SESAM TUTORIAL

GeniE

Semisubmersible Panel and Structural (FE) Modelling

Valid from program version 8.2

Sesam Tutorial

GeniE – Semisubmersible Panel and Structural (FE) Modelling

Date: August 2021

Valid from GeniE version 8.2

Prepared by: Digital Solutions at DNV

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

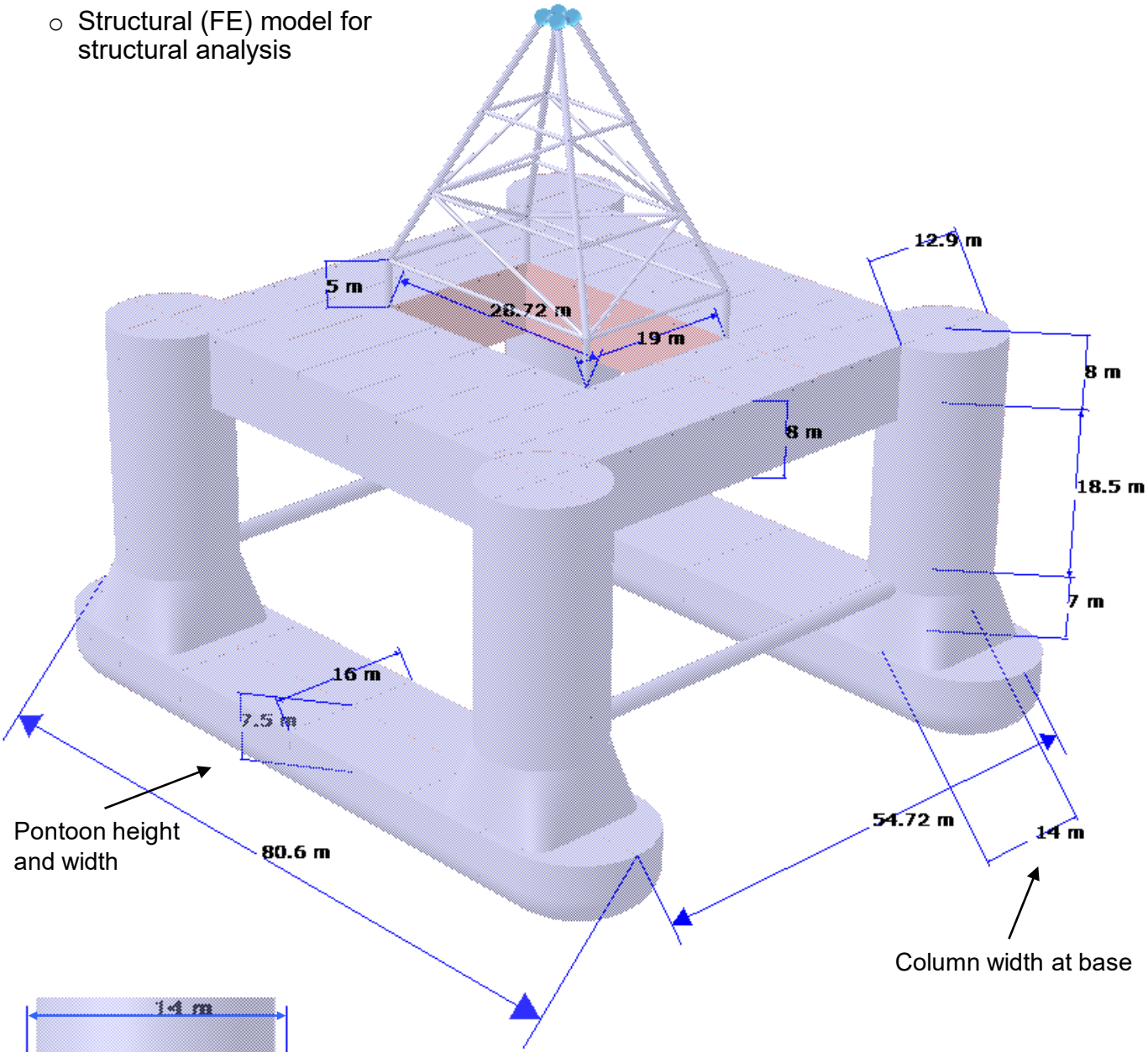
E-mail sales: digital@dvn.com

TABLE OF CONTENTS

1. Introduction Page 4
2. Units, Materials, Cross Sections, Plate Thicknesses Page 5
3. Create Guiding Geometry for Pontoon Page 9
4. Create Pontoon Page 14
5. Create Column Page 18
6. Create Panel Model (T1.FEM) Page 25
7. Create Full Model Page 29
8. Create Morison Model (T2.FEM) Page 33
9. Create Derrick Page 34
10. Create Compartments Page 35
11. Create Equipments Page 38
12. Add Support Points Page 40
13. Create FE Structural Model (T3.FEM) Page 41

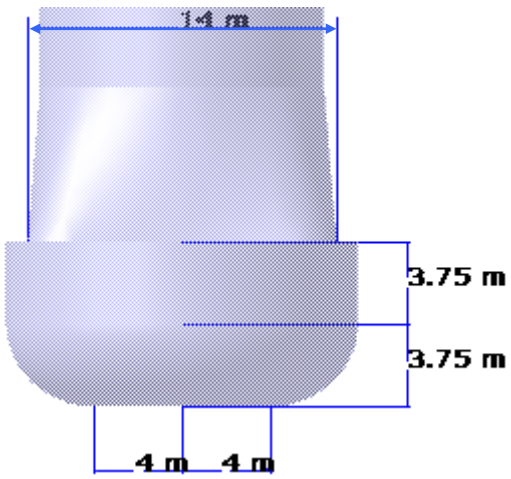
1 INTRODUCTION

- In this tutorial a model of a semisubmersible is created and based on this the following analysis models are created:
  - Panel model and Morison model for hydrostatic and hydrodynamic analysis
  - Structural (FE) model for structural analysis



Pontoon height and width

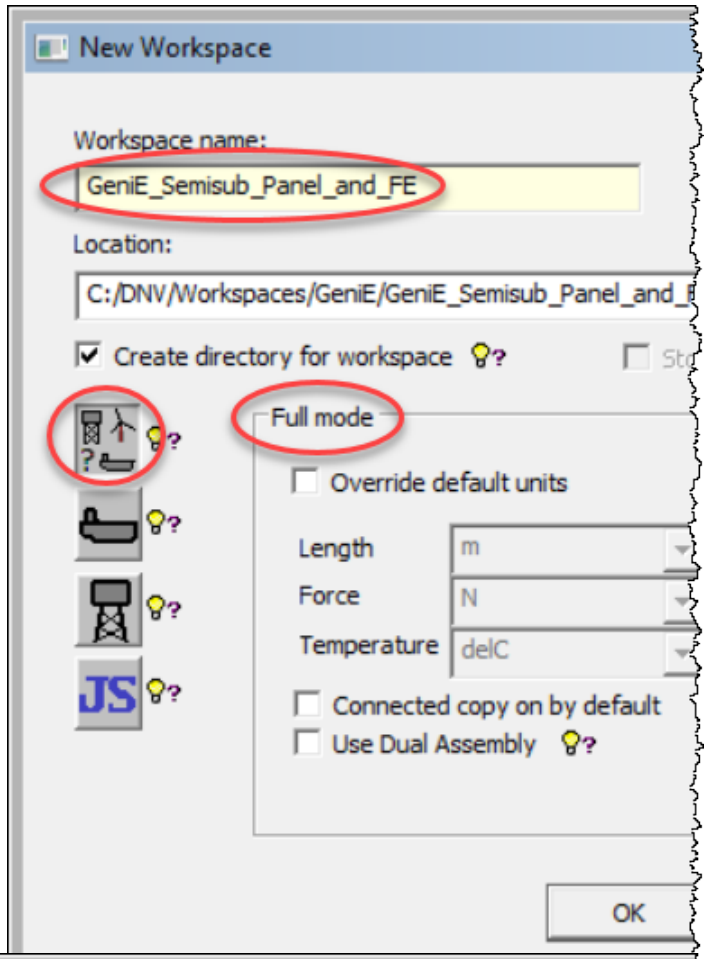
Column width at base



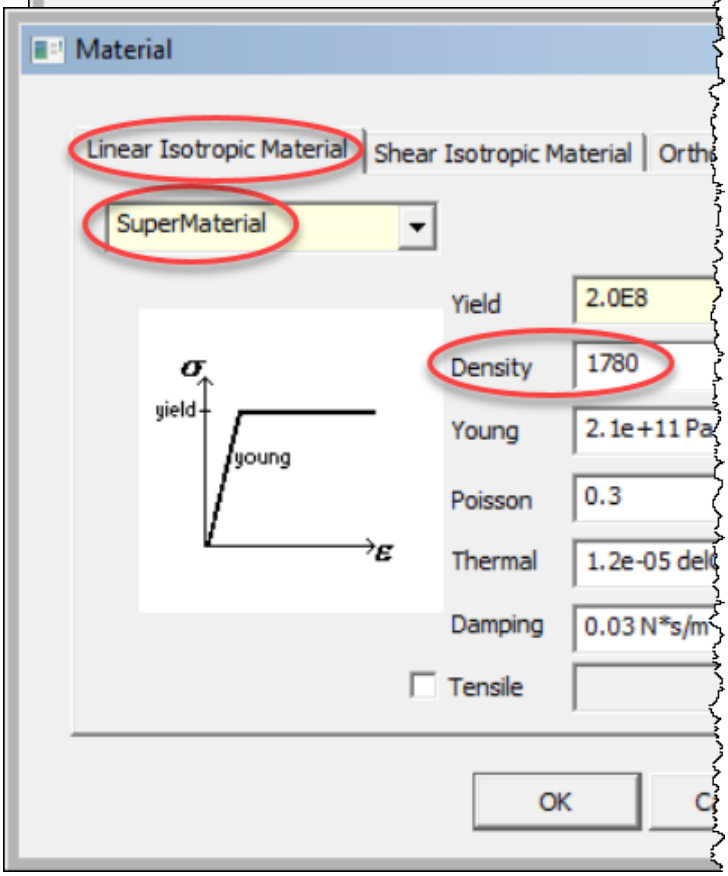
Pontoon cross section

2 UNITS, MATERIALS, CROSS SECTIONS, PLATE THICKNESSES

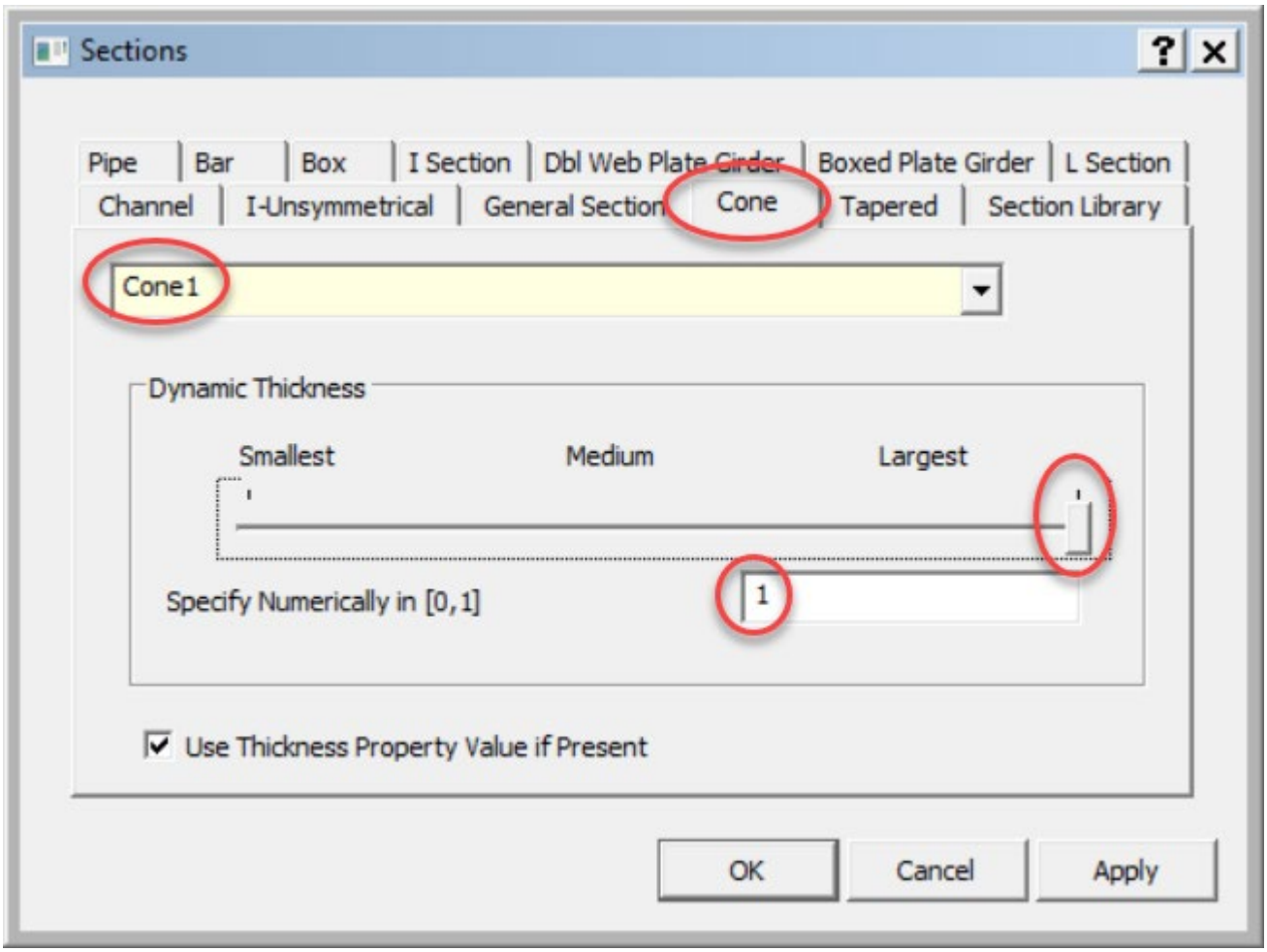
- Start GeniE and open a new workspace.
  - Give a *Workspace name*.
  - Accept default units m and N and click *OK*.
    - Unless otherwise specified, all values in this tutorial are in these units.
  - Make sure *Full mode* is selected as this tutorial involves curved modelling.



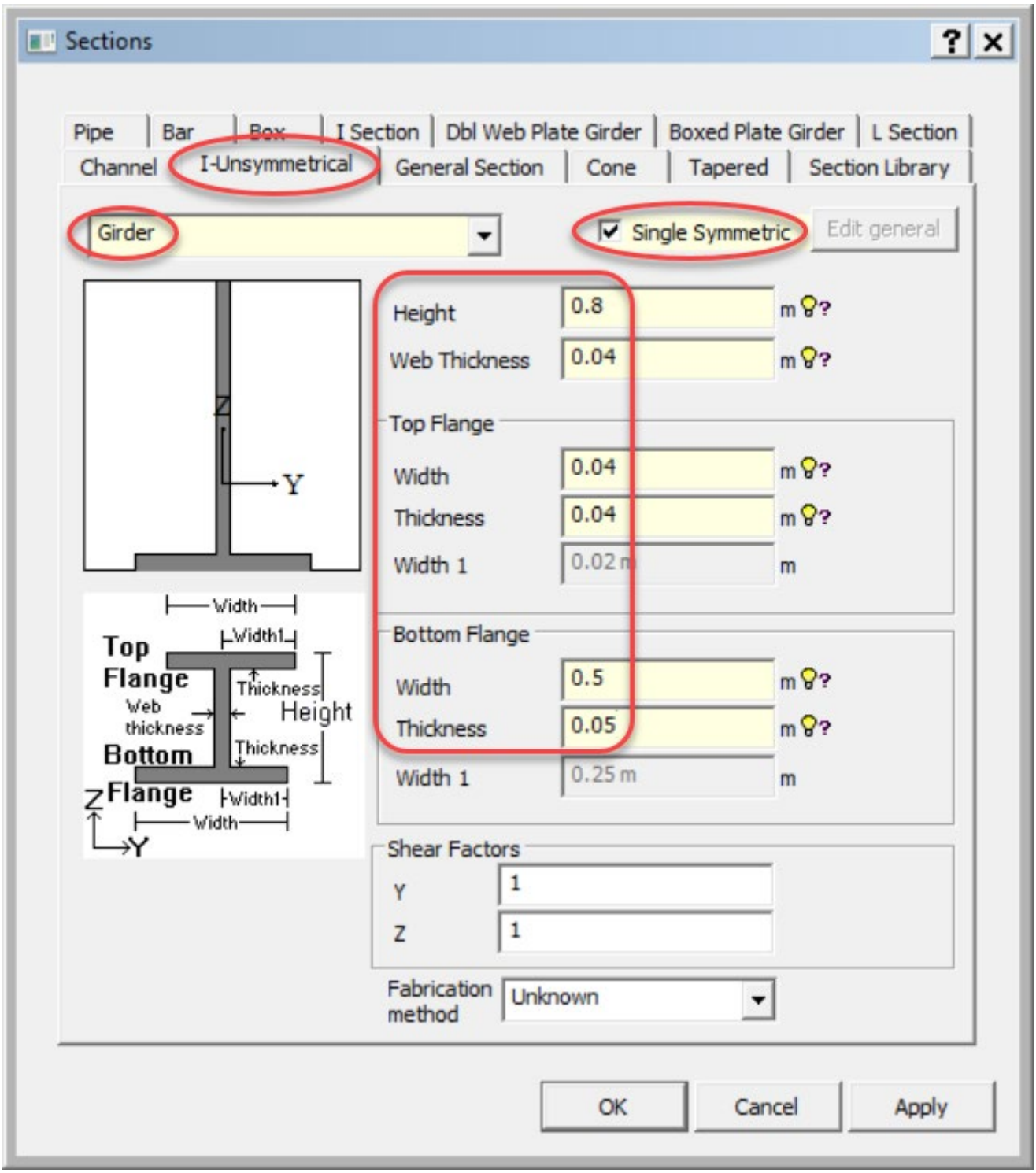
- Define two materials:
  - Steel
    - Yield strength = 2.0E8 Pa
    - ρ = 7850 kg/m^3
    - E = 2.1E11 Pa
    - υ = 0.3
  - SuperMaterial
    - Yield strength = 2.0E8 Pa
    - ρ = 1780 kg/m^3
    - E = 2.1E11 Pa
    - υ = 0.3
  - The SuperMaterial is for the very thick plates and shells used for most of the structure.
  - Thick plates with low material density are used to represent stiffened plates thereby simplifying the modelling job substantially.
  - Set SuperMaterial as the default material.



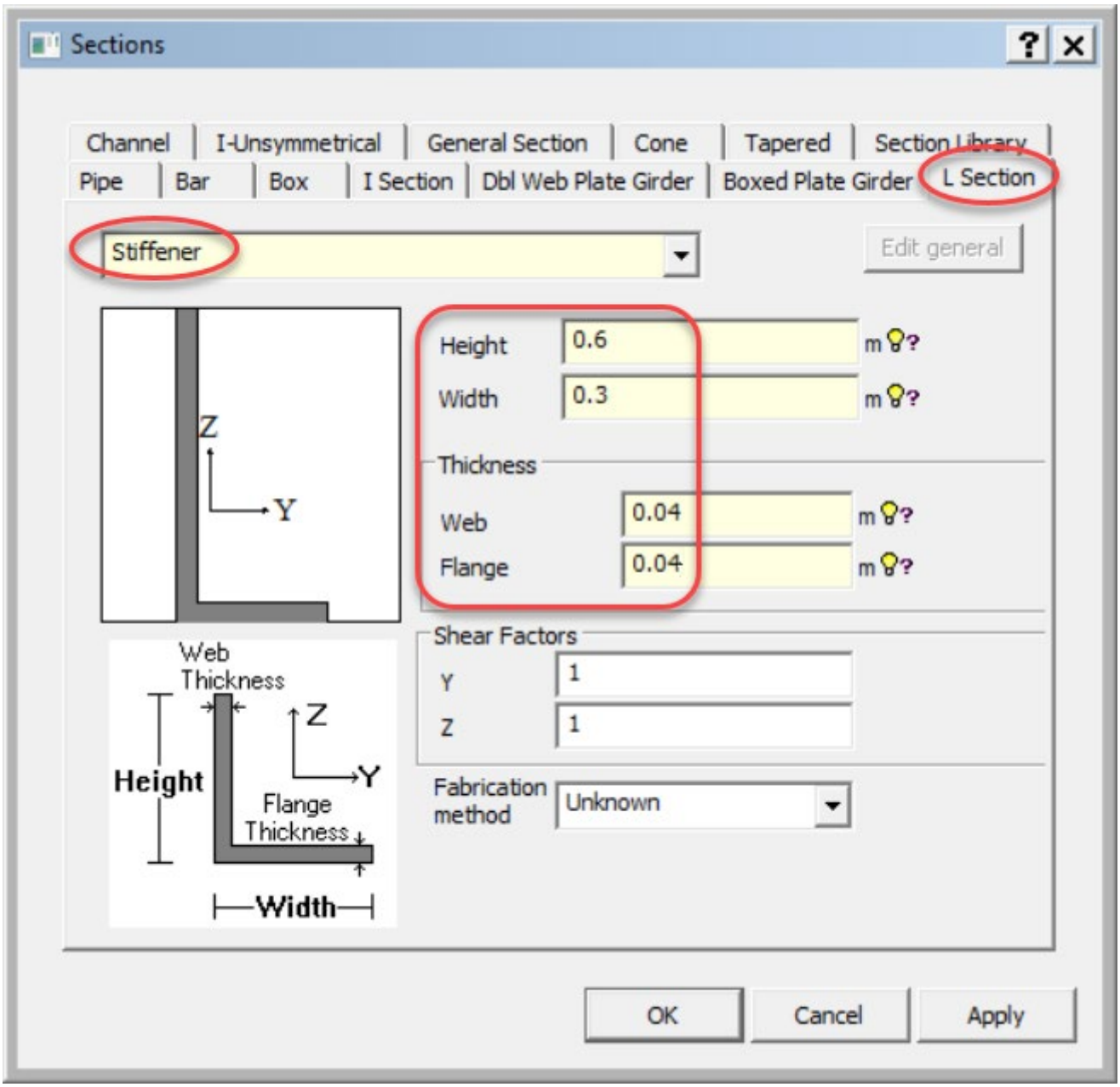
- Create beam cross sections:
  - Pipe1: Diameter = 2 m and thickness = 0.1 m (for braces between columns)
  - Pipe2: Diameter = 1 m and thickness = 0.05 m (for derrick)
  - Pipe3: Diameter = 0.8 m and thickness = 0.04 m (for derrick)
  - Cone1: Cone section with *Dynamic Thickness* = *Largest* (numerically = 1), see below



- Girder: T section modelled as an un-symmetric I section, see below for data



- Stiffener: L section, see below for data



- Set Stiffener as default beam cross section.

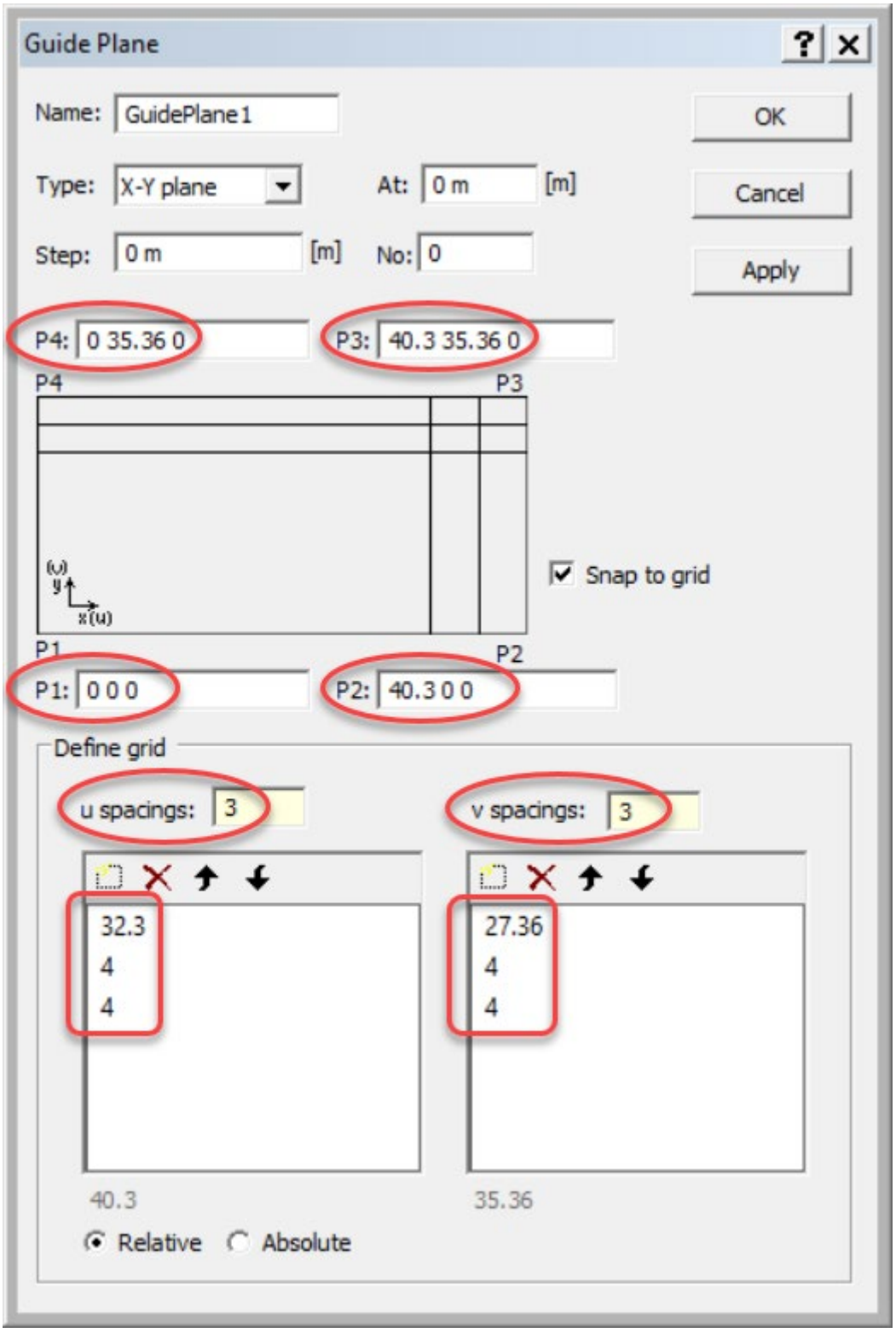
- Create plate thicknesses:
  - Th03: thickness = 0.03 m
  - Th35: thickness = 0.35 m
  - Th50: thickness = 0.50 m

- Set Th50 as default.

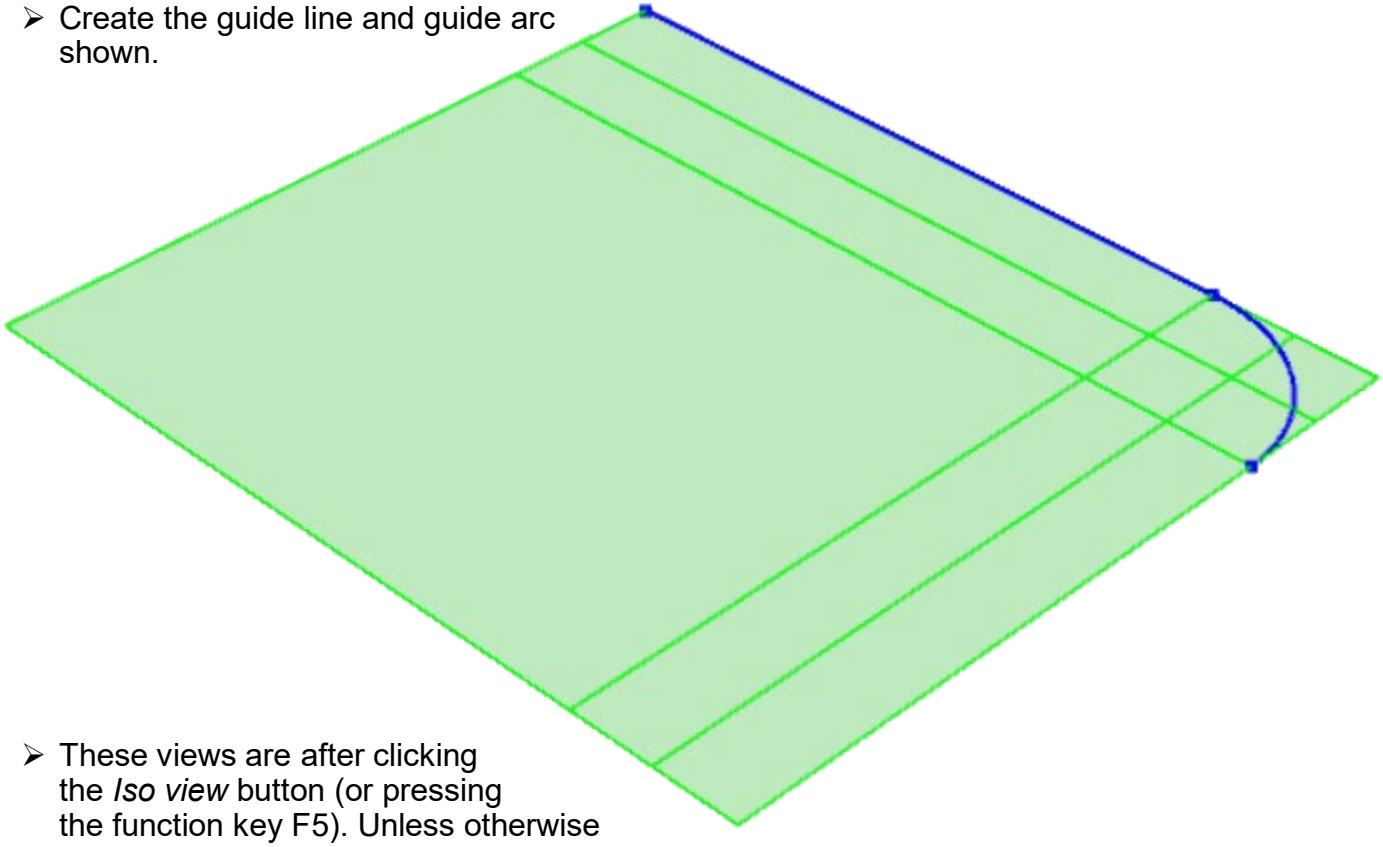
3 CREATE GUIDING GEOMETRY FOR PONTOON

➢ Create a guide plane as shown below. This will be at the pontoon top.

- After entering the last *u spacing* value (4) click another *u spacing* value (32.3 or 4) to see the guide plane sketch in the dialog updated.
- Do the same after entering the last *v spacing* value (4).

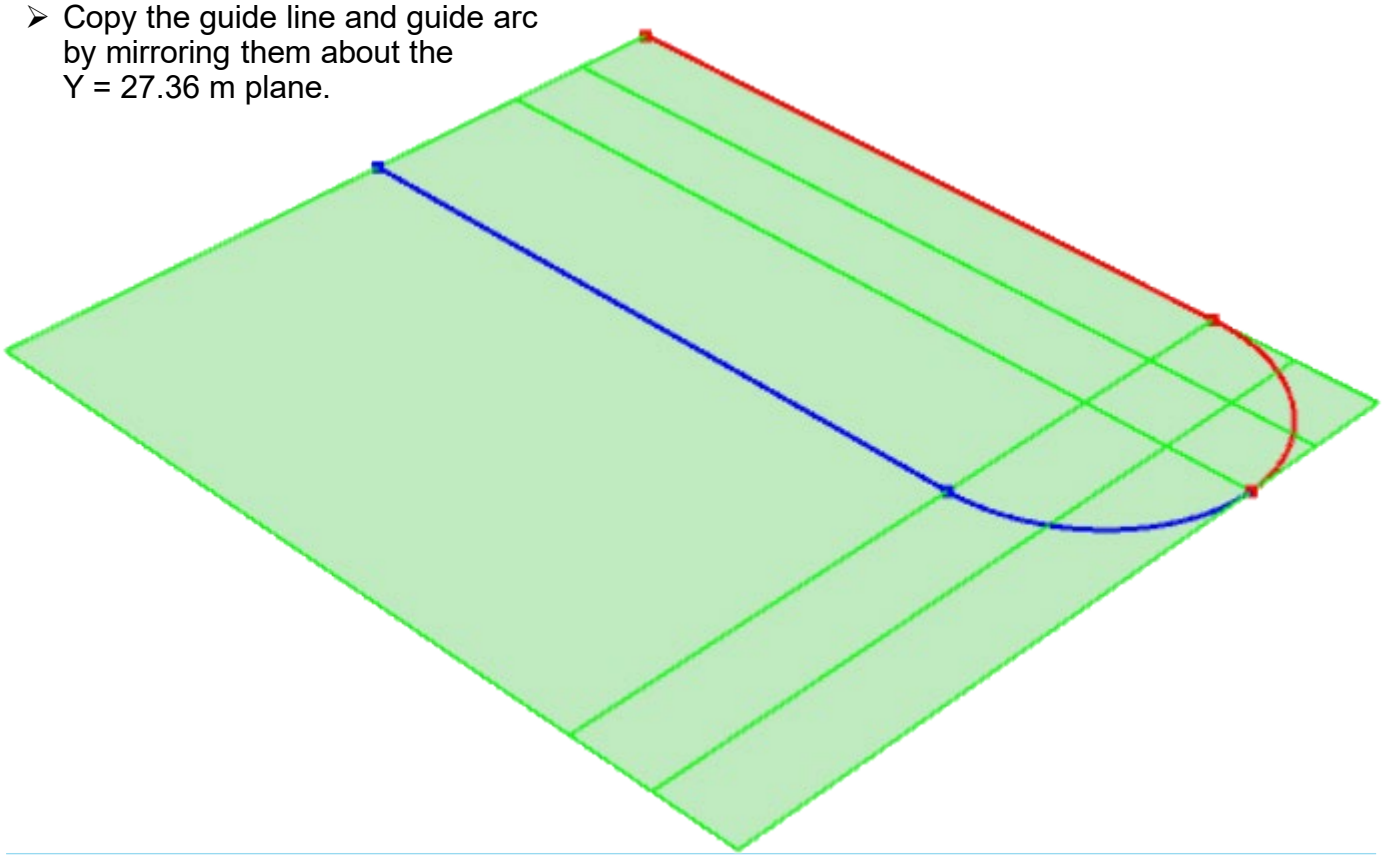


- Create the guide line and guide arc shown.

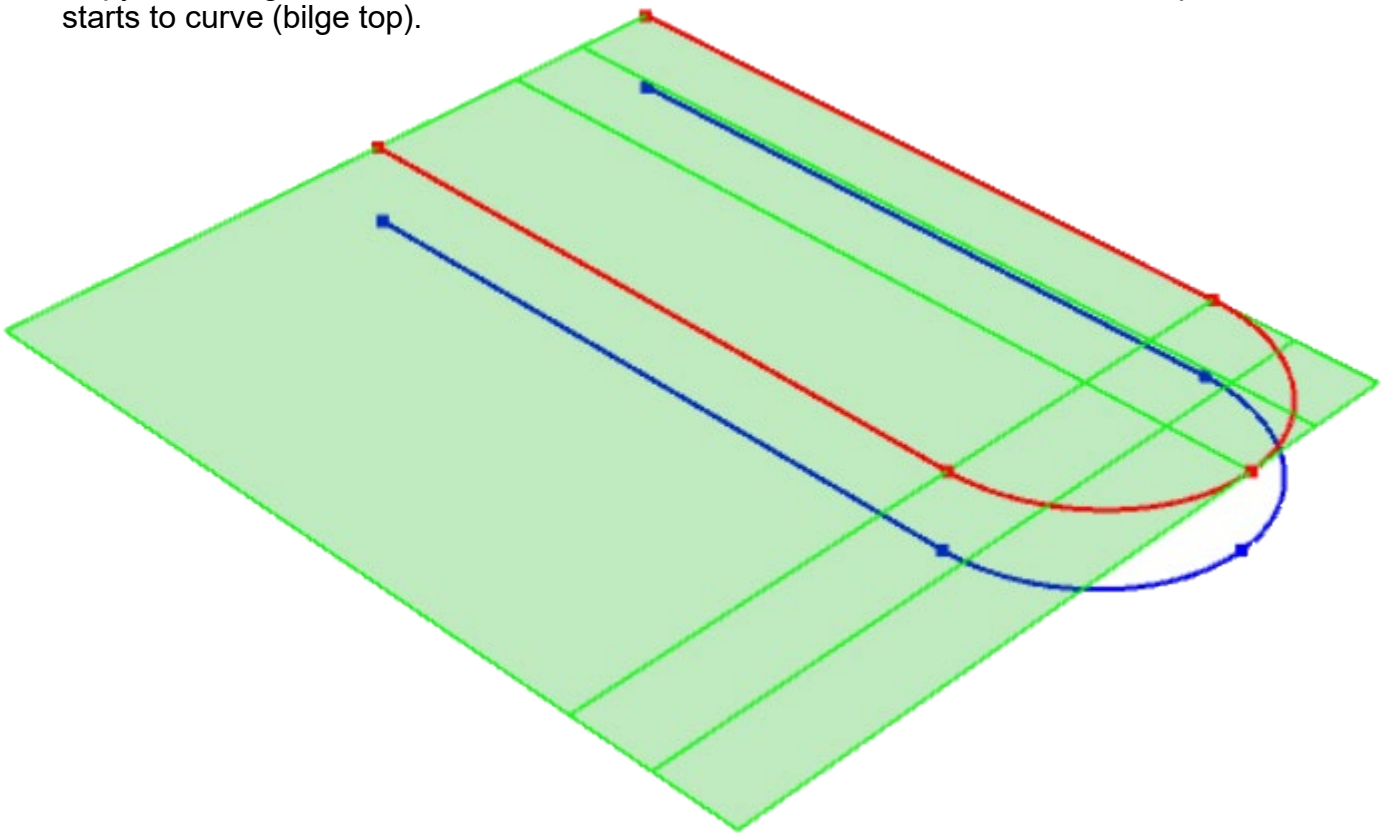


- These views are after clicking the *Iso view* button (or pressing the function key F5). Unless otherwise stated or shown, all views in this tutorial are using this viewpoint, or approximately this viewpoint.

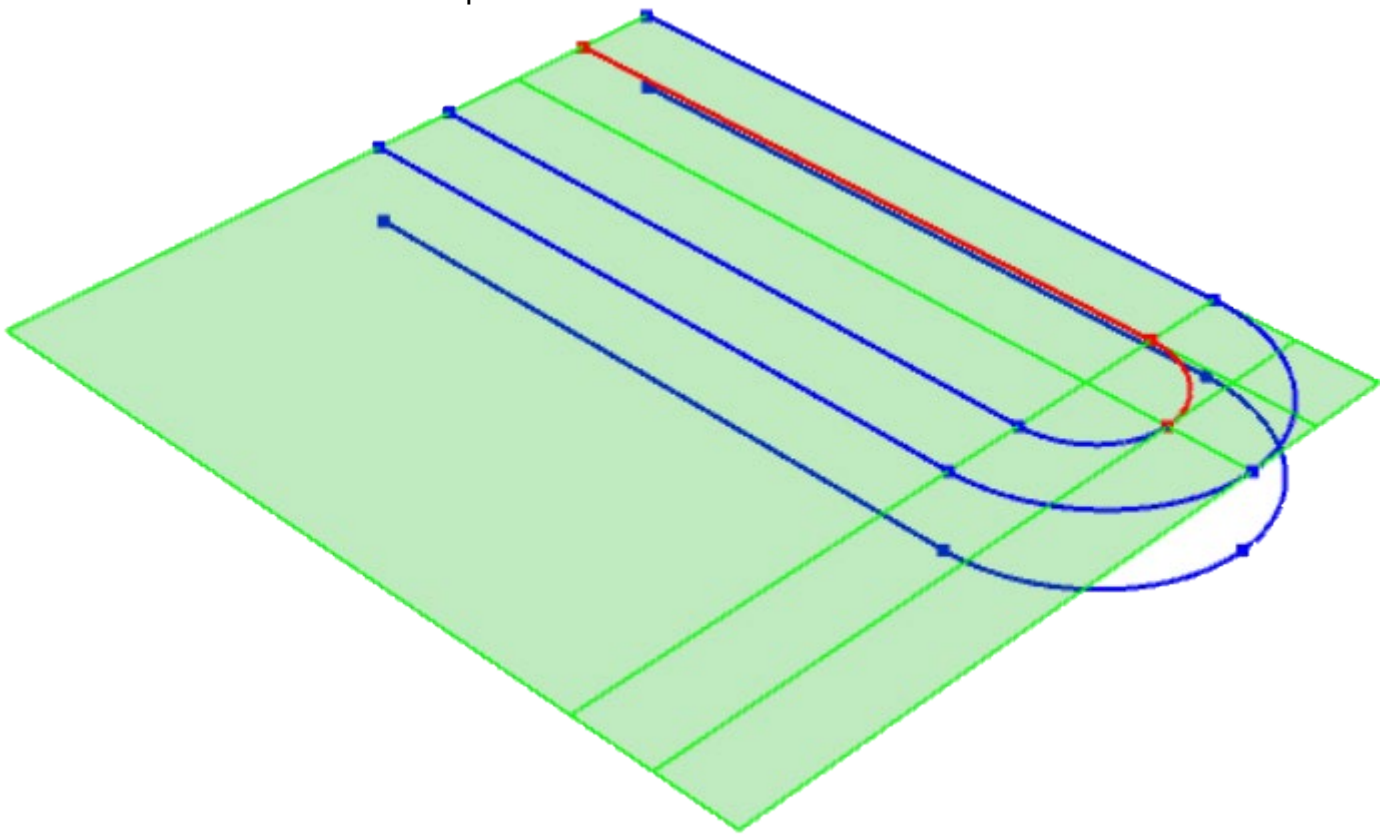
- Copy the guide line and guide arc by mirroring them about the Y = 27.36 m plane.



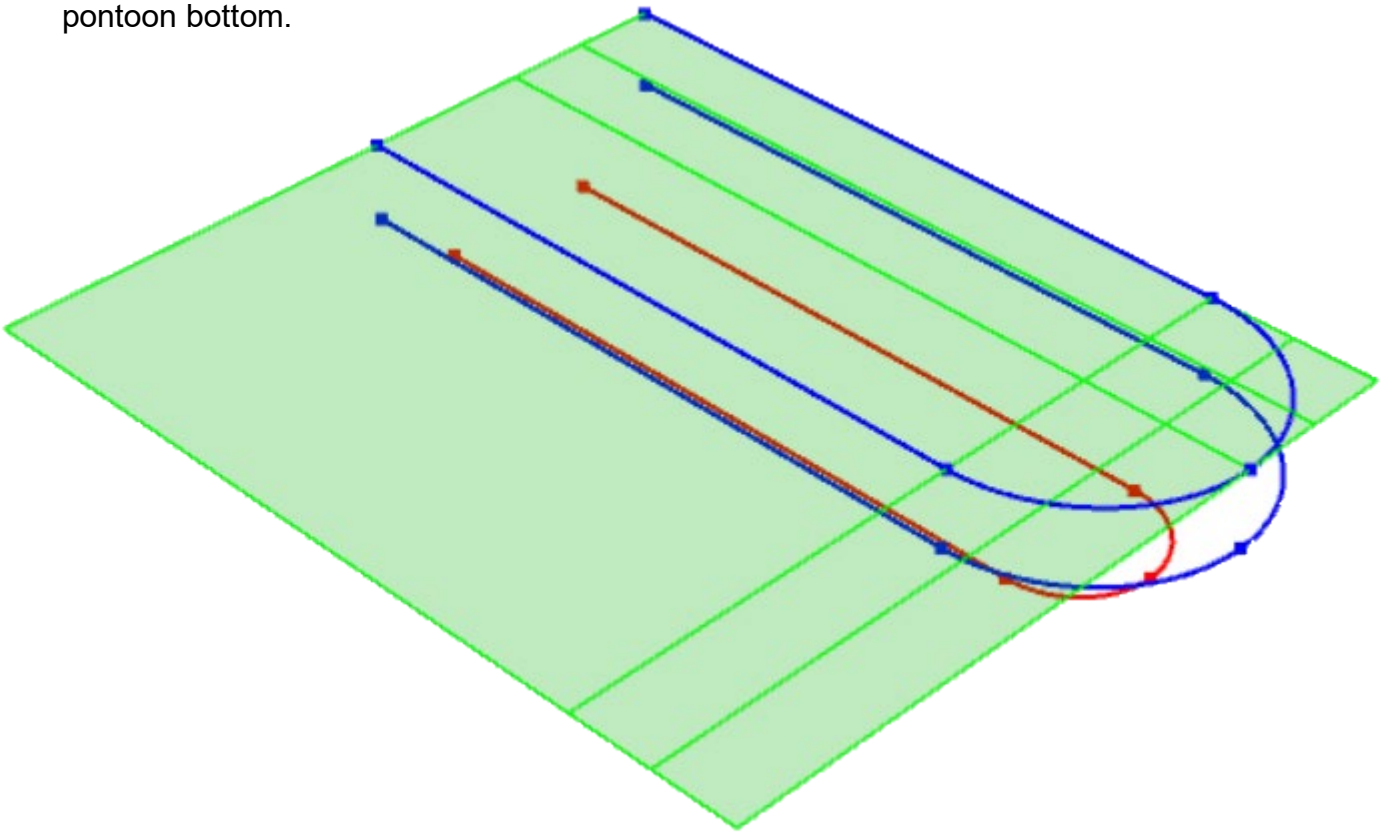
- Copy the four guide curves a distance of 3.75 m downwards. This is where the pontoon starts to curve (bilge top).



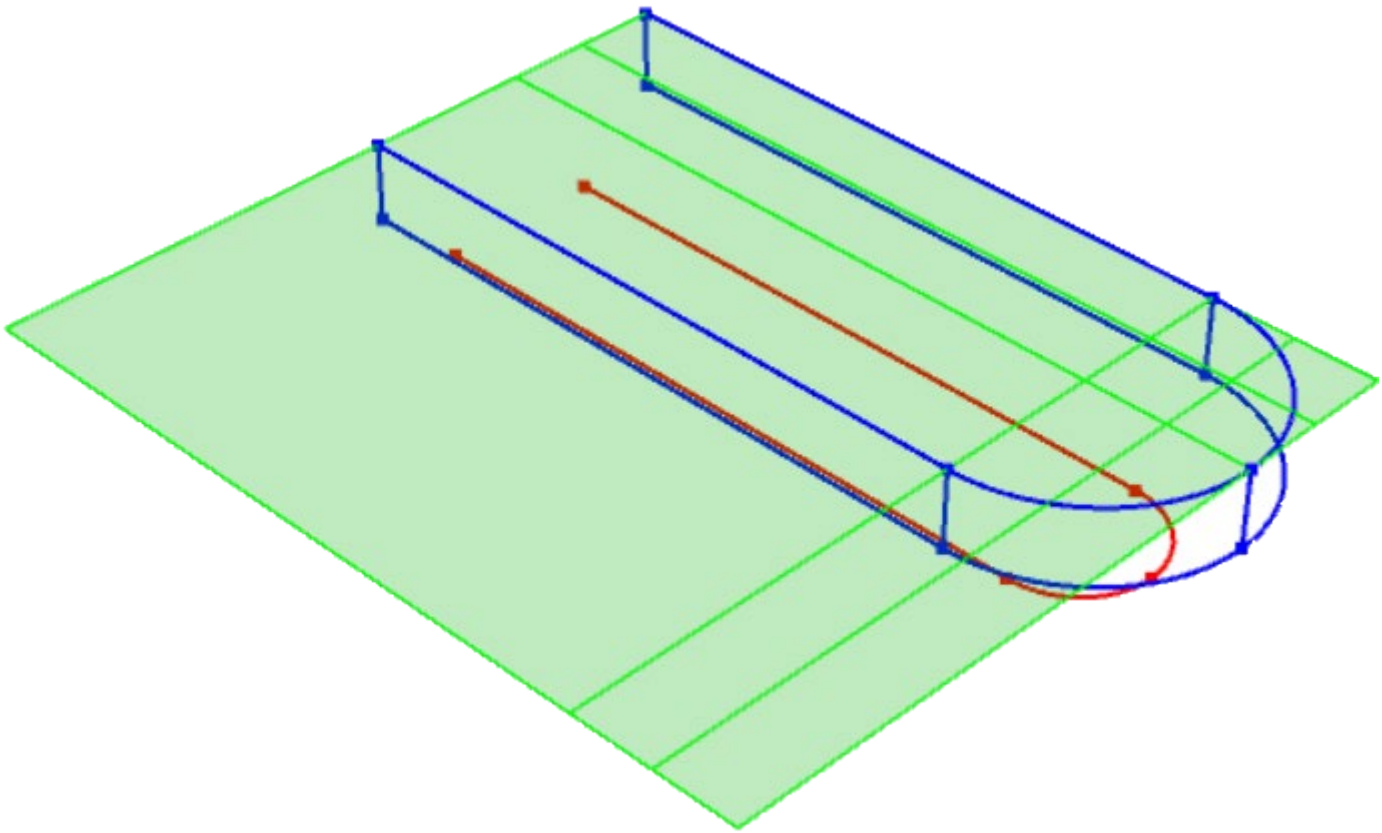
- Create a new guide line and guide arc as shown in red below. Then copy these by mirroring them about the Y = 27.36 m plane.



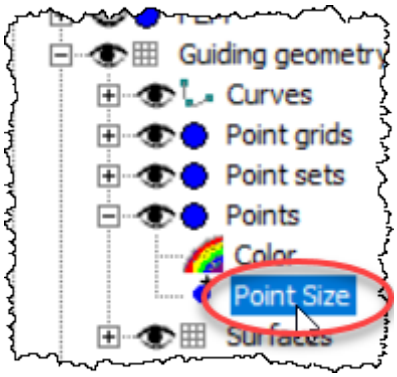
- Move the last created four guide curves downwards a distance of 7.5 m. This is at the pontoon bottom.



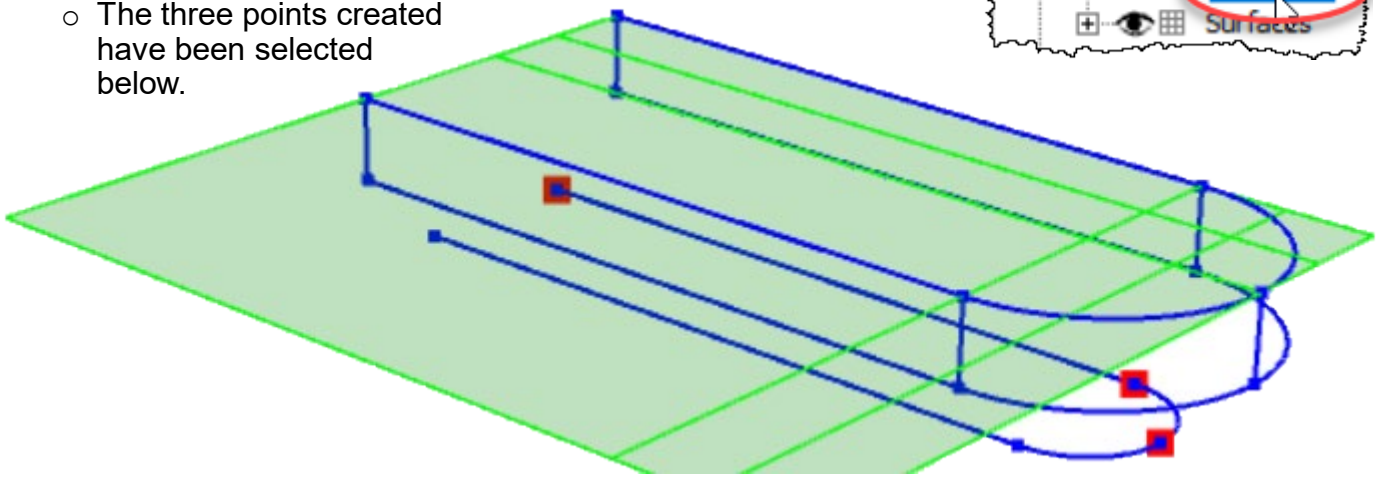
- Create five vertical guide lines for the vertical sides of the pontoon as shown below.



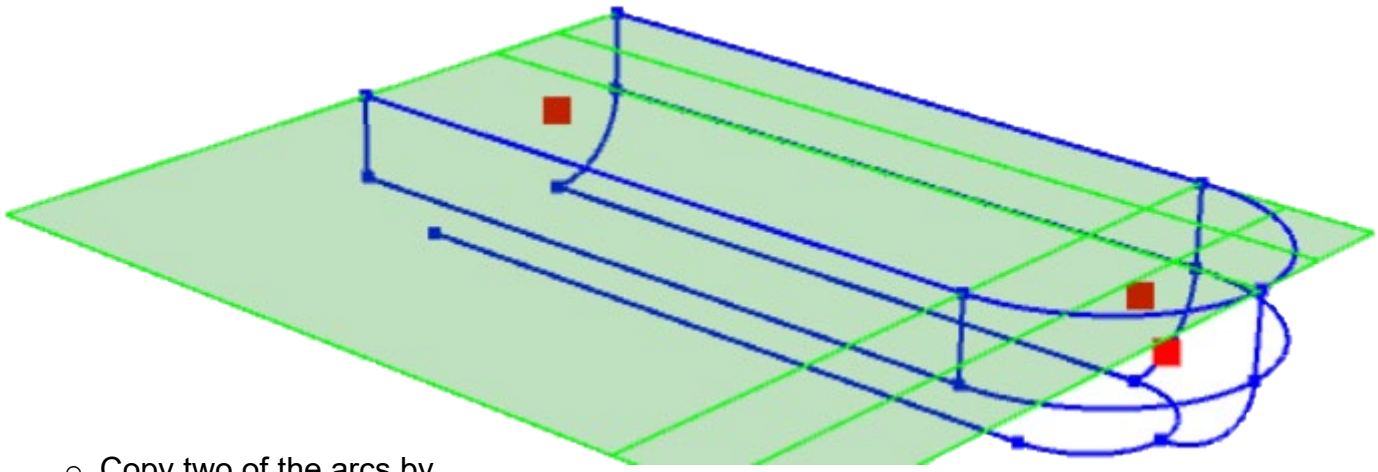
➢ To create the cylindrical and spherical lower parts of the pontoon, arcs in vertical planes are required. To create these arcs centre points are needed. Create the three guide points shown below.



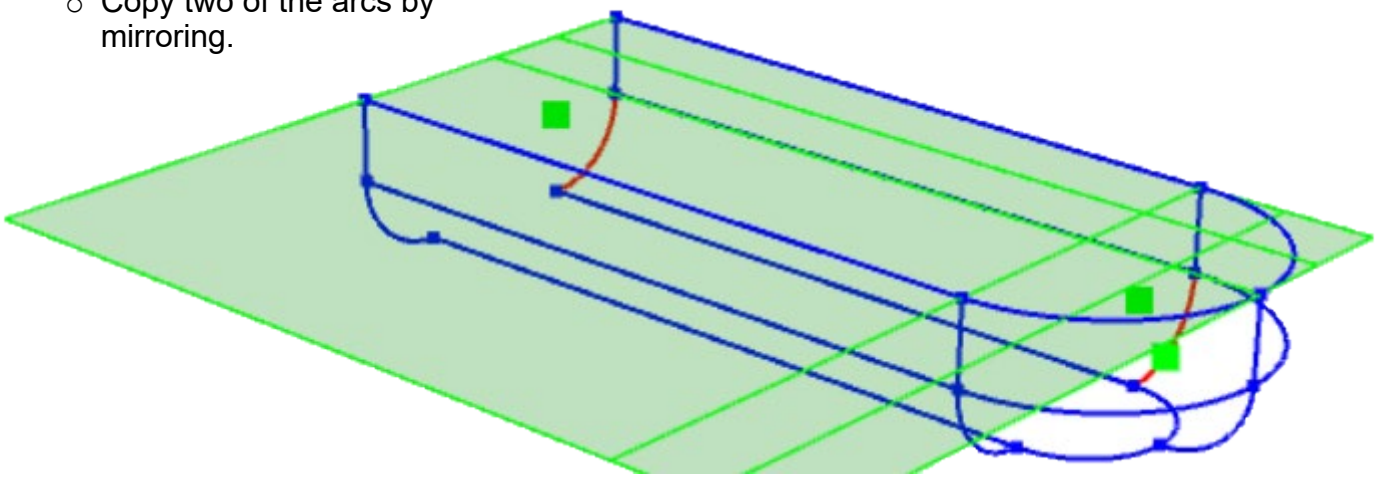
- Use *View* | *Options* | *Settings* and double-click *Point Size* shown to the right to increase the point sizes thereby making them more easily visible.
- The three points created have been selected below.



- Move the three points upwards a distance of 3.75 m and then use these as centre points for creating the three arcs in vertical planes shown below.

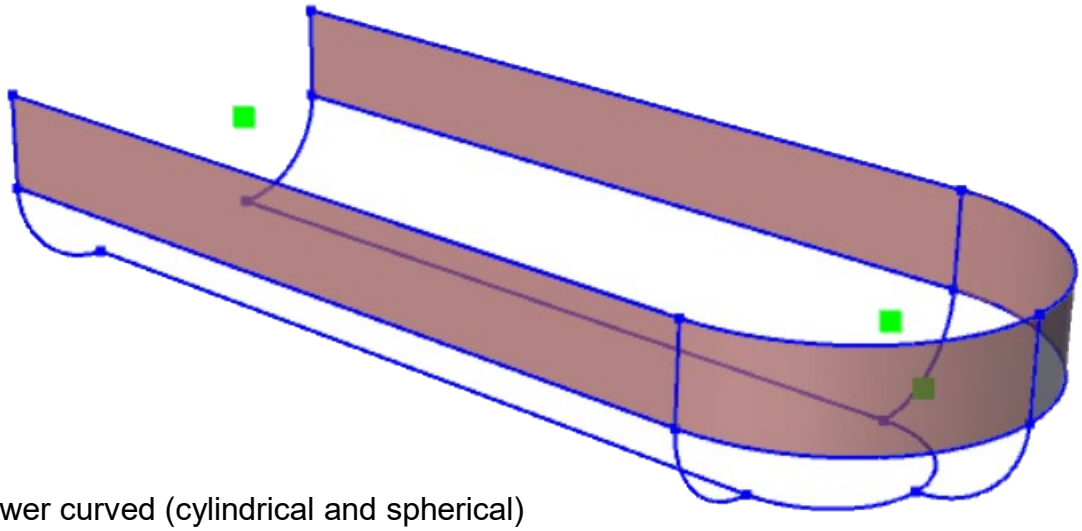


- Copy two of the arcs by mirroring.

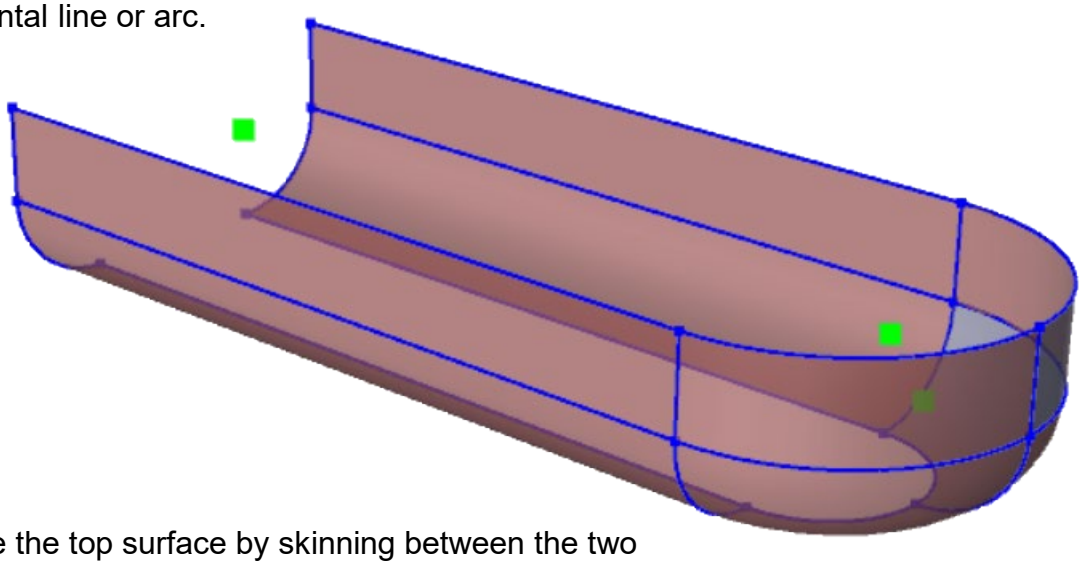


4 CREATE PONTOON

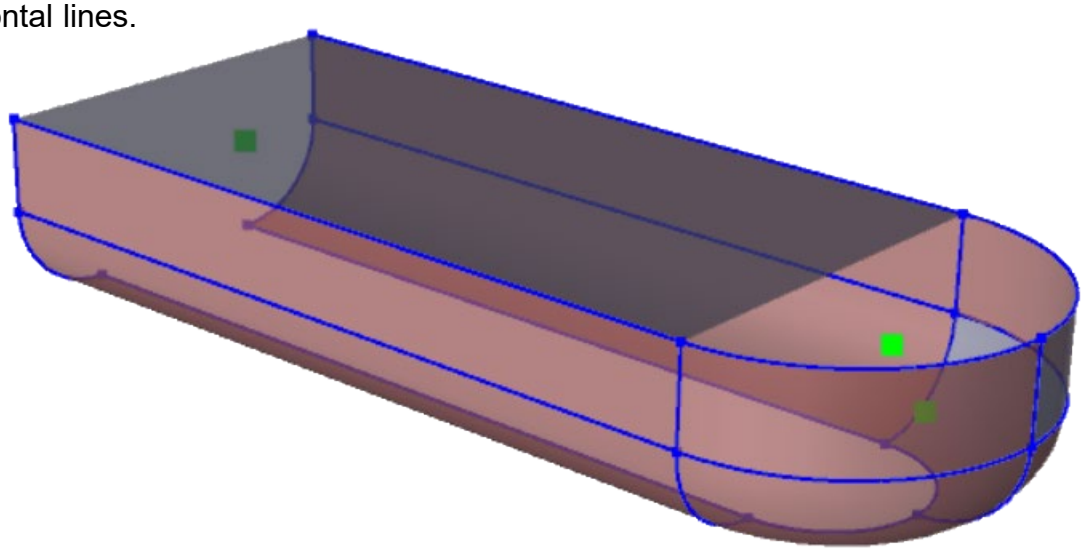
- Use *Structure* | *Free Form Shells* | *Skin/Loft Curves* to create the vertical pontoon sides. Click the upper horizontal lines/arcs followed by double-clicking the corresponding lower lines/arcs. Ensure that the plate thickness property is Th50 for all surfaces.



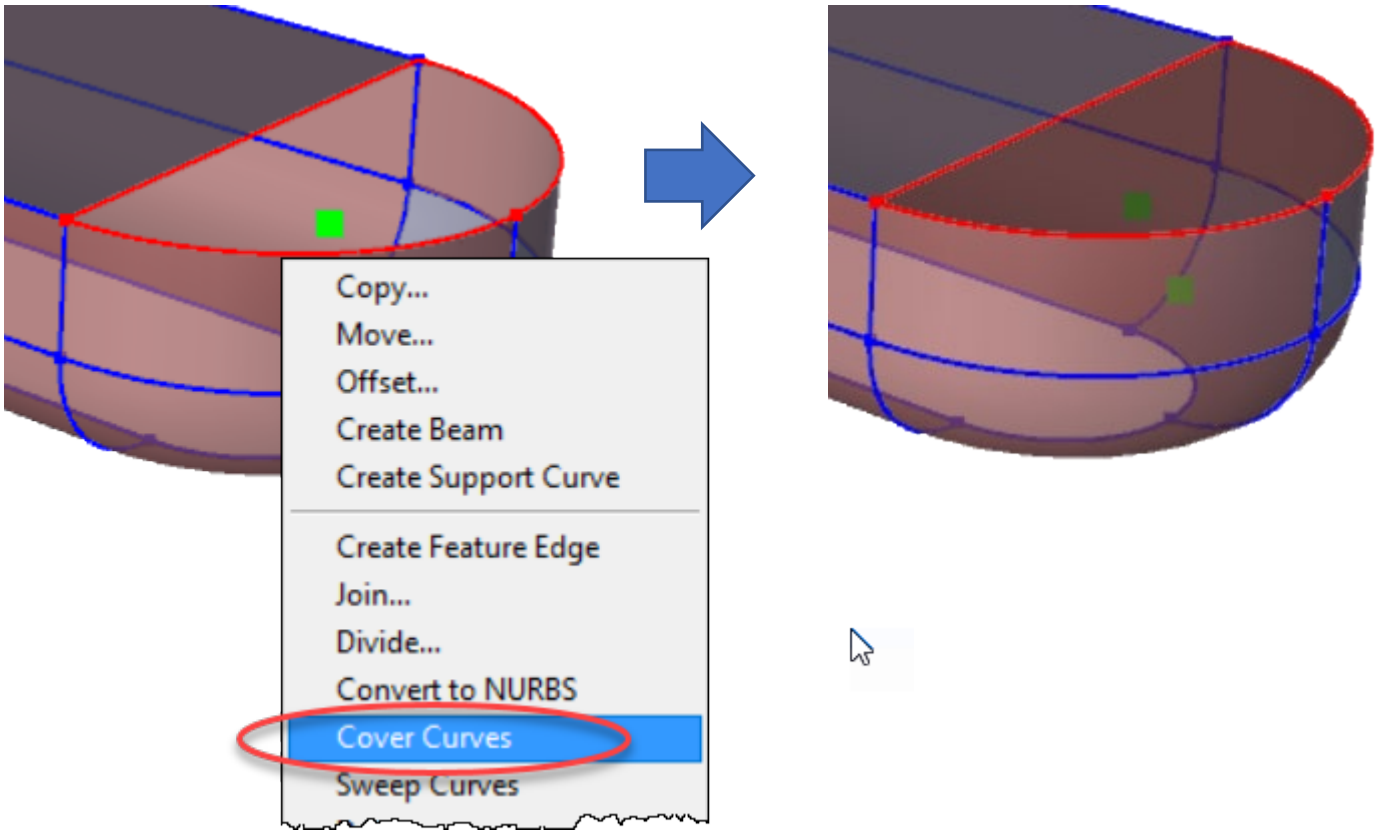
- The lower curved (cylindrical and spherical) surfaces of the pontoon are created by *Structure* | *Free Form Shells* | *Sweep a Profile along a Trajectory*. Click an arc in the vertical plane (the profile) and then the trajectory being a horizontal line or arc.



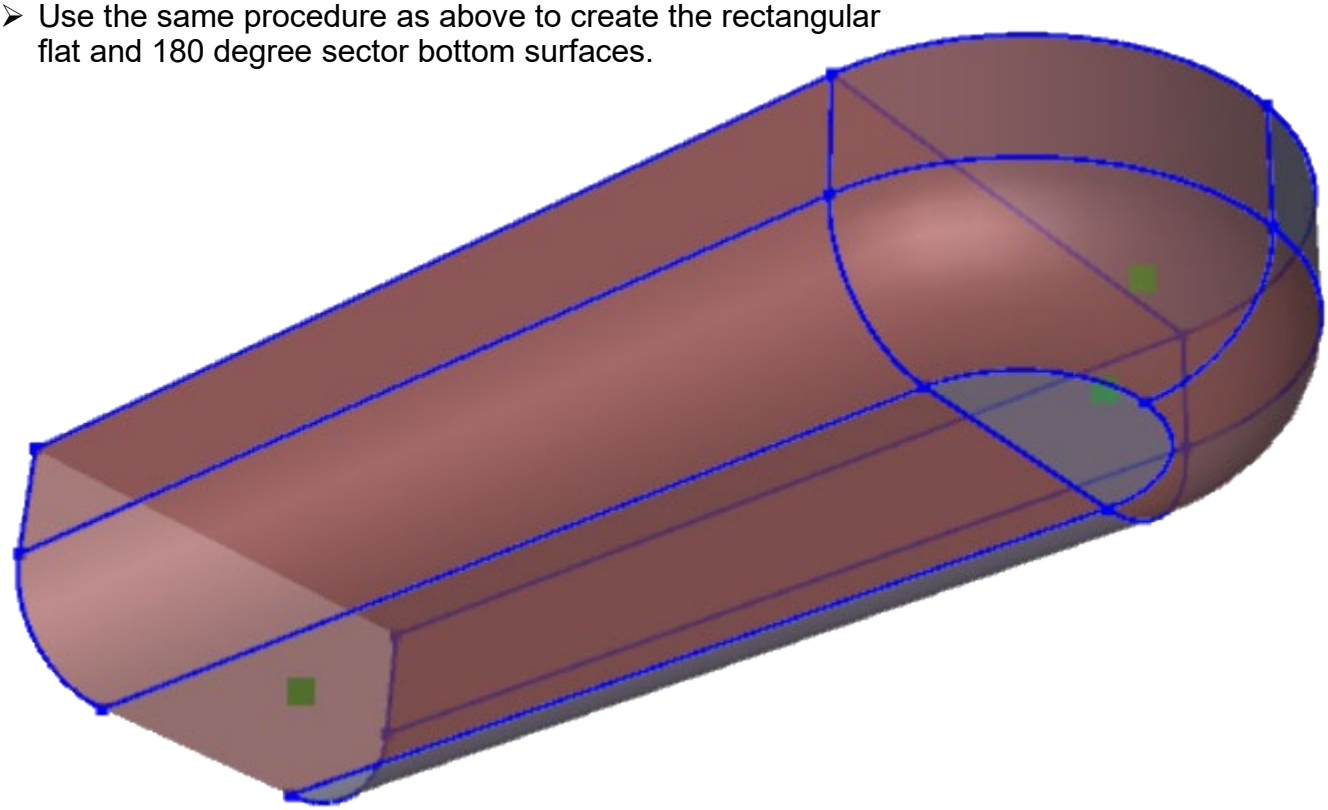
- Create the top surface by skinning between the two horizontal lines.



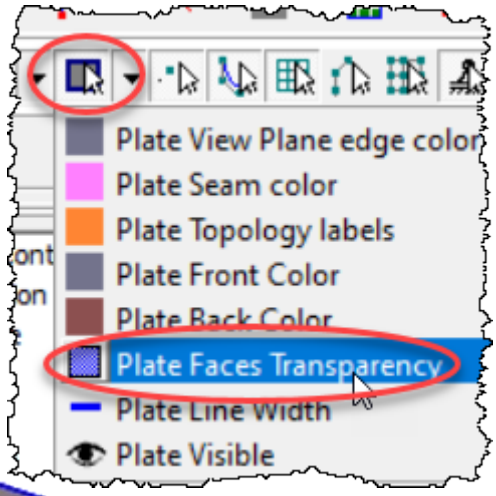
➢ To cover the 180 degree sector of the top surface, create first a guide line as shown, select the surrounding guide line and curves, right-click and select *Cover Curves*.

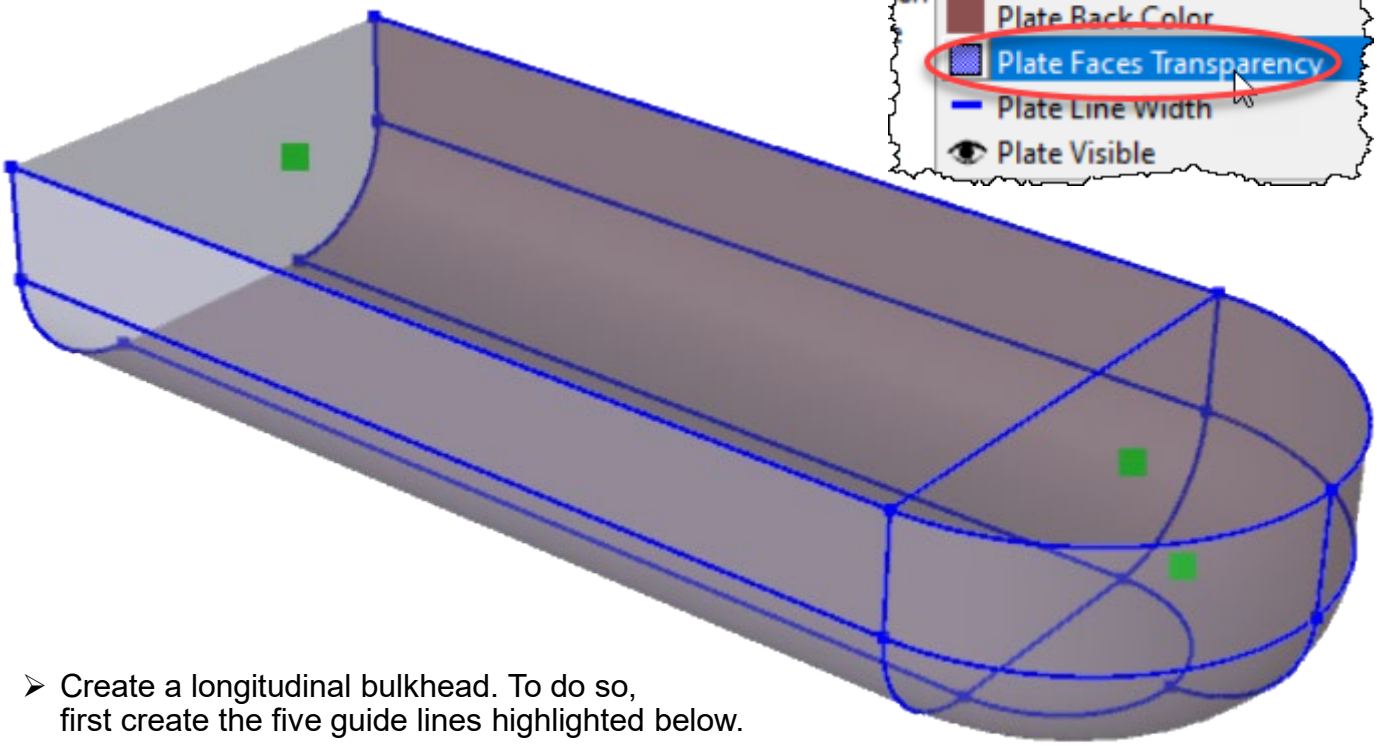


➢ Use the same procedure as above to create the rectangular flat and 180 degree sector bottom surfaces.

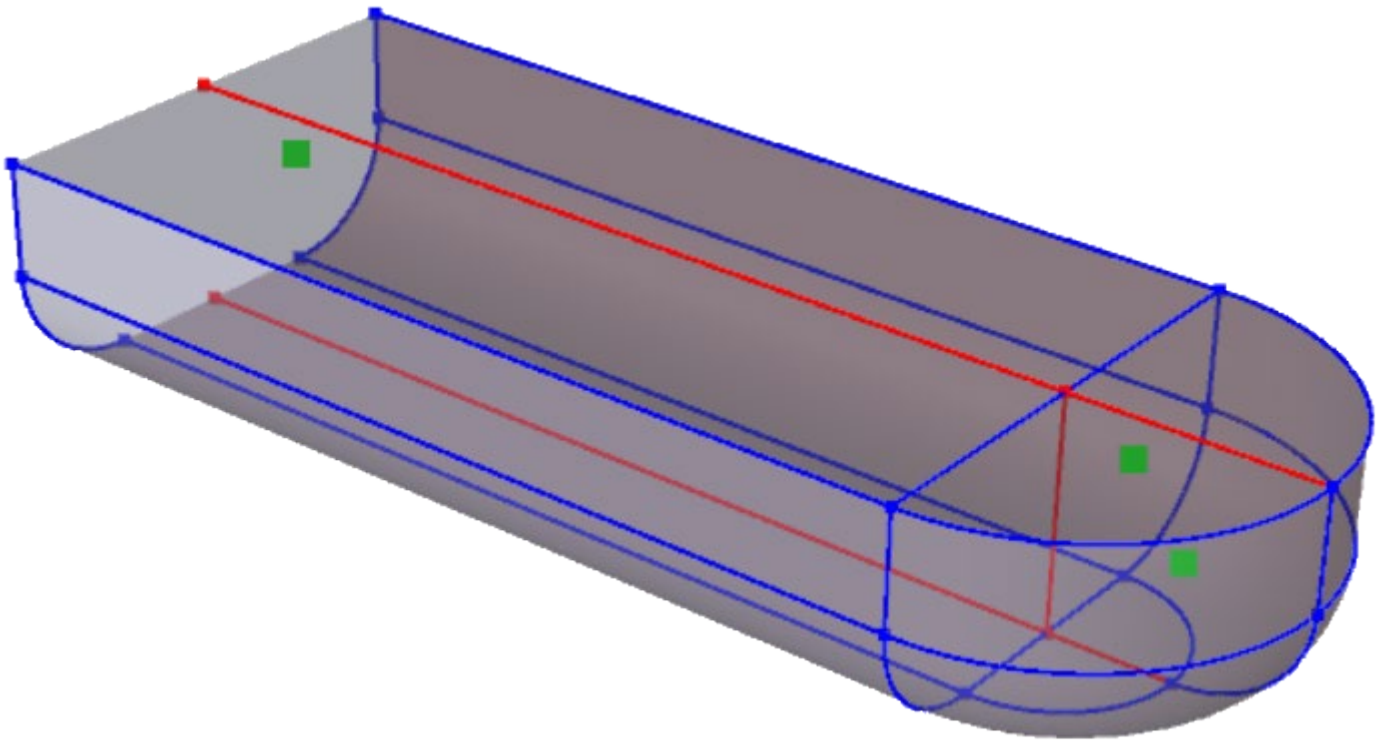


- Flip surface normals so that all normals point consistently outwards, i.e. bluish on the outside.
  - To ease viewing inside the pontoon right-click the *Plate selection* button and select *Plate Faces Transparency* to increase the transparency.

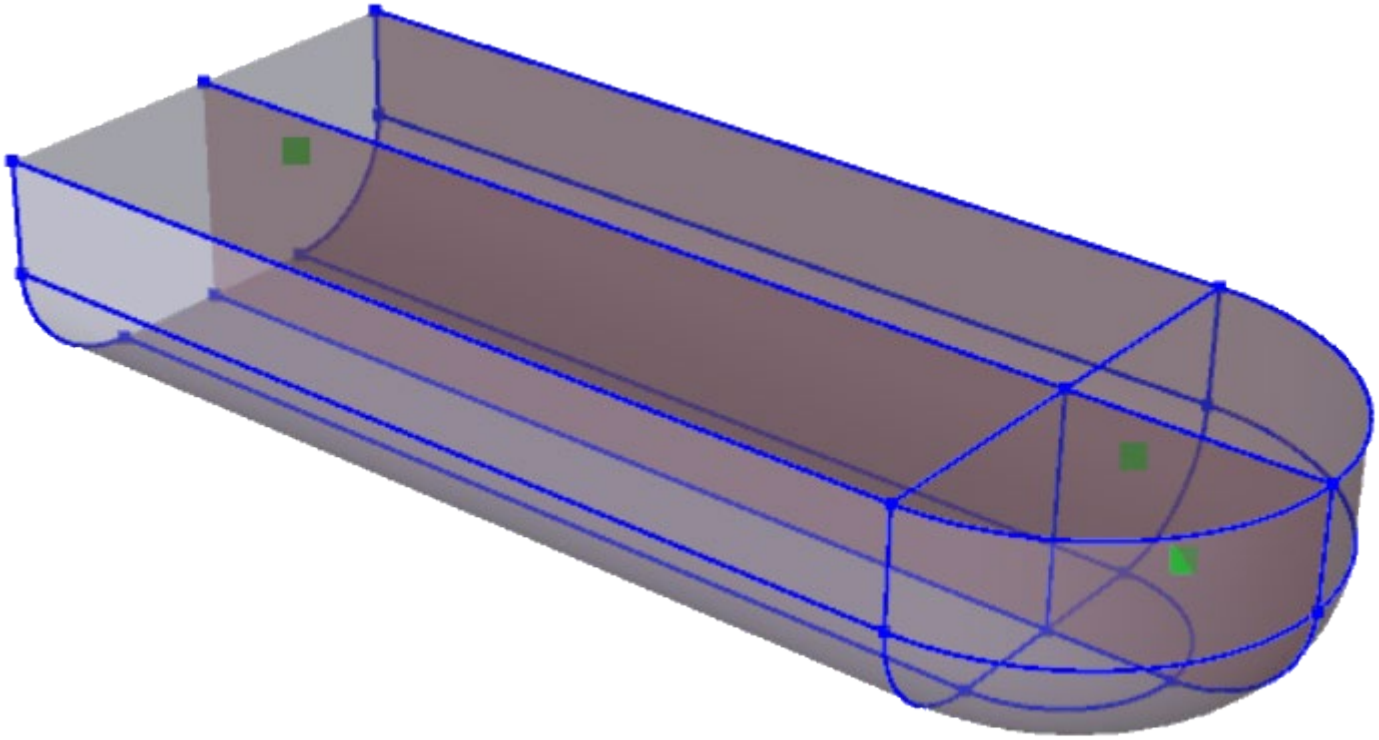




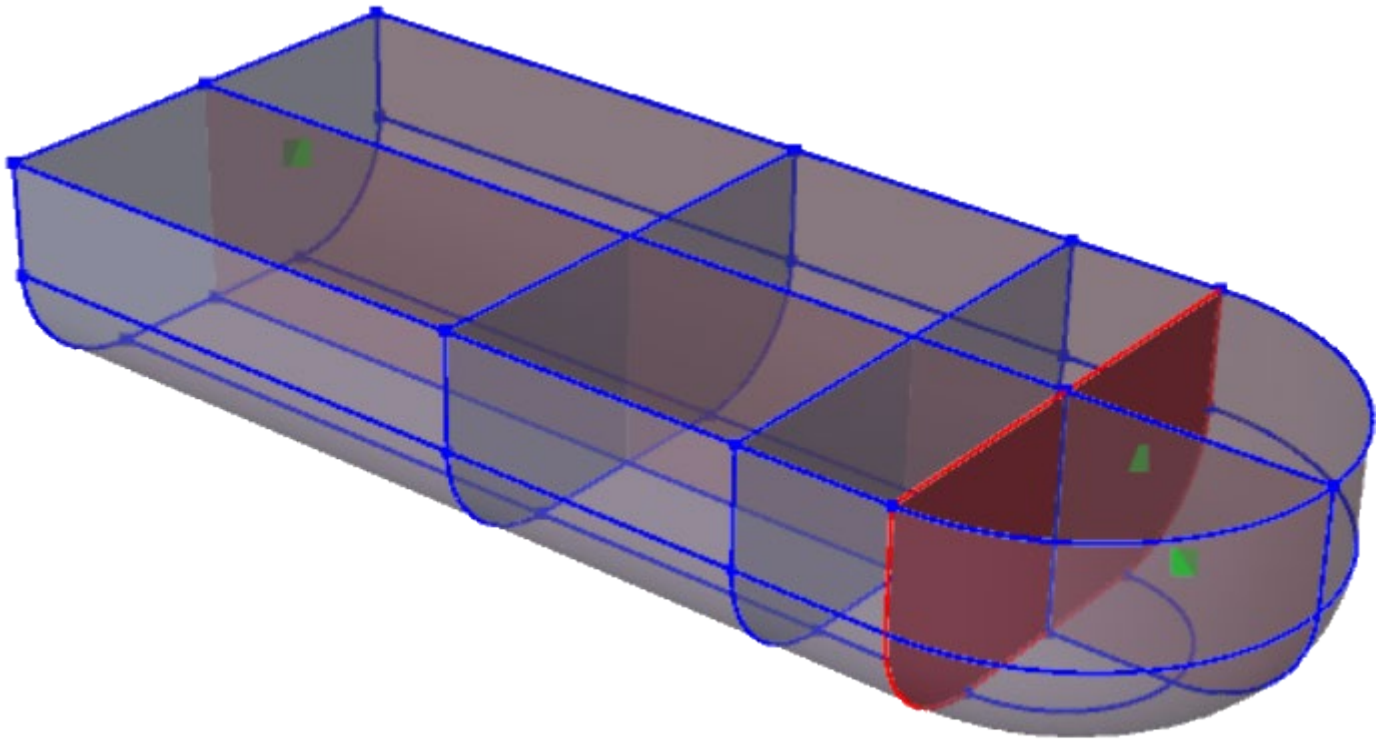
- Create a longitudinal bulkhead. To do so, first create the five guide lines highlighted below.



- Then use a skinning operation for the rectangular surface and a cover operation for the remaining part of the bulkhead.

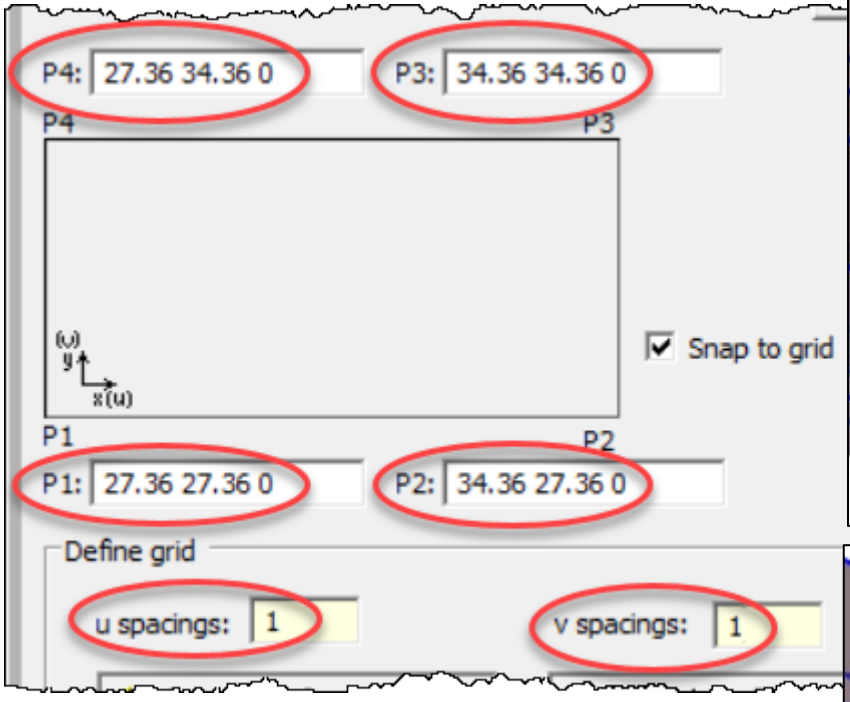


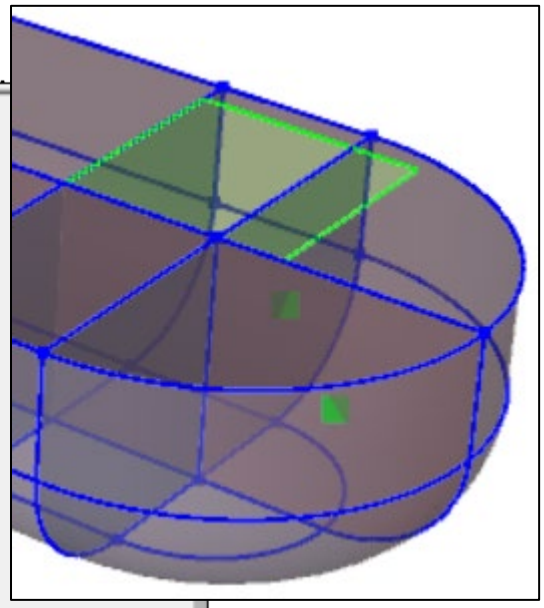
- Create a transverse bulkhead at X = 32.3 m and copy this three times, distances −4.94, −14.94 and −32.2 in X-direction.



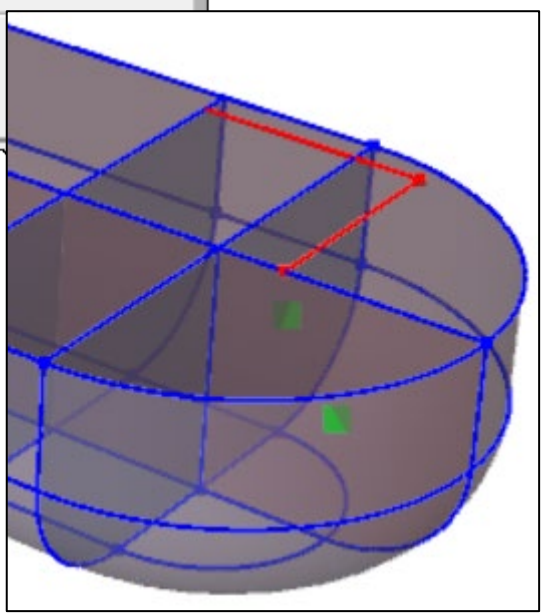
5 CREATE COLUMN

- Create a guide plane for the base of the column.

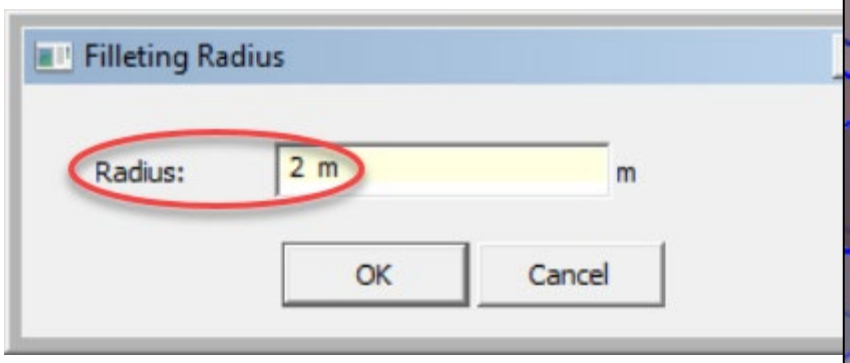


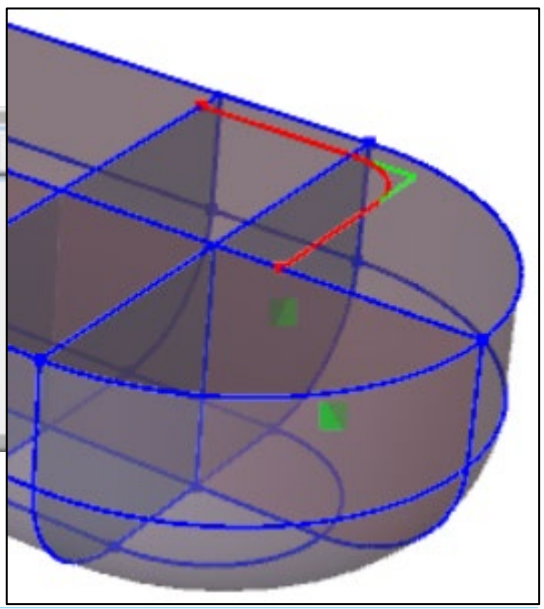


- Create the two guide lines highlighted to the right.



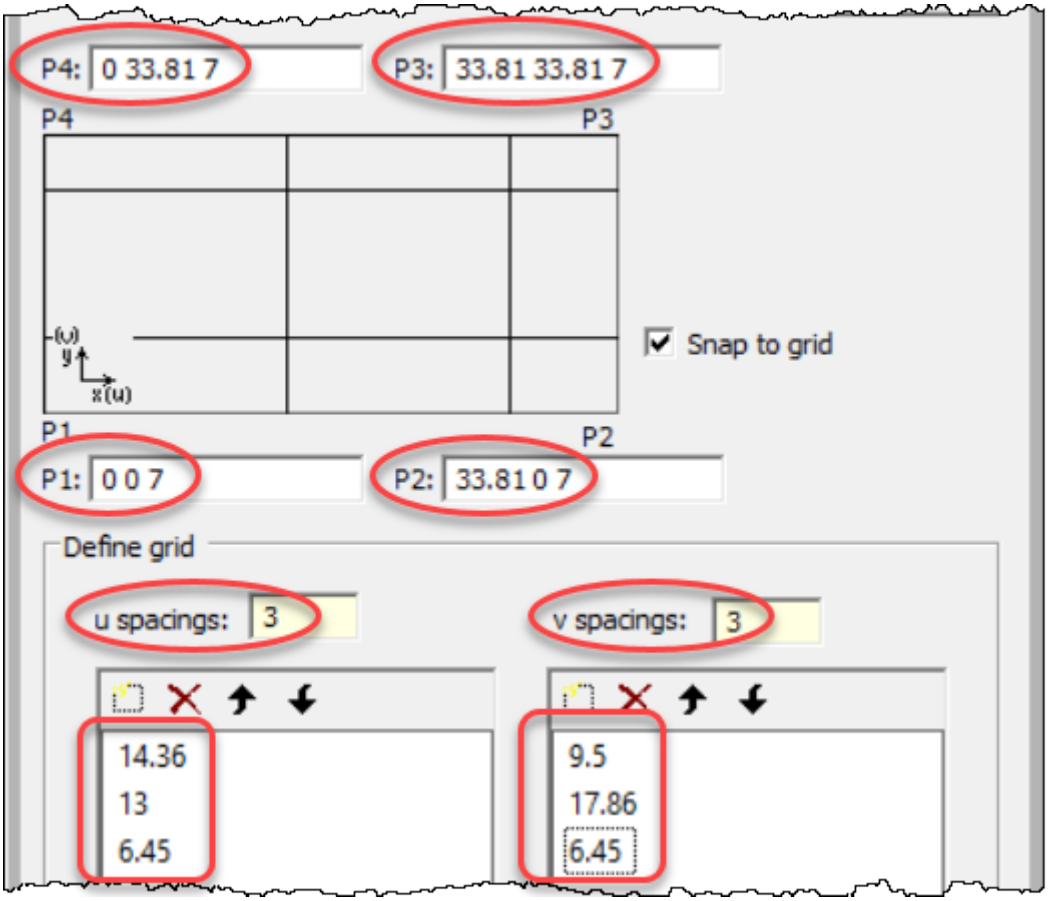
- Use *Guiding Geometry* | *Conic Sections* | *Circular Fillet* to round off the corner between the new guide lines. After clicking the two guide lines the *Filleting Radius* dialog below appears. Give a radius of 2 m.

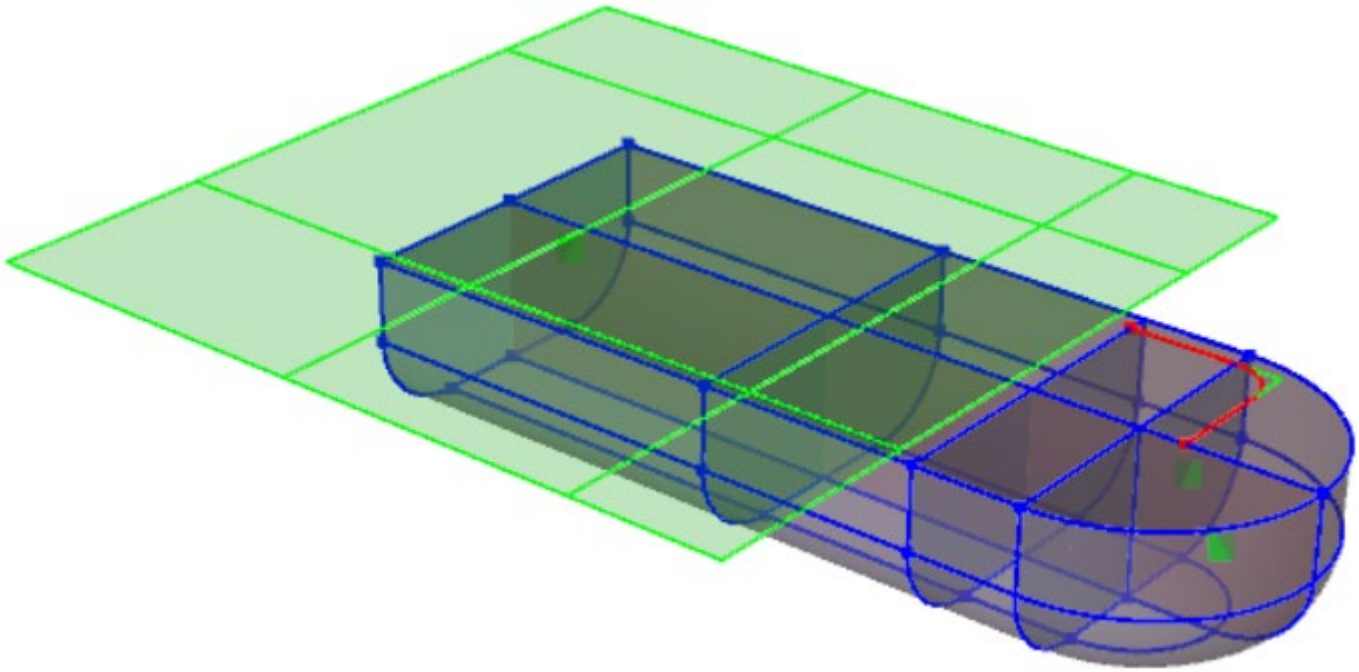




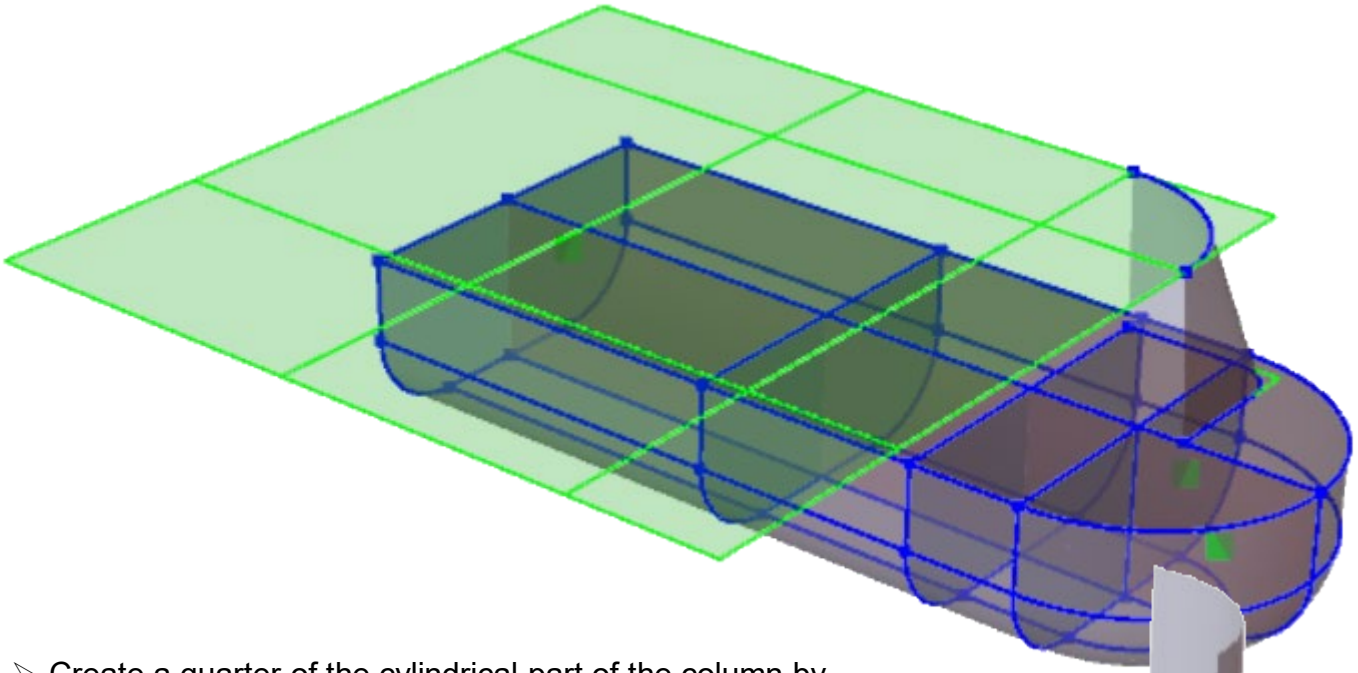
➢ Create a guide plane at elevation 7 m for the base of the cylindrical part of the column.

- After entering the last *u spacing* value (6.45) click another *u spacing* value (14.36 or 13) to see the guide plane sketch in the dialog updated.
- Do the same after entering the last *v spacing* value (6.45).

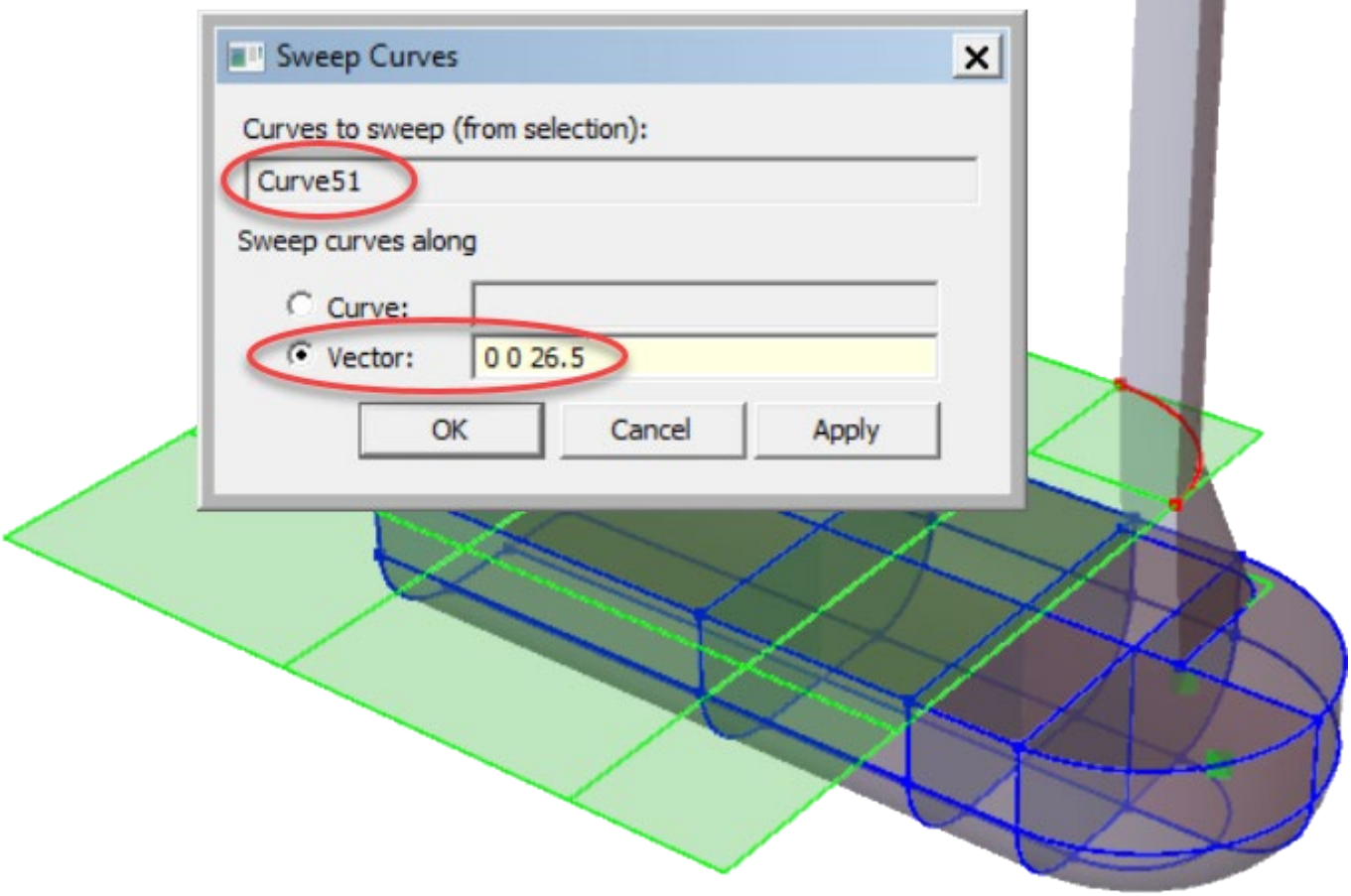




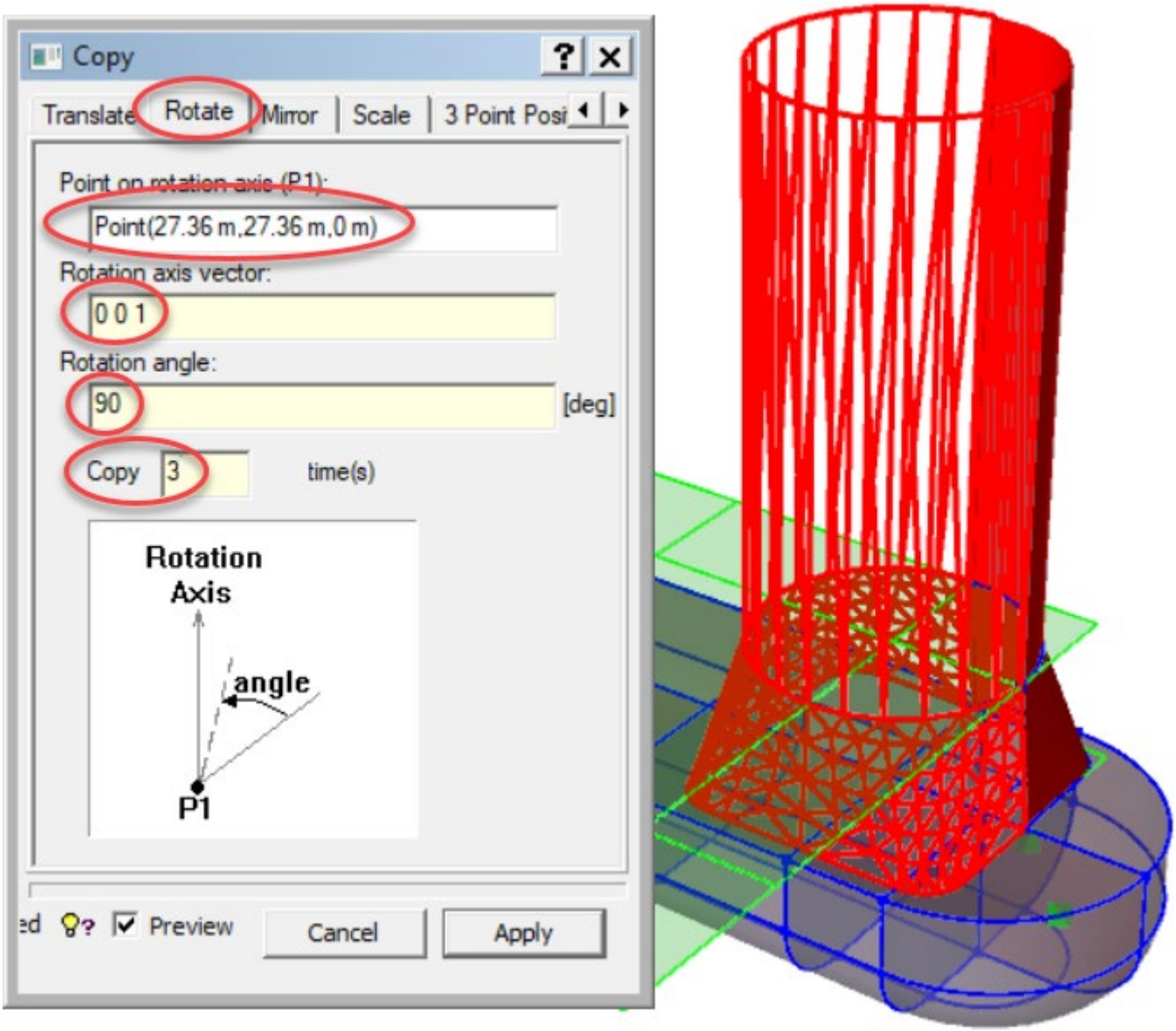
- Create the arc shown at elevation 7 m and do a skinning operation to create a quarter of the transition from the column base to its cylindrical part.



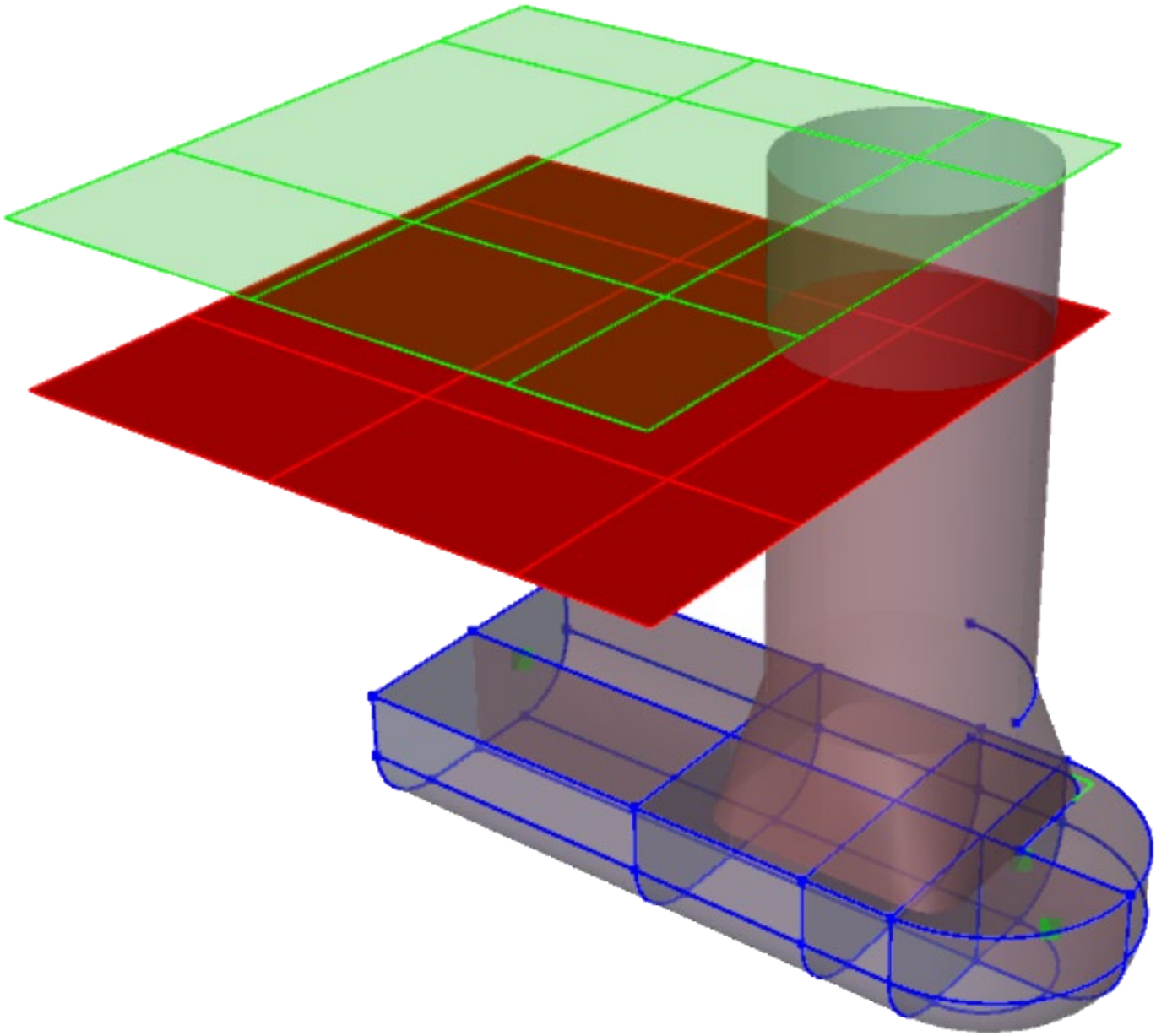
- Create a quarter of the cylindrical part of the column by right-clicking the arc (Curve51 in this case) to sweep it upwards a distance of 26.5 m.



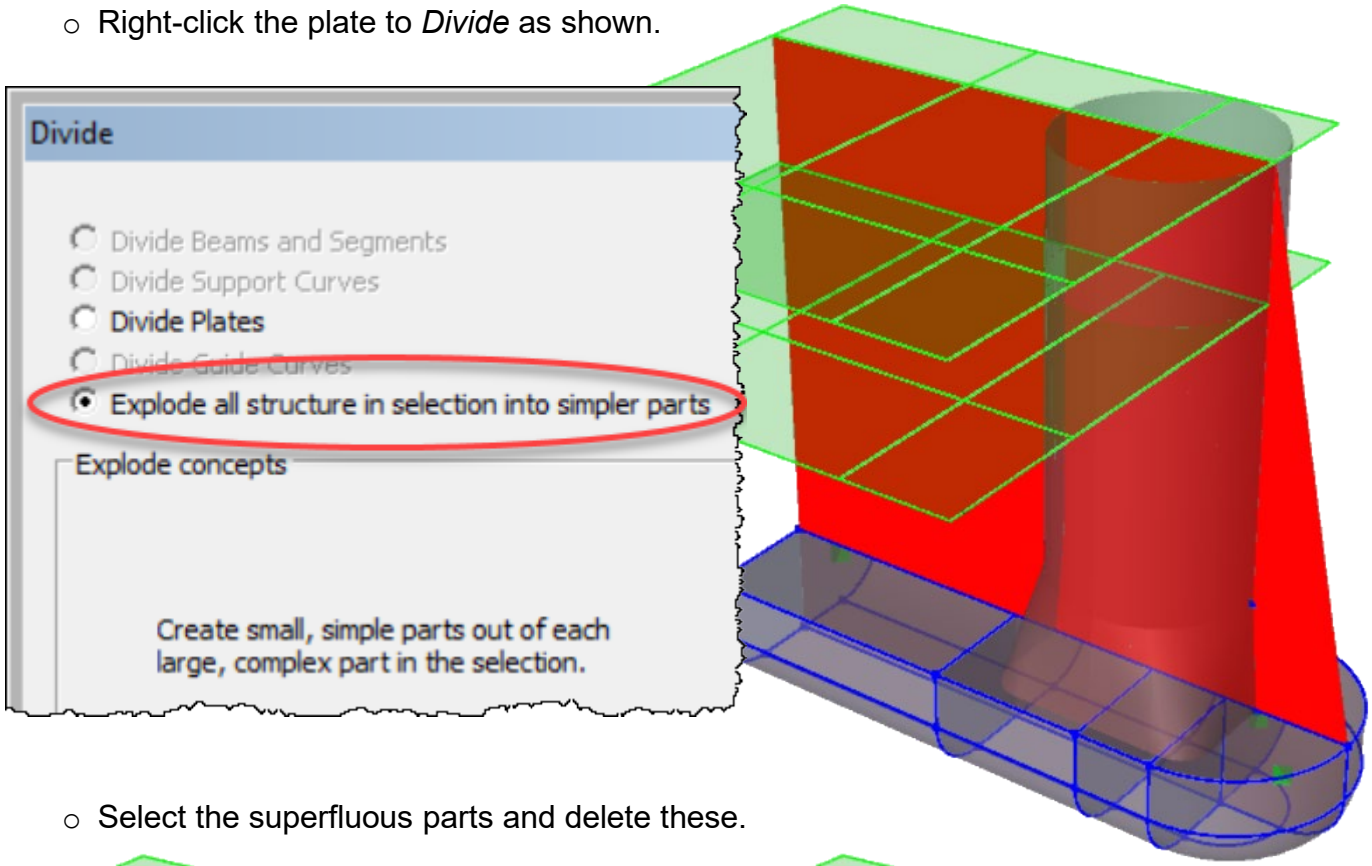
➢ Copy the quarter part of the column (conical transition plus cylindrical part) three times to create the complete column.



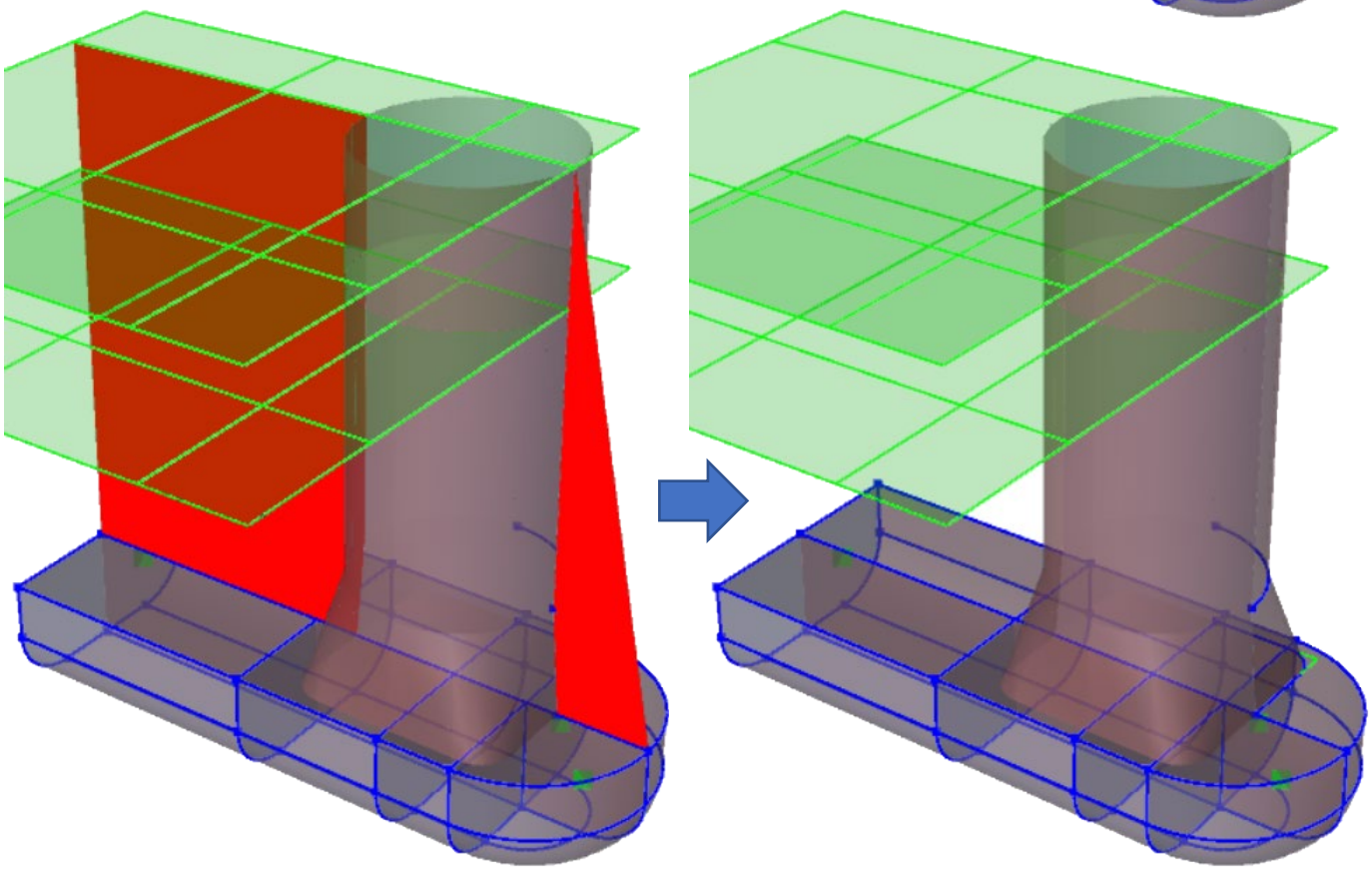
- Move the guide plane at 7 m upwards to the lower deck at elevation 25.5, i.e. 18.5 m upwards. Then copy this guide plane up to the upper deck at elevation 33.5 m, i.e. 8 m upwards.



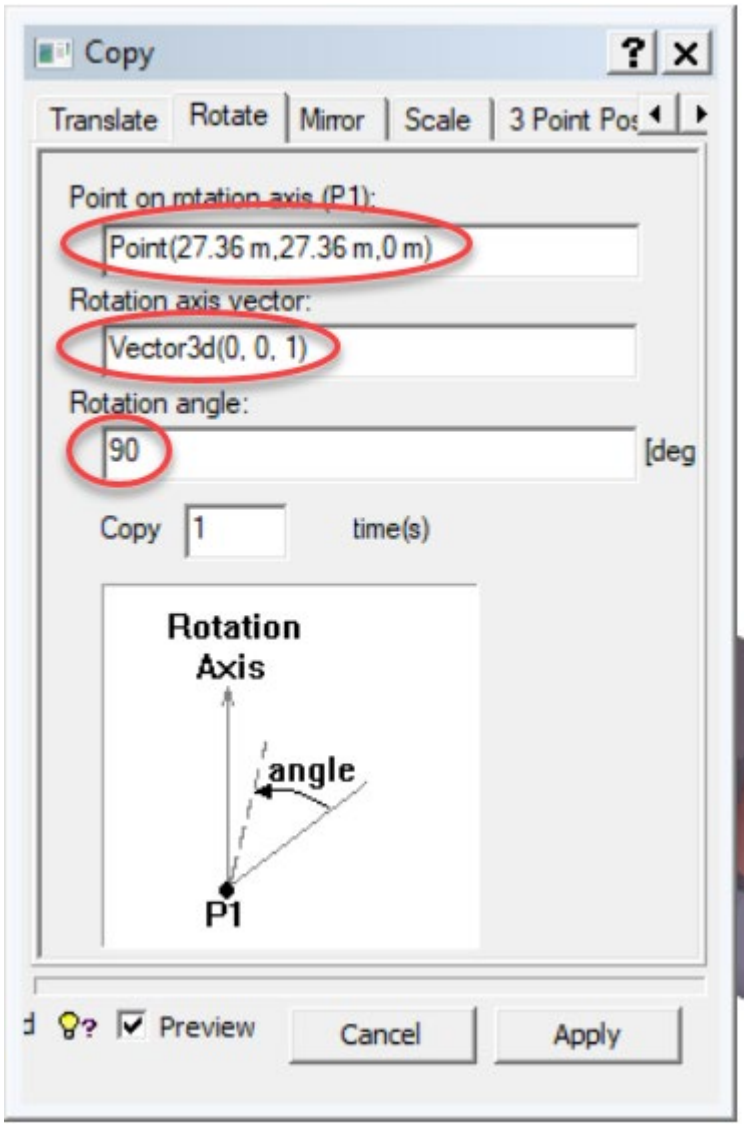
- Create an internal bulkhead for the column by first creating a plate extending outside the column and then trimming this.
  - Right-click the plate to *Divide* as shown.

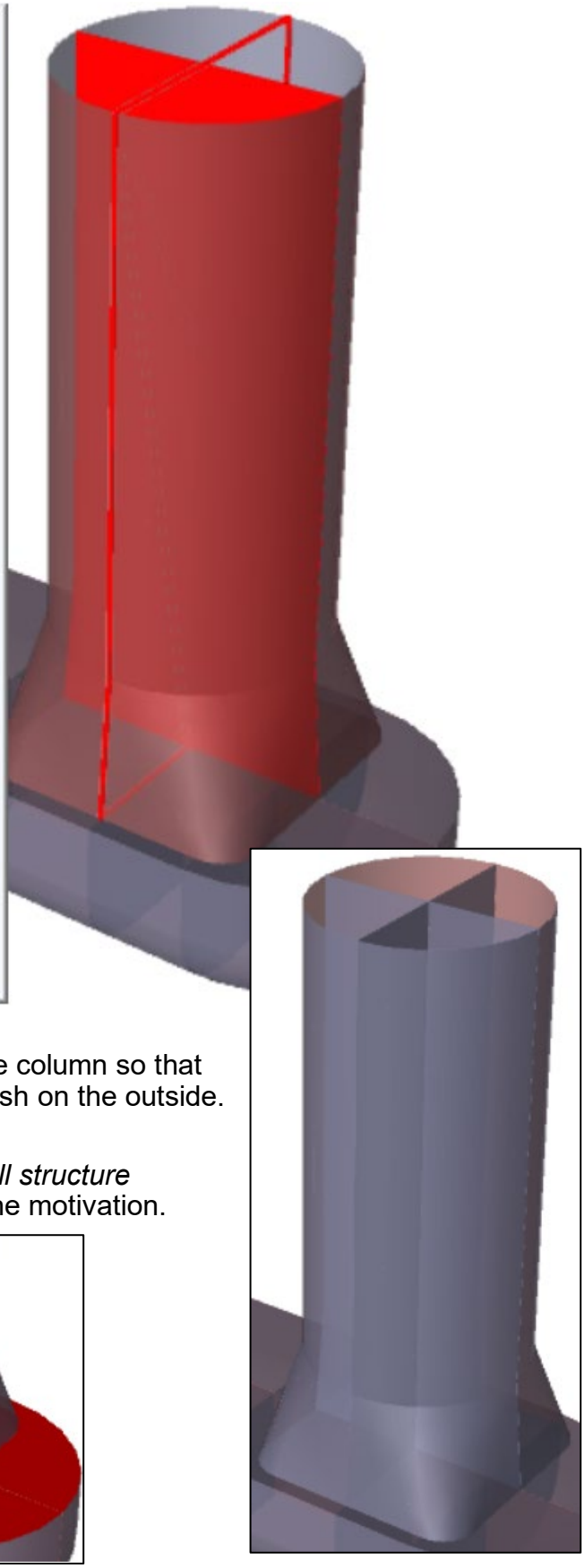


  - Select the superfluous parts and delete these.



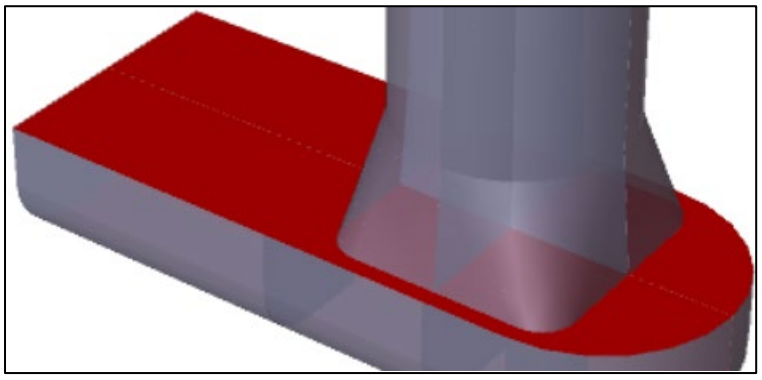
➢ Copy the remaining part inside the column, i.e. the bulkhead, by rotating it 90 degrees.





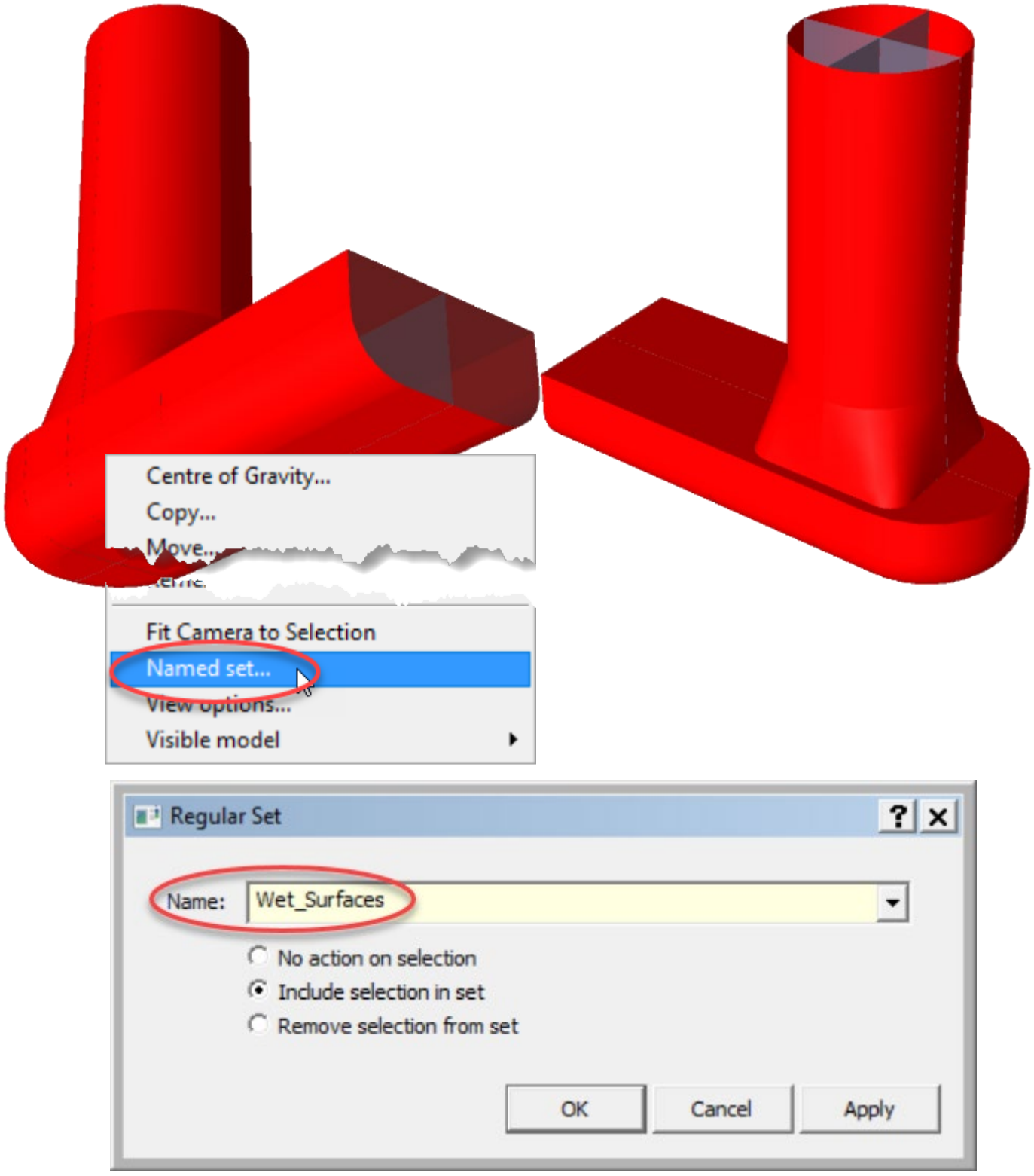
➢ Flip surface normals for external surfaces of the column so that all normals point consistently outwards, i.e. bluish on the outside.

➢ Divide the top plates of the pontoon (*Explode all structure in selection into simpler parts*). See below for the motivation.



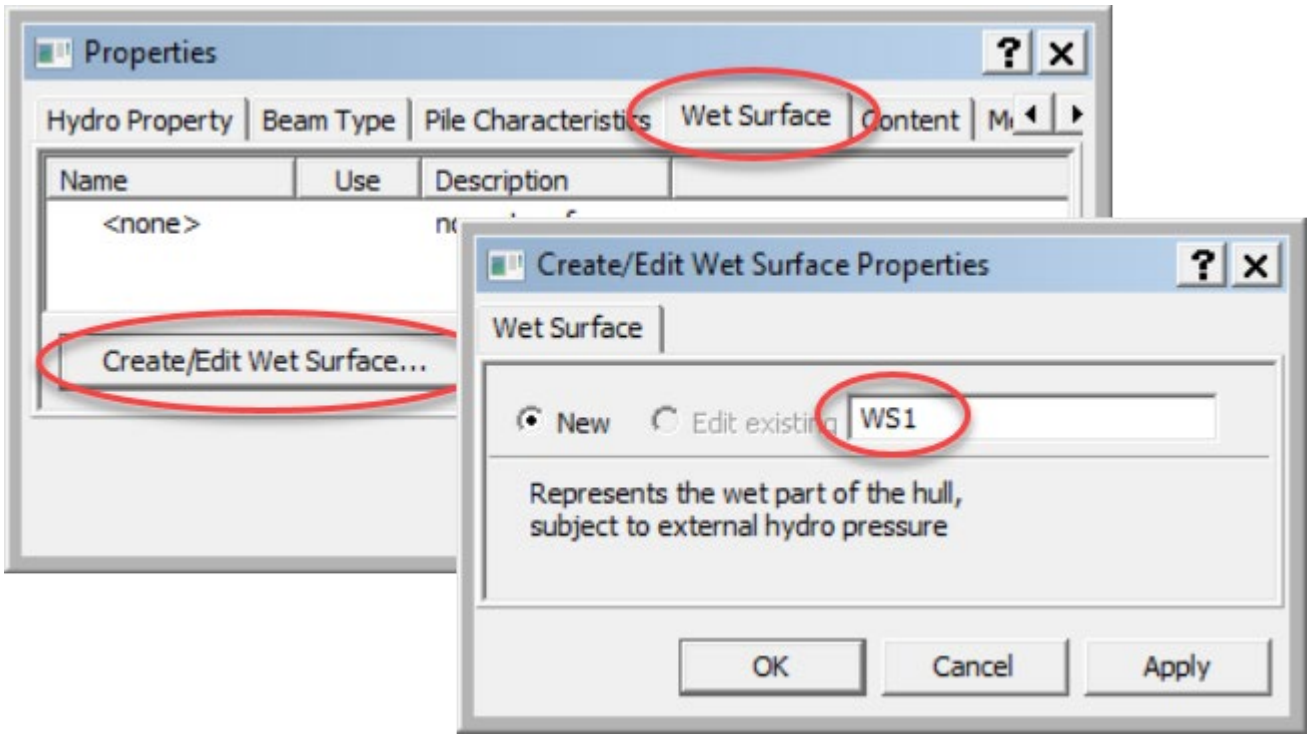
6 CREATE PANEL MODEL (T1.FEM)

- Select all wet surfaces (those subjected to water) and put them into a set named Wet_Surfaces.

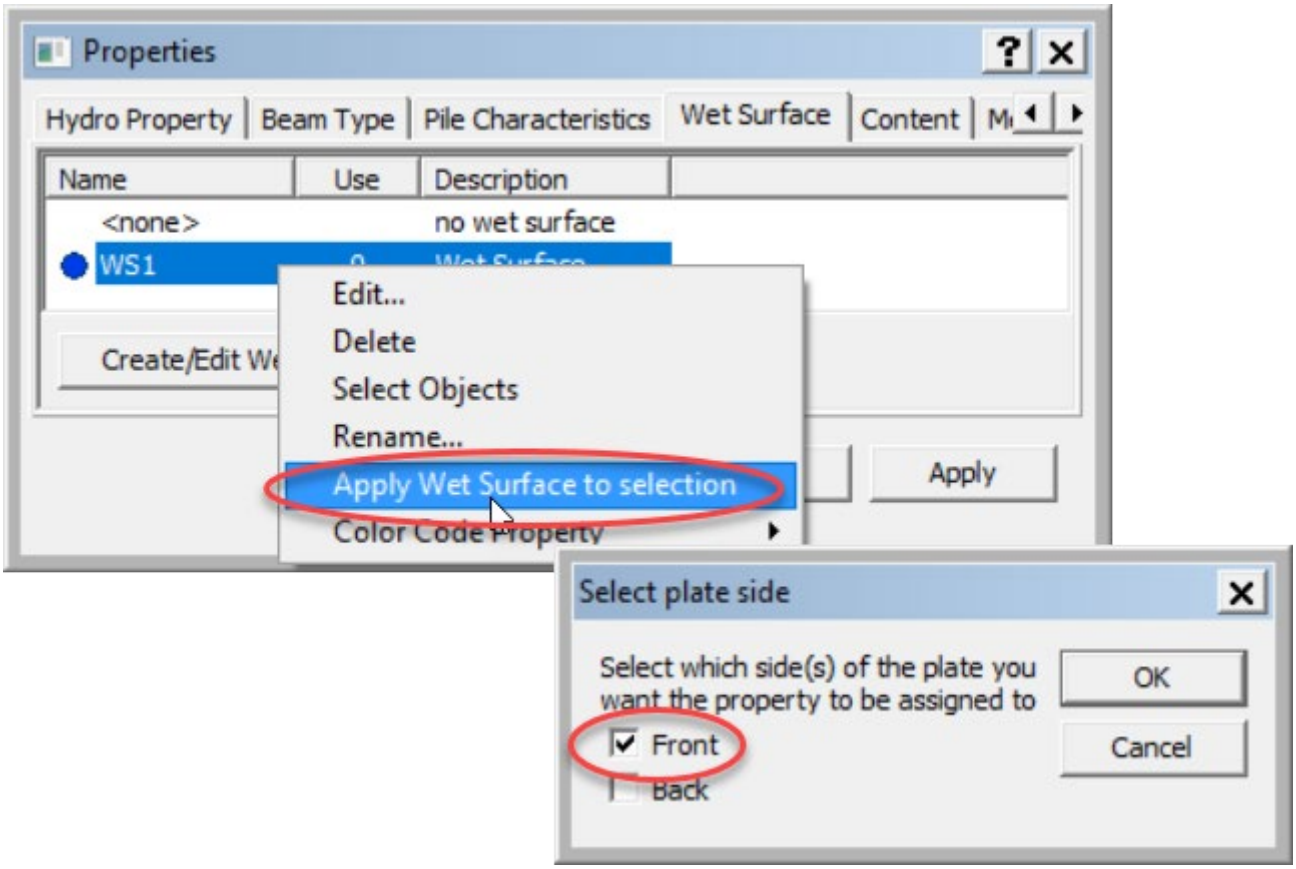


- Use *Edit* | *Properties* | *Mesh Property* to create a mesh property with *Element Length* = 5 m and assign this to the new set.

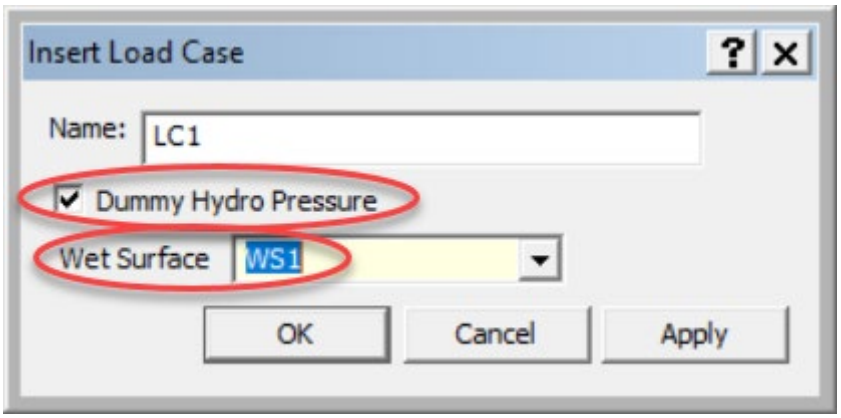
➢ Use *Edit* | *Properties* | *Wet Surface* to create a wet surface property named WS1.



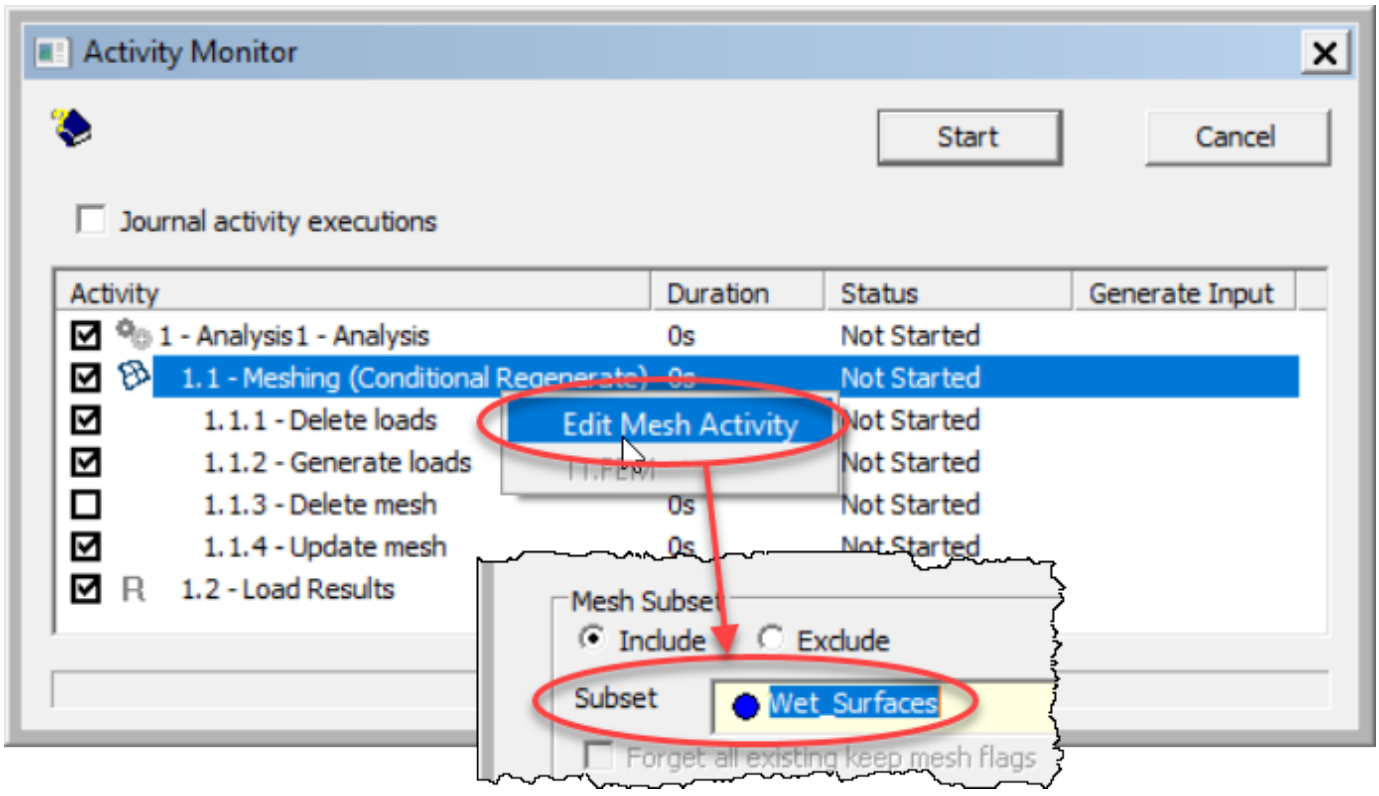
➢ Assign this wet surface property to the set named Wet_Surfaces and select the *Front* side, i.e. the outside surfaces.



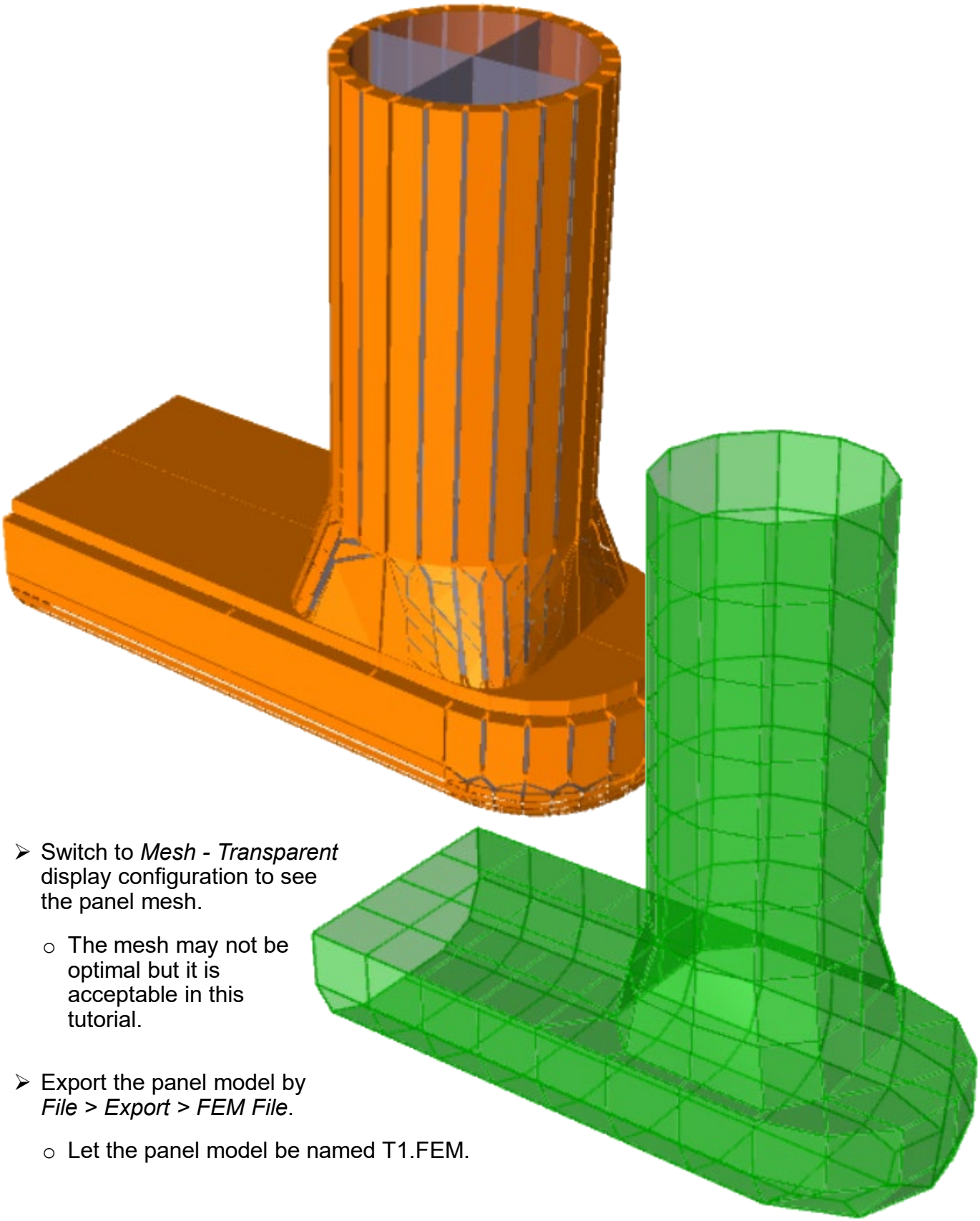
- Create a so-called *Dummy Hydro Pressure* load case. This refers to the wet surfaces and is used by HydroD/Wadam to identify surfaces for which hydrodynamic loads shall be calculated. This load case is hence an essential part of the panel model.
  - Use *Loads* | *Load Case* to open the *Insert Load Case* dialog, check *Dummy Hydro Pressure* and select the wet surface recently created.



- Create a panel mesh for the wet surfaces only.
  - Use *Mesh & Analysis* | *Create Mesh* (or Alt+M) to create an analysis activity including only meshing.
  - In the *Activity Monitor* right-click the meshing activity to *Edit Mesh Activity*.
  - In the *Mesh activity* dialog select the set Wet_Surfaces.



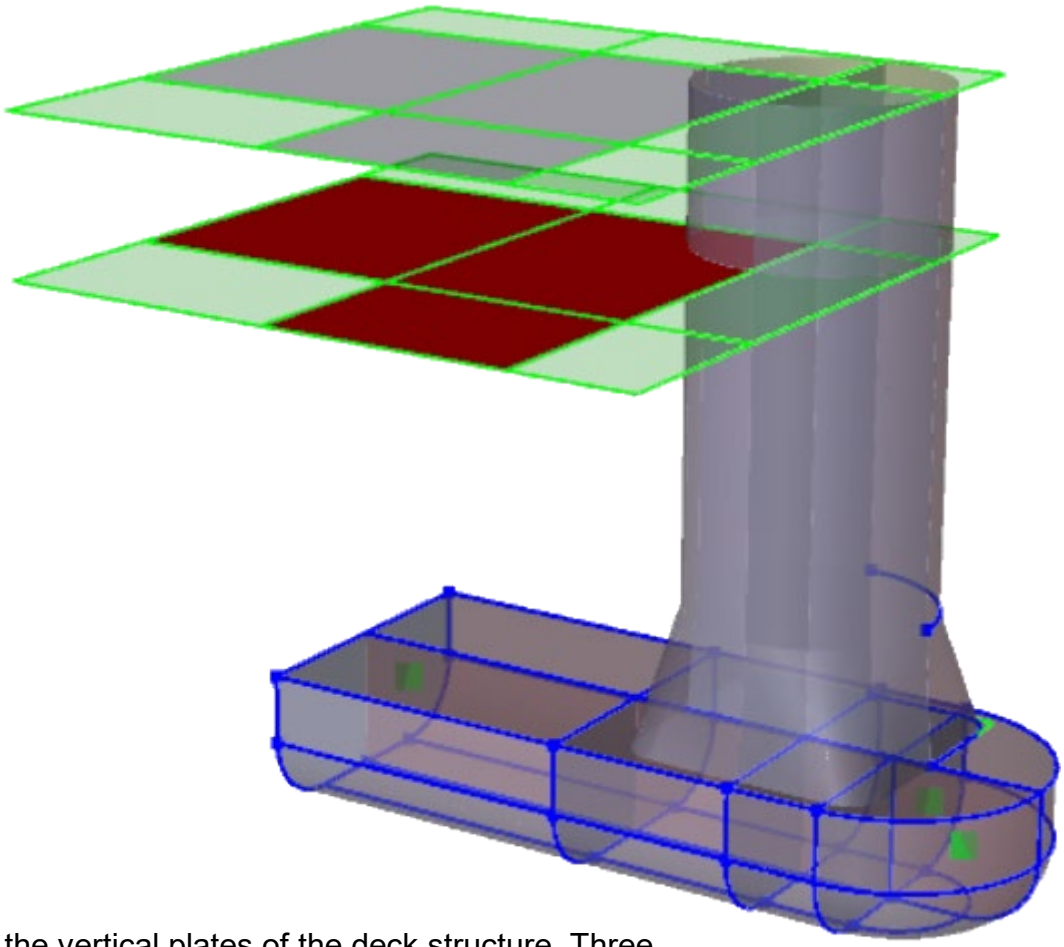
- Click *Start* in the *Activity Monitor* and see that the dummy load case is displayed on the concept model.



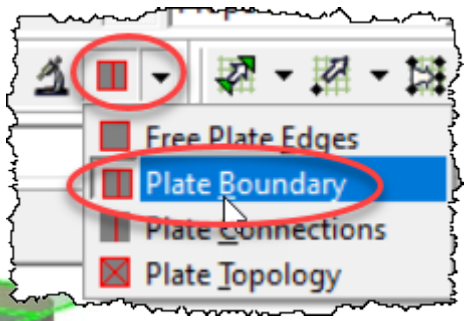
- Switch to *Mesh - Transparent* display configuration to see the panel mesh.
  - The mesh may not be optimal but it is acceptable in this tutorial.
- Export the panel model by *File > Export > FEM File*.
  - Let the panel model be named T1.FEM.

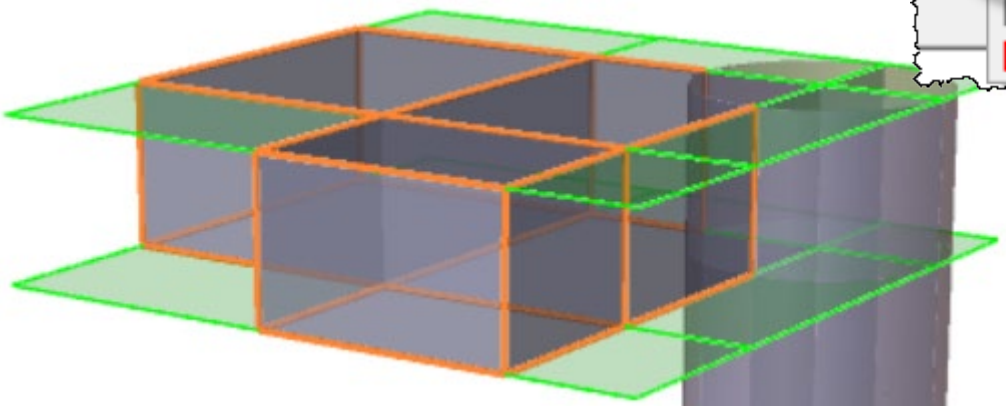
7 CREATE FULL MODEL

- Before creating deck plates set the default thickness to Th35.
  - Create the three horizontal plates of the lower deck and copy these to the upper deck.
  - Change the thickness to Steel and material to Th03 for the three plates of the upper deck. (These plates will have girders and stiffeners and shall therefore have normal material and thickness.)

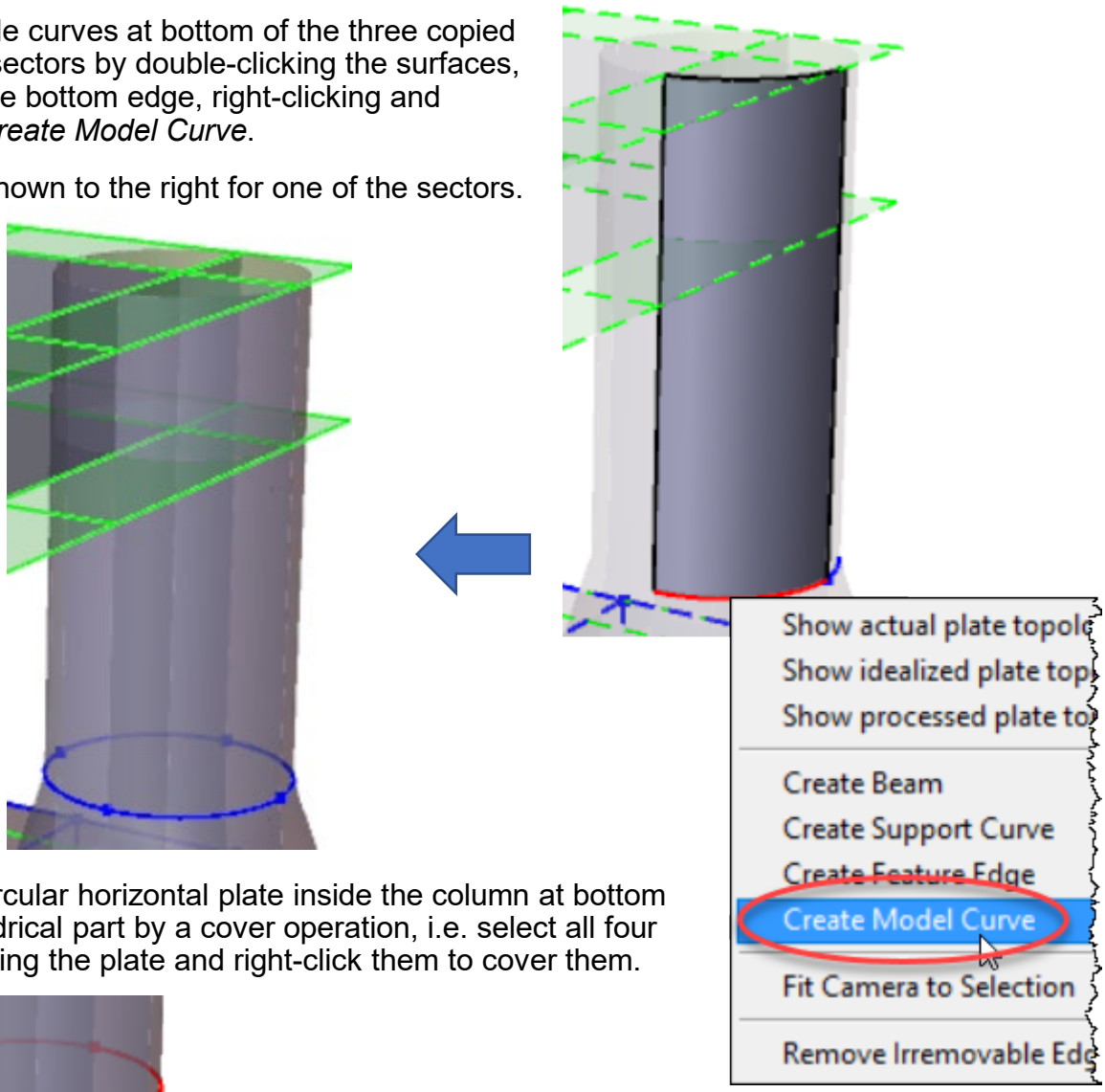


- Create the vertical plates of the deck structure. Three vertical plates in both the XZ-plane and YZ-plane. The plates stop at the column cylinder wall (there are already vertical plates inside the column).
  - *Plate Boundaries* are shown to highlight the six plates to create.

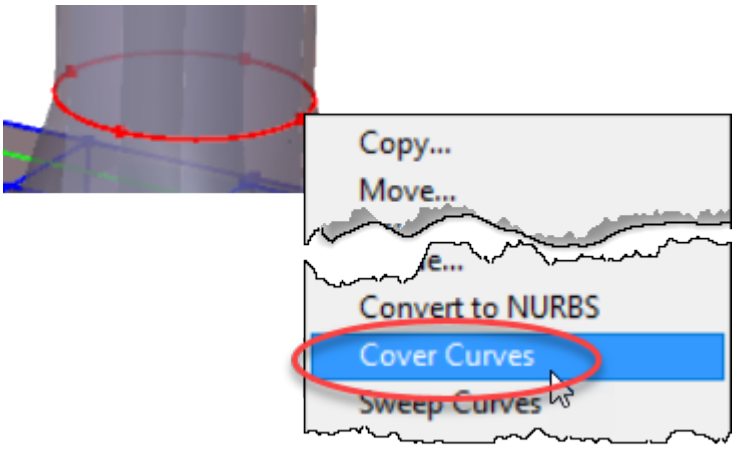




- Create guide curves at bottom of the three copied cylindrical sectors by double-clicking the surfaces, selecting the bottom edge, right-clicking and selecting *Create Model Curve*.
  - This is shown to the right for one of the sectors.



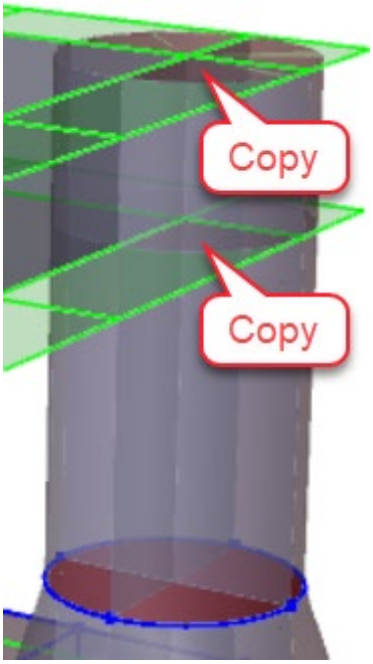
- Create a circular horizontal plate inside the column at bottom of the cylindrical part by a cover operation, i.e. select all four arcs enclosing the plate and right-click them to cover them.



- Copy this circular plate up to the lower deck. Note that the 90° sector overlapping the existing plate of the deck will not be copied, i.e. only 3/4 of the circular plate will be copied.

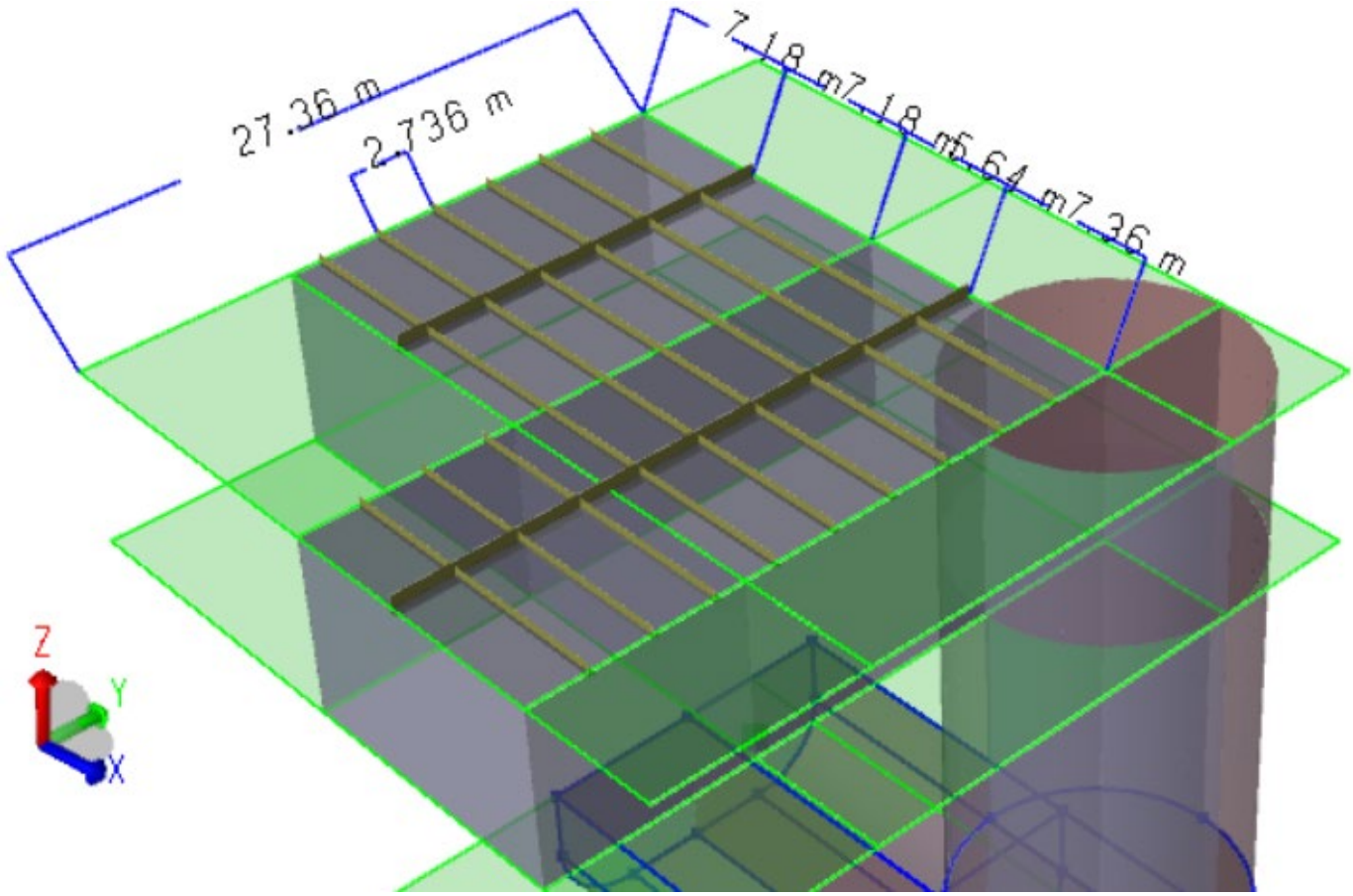
- Copy the circular plate up to the upper deck.

- The result is shown to the right.

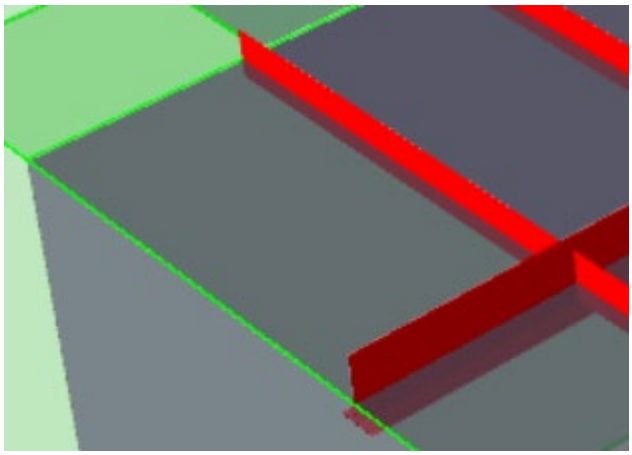


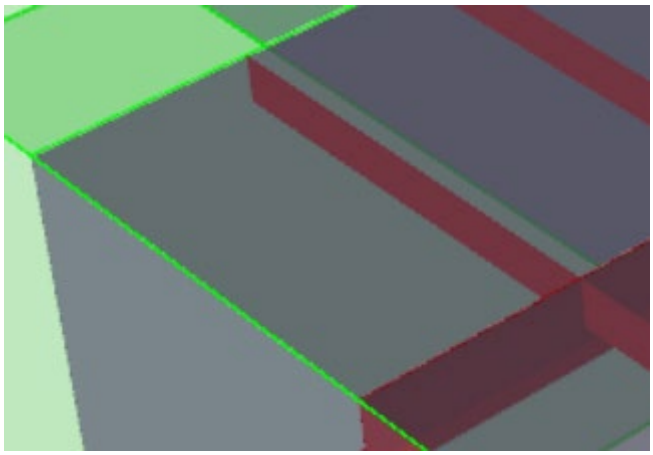
- Set default material to Steel.

- Create girders and stiffeners in the top deck.
  - The girders (cross section Girder) in Y-direction are positioned as shown.
  - The 9 stiffeners (cross sections Stiffener) in the X-direction are evenly spaced over the deck width of 27.36, i.e. a spacing of 2.736.



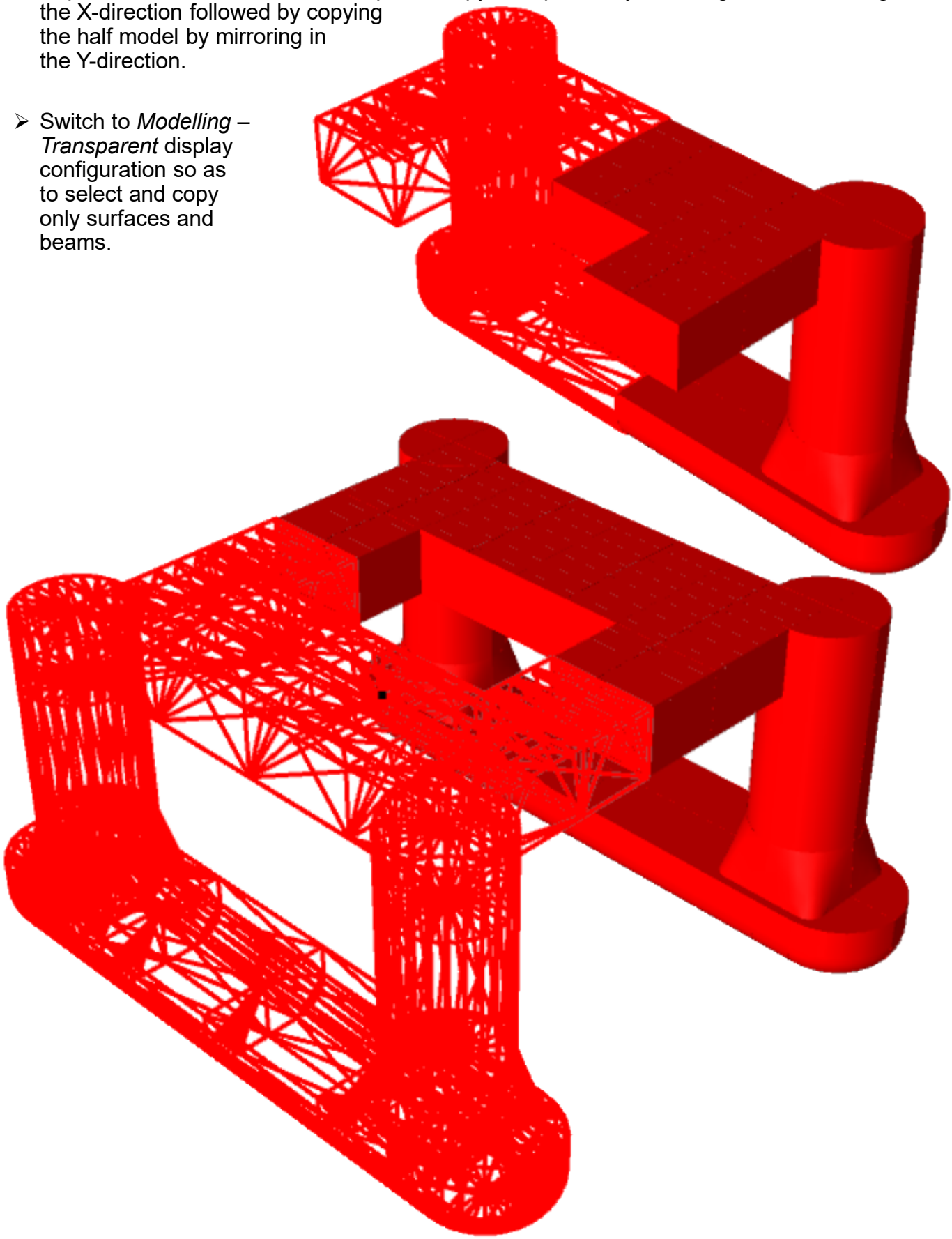
- Flush all beams at top with the plate. Before to the left and after to the right.





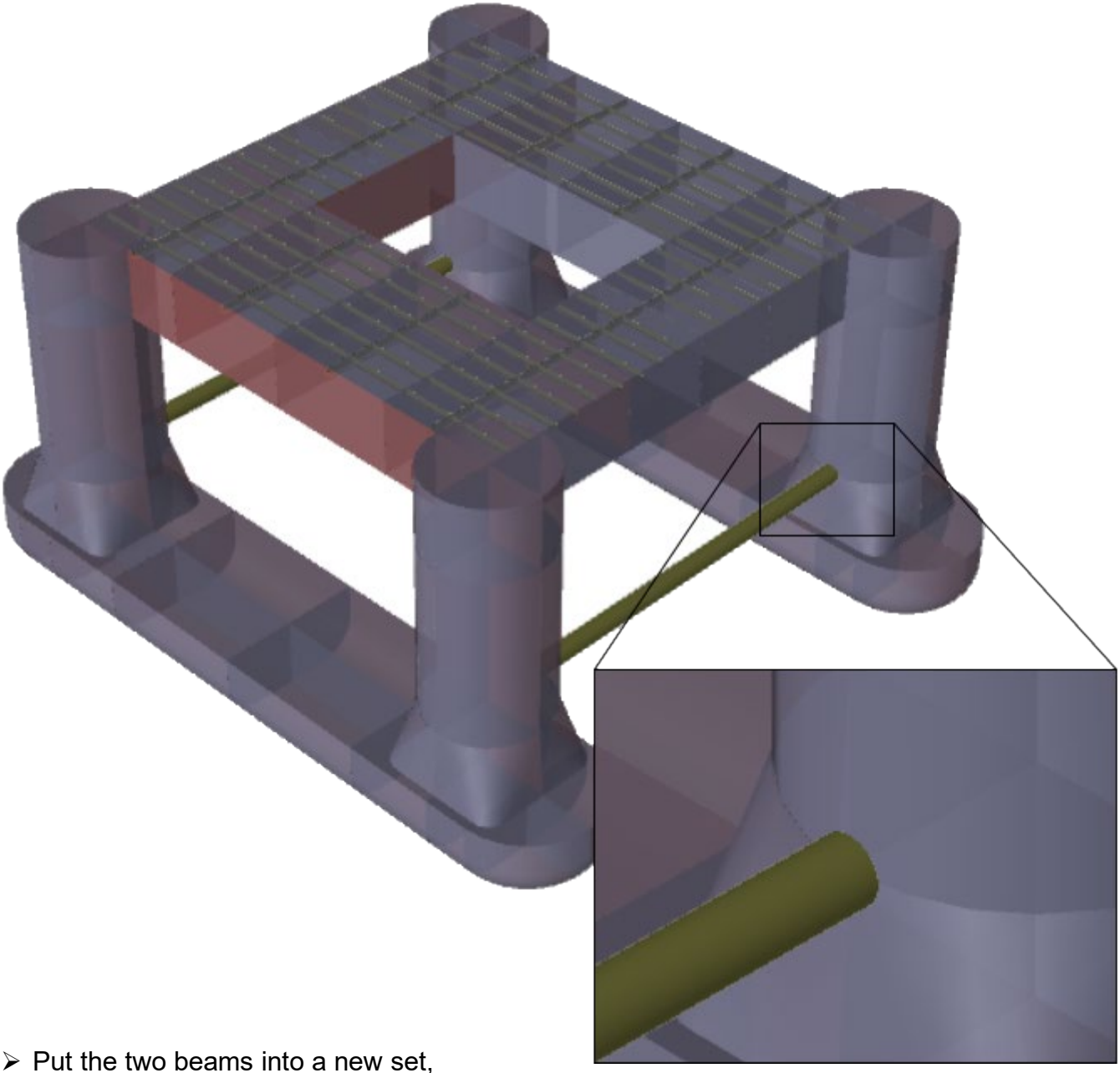
- A quarter of the model is now complete. Copy this quarter by mirroring it about the origin in the X-direction followed by copying the half model by mirroring in the Y-direction.

- Switch to *Modelling – Transparent* display configuration so as to select and copy only surfaces and beams.



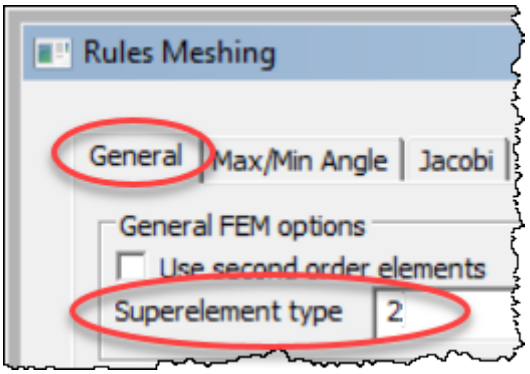
8 CREATE MORISON MODEL (T2.FEM)

- The columns are connected by two horizontal braces. These are modelled as beam elements with section Pipe1 and material Steel. The braces connect with the columns at the top of the conical transition as show below.



- Put the two beams into a new set, named e.g. MorisonBeams.

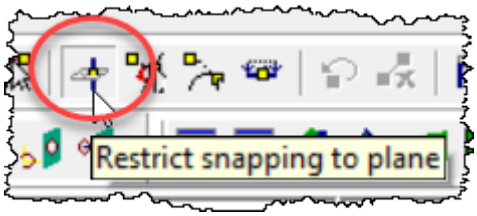
- Use *Edit* > *Rules* > *Meshing Rules* to set the superelement number to 2.

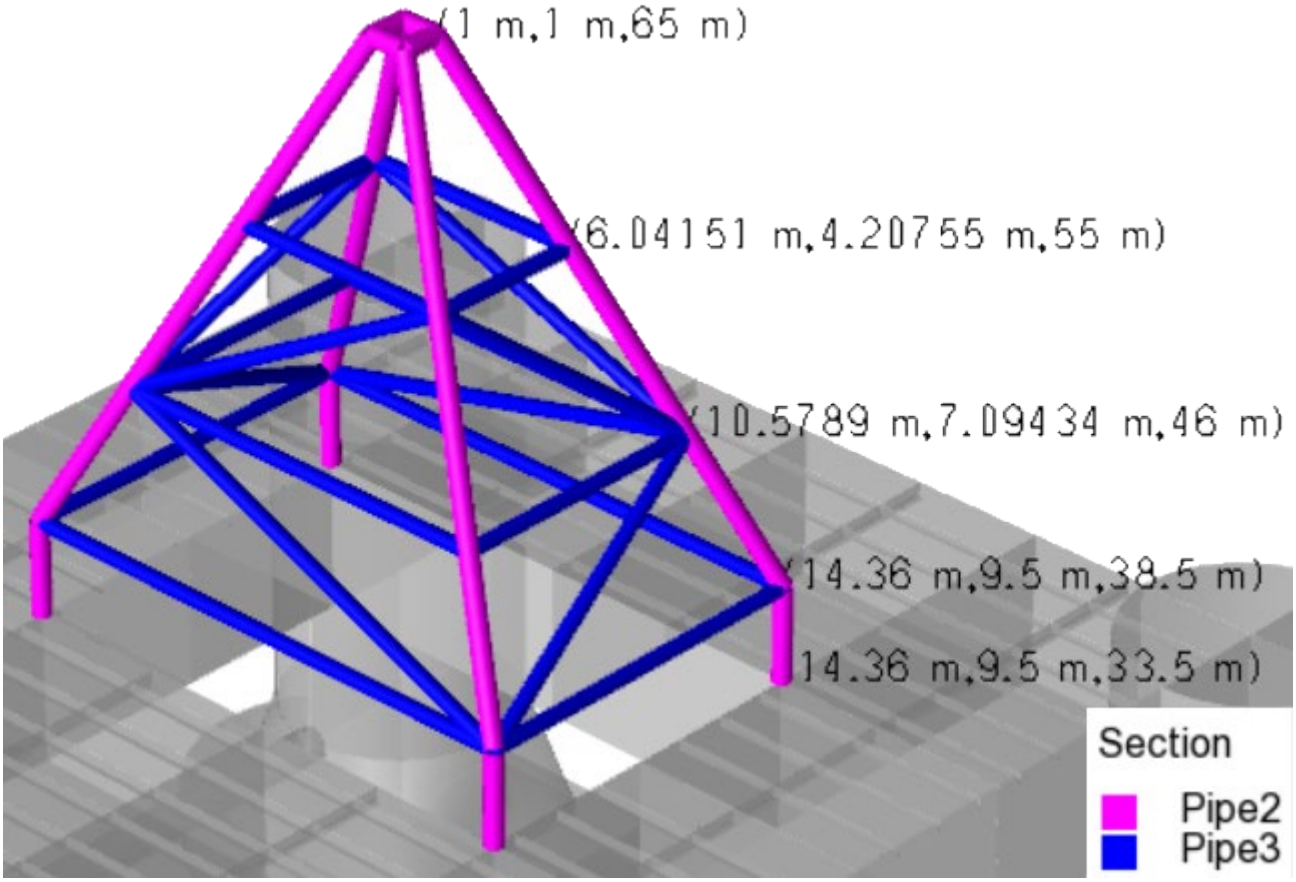


- Open the existing analysis activity, edit the meshing activity to select the set MorisonBeams for meshing only, create a mesh for the two beams (no mesh density produces a single beam element per brace) and export as T2.FEM.

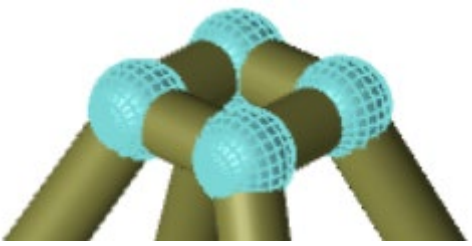
9 CREATE DERRICK

- Create the derrick shown below, using, for example, the following procedure:
  - Start with one the lower vertical legs with cross section Pipe2 and height 5.
  - Continue with the sloping leg, also cross section Pipe2.
  - Form the three remaining sets of legs by two mirror copying operations.
  - Add the four short horizontal beams on top, again with Pipe2.
  - Add the horizontal bracings with cross section Pipe3. Use the snap plane functionality for the two upper elevations.
  - Add the diagonal bracings with cross section Pipe3.



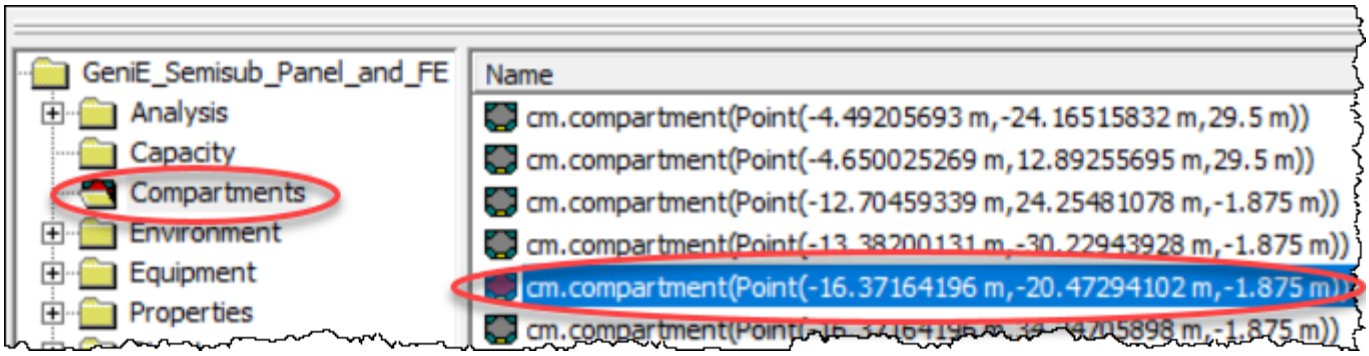


- Use *Structure* | *Point Mass* to add four mass points at top of the derrick, each with mass 2.0E5 kg.

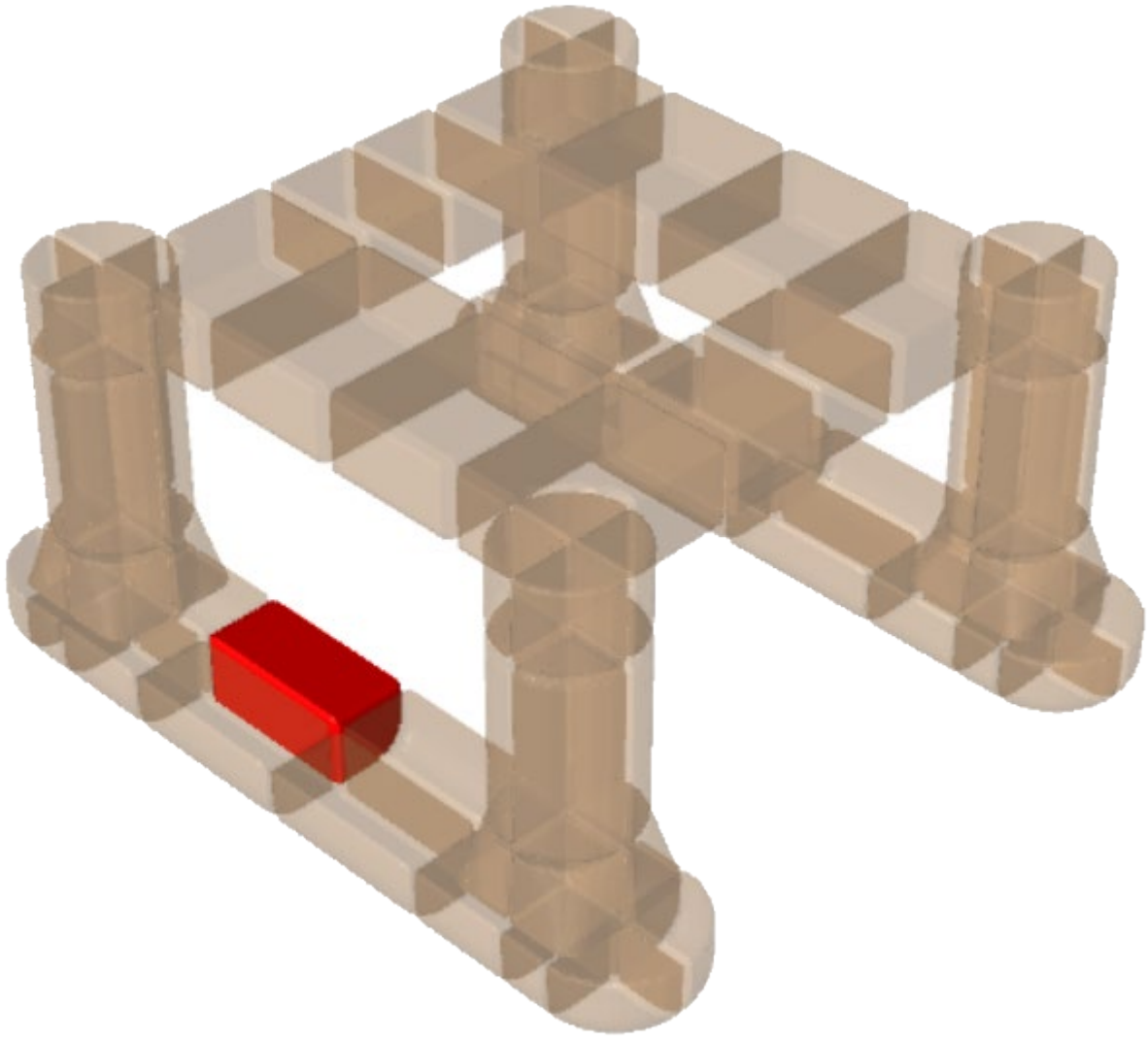


10 CREATE COMPARTMENTS

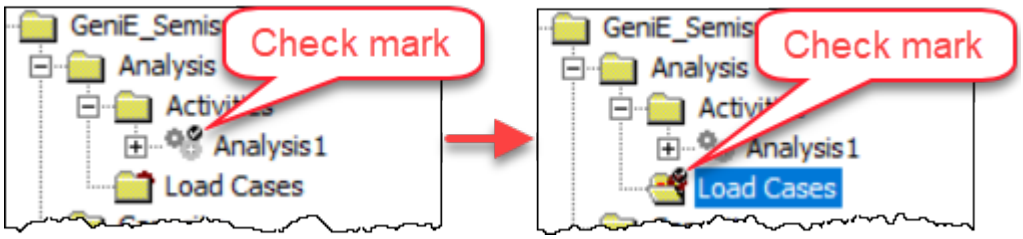
- The pontoons have ballast tanks filled with water. These are created as compartments in GeniE and taken into account in the subsequent hydrodynamic analysis in HydroD/Wadam.
  - Use *Loads* | *Compartment* | *Compartment Manager* to open the *Compartment Manager* dialog and click *OK*.
  - By this action GeniE will search for *all* closed compartments in the entire model and list these in the *Compartments* folder as shown below. Compartments selected in the browser are highlighted in the display as shown further below.



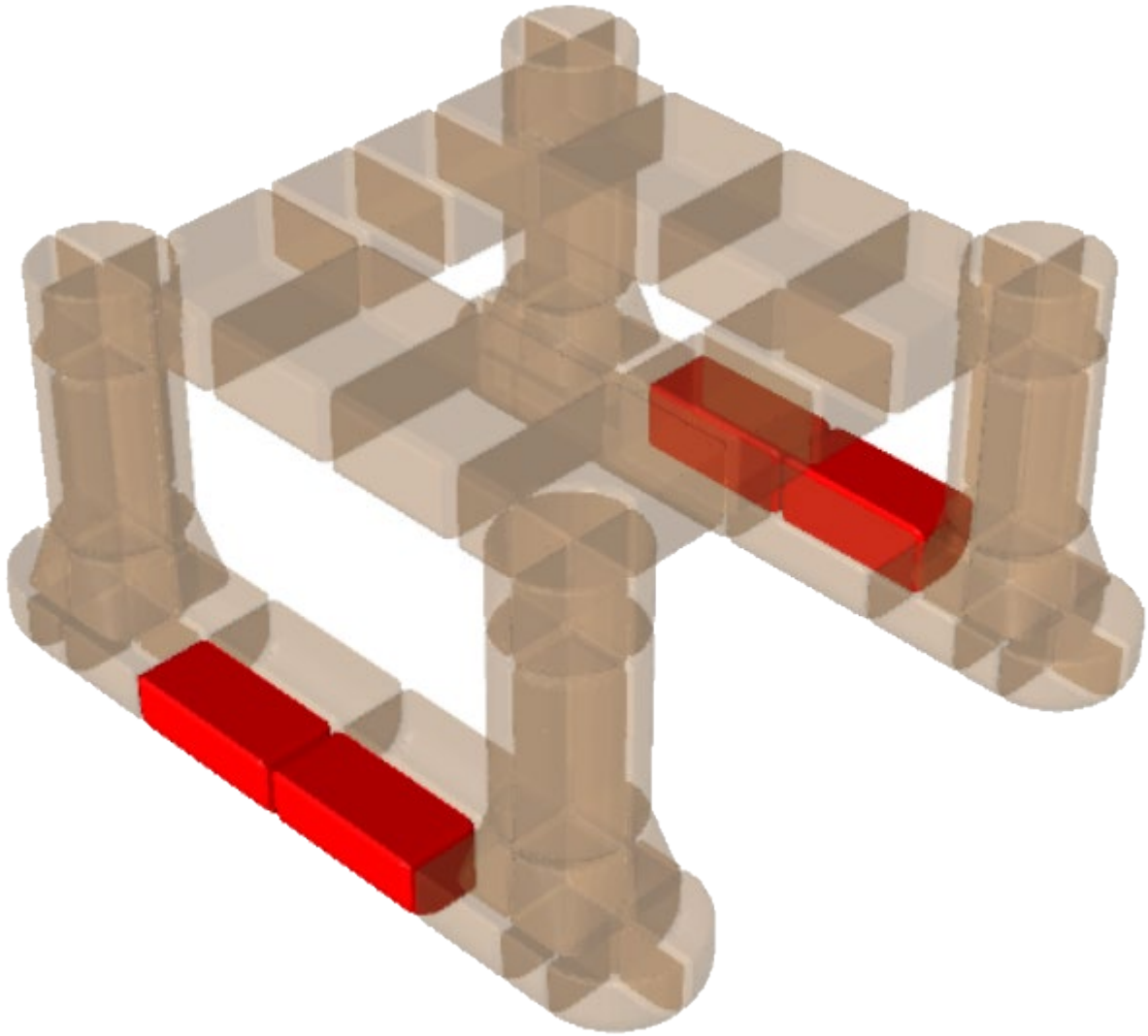
  - Switch to *Compartments* display configuration to see the compartments only.



- A separate load case must be defined for each compartment to be accounted for in HydroD/Wadam. It is a requirement that these load cases are numbered consecutively after the dummy hydro pressure load case for the external water pressure.
  - These compartment load cases should not pertain to any particular analysis activity. Right-click the *Analysis* | *Load Cases* folder to make this folder active rather than an analysis activity.

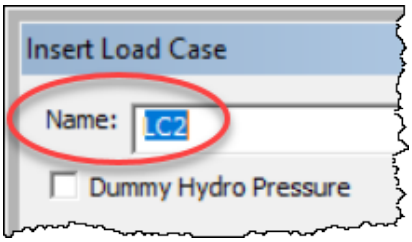


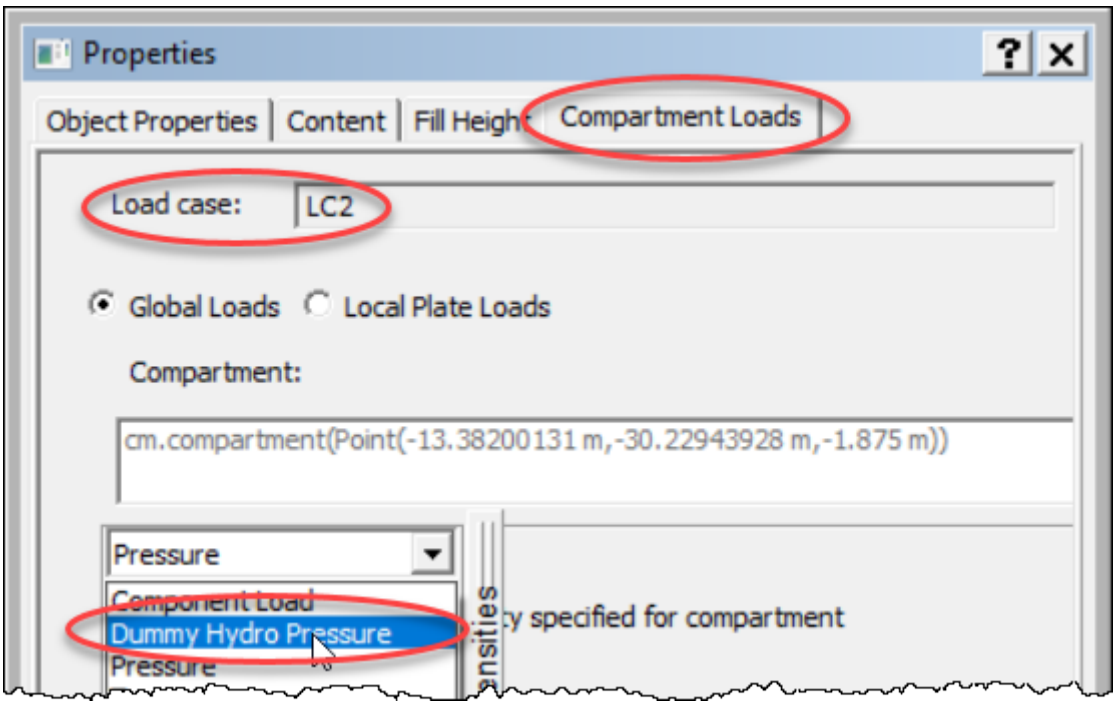
- In this tutorial we want the four pontoon compartments shown below to be accounted for in the HydroD/Wadam analysis.



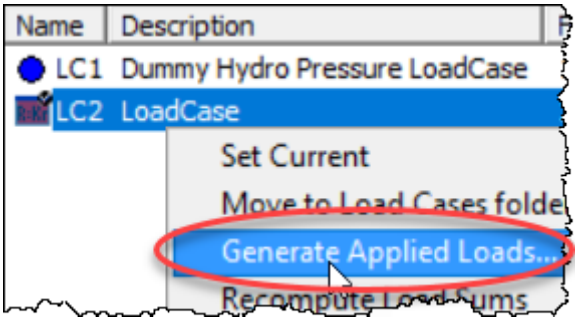
➢ The procedure for each of the four compartments is:

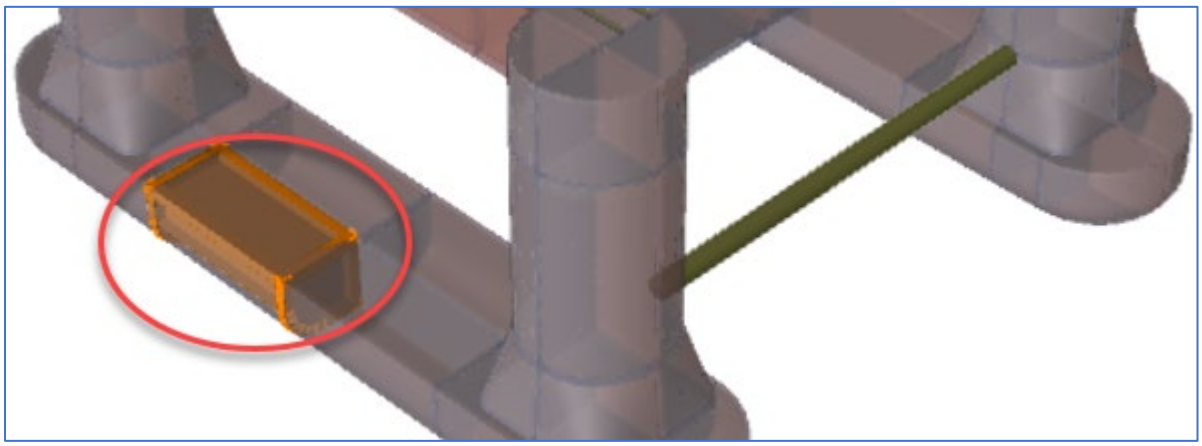
- Create a new load case that will then be the current load case. Do not check *Dummy Hydro Pressure*.
- Right-click a compartment to open the *Properties* dialog. In the *Compartment Loads* tab select *Dummy Hydro Pressure* as shown.





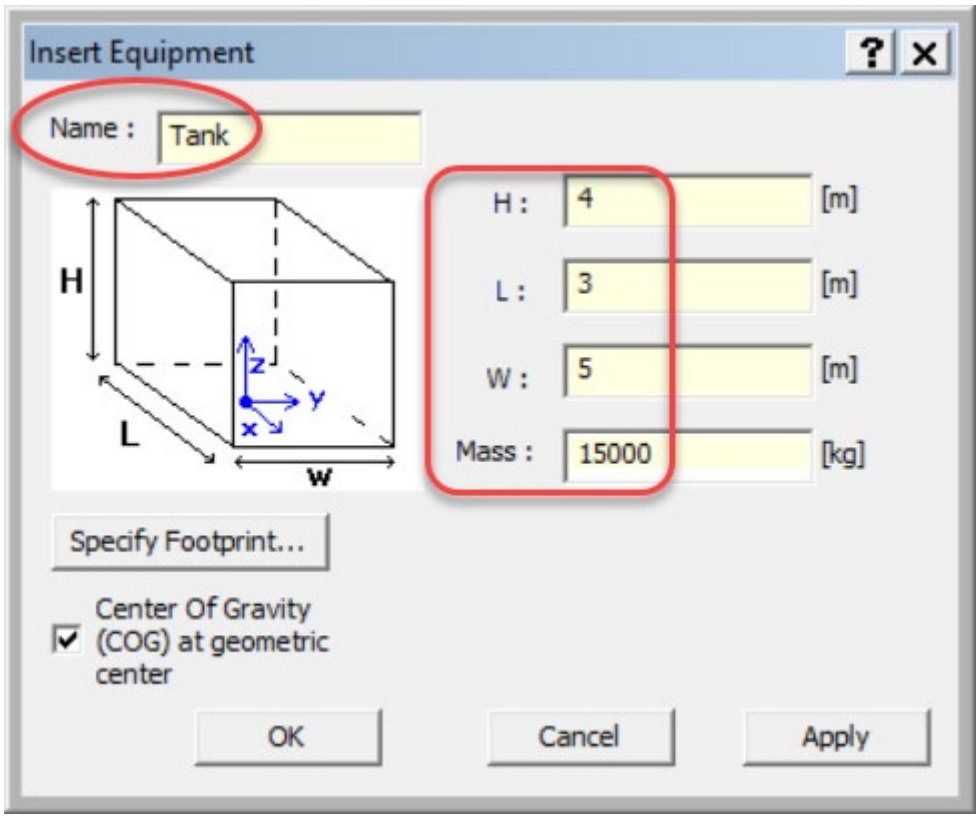
- Right-click the load case in the browser to *Generate Applied Loads*.
- Switch to *Modelling - Transparent* display configuration to see the dummy hydro pressure.



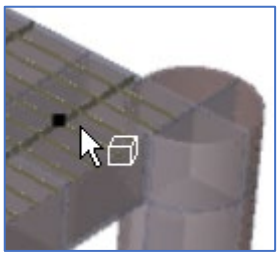


11 CREATE EQUIPMENTS

- There are four identical equipments on the upper deck. Use *Loads* > *Prismatic Equipment* to create an equipment as shown.

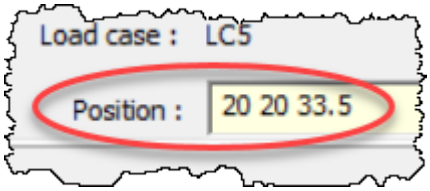


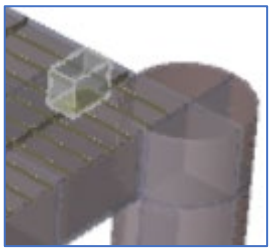
- Equipments are placed in the model in the context of a load case.
  - As explained later, the equipments shall in this tutorial be represented as mass rather than line loads. This is because the FE model (T3.FEM to be created later) will be used as mass model in a hydrodynamic analysis in addition to as structural model in a structural analysis.
  - When an equipment is represented as load case independent mass, it is in effect removed from the load case in which it was placed. For this reason one of the compartment load cases (LC2 – LC5) may be used for placing the equipment rather than creating an additional load case.
  - Set one of the compartment load cases, e.g. LC5, as current.



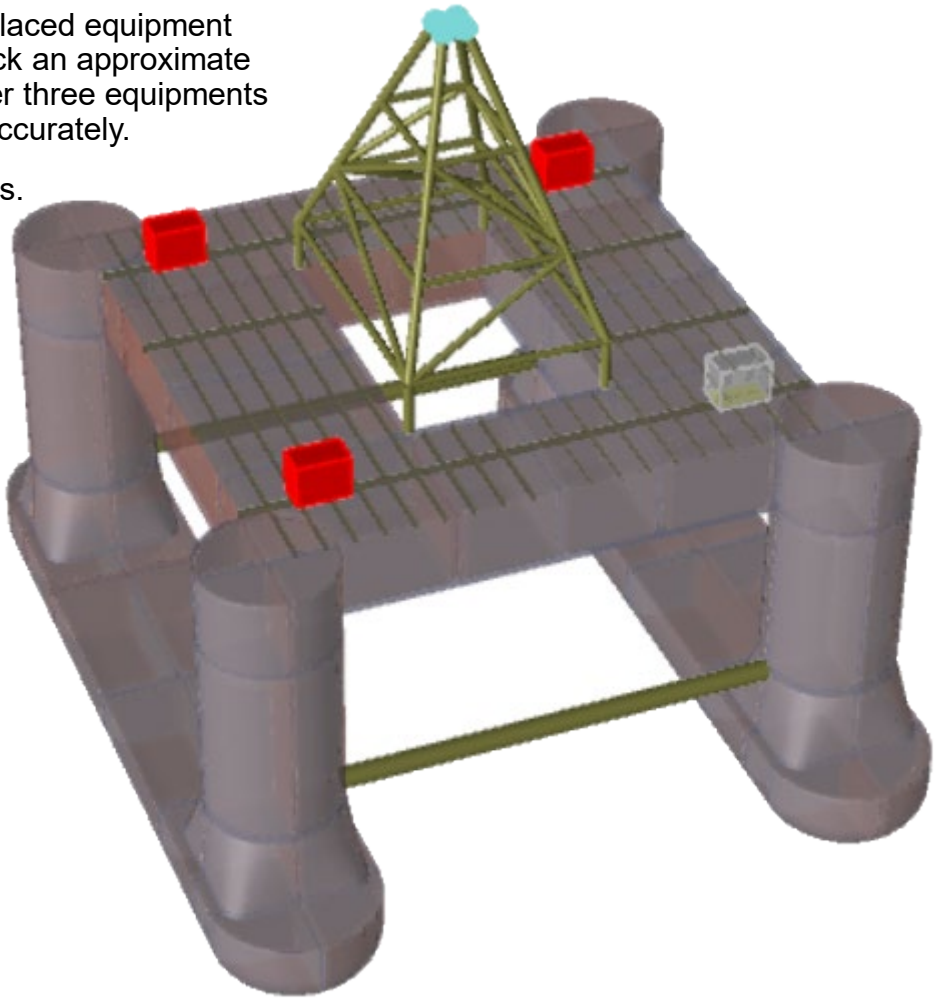
- Select the equipment in the *Equipment* folder in the browser, right-click and select *Place in Loadcase*.
  - Click an approximate position for the equipment.

- Select the placed equipment, right-click and select *Properties*.
  - In the *Object Properties* tab of the *Properties* dialog correct to *Position* (20, 20, 33.5).



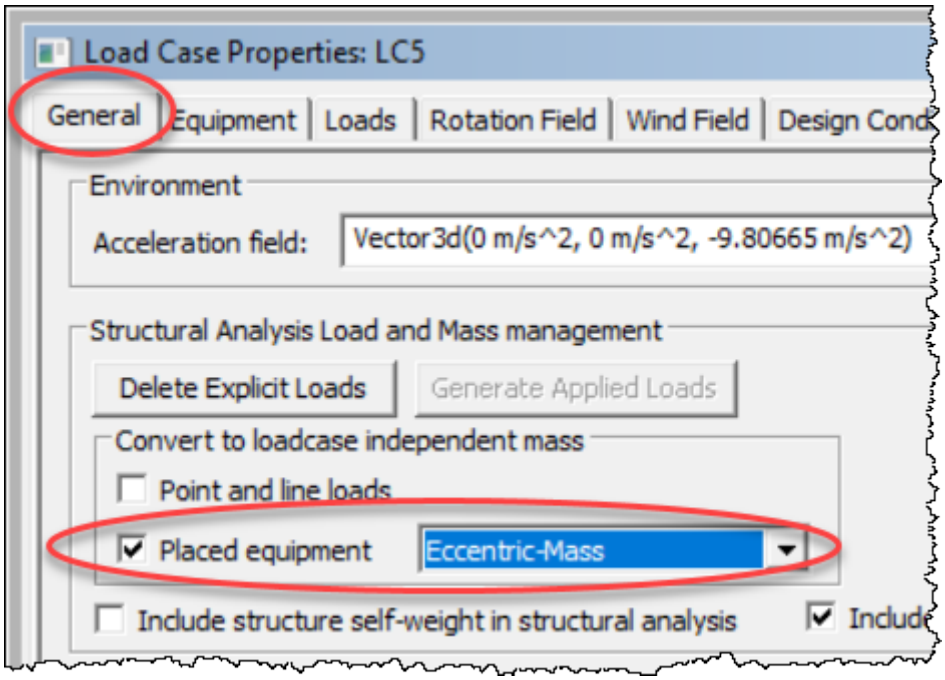


- Select and right-click the placed equipment to select *Place a copy*. Click an approximate position for one of the other three equipments followed by positioning it accurately.
  - Do so for all three copies.
  - The proper positions are:
    - (20, -20, 33.5)
    - (-20, 20, 33.5)
    - (-20, -20, 33.5)



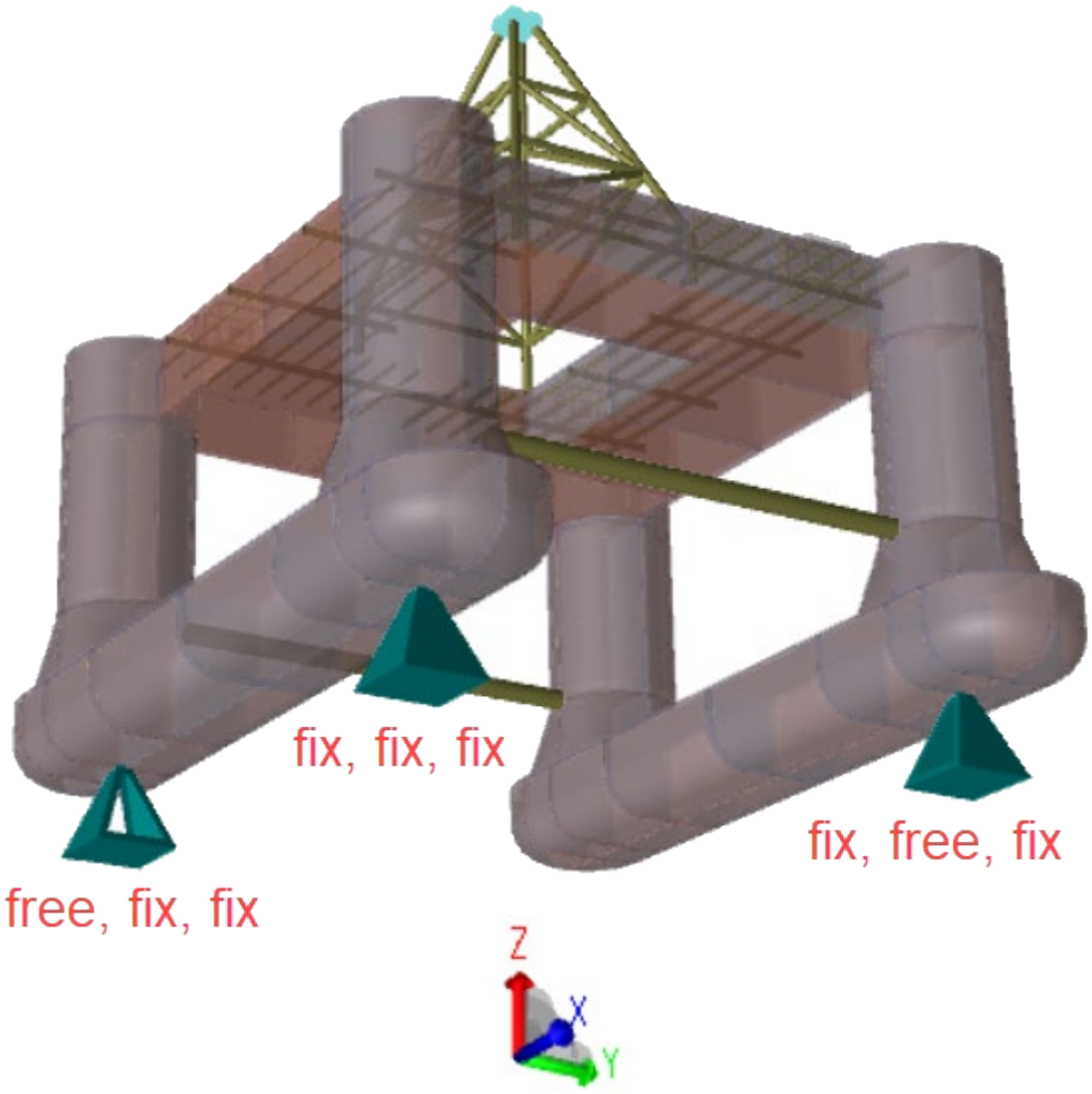
- The three copies are highlighted to the right.

- In the *Load Cases* folder of the browser, select the load case containing the equipment (LC5?), right-click and select *Properties*. In the *General* tab check *Placed equipment* and select *Eccentric-Mass* from the pulldown menu.



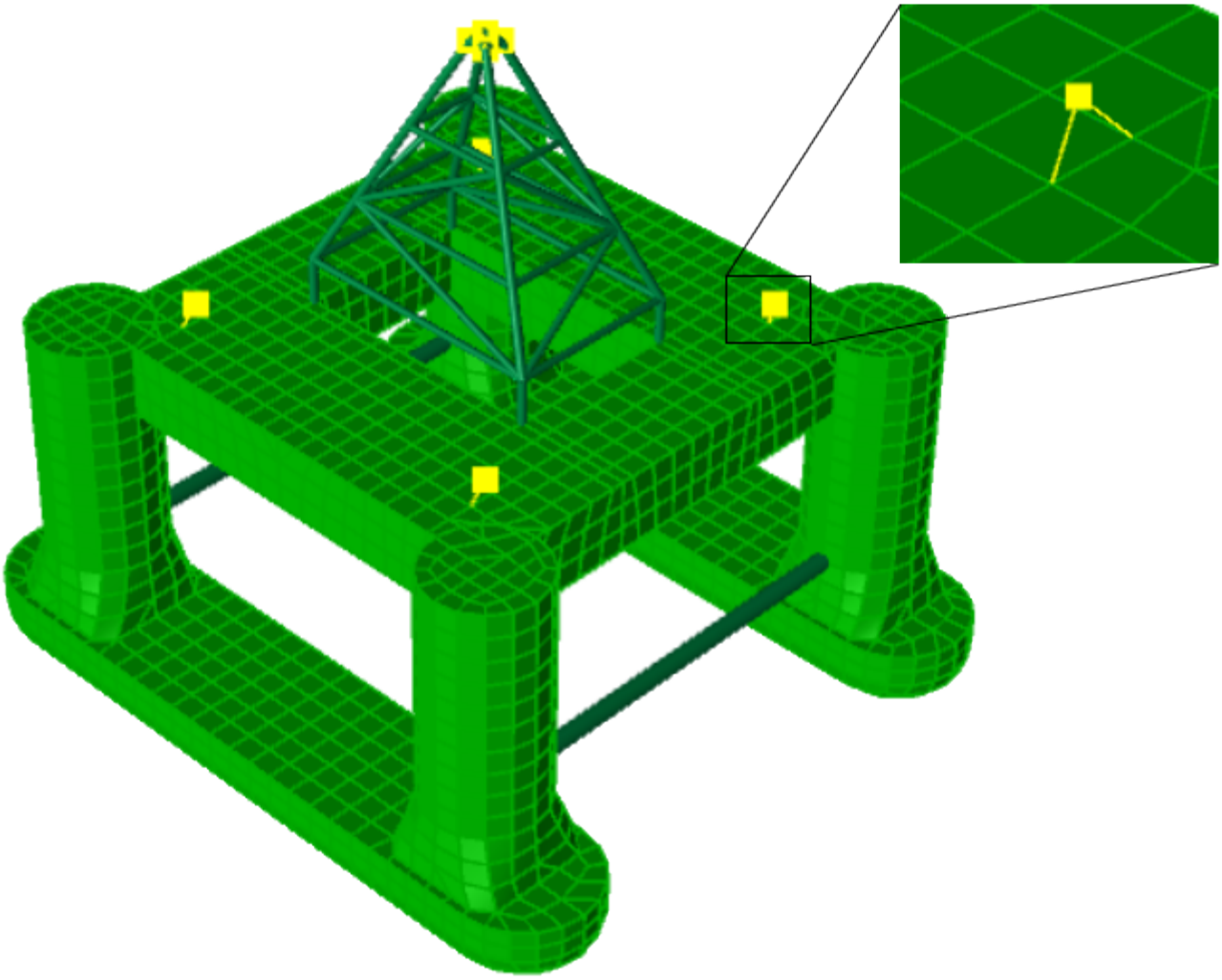
12 ADD SUPPORT POINTS

- Even though the semisubmersible is floating, the FE structural model must be supported against rigid body displacements or else the structural analysis will fail. These supports will have no effect on the hydrodynamic analysis in HydroD/Wadam. The hydrodynamic analysis should balance the rigid body acceleration against the hydrodynamic forces so that the reaction forces in the supports should be approximately zero.
  - Add the three support points shown below. All rotational degrees of freedom for all three are free. The fixation of the translational degrees of freedom is indicated in the figure.



13 CREATE FE STRUCTURAL MODEL (T3.FEM)

- Use *Edit* | *Properties* and open the *Mesh Property* tab to create a new mesh property with *Element Length* 3 m.
  - Assign this density to all plates and shells, i.e. to the whole model except the beams.
- Use *Edit* > *Rules* > *Meshing Rules* to set the superelement number to 3.
- Open the *Activity Monitor* of the existing analysis (Alt+D) and edit the meshing activity to set *Mesh Subset* to <*None*>. I.e. the complete model will be meshed. Run the analysis to create a FE mesh.
- Notice the eccentric masses representing the equipments.



- Export the model as name T3.FEM.
- The FE mesh is rather coarse and there are some badly shaped elements. In a realistic analysis, efforts should be put into refining the mesh in areas and improving element shapes.

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