SESAM TUTORIAL

GeniE

Curved Structure Modelling – Semisubmersible Pontoons

Valid from program version 8.2

Sesam Tutorial

GeniE – Curved Structure Modelling – Semisubmersible Pontoons

Date: July 2021

Valid from GeniE version 8.2

Prepared by: Digital Solutions at DNV

E-mail support: software.support@dnv.com

E-mail sales: digital@dvn.com

TABLE OF CONTENTS

1 INTRODUCTION

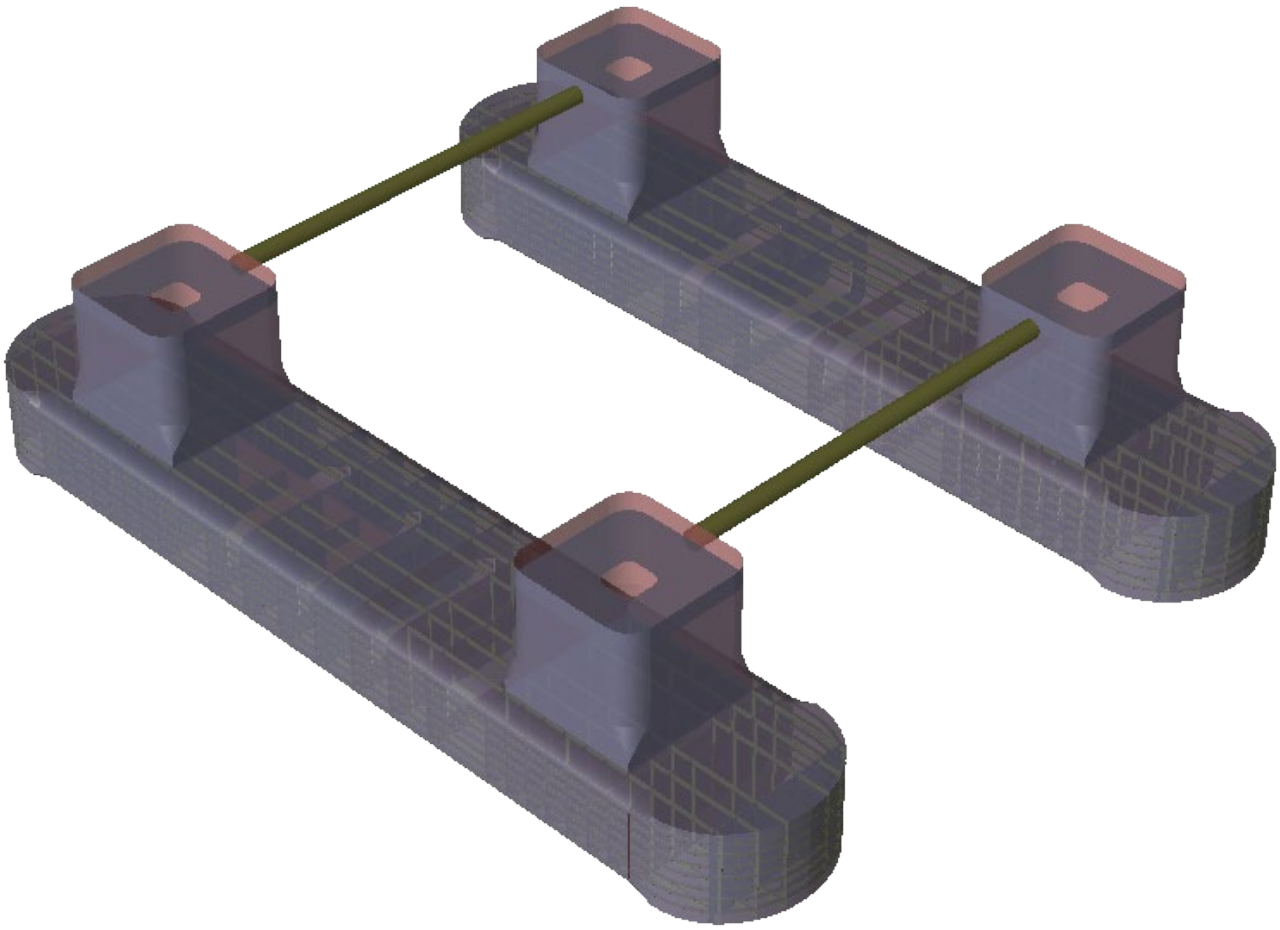
- In this tutorial a model of semisubmersible pontoons with columns is created for hydrostatic, hydrodynamic and stress analysis. The deck is not included.
  - How to do these analyses is, however, not covered in this tutorial.

- The following modelling topics are covered:
  - Flat and curved plates
  - Straight and curved beams
  - Basic FE mesh control

- The tutorial presuppose some basic knowledge in GeniE's GUI.

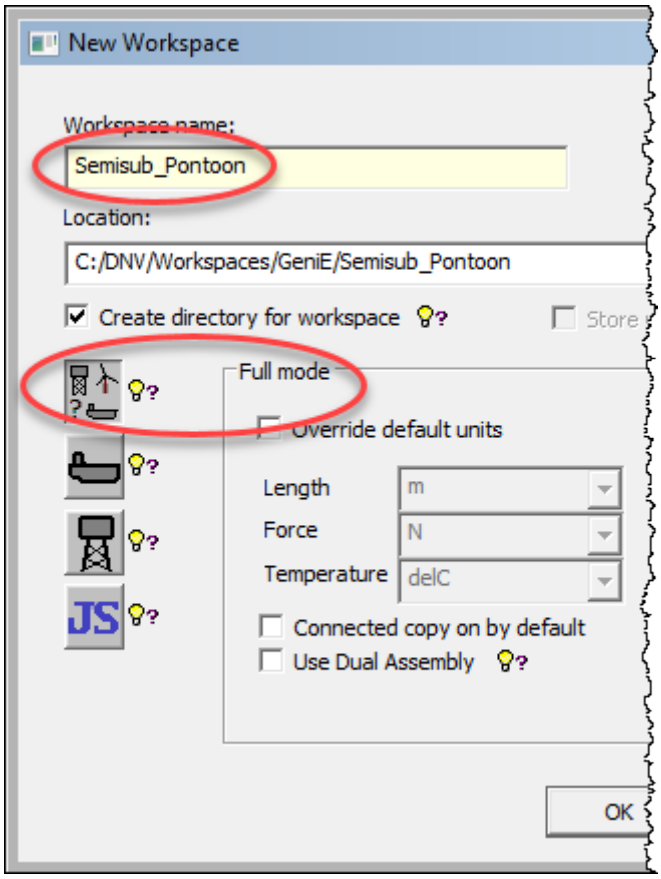
- A GeniE input file for creating the model is provided.

- The appearance of the GUI and dialogs in later versions of GeniE may change.

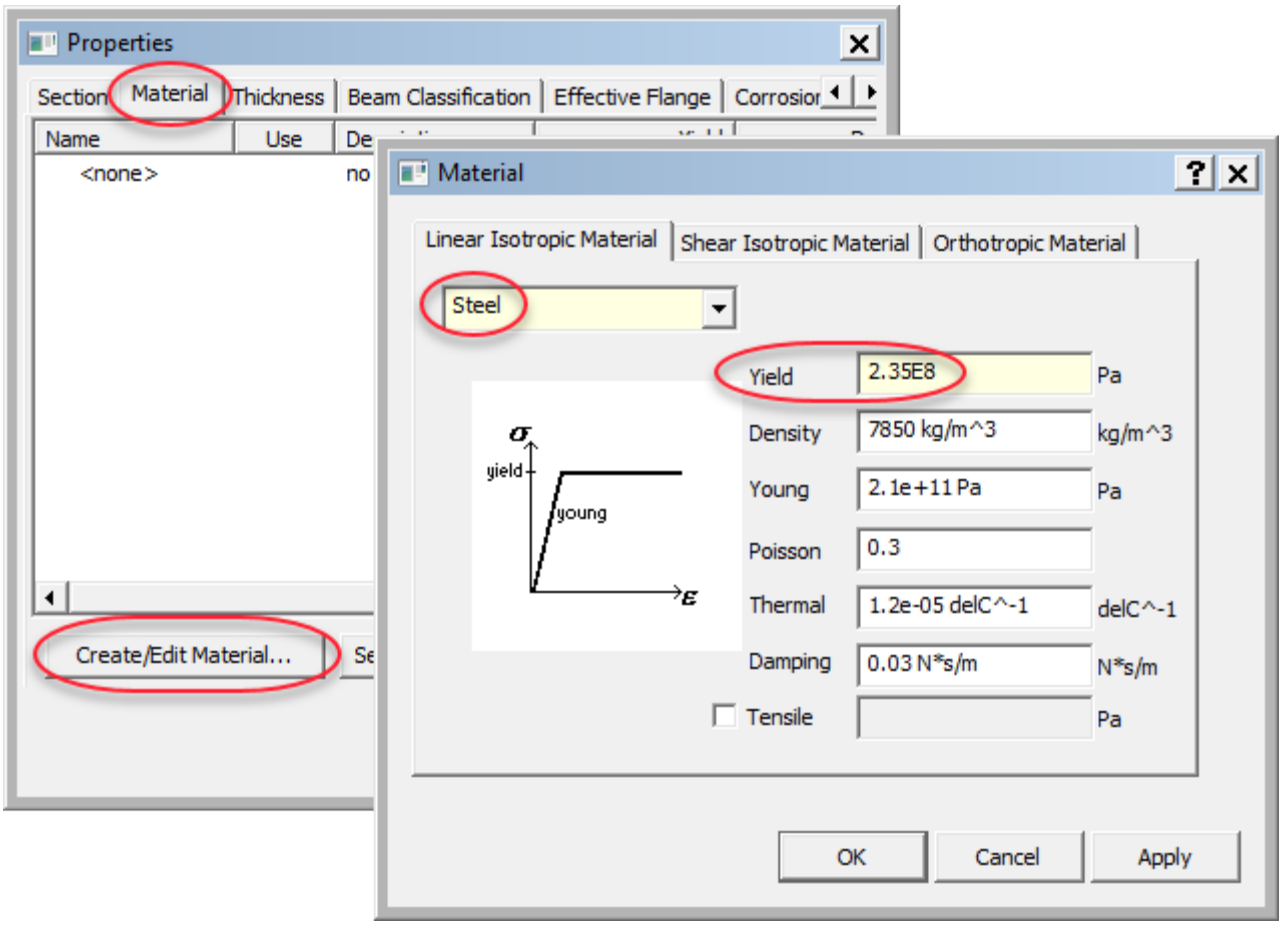


2 UNITS, MATERIAL, CROSS SECTIONS, PLATE THICKNESSES

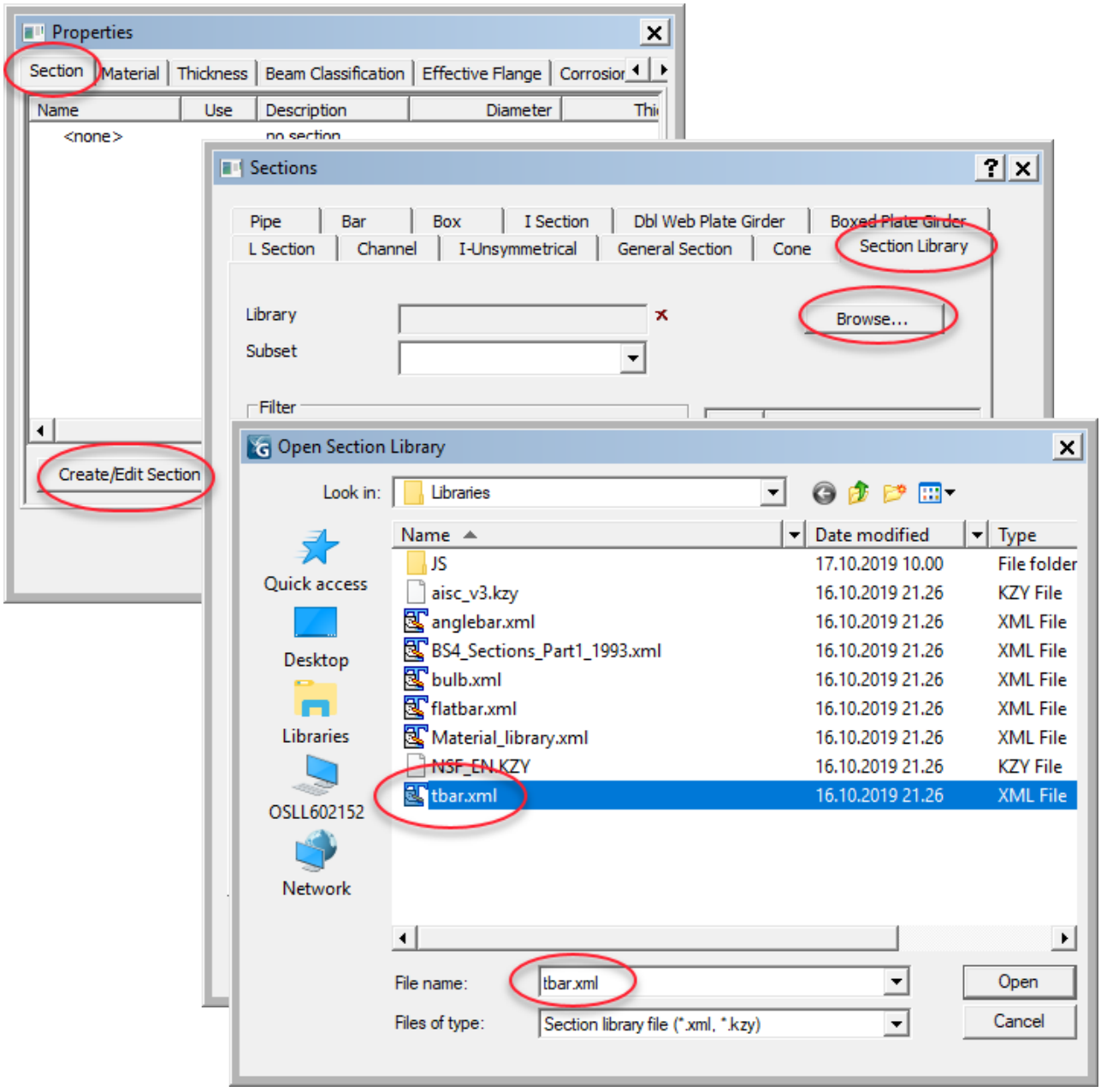
- Start GeniE and open a new workspace.
  - Give a *Workspace name*.
  - Accept default *Output Units* m and N and click *OK*.
    - Unless otherwise specified, all values in this tutorial are in these units.
  - Use *Full mode* as curved geometry shall be modelled.



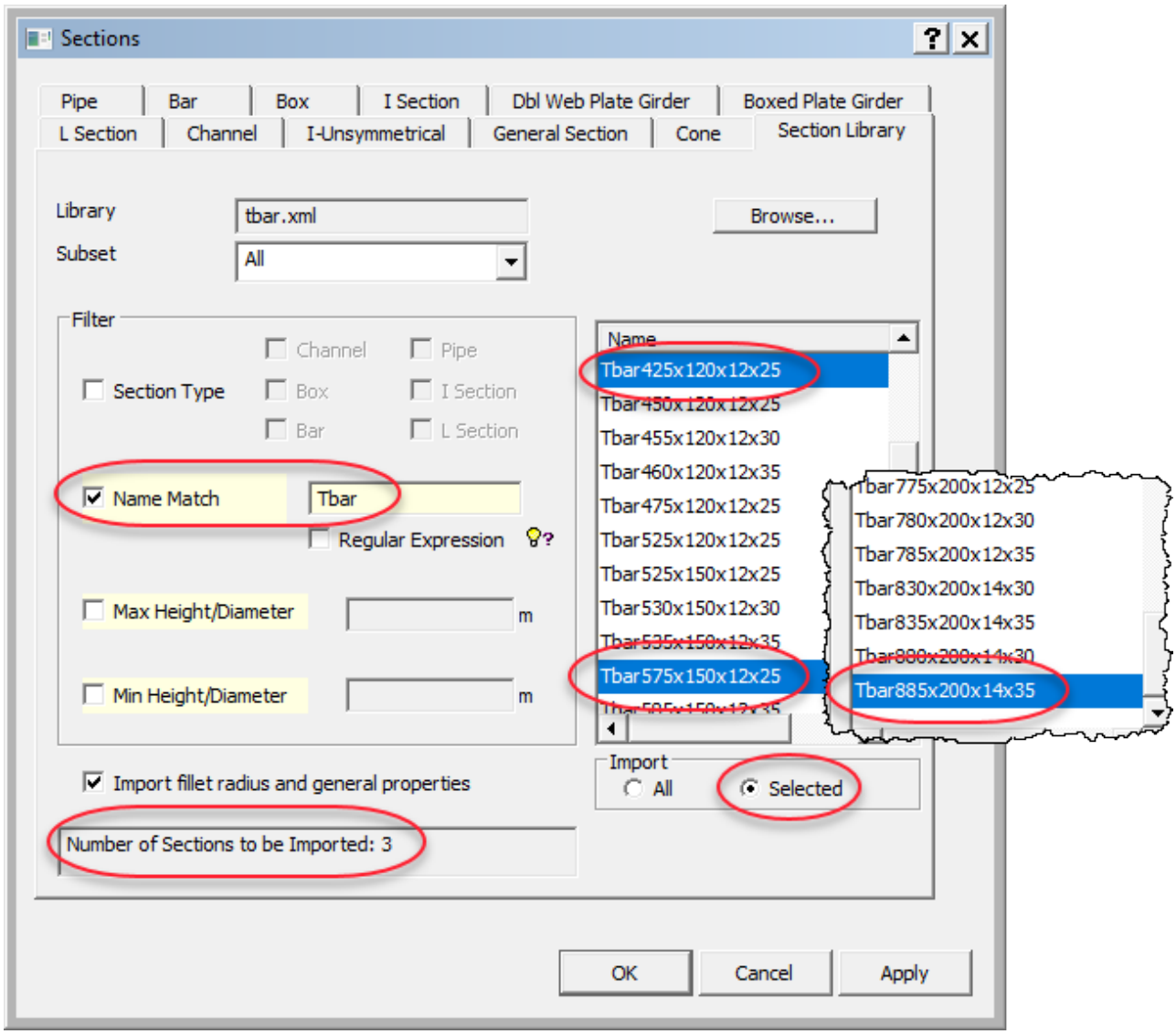
- Define steel material.
  - Use *Edit* | *Properties* to open the *Properties* dialog.
  - In the *Material* tab click *Create/Edit Material*.
  - In the *Material* dialog give a material name and a *Yield* value. Accept default values and click *OK*.



- 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.

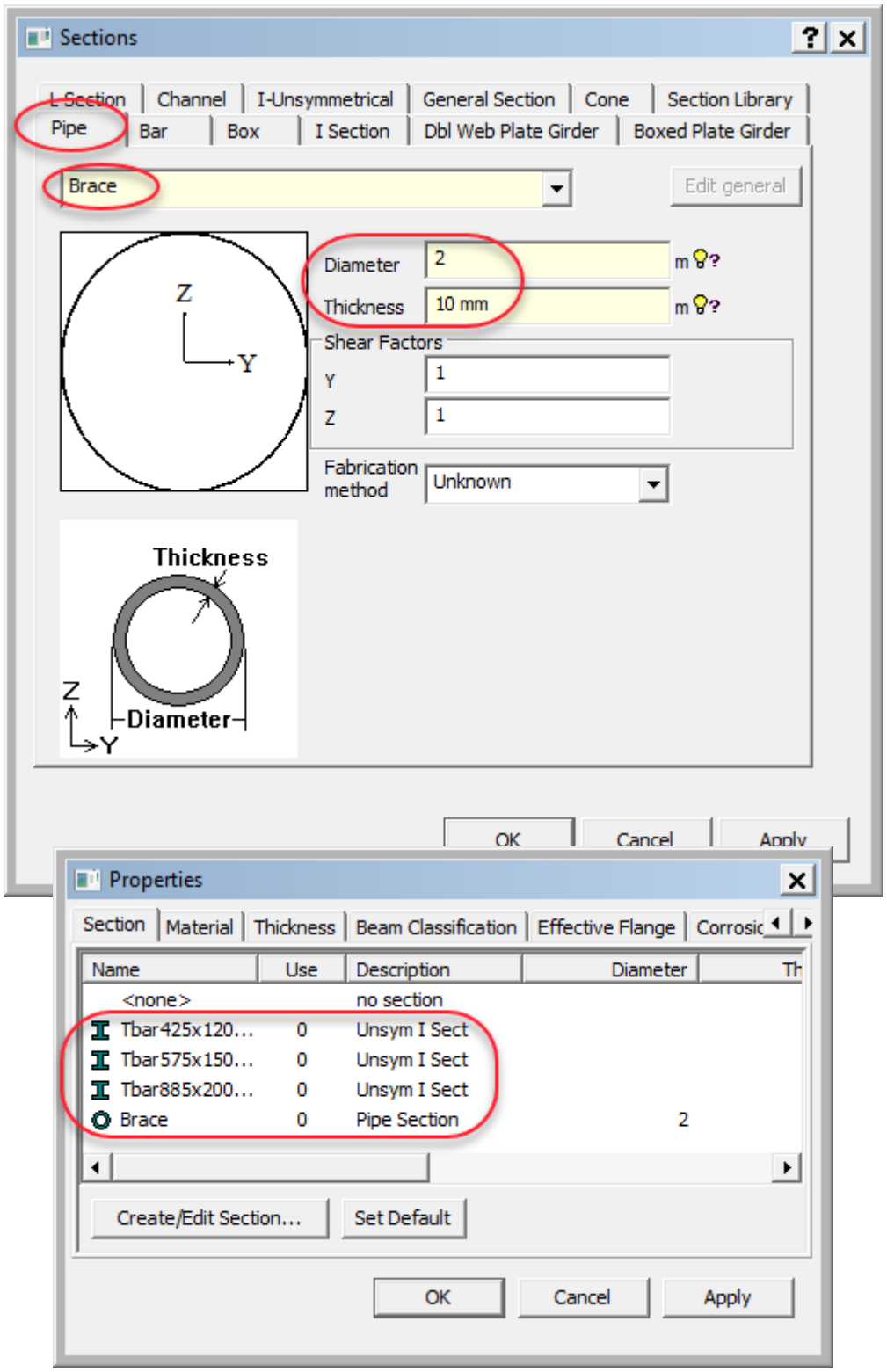


- We want to import the following Tbar sections into our workspace:
  - Tbar425x120x12x25
  - Tbar575x150x12x25
  - Tbar885x200x14x35
- To ease the selection check *Name Match* and enter Tbar and see that only sections with name containing the string Tbar are listed.
- Having made the selection make 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.



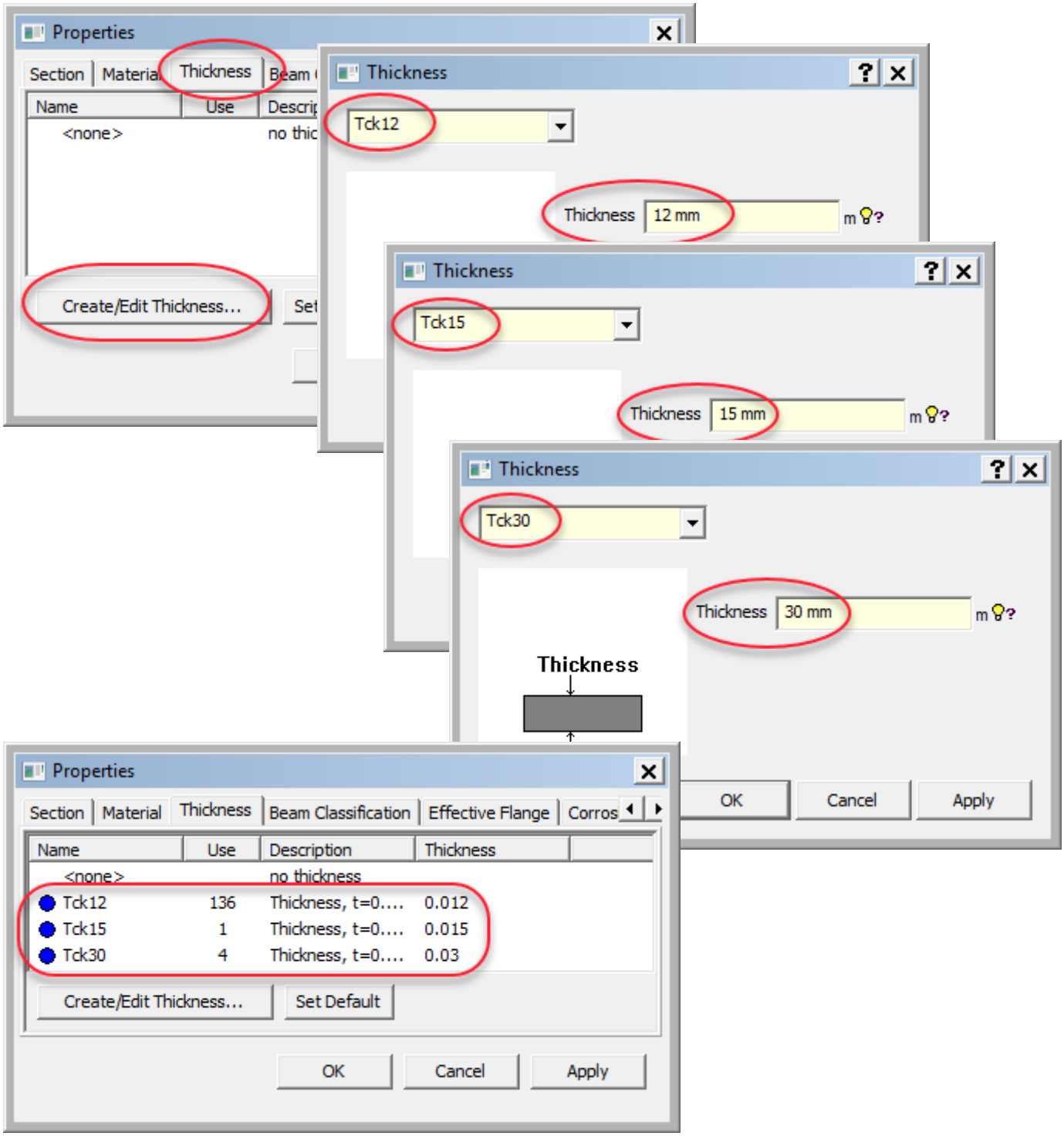
- Again, click *Create/Edit Section* in the *Section* tab of the *Properties* dialog to open the *Sections* dialog. Now create a beam cross section of type *Pipe* named Brace with data as given below.

- See all imported and manually created sections listed in the *Section* tab of the *Properties* dialog as shown below.



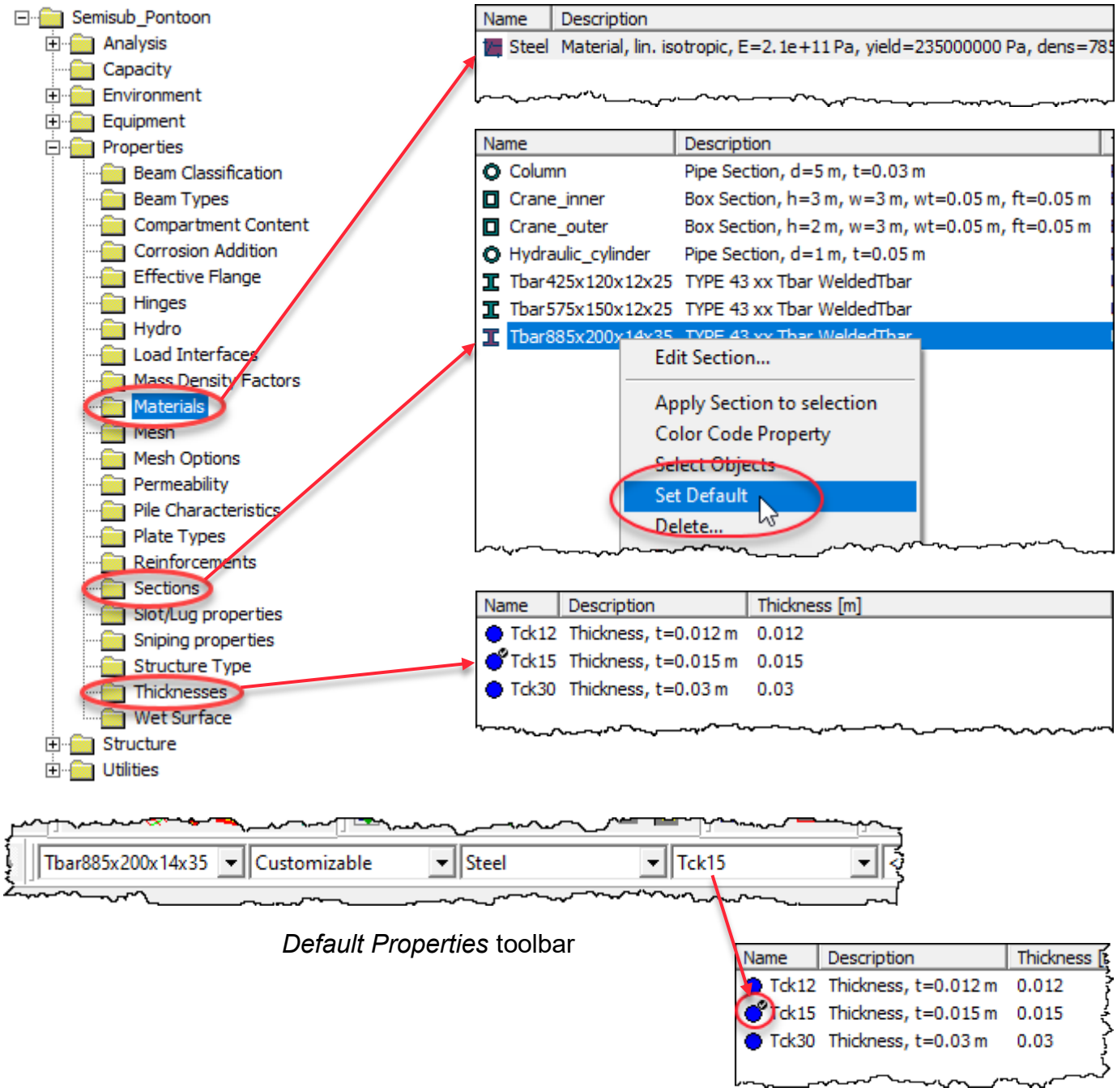
➢ Define three plate thicknesses Tck12, Tck15 and Tck30.

- 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 be accessed from there.

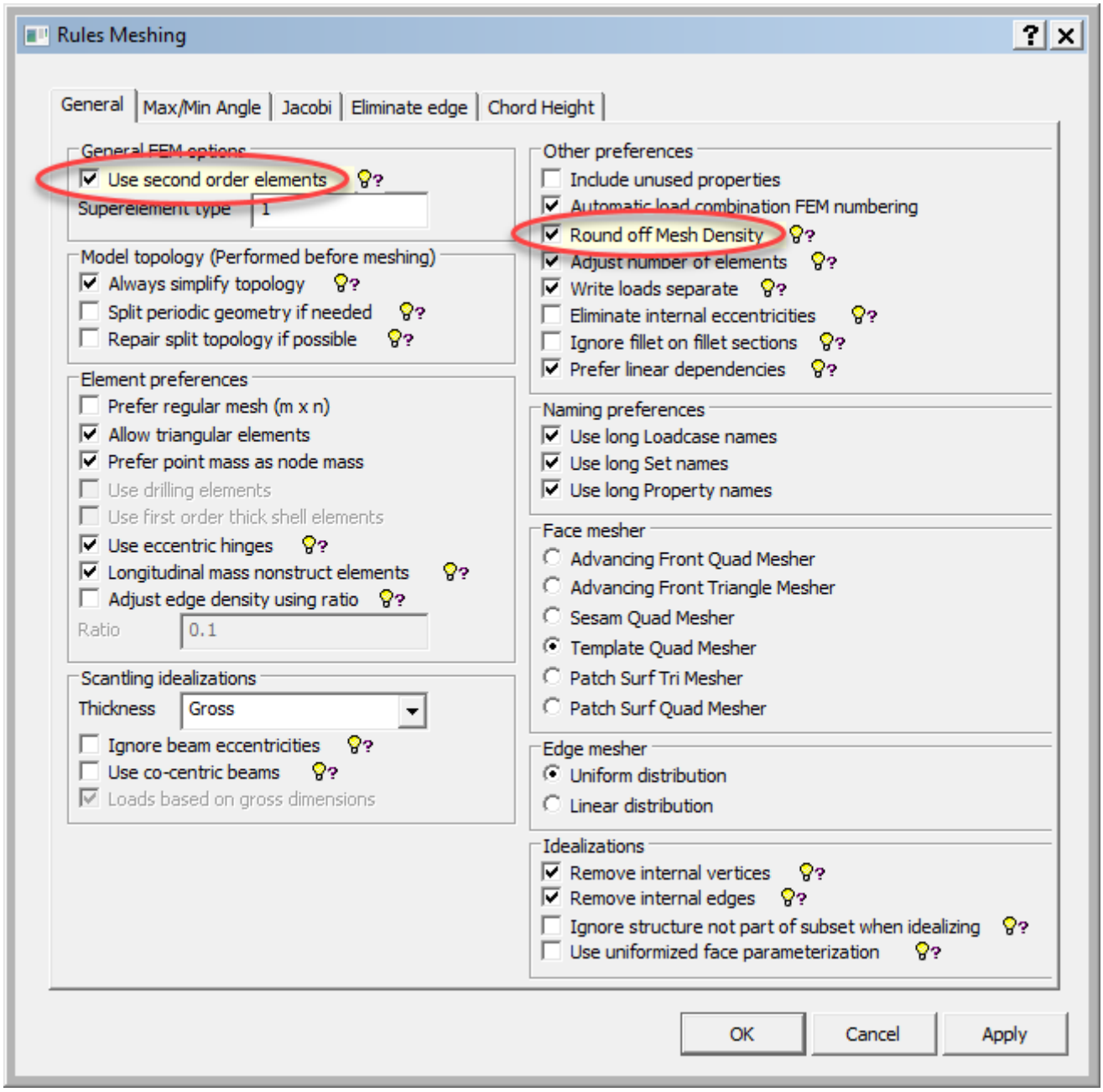
- Now set default material, section and thickness. I.e. properties assigned to objects at their creation. This can be done by right-clicking as shown below for Tbar885x200x14x35. This can also be done through the *Default Properties* toolbar as shown further below.
  - Set material Steel, section Tbar885x200x14x35 and thickness Tck15 as default.
  - Notice the small check mark in the browser for Tck15 indicating this is default.



Default Properties toolbar

➢ Use *Edit* | *Rules* | *Meshing Rules* to adjust FE mesh settings.

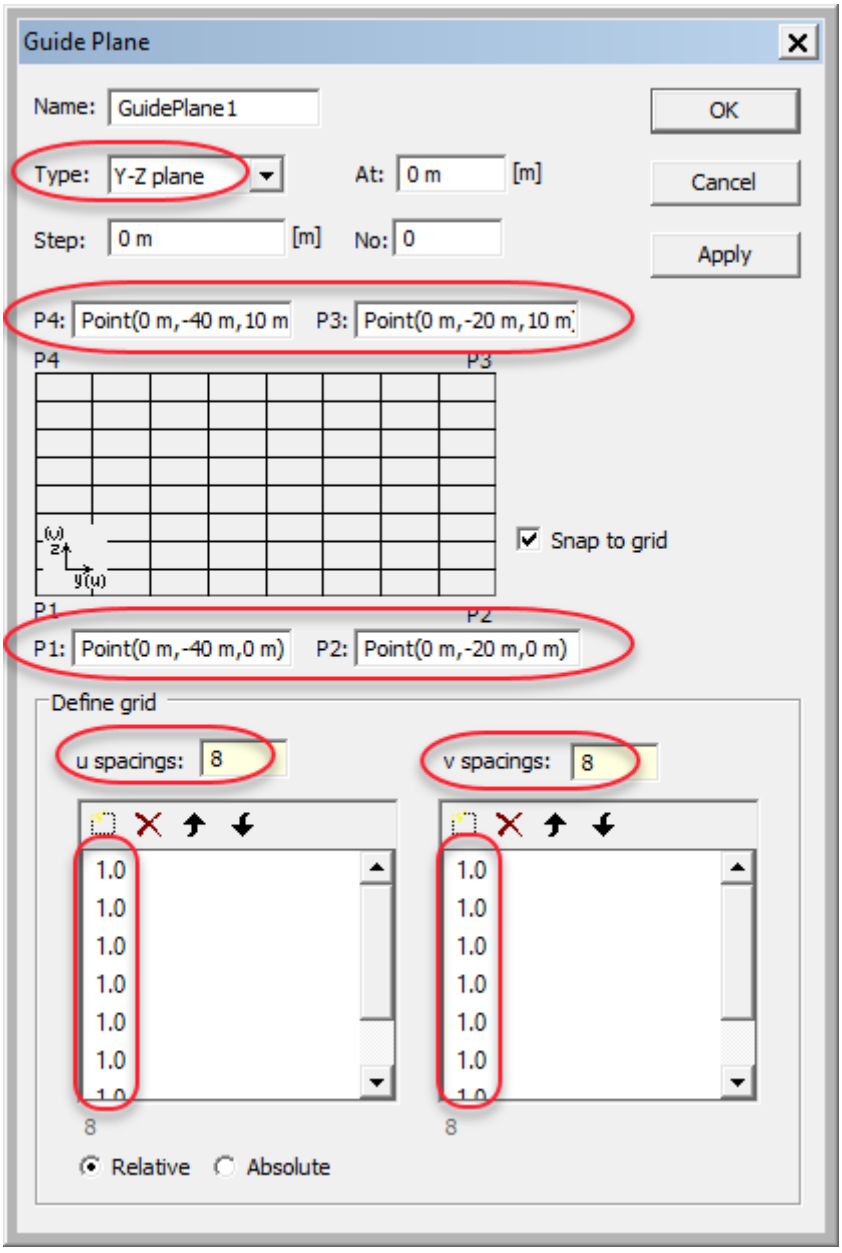
- Check *Use second order elements*, i.e. select 8-node curved quadrilateral and 6-node curved triangular elements rather than 4-node flat quadrilateral and 3-node triangular elements.
- Check *Round off Mesh Density* to allow some flexibility in the FE mesh generation, this allows somewhat larger elements than otherwise specified.
- Accept other default settings.



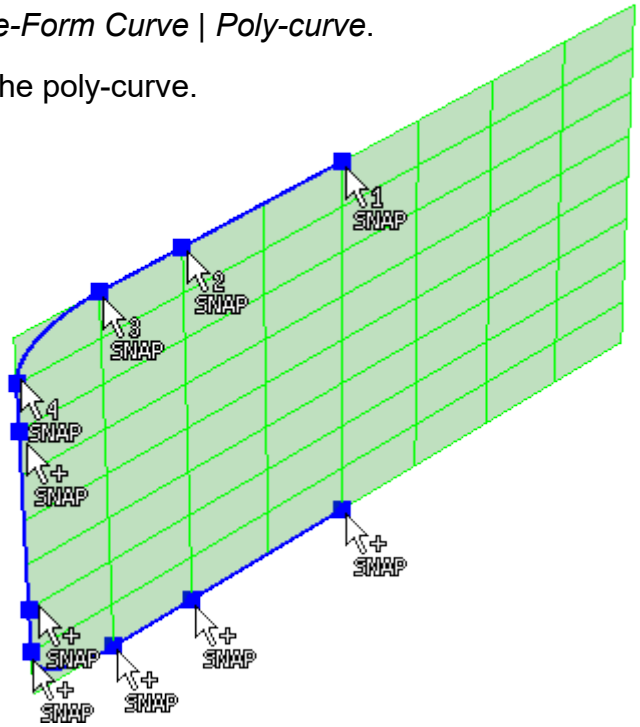
3 CREATE OUTER HULL PRISMATIC PART

- Use *Guiding Geometry* | *Planes* | *Guide Plane Dialog* to create a guide plane.
  - Change to *Type: Y-Z plane* and give coordinates as specified for the corner points *P1-P4*. I.e. modify the Y-coordinate values only.
  - Give value 8 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.

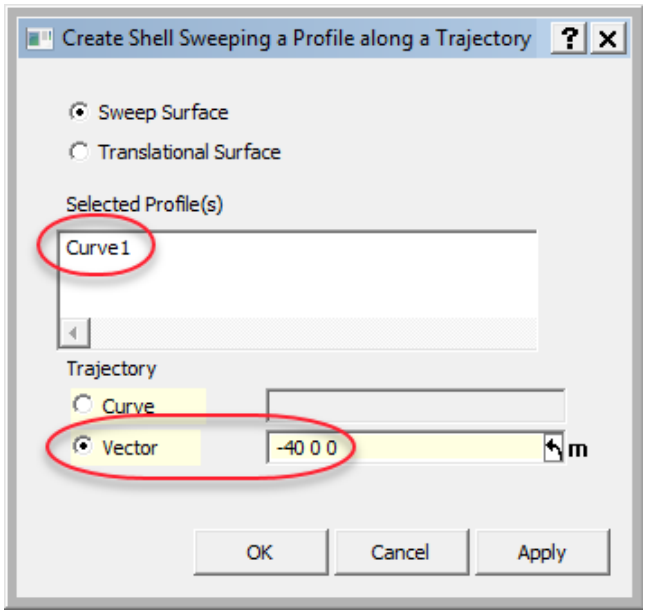
- Provided the *Default display* configuration is chosen the guide plane will be displayed. Press the *Iso view* button, , or F5, to fit the guide plane in the display.

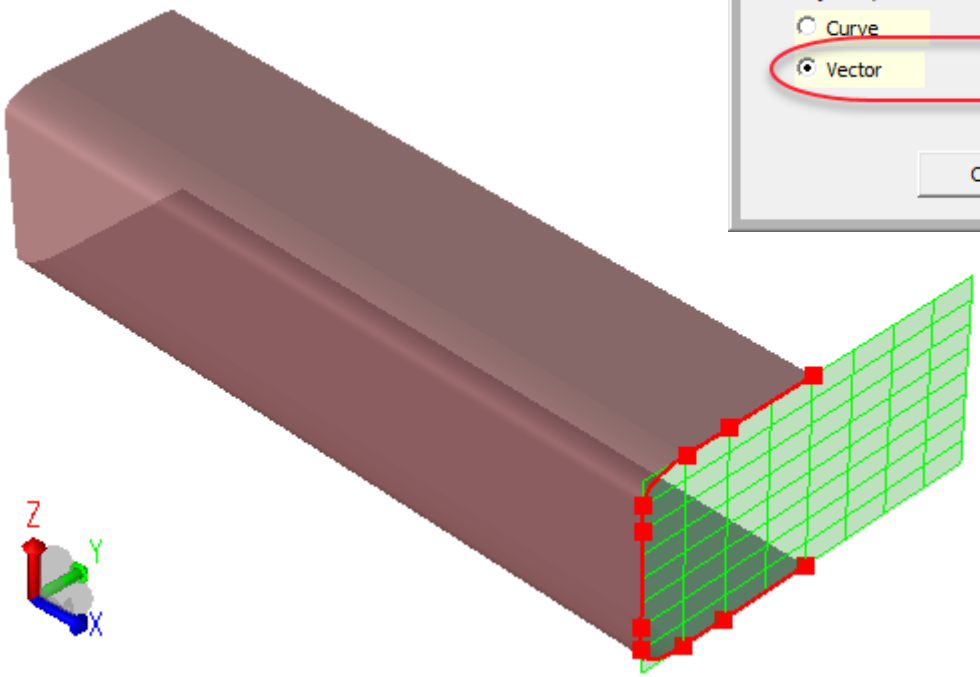


- Create a poly-curve by *Guiding Geometry* | *Free-Form Curve* | *Poly-curve*.
  - The intermediate points define the shape of the poly-curve.
  - Double-click the end point.

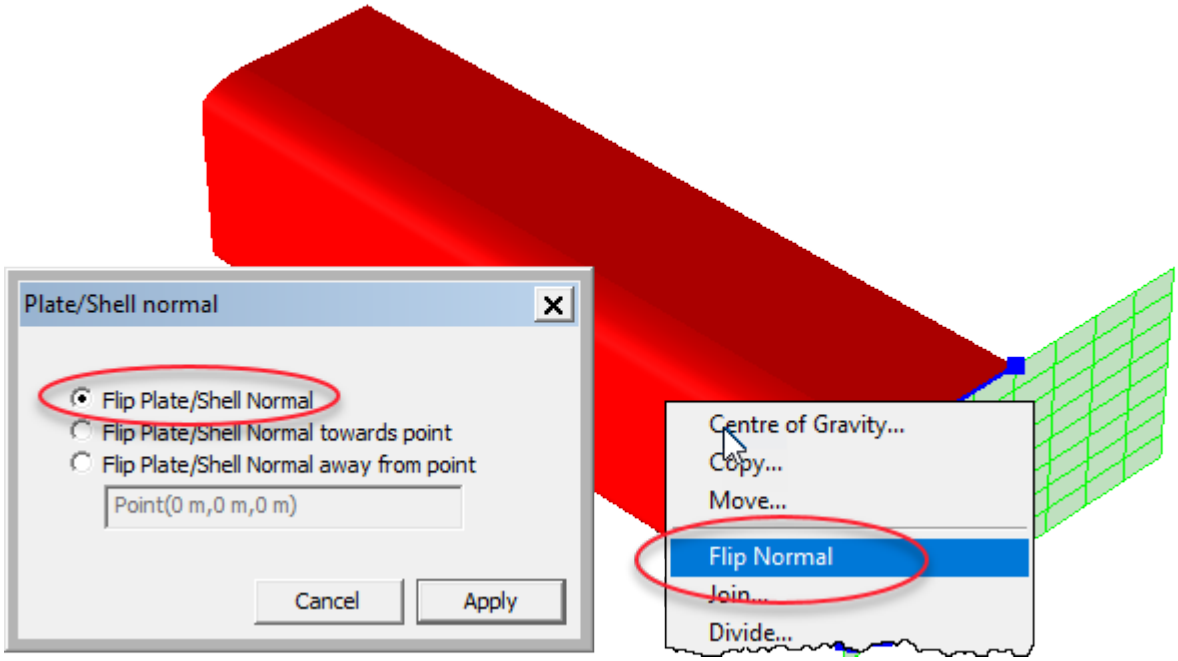


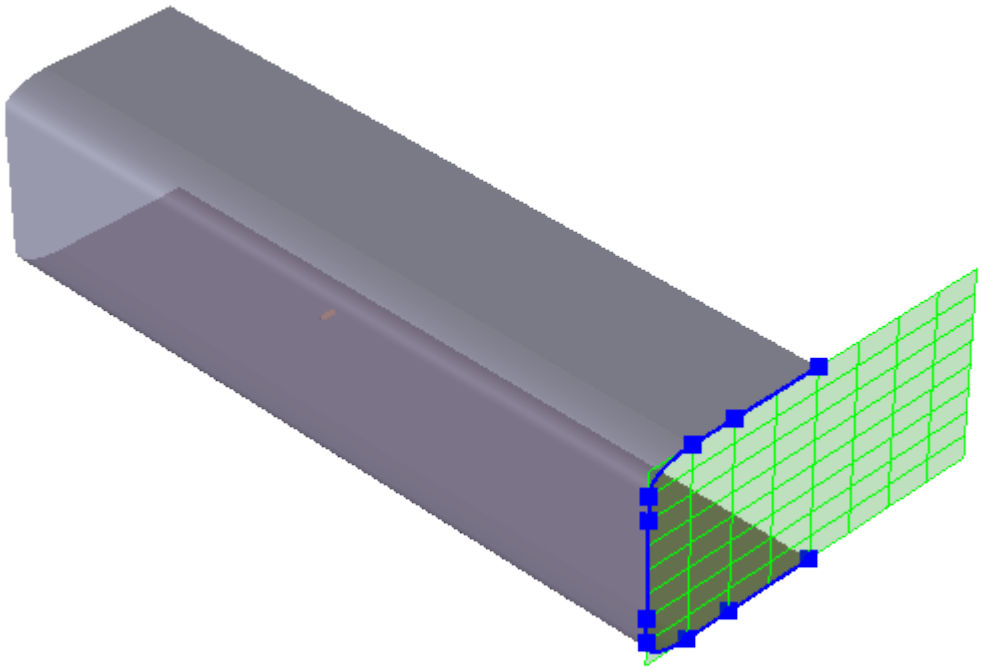
- Use *Structure* | *Free Form Shells* | *Sweep Curves Dialog* to create the outer hull of the pontoon by extruding the poly-curve a vector of (−40,0,0).
  - Click the guide curve before or after opening the dialog.
  - Type in the vector and click *OK*.





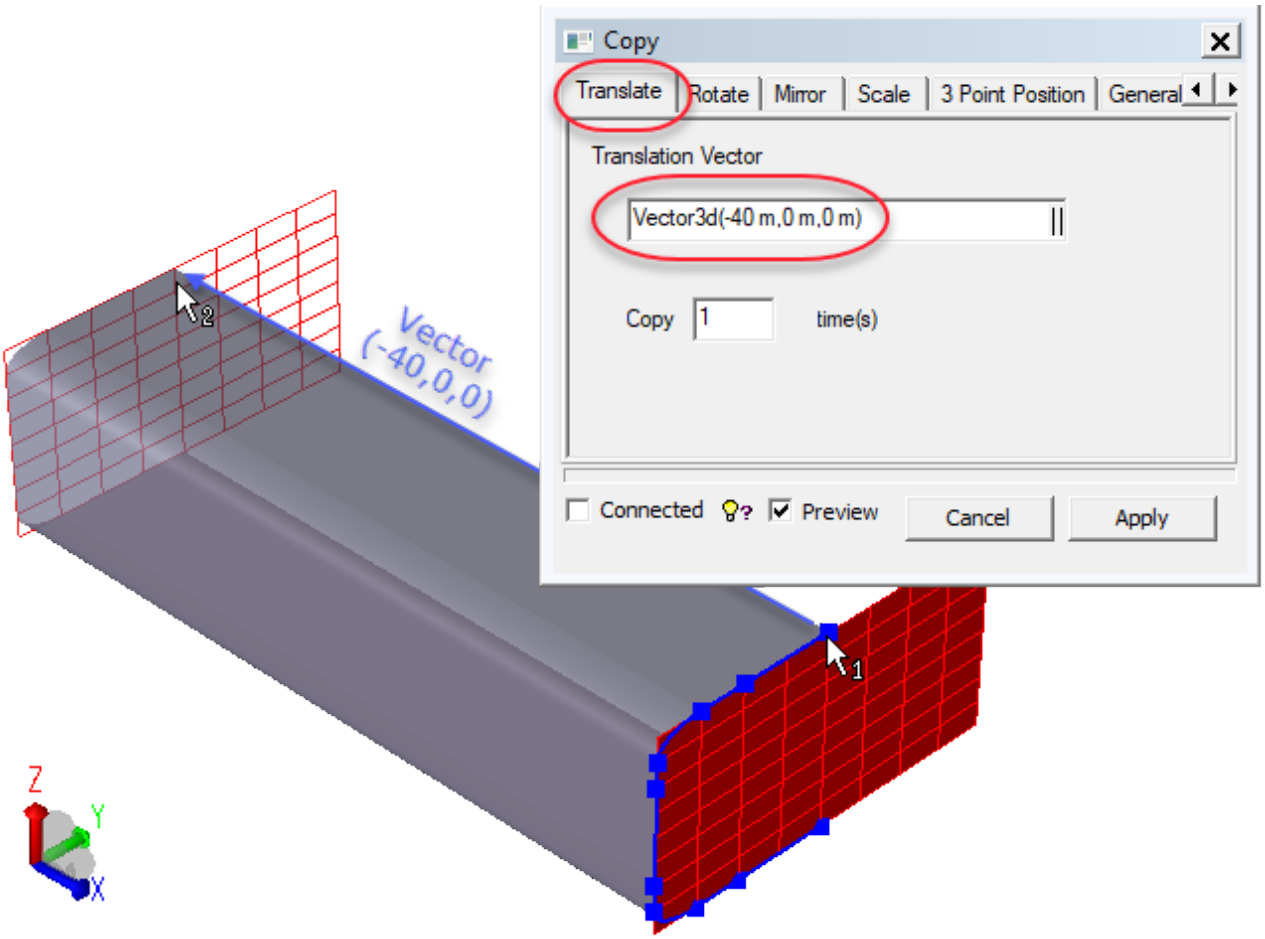
- Depending on the direction of clicking when creating the guide curve, the extruded surface may be reddish meaning that the surface normal points inwards. A normal requirement for a surface for which hydrodynamic loads shall be computed is that the surface normal should points outwards, i.e. its colour should be bluish.
  - Select the surface, right-click and click *Flip Normal* to switch the surface normal.



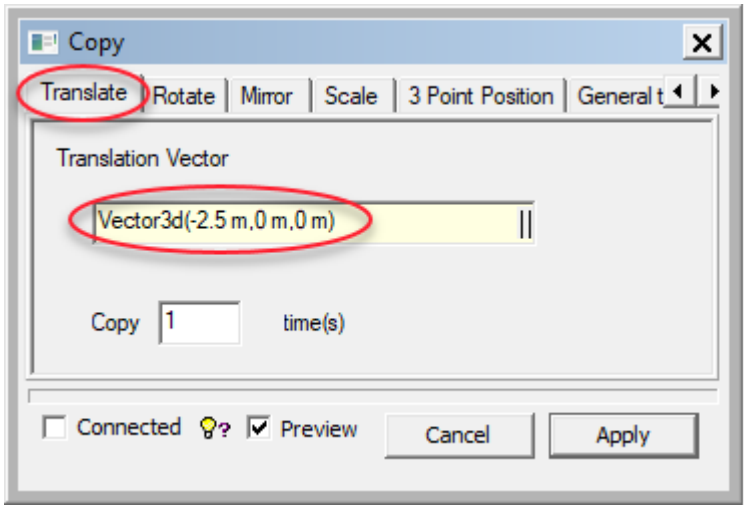


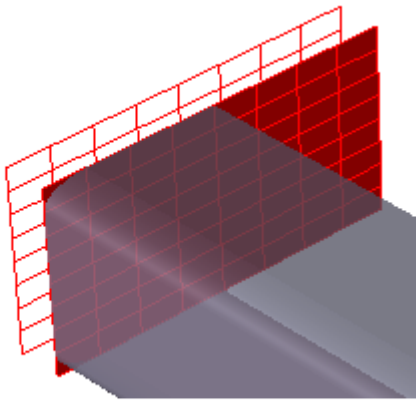
4 CREATE OUTER HULL TRANSITION PART

- Copy the guide plane to the other end of the hull already created.
  - Fetch the vector from the display by clicking twice as shown.

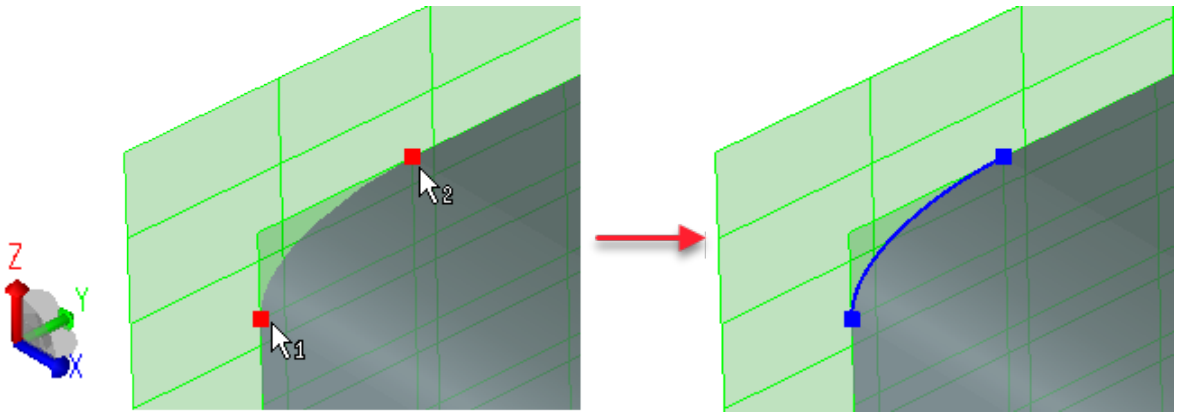


- Copy the new guide plane 2.5 m in negative X-direction.

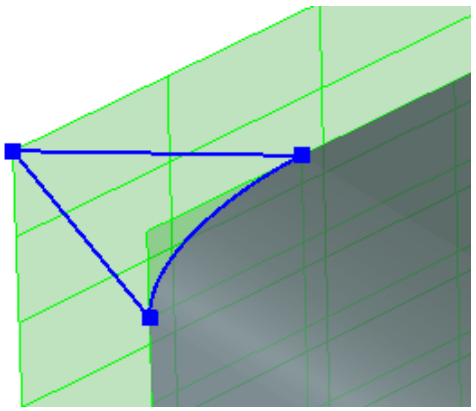




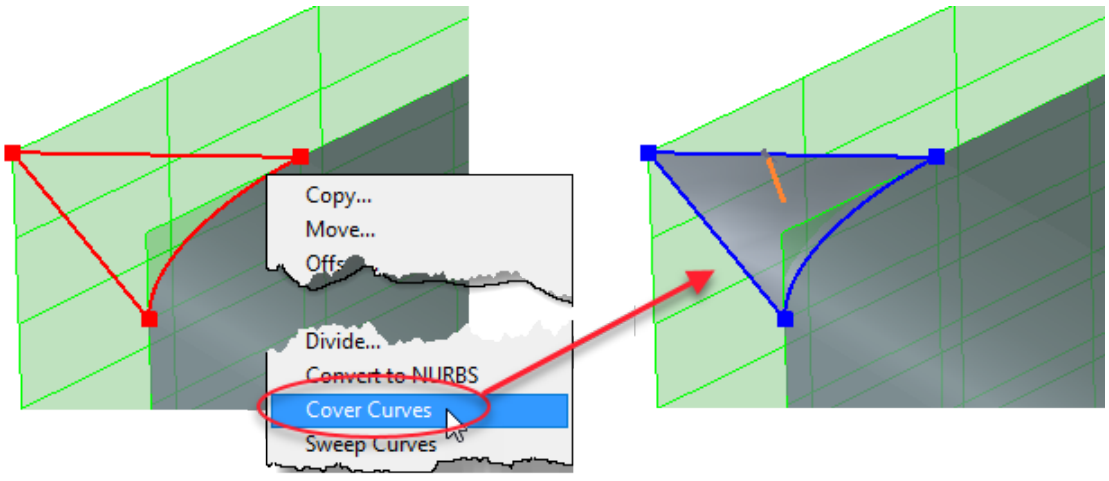
- Create a model curve and guide lines for creating a surface transition between the rounded and rectangular part of the hull.
  - Use *Guiding Geometry* | *Curves on Surfaces* | *Model Curve* and click twice as shown. Make sure you click the surface and not the guide plane.



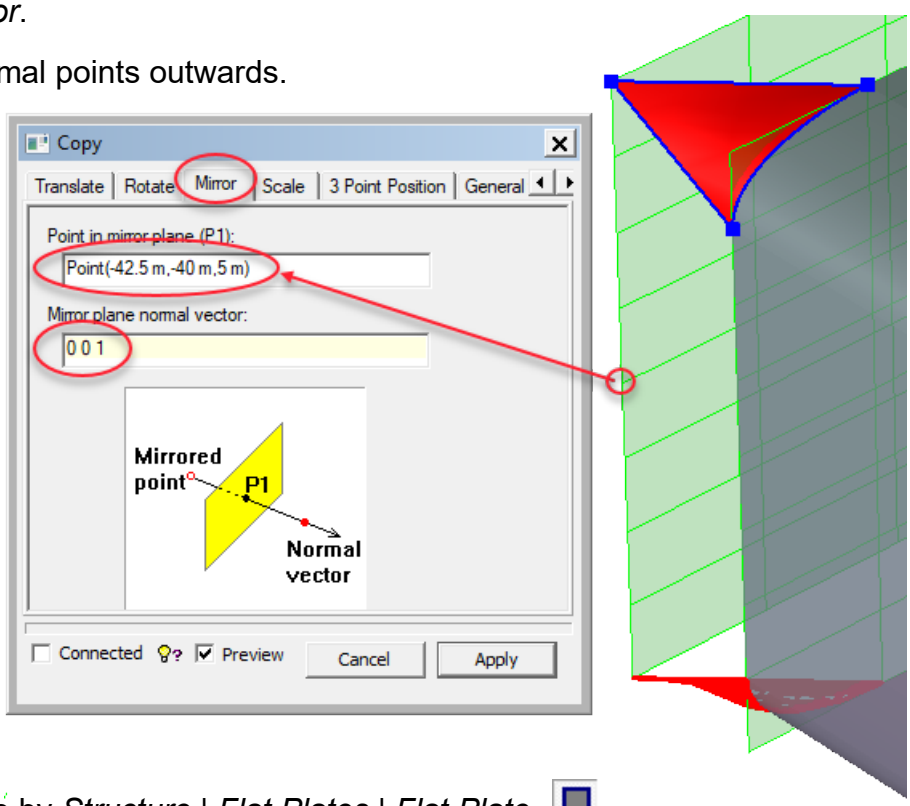
  - Use *Guiding Geometry* | *Lines* | *From Two Points,* , to connect the model curve with the guide plane corner by two straight guide lines as shown to the right.



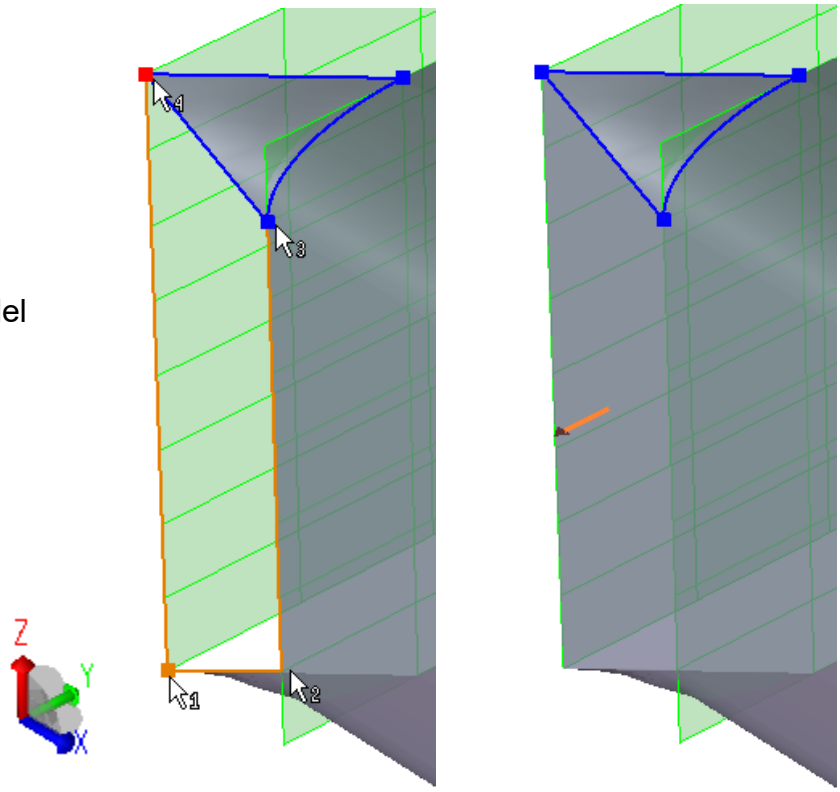
- Create the surface transition by selecting all three guide lines/curves, right-clicking and *Cover Curves*. Label the surface normal (*Labels* | *Local Coordinate System)* to confirm that the surface normal points outwards. If not, flip it.



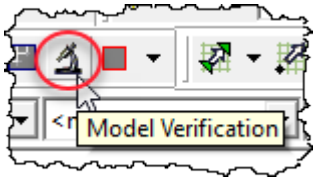
- Copy the transition part from the upper to the lower corner by mirroring. The *Point in mirror plane (P1)* is fetched from the display (only the Z value of this point has relevance). Fill in the *Mirror plane normal vector*.
  - Verify that the surface normal points outwards.



- Create the plane side surface by *Structure* | *Flat Plates* | *Flat Plate,* .
  - Double-click the 4th point to complete the definition.
  - Verify that the surface normal points outwards.

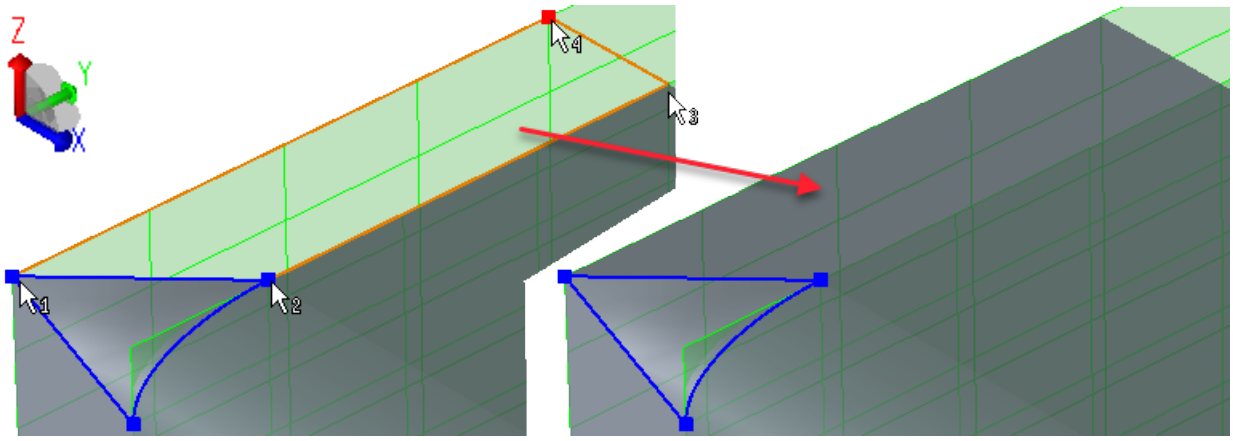


- Check the consistency of the model by clicking the *Model Verification* button and correct any problems detected.



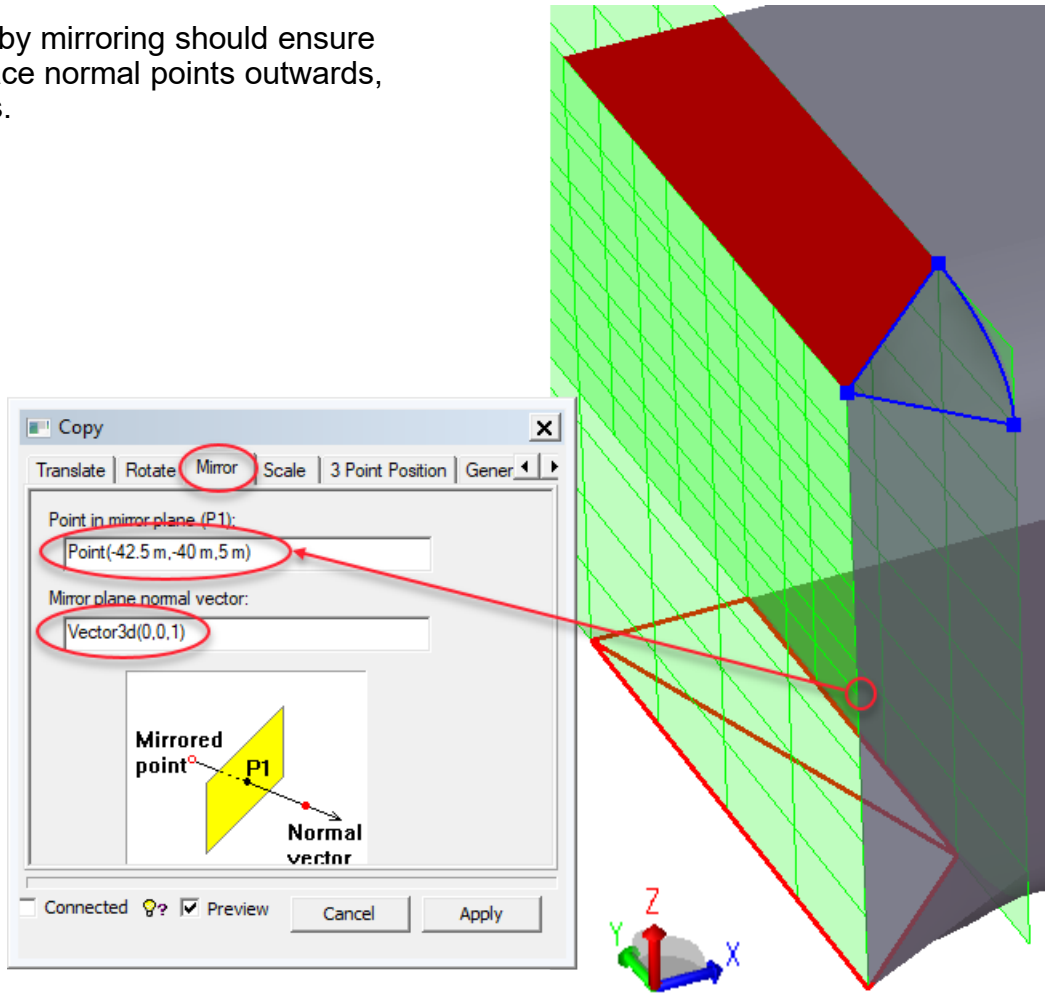
➢ Create the top horizontal plane surface.

- Verify that the surface normal points outwards.



➢ Copy the top horizontal plane surface by mirroring down to form the lower horizontal plane surface.

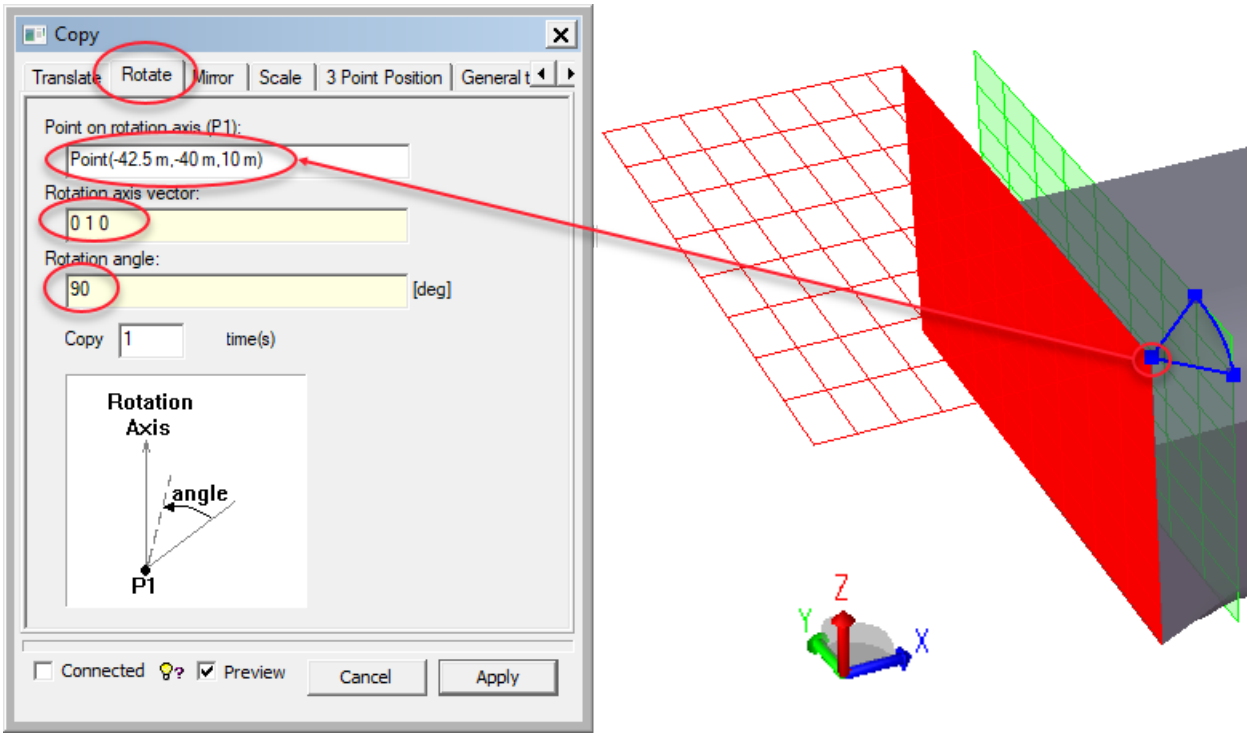
- The copying by mirroring should ensure that the surface normal points outwards, but verify this.



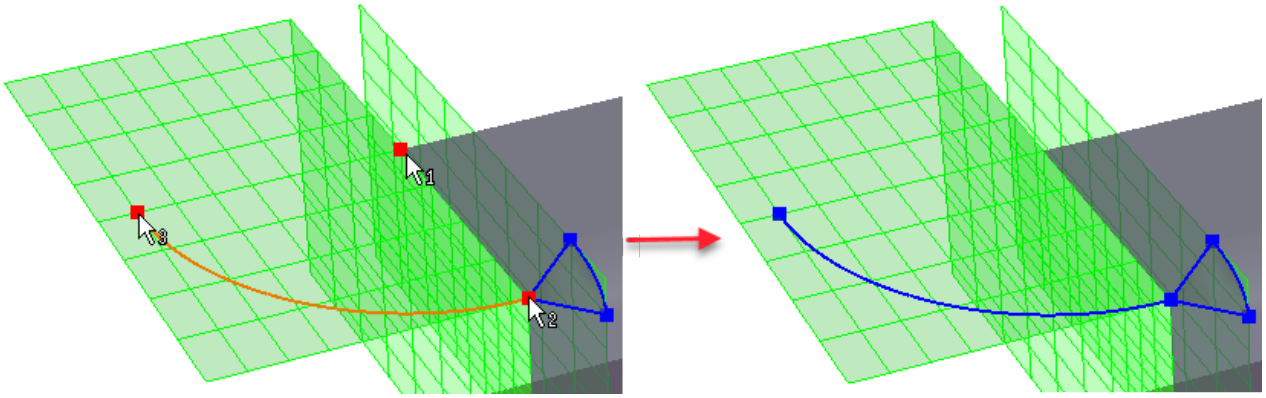
5 CREATE OUTER HULL FORE PART

➢ Copy the guide plane by a rotation as shown.

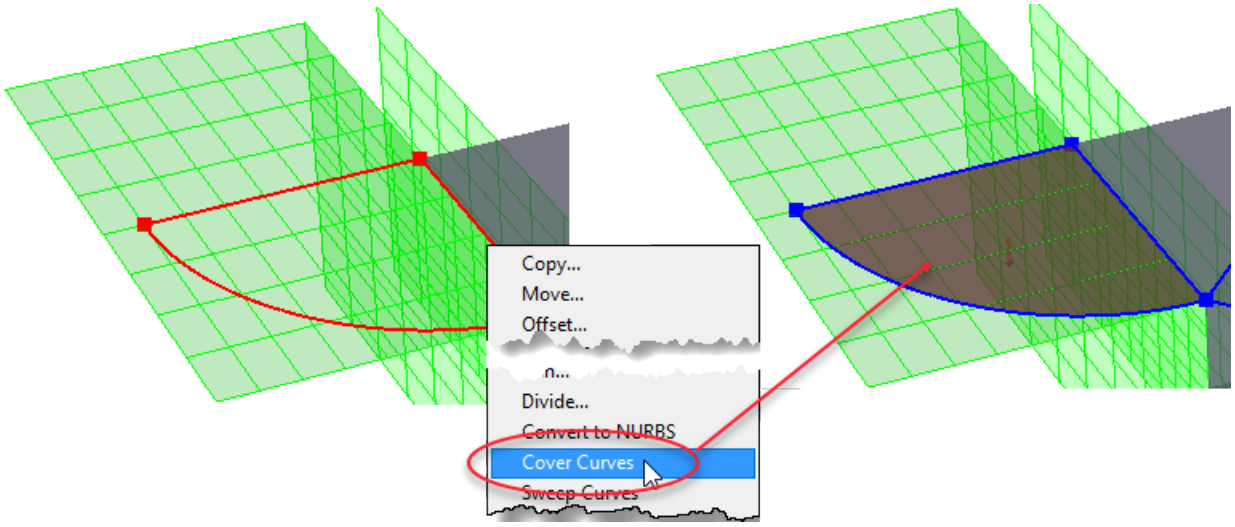
- Fetch the *Point on rotation axis (P1)* from the display. (The Y coordinate is of no consequence.)
- Fill in the *Rotation axis vector* and the *Rotation angle*.



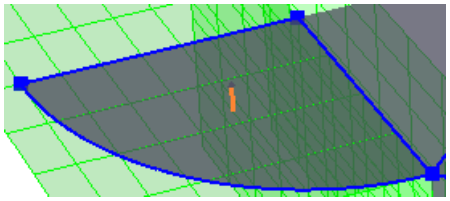
➢ Use *Guiding Geometry* | *Conic Section* | *Elliptic Arc from Center and Two Points*  to create a guide curve, an ellipse, as shown.



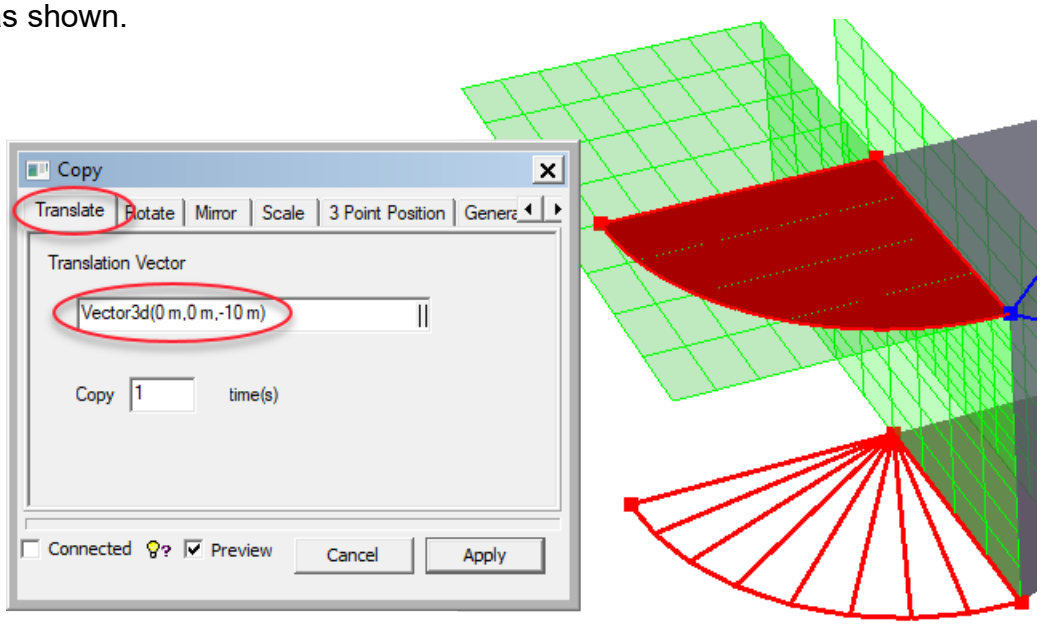
➢ Create two straight guide lines as shown and then create a plane surface by a cover operation.



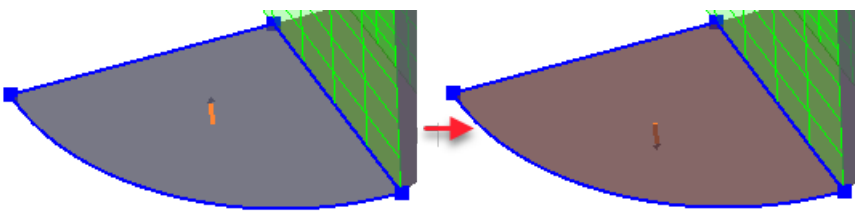
- If the colour is reddish, i.e. the surface normal points downwards, select the surface, right-click and *Flip Normal*.



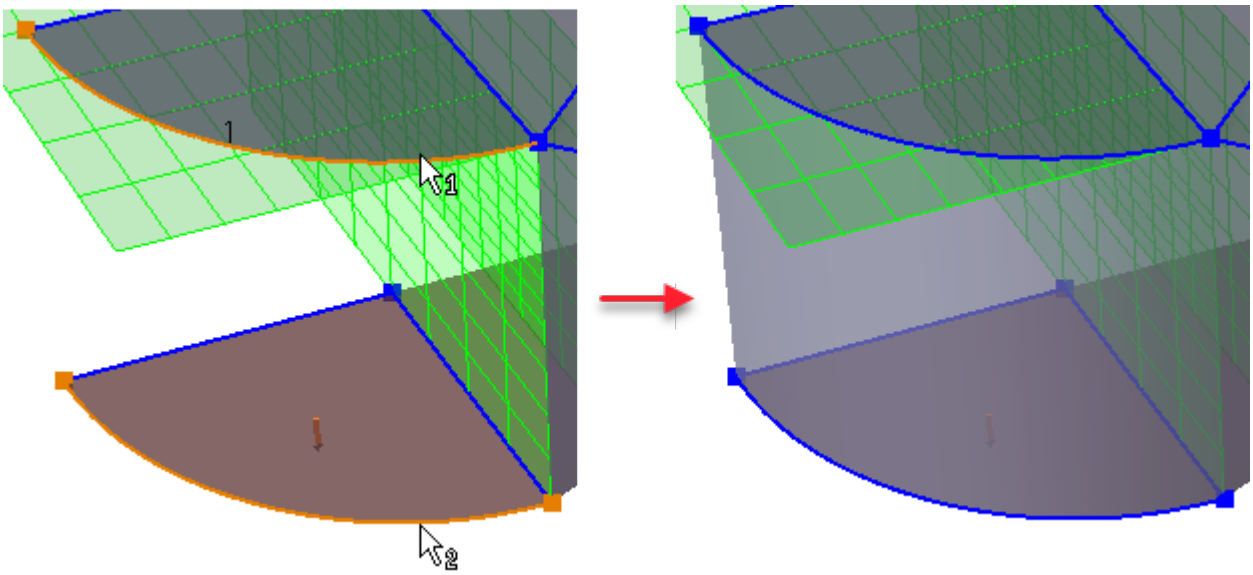
➢ Copy the upper surface including the three bordering guide lines/curves to form the lower surface as shown.



- The normal of the lower surface must be flipped to point downwards.

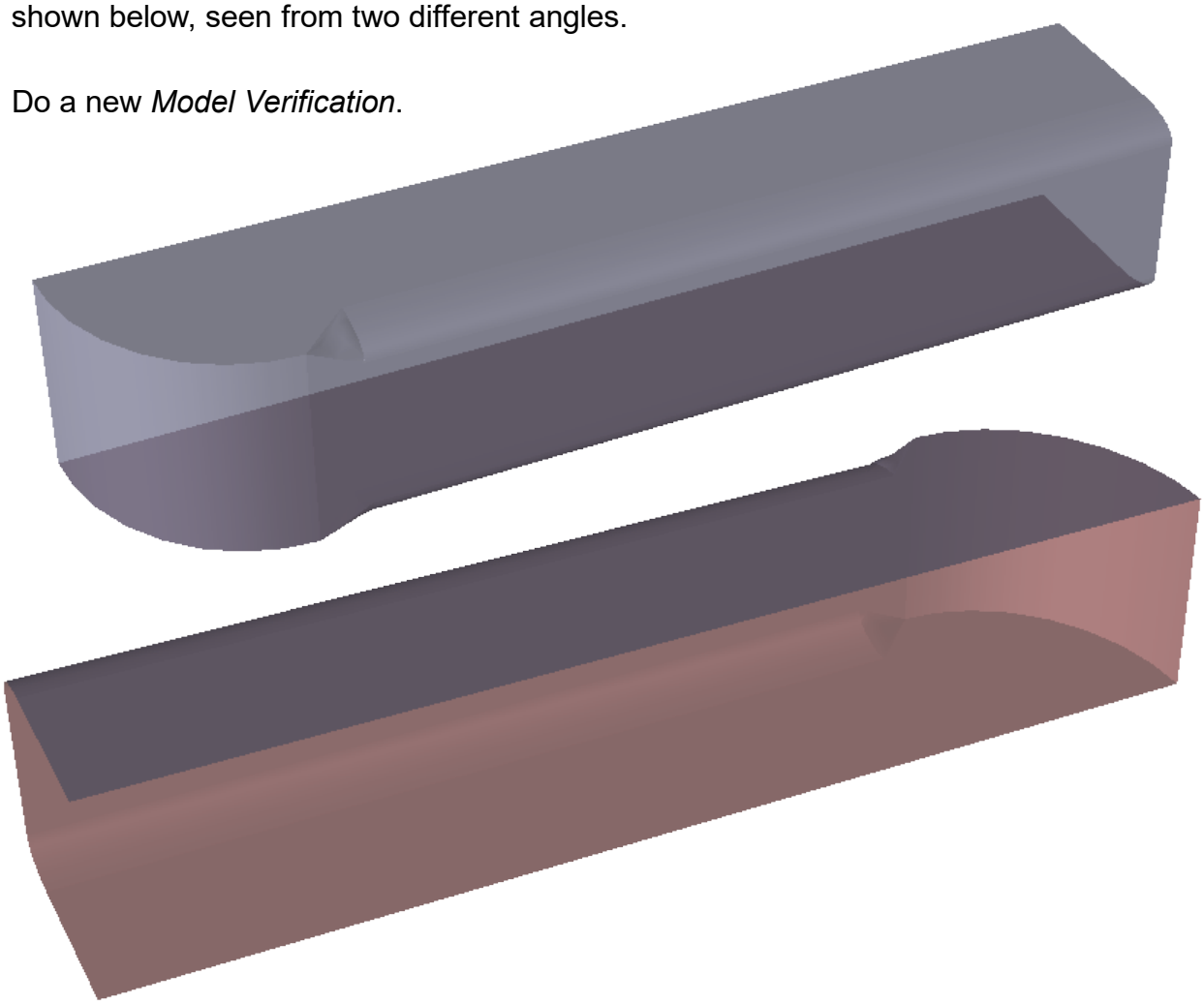


- Use *Structure* | *Free Form Shells* | *Skin/Loft Curves,*  , to create the curved side surface as shown. Double-click the second curve. Flip the surface normal if necessary.



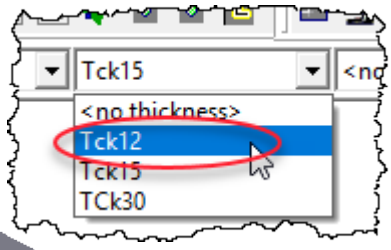
- Switching to *Modelling* - *Transparent* display configuration the model should look like as shown below, seen from two different angles.

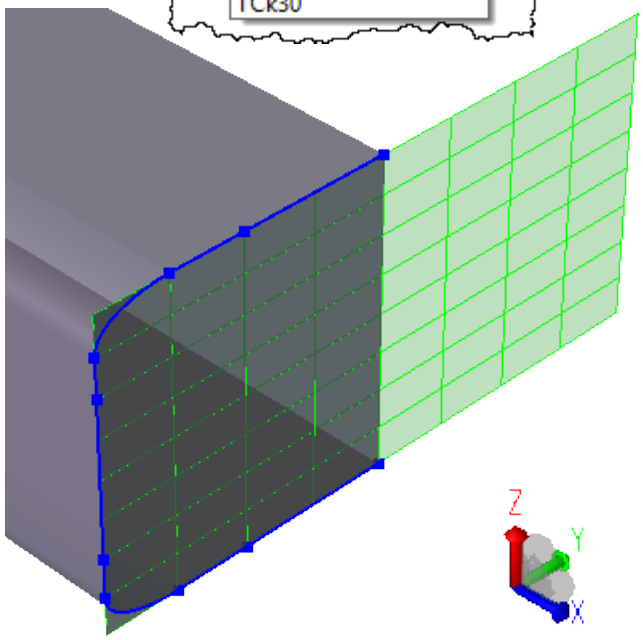
- Do a new *Model Verification*.

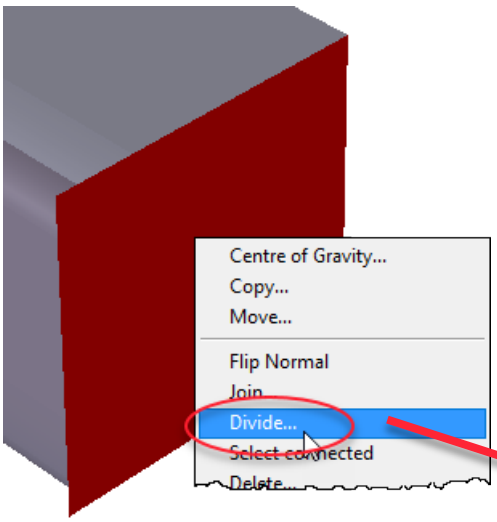


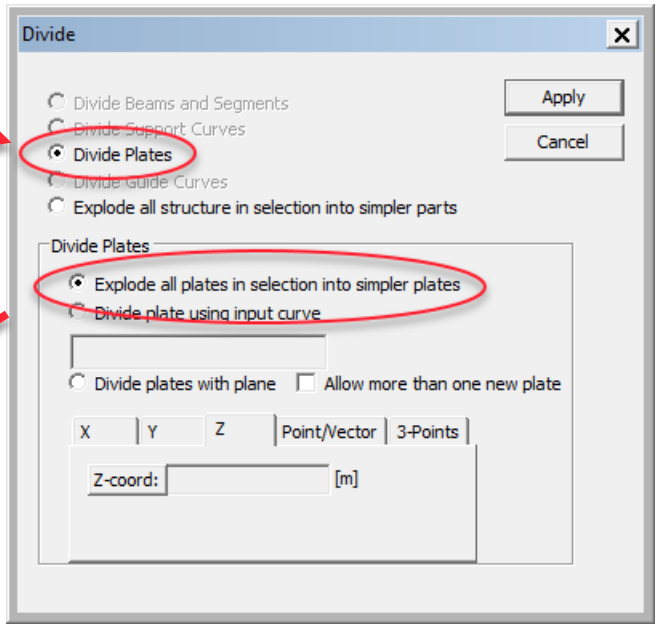
6 CREATE BULKHEADS AND WEB FRAMES

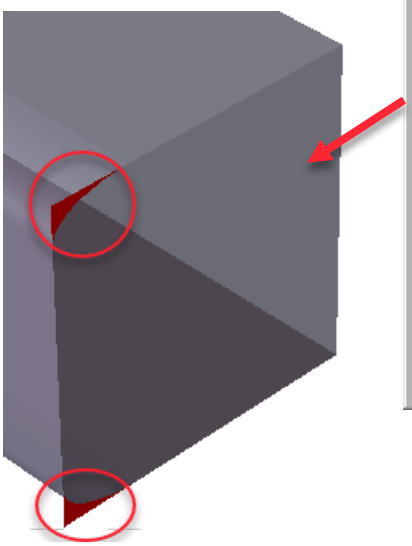
- Using the *Default display* configuration, create a bulkhead with thickness 12 mm at X = 0.
  - Change default thickness to Tck12.
  - Create a flat plate as shown at X = 0 by clicking the guide plane.
  - Select the plate, right-click and select *Divide* and *Explode all plates in selection into simpler plates* as shown below.
  - Select the superfluous parts and delete them by hitting the Delete button on the keyboard.





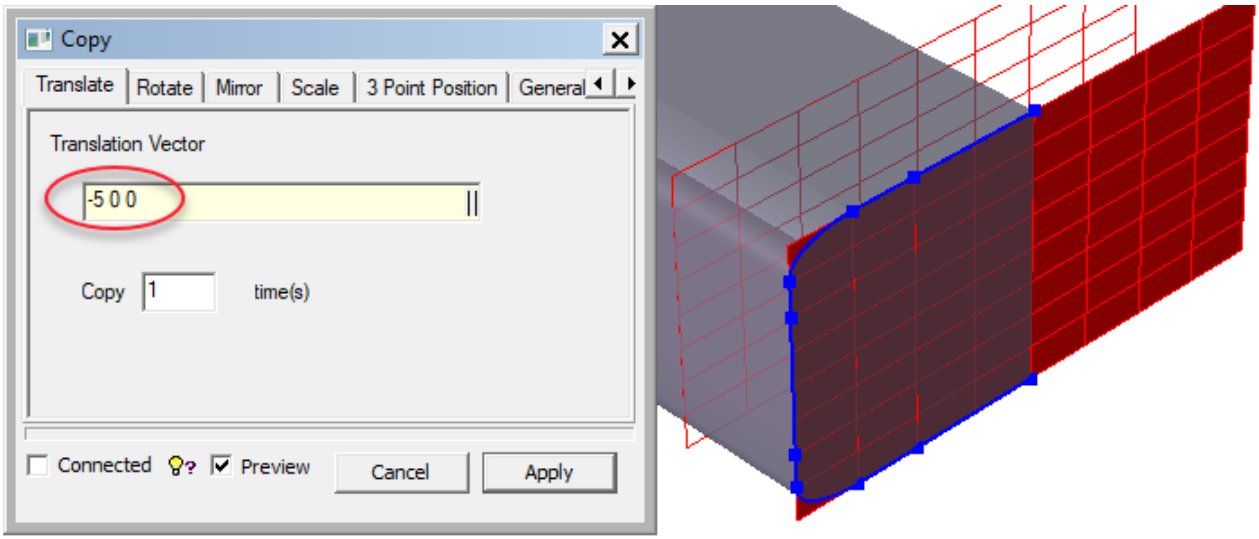




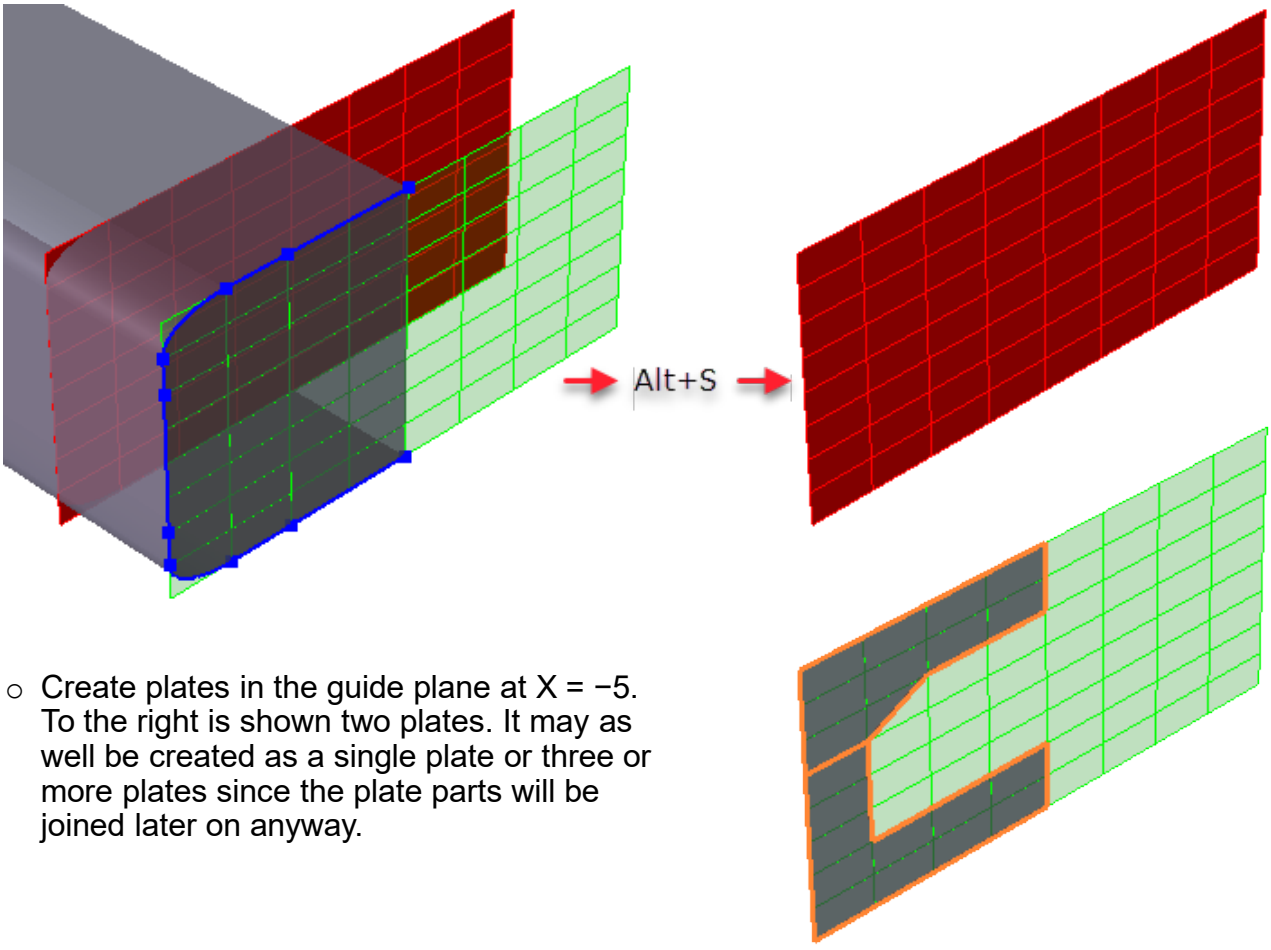


➢ Create a web frame with thickness 12 mm at X = −5 m.

- Copy the guide plane at X = 0 a distance of 5 m in negative X-direction.

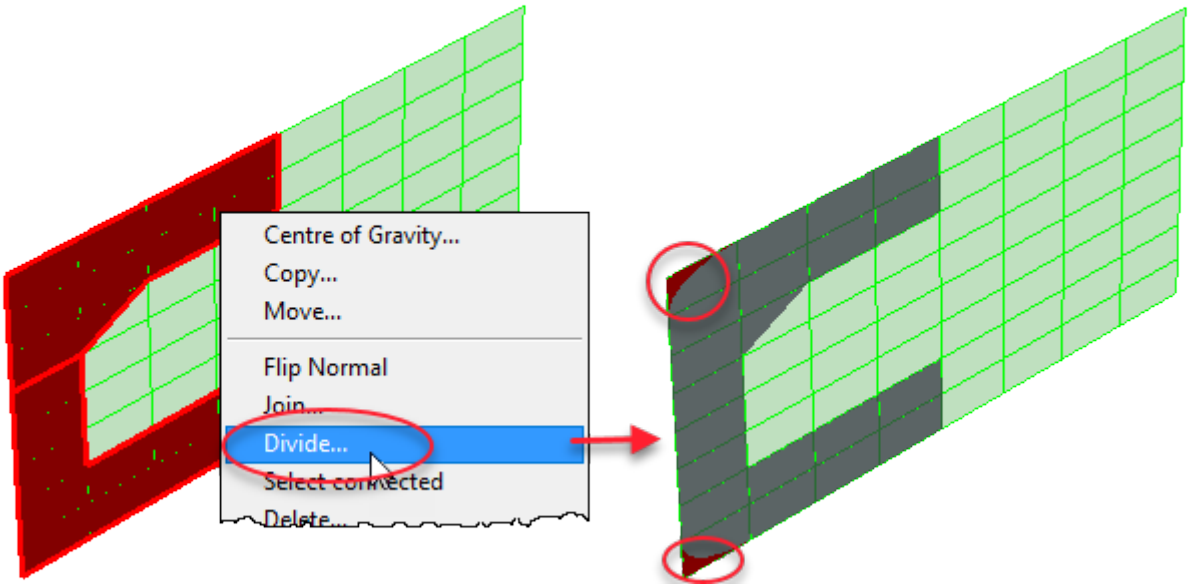


- To ease modelling the web frame show only the new guide plane. *Alt+S* is a shortcut to right-clicking and selecting *Visible model* | *Show selection only*.

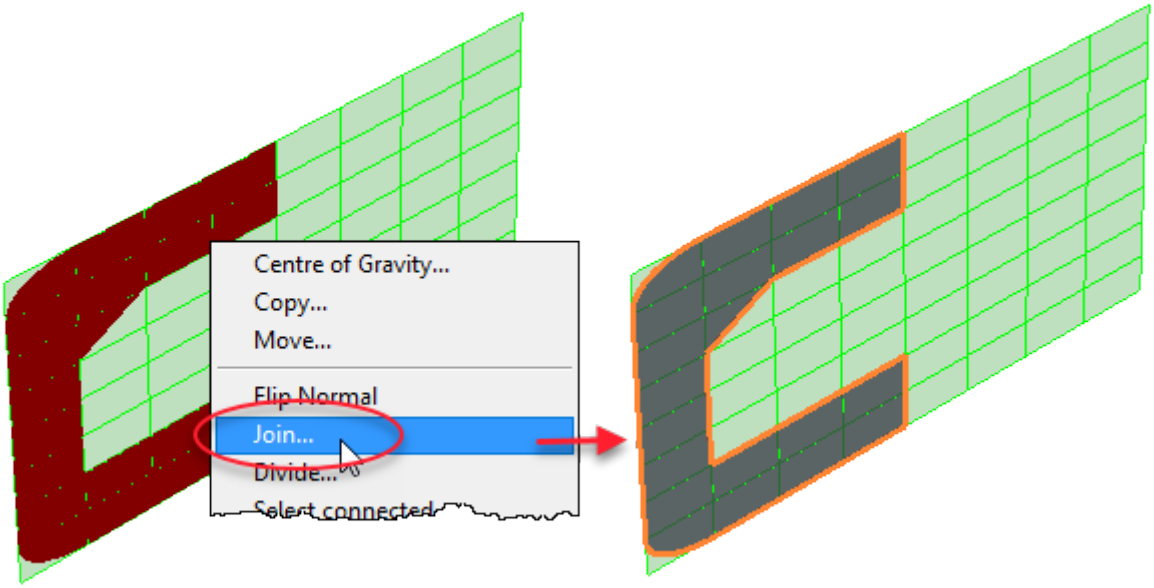


- Create plates in the guide plane at X = −5. To the right is shown two plates. It may as well be created as a single plate or three or more plates since the plate parts will be joined later on anyway.

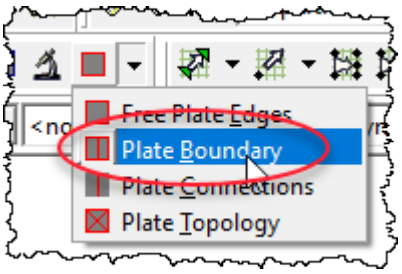
- Select the plates that must be trimmed, right-click, select *Divide* and *Explode all plates in selection into simpler plates*, and finally delete the superfluous parts.



- Select all plate parts, right-click and select *Join* to join them into a single plate.

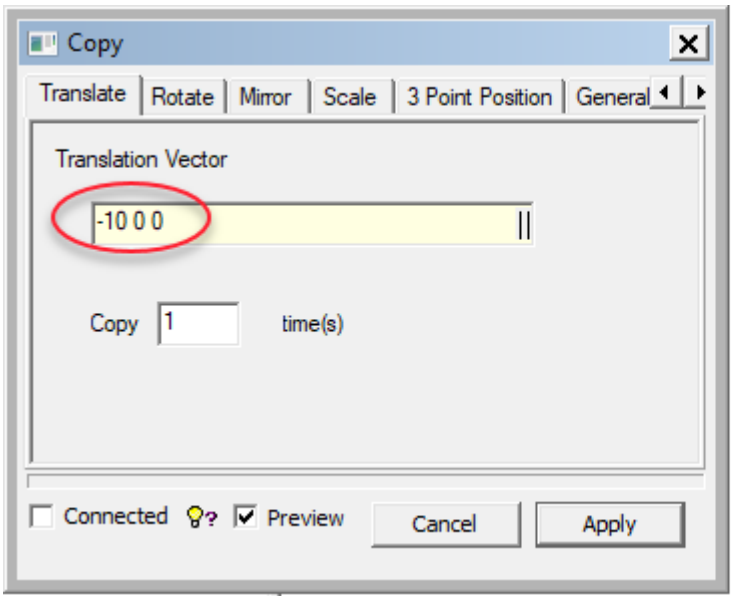


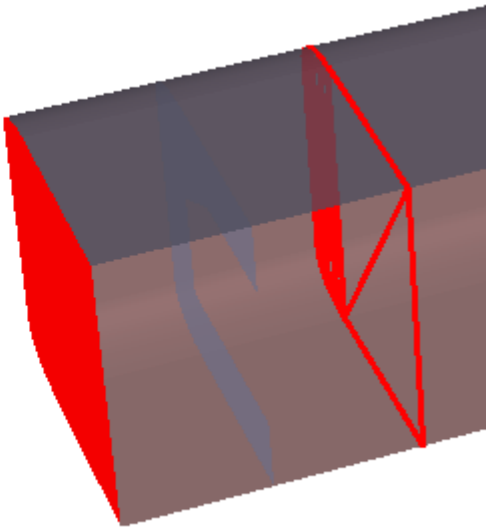
- To see the boundaries of individual plates press the *Plate Boundary* button as shown to the right.



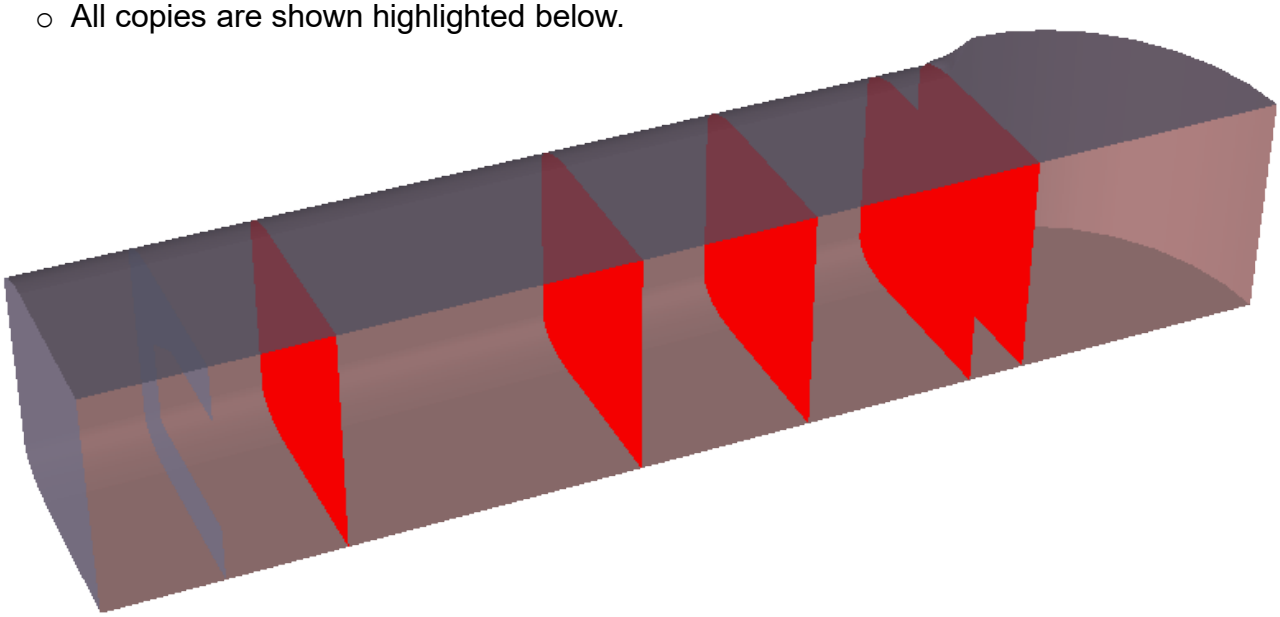
- Show the whole model by *Alt+A* (shortcut to right-clicking and selecting *Visible model* | *Show all*). And switch to *Modelling - Transparent* display configuration.

- Copy the bulkhead at X = 0 to X positions −10 m, −22.5 m, −30 m, −37.5 m and −40 m.



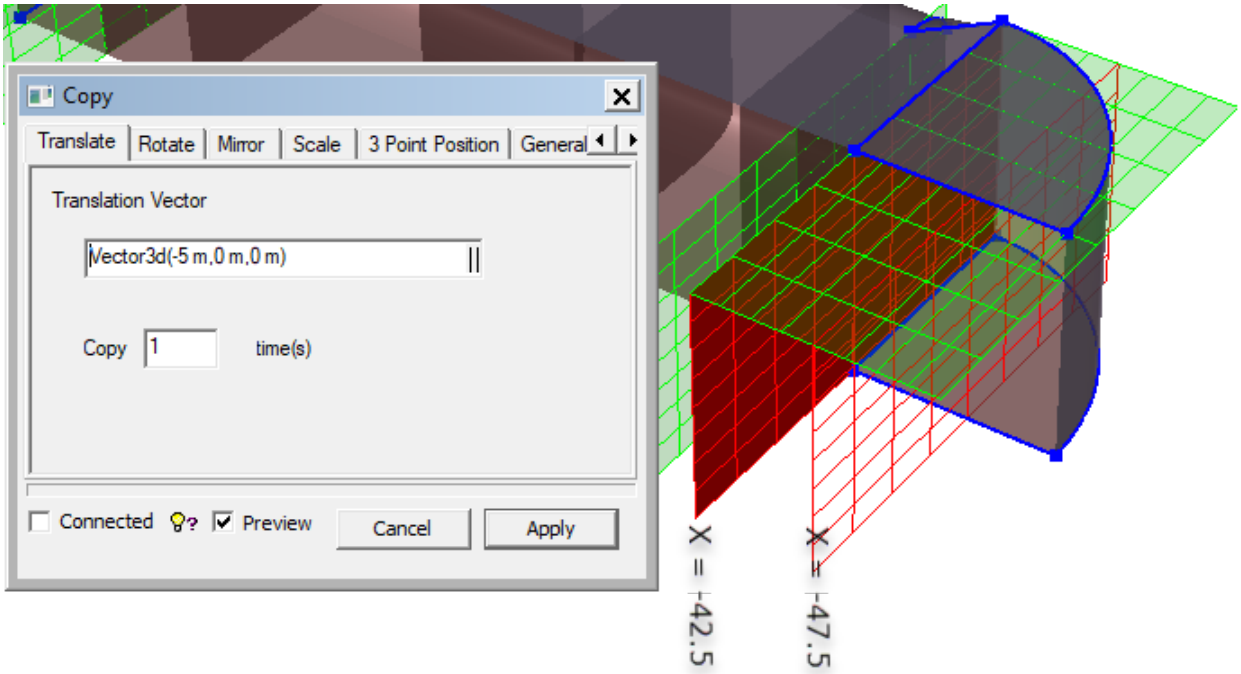


  - All copies are shown highlighted below.

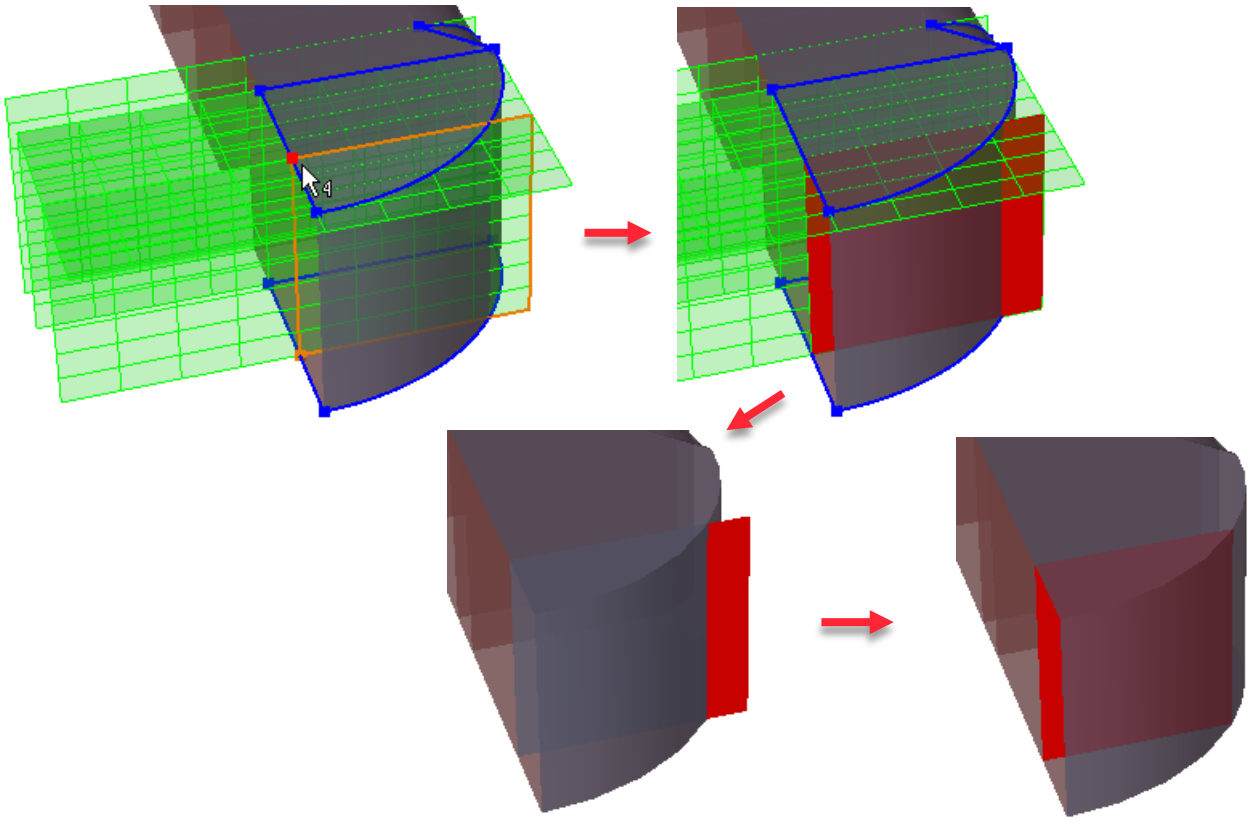


➢ Create a bulkhead at X = −47.5.

- First copy a guide plane to that position.

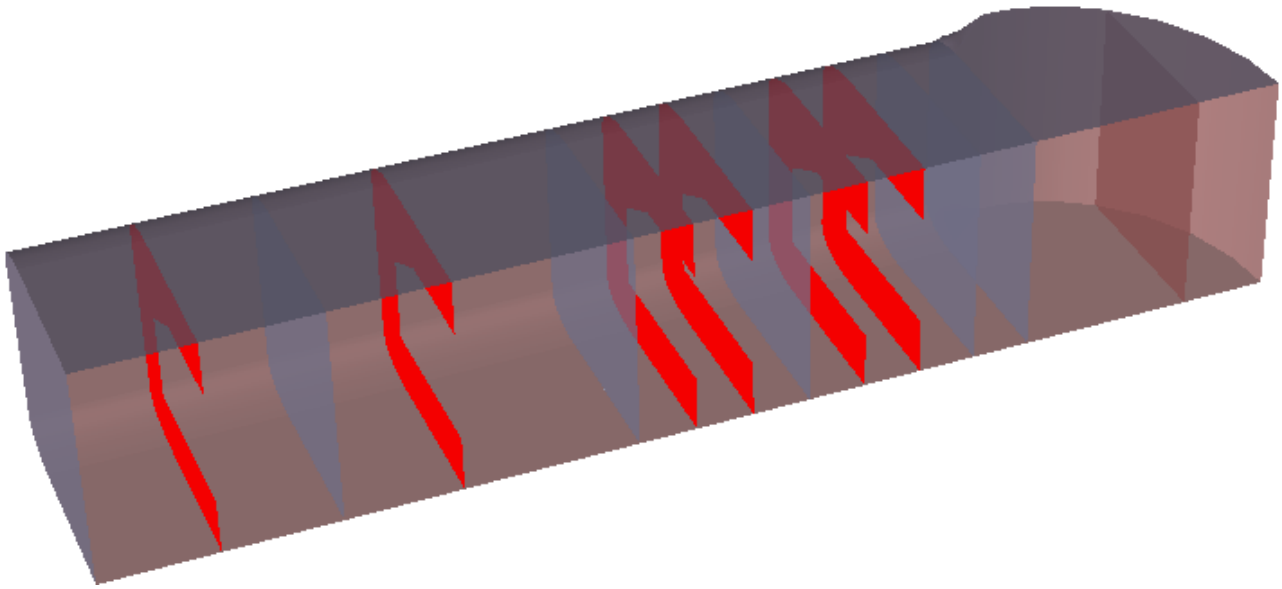


- Then create a plate and trim it as shown below.

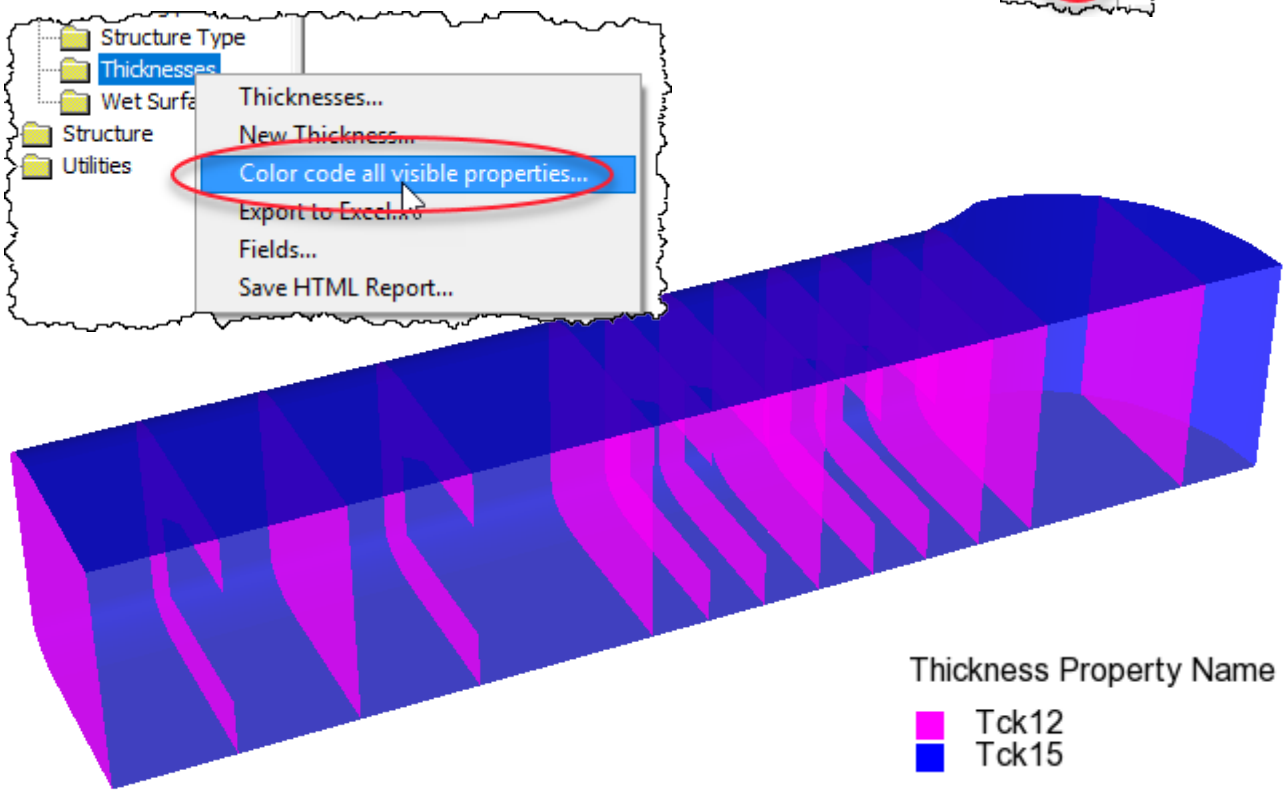


- Copy the web frame at X = −5 to X positions −15 m, −25 m, −27.5 m, −32.5 m and −35 m. All web frames are shown below highlighted. (The bulkheads and web frames do not need consistent plate normals.)

- Do a new *Model Verification*.



- Verify plate thicknesses by right-clicking the *Thicknesses* folder in the browser and selecting *Color code all visible properties* as shown below.
  - Toggle current colours by *Property color code of all visible structure*: 

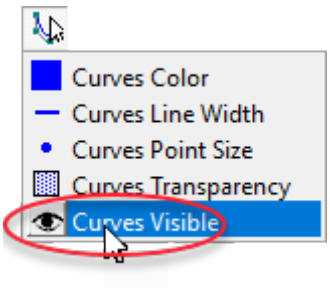


7 CREATE THE COLUMN

- Create a quarter of the column.

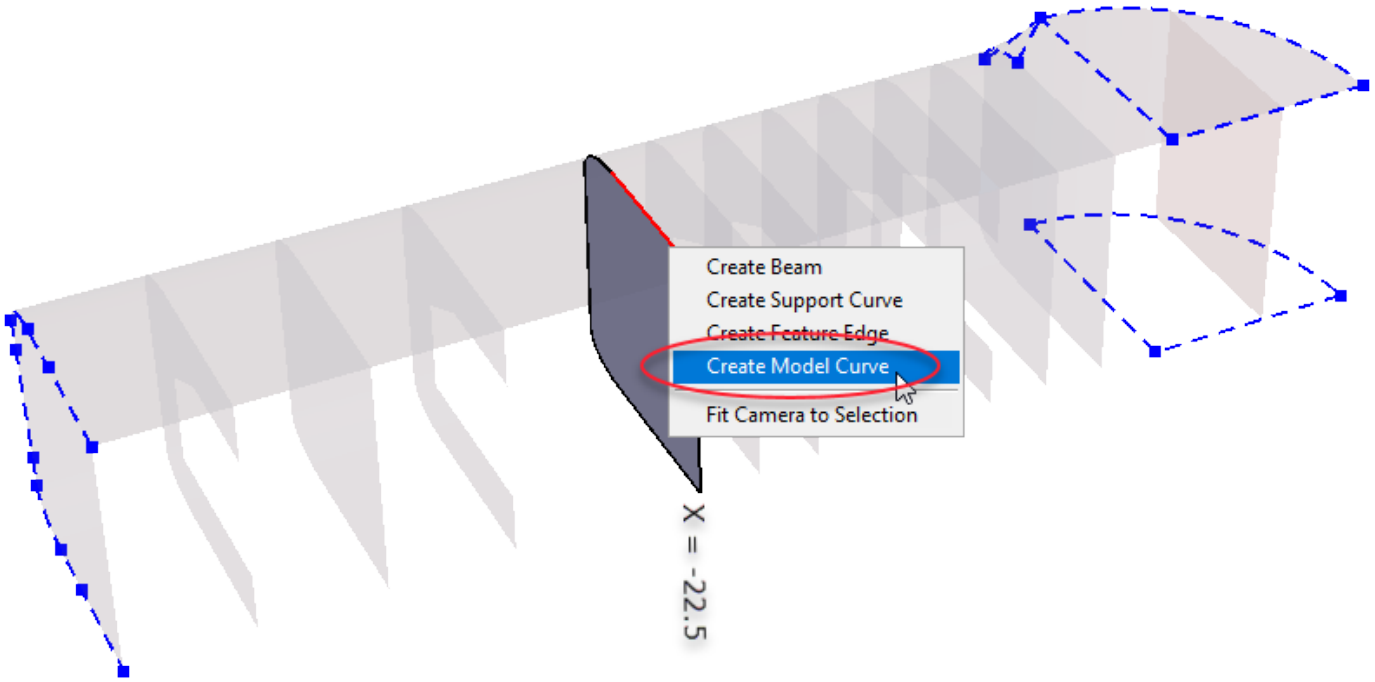
- The creation is based on using guide lines. To see the guide lines the *Default display* configuration can be used. Alternatively, to avoid seeing also the guide planes thereby congesting the display, use *Modelling - Transparent* and right-click the *Guide curve selection* button and open the eye symbol.

  - Note that changes to visibility, colour, transparency, etc. by right-clicking the selection buttons is stored in the registry and is therefore persistent. You may, therefore, want to set it back or use *View* | *Options* | *Settings* (or Alt+O) and click *Restore defaults* to set back to installation settings.



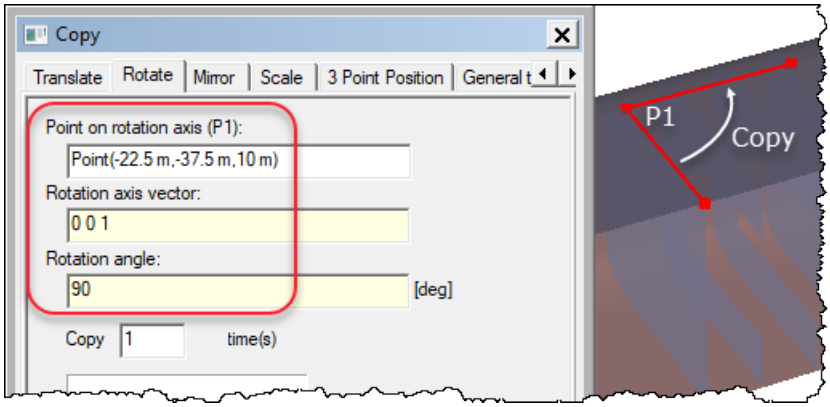
- Create guide lines and curves to be used for creating the column.

  - Double-click the bulkhead at X = −22.5 m, select the edge at top, right-click it and select *Create Model Curve* as shown below.

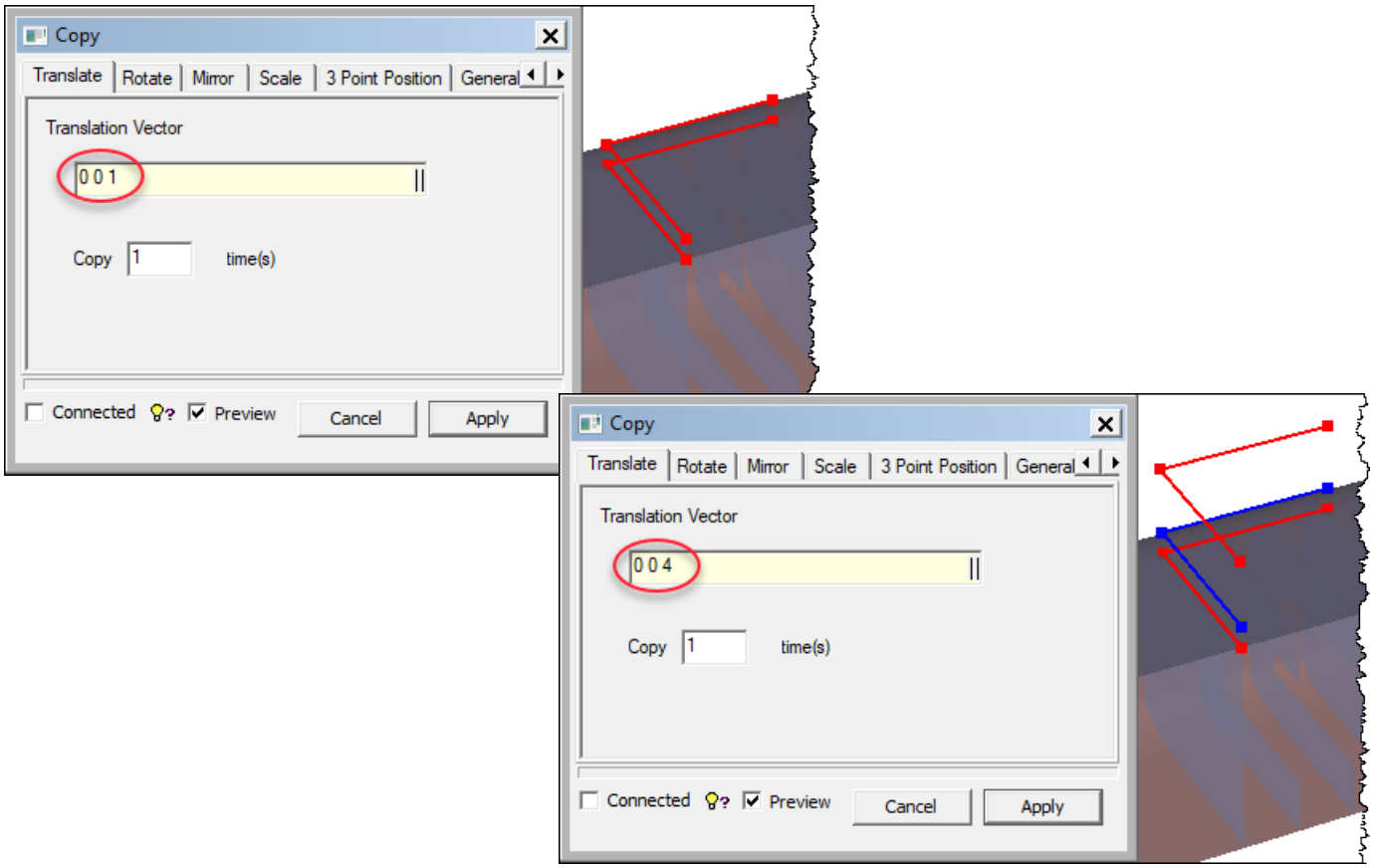


  - Double-click outside the model to return to normal display mode.

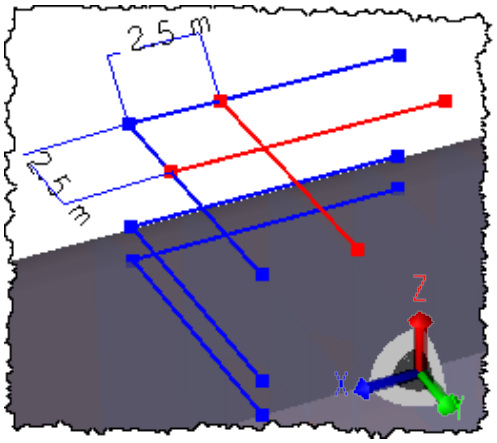
  - Select the new guide line and copy it by a rotation as shown.



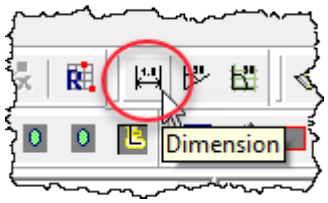
- Select the two guide lines and copy them upwards 1 m and 4 m.



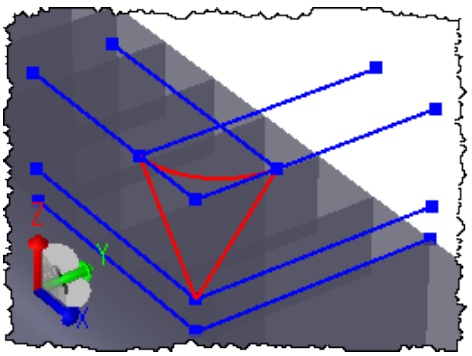
- Copy the two upper guide lines at Z = 14 m. One is copied −2.5 m in X-direction and the other 2.5 m in Y-direction to achieve the result shown to the right. The copies are highlighted.



  - Measurements are inserted using the *Dimension* button and deleted by the *Refresh graphics* button:



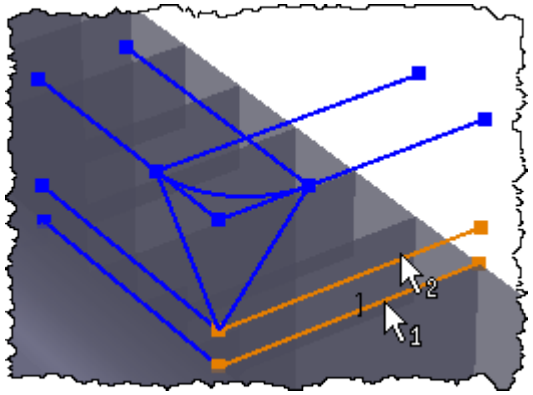
- Create a guide curve using *Guiding Geometry* | *Conic Section* | *Elliptic Arc from Center and two Points*. The arc is highlighted to the right.

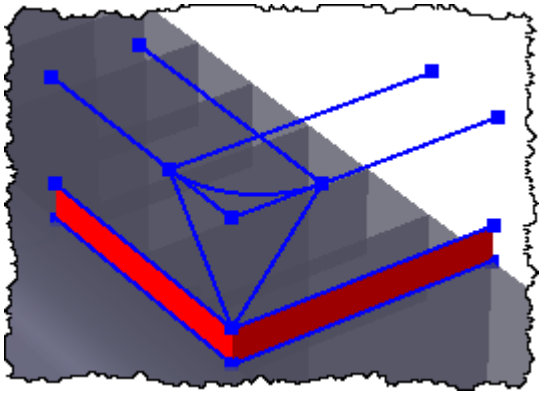


- Create two guide lines using *Guiding Geometry* | *Lines* | *From Two Points*. The two lines are highlighted to the right.

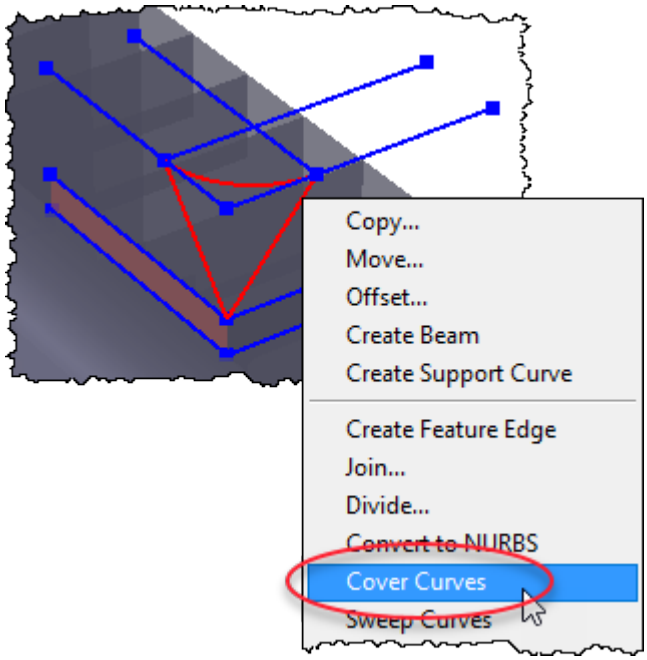
- Change to default plate thickness Tck30.

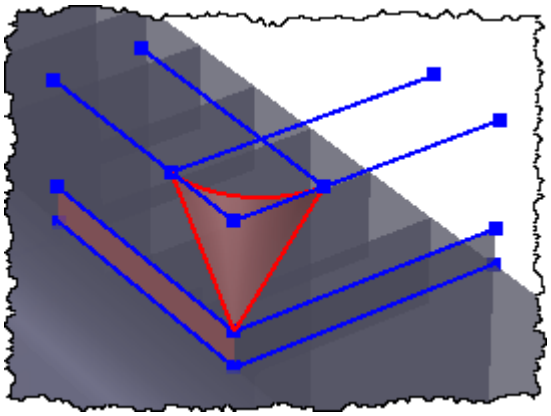
- Use *Structure* | *Free Form Shells* | *Skin/Loft Curves* to create the lower part of the column. Click the first guide line and double-click the second guide line as shown. Do the operation twice to create the two surfaces highlighted to the right below.

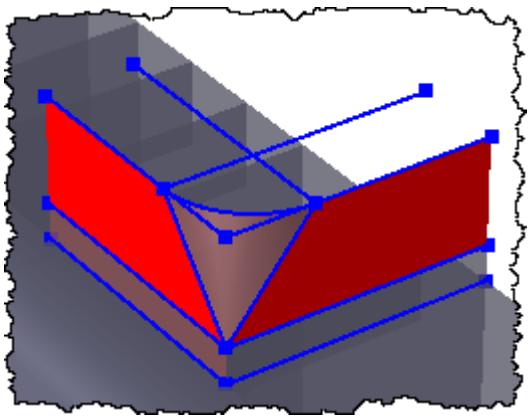




- Select the three guide lines/curves enclosing the transition from sharp to curved corner, right-click and select *Cover Curves* to create the surface shown to the right below.

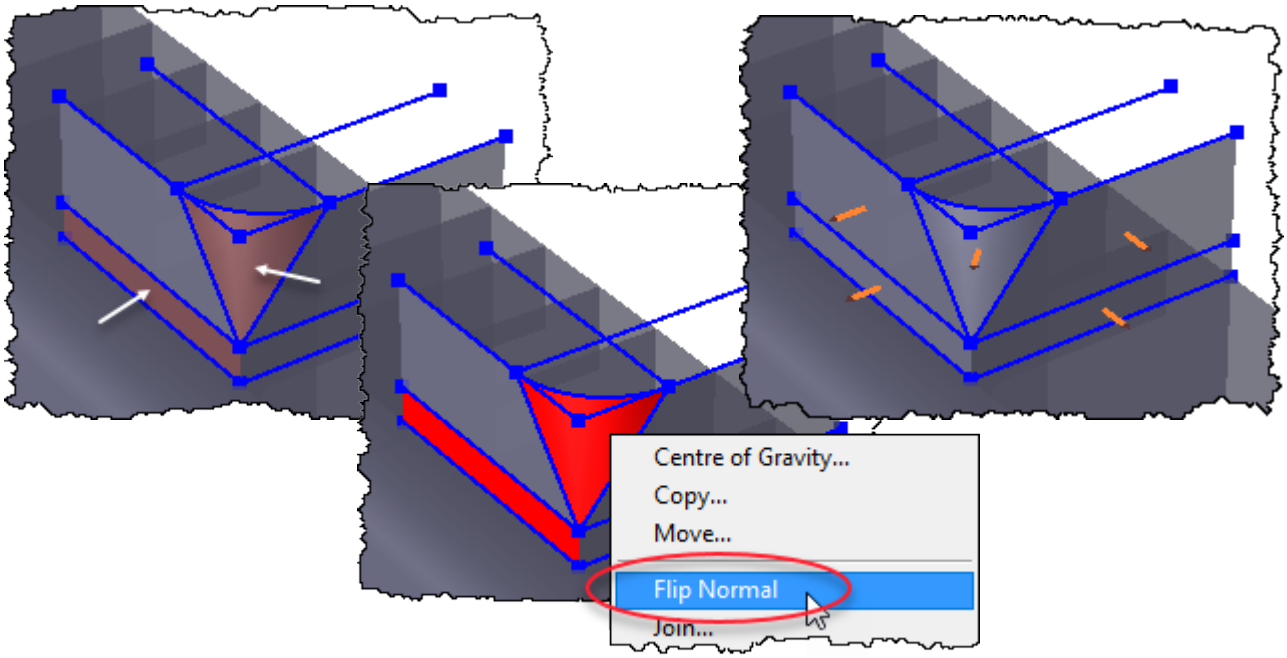




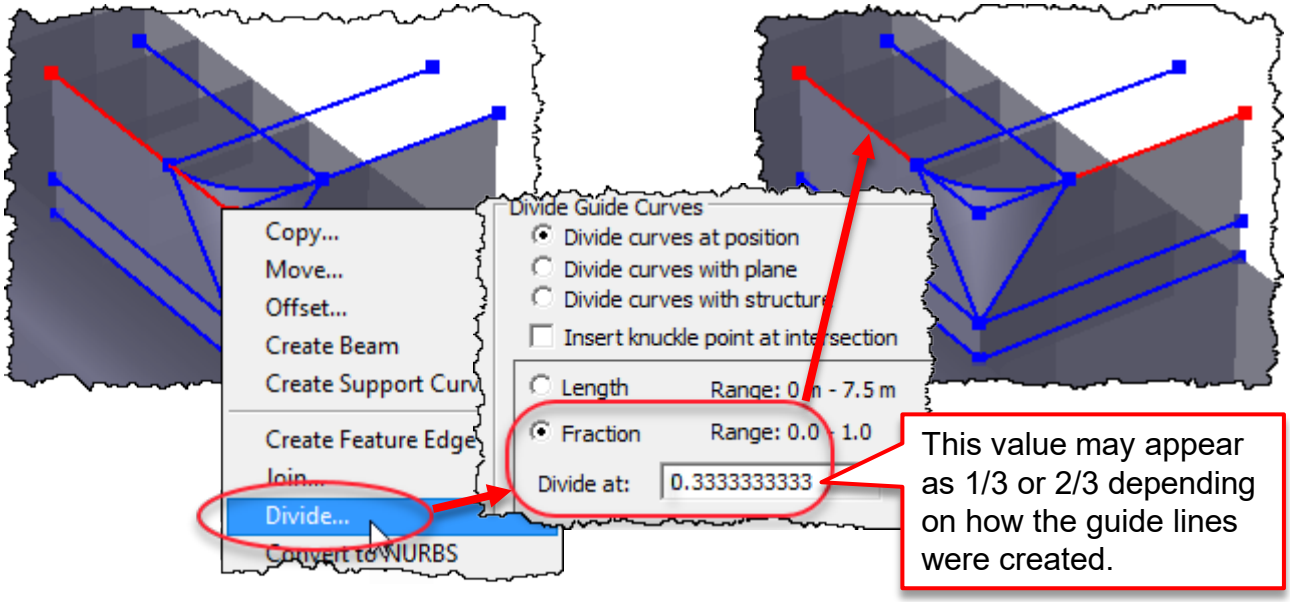


- Use *Structure* | *Flat Plates* | *Flat Plate* to create the two quadrilateral plates highlighted to the right. Double-click the fourth corner to close the creation process for a plate.

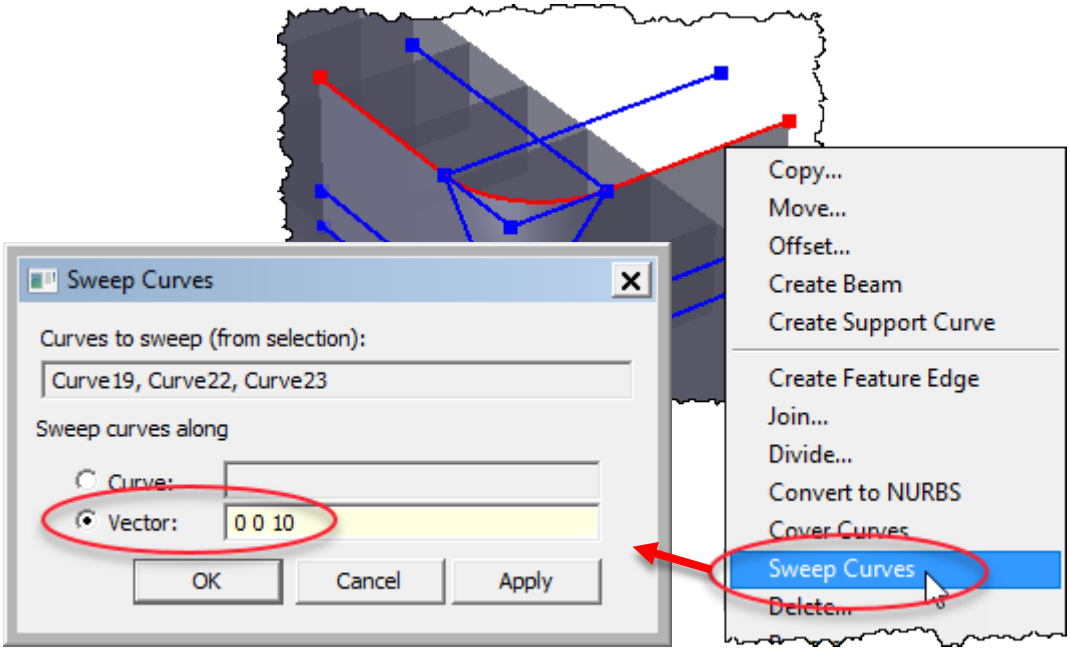
- Depending on the creation process, some of the surfaces may have their surface normals pointing inwards. Select these, right-click and select *Flip Normal* to make all surface normals pointing outwards. The surfaces should be bluish when seen from the outside. The surface normal may also be displayed by selecting the surfaces, right-clicking and selecting *Labels* | *Local Coordinate System*. Remove the surface normals by selecting the surfaces, right-clicking and selecting *Labels* | *Clear Labels*.



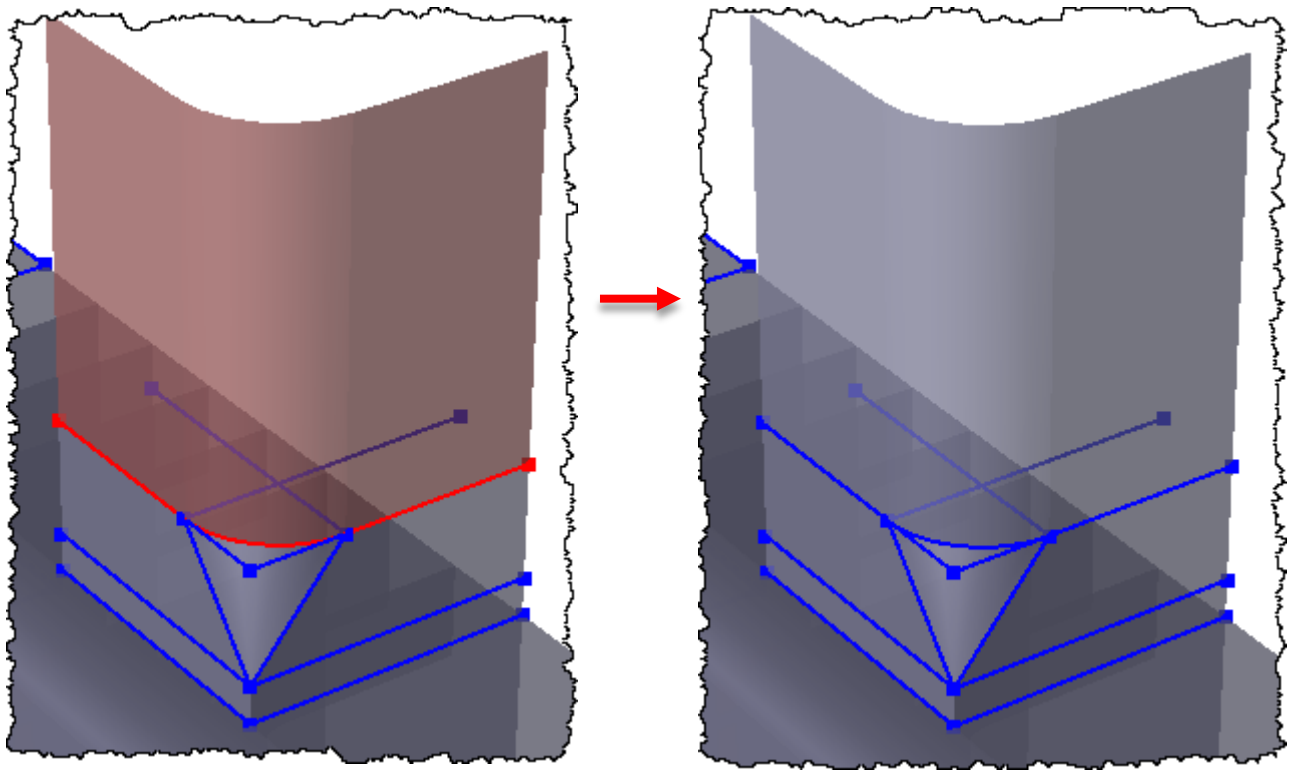
- The two guide lines on top of the flat column sides must be split as shown below. Click the split point to insert the fraction into the *Divide* dialog. The division should produce the guide lines highlighted to the right below.



➢ Create the upper part of the column by *Structure* | *Free Form Shells* | *Sweep Curves Dialog*, or more easily, select the guide lines/curves, right-click and select *Sweep Curves* to open the *Sweep Curves* dialog. In the dialog specify the sweep vector (0,0,10).



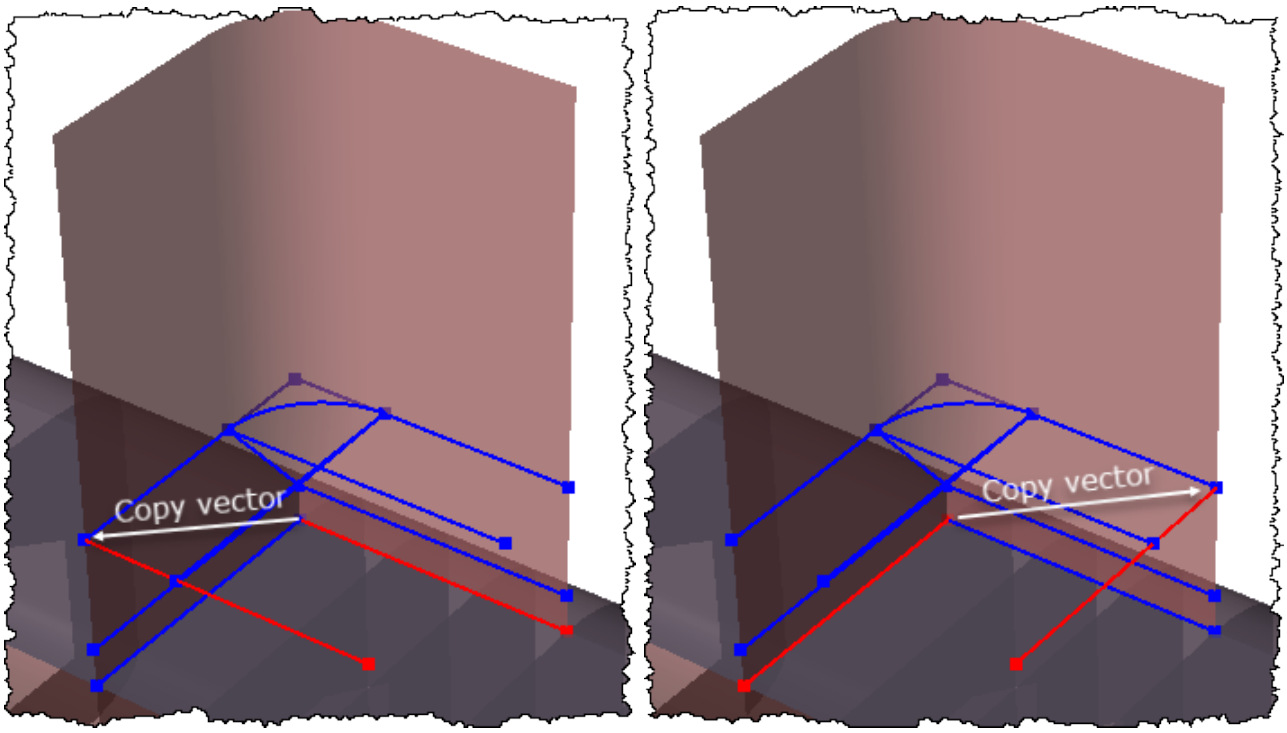
➢ If the new surfaces have their surface normals pointing inwards, flip them.



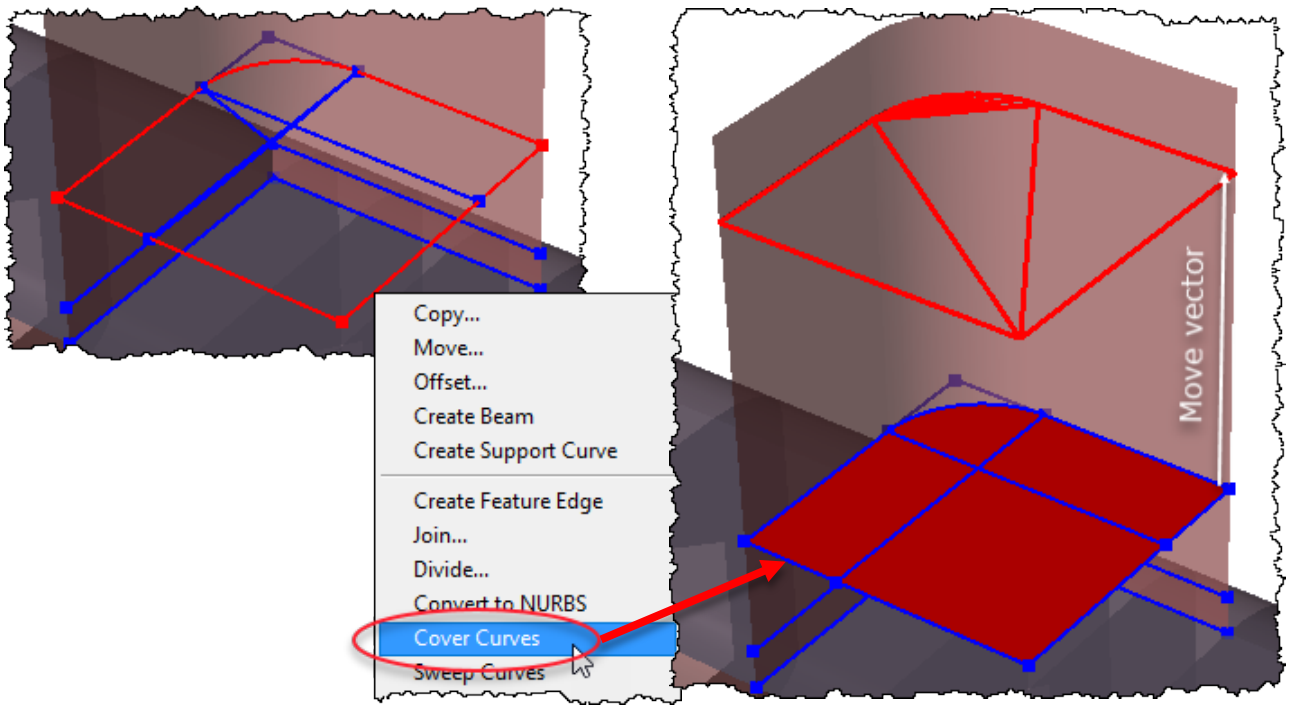
➢ Create a deck at elevation 22 m in the column.

➢ Change to default plate thickness Tck12.

- Copy two guide lines at elevation 10 m to elevation 14 m as shown. Fetch the copy vectors from the display.

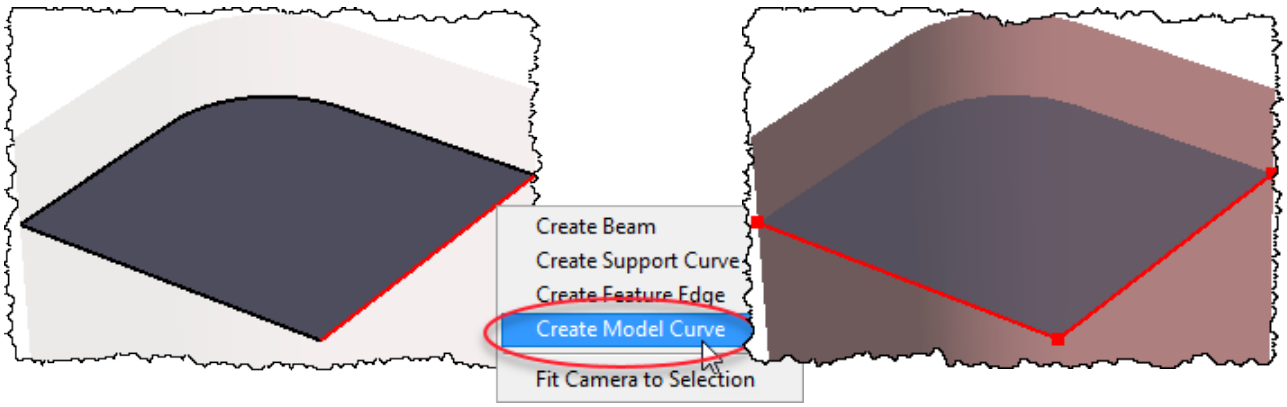


- Create a deck at elevation 14 m by a cover operation as shown below. Then move the deck up from Z = 14 m to Z = 22 m, i.e. a move vector (0,0,8).

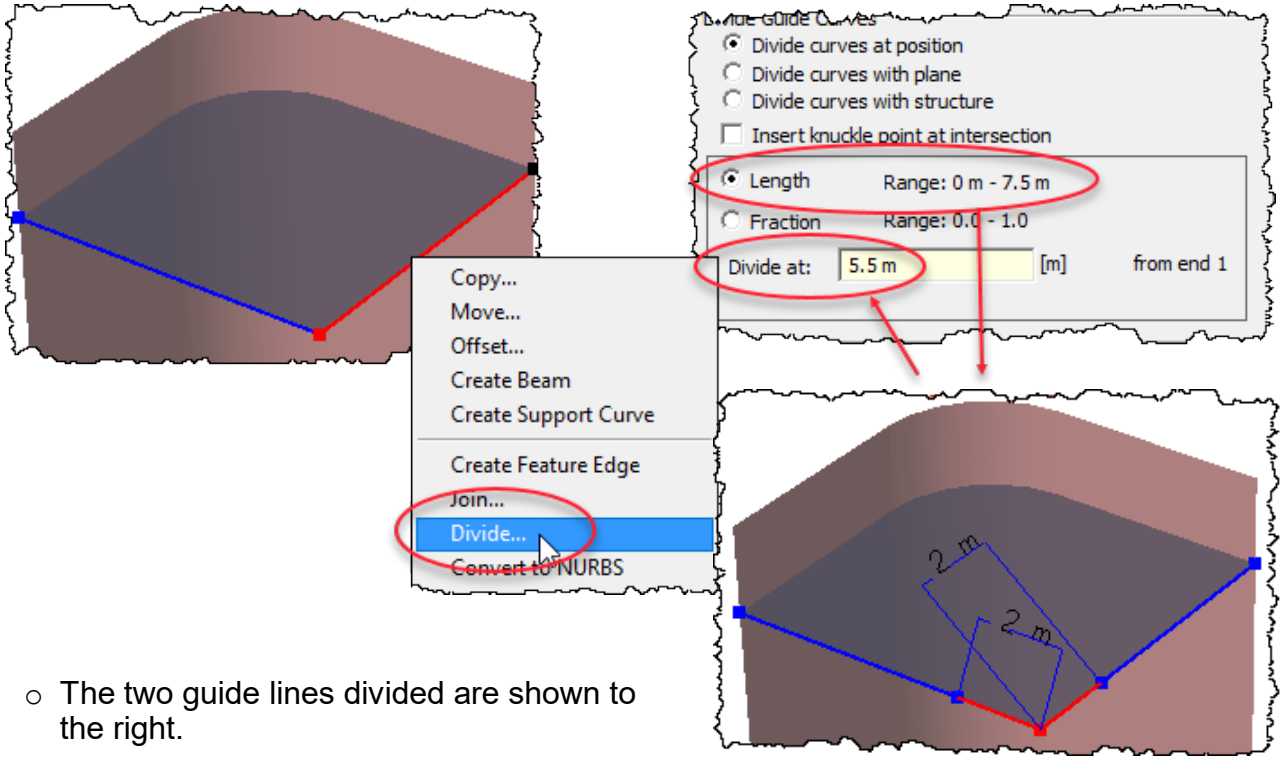


➢ Make an opening in the deck at elevation 22 m.

- Create a guide line by double-clicking the deck, selecting an edge and clicking *Create Model Curve*. Do so to create the two guide curves shown to the right below.

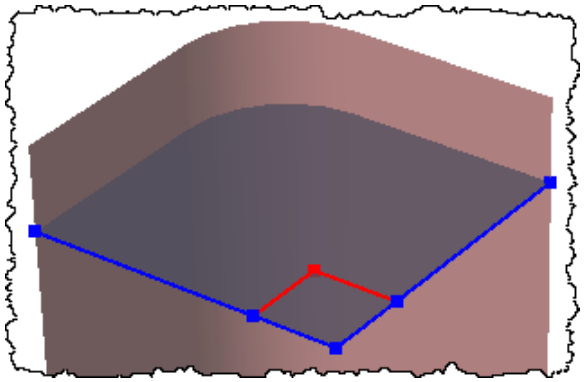


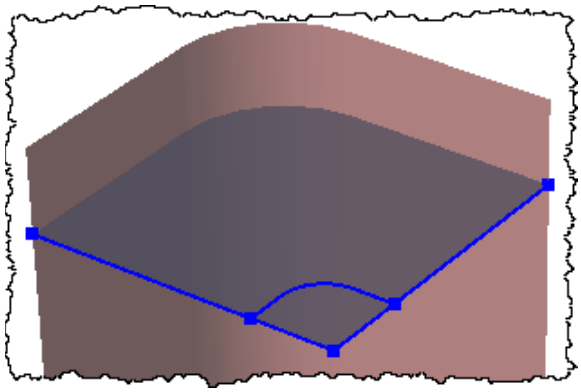
- Divide each of the two new guide lines at 2 m from the column centre point.
  - Select a line, right-click and select *Divide*.
  - In the *Divide* dialog select to divide at a given length *from end 1*. But unless you happen to know which end is end 1 you don't know whether to divide at 2 m or 5.5 m (see that the *Range* is given as '0 m - 7.5 m').
  - Simply try one value and if incorrect, use Ctrl+Z to undo and try the other.



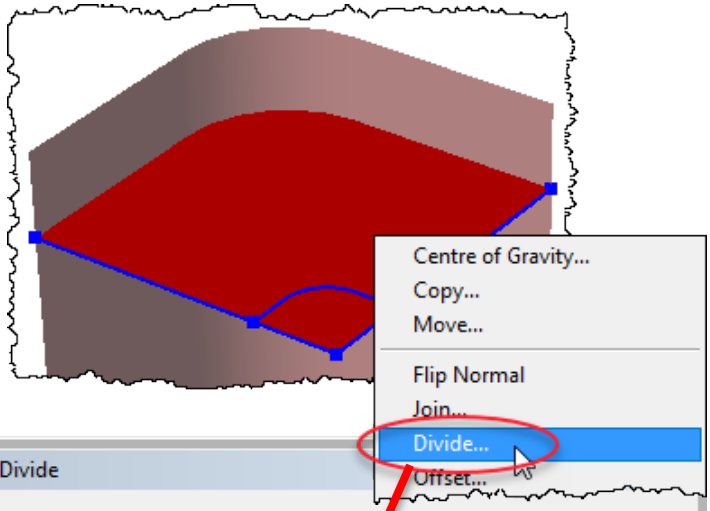
- The two guide lines divided are shown to the right.

- Copy the two guide lines to form the two new guide curves highlighted below.
- Use *Guiding Geometry* | *Conic Sections* | *Circular Fillet* to join the two guide lines by a fillet of radius 1 m.

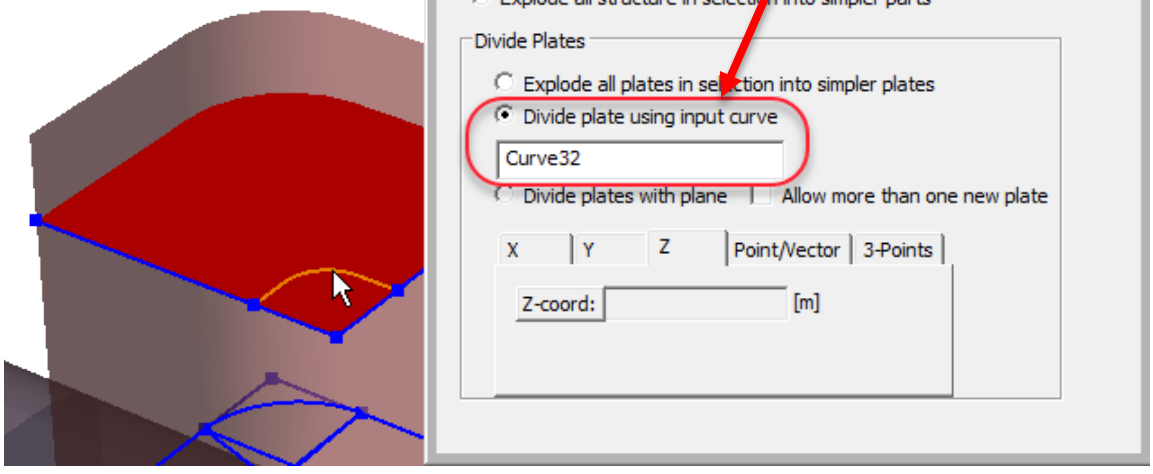




- Select the deck, right-click and select *Divide*.

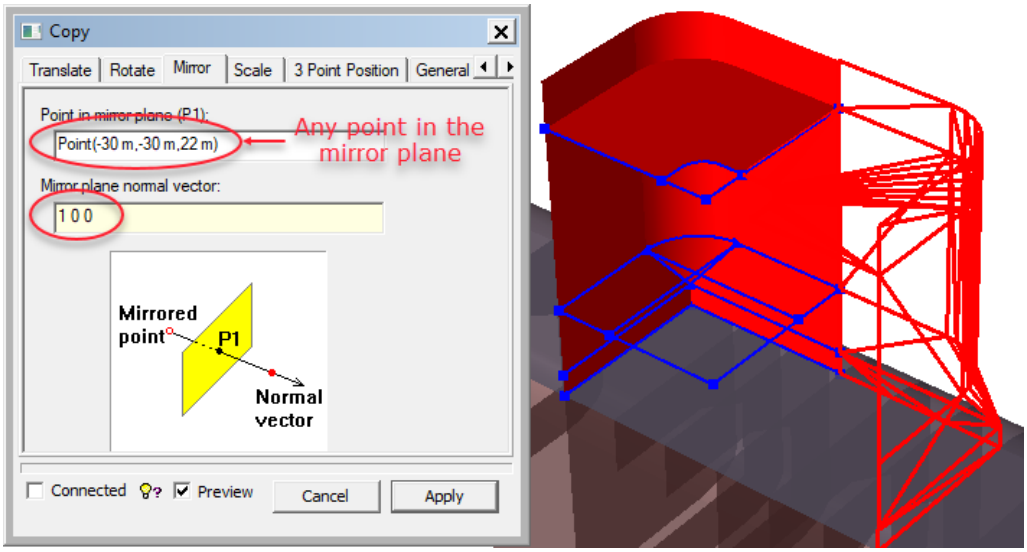


- In the *Divide* dialog select *Divide plate using input curve* and click the joined guide line.



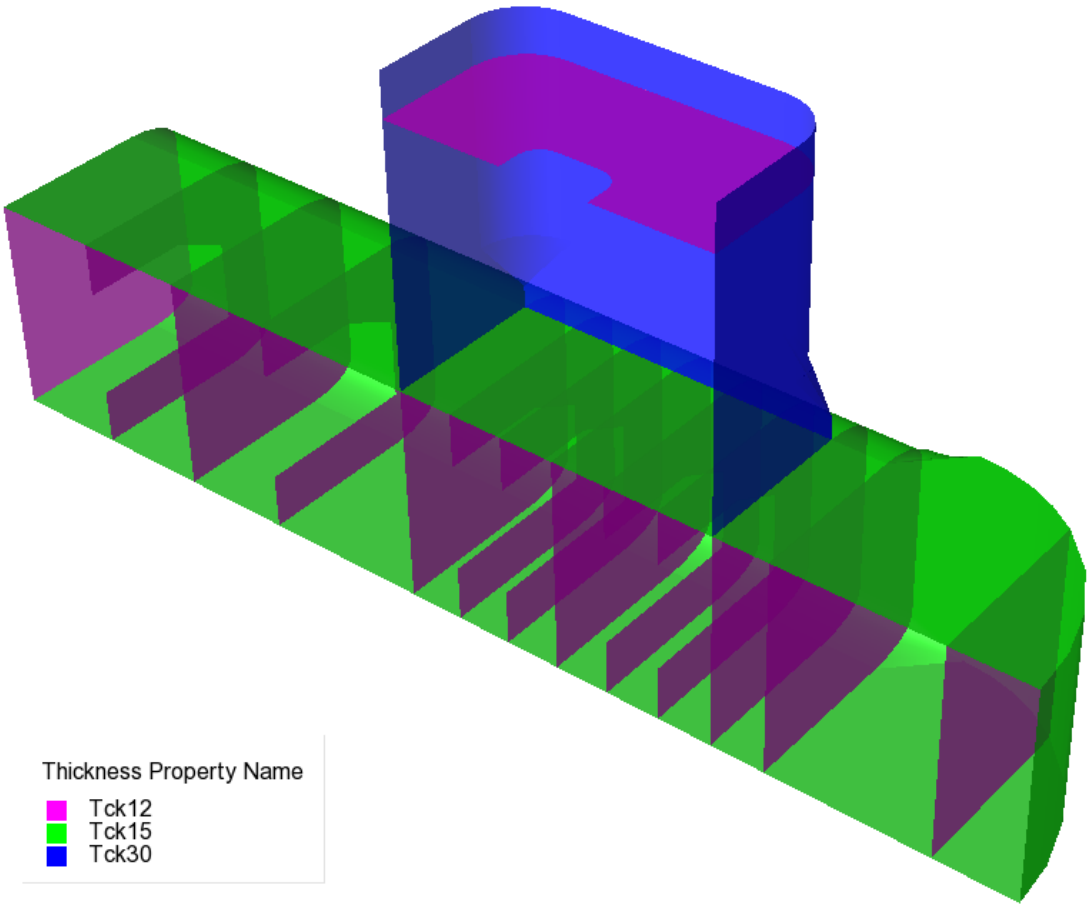
- Delete the plate part in the middle of the column to create an opening.

- Copy by mirroring the column as shown below. Make sure to select all surfaces constituting the column.



- At this point you may want to switch off display of guide curves.

- Colour coding the plate thicknesses, the model should appear as shown below.

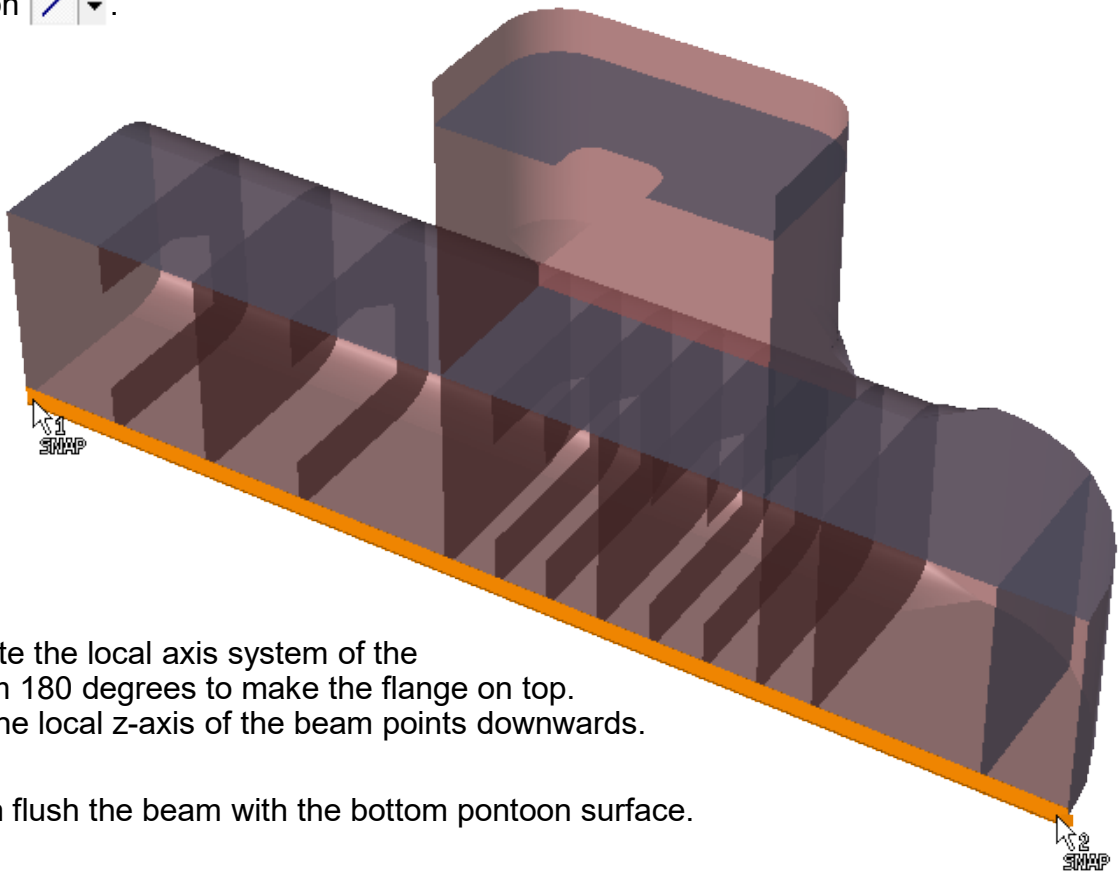


- Do a new *Model Verification*. Investigate and correct any problems.

8 CREATE PONTOON STIFFENERS

- Create longitudinal stiffener beams at bottom of the pontoon.
  - Make sure the default beam cross section is Tbar885x200x14x35.

- Create a beam using *Structure* | *Beams and Piles* | *Straight Beam*, or press the *Beam* button .

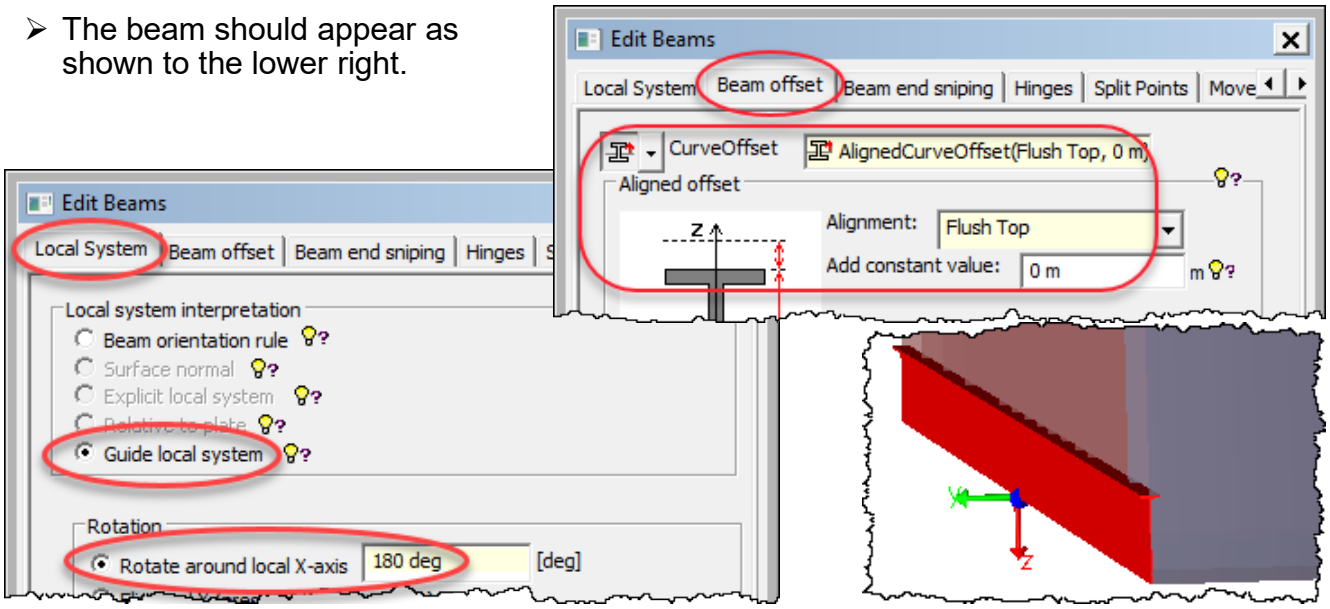


- Rotate the local axis system of the beam 180 degrees to make the flange on top. I.e. the local z-axis of the beam points downwards.

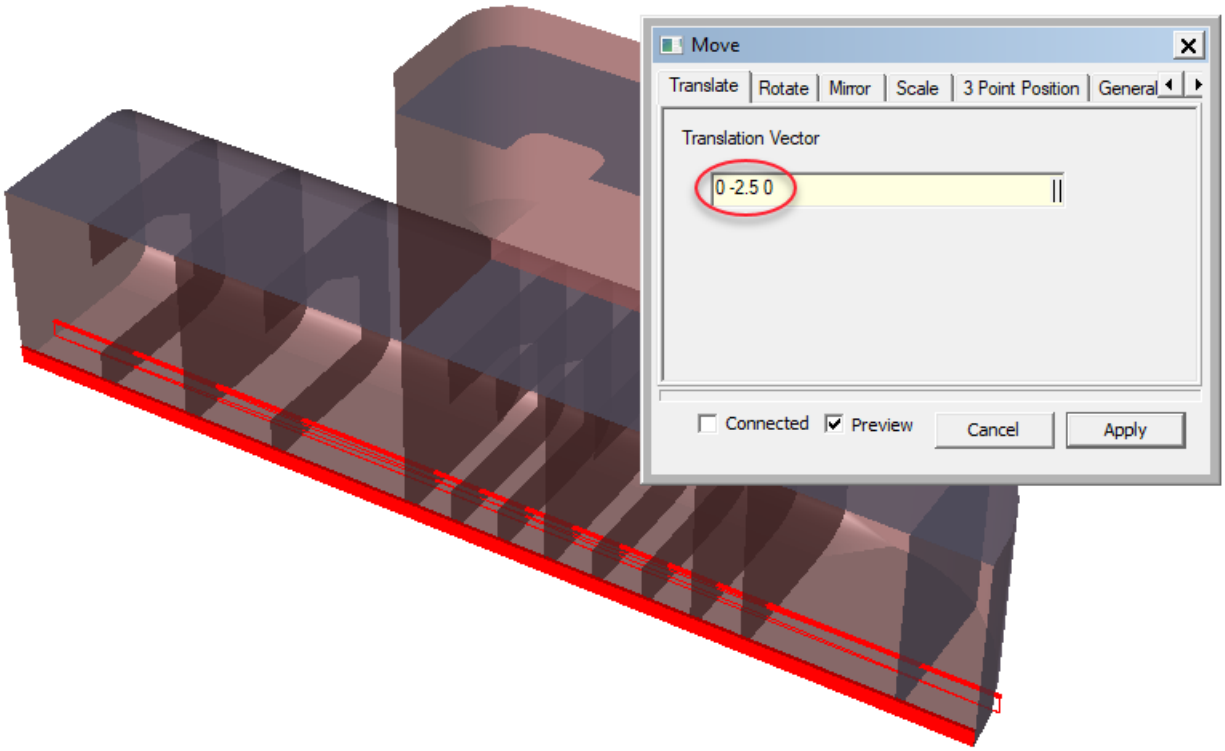
- Then flush the beam with the bottom pontoon surface.

- Both these operations are done by right-clicking the beam, selecting *Edit Beam* and selecting the tabs *Local System* and *Beam offset*.

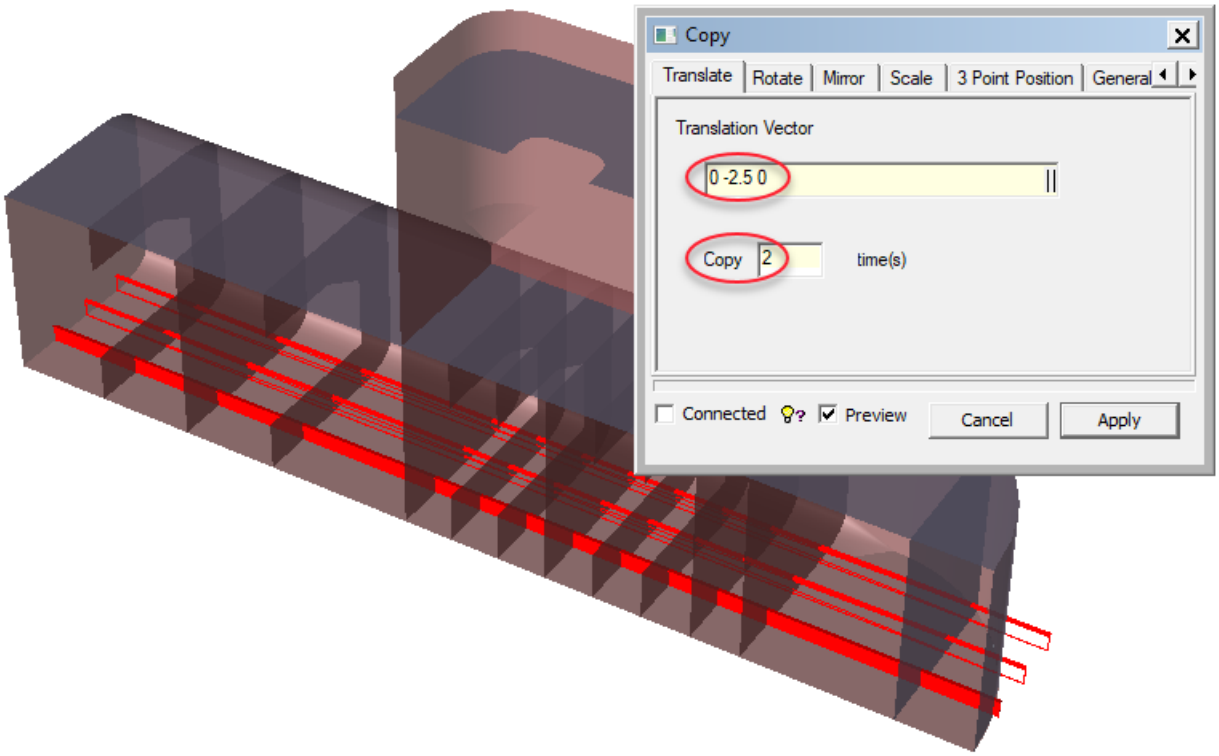
- The beam should appear as shown to the lower right.



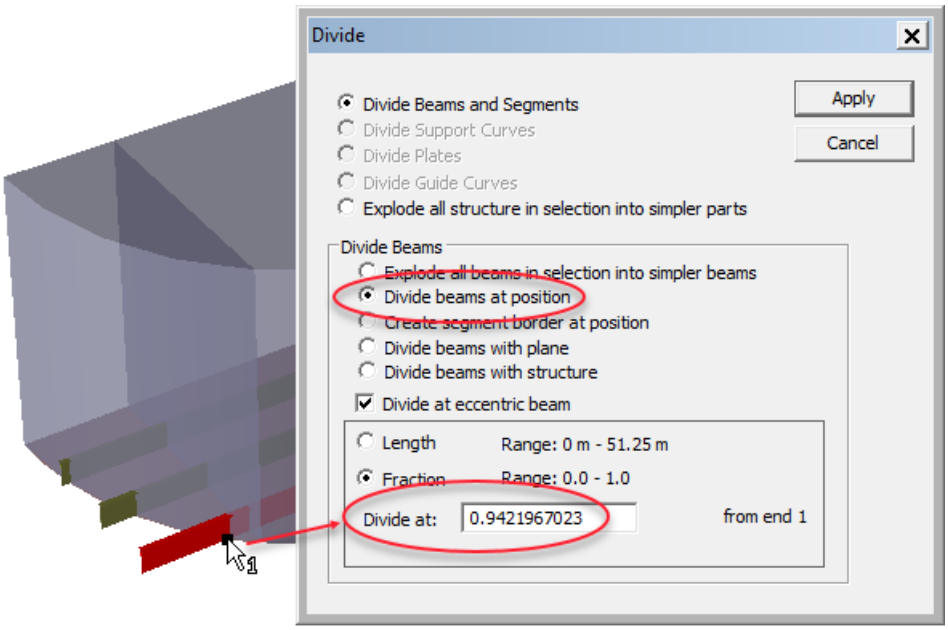
- Move the beam from the centre line of the pontoon with Y = −30 m to its proper position at Y = −32.5 m. I.e. a move vector (0,−2.5,0).



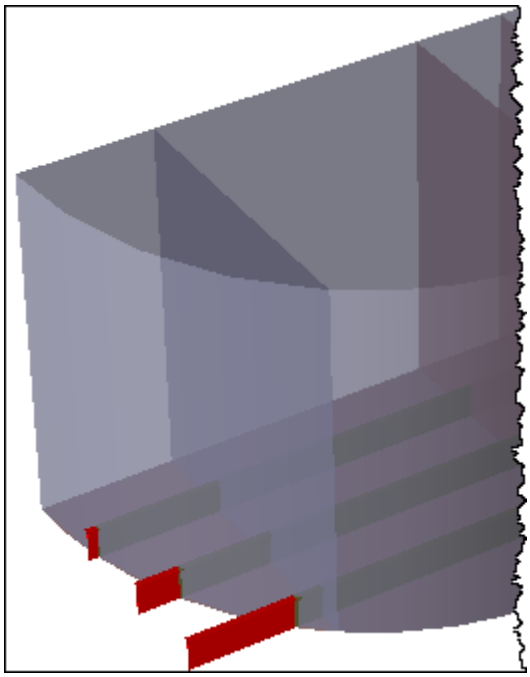
- Copy the stiffener to positions Y = −35 m and Y = −37.5 m. I.e. a copy vector (0,−2.5,0) twice.

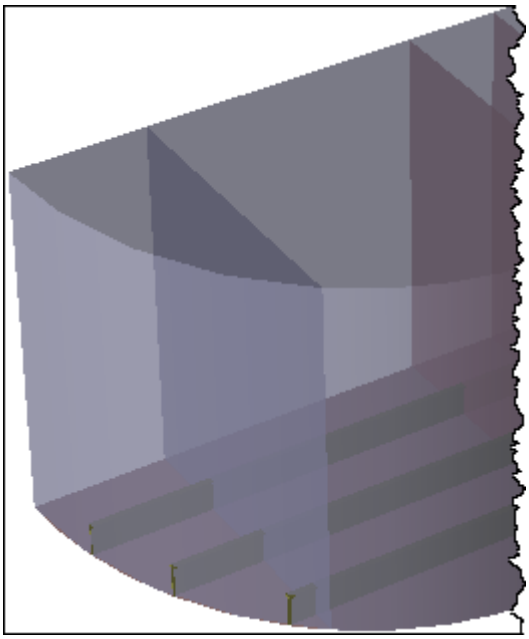


➢ Notice that the stiffeners protrude from the pontoon hull in one end. Trim them by dividing them where they intersect the pontoon surface. This is shown below for one of the stiffeners. Repeat this for the other two stiffeners.



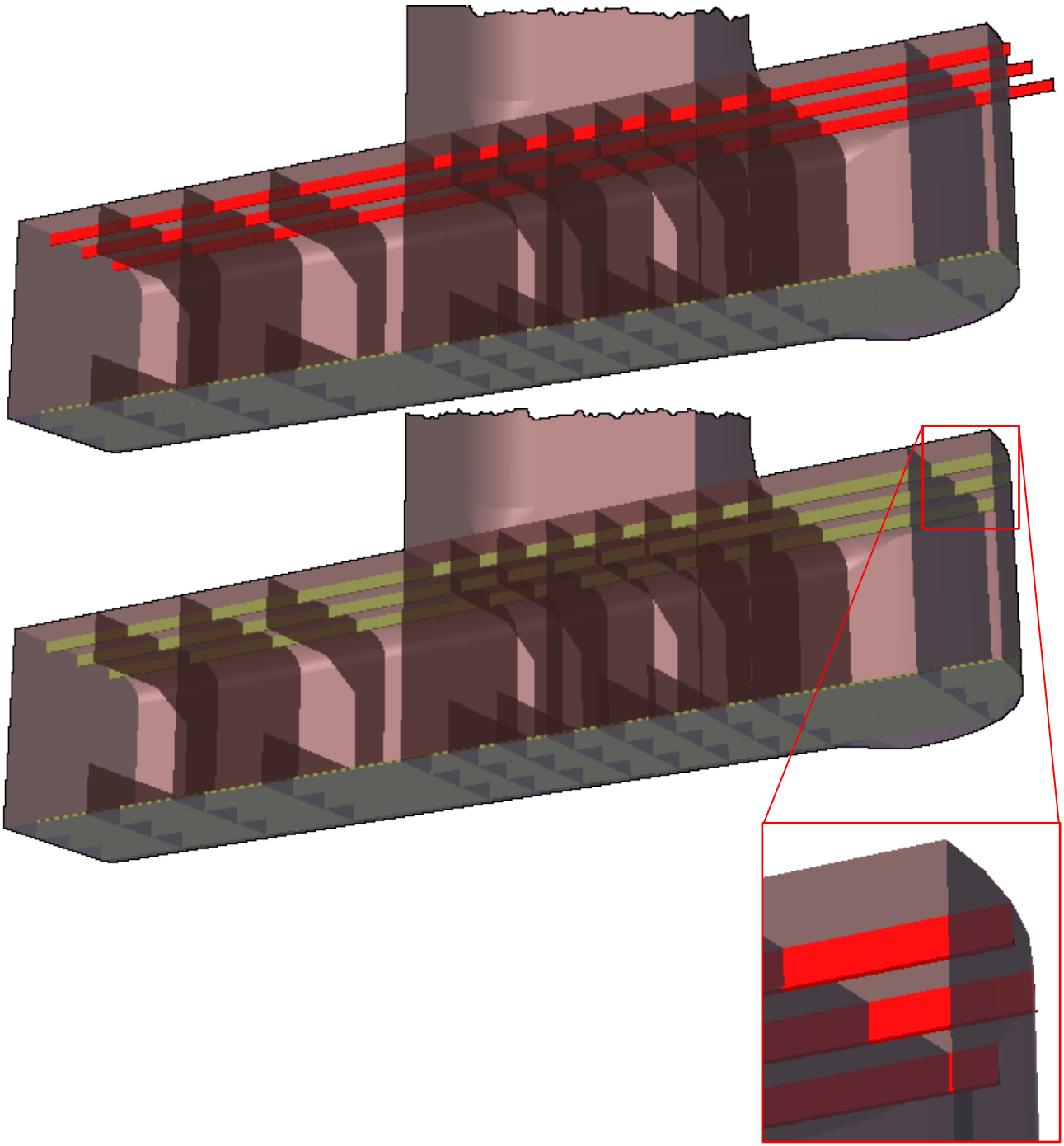
➢ Delete the protruding parts.





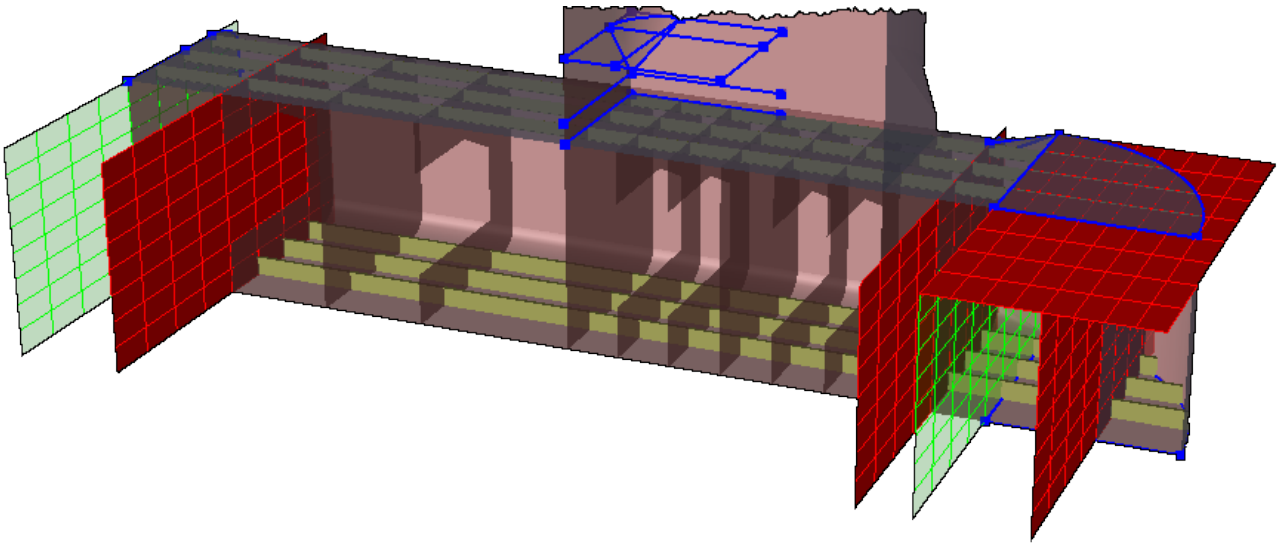
- Create longitudinal stiffener beams at top of the pontoon.
  - The beam cross section is Tbar575x150x12x25, so change the default to this.

- Follow the same procedure as for the stiffeners at bottom. I.e. first create the stiffener at the pontoon centre line and flush it with the pontoon surface. Note that in this case the local z-axis should point upwards. Move it −2.5 m in Y-direction. Then copy it twice −2.5 m in Y-direction. Finally, delete the protruding parts.

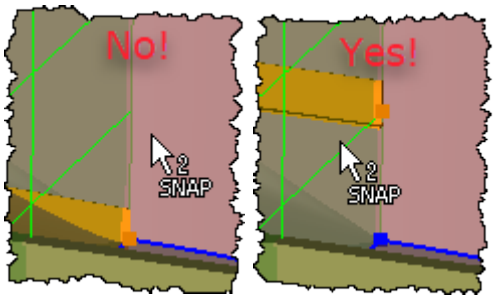


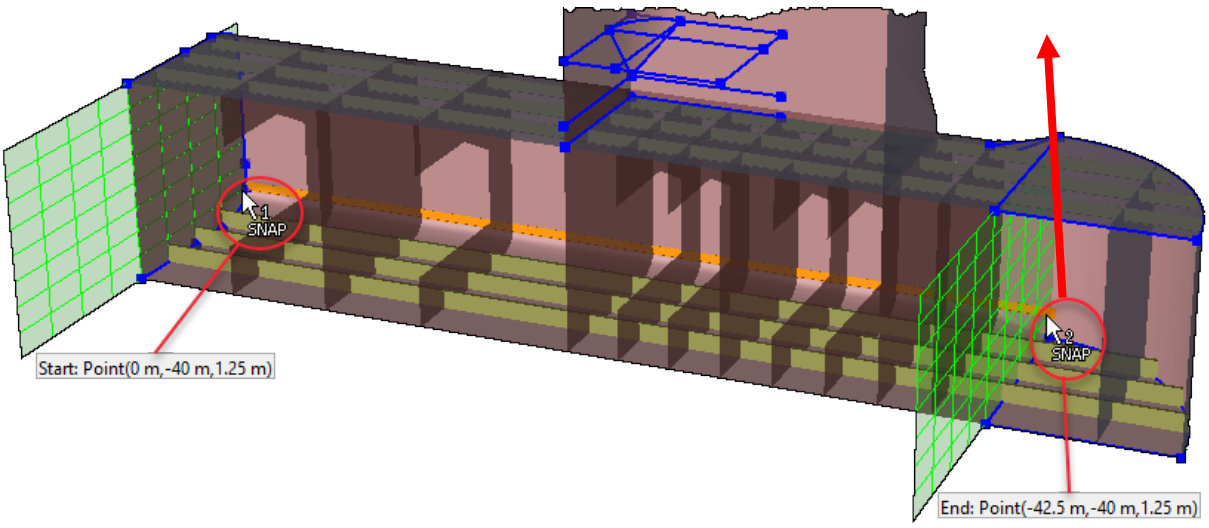
- Create longitudinal stiffener beams at the side of the pontoon.
  - The beam cross section is Tbar425x120x12x25, so change the default to this.

- To ease the creation of the stiffeners switch to *Default display* configuration, rotate to see into the pontoon and remove the four guide planes highlighted in red below from the display by Alt+minus.



- The lowermost stiffener at the side of the pontoon should be positioned using the vertical guide planes at X = 0 m and X = −42.5 m as shown below. Make sure the coordinates for the start and end points appearing in the lower left corner of the GeniE window are correct before clicking.
  - Note that to snap to a point in a guide plane the pointer must hover over the guide plane when clicking as shown to the right.

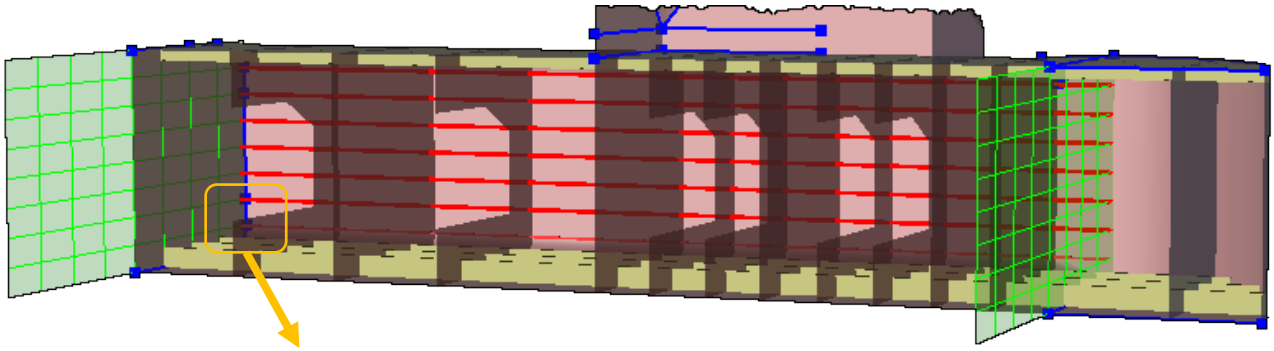


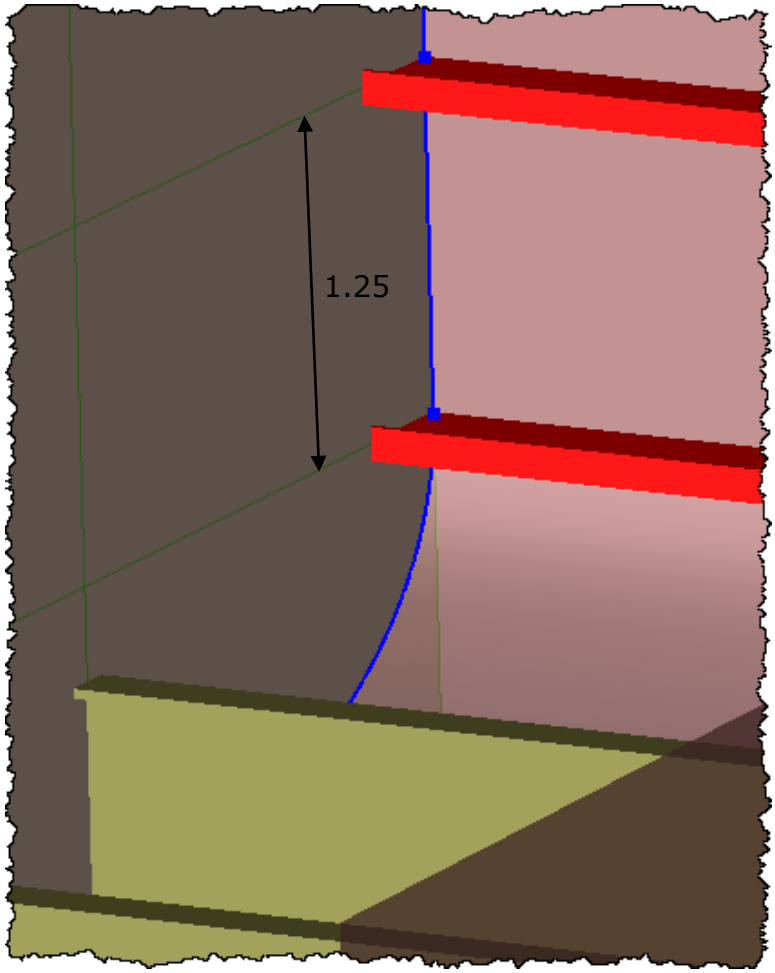


- Rotate the section to be perpendicular to the side surface and flush the beam with the surface.

- Copy the beam six times upwards a distance of 1.25 m, i.e. use the copy vector (0,0,1.25). The stiffeners should match the guide plane spacing.

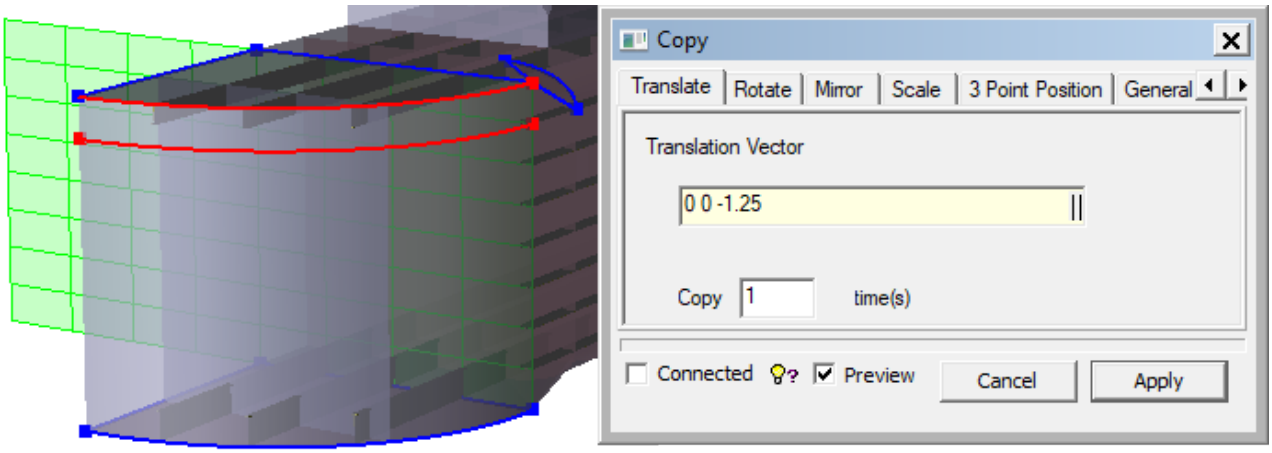
- The result of the two operation above is shown below. All seven stiffeners are highlighted in red.





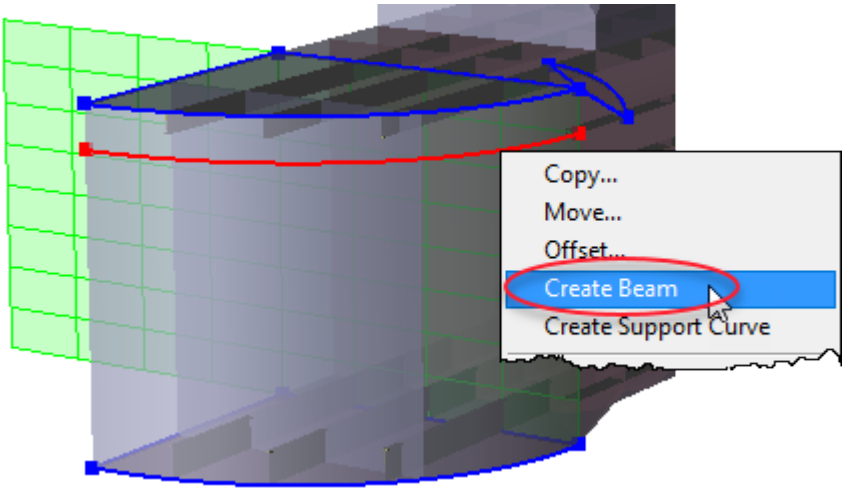
- Create horizontal stiffener beams in the cylindrical fore part of the pontoon.
  - The beam cross section is Tbar425x120x12x25, so keep this as default.

- Copy the guide curve at top of the cylindrical part downwards a distance of 1.25 m, i.e. using the copy vector (0,0,−1.25).



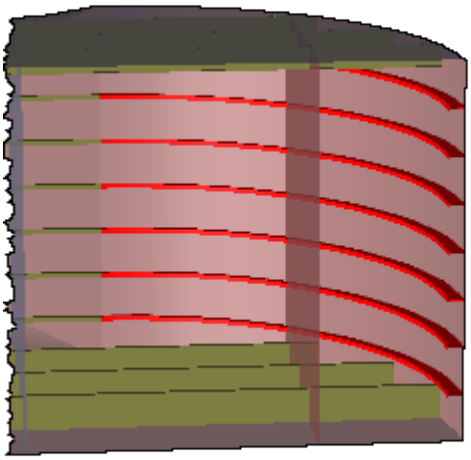
- Select the new guide curve, right-click and select *Create Beam*.

- Rotate the section to be perpendicular to the side surface and flush the beam with the surface.



- Copy the beam six times downwards a distance of 1.25 m, i.e. use the copy vector (0,0,−1.25). The stiffeners should match the guide plane spacing.

- The result of the operations above is shown to the right. All seven stiffeners are highlighted in red.



- Do a new *Model Verification*. Investigate and correct any problems.

- At this point you may switch to *Modelling - transparent* display configuration.

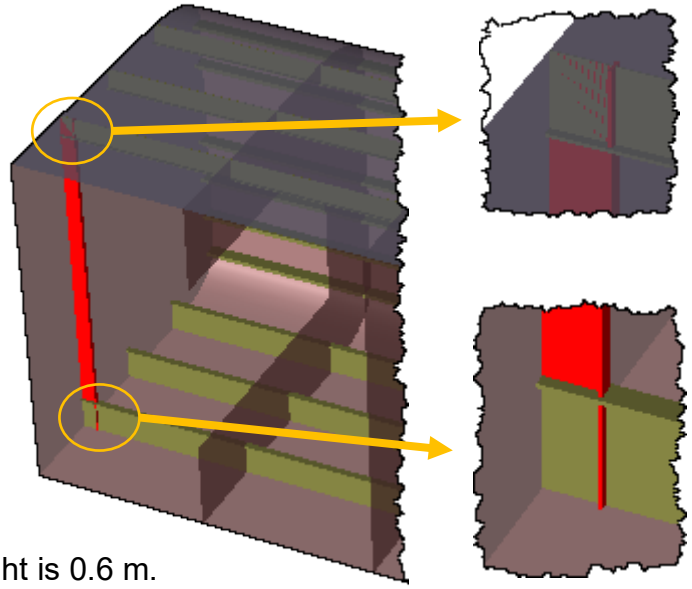
9 CREATE BULKHEAD STIFFENERS

➢ Create vertical stiffener beams for the bulkheads.

- The beam cross section is Tbar425x120x12x25, so keep this as default.

➢ Create first one of the stiffeners at X = 0 m. Orientate the section perpendicular to the bulkhead and flush it to the surface.

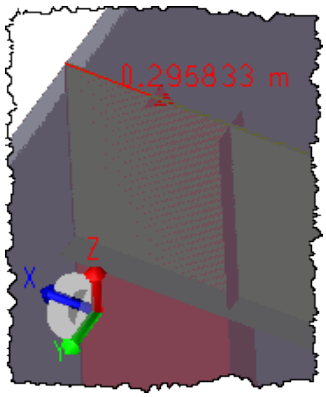
- As seen to the right, in the upper and lower ends, the stiffener overlaps the horizontal longitudinal stiffeners of the bottom and top surfaces of the pontoon.



- Therefore, add eccentricities (offsets) in the axial direction (vertical) in each end equal to the section heights of the corresponding horizontal longitudinal stiffeners.
  These section heights are found in the *Sections* folder in the browser. See below.
  - Upper end (Tbar575x150x12x25): Height is 0.6 m.
  - Lower end (Tbar885x200x14x35): Height is 0.92 m.

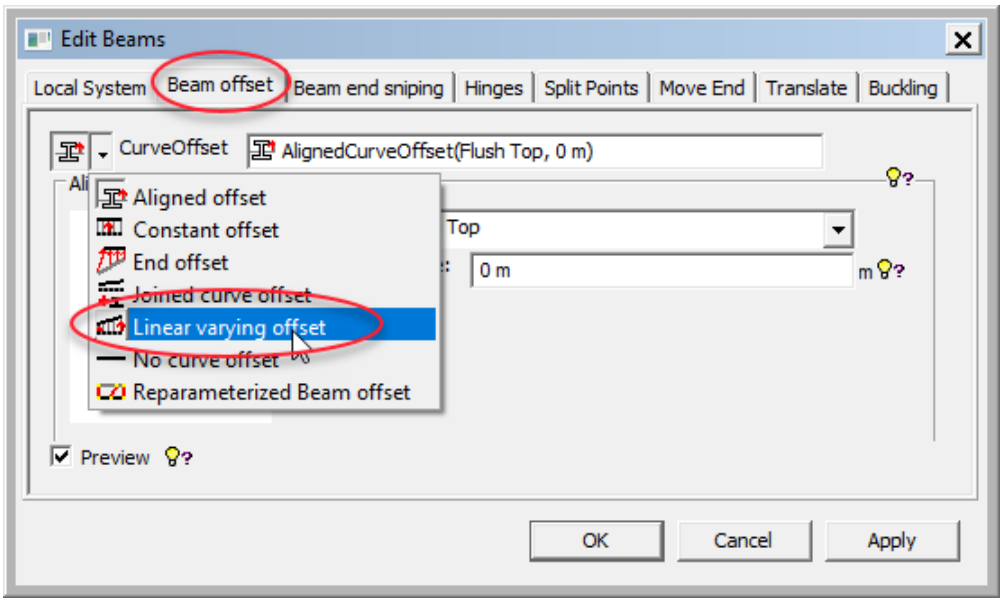
Name	Description	Type	Diameter [m]	Thickness [m]	Height [m]	W
Brace	Pipe Section, d=2 m, t=0.01 m	Pipe Section	2	0.01	2	2
Tbar425x120x12x25	TYPE 43 xx Tbar WeldedTbar	Unsym I Sect			0.45	0
Tbar575x150x12x25	TYPE 43 xx Tbar WeldedTbar	Unsym I Sect			0.6	0
Tbar885x200x14x35	TYPE 43 xx Tbar WeldedTbar	Unsym I Sect			0.92	0

- To combine flushing with axial eccentricities *Aligned offset* must be replaced by *Linear varying offset*. Therefore, make a note of the eccentricity calculated by the *Aligned offset* so that this can be manually added in combination with the axial eccentricities.
  - Right-click and select *Labels* | *Eccentricities* to see that the *Aligned offset* has yielded an eccentricity of 0.295833 in negative X-direction. This is constant along the beam.

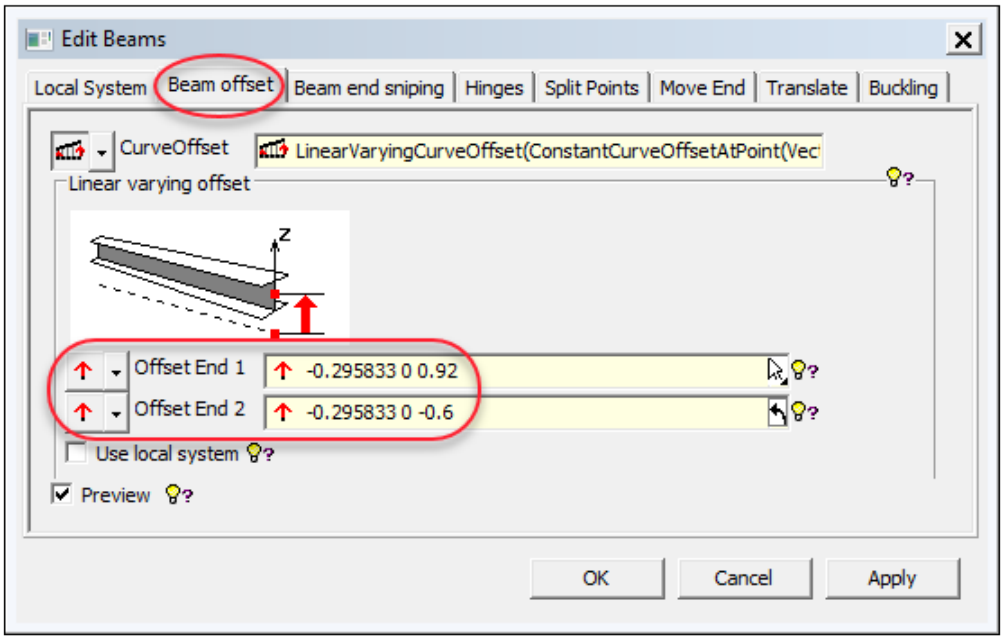


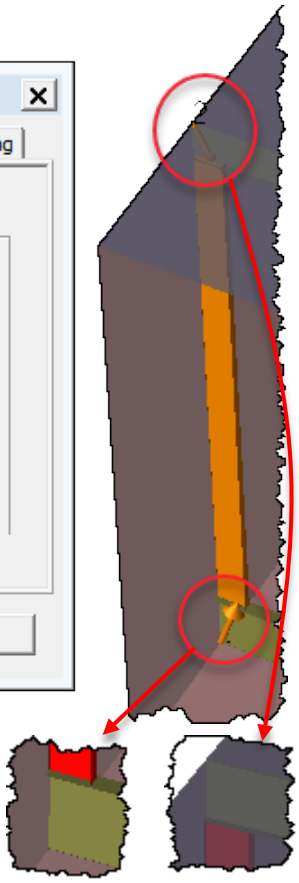
  - The eccentricity vectors in the upper and lower ends should therefore be:
    - Upper: (−0.295833,0,−0.6)
    - Lower: (−0.295833,0,0.92)

- Right-click to open the *Beam offset* tab of the *Edit Beams* dialog and change to *Linear varying offset* as shown below.



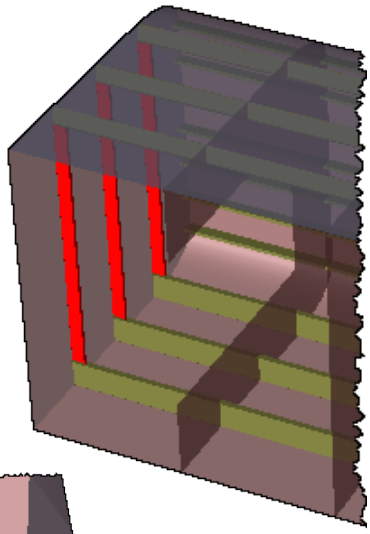
- Then fill in the eccentricity vectors as show below. Which end is 1 and which is 2 depends on how the beam was created. When entering the eccentricities the two ends are labelled (the numbers are a bit vague and may be difficult to see).



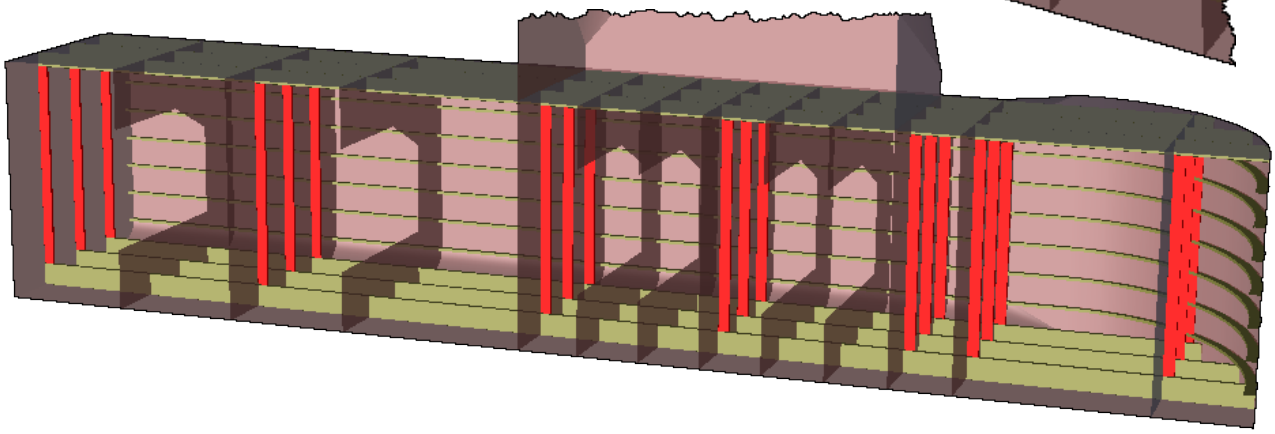


- Check that the offsets are as shown to the right.

- Copy the bulkhead stiffener twice so that there are three stiffeners for the bulkhead as shown to the right.

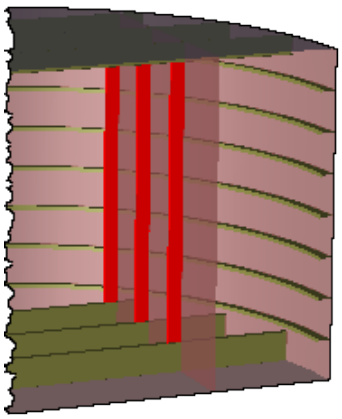


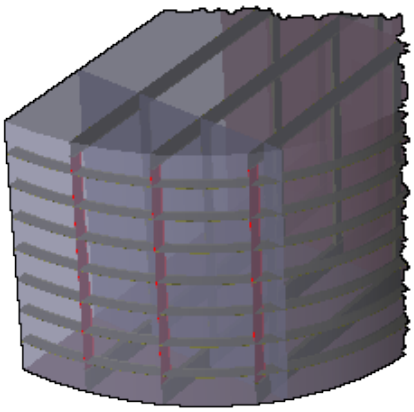
- Copy the three stiffeners six times so that all seven bulkhead have stiffeners as shown below.



- The flanges of the stiffeners at the bulkhead at X = −47.5 m (in the fore part) should point in the opposite direction. Select these and rotate them 180°, see the figure to the left below.
  - Doing so, the sign of the manually added flushing in X-direction must be inversed. I.e. change −0.295833 to 0.295833.

- Copy each of the three stiffeners at the bulkhead at X = −47.5 m to the cylindrical hull of the pontoon. The copy vectors will be different for each. See the figure to the right below.

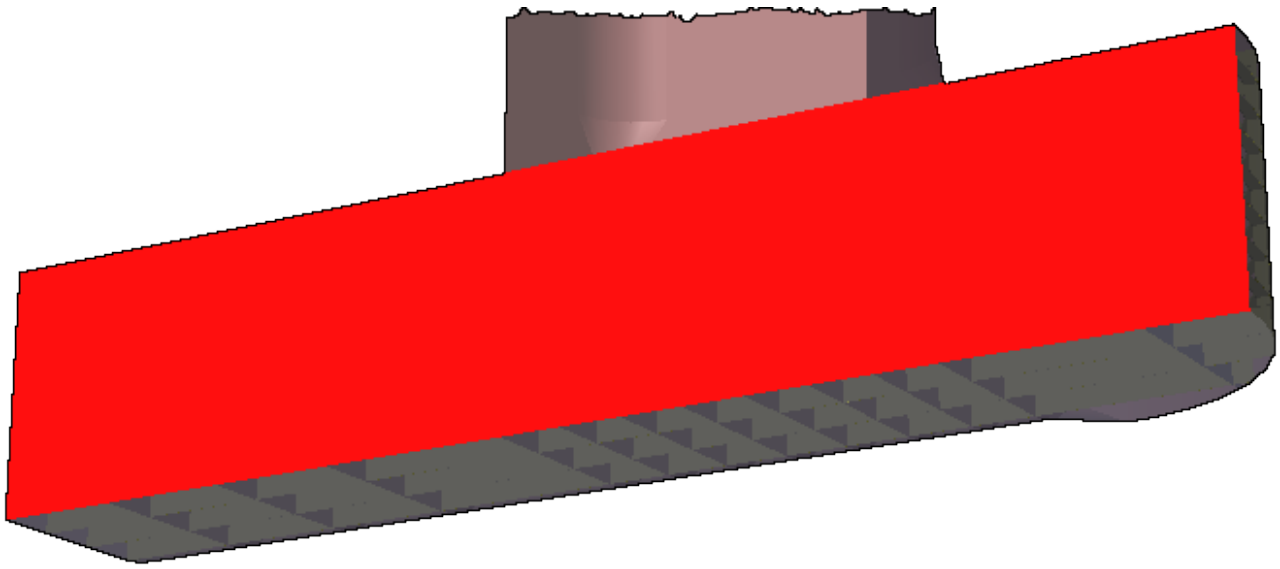




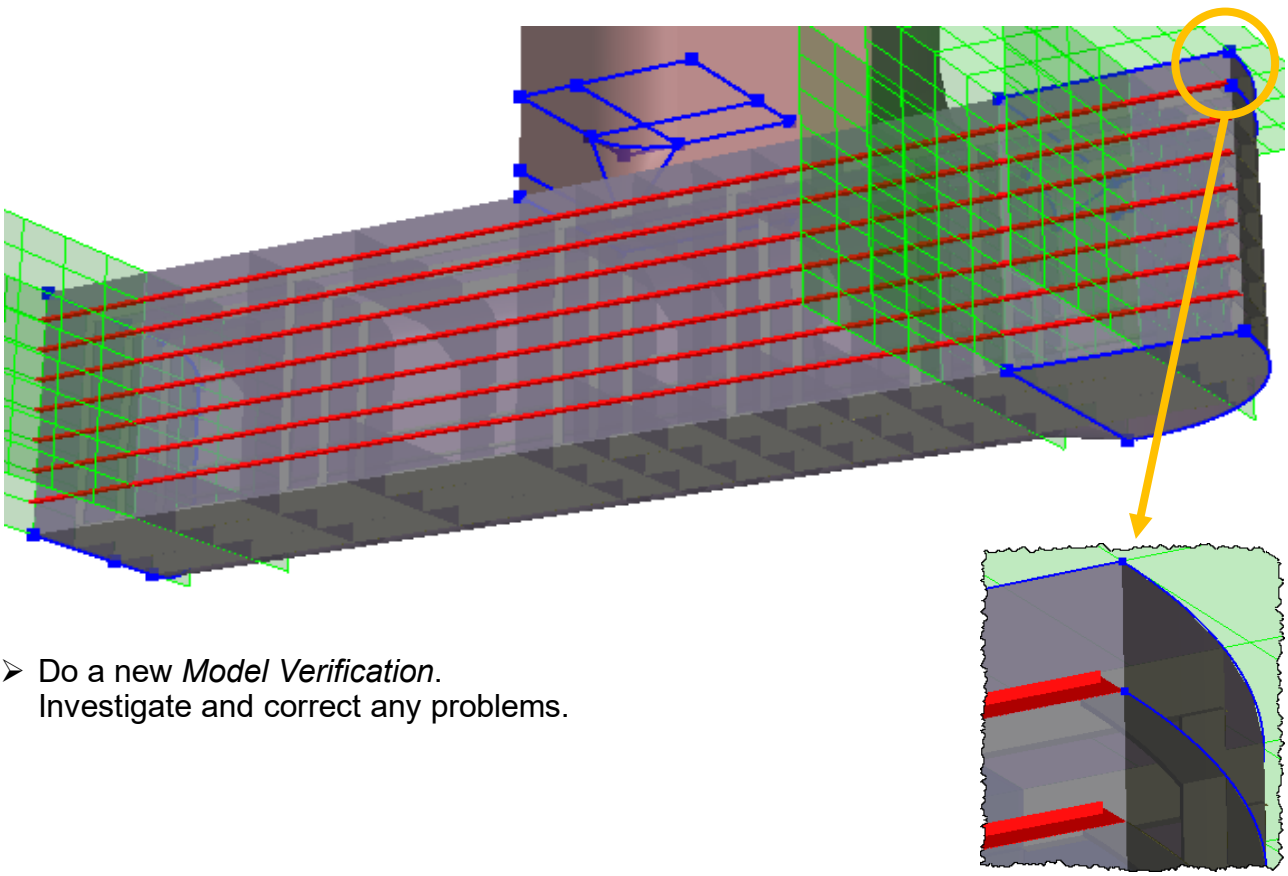
- Do a new *Model Verification*. Investigate and correct any problems.

10 CREATE LONGITUDINAL BULKHEAD

- Create a longitudinal bulkhead with thickness to 15 mm (Tck15) in the whole length of the pontoon. The bulkhead is highlighted below.



- Create longitudinal stiffener beams with cross section Tbar425x120x12x25 for the longitudinal bulkhead as shown below. Orientate the beam axis systems properly and flush them to the side of plate.
  - To ease the creation switch to *Default display* configuration.



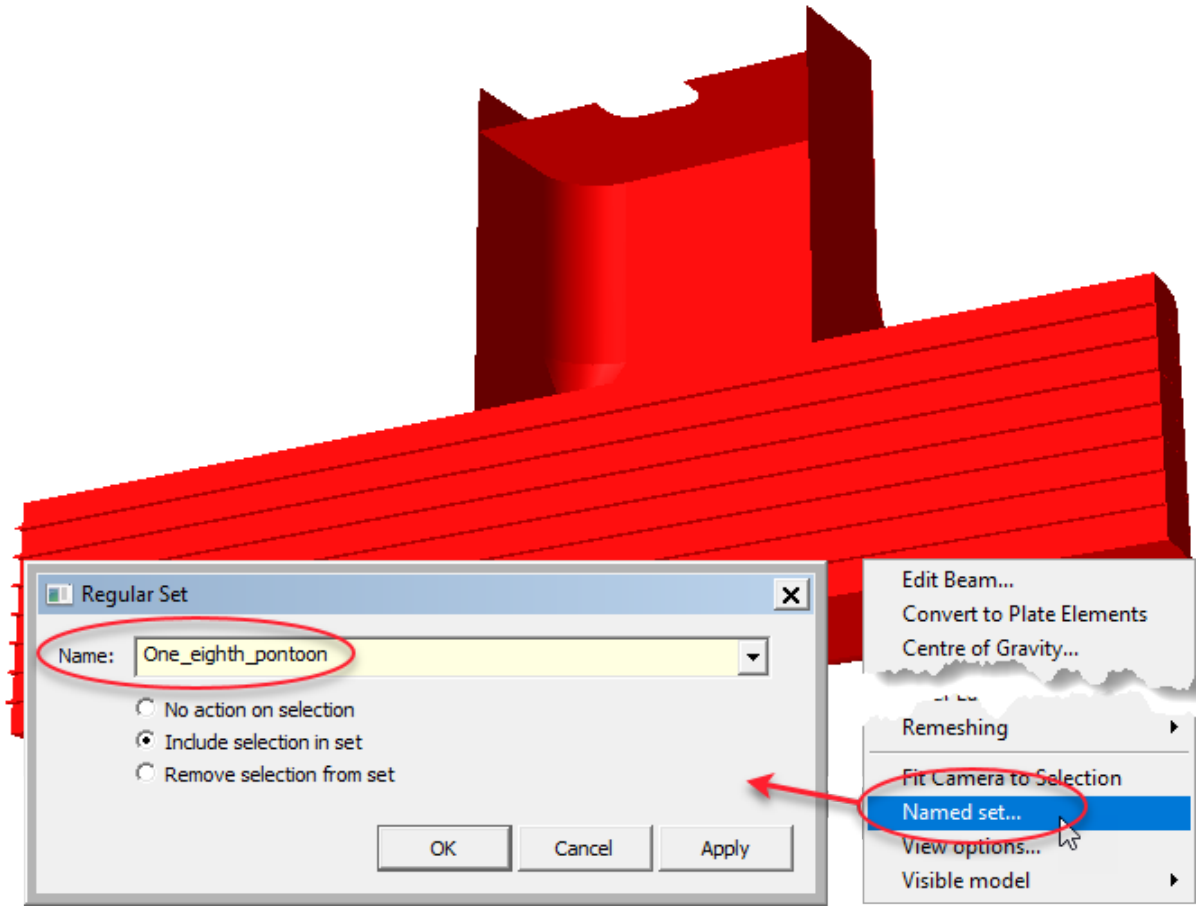
- Do a new *Model Verification*.
  Investigate and correct any problems.

11 CREATE SETS

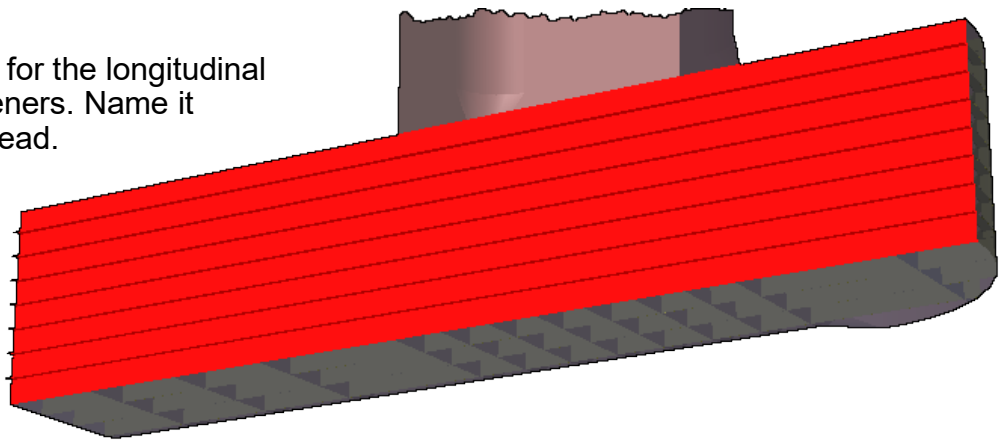
- Create sets for later retrieval, both in GeniE and other programs.

- Switch to *Modelling - transparent* display configuration.

- Select the whole model created so far, right-click to select *Named set* and put the selection into a set named One_eighth_pontoon as shown below.



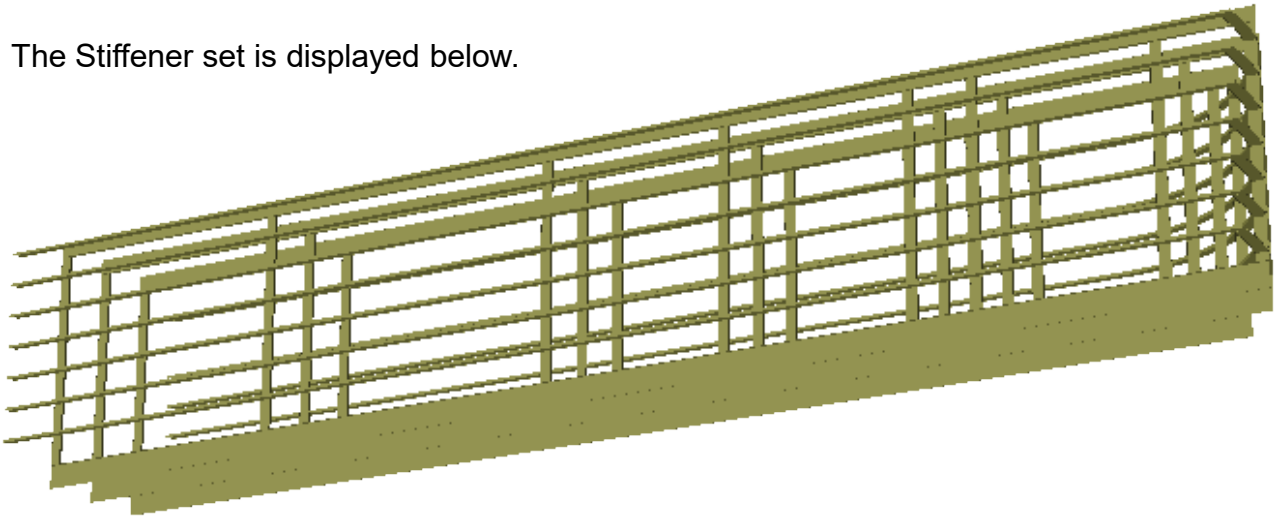
- Create another set for the longitudinal bulkhead with stiffeners. Name it Longitudinal_bulkhead.



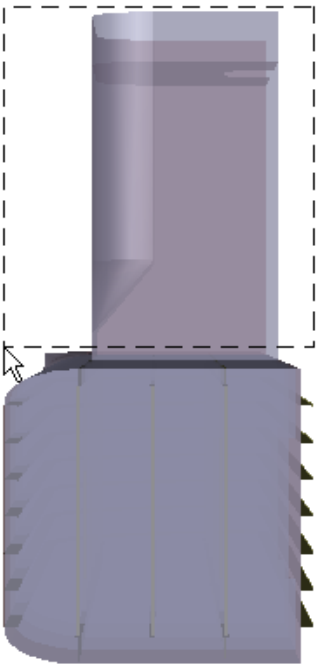
- Find the sets in the browser in the *Utilities* | *Sets* | *Regular Sets* folder. They may be selected followed by Alt+S to show only the set and Alt+minus to remove the set from the display. Alt+A shows all.

- Create a set named Stiffeners for all stiffener beams in the model. There are alternative ways of selecting all beams in the model:
  - Lift the *Plate selection* button and drag a rubberband around the whole model.
  - Right-click the *Plate selection* button and make plates invisible by closing the eye symbol followed by dragging a rubberband around all beams.
  - In the *Properties* | *Sections* folder in the browser select all sections, right-click and press *Select Objects*.
  - In the *Structure* folder in the browser click the *Name* or *Description* header to sort the objects and select by click and Shift+click.

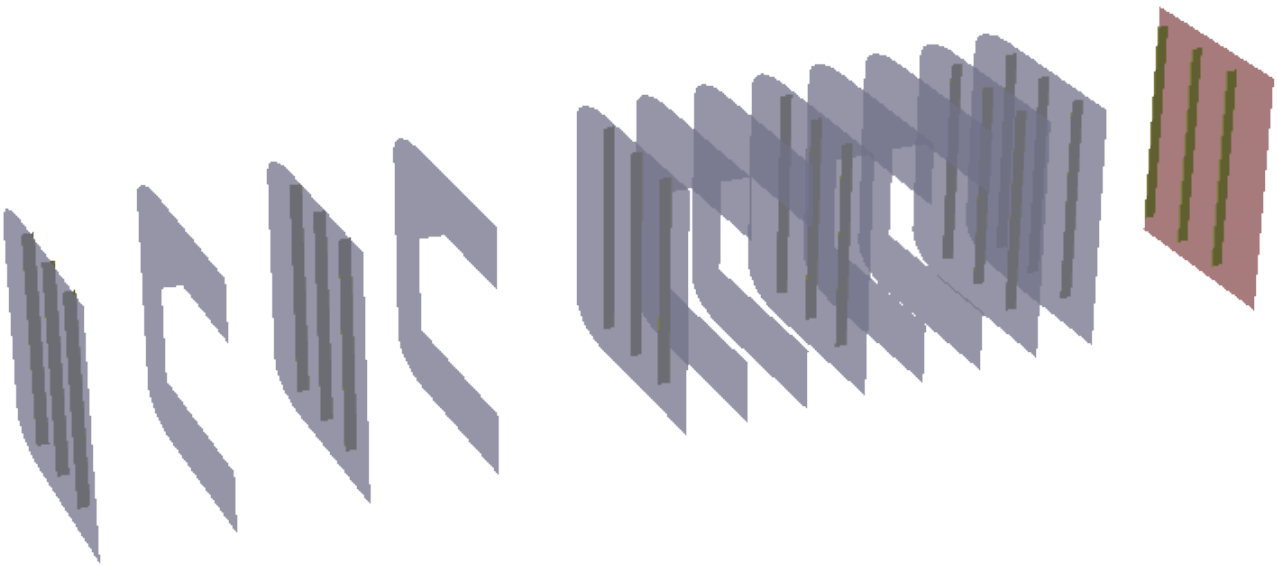
- The Stiffener set is displayed below.



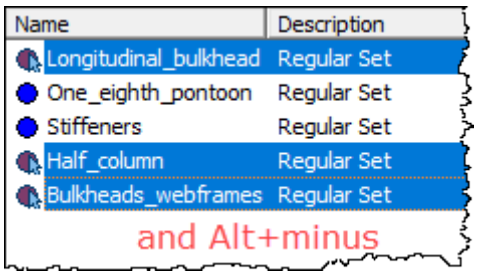
- Create a set named Half_column for the column part. Select it for instance by viewing the model in the X-direction (F6) and dragging a rubberband from right to left as shown to the right.
  - Provided selection mode is set to *Enclosed or touched by rubberband*, dragging from right to left is touched selection and dragging from left to right is enclosed selection.

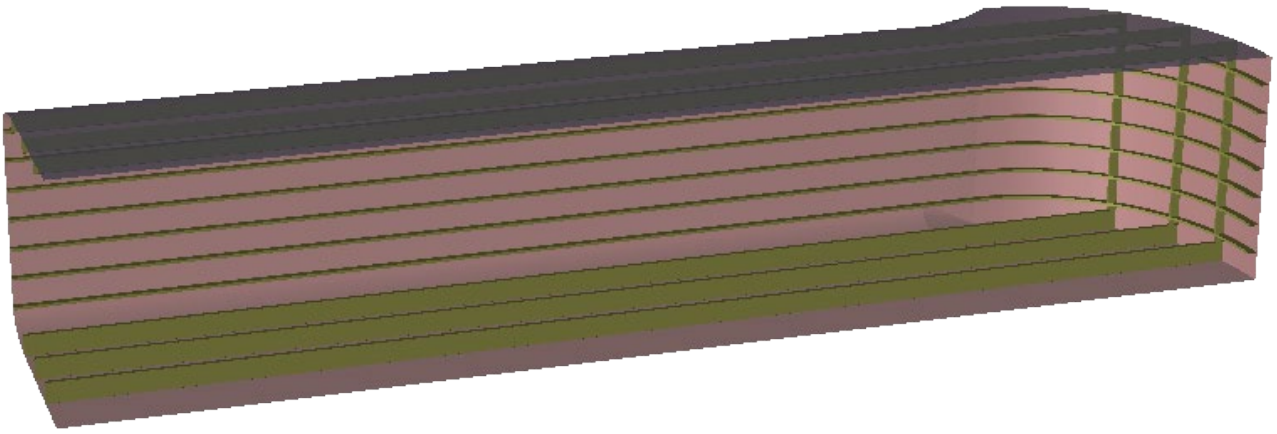


- Create a set named Bulkheads_webframes containing all bulkheads and web frames with stiffeners as shown below.



- Create a set named Outer_hull containing the hull with stiffeners as shown below.
  - The selection may be established by first showing all (Alt+A). Then selecting the sets Longitudinal_bulkhead, Half_column and Bulkheads_webframes in the browser (*Utility* | *Sets* | *Regular Sets*) and using Alt+minus to remove these from the display. Finally, drag a rubberband around the remainder and put it into a set named Outer_hull.



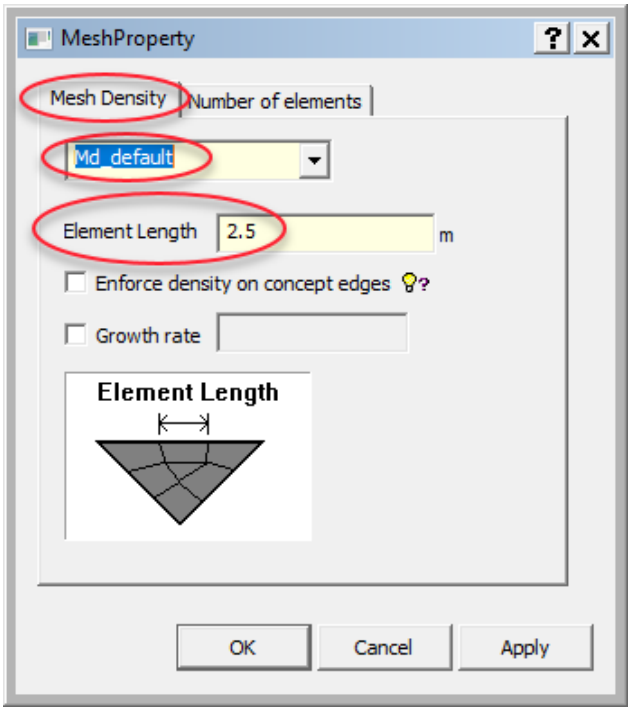


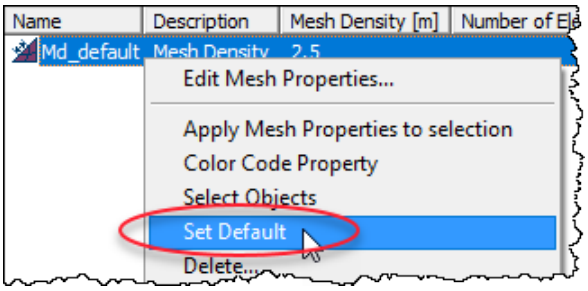
- You may create more sets. Sets facilitate selection and is useful in the further work in GeniE as well as in subsequent programs like the postprocessor Xtract.

12 CREATE A FE MESH

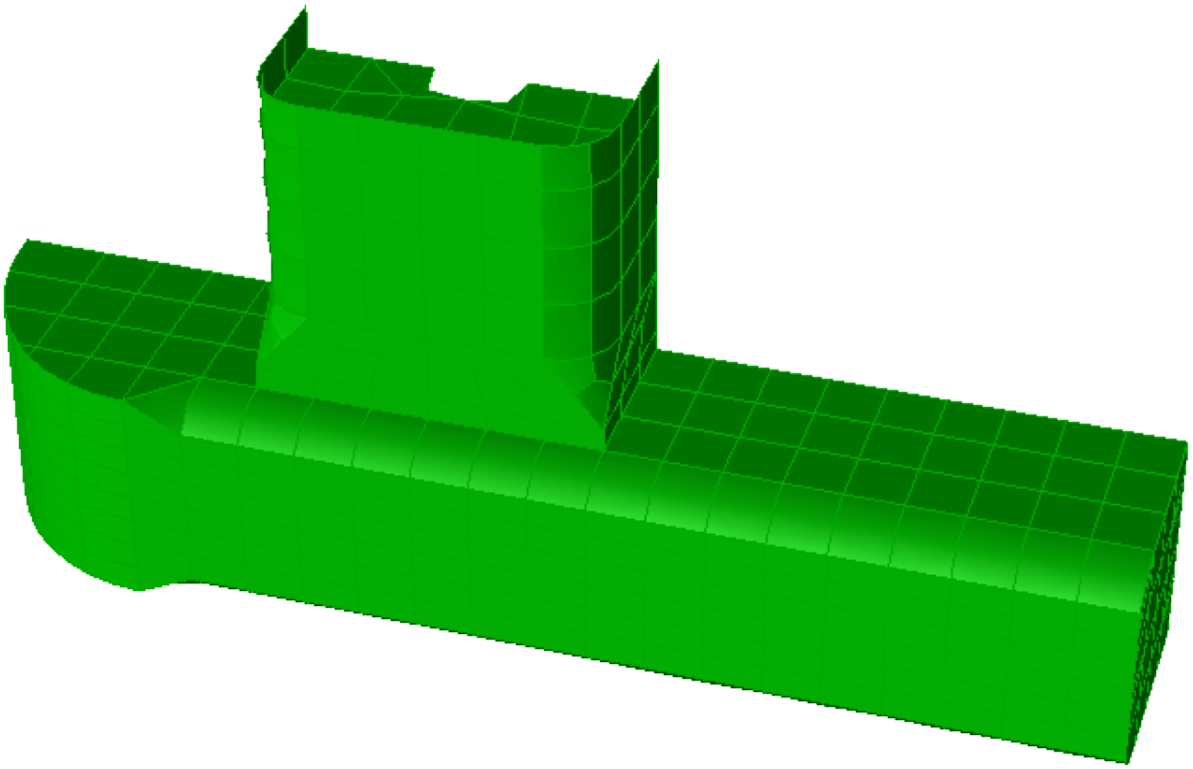
➢ Create a FE mesh for the one eighth part of the complete semisubmersible using a mesh density of 2.5 m.

- Right-click the *Properties* | *Mesh* folder in the browser and select *New Mesh Property*.
- Define a mesh property named Md_default with *Element Length* 2.5 m as shown to the right.
- Right-click the property Md_default found in the browser and click *Set Default* as shown below.

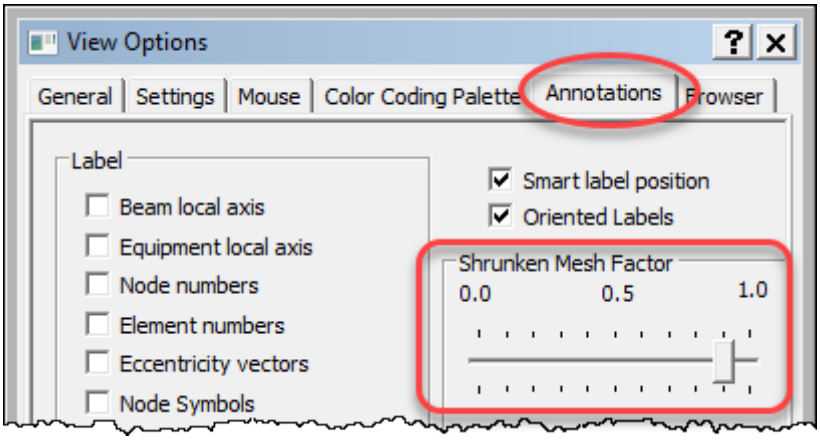




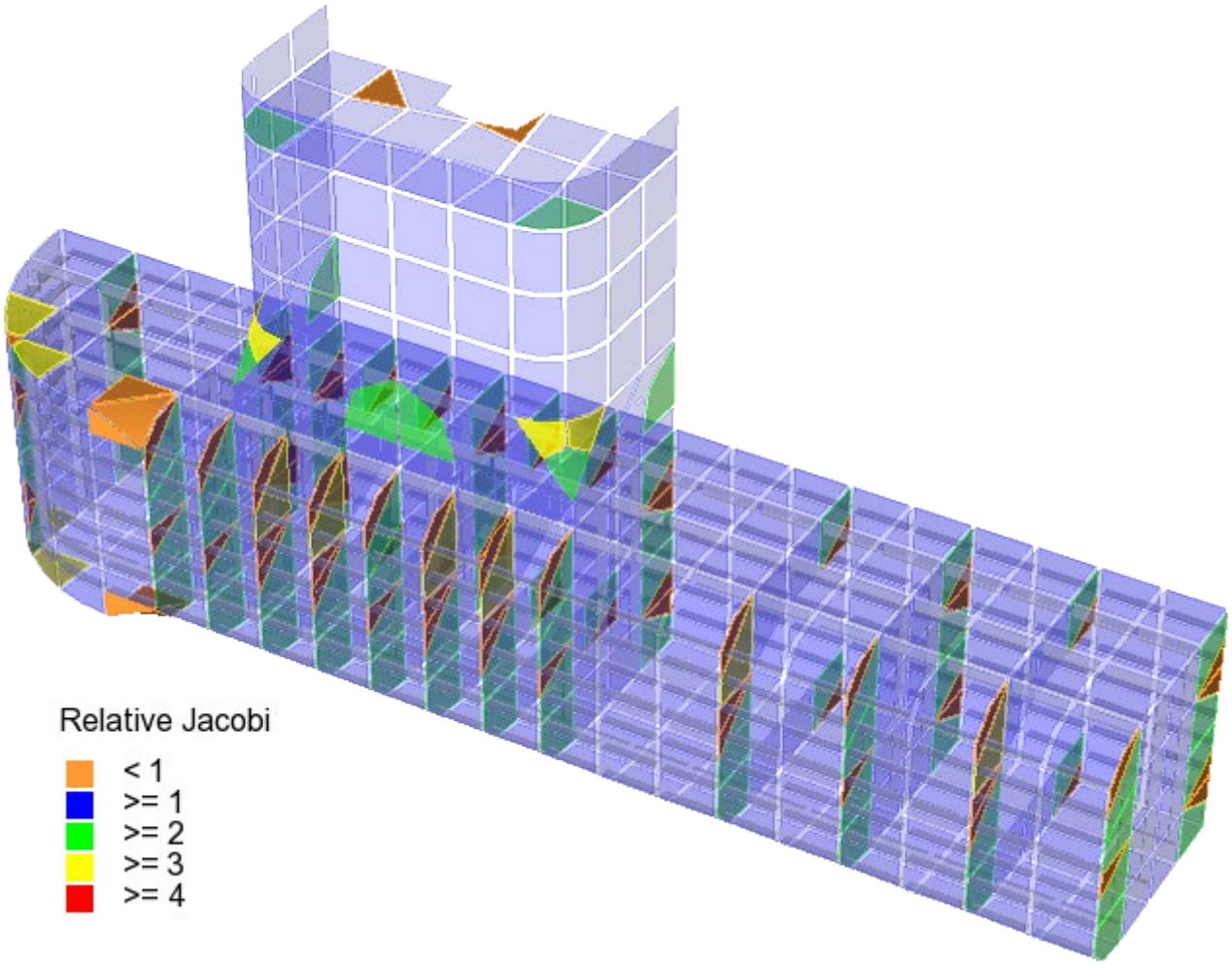
- Use *Mesh & Analysis* | *Create Mesh* (or Alt+M) to open the *Create Activity* dialog. Do not select any of the *Available activities* in the dialog. Click *OK* to open the *Activity Monitor*.
- Click *Start* in the *Activity Monitor* to create the mesh.
- Switch to *Mesh - All* display configuration to see the mesh.



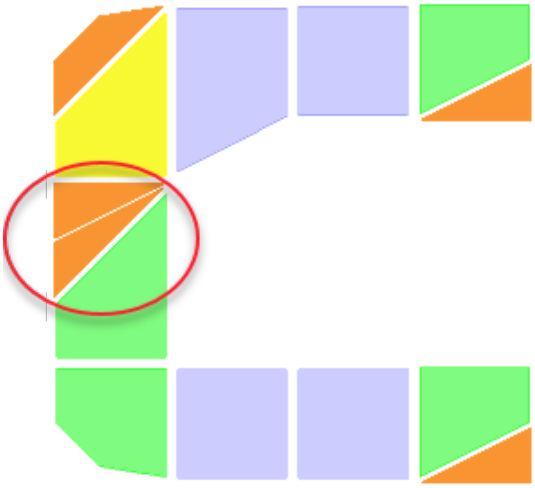
- *View* | *Options* | *Annotations* opens the *Annotations* tab of the *View Options* dialog. Set *Shrunken Mesh Factor* to 0.9.

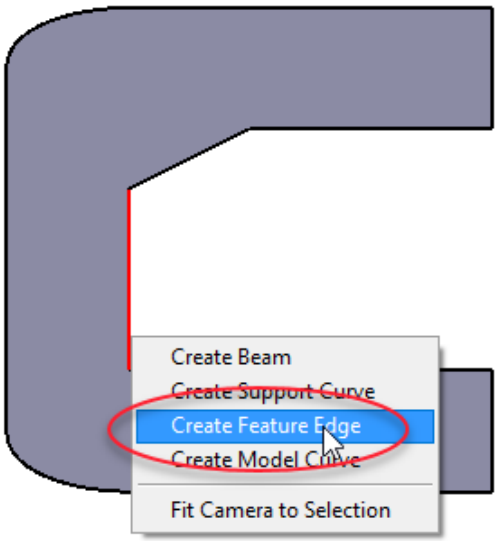


- Select the whole model (drag a rubberband around the mesh), right-click and select *ColorCode* | *Mesh* | *Relative Jacobi*. The display below appears. The colours orange, yellow and red identify suboptimal elements. Some of these are difficult to avoid and some may be improved by various means.

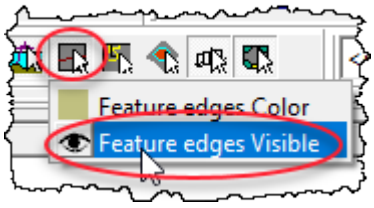


- The meshes of the web frames are suboptimal with two triangles next to each other. Select one of the web frames, switch off colour coding and switch to *Modelling - Transparent* display configuration, double-click it, select the edge and right-click to *Create Feature Edge* as shown below.

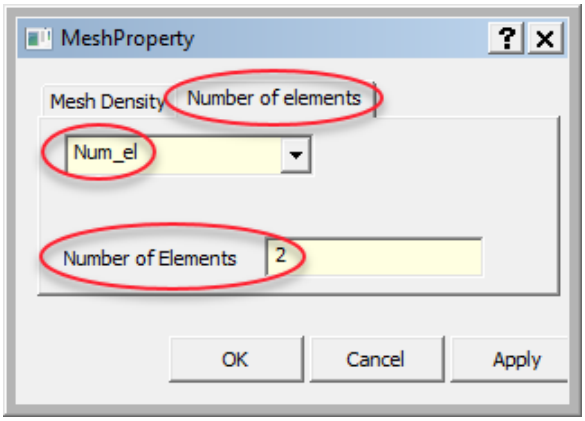




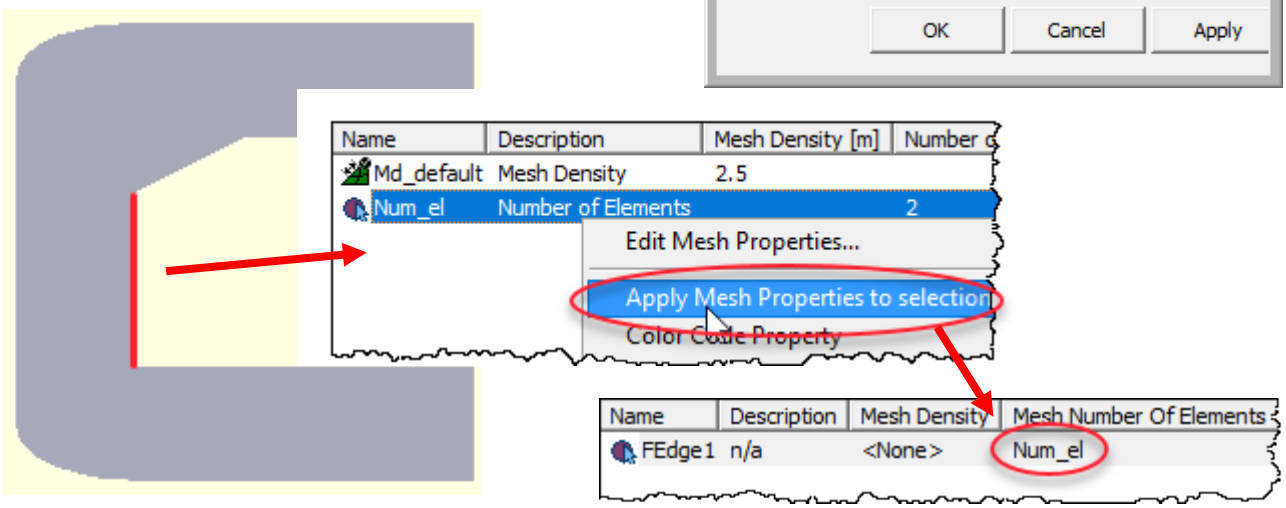
- Double-click outside the model to return to normal display mode. To see the feature edge, right-click the *Feature edge selection* button and open the eye symbol. Also press the button down to enable selecting feature edges. Select the feature edge in the display or in the *Structure* | *Features* folder in the browser (the latter may be easier).



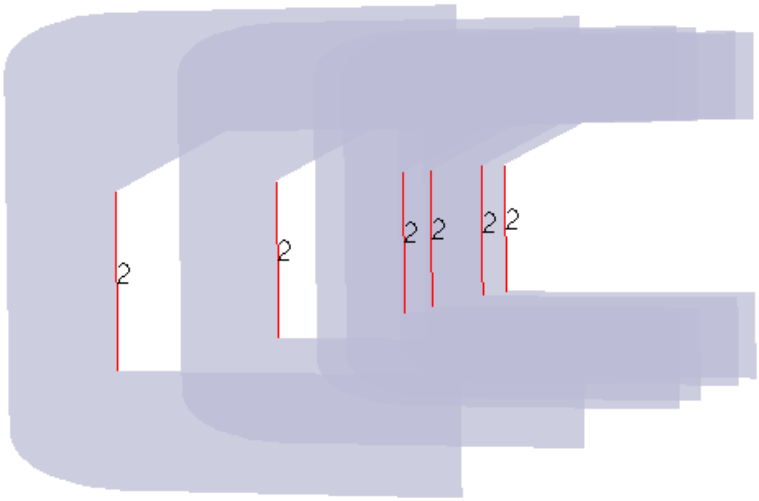
- Right-click the *Properties* | *Mesh* folder to create a new mesh property named Num_el with *Number of Elements* set to 2.



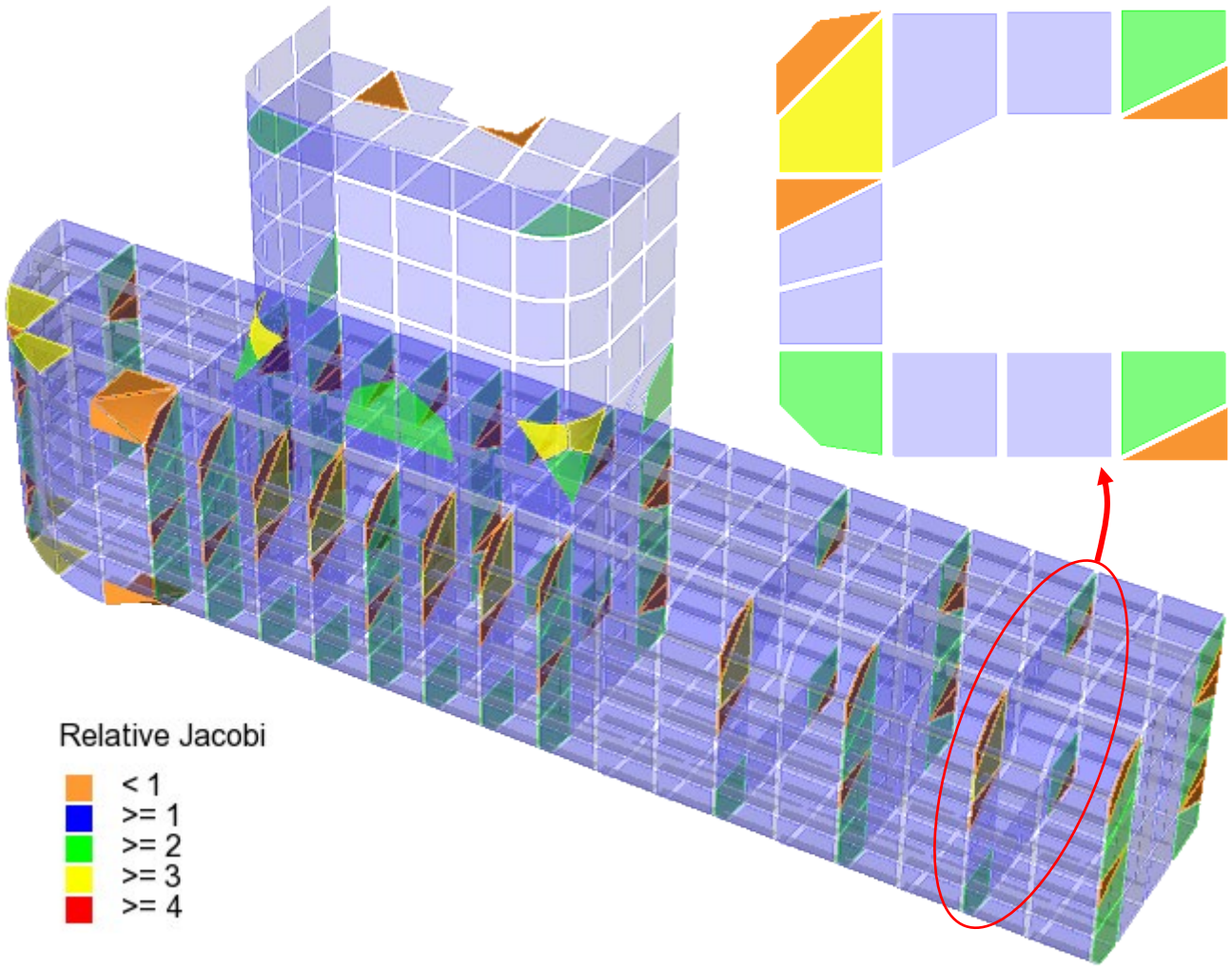
- With the feature edge selected right-click the new mesh property and apply it.



- Copy the featured edge to all web frames. Having done so, select all feature edges, right-click and select *Labels* | *Property values* | *Number of Elements* and see that the assignment of mesh property follows the copy.

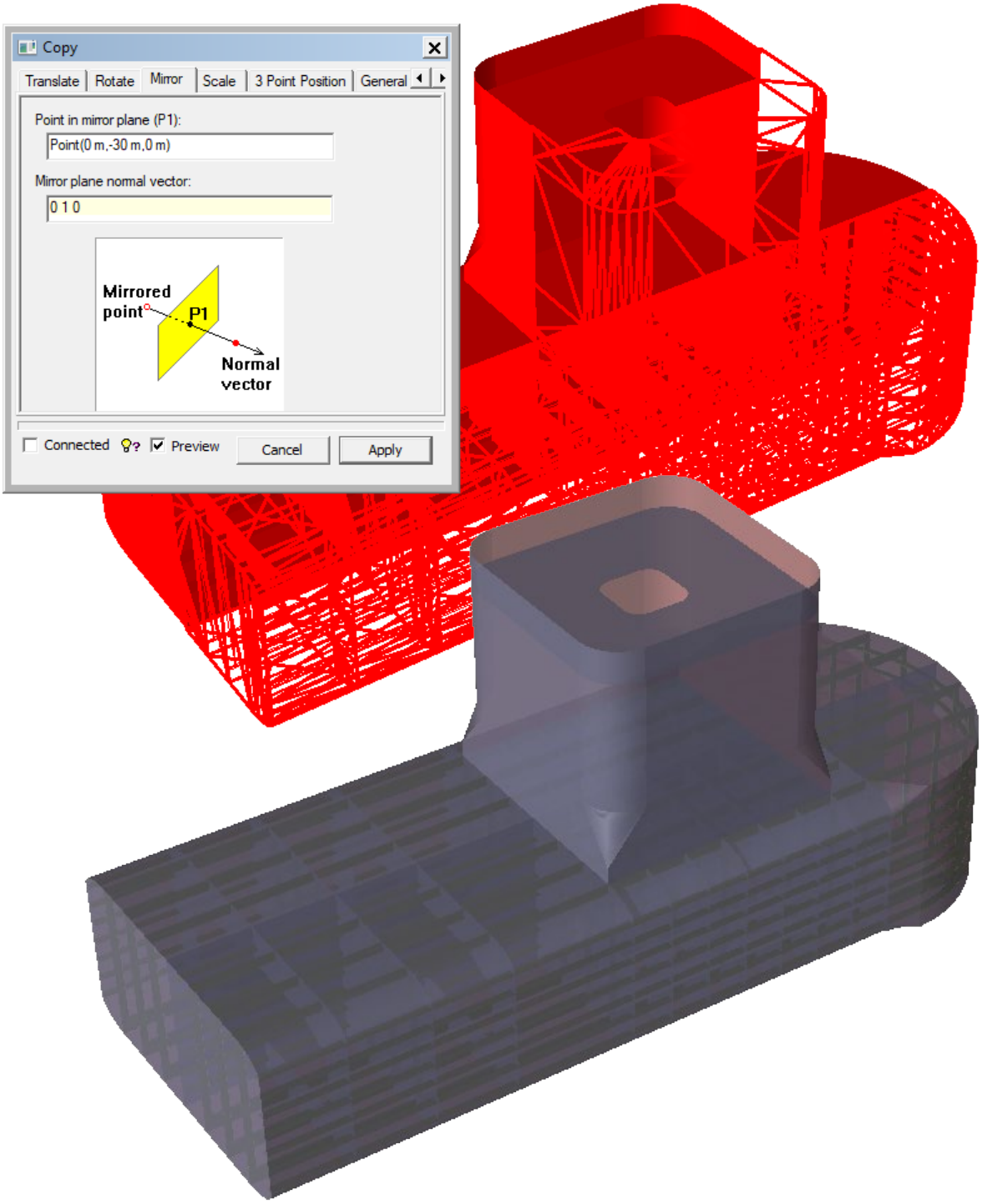


- Create a new mesh (Alt+D and *Start*) and see that the mesh of the web frames is improved.



13 CREATE HALF A PONTOON

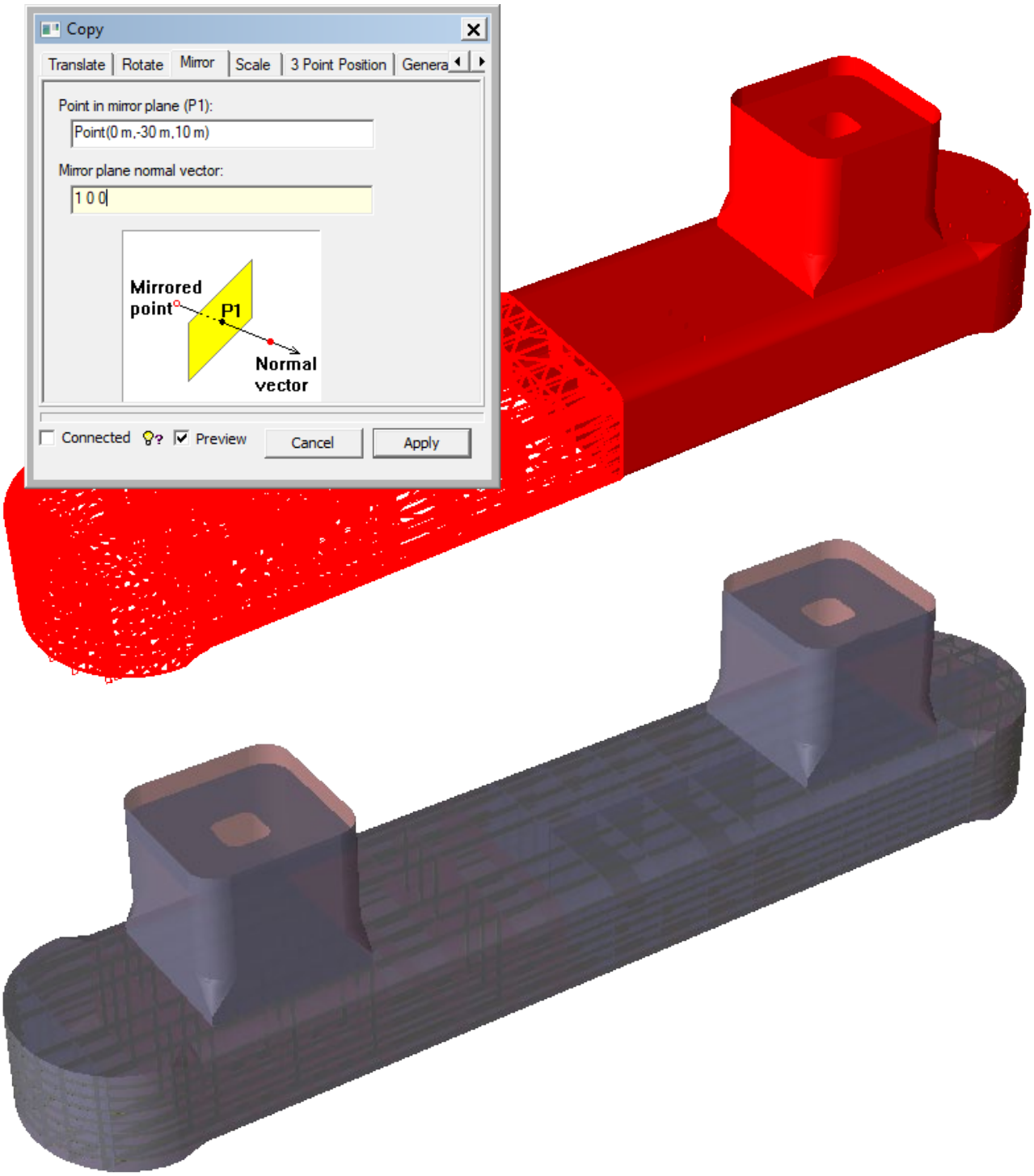
- Create half a pontoon by copying the model by mirroring.
  - The longitudinal bulkhead with stiffeners should not be copied as this is in the mirror plane.
  - Select the set Longitudinal_bulkhead and press Alt+minus to remove it from the display. (An attempt at copying also the bulkhead would cause a message saying it cannot be copied since plates cannot overlap.)
  - After the copying remember to use Alt+A to include the set Longitudinal_bulkhead in the display or else further copying may exclude this part.



14 CREATE A COMPLETE PONTOON

➢ Create a complete pontoon by copying the model by mirroring.

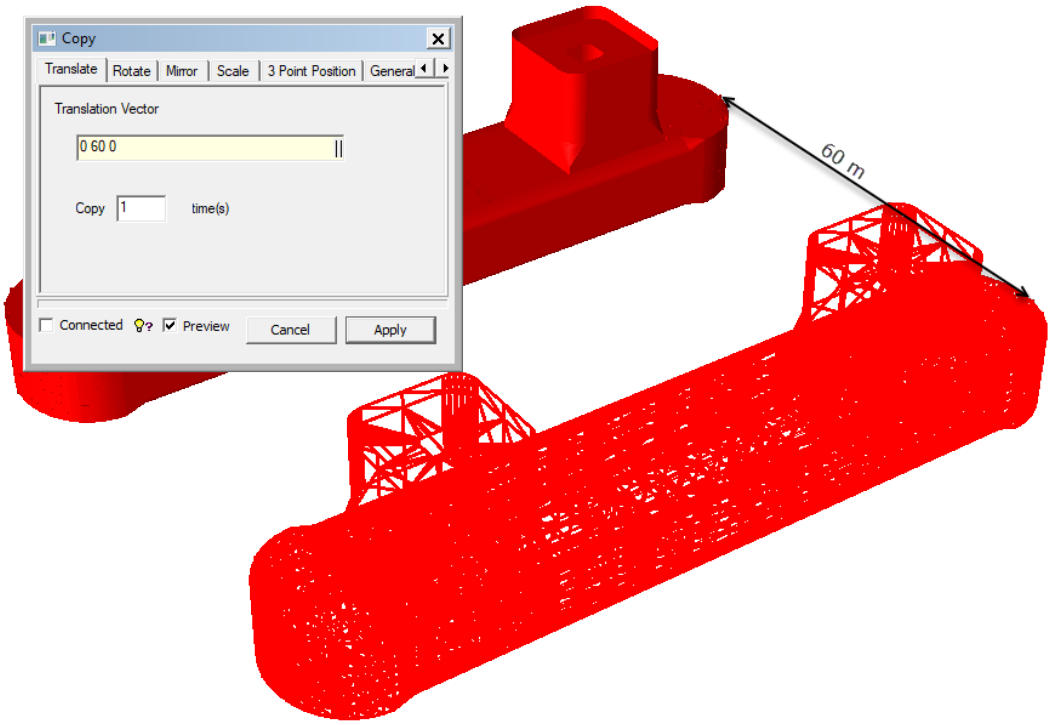
- The transverse bulkhead at X = 0 m with stiffeners should not be copied so remove this from the display before selecting and copying.
- After the copying remember to use Alt+A to include the transverse bulkhead at X = 0 in the display or else further copying may exclude this part.



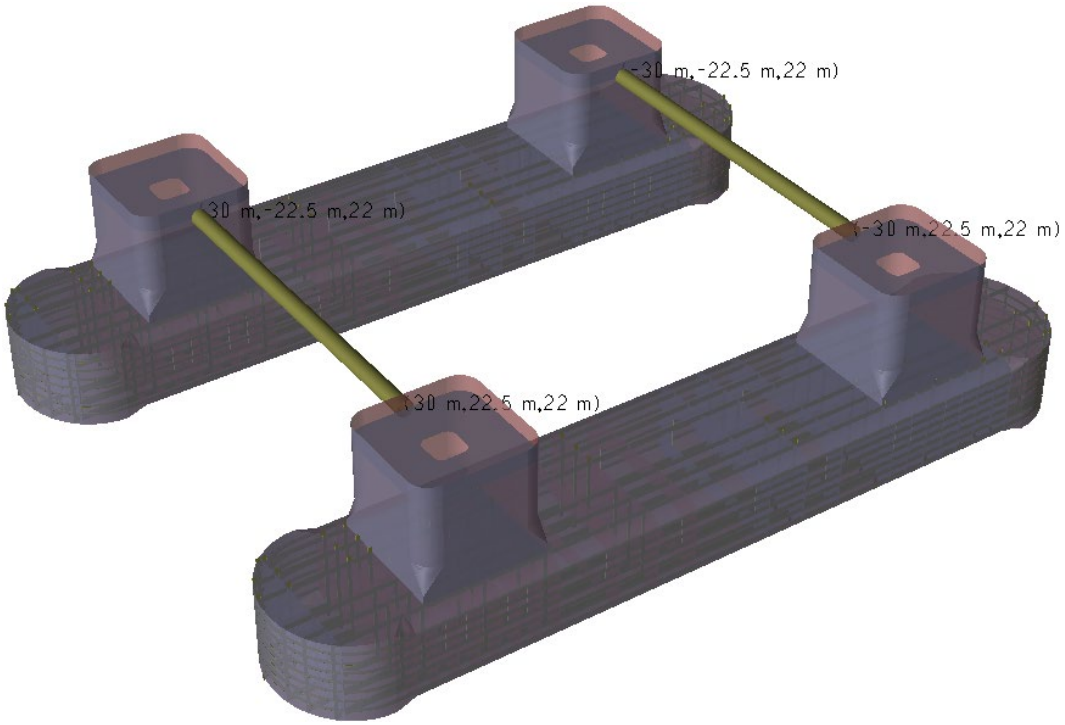
- Do a new *Model Verification*. Investigate and correct any problems.

15 CREATE THE COMPLETE SEMISUBMERSIBLE MODEL

- Create the other pontoon by copying 60 m in Y-direction.



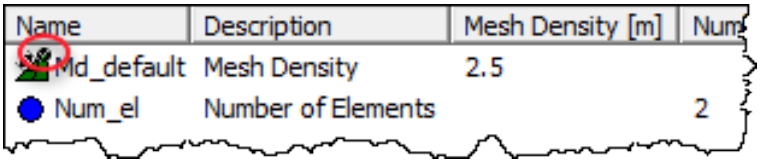
- After the copying, create the two braces with beam section Brace. The end coordinates of the braces are labelled (right-click and *Labels* | *Coordinates*).



- Do a new *Model Verification*. Investigate and correct any problems.

16 CREATE FE MESH FOR THE COMPLETE MODEL

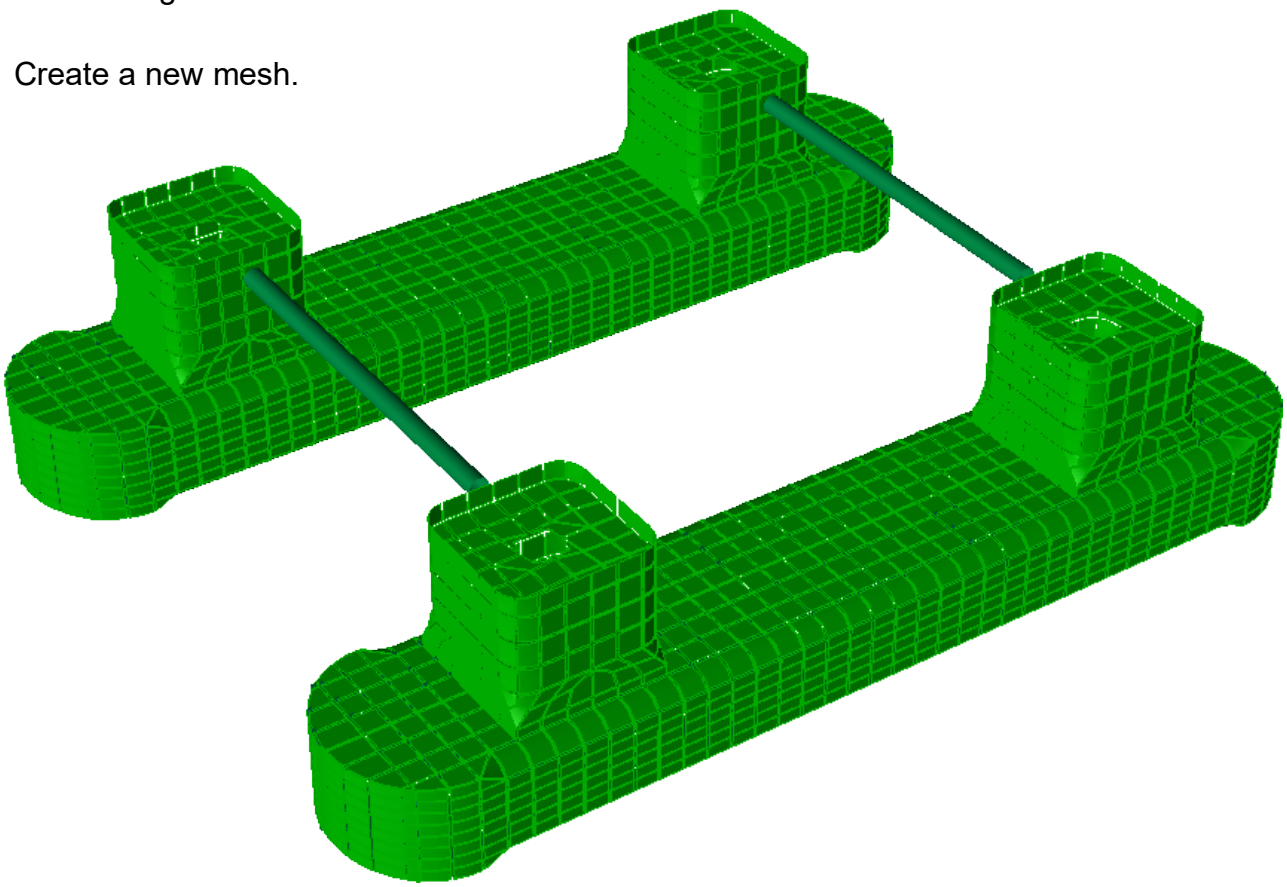
- Md_default is currently the default mesh property. This can be seen by the small check mark in the browser as shown below.



- This means that this property is assigned to all plates, shells and beams for which there are no mesh property specifically assigned.

- However, there is no point in subdividing the braces in more than single beam elements. Therefore, create a new mesh property named Single_elem with number of elements set to 1 and assign this to the two braces.

- Create a new mesh.



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.

DNV | digital@dnv.com | dnv.com/digital