
- A GeniE input file for creating the model is provided.
- The appearance of the GUI and dialogs in later versions of GeniE may change.

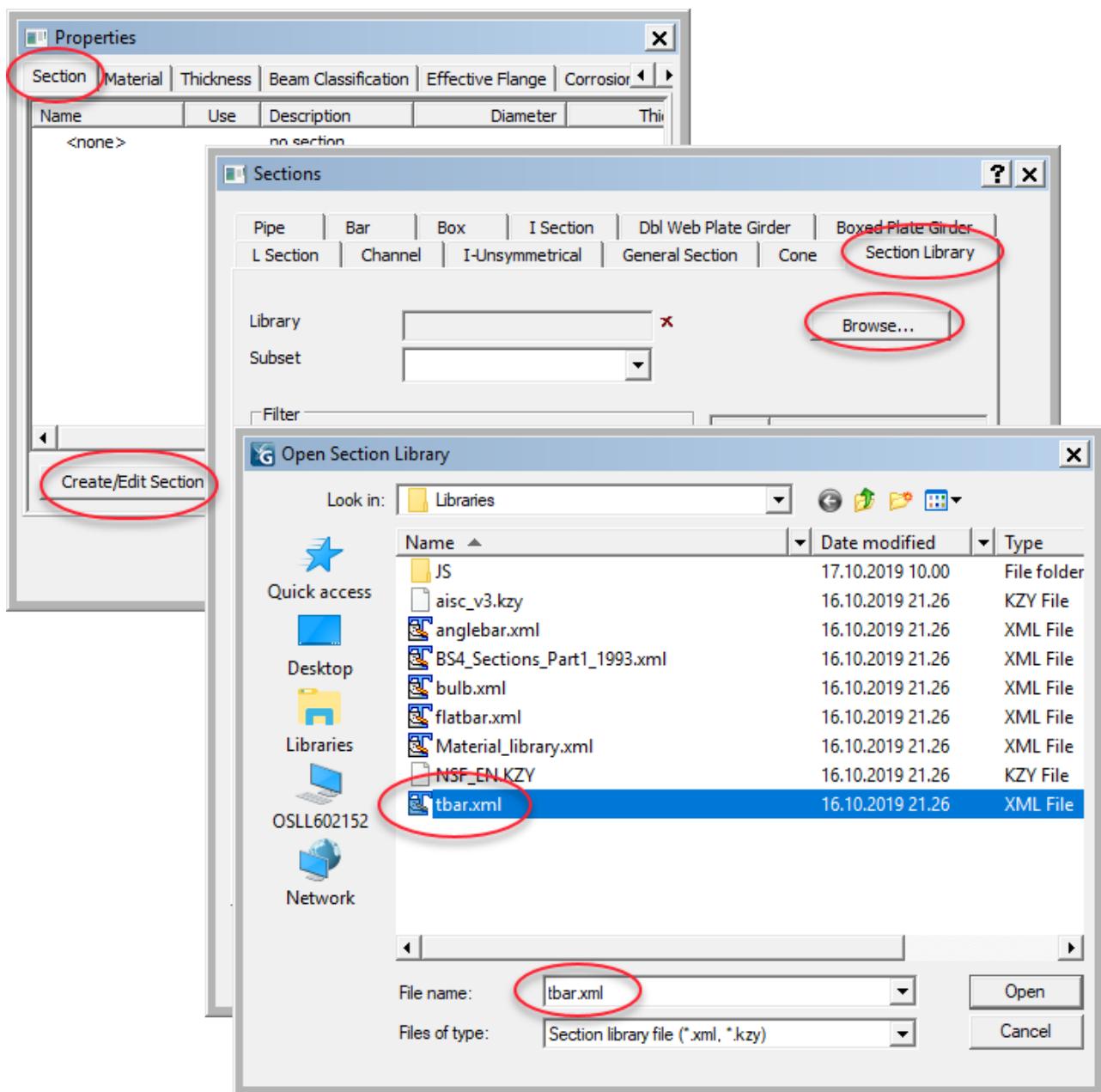


## 2 UNITS, MATERIAL, CROSS SECTIONS, PLATE THICKNESSES

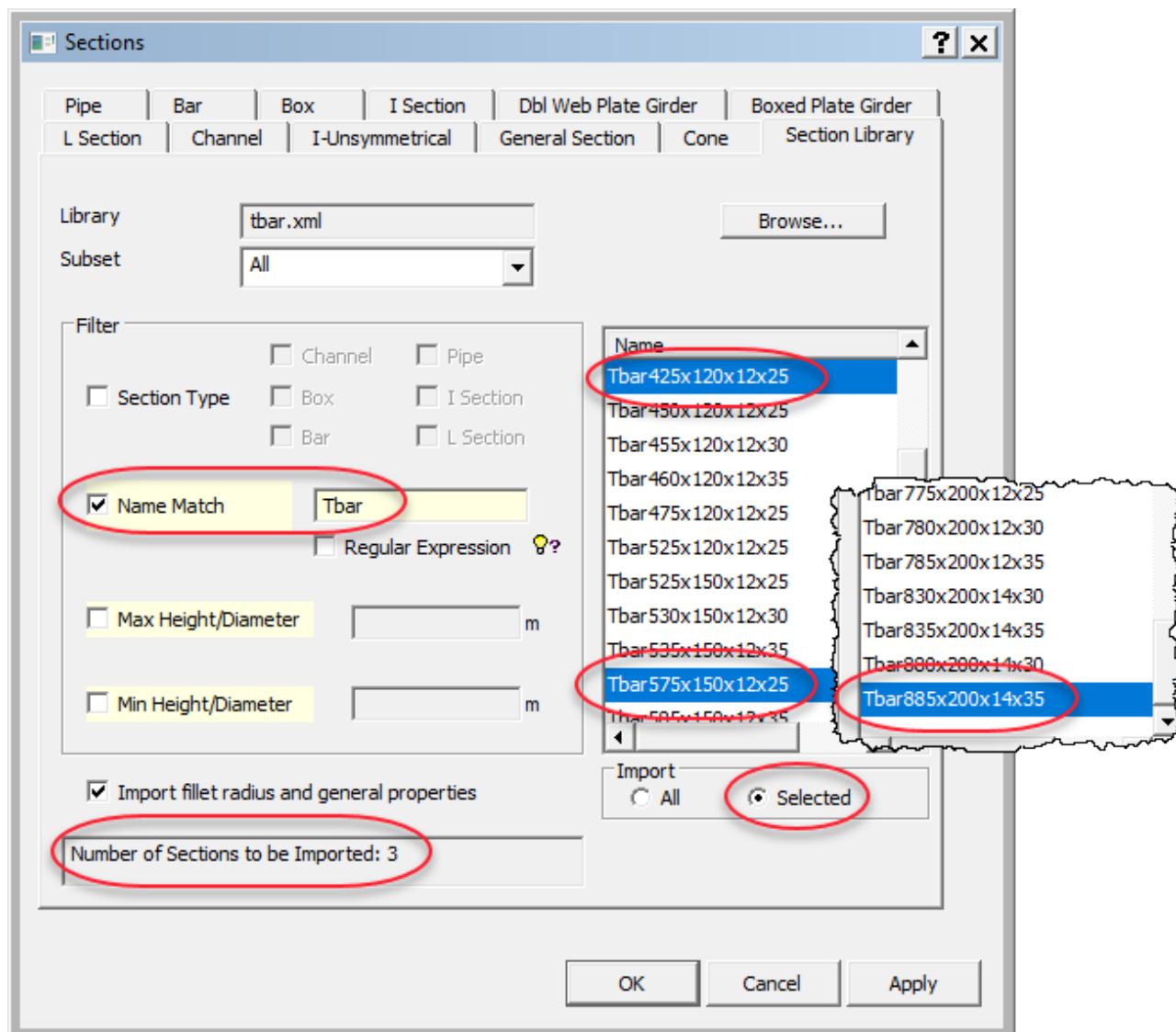
- Start GeniE and open a new workspace.
  - Give a *Workspace name*.
  - Accept default units m and N and click **OK**.
    - Unless otherwise specified, all values in this tutorial are in these units.
  - Make sure *Full mode* is selected as this tutorial involves curved modelling.
- Define steel material.
  - Use *Edit | Properties* to open the *Properties* dialog.
  - In the *Material* tab click *Create/Edit Material*.
  - In the *Material* dialog give a material name and a *Yield* value. Accept default values and click **OK**.



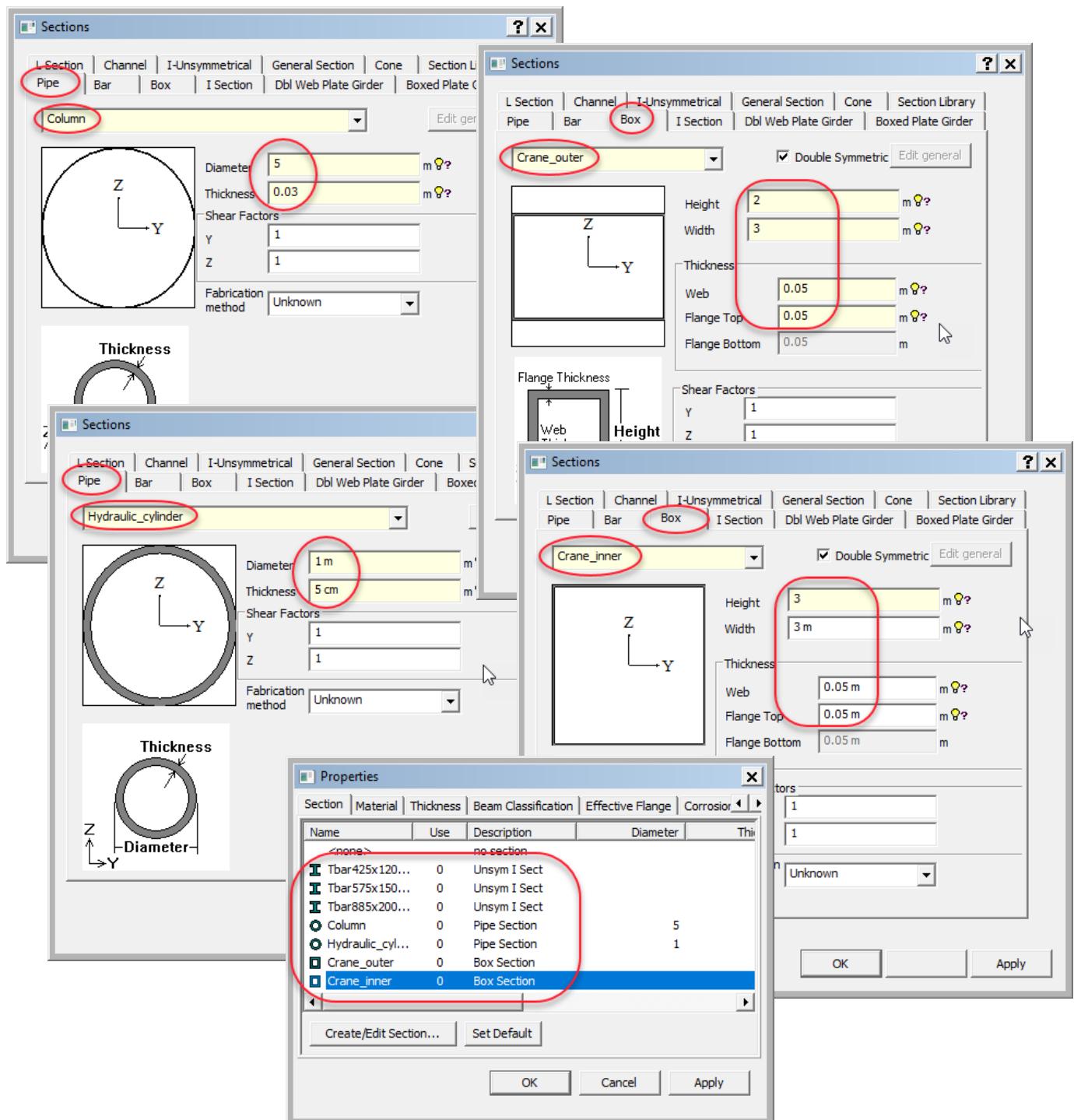
- Define beam cross section properties by importing from a library.
  - Use *Edit | Properties* to open the *Properties* dialog. In the *Section* tab click *Create/Edit Section* to open the *Sections* dialog.
  - In the *Sections* dialog go to the *Section Library* tab and click *Browse* and find the beam cross section library named *tbar.xml* (part of the GeniE installation).
  - Click *Open* and see several sections listed.



- We want to import the following Tbar sections into our workspace:
  - Tbar425x120x12x25
  - Tbar575x150x12x25
  - Tbar885x200x14x35
- To ease the selection check *Name Match* and enter Tbar and see that only sections with name containing the string Tbar are listed.
- Having made the selection make sure the *Import* radio button *Selected* is chosen and see the text *Number of Sections to be Imported:* 3 appears.
- Click *OK* and see the three sections appear in the *Properties* dialog.

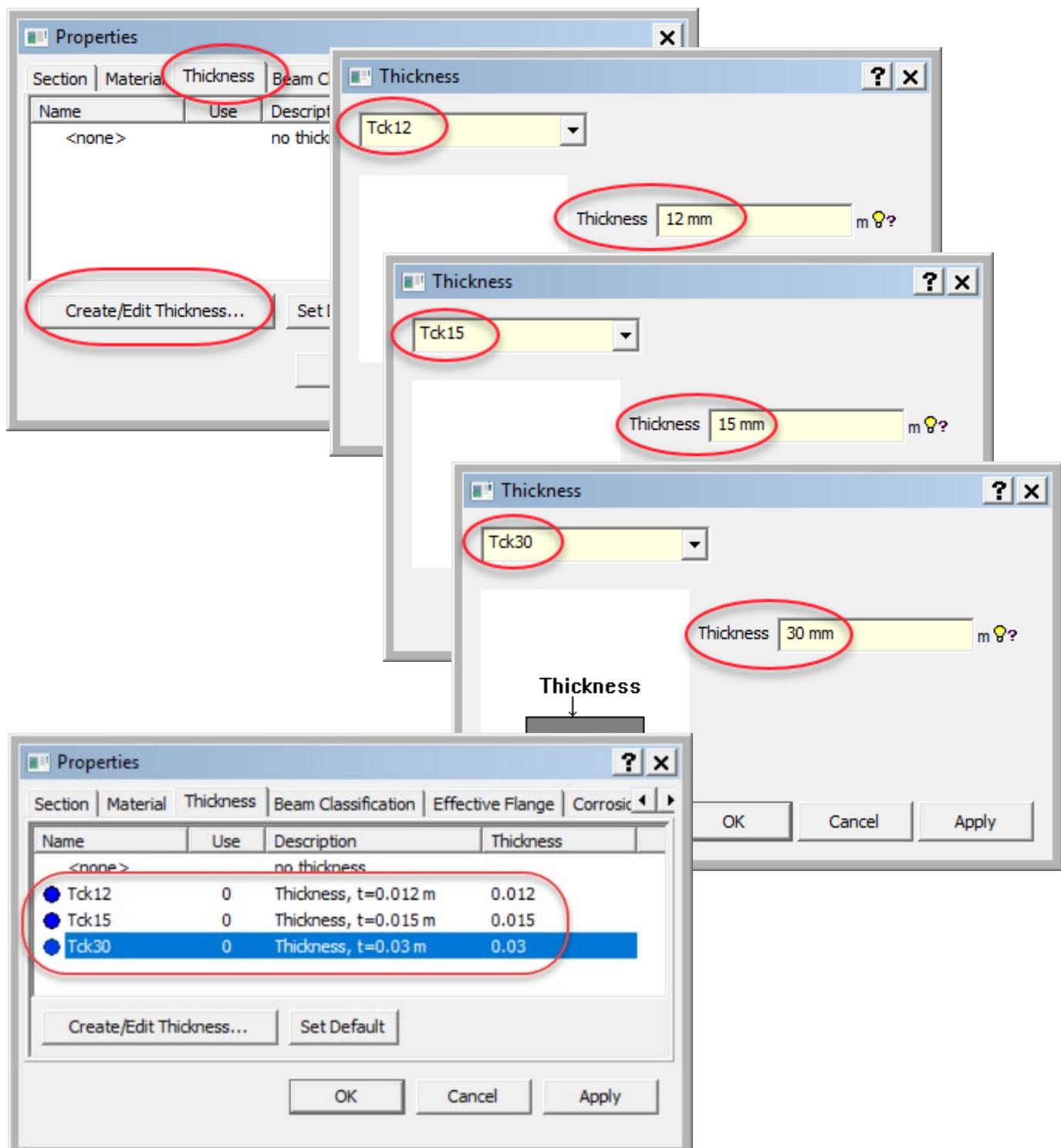


- Again, click *Create/Edit Section* in the *Section* tab of the *Properties* dialog to open the *Sections* dialog. Now create beam cross section properties manually:
  - The two *Pipe* sections *Column* and *Hydraulic\_cylinder* with data as given below
  - The two *Box* sections *Crane\_outer* and *Crane\_inner* with data as given below
- See all imported and manually created sections listed in the *Section* tab of the *Properties* dialog as shown below.



➤ Define three plate thicknesses Tck12, Tck15 and Tck30.

- Use *Edit | Properties* to open the *Properties* dialog. In the *Thickness* tab click *Create/Edit Thickness* to open the *Thickness* dialog.
- Create the plate thicknesses with data as shown below.
- See all manually created thicknesses listed in the *Thickness* tab of the *Properties* dialog as shown below.



- The material, beam cross section and plate thickness properties are found in the browser and may accessed from there.
- Now set default material, section and thickness. I.e. properties that will be assigned to beams and plates/shells at their creation. This can be done by right-clicking as shown below for Tbar885x200x14x35. This can also be done through the *Default Properties* toolbar as shown further below.

  - Set material Steel, section Tbar885x200x14x35 and thickness Tck15 as default.
  - Notice the small check mark in the browser for Tck15 indicating this is default thickness.

The screenshot illustrates the process of setting default properties in GeniE. It shows the software's interface with several windows and toolbars.

**Browser Tree:** On the left, the 'Crane\_Pedestal\_Tutorial' project tree is visible. Nodes under 'Properties' are circled in red: 'Materials', 'Sections', and 'Thicknesses'. A red arrow points from the 'Materials' node to the top table.

**Material Table:** A table showing material properties. The 'Steel' entry is selected.

Name	Description
Steel	Material, lin. isotropic, E=2.1e+11 Pa, yield=235000000 Pa, dens=7850

**Section Table:** A table showing beam cross section properties. The 'Tbar885x200x14x35' entry is selected.

Name	Description
Column	Pipe Section, d=5 m, t=0.03 m
Crane_inner	Box Section, h=3 m, w=3 m, wt=0.05 m, ft=0.05 m
Crane_outer	Box Section, h=2 m, w=3 m, wt=0.05 m, ft=0.05 m
Hydraulic_cylinder	Pipe Section, d=1 m, t=0.05 m
Tbar425x120x12x25	TYPE 43 xx Tbar WeldedTbar
Tbar575x150x12x25	TYPE 43 xx Tbar WeldedTbar
<b>Tbar885x200x14x35</b>	<b>TYPE 43 xx Tbar WeldedTbar</b>

**Context Menu:** A context menu is open over the selected section. The 'Set Default' option is highlighted and circled in red.

**Thickness Table:** A table showing plate thickness properties. The 'Tck15' entry is selected.

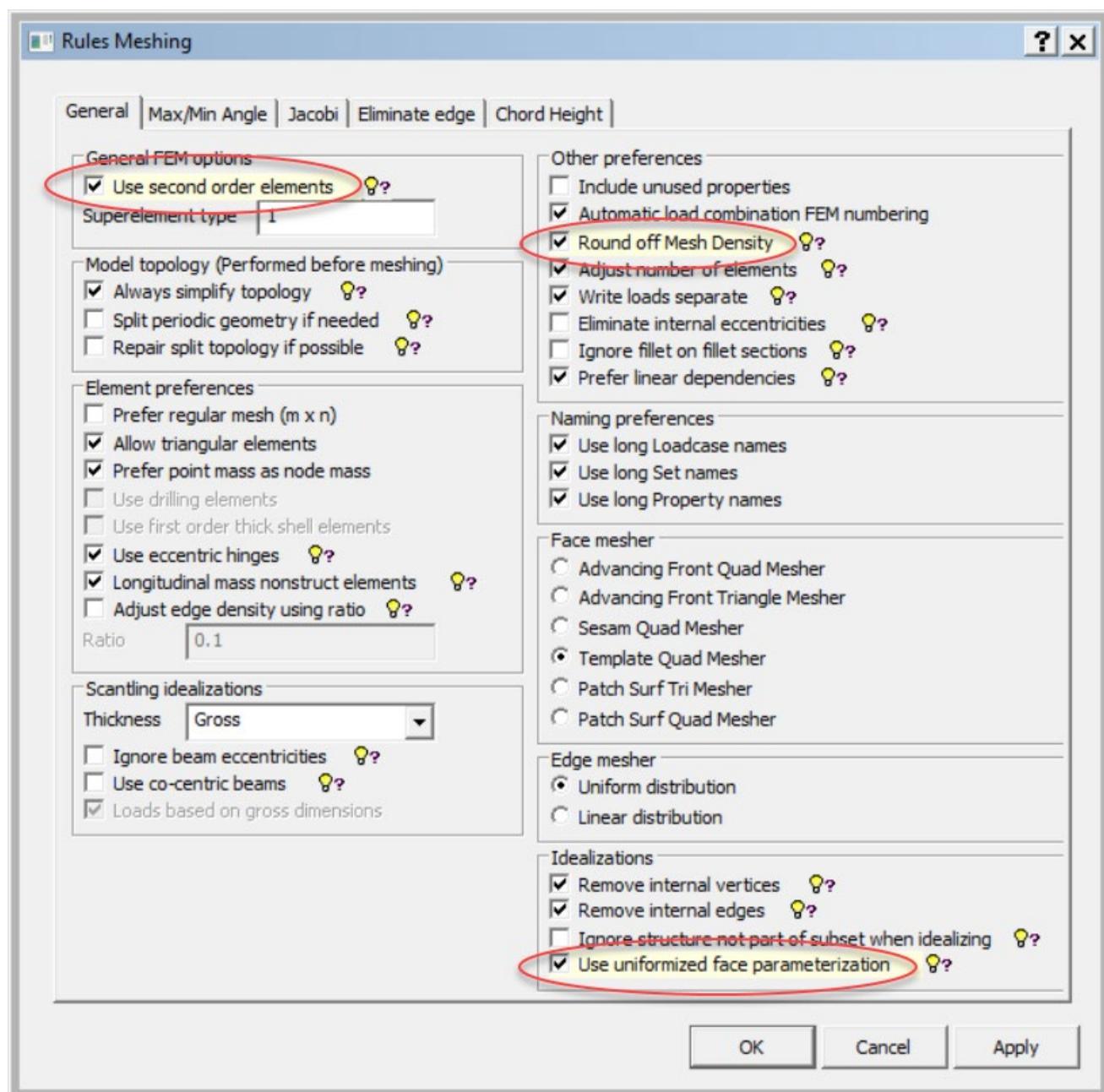
Name	Description	Thickness [m]
Tck12	Thickness, t=0.012 m	0.012
<b>Tck15</b>	<b>Thickness, t=0.015 m</b>	<b>0.015</b>
Tck30	Thickness, t=0.03 m	0.03

**Default Properties Toolbar:** At the bottom, the toolbar shows the selected section ('Tbar885x200x14x35'), material ('Steel'), and thickness ('Tck15').

**Thickness Table (Toolbar):** A smaller table showing the thickness list, with 'Tck15' circled in blue.

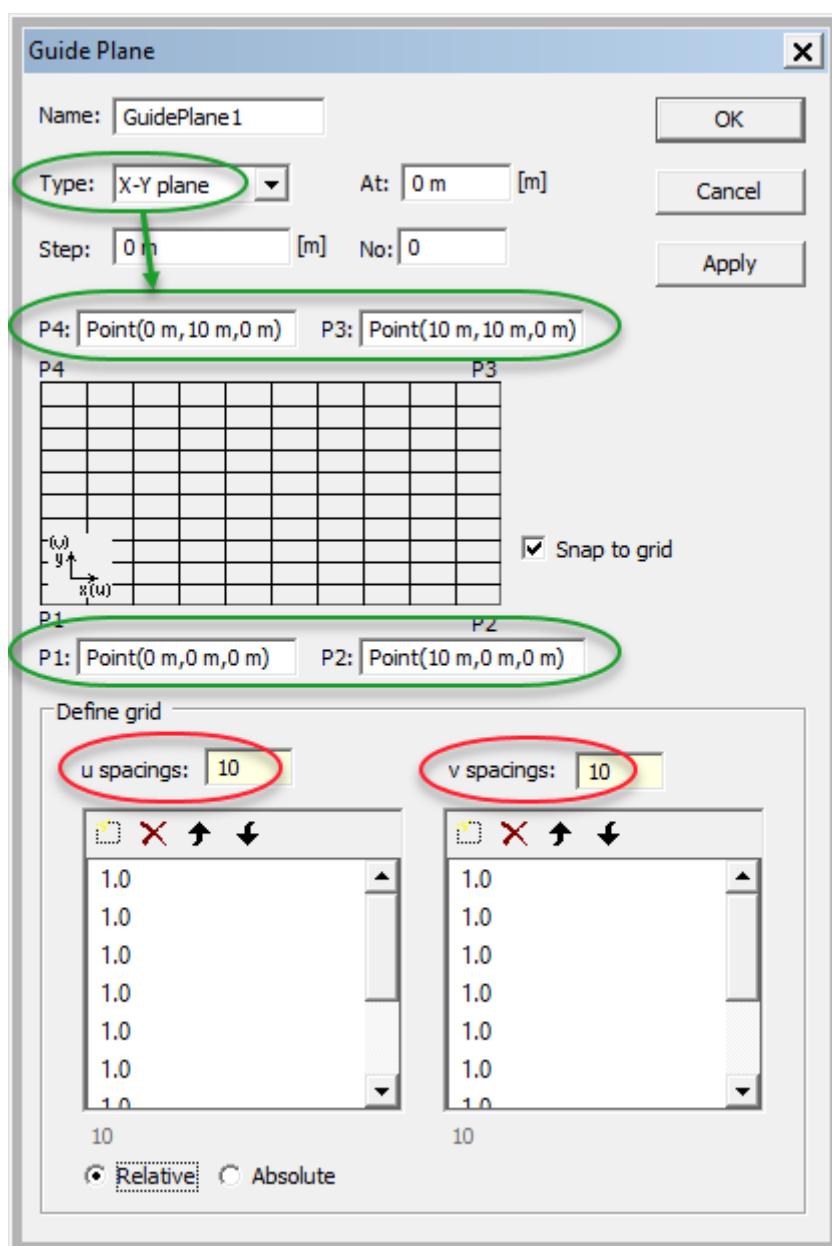
Name	Description	Thickness
Tck12	Thickness, t=0.012 m	0.012
<b>Tck15</b>	<b>Thickness, t=0.015 m</b>	<b>0.015</b>
Tck30	Thickness, t=0.03 m	0.03

- Use *Edit | Rules | Meshing Rules* to adjust FE mesh settings.
  - Check *Use second order elements*, i.e. select 8-node curved quadrilateral and 6-node curved triangular elements rather than 4-node flat quadrilateral and 3-node triangular elements.
  - Check *Round off Mesh Density* to allow some flexibility in the FE mesh generation, this allows somewhat larger elements than otherwise specified.
  - Check *Use uniformized face parameterization* to improve coarse FE meshes for curved surfaces. Do not use this option for fine meshes for large models as the mesh generation performance is significantly reduced.
  - Accept other default settings.



### 3 GUIDING GEOMETRY

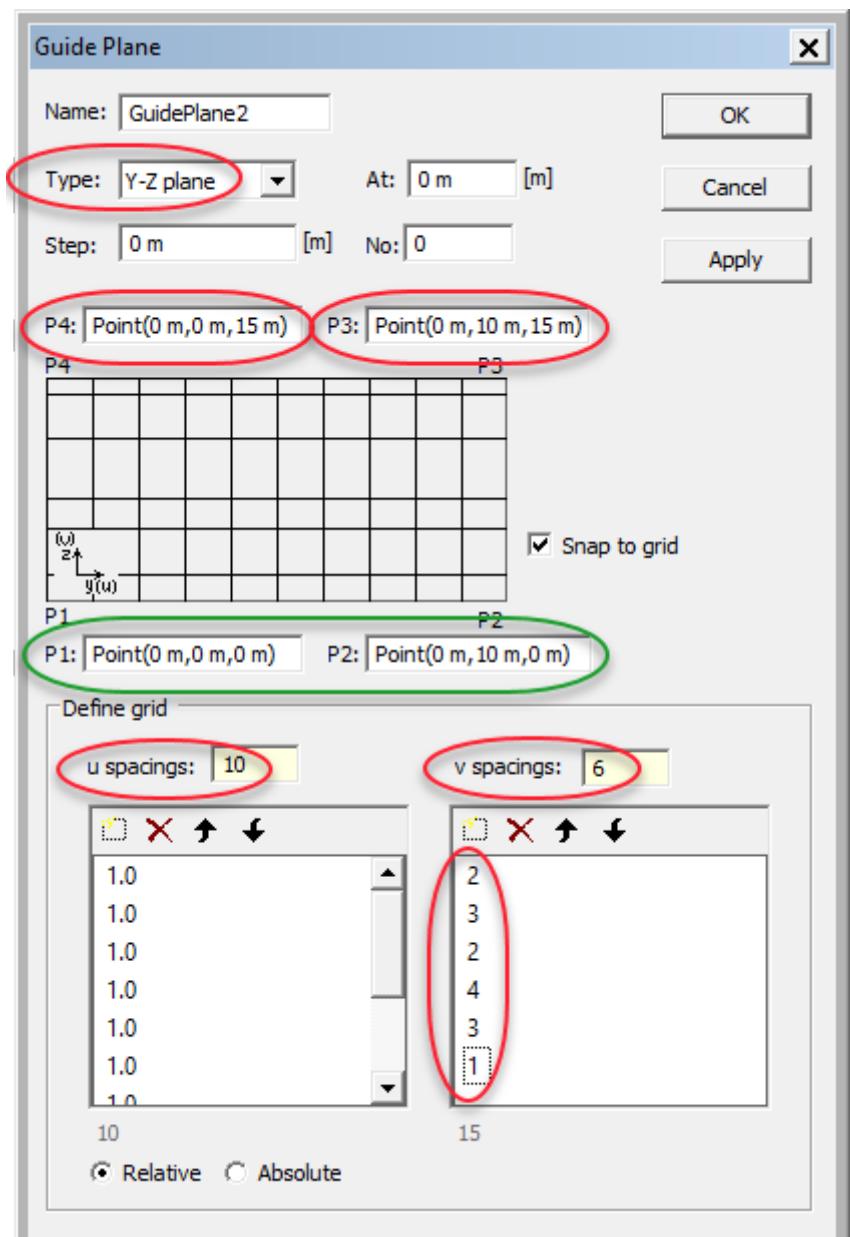
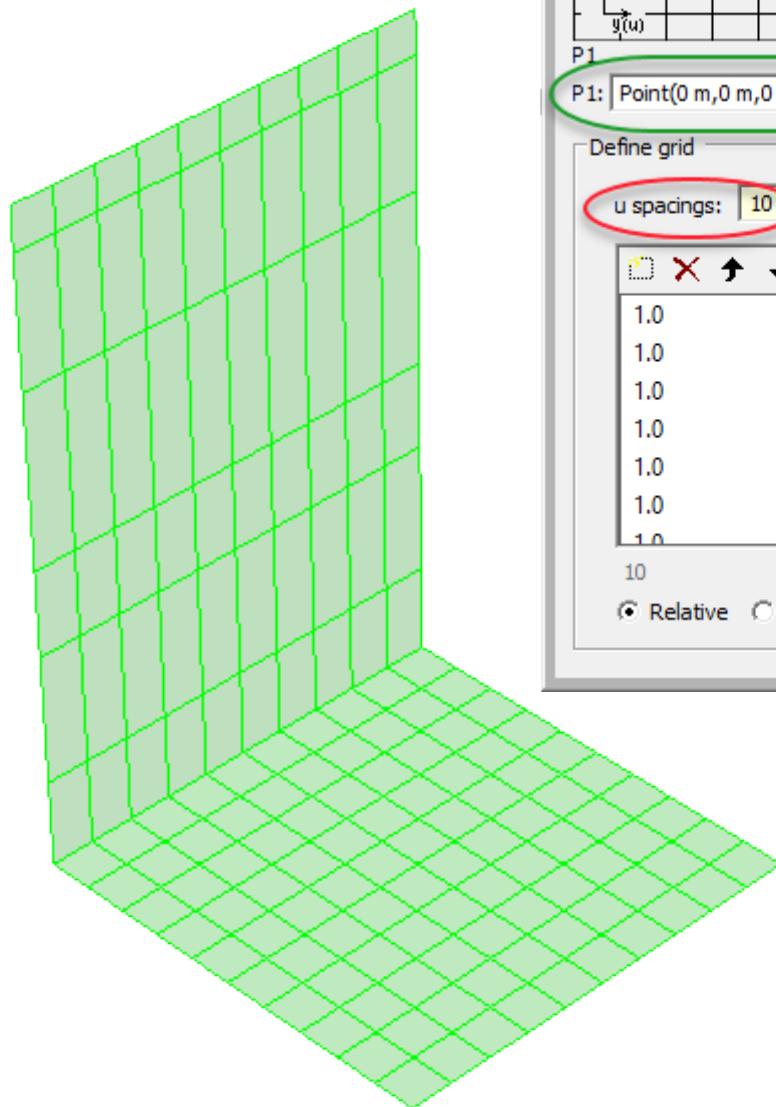
- Use *Guiding Geometry | Planes | Guide Plane Dialog* to create a guide plane with values as shown.
  - Accept default *Type: X-Y plane* and the coordinates for the corner points *P1* to *P4*.
  - Give value 10 for both *u spacings* and *v spacings* and let all spacings be 1.0, i.e. uniform spacing in both directions.
  - Click OK to create the guide plane.
- Provided the *Default display* configuration is chosen the guide plane will be displayed. Press the *Iso view* button, , or F5, to fit the guide plane in the display.



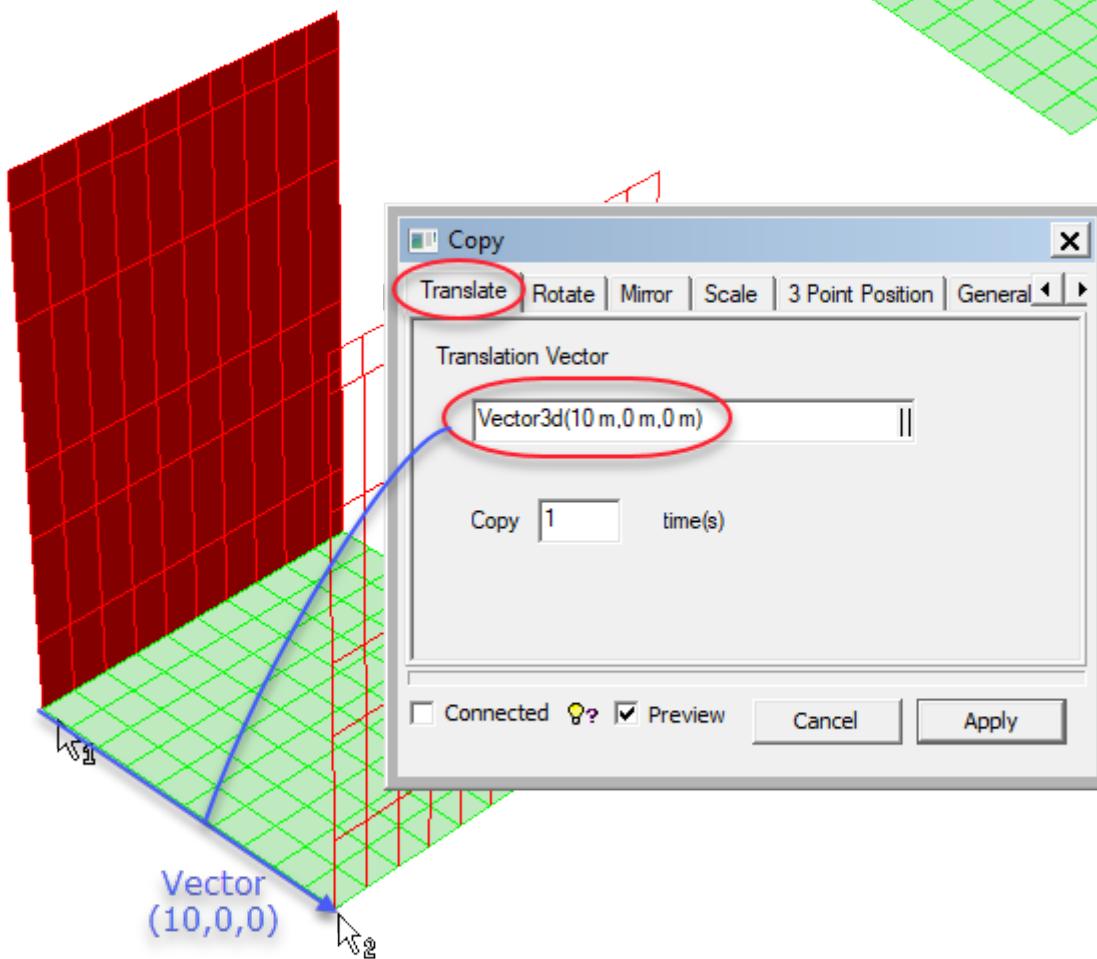
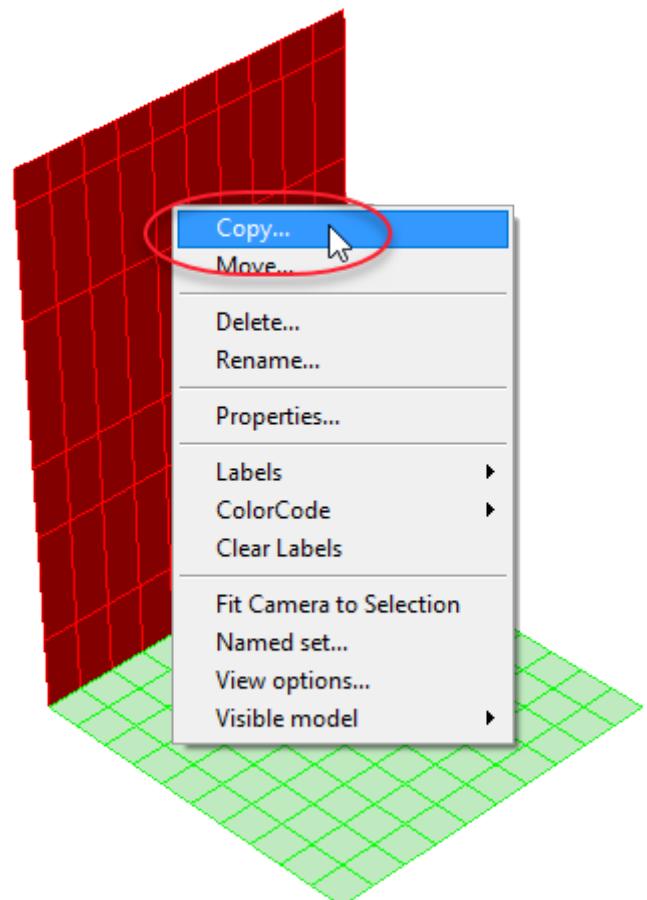
➤ Create a new guide plane with values as shown.

- Change to *Type: Y-Z plane* and adjust the coordinates of points *P4* and *P3*.
- Give 10 for *u spacings* and 6 for *v spacings*.
- Adjust *v spacings* as shown. (These are relative spacings.)
  - Note: Click individual items to change. Do not use Tab or Enter.
- Click OK to create the guide plane.

➤ Press F5 and see the display of the two guide planes.

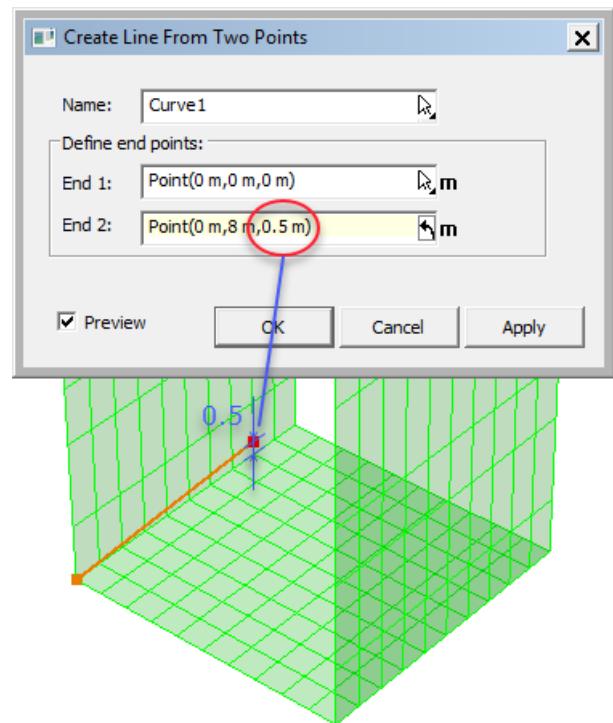
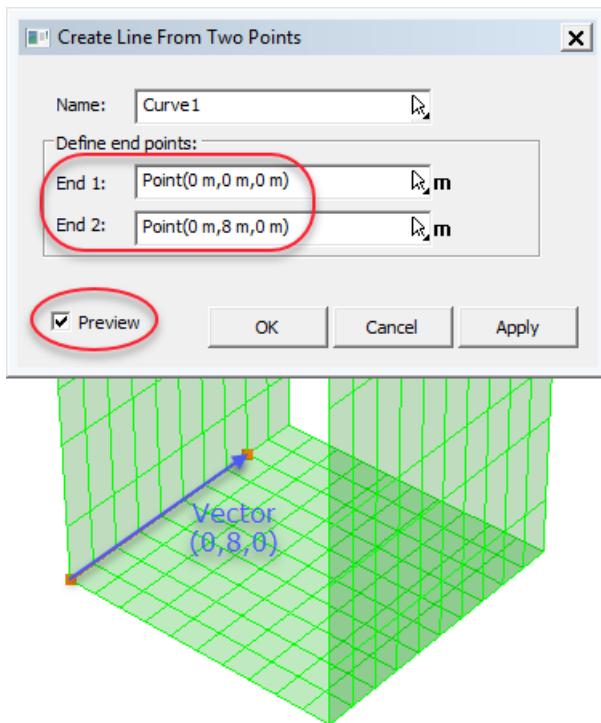


- Copy the vertical guide plane (GuidePlane2) as shown.
- Select the plane, right-click and press **Copy**.
- In the *Translate* tab of the **Copy** dialog fill in the vector by fetching it from the display by clicking twice as shown. Note that any two clicks in the model representing the desired vector (10,0,0) may be used.
- Click **Apply** and see the copy appear.

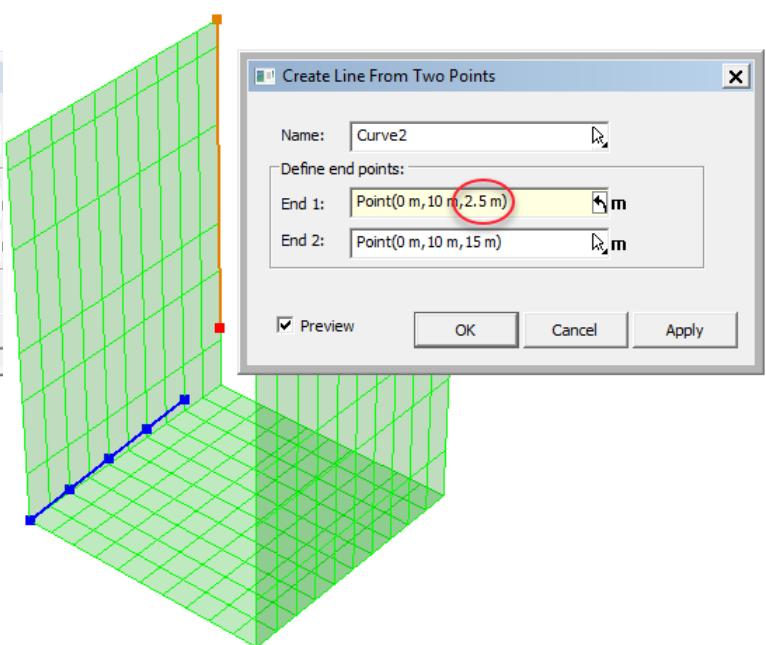
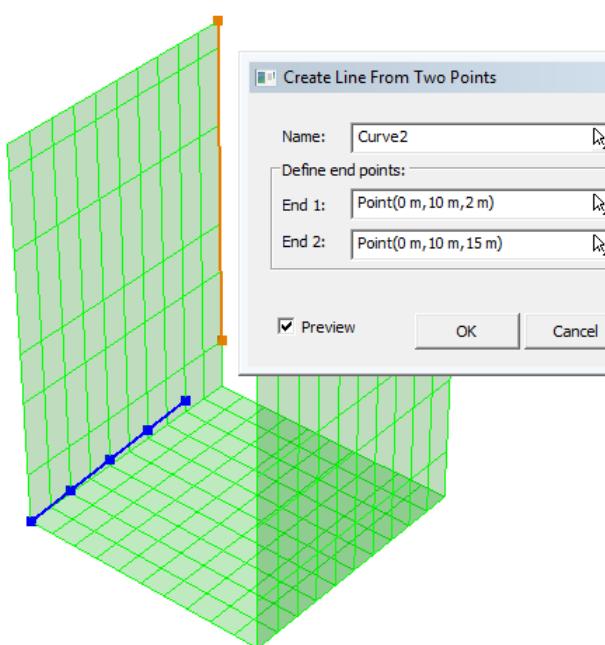


- Use *Guiding Geometry | Lines | Guide Line Dialog* to create the guide line below. Since there is no clickable point for *End 2* click point below and correct Z value.

- Check *Preview* to see the guide line position before creating it.

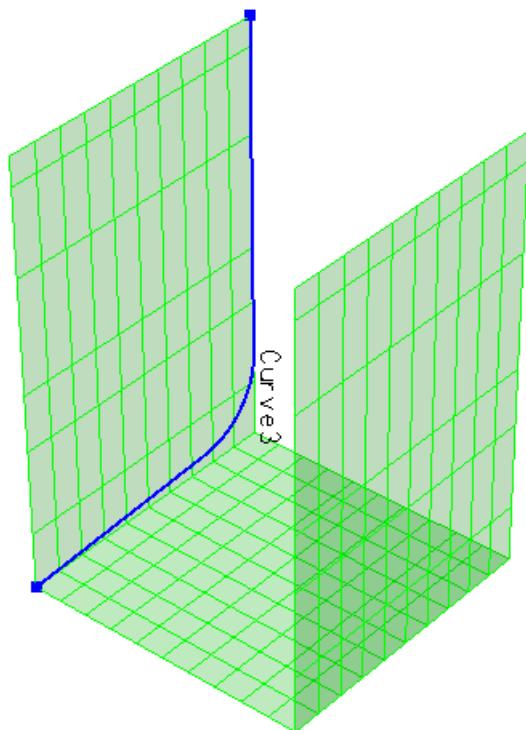
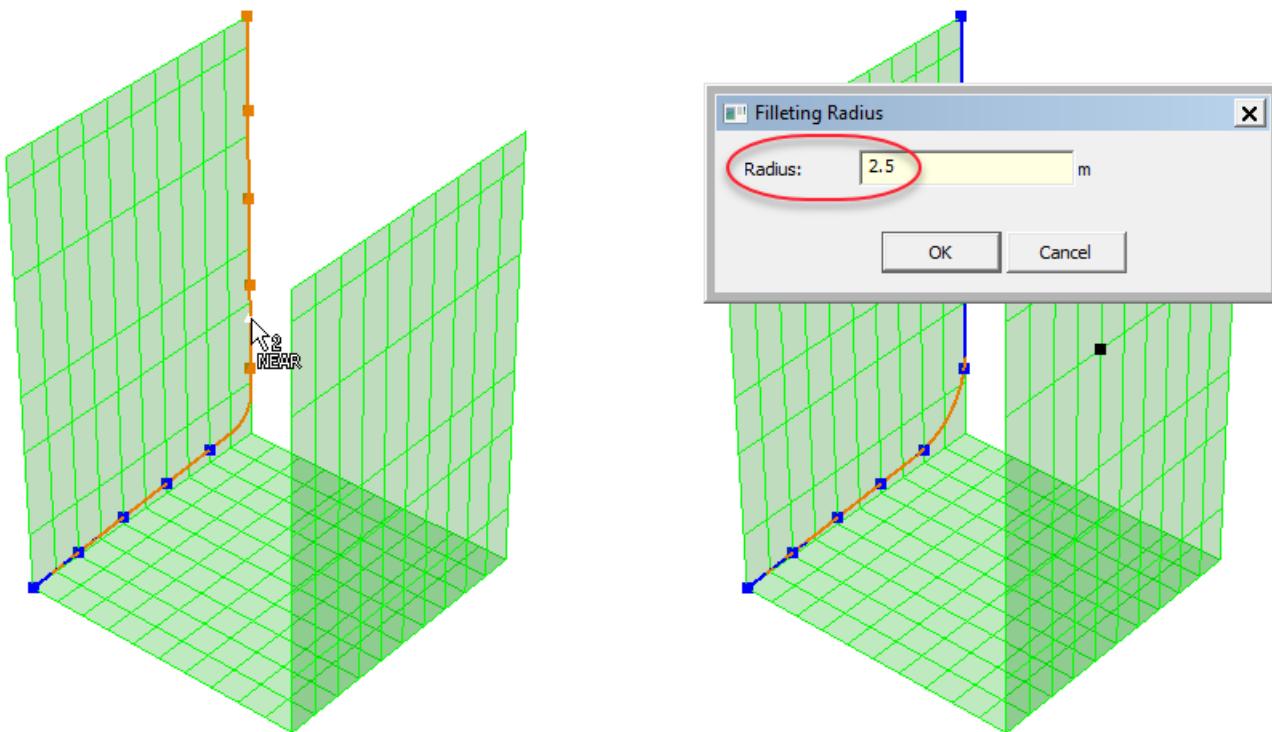


- Create a second vertical guide line in a similar way. First click lower and upper points in the display, then correct Z coordinate of *End 1* from 2 to 2.5.

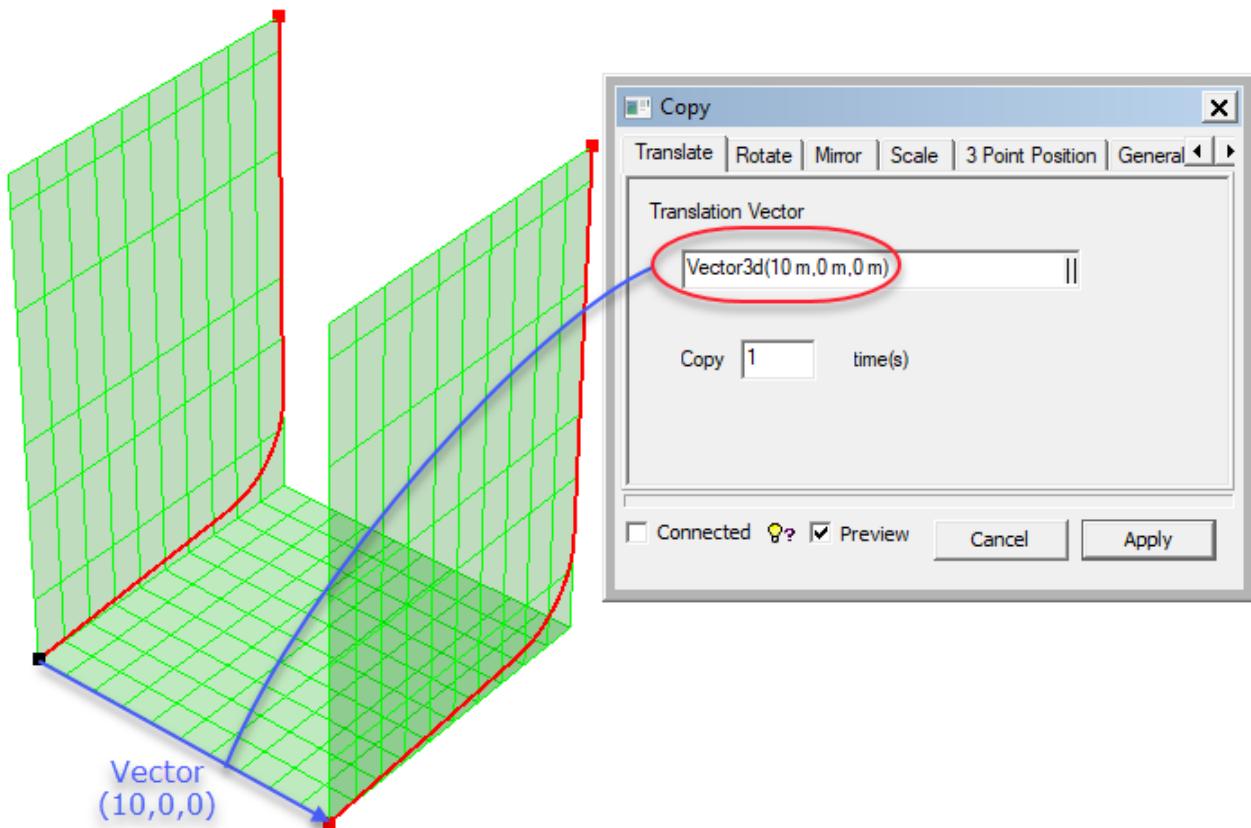


## 4 CREATE OUTER HULL

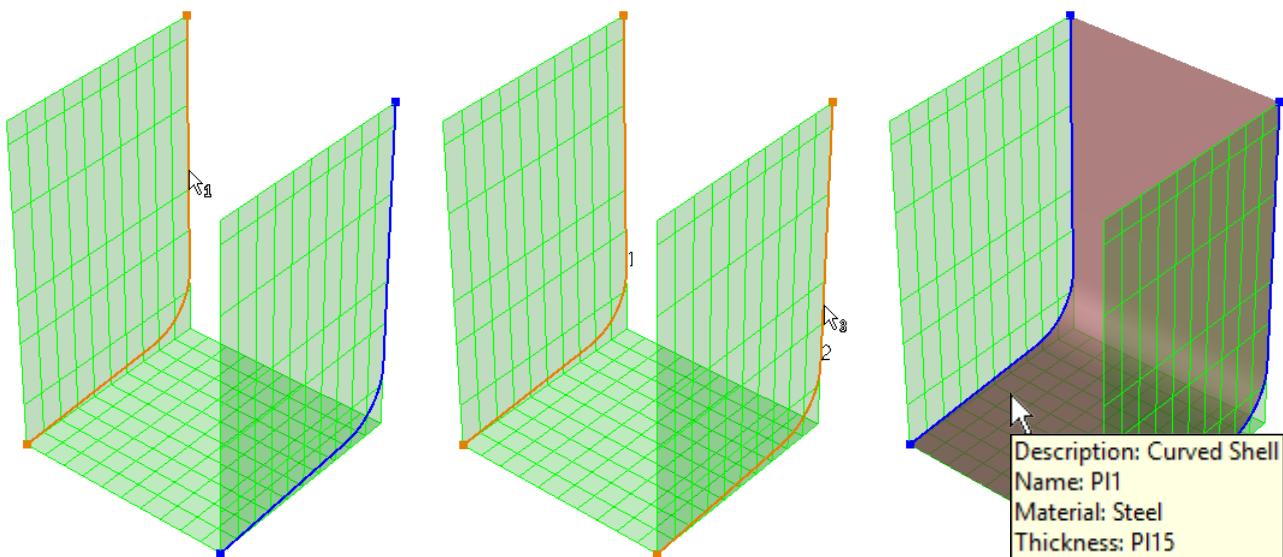
- Use *Guiding Geometry | Conic Sections | Circular Fillet* to join the two guide lines by a fillet curve.
- After clicking the two lines the *Filletting Radius* dialog appears in which the radius 2.5 is entered.
  - Clicking *OK* joins the two guide lines into a single guide curve.



- Copy the guide curve as shown below.

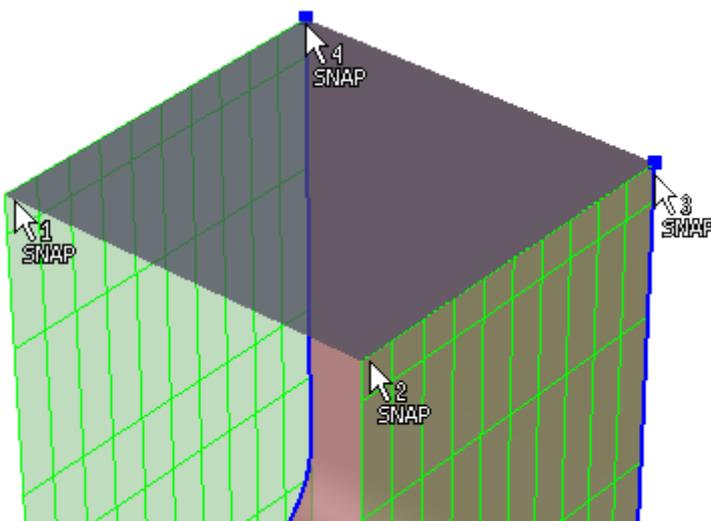


- Use *Structure | Free Form Shells | Skin/Loft Curves* to create the hull. Click the curves in sequence and close by double-clicking the last curve. In this case only two curves.
- The hull will have thickness 15 mm since Tck15 is currently the default thickness. This can be verified by hovering the mouse over the plate.

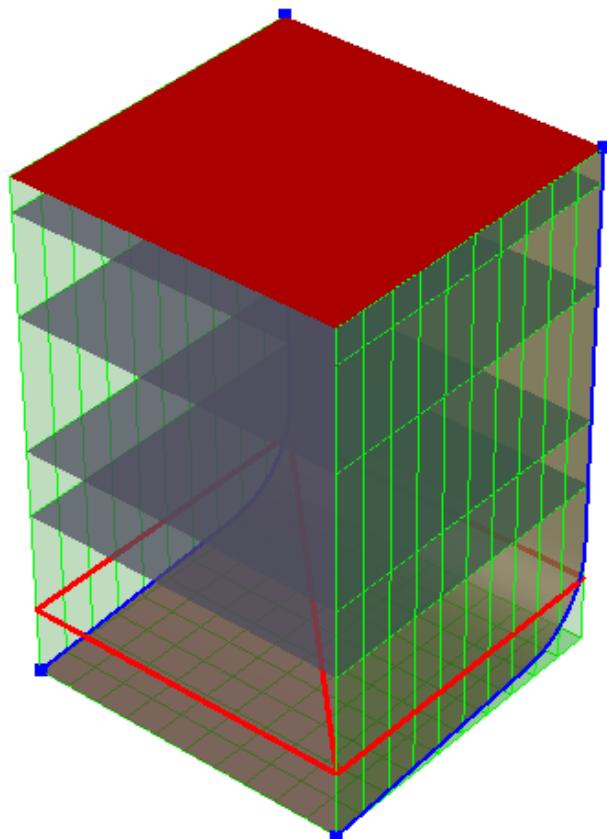
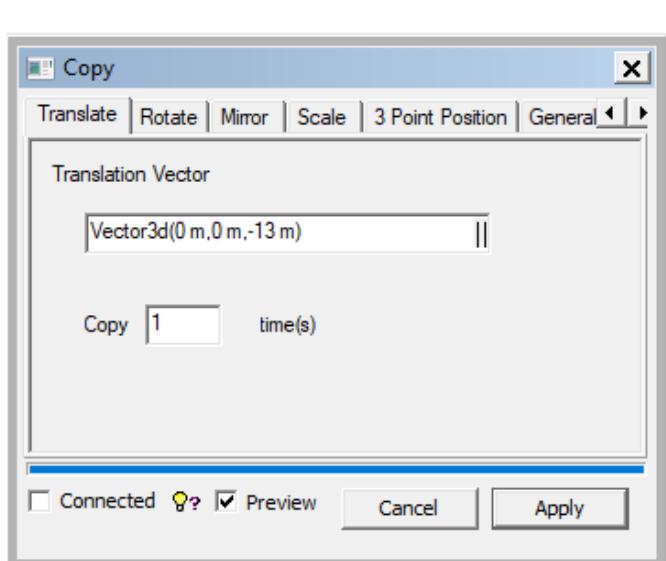


## 5 CREATE DECK PLATES

- The deck plate thicknesses are 12 mm so change default thickness to Tck12.
- Use *Structure | Flat Plates | Flat Plate*, or simply press the *Plate* button  , to create a deck at top level. Click the four corners in sequence, double-click the last corner.

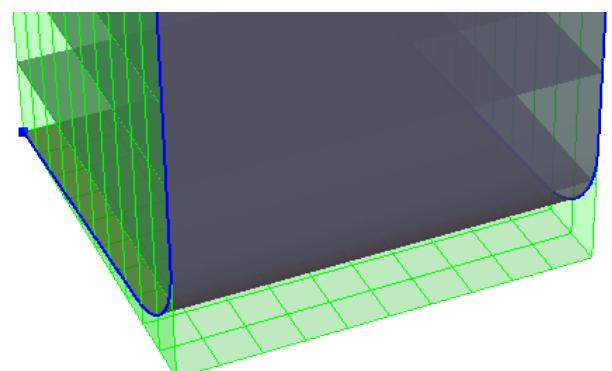
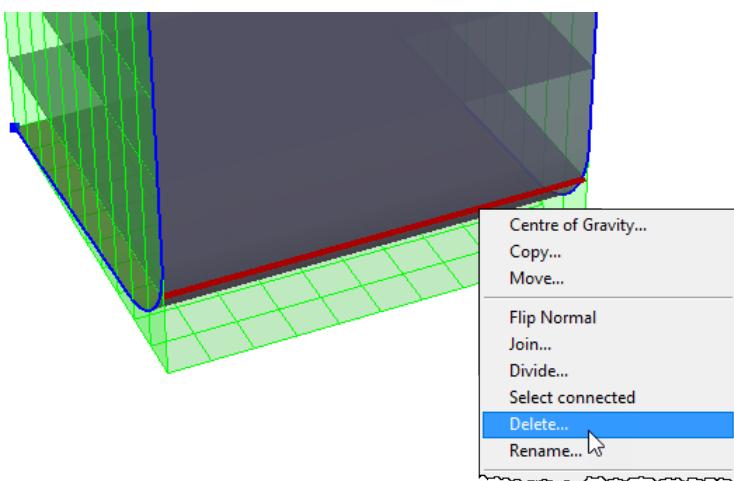
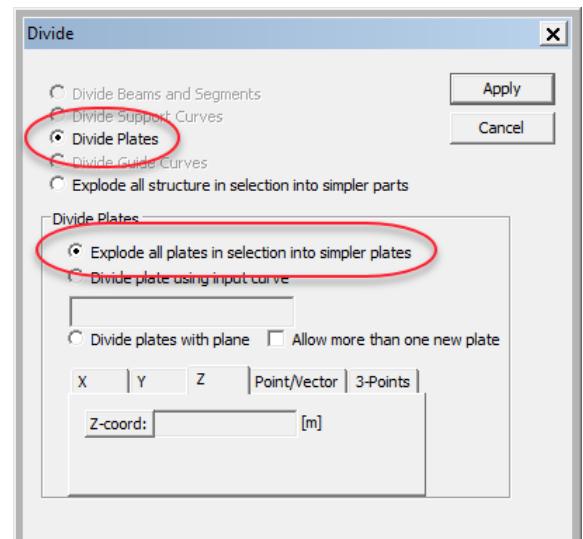
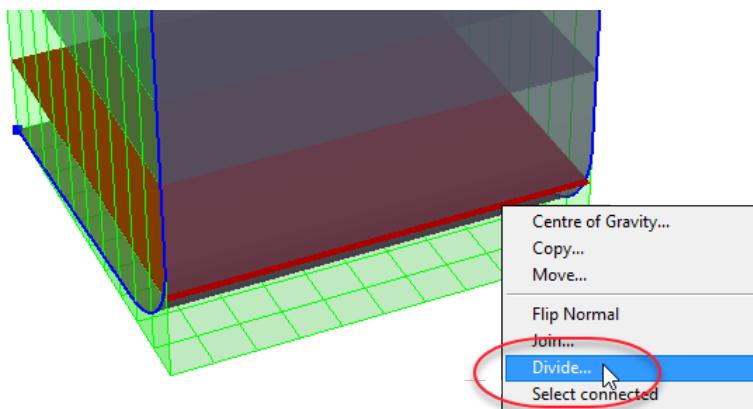
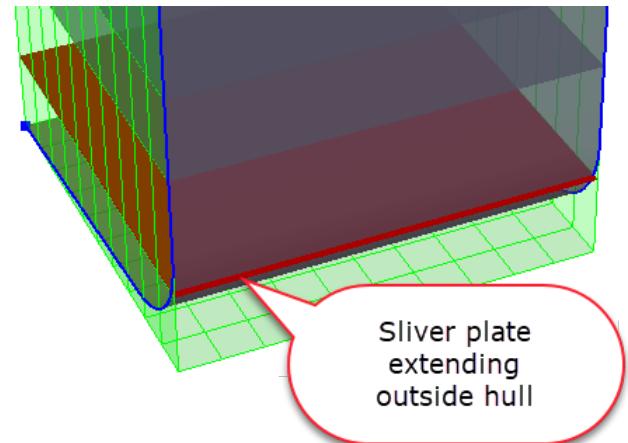


- Copy the top deck at elevation 15 to elevations 14, 11, 7, 5 and 2, i.e. to where there are clickable points in the vertical guide planes. So, all copy vectors may be fetched from the display. This operation is shown below for the deck at elevation 2.



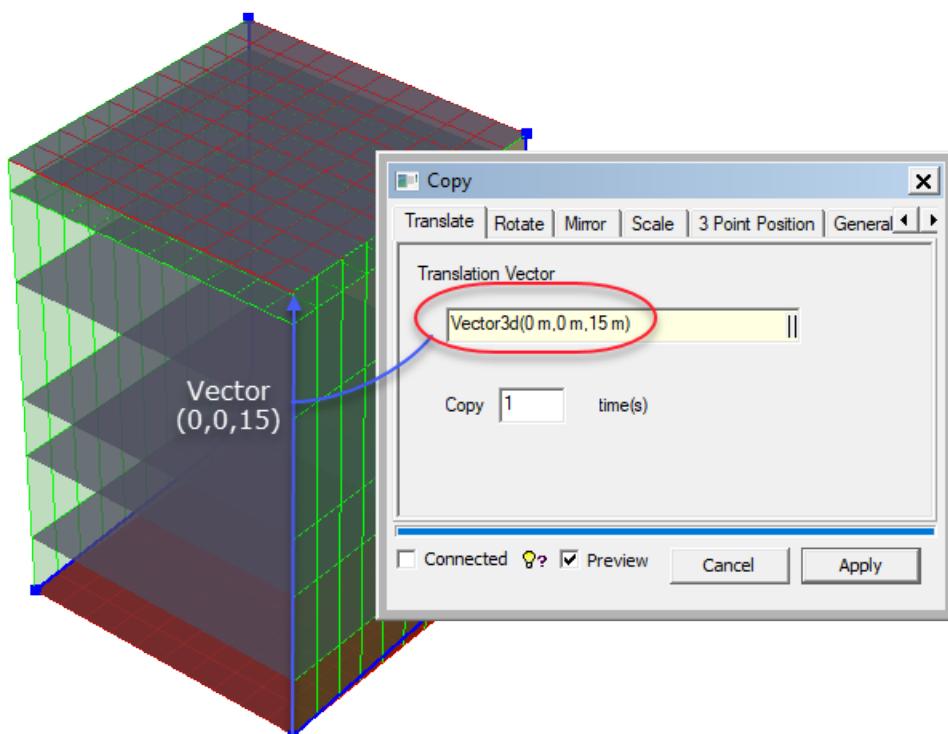
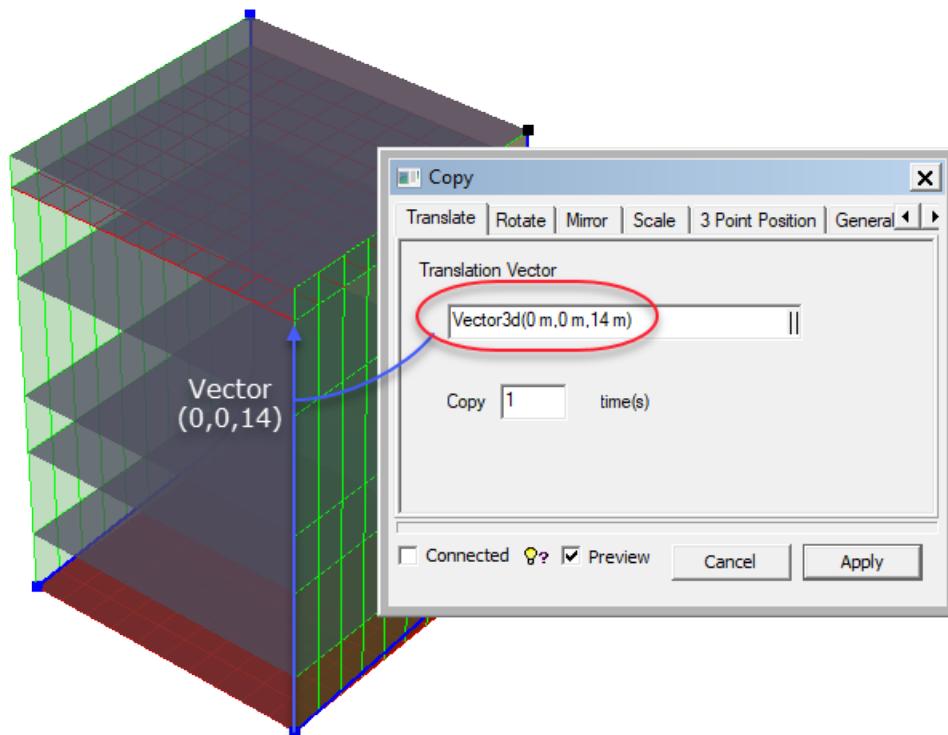
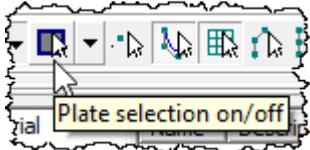
- The deck at elevation 2 extends a bit outside the hull surface. This is trimmed off as explained in the following.

- Select the plate, right-click and press *Divide*.
- In the *Divide* dialog select *Divide Plates*, *Explode all plates in selection into simpler plates* and click *Apply*.
- Select the sliver plate, i.e. the bit extending outside the hull, right-click and press *Delete*. (Or press the Delete key on the keyboard.)
- The end result is shown to the lower right.

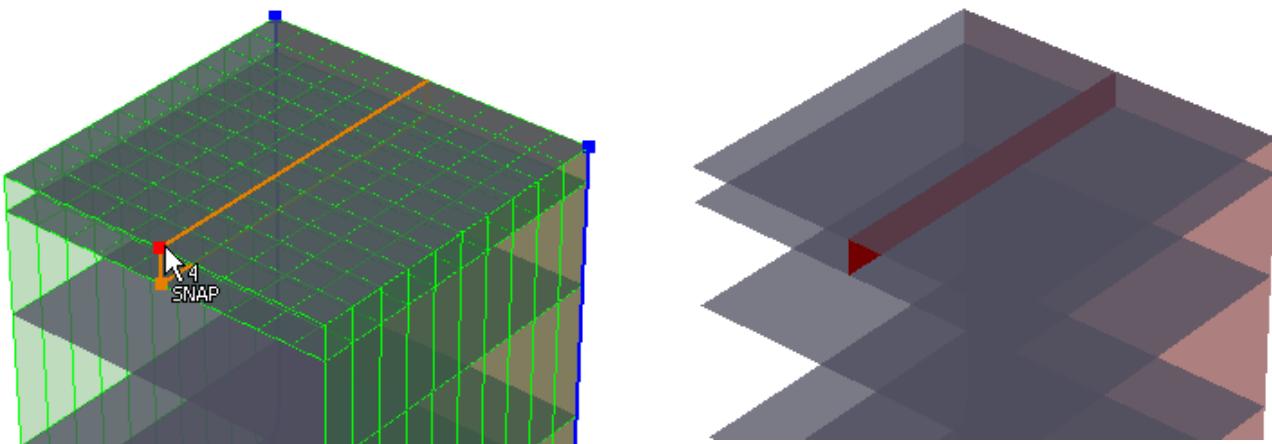


## 6 CREATE VERTICAL STIFFENER PLATES

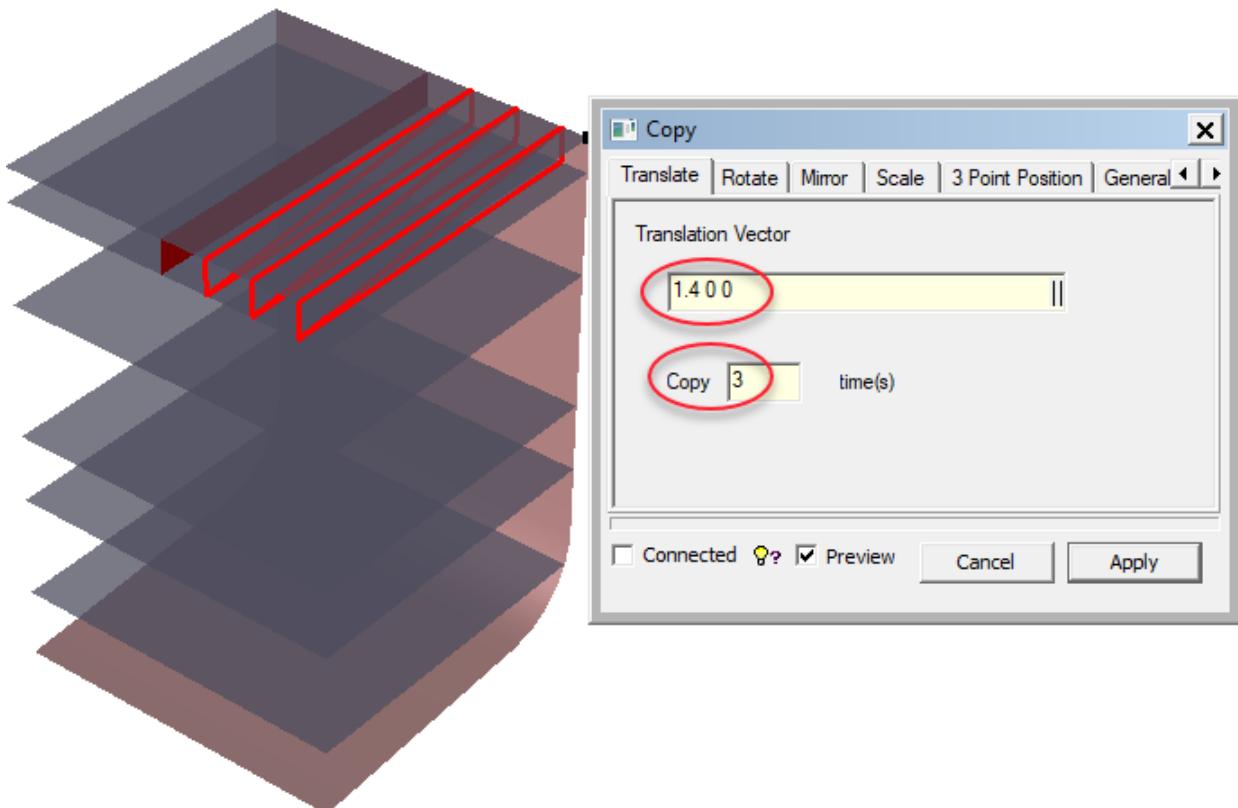
- Copy the horizontal guide plane (GuidePlane1) up to elevations 14 and 15. To enable selecting the guide plane at elevation 0 rather than the surface lift the *Plate selection on/off* button:



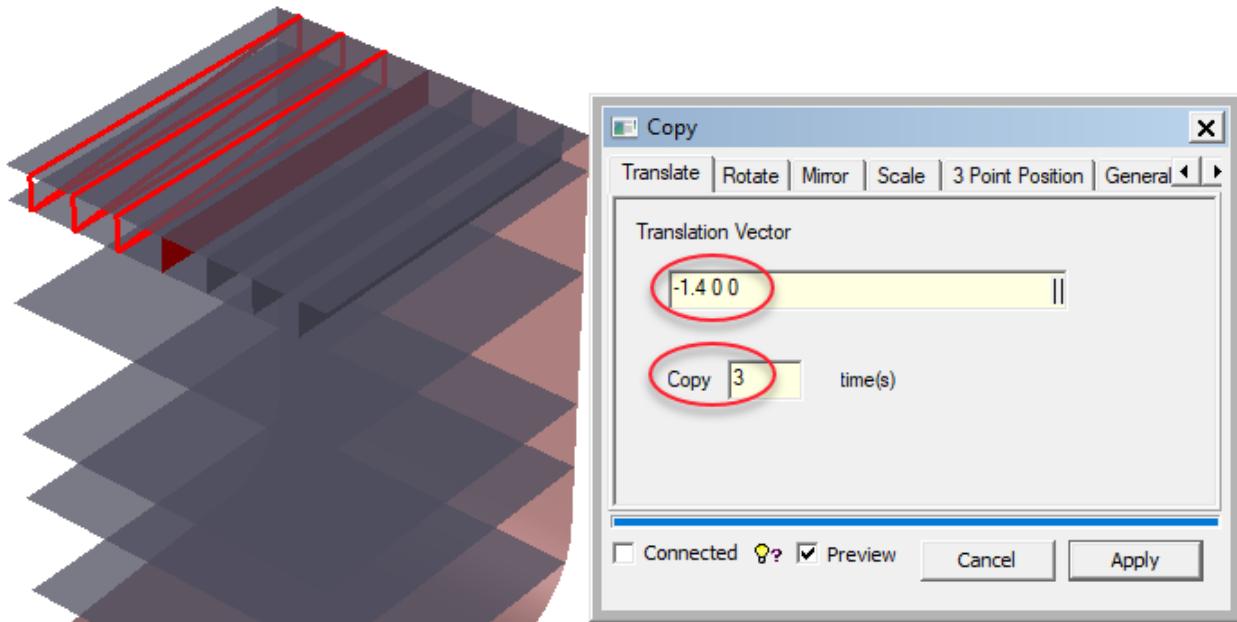
- Create a vertical stiffener plate in the YZ-plane in between the elevations 14 and 15 and at X = 5.
  - Having created the plate switch to *Modelling - Transparent* as the guide planes are not needed for the subsequent operations.



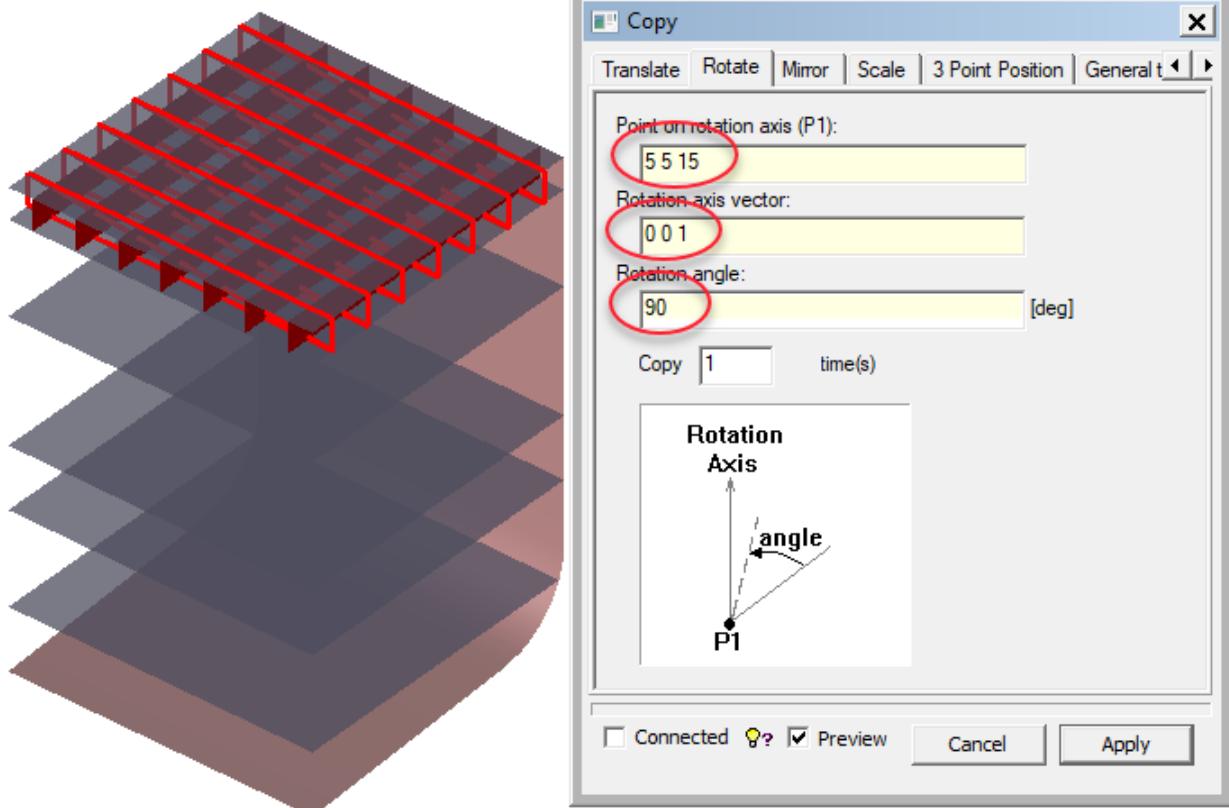
- Press the *Plate selection on/off* button to enable selecting and clicking plates.
- Copy the vertical plate three times a distance of 1.4 in the X-direction, i.e. type in the vector (1.4,0,0) as shown below.



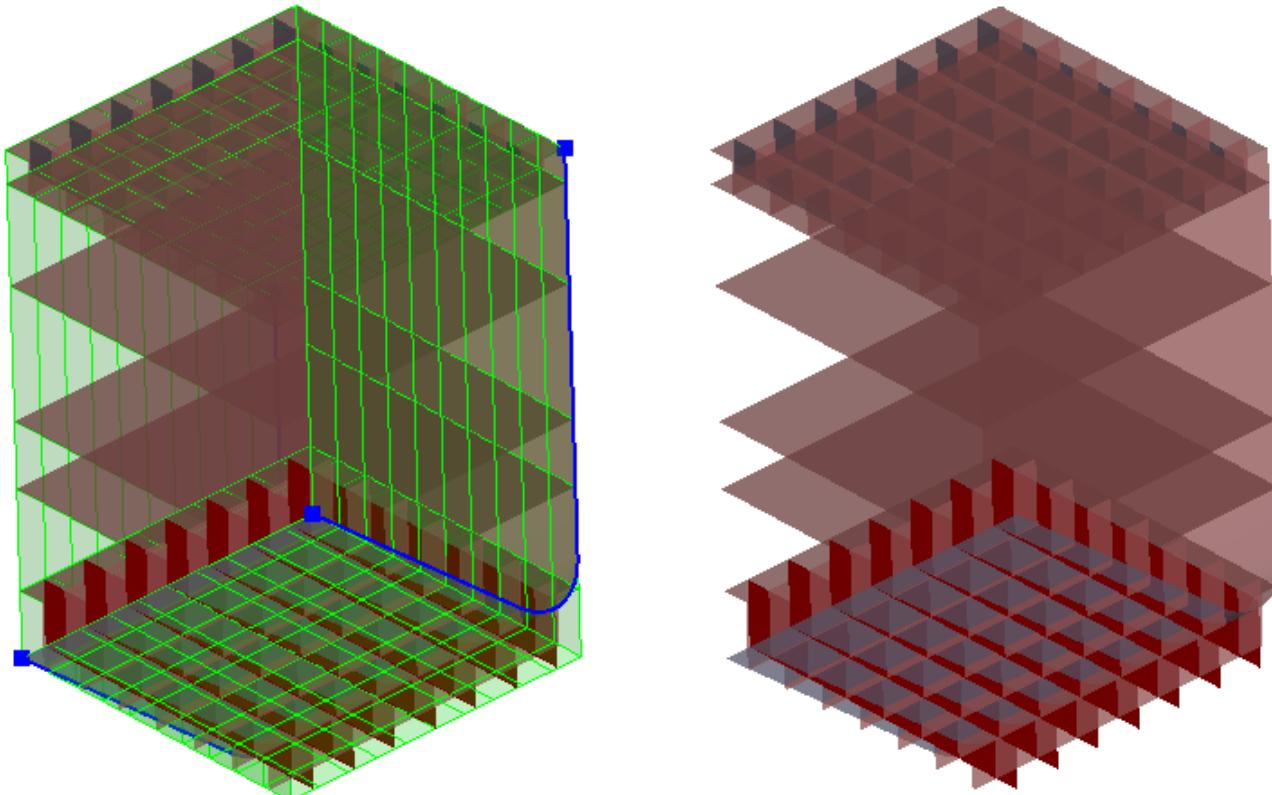
- Then copy the same plate three times a distance of  $-1.4$  in the X-direction, i.e. type in the vector  $(-1.4, 0, 0)$  as shown below.



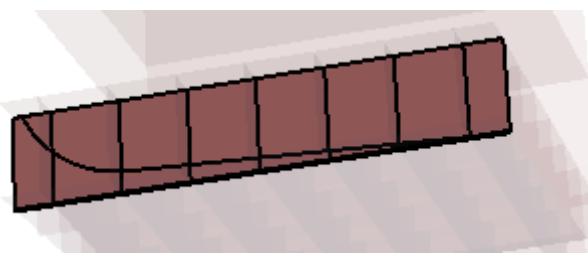
- Copy the vertical stiffener plates by a 90-degree rotation about the vertical axis to create a stiffener grid. The rotation point is  $(5, 5, 15)$ . (The Z value has no consequence).



- Create vertical stiffener plates between the lowest deck and the hull by copying the horizontal guide plane at elevation 0 (GuidePlane1) up to elevation 2 (use *Default display* configuration) and then following the same procedure as for the vertical stiffener plates between the decks at elevations 14 and 15.



- Double-click one of the stiffener plates in the XZ-plane and see that it has an internal curved edge. This is because the stiffener plates in the XZ-plane are copies of the stiffener plates in the YZ-plane and these stiffeners are intersected by the hull.

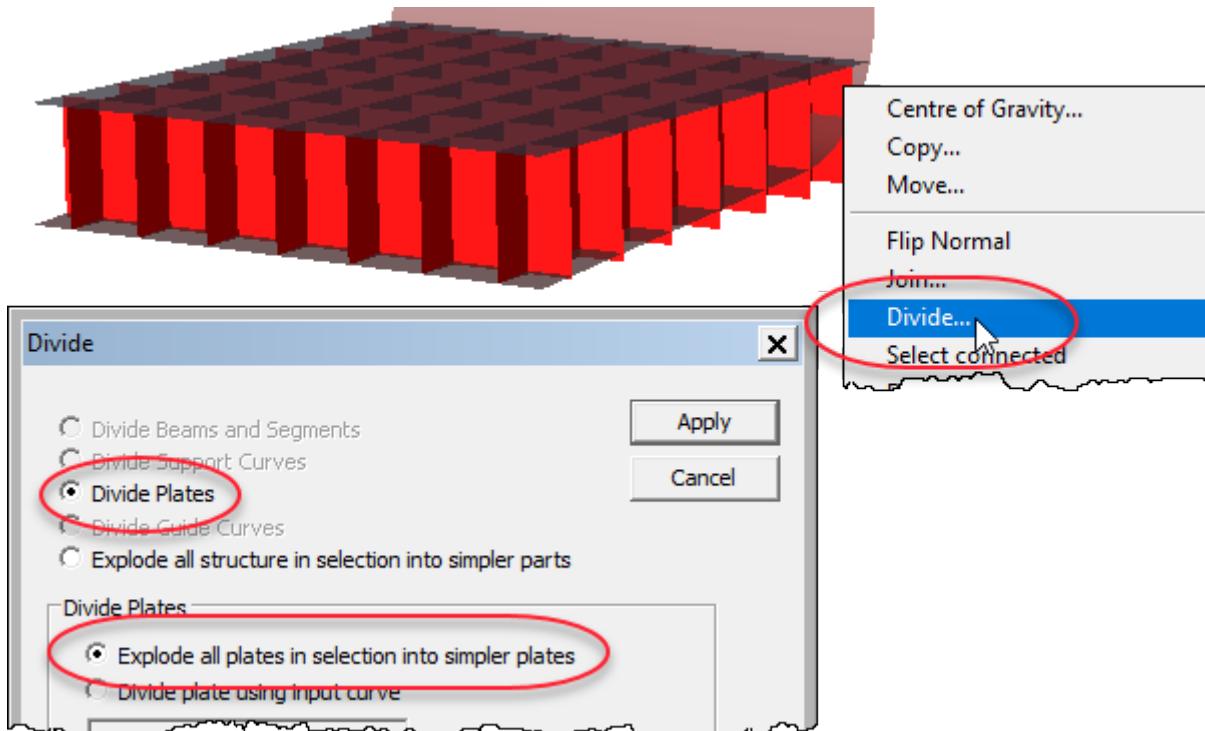


- Use *Structure | Topology | Simplify Topology* to remove superfluous internal edges.

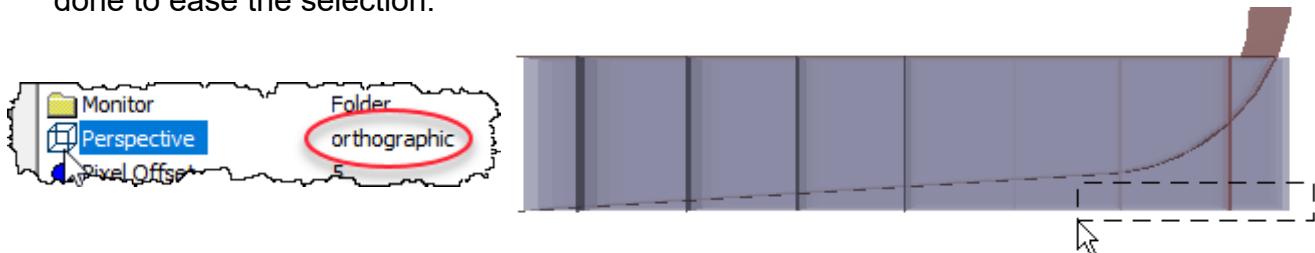


- Double-click outside the model to revert to normal display mode.

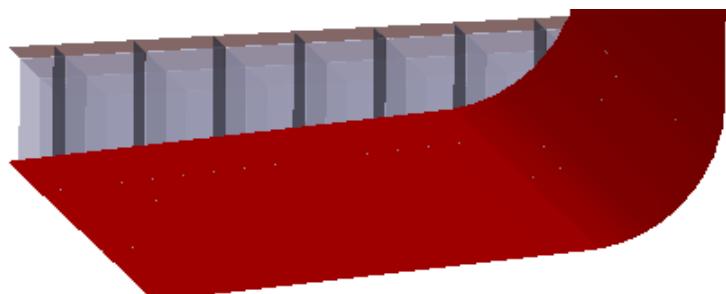
- The vertical stiffener plates intersect the hull so they need to be divided by *Explode all plates in selection into simpler plates* and thereafter trimmed.



- To ease the selection of the many protruding plate parts view the model along the X axis (F6), zoom in and select by touching mode (drag rubberband from right to left). Switching from perspective to orthographic view by *View | Options | General* may optionally be done to ease the selection.



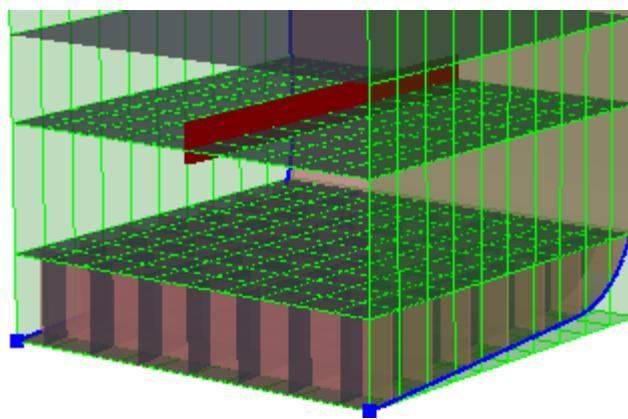
- Using this selection method a few times combined with selecting by clicking missed parts should allow you to delete all protruding parts. Make sure all protruding parts are deleted by zooming and rotating the model. Selecting the hull may help in locating protruding blue-grey parts.



- You may want to return to perspective view.

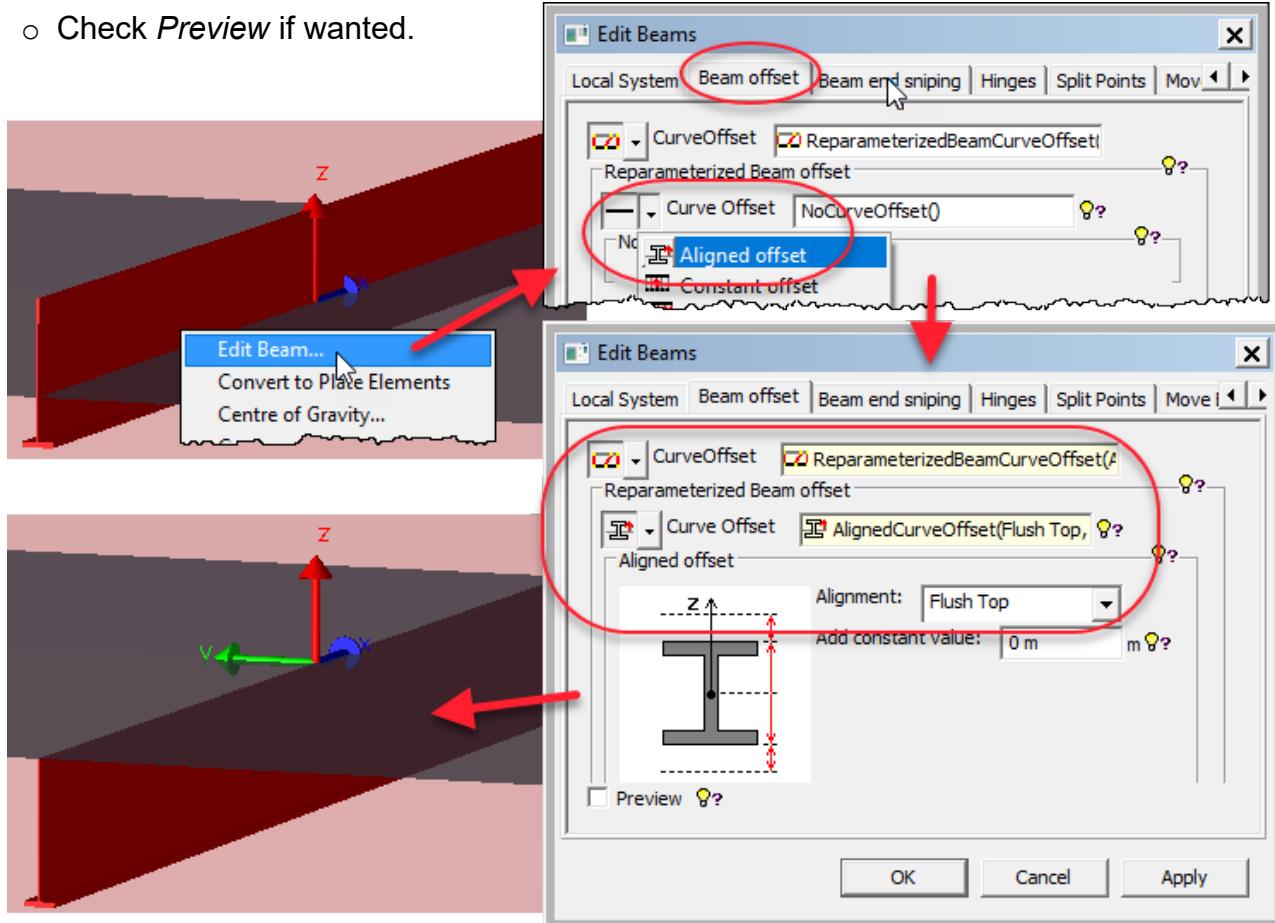
## 7 CREATE STIFFENER BEAMS FOR DECKS

- Copy the horizontal guide plane (GuidePlane1) up to elevation 5. The guide plane may optionally be selected in the browser under *Utilities | Guiding Geometry*.
- Use *Structure | Beams and Piles | Straight Beam*, or press the *Beam* button  , to create a stiffener beam with section Tbar885x200x14x35 for the deck at elevation 5. The beam is oriented in the Y-direction and at position X = 5.

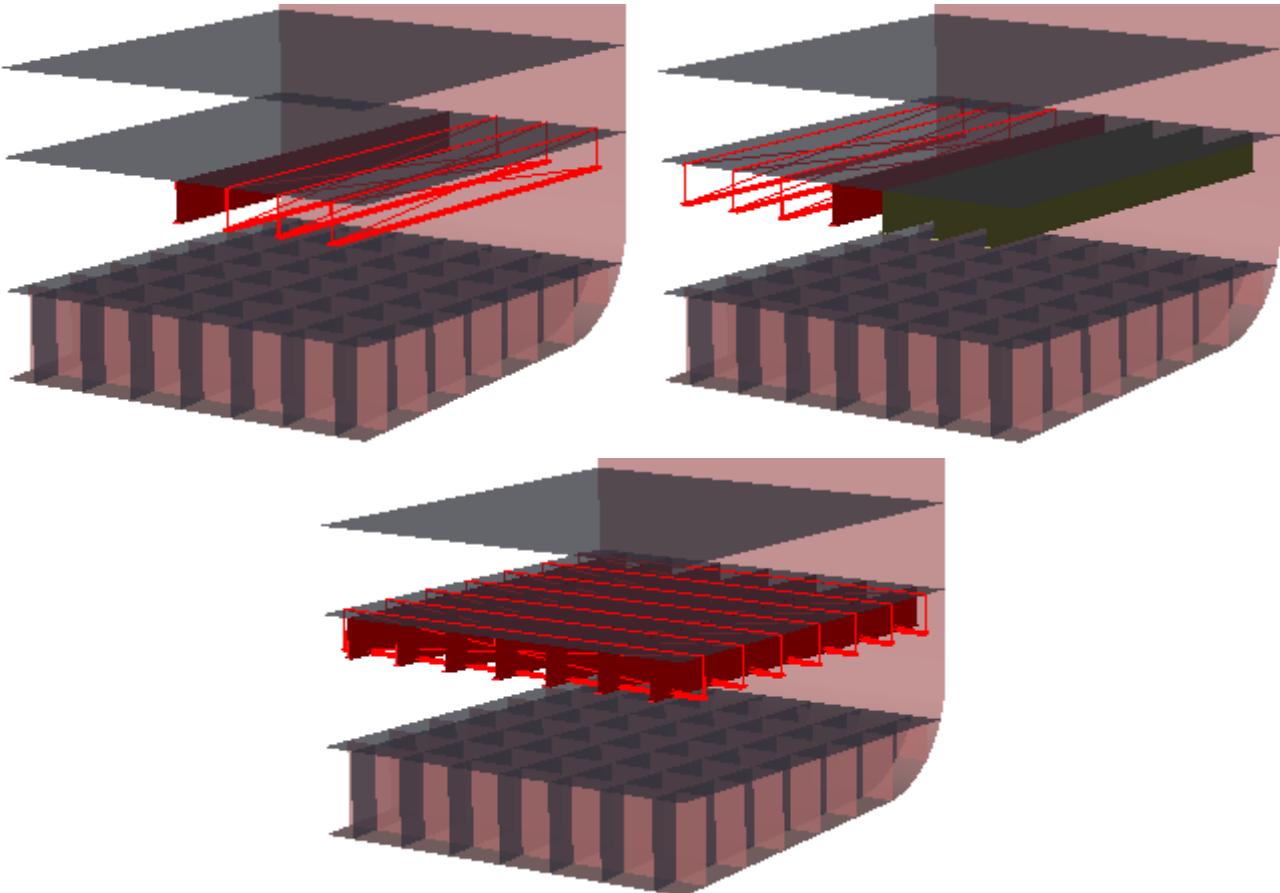


- The orientation of the cross section is correct (local z axis pointing upwards) but it must be assigned an offset (eccentricity) for the top to be flush with the plate. Do this by selecting and right-clicking the beam, selecting *Edit Beam* and in the *Edit Beams* dialog, go to the *Beam offset* tab. For *Curve Offset* select *Aligned offset* and set *Alignment* to *Flush Top*.

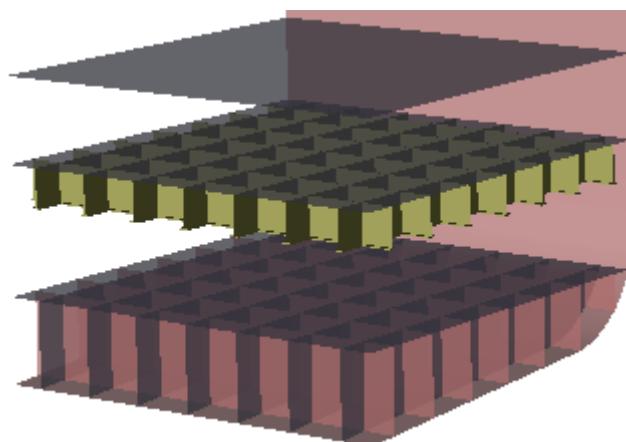
- Check *Preview* if wanted.



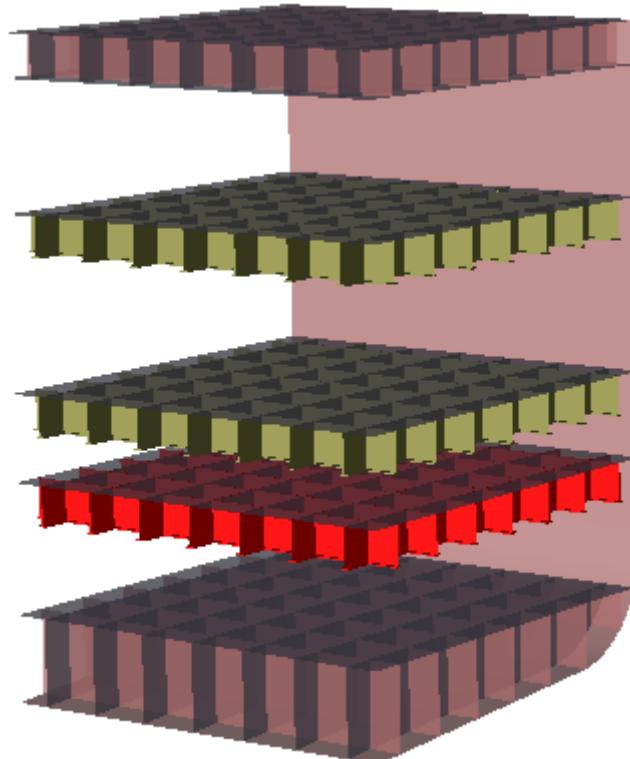
- Copy the stiffener beam three times a distance of 1.4 in the X-direction, and three times a distance of -1.4 in the X-direction, i.e. in the same way as for the stiffener plates.



- Also, as for the stiffener plates, copy the stiffener beams by a 90-degree rotation to create a stiffener grid.

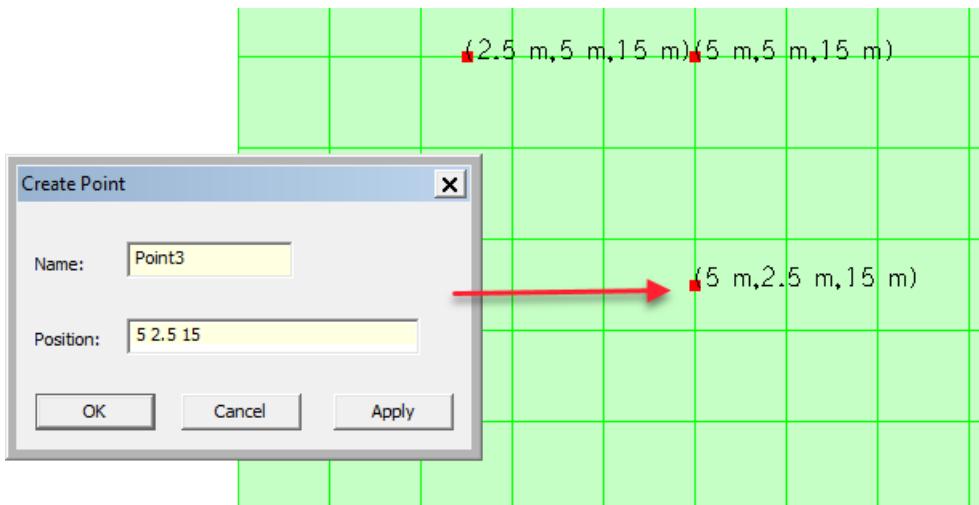


- Copy the stiffener grid to elevations 7 and 11. Selecting all stiffener beams but no plates and surfaces is easily done by lifting the *Plate selection on/off* button while using the *Modelling - Transparent*. Note that to be able to fetch copy vectors from the display press the *Plate selection on/off* button to enable clicking plates once the beams have been selected.

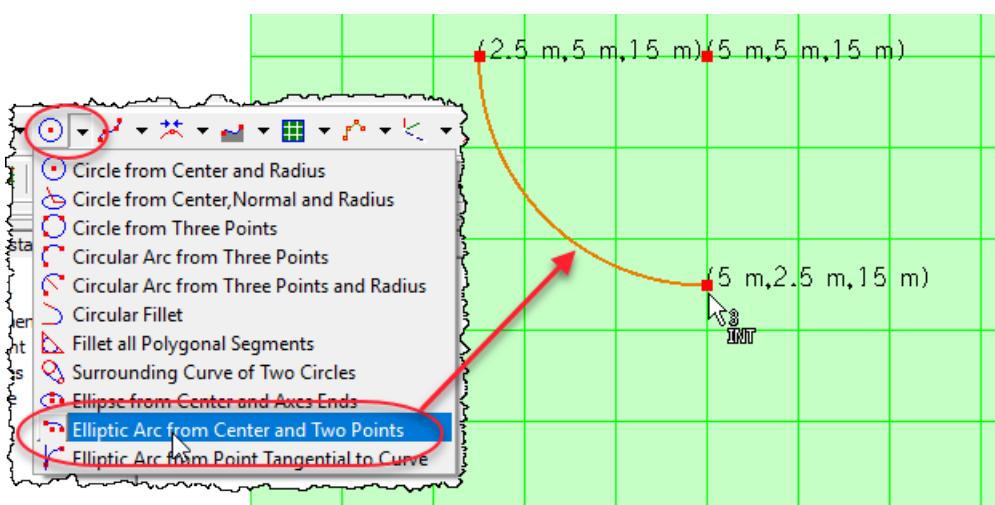


## 8 CREATE COLUMN

- To ease the work, switch to *Default display* configuration, select the top guide plane at elevation 15 (GuidePlane5) and display only this guide plane by Alt+S.
- View from above by clicking the *View from Z* button, , or pressing F8.
- Create a guide curve forming a 90-degree arc as shown.
  - First create the centre and end points of the arc by *Guiding Geometry | Points | Guide Point Dialog*. The coordinates of the three points are given below.



- Then create the guide curve being an arc by clicking the three points as shown.
  - Press the *Point selection* button, , to enable clicking points.
  - While clicking, notice the information in the lower left corner of the GeniE window telling which point of the arc is expected and the coordinates of the current hovering point: **Center: Point(5 m, 5 m, 15 m)**



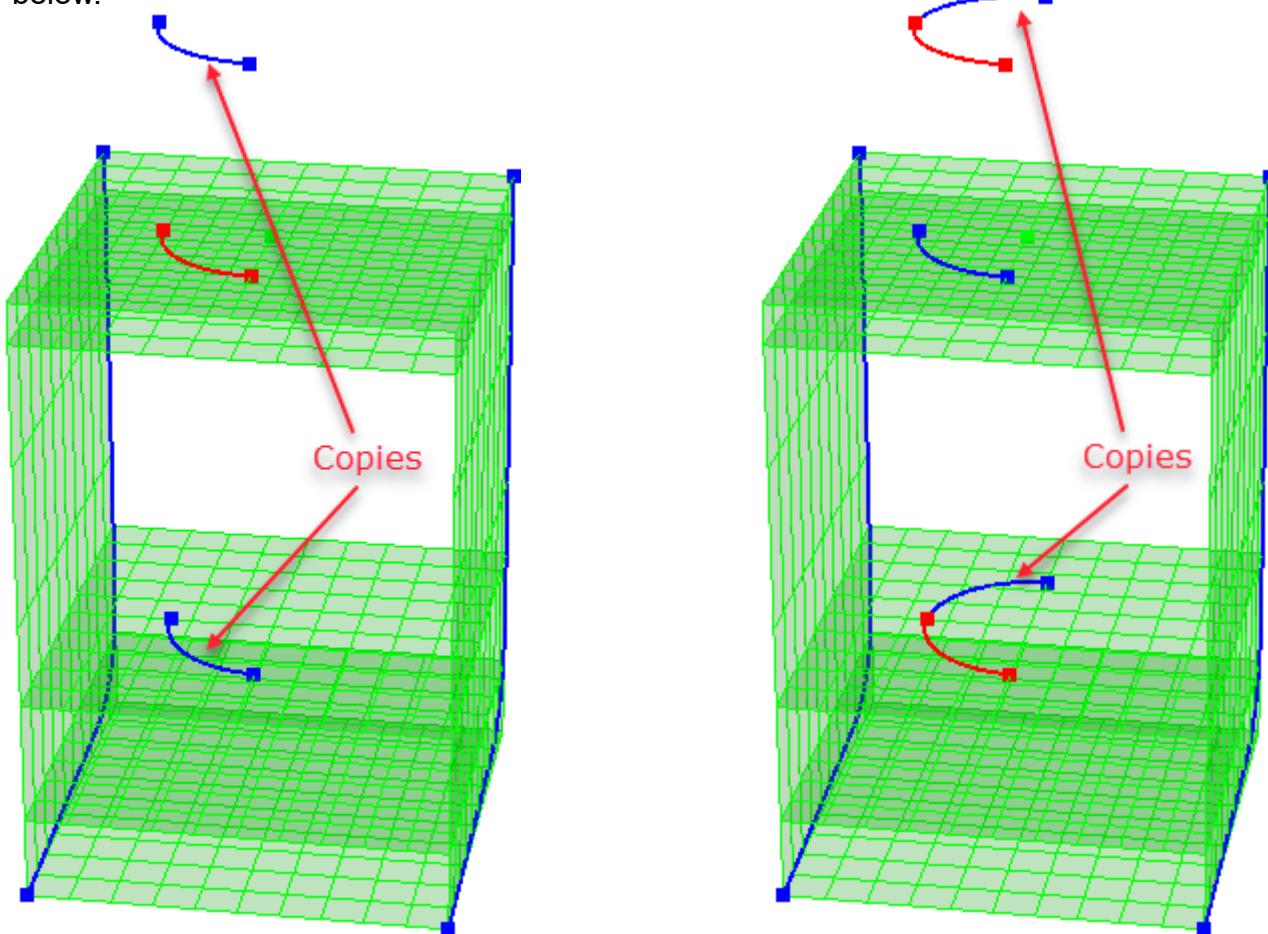
➤ Use Alt+A to display the whole model and rotate to a better view point, e.g. by pressing F5.

➤ Copy the guide curve at elevation 15 to elevations 5 and 20, i.e. use the copy vectors  $(0,0,-10)$  and  $(0,0,5)$ , respectively. See the figure to the left below.

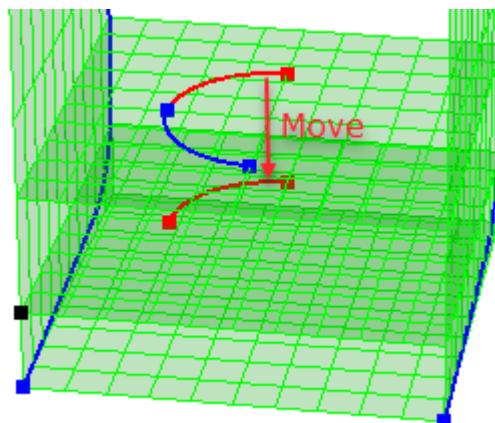
- In the figures below the plates and beams are hidden by right-clicking the respective selection buttons and closing the eye symbol. To the right, this is shown for the beams. Note that this setting is stored in the registry and should, therefore, be reset later.



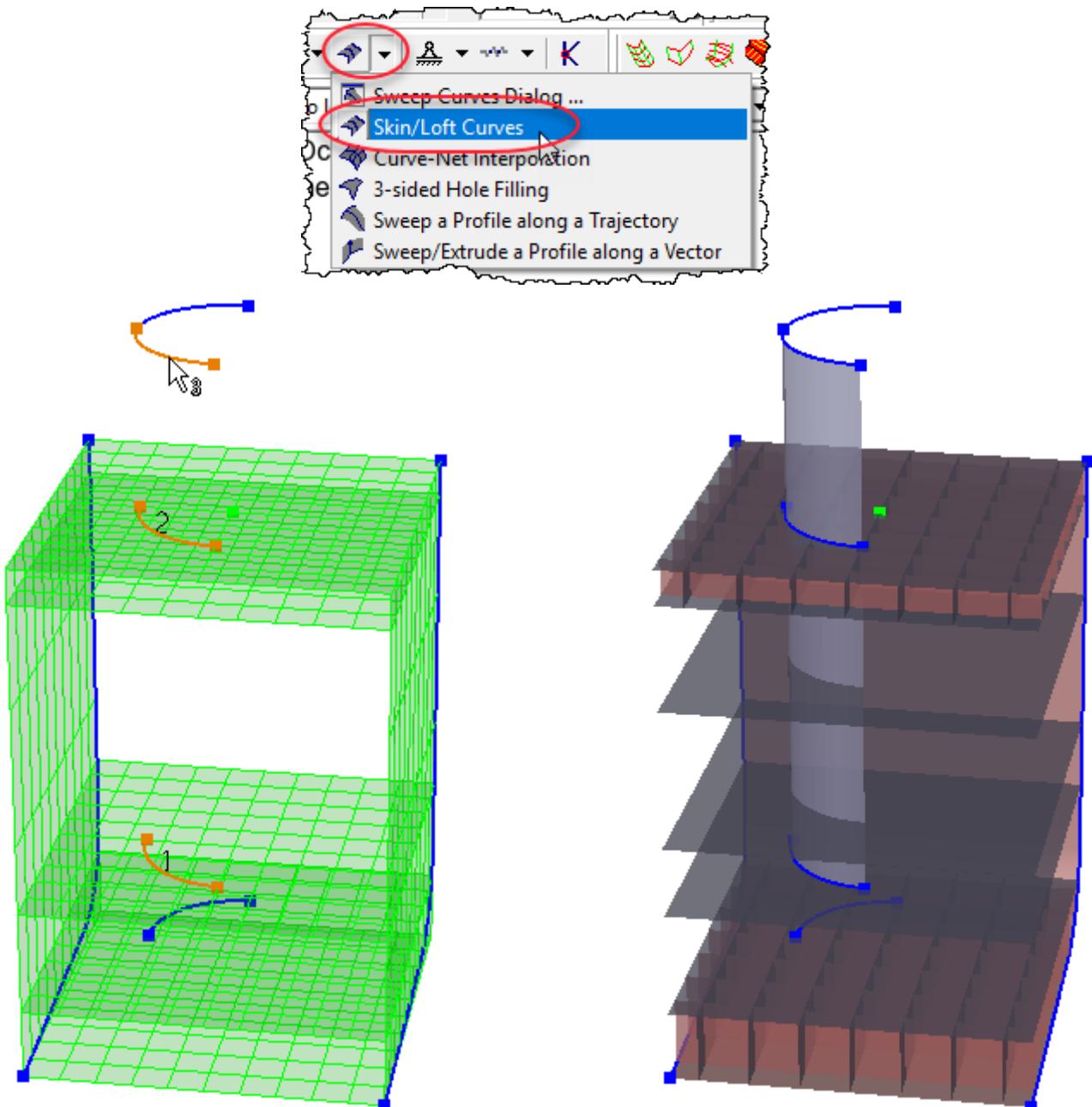
➤ Copy by a 90-degree rotation the upper and lower guide curves as shown to the right below.



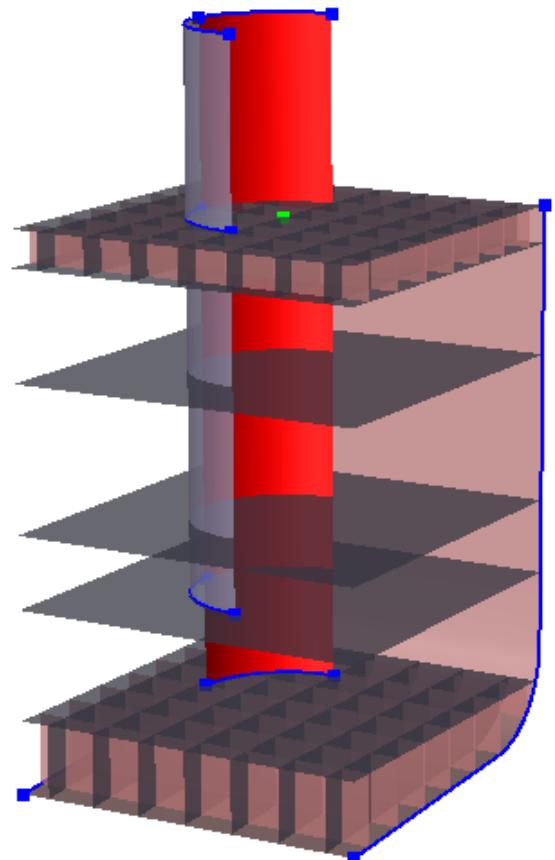
➤ Select the guide curve highlighted to the right and move it from elevation 5 to elevation 2, i.e. a move vector  $(0,0,-3)$ .



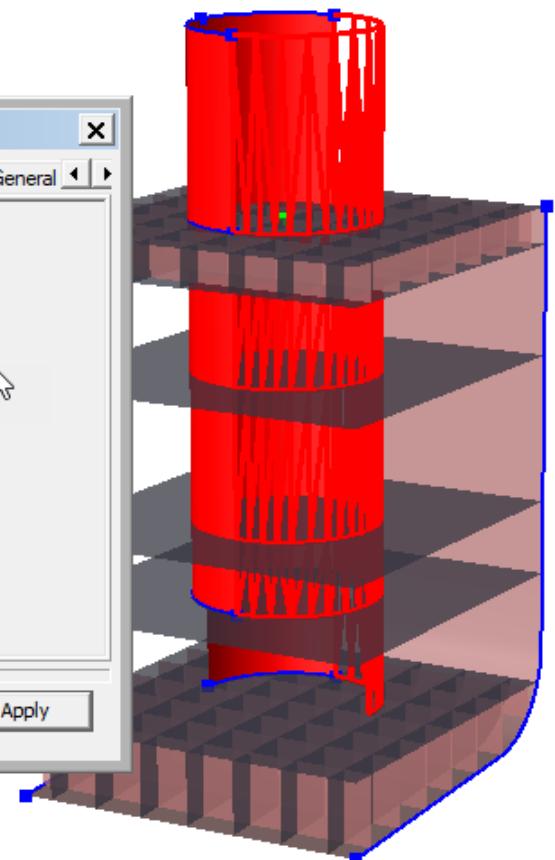
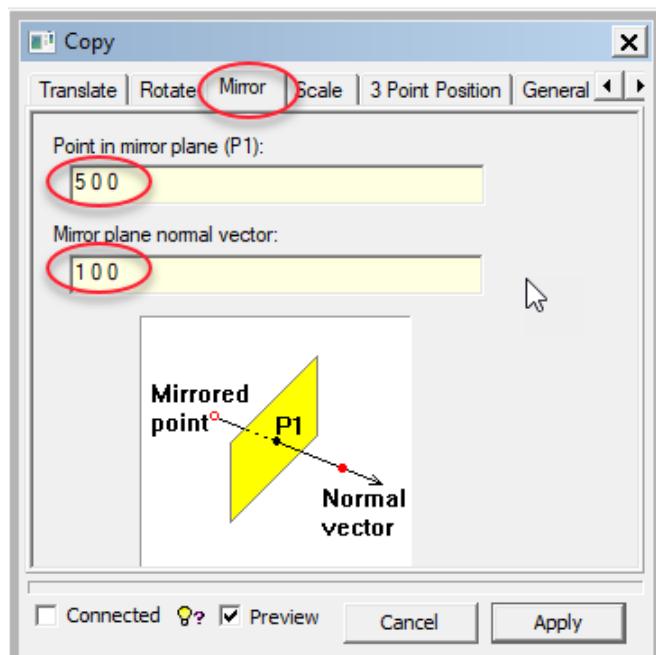
- Create a part of the column by skinning three guide curves as shown below.
- Prior to creating the surface set default plate thickness to Tck30.
  - Remember to double-click the third and last guide curve to close the skinning operation.
  - The result of the skinning operation is shown to the lower right with the eye symbol opened for plate selection button, , and closed for the guide plane selection button, .



- Do another skinning operation to create the surface highlighted to the right.

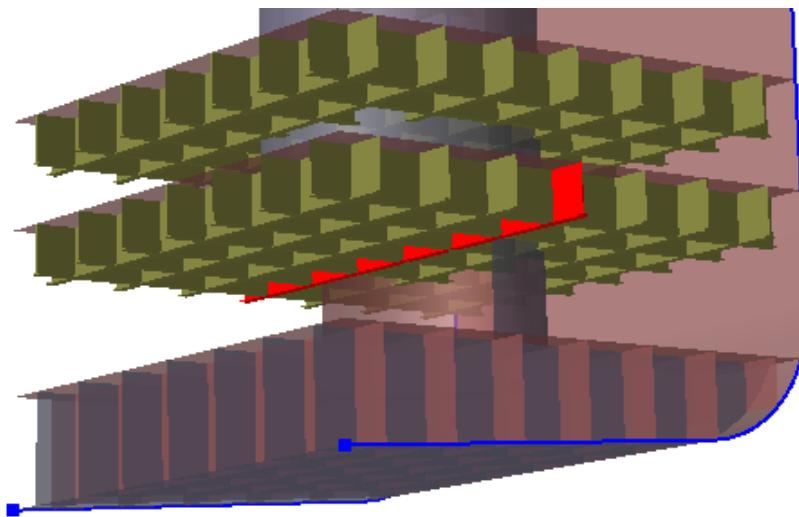


- Copy the two surfaces by mirroring them in the X-direction to complete the column as shown below.

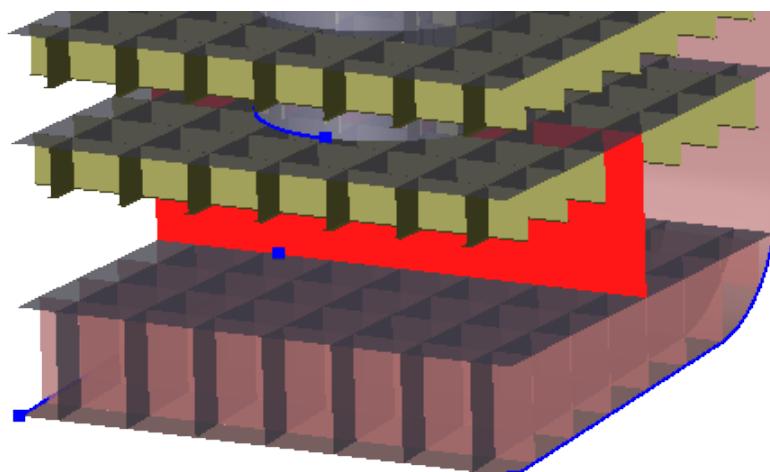
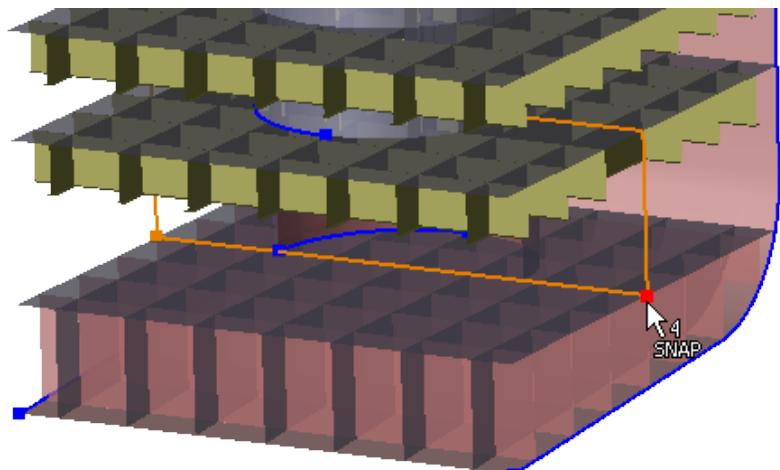


- Replace the stiffener beam at elevation 5 highlighted below with a vertical plate between elevations 2 and 5.

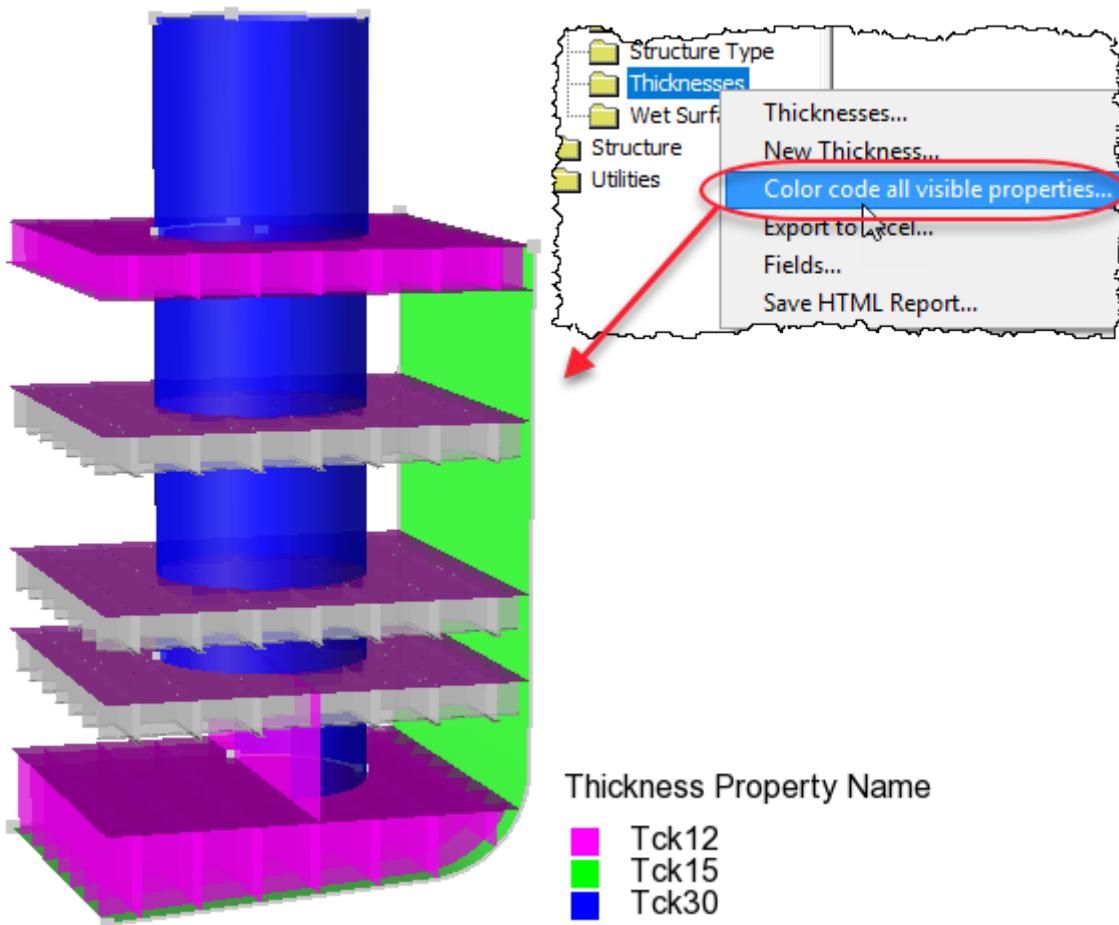
- Open the eye symbol for beam selection button, , and delete the highlighted beam.



- Set default plate thickness to Tck12.
- Create the plate. To click the four corner points you may need to rotate and zoom. This can be done in the middle of the operation.
  - The preview in orange lines only appears when the 4<sup>th</sup> point is coplanar with the three other points.



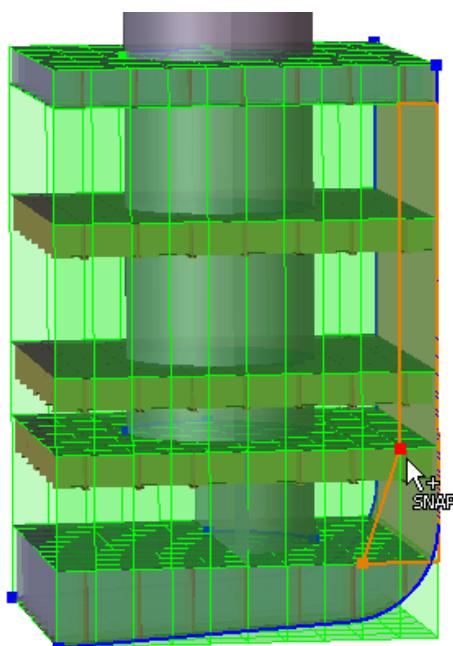
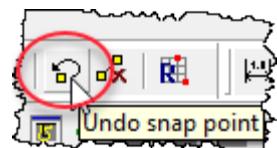
- Verify the plate thicknesses by colour coding. Right-click the *Thicknesses* folder in the browser and press *Color code all visible properties* as shown.
- Deactivate the colour coding by lifting the *Property color coding of all visible structure* button,  , (actually toggling the current colour coding) to return to normal view.



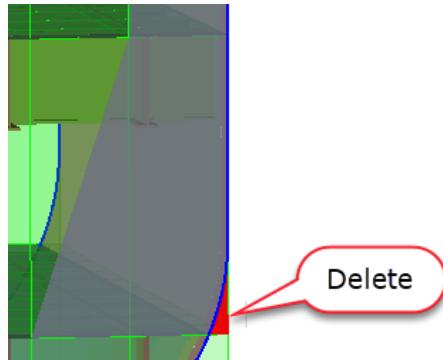
- Use *Structure | Topology | Verify model* to verify the geometric consistency of the model at this point.

## 9 CREATE WEB FRAMES

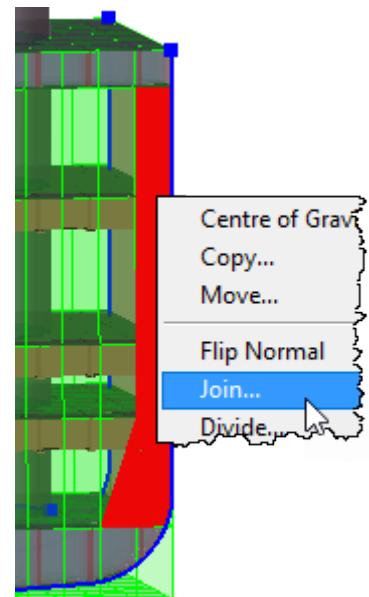
- Create three web frames with thickness 12 mm, i.e. keep the default thickness.
- Use *Default display* configuration and lift the beam and plate selection buttons to make sure clicking snaps to the guide plane.
- Create a plate as shown below in the guide plane at X = 10. It doesn't matter which of the five points is the start point. Double-click the fifth point.
  - Note that if you click one or more erroneous points use the *Undo snap point* button one or more times and continue clicking the proper points.



- The plate intersects the hull, so divide it by *Explode all plates in selection into simpler plates* and thereafter trim it as shown to the right.

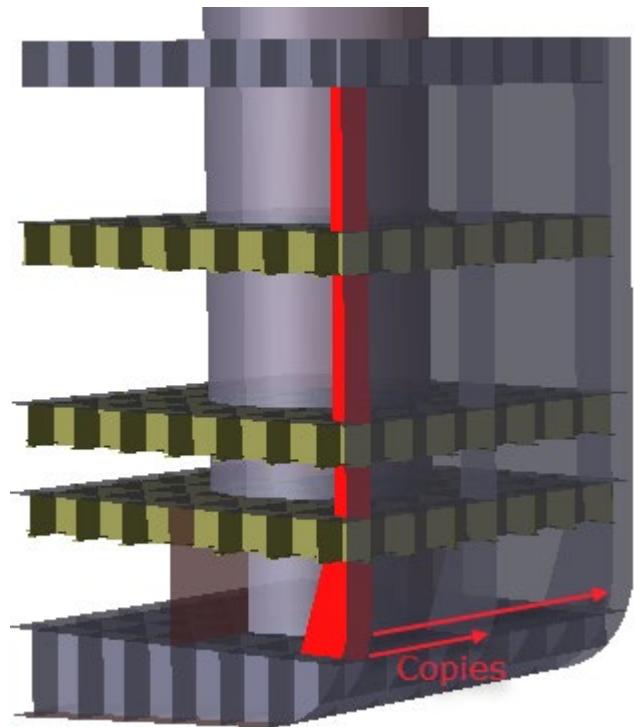
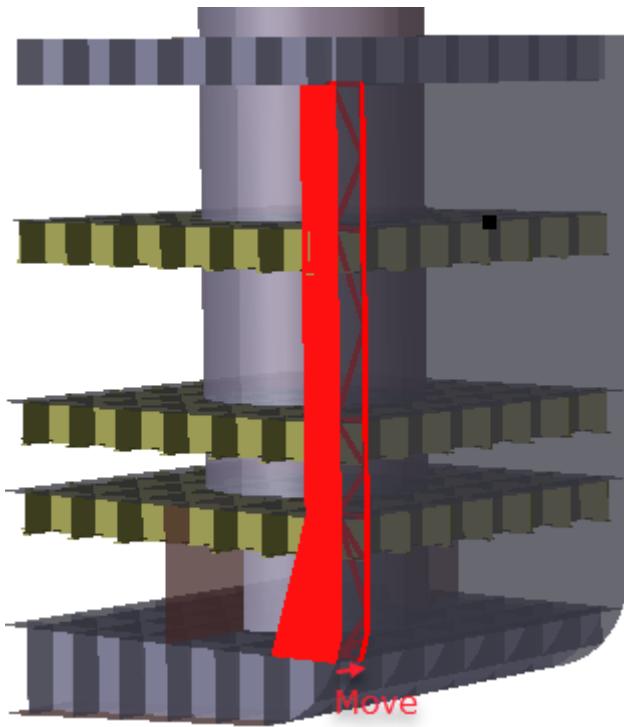


- Join the remaining parts of the plate as shown to the right.



- Alternatively, divide the plate by the guide curve along the hull in which case there is no need for joining.

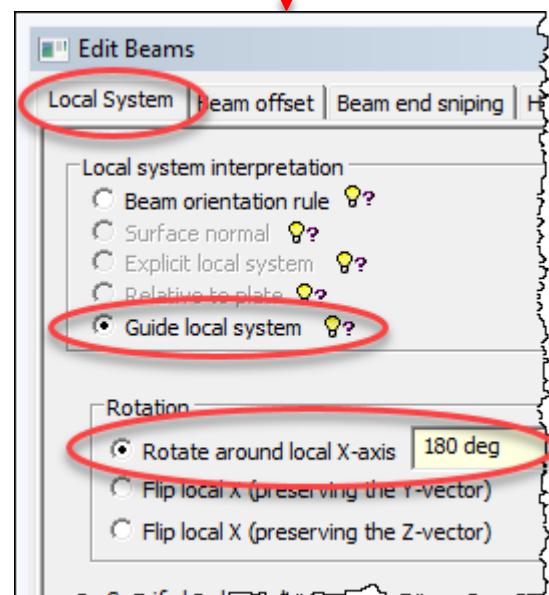
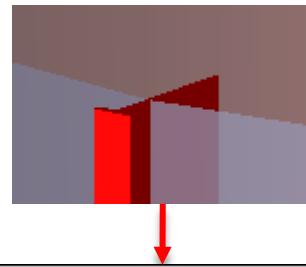
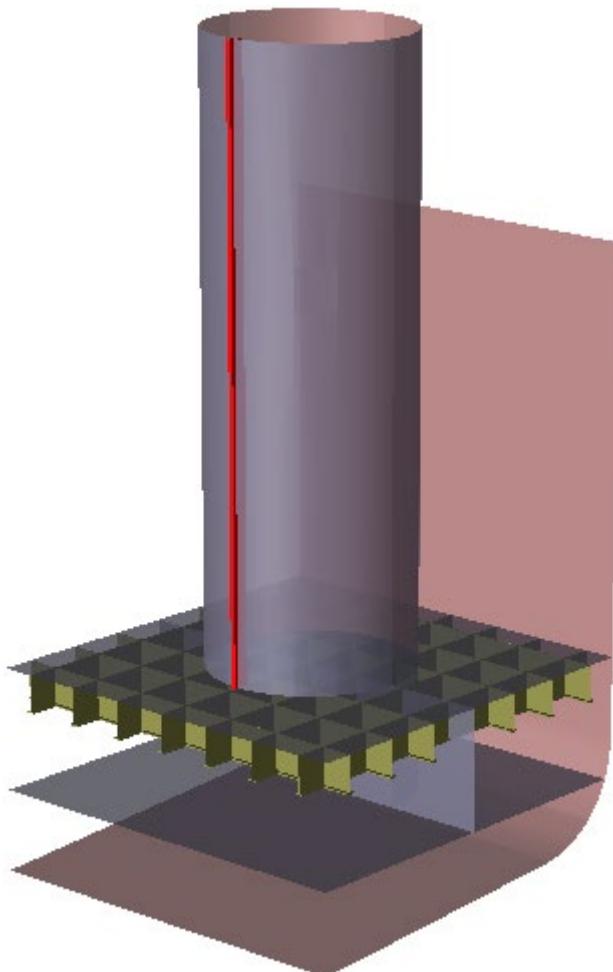
- Move and copy the plate to form three web frames.
  - First move it to match the first stiffener in the YZ-plane, i.e. a vector  $(-0.8,0,0)$ .
  - Then copy it to the mid-position and the other side of the column by using copy vectors  $(-4.2,0,0)$  and  $(-8.4,0,0)$ , respectively.



- Use *Structure | Topology | Verify model* to verify the geometric consistency of the model.

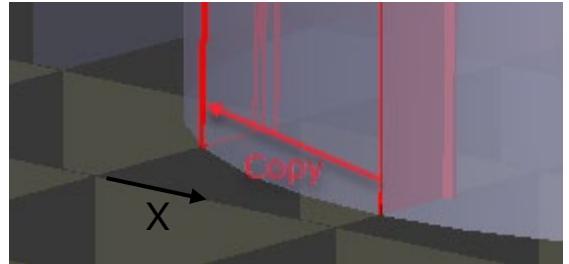
## 10 CREATE COLUMN STIFFENERS

- Create vertical stiffeners inside the column. Use beam section property Tbar425x120x12x25. I.e. set this as default.
- Use *Default display* configuration and display only parts of the model by selection and Alt+S.
- Create the highlighted beam (at X = 5, from Z = 5 to 20).
- Zoom in and notice that the flange is oriented the wrong way. Select it, right-click and press *Edit Beam* and in the *Local System* tab of the *Edit Beams* dialog select *Guide local system* and *Rotate around local X-axis 180 deg*.
- Thereafter, go to the *Beam offset* tab, for *Curve Offset* select *Aligned offset* and set *Alignment* to *Flush Top*.



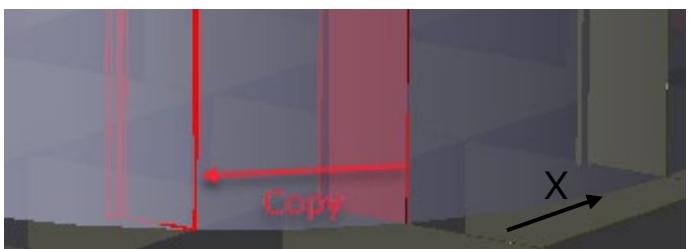
- Copy the stiffener beam as shown in the figure to the right. The figure shows a zoom in on the lower end of the stiffener beam at elevation Z = 5 m.

- The copy vector should be  
 $(-1.4 \text{ m}, 0.4287684823 \text{ m}, 0 \text{ m})$



- Copy the copy as shown.

- The copy vector should be  
 $(-0.6712315177 \text{ m}, 0.6712315177 \text{ m}, 0 \text{ m})$

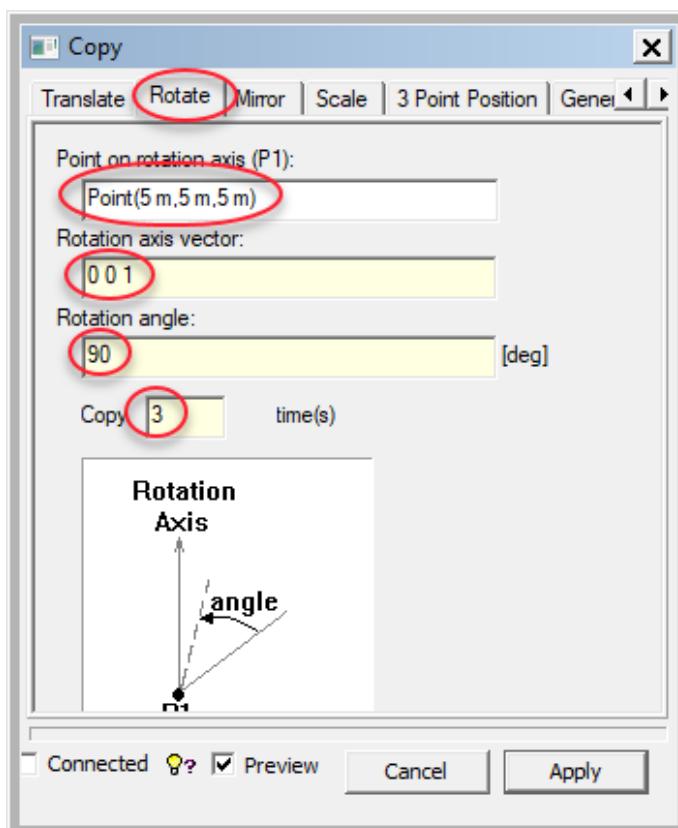


- The orientation of the local axis system of the last copy is incorrect. The web should be in XZ-plane as shown to the right.

In the *Local System* tab of the *Edit Beams* dialog, select *Guide local system* and *Rotate around local X-axis* 90 degrees or -90 degrees depending on in which direction the beam was created.

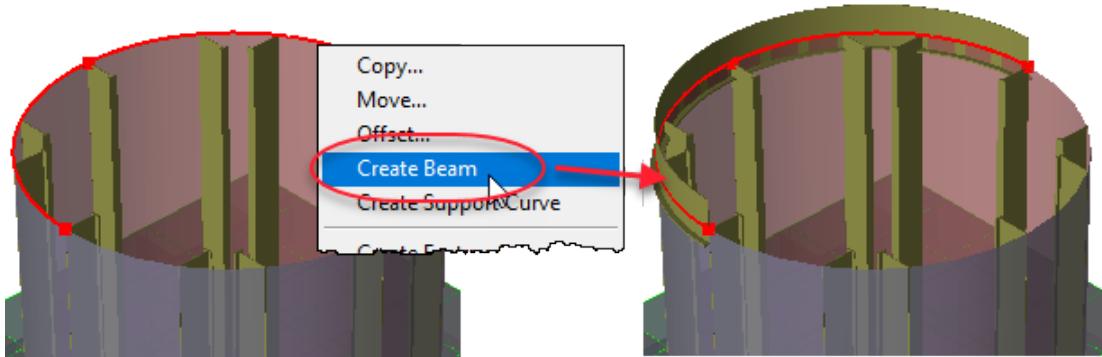


- Copy the three stiffener beams as shown below.

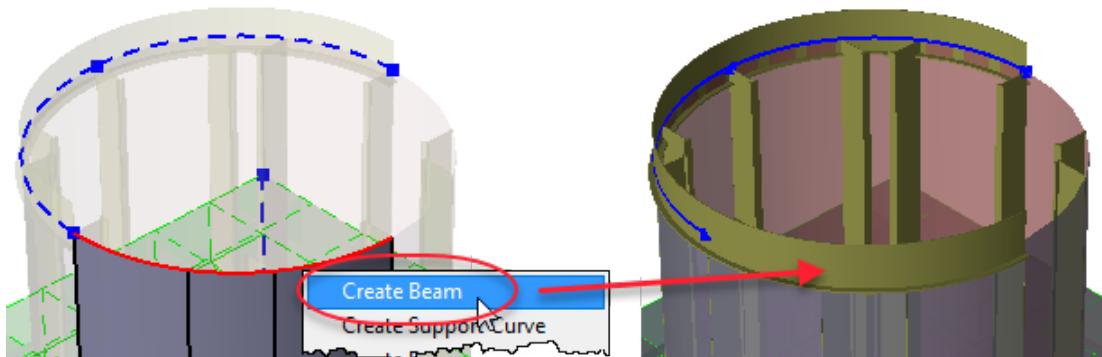


- Create a ring stiffener beam with cross section Tbar575x150x12x25 at top of the column. Do this in three different ways as advised below.

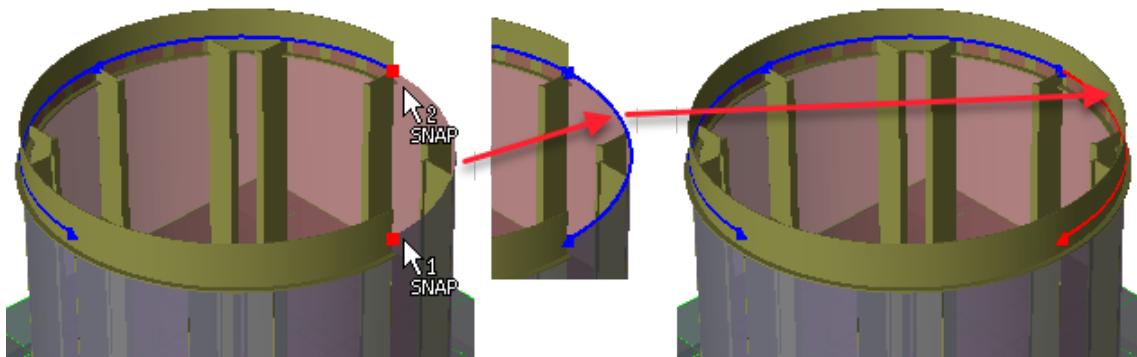
- Set Tbar575x150x12x25 as default beam cross section.
- Use the two 90-degree guide curve arcs (use *Default display* configuration) to create beams. Select them, right-click and click *Create Beam*.



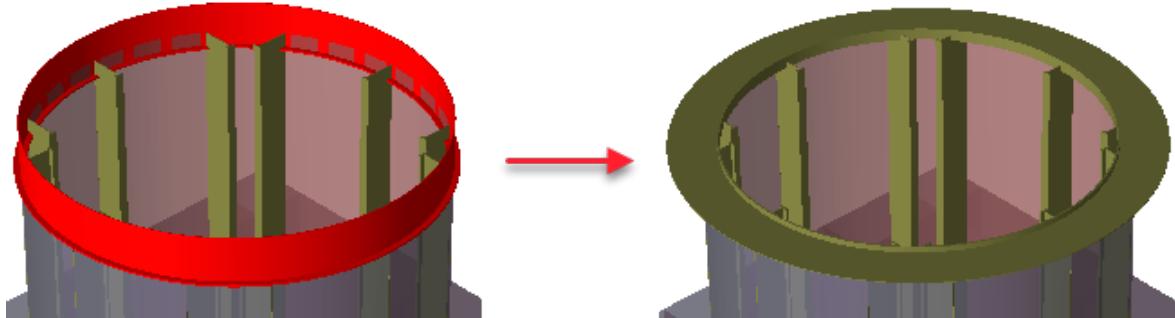
- Double-click a 90-degree sector of the column to display the external and internal edges of the surface. Select the upper edge (three arc segments), right-click and click *Create Beam*. Double-click anywhere outside the model to return to normal display.



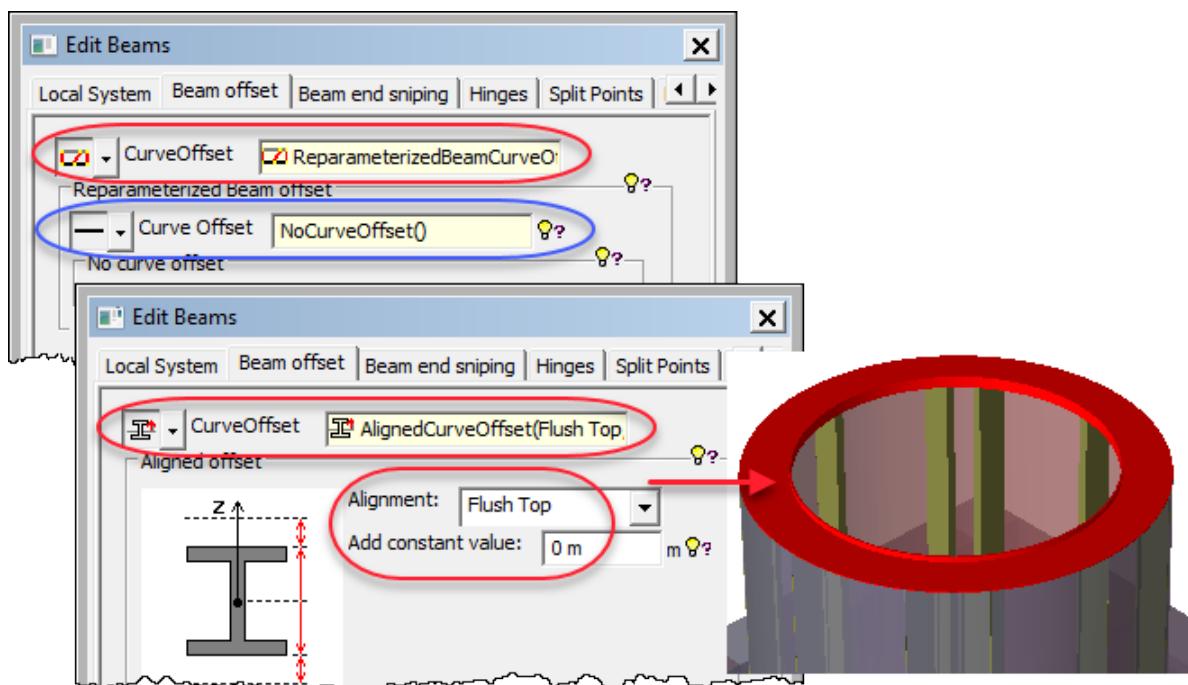
- Create a guide curve of type *Model Curve* by *Guiding Geometry | Curves on Surfaces | Model Curve* or pressing the *Create Curve on the Model* button, . Click the two ends of the guide curve. Then select the guide curve and create a beam in the same was as above.



- Rotate the local axis system 90 degrees so that the flange is on the inside of the column. Note that depending on how the stiffener beams were created they may have different orientation of local axes and therefore need negative or positive rotation around the local x-axis. (First rotate all and then rotate the incorrect ones 180 degrees.)

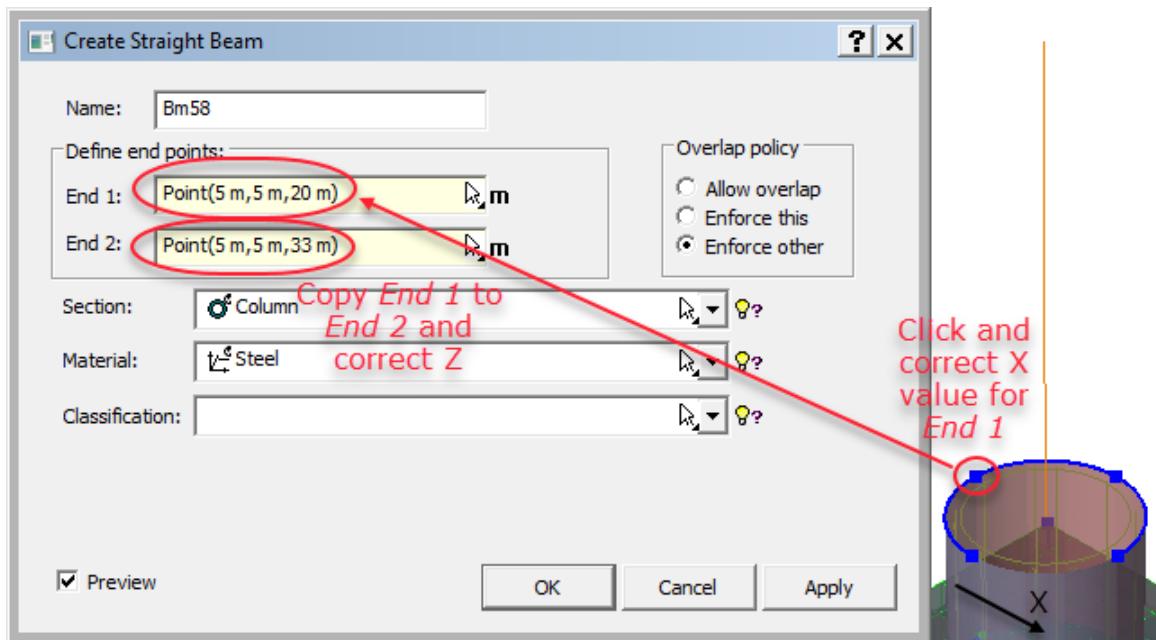


- Flush the top of the section with the column surface.
- By default, a property named *ReparameterizedBeamCurveOffset* is assigned to beams. It has a potential effect on beam offsets but is *not* an offset in itself. The property is relevant for frame structures with straight beams only. For curved beams it should be switched off or else the beam offsets may be incorrect.
- Therefore, to flush the sections properly the *ReparameterizedBeamCurveOffset* must be replaced by *Aligned offset*. I.e., select *Aligned offset* in the field encircled in red below rather than the in field encircled in blue.
- Note that through *Edit | Rules | Beam creation* the default may be changed from *ReparameterizedBeamCurveOffset* to *Aligned offset*. If this is done prior to creating the ring stiffener beam then the desired flushing is done automatically.

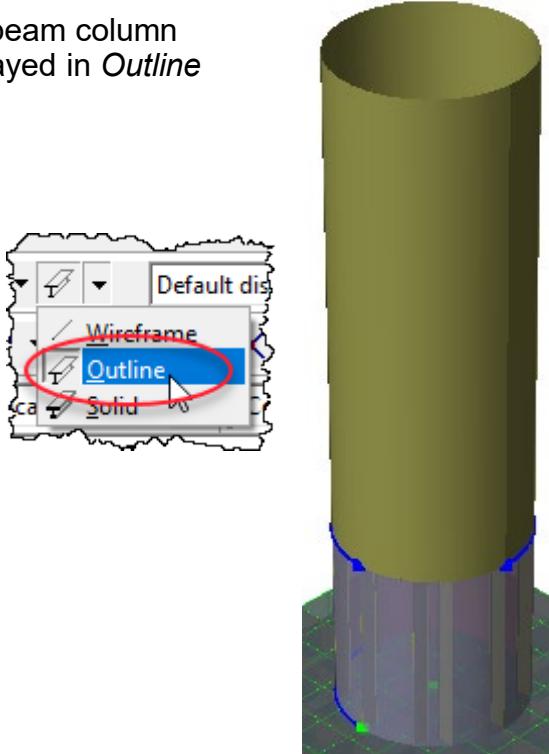


## 11 CREATE CRANE

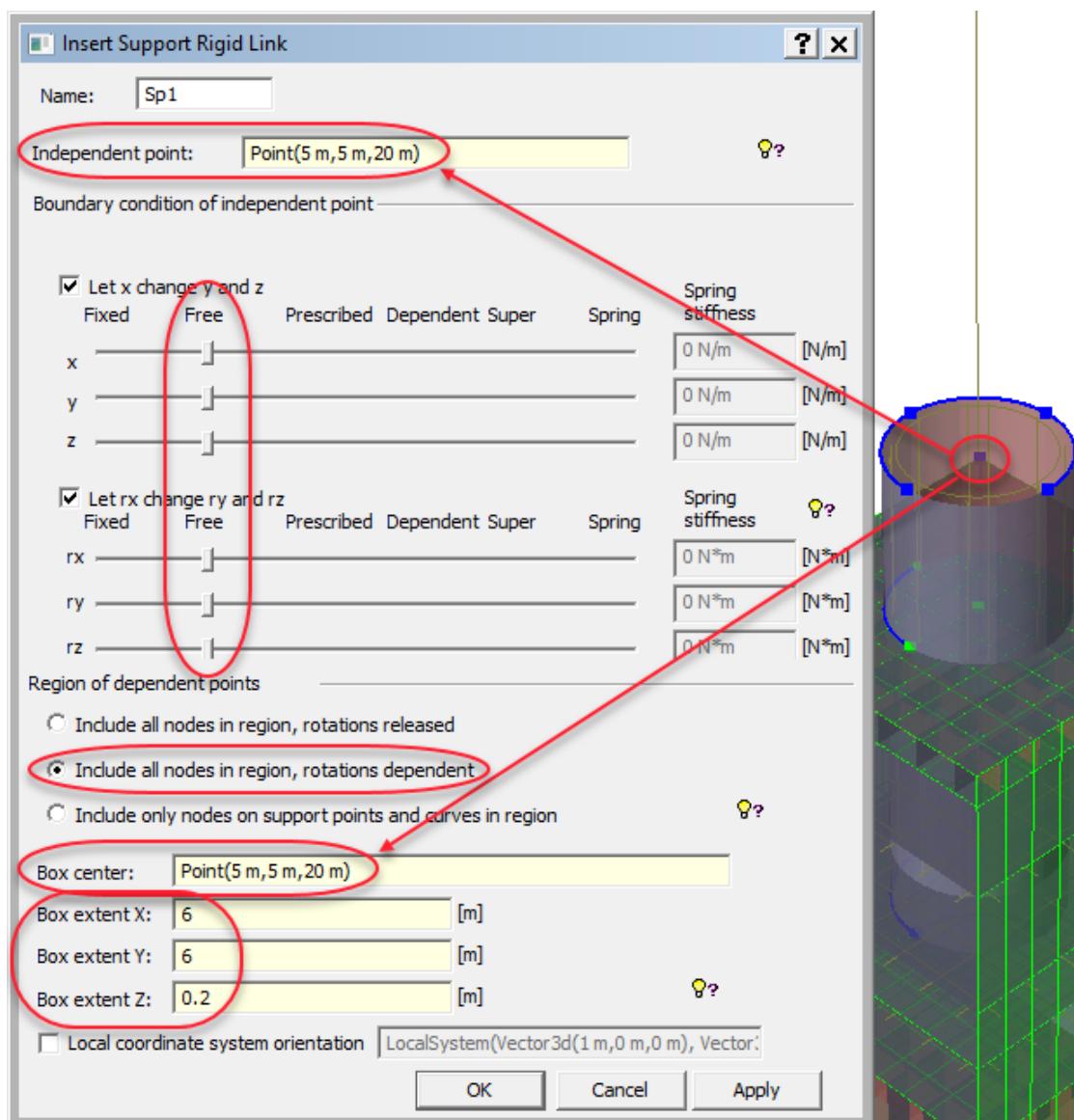
- Create a beam model for the upper part of the crane column.
- Set *Column* as default beam cross section.
- Through the *Create Straight Beam* dialog (*Structure | Beam and Piles | Straight Beam Dialog*) insert the column part of the crane. The coordinates may be found as shown below. The beam preview below is in *Wireframe* view.



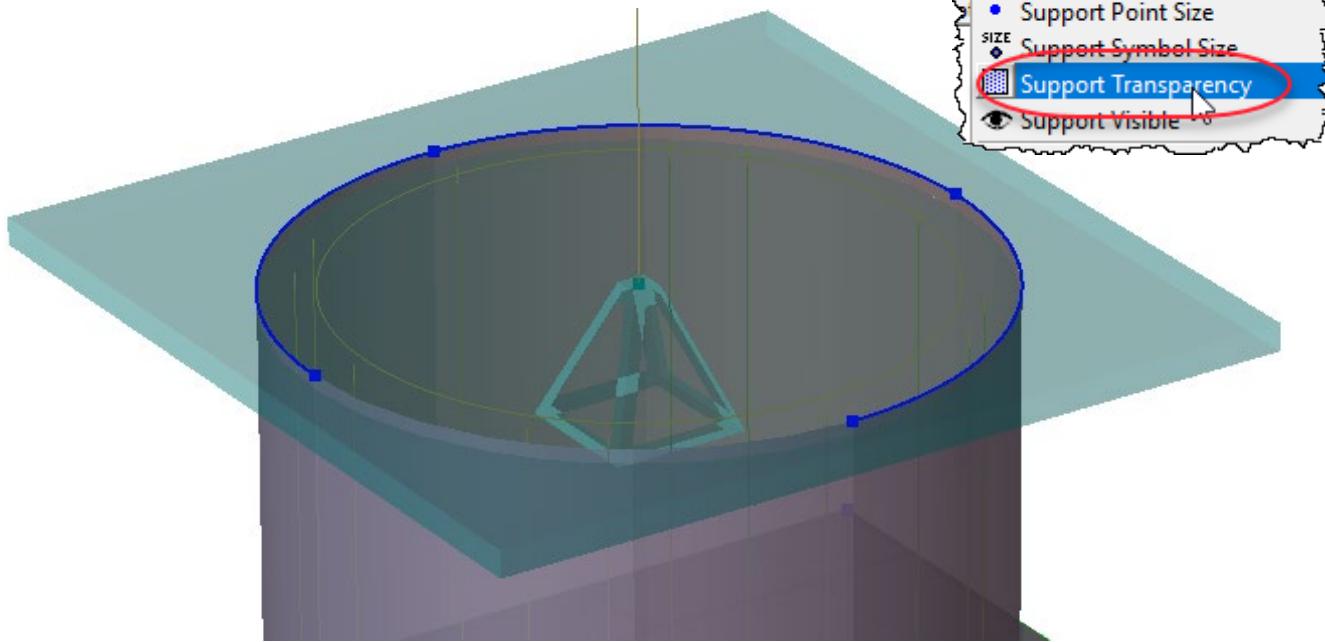
- The beam column displayed in *Outline* view.



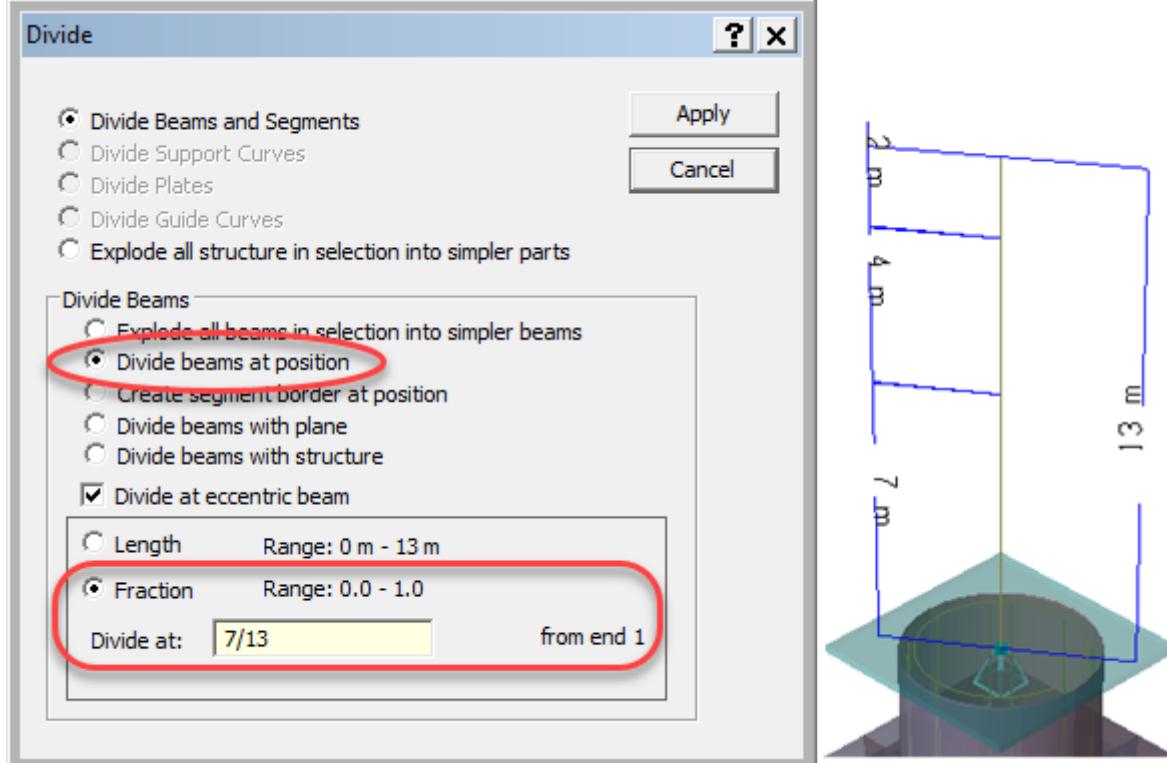
- The beam part of the vertical column shall be coupled with the shell part of the column using so-called *Support Rigid Link*. This coupling ensures proper force and moment transfer and that plane sections through the shell part of the column remain plane in accordance with standard beam theory.
- *Structure | Support | Support Rigid Link Dialog* opens the dialog below. The *Independent point* is the node at the lower end of the beam. The boundary conditions for this node is free for all 6 dofs. Select *Include all nodes in region, rotations dependent* so that also the rotational dofs are dependent. The dependent points shall be the nodes at the upper end of the shell part of the column. The *Box center* and *Box extent* forms a box enclosing the nodes being dependent of the *Independent point*.



- To verify the *Support Rigid Link* make support symbols transparent by right-clicking the *Support selection* button, e.g. 75 % transparent, and zoom in.

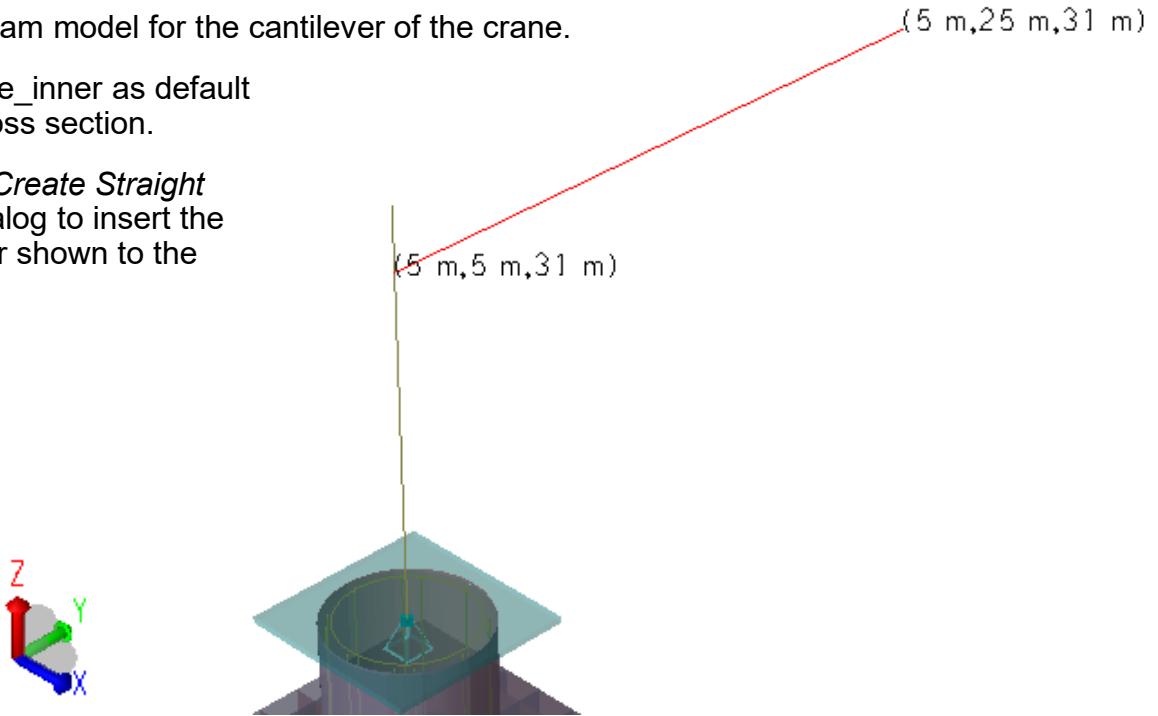


- Divide the beam part of the column as shown. Divide using the *Length* or *Fraction* options in the *Divide* dialog as desired. Note that arithmetic operations may be done as shown to divide it at  $7/13$  measured from end 1 which is the lower end.
- The reason for these divisions becomes evident in the next steps.

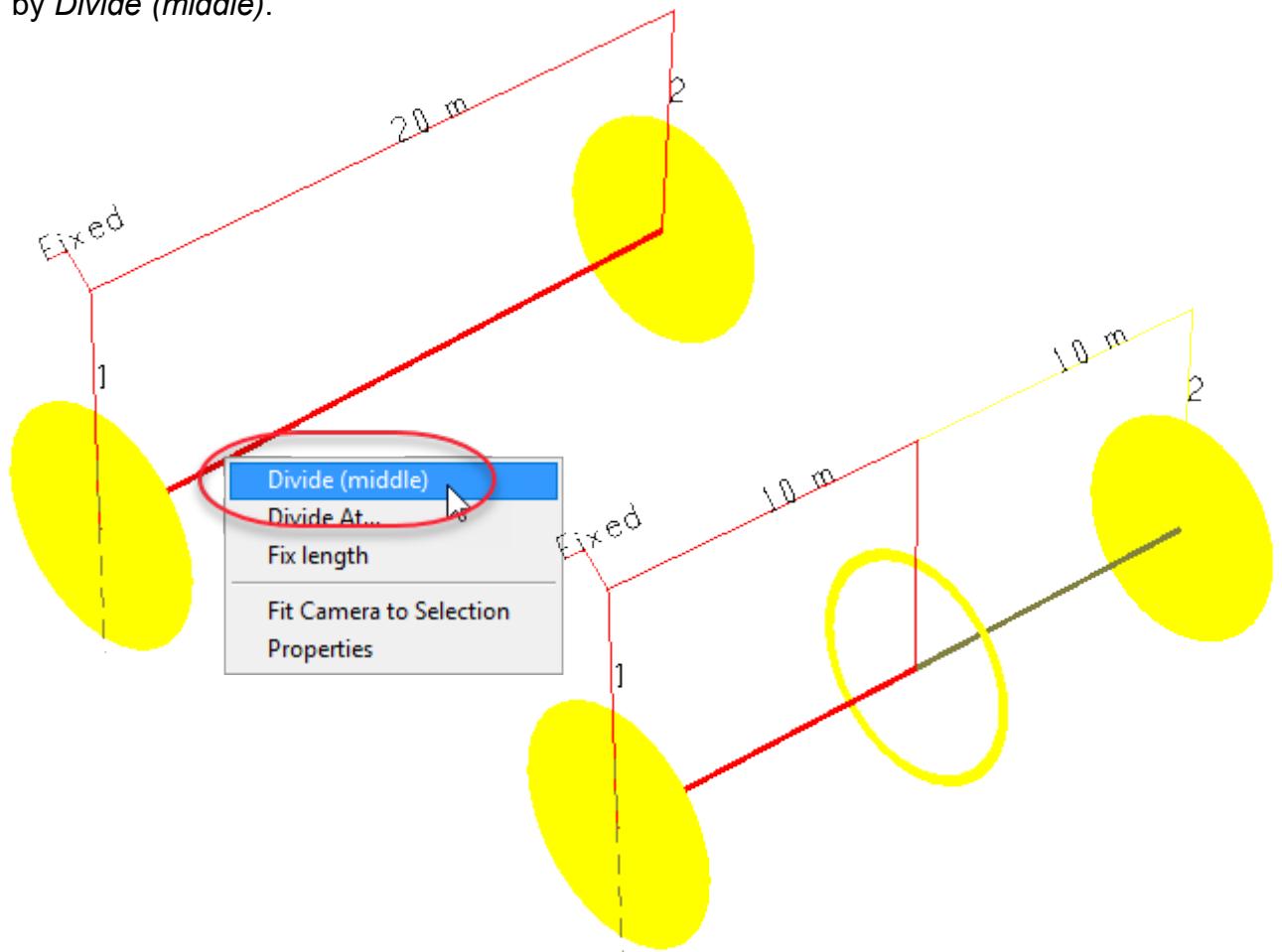


- Create a beam model for the cantilever of the crane.

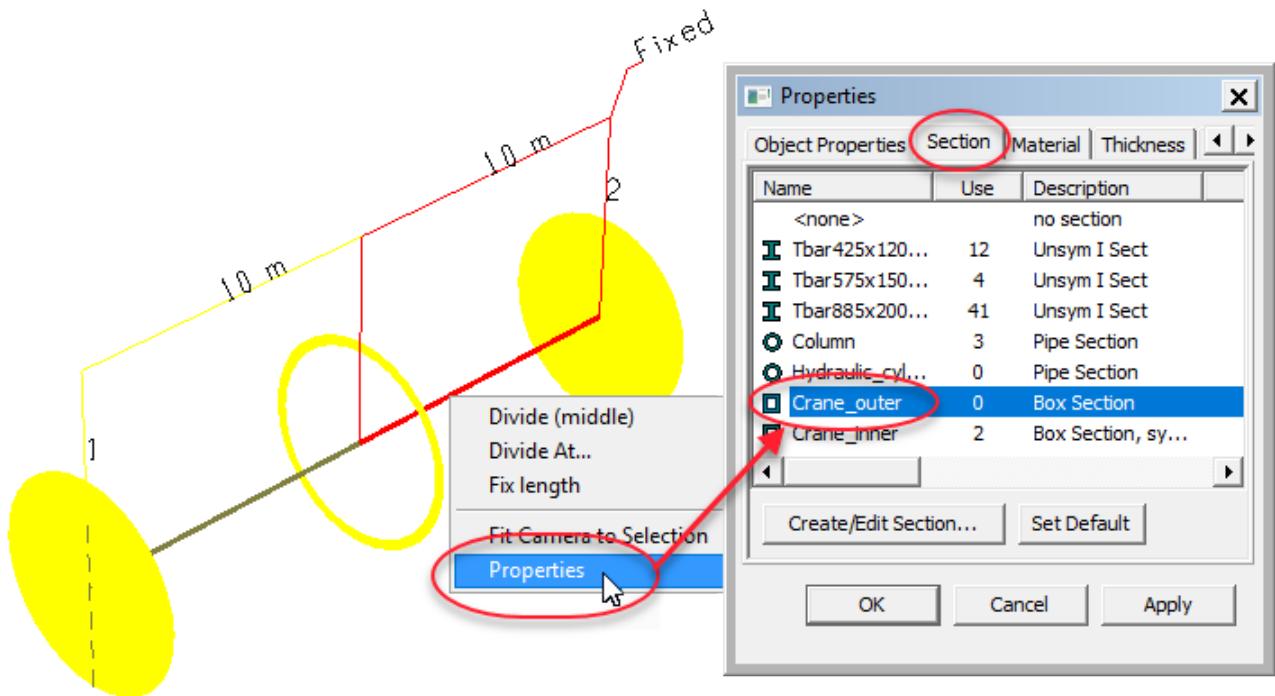
- Set Crane\_inner as default beam cross section.
- Use the *Create Straight Beam* dialog to insert the cantilever shown to the right.



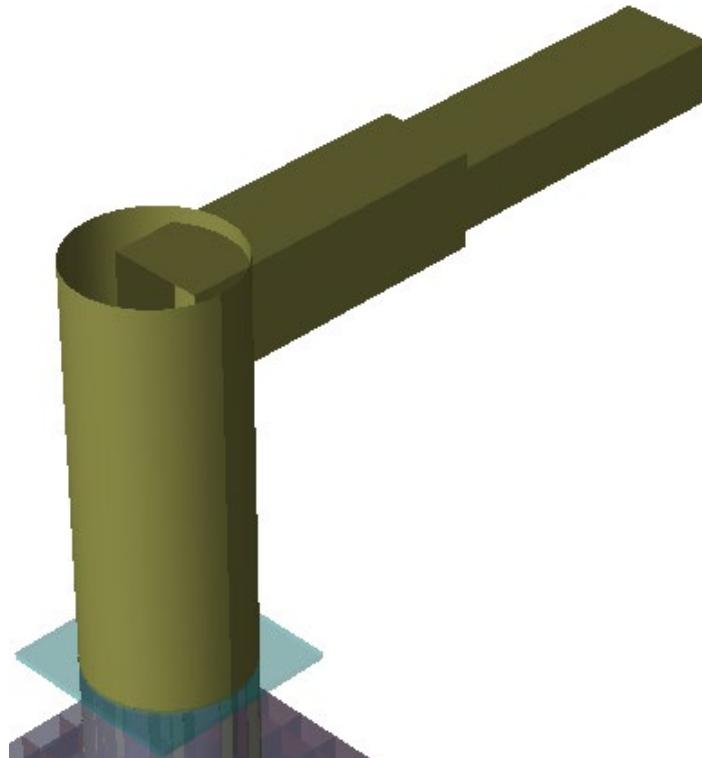
- Double-click the cantilever beam to enter segmentation mode and create two segments by *Divide (middle)*.



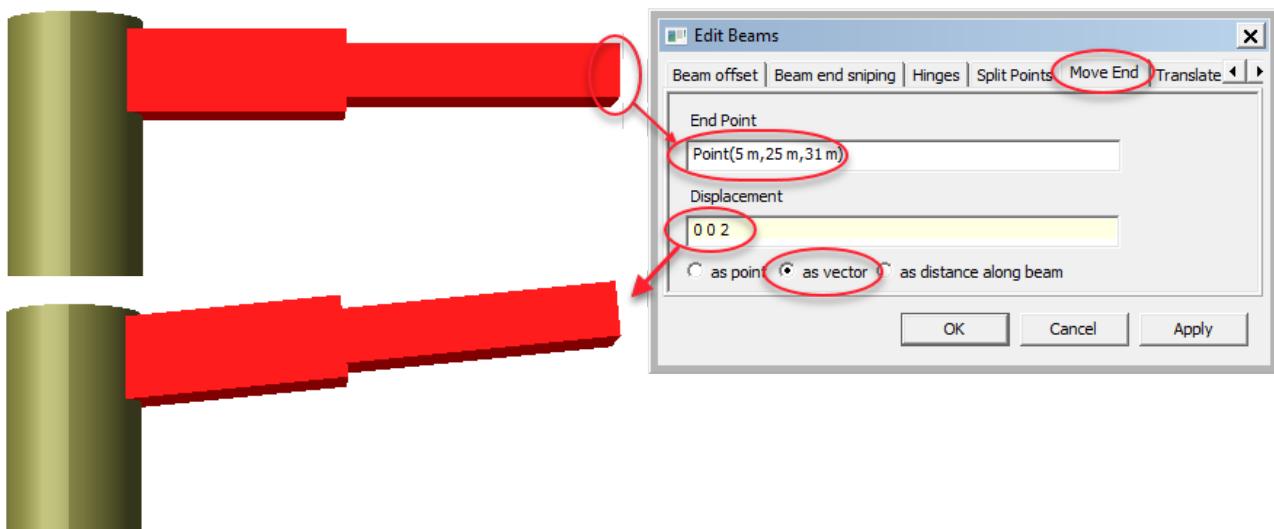
- Select the outer beam segment and change the cross section to Crane\_outer by right-clicking, selecting *Properties*, selecting the proper section and clicking *OK*.



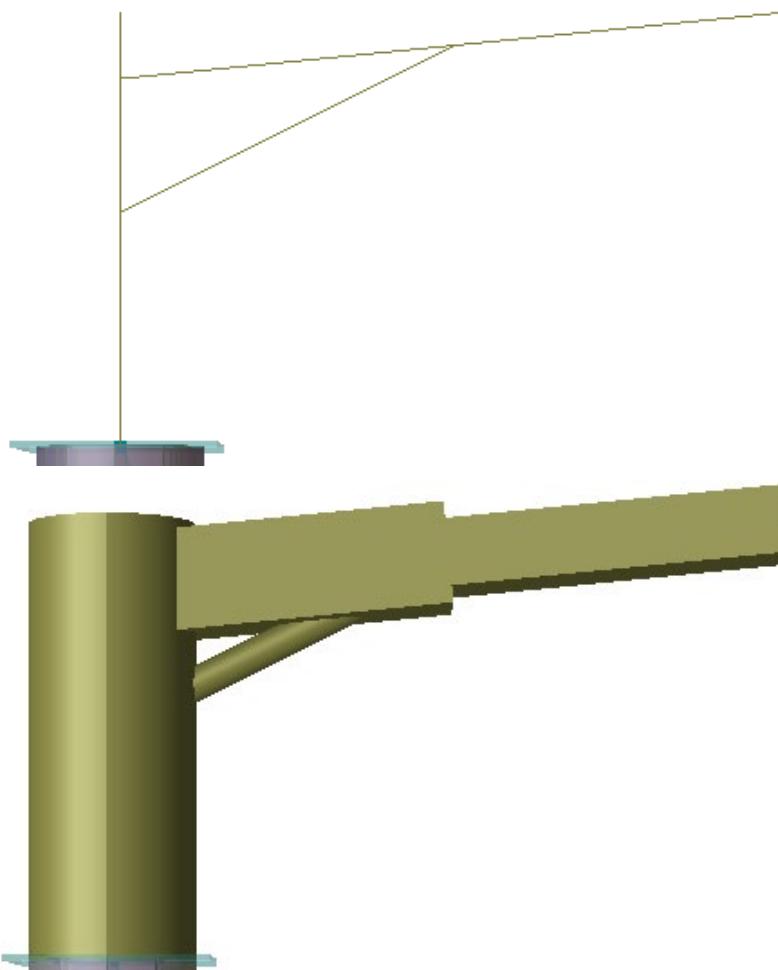
- Double-click anywhere outside the model to leave segmentation mode, switch to *Outline* view and see the result.



- Move outer end of the cantilever a distance of 2 up using the *Edit Beams* dialog.



- Create the hydraulic cylinder beam with section Hydraulic\_cylinder as shown below both in *Wireframe* view and *Outline* view.

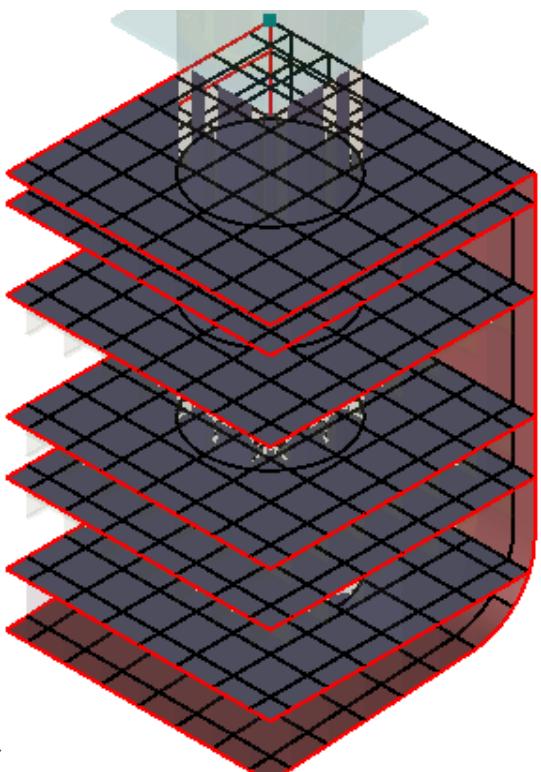
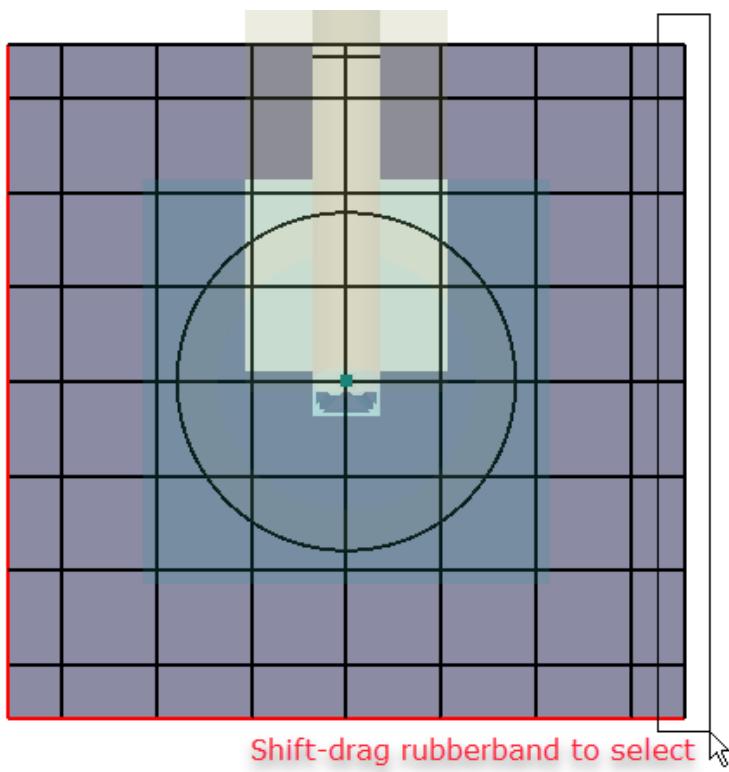
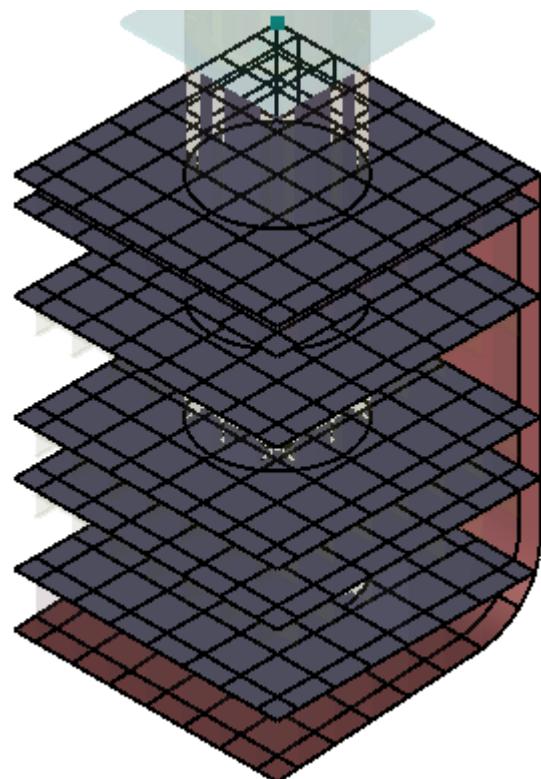


## 12 BOUNDARY CONDITIONS

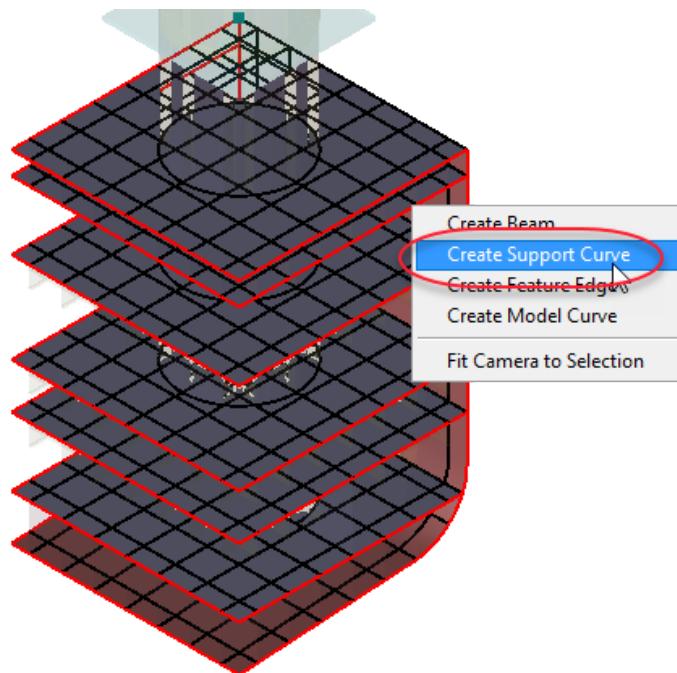
- The boundary conditions along outer edges of decks and the hull are fixed for all translations and free for all rotations.

- To select all outer edges do as follows:

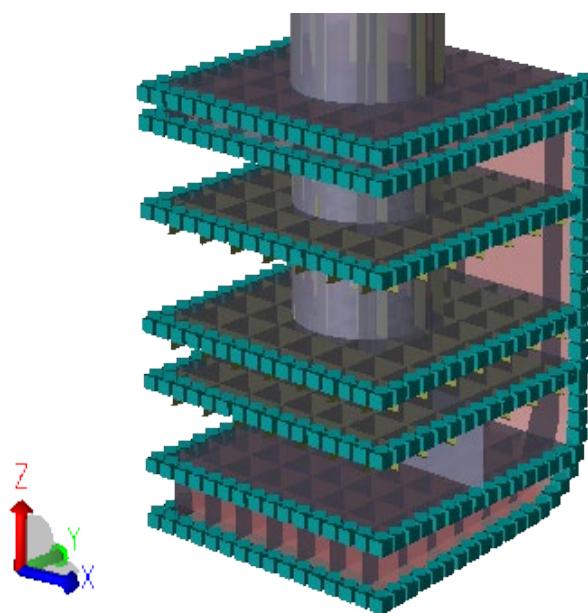
- Select the hull and all decks by click and shift-clicks.
- While keeping the Shift key pressed double-click *outside* the model.
- This will display all selected plates with external and internal edges shown.
- Switch to *Orthographic* view by *View | Options | General* and clicking the icon to the left of the property *Perspective*. Its value changes to *orthographic*. Then press F8 to view the model from above. This allows selecting external plate edges using rubberband selection (dragging from left to right) as shown below.
- The selected edges are shown to the lower right.



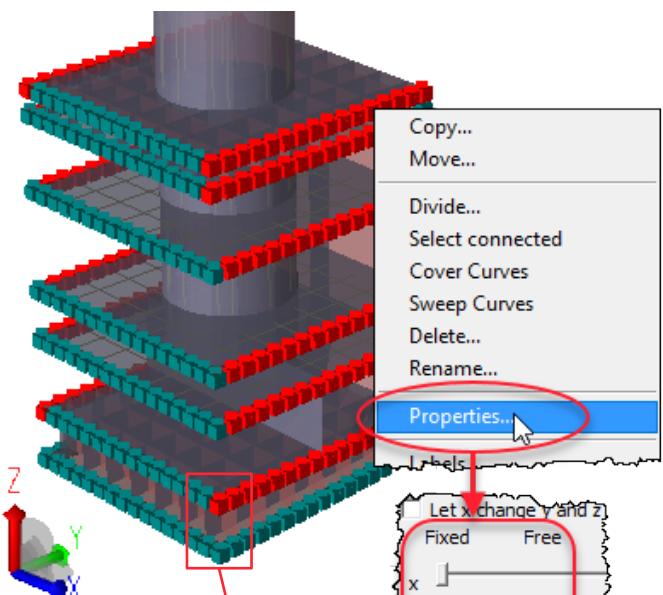
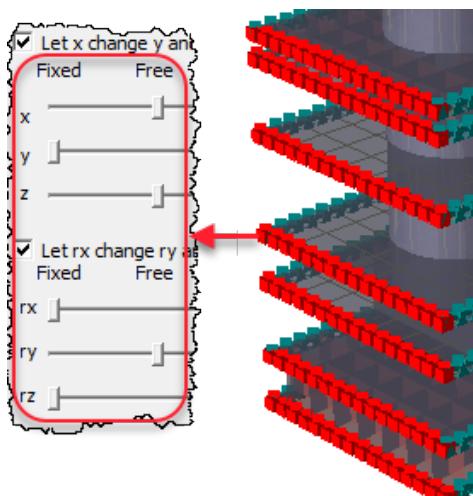
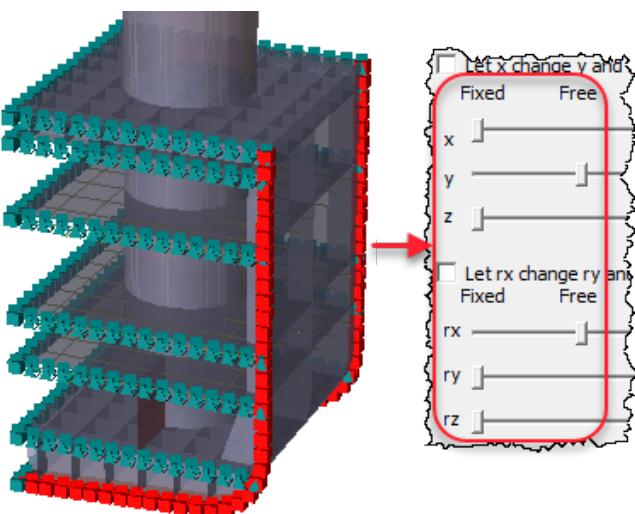
- Right-click the selected edges to create support curves for all.



- Double-click anywhere outside the model to return to normal display mode as shown below.
- By default, all support curves have boundary conditions fixed for all six dofs.
- Change support curves into symmetry conditions as explained next page. (Symmetry conditions are selected in the absence of a better alternative.)
- You may want to return to perspective view.

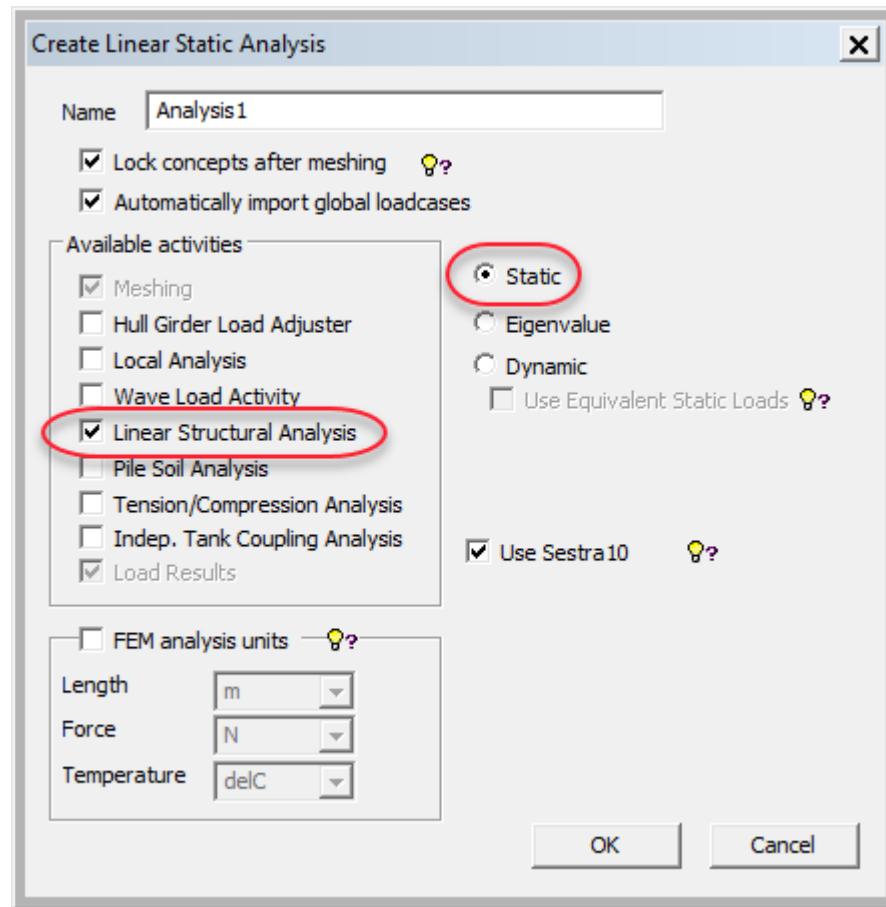


- Select all support curves for the decks in the two YZ-planes, right-click and select *Properties*. In the *Support* tab of the *Properties* dialog set boundary conditions as shown to the right.
  - Select the support curves of the hull in the two YZ-planes and give boundary conditions as shown below.
  - Select all support curves in the XZ-plane and give boundary conditions as shown below.
- Zoom in and see the support curve symbols illustrating the boundary conditions.

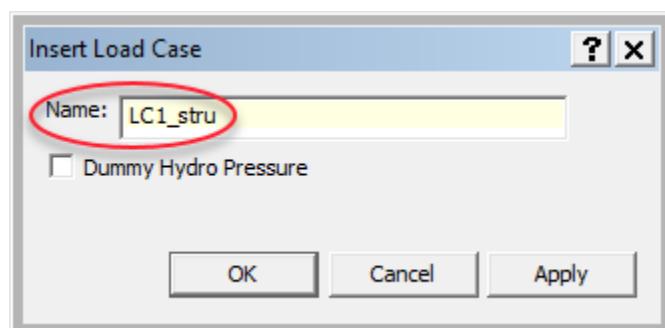


## 13 ANALYSIS ACTIVITY AND LOADS

- >Create an analysis activity by *Mesh & Analysis | Activity Monitor* (or Alt+D). Accept the default settings of *Linear Structural Analysis* and *Static*.



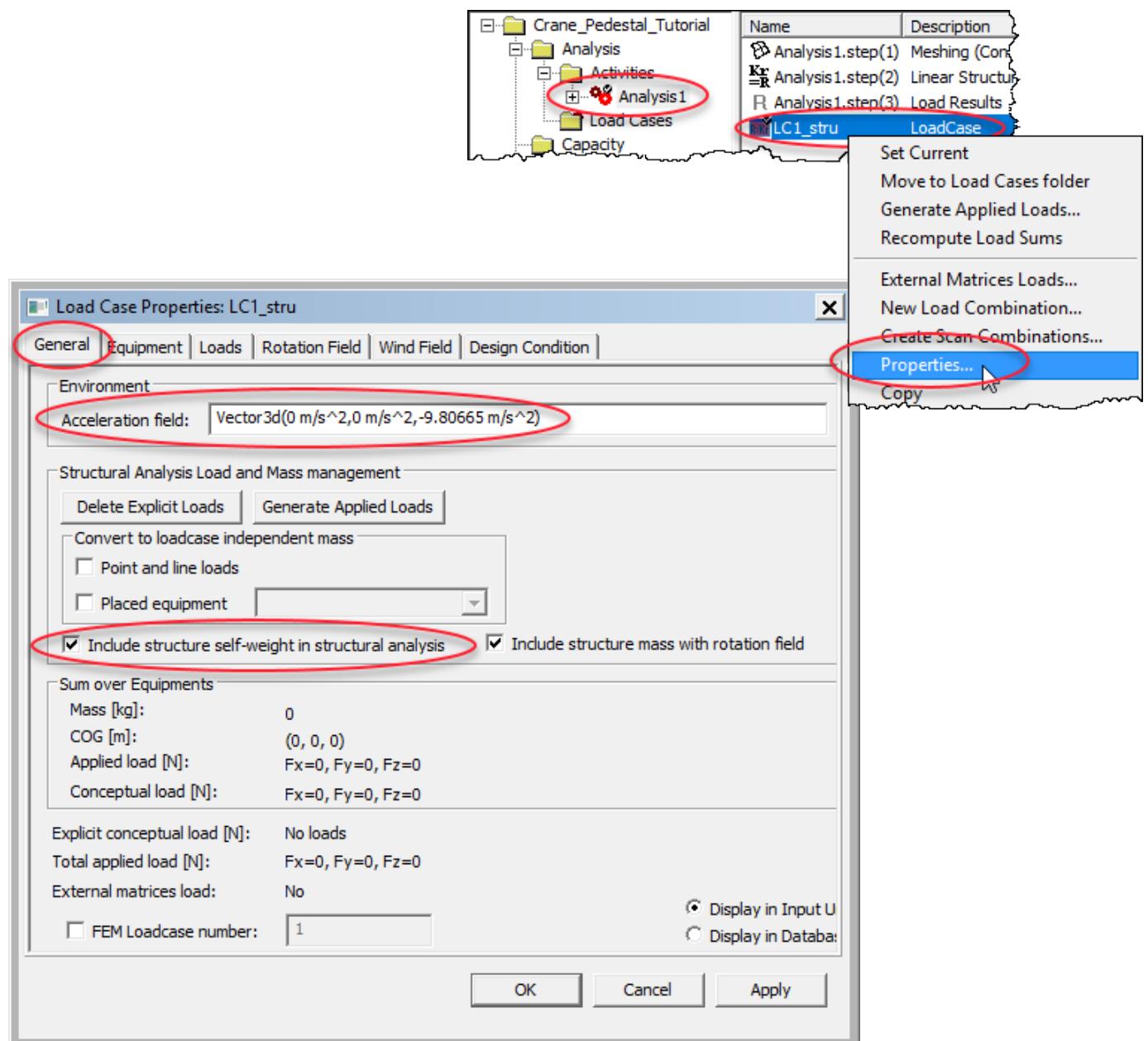
- Clicking *OK* in the above dialog opens the *Activity Monitor*. Cancel this dialog as we don't want to run the analysis yet.
- Create a load case for structure self weight by *Loads | Load Case* and give it the name LC1\_Stru.



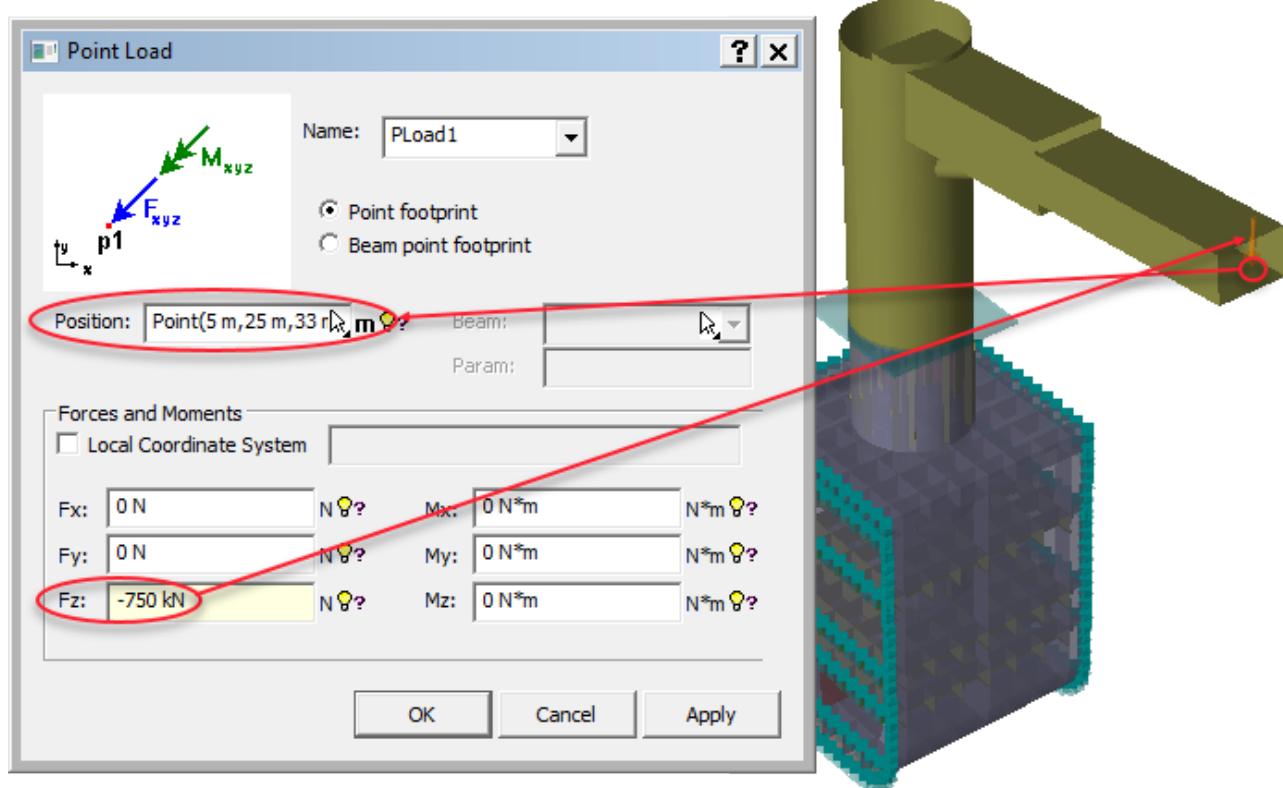
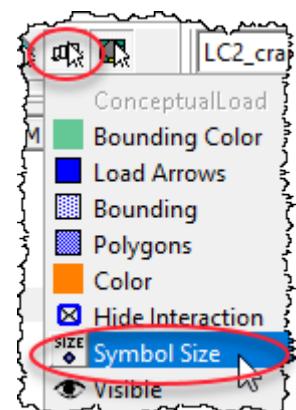
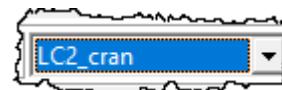
➤ Find the load case in the analysis folder.

- Note: When creating an analysis activity it becomes active. This means that when creating load cases they will belong to the analysis activity. A small check mark indicates that Analysis1 is active.
  - If right-clicking the *Load Cases* folder and selecting *Set Active* then loads created will be stored in the *Load Cases* folder. These loads may then be imported manually or automatically into any analysis activity.

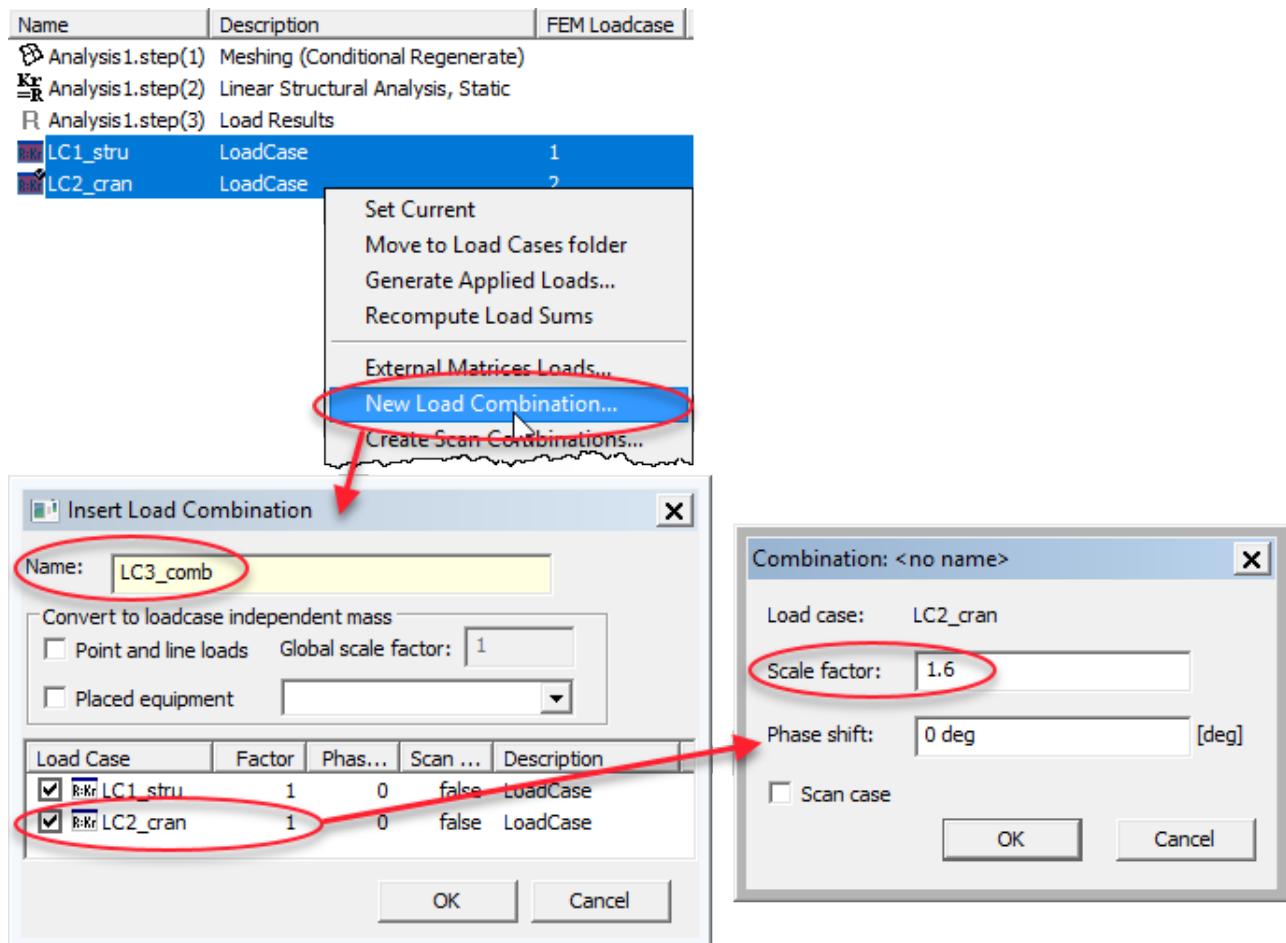
➤ Right-click the load case LC1\_stru, click *Properties* and in the *General* tab of the *Load Case Properties* dialog check the *Include structure self-weight in structural analysis* option. Notice that the default vector in the *Acceleration field* is suitable. Click *OK* in the dialog. The self weight load case is now created.



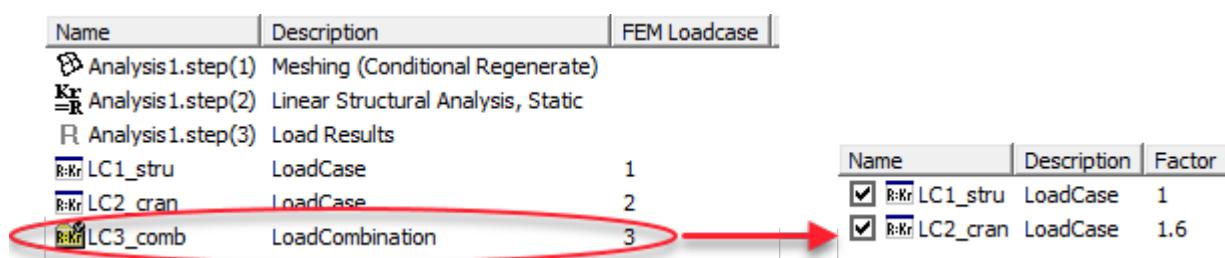
- Create a load case for the crane load by *Loads | Load Case* and give it the name LC2\_cran.
  - This load case is now the current one and explicit loads will be put into this.
  - Note that right-clicking and selecting *Set Current* is used to set a load case current. Or use the load case selector:
  
- Create a downwards pointing load of 750,000 N (750 kN) at the crane tip by *Loads | Explicit Load | Point Load*.
  
- The point load is displayed.
  - Increase the size of the load arrow by right-clicking the *Load selection* button and clicking *Symbol Size*.
  - You may have to click the *Refresh graphics* button, !, to see the effect of the increased size.



- Create a load LC3\_comb as a combination of LC1\_stru with a factor of 1 and LC2\_cran with a factor of 1.6, the latter factor to account for amplification due to ship motion.
- Select the two load cases found in the analysis folder, right-click and select *New Load Combination*.
- In the *Insert Load Combination* dialog double-click LC2\_cran to set the scaling factor.

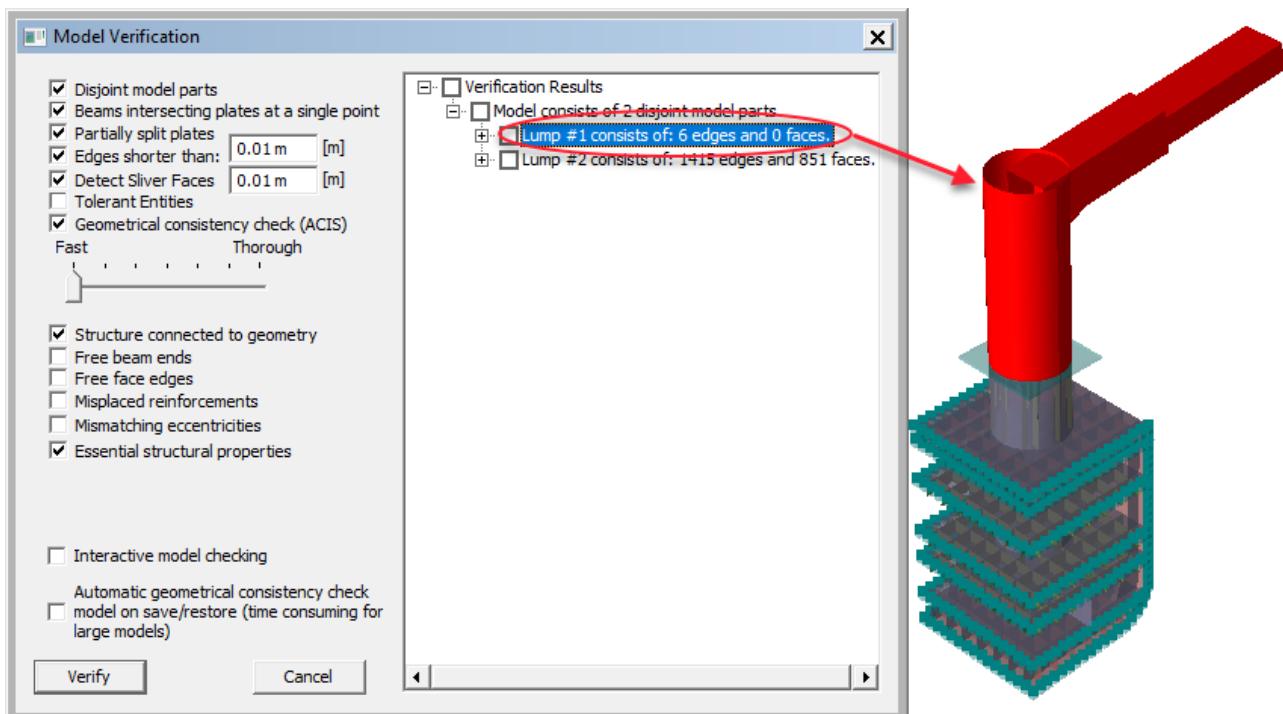


- The load combination is now found in the analysis folder together with the load cases.
- Double-click the load combination to see details.



## 14 VERIFY MODEL

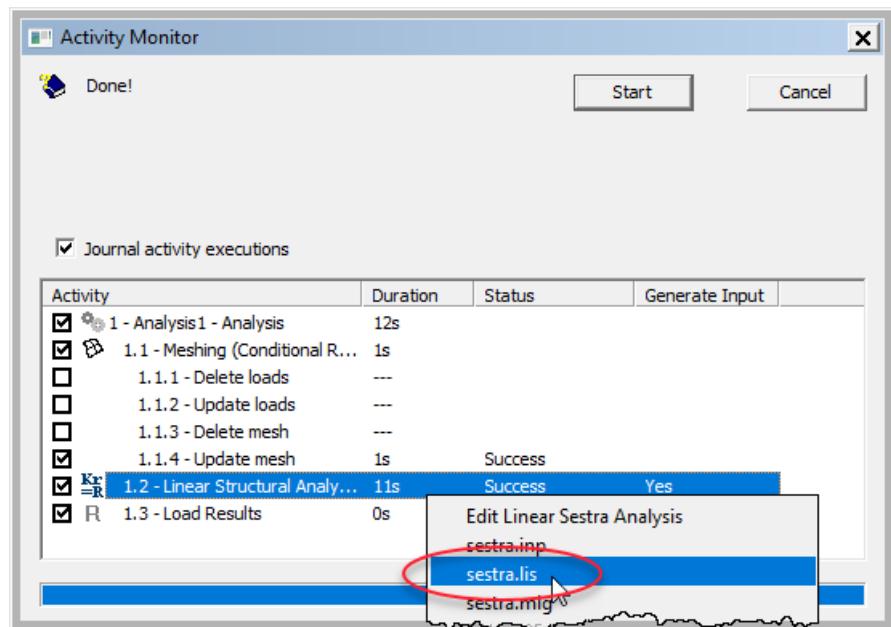
- Use *Structure | Topology | Verify Model*, or click  to open the *Model Verification* dialog.
- Accept all default settings in the dialog and press *Verify*.
- The only *Verification Results* that should appear is *Model consists of 2 disjoint model parts*. Expand this item and click one of the two parts.
  - If there are other issues *Edges shorter than*, *Partially split plate* and *Sliver face* then these problems must be investigated and if necessary rectified.
- The two parts are the upper beam crane part and the lower plate/shell part.
- The reason for these being disjoint is that they are not directly connected. They are only coupled through a support rigid link.
- But knowing that in this case the two disjoint parts are in reality connected the *Verification Results* are acceptable.



## 15 RUN ANALYSIS WITH COARSE MESH

- Open the *Activity Monitor* by *Mesh & Analysis | Activity Monitor* (or ALT+D) and click *Start* to run the linear static analysis.

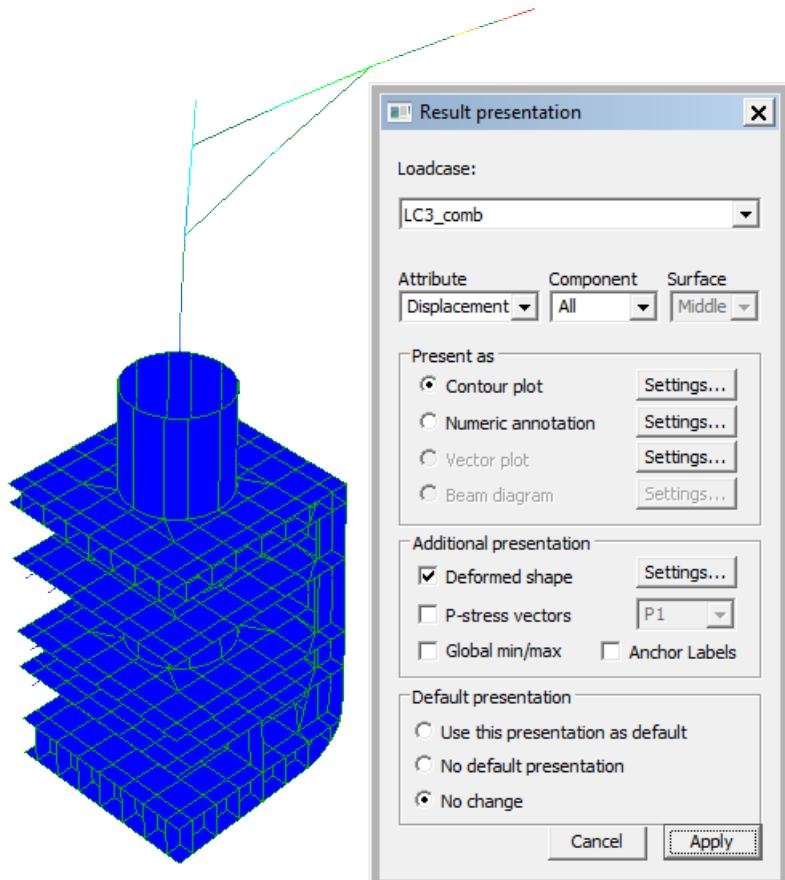
- Open the *Sestra.lis* file containing results for verification:
  - Mass centre
  - Mass matrix
  - Load sum
  - Reaction sum
  - Sum of loads and reactions that should be approximately zero.



- View some results by switching to *Results - with Mesh* and open the *Result presentation* dialog by Alt+P.

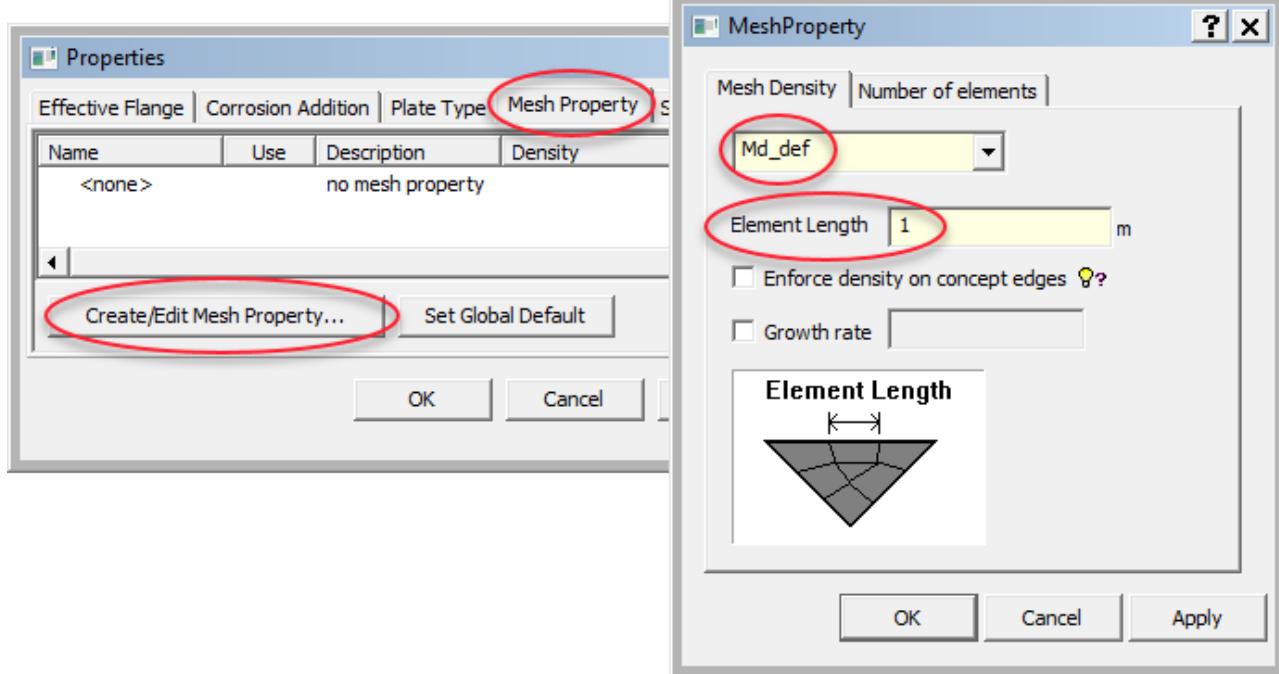
- Select case, e.g. the combination, and *Attribute*, *Component* and, if relevant, *Surface*.
- Displacements are shown to the right. The displacements of the crane, the beam part of the model, dominate.

- In this analysis no mesh property was set. The FE mesh is, therefore, determined solely by the geometry. I.e. the elements are as large and few as possible given the geometry.



## 16 RUN ANALYSIS WITH FINER MESH

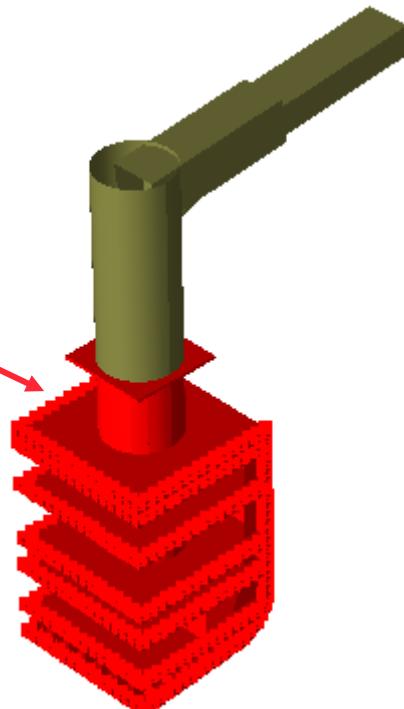
- Create a mesh property with mesh edge length set to maximum 1 m.
  - Use *Edit | Property* to open the *Properties* dialog. In the *Mesh Property* tab click *Create/Edit Mesh Property* to define the mesh density *Md\_def* with *Element Length* set to 1.



- Select the deck and hull part of the model and assign the mesh property to this part. The mesh property is found in the *Properties | Mesh* folder.
  - Alternatively, select all plates from the *Structure* folder in the browser and assign the mesh property to these only with the same effect.

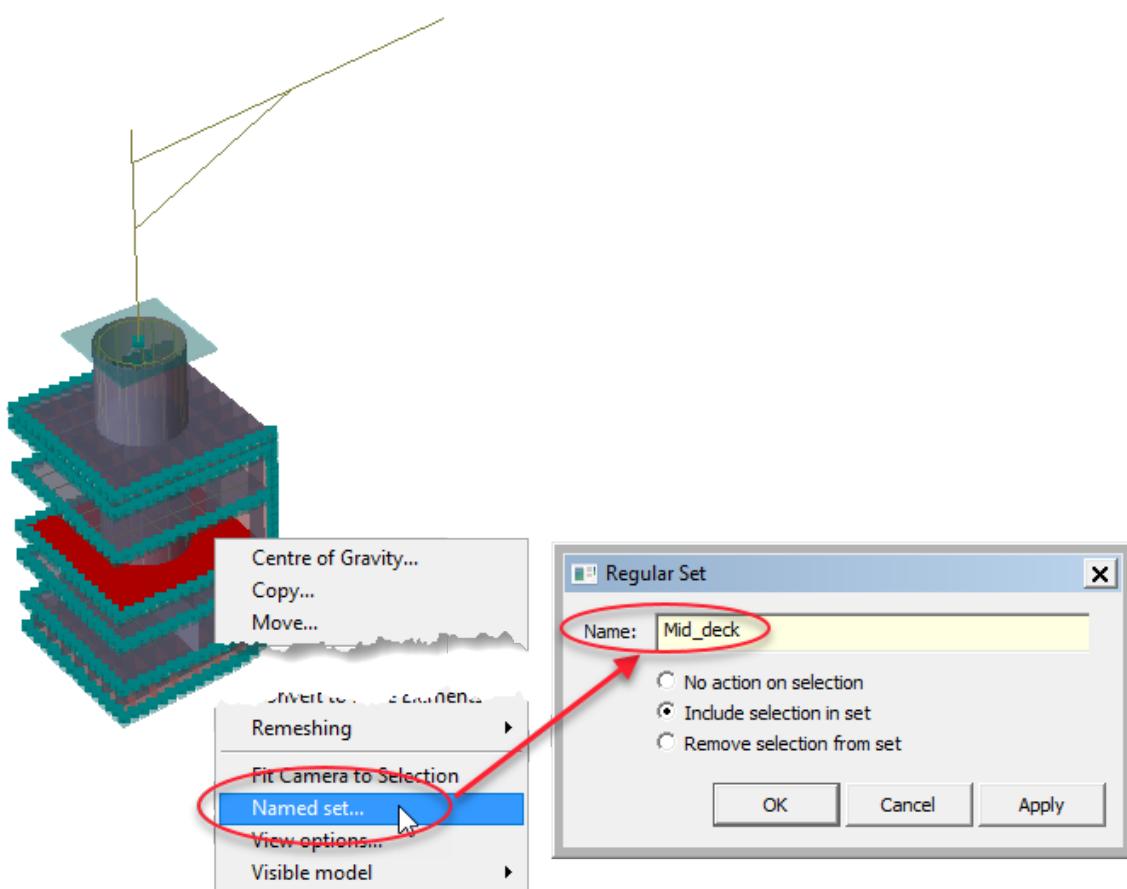
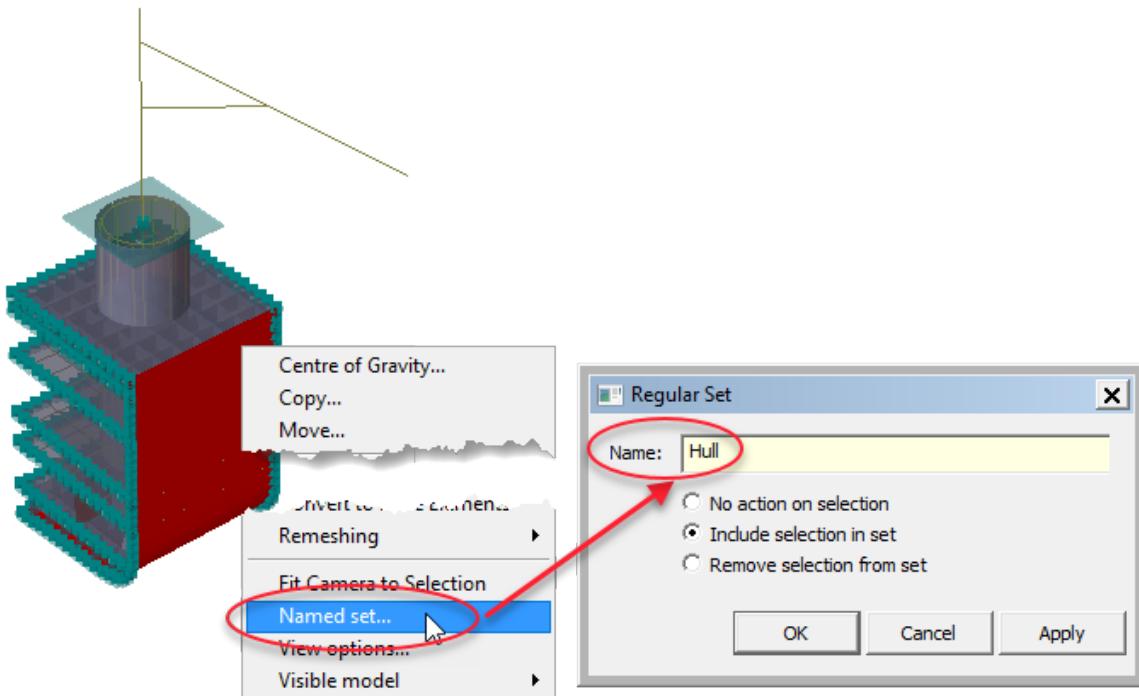


- The reason for assigning the mesh property to the deck and hull part only is that for a beam model (the crane part) there is nothing to gain in accuracy of results by refining the mesh for beams.
- Re-run analysis by Alt+D and clicking *Start*.
- Investigate the results.

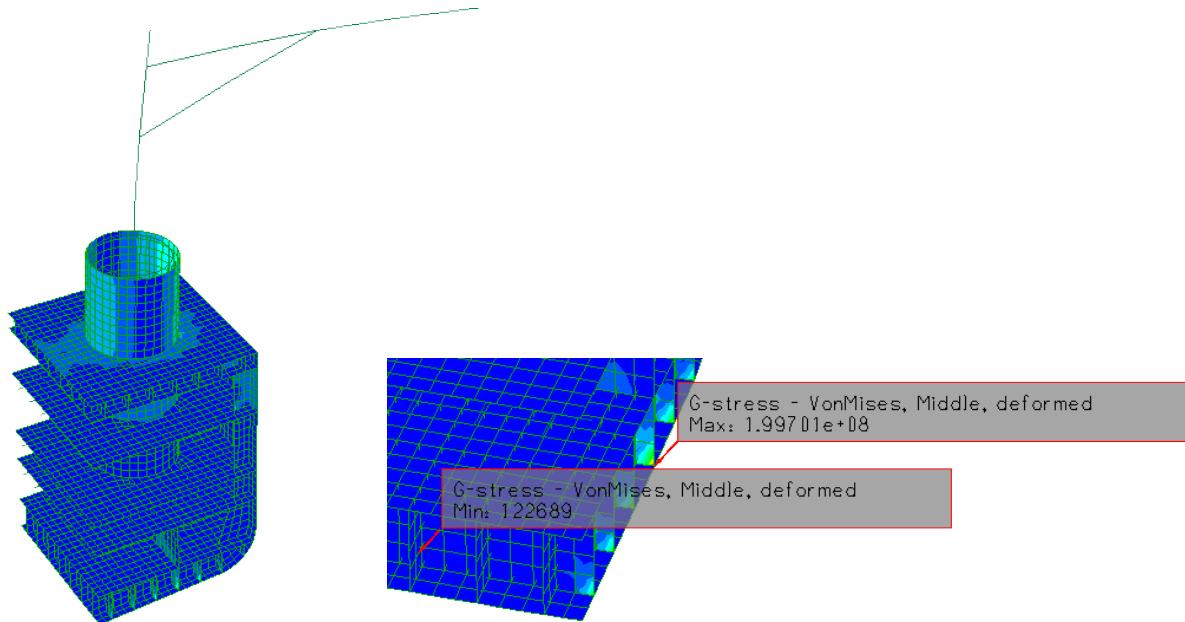


## 17 CREATE SETS AND RUN ANALYSIS WITH EVEN FINER MESH

- Create a few sets. A couple of examples are shown below. The sets are found in the *Utilities | Sets | Regular Sets* folder.

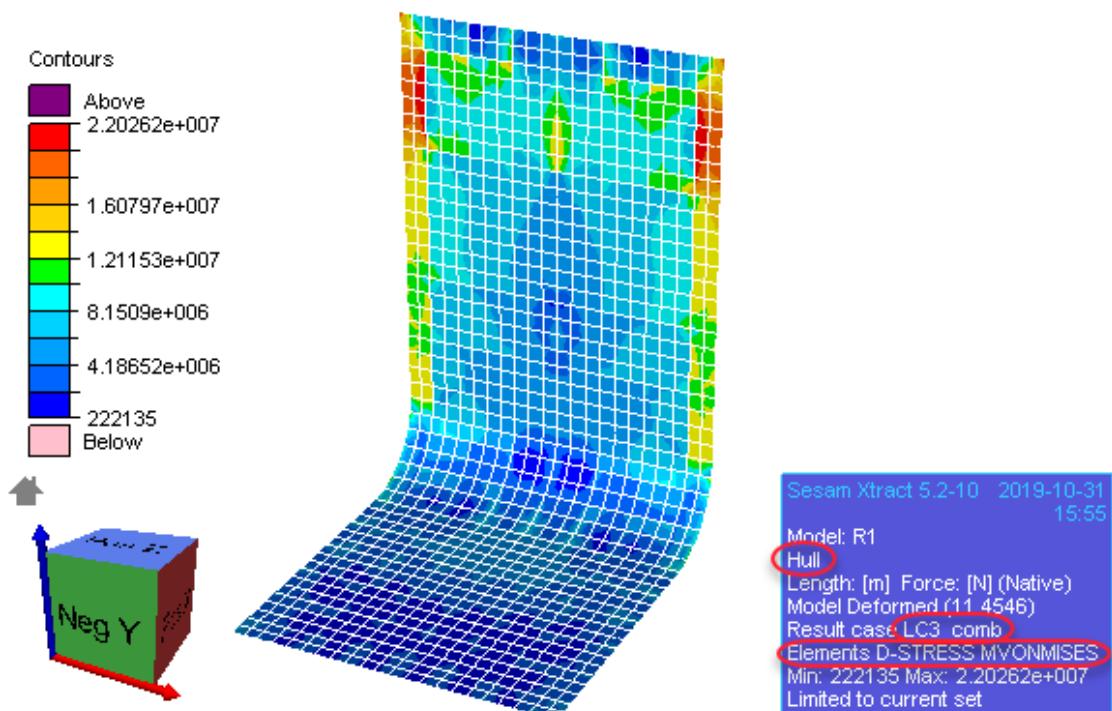


- Change the mesh density from 1 m to 0.5 m and rerun the analysis. The G-stress VonMises in the middle surface is shown below for the load combination. The maximum and minimum values are highlighted for a zoomed in part. (The position of the lowest von Mises stress may vary and is relevant.)



- Start Xtract to investigate the results further by *Results | Advanced Results (Xtract)*.

- In Xtract, *Elements D-STRESS MVONMISES* corresponds to the stress presented by GeniE above. The sets defined in GeniE are available in Xtract. The set named Hull has been selected below.



## 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.