

SESAM TUTORIAL

GeniE

Ship Cargo Rail

Valid from program version 8.3



Sesam Tutorial

GeniE – Ship Cargo Rail

Date: September 2021

Valid from GeniE version 8.3

Prepared by: Digital Solutions at DNV

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

E-mail sales: digital@dnv.com

© DNV AS. All rights reserved

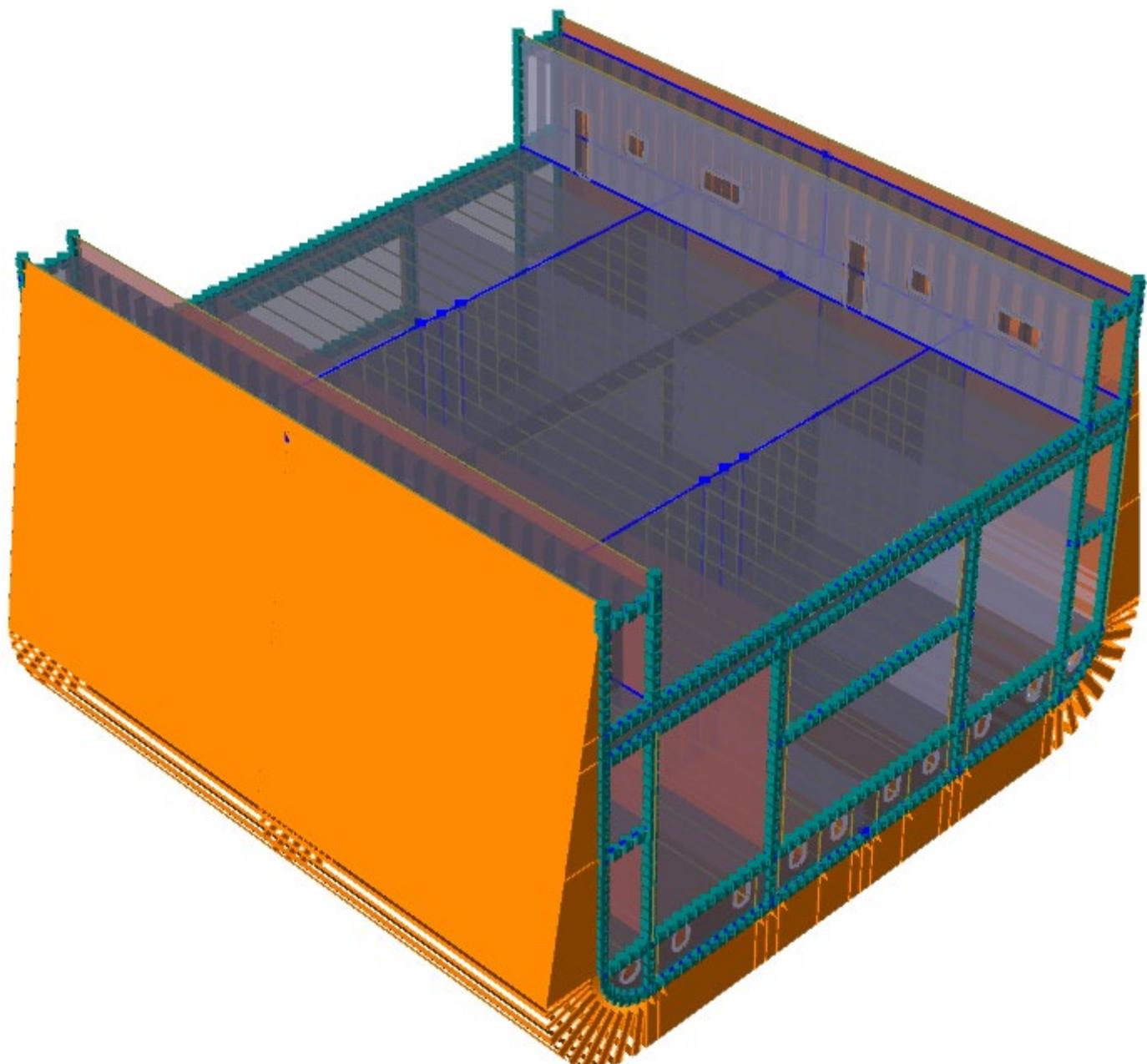
This publication or parts thereof may not be reproduced or transmitted in any form or by any means, including copying or recording, without the prior written consent of DNV AS.

TABLE OF CONTENTS

1. Introduction	Page 4
2. Units, Material, Cross Sections, Plate Thicknesses	Page 5
3. Create Guiding Geometry	Page 12
4. Create Plate with Stiffeners at X = -10 m	Page 15
5. Create Web Frame with Stiffeners at X = -5 m	Page 19
6. Copy Web Frame with Stiffeners	Page 26
7. Create Hull	Page 27
8. Modify Thicknesses	Page 29
9. Create Decks with Stiffeners	Page 34
10. Create the Cargo Rail	Page 36
11. Copy the Model	Page 42
12. Introduce Boundary Conditions	Page 44
13. Create Loads	Page 45
14. Run the Analysis	Page 48

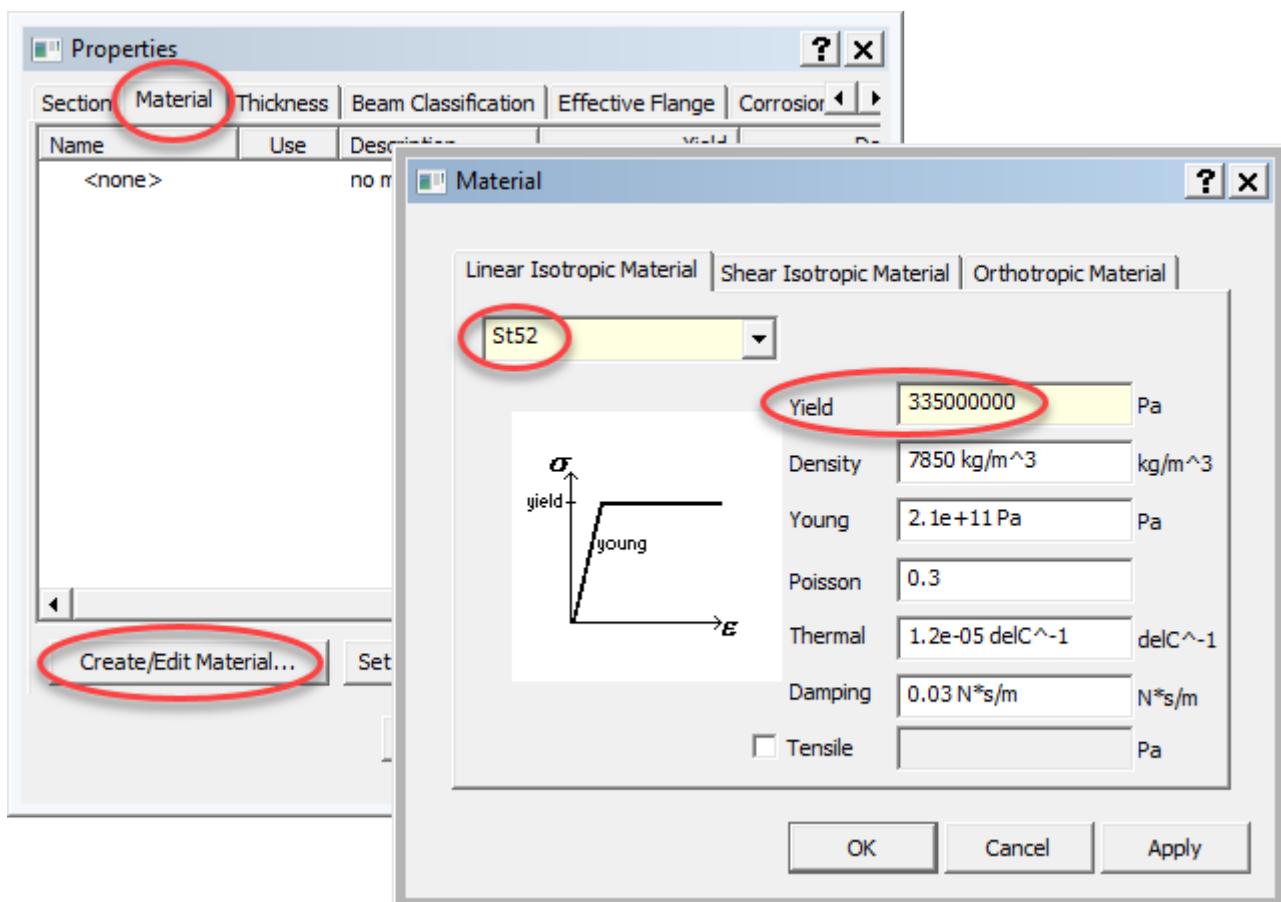
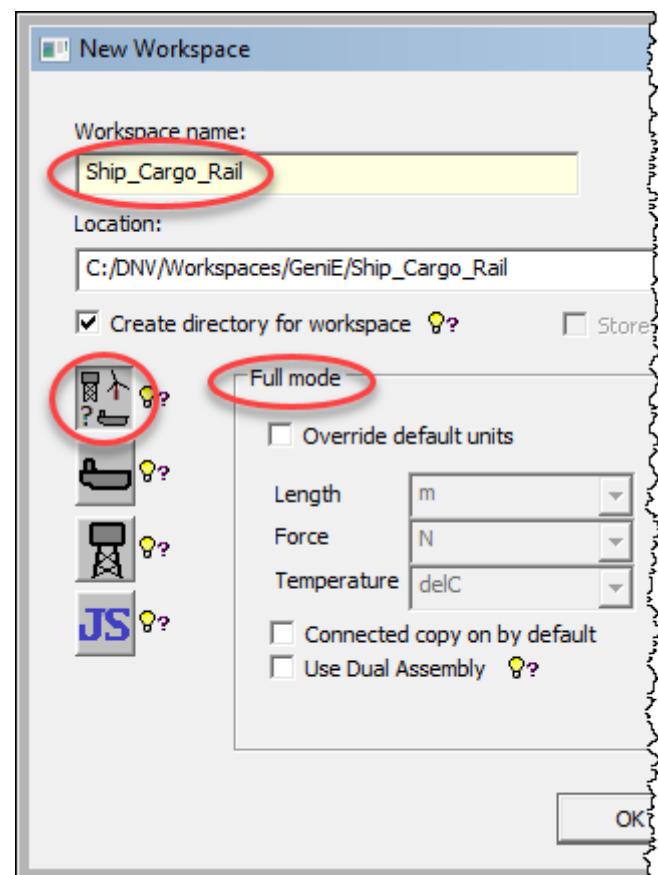
1 INTRODUCTION

- In this tutorial a model of a ship section with a cargo rail is created and analysed.
- The focus is on modelling and less on defining loads and controlling the FE mesh.
- The workshop is accompanied by the GeniE input file Ship_Cargo_Rail_input.js.

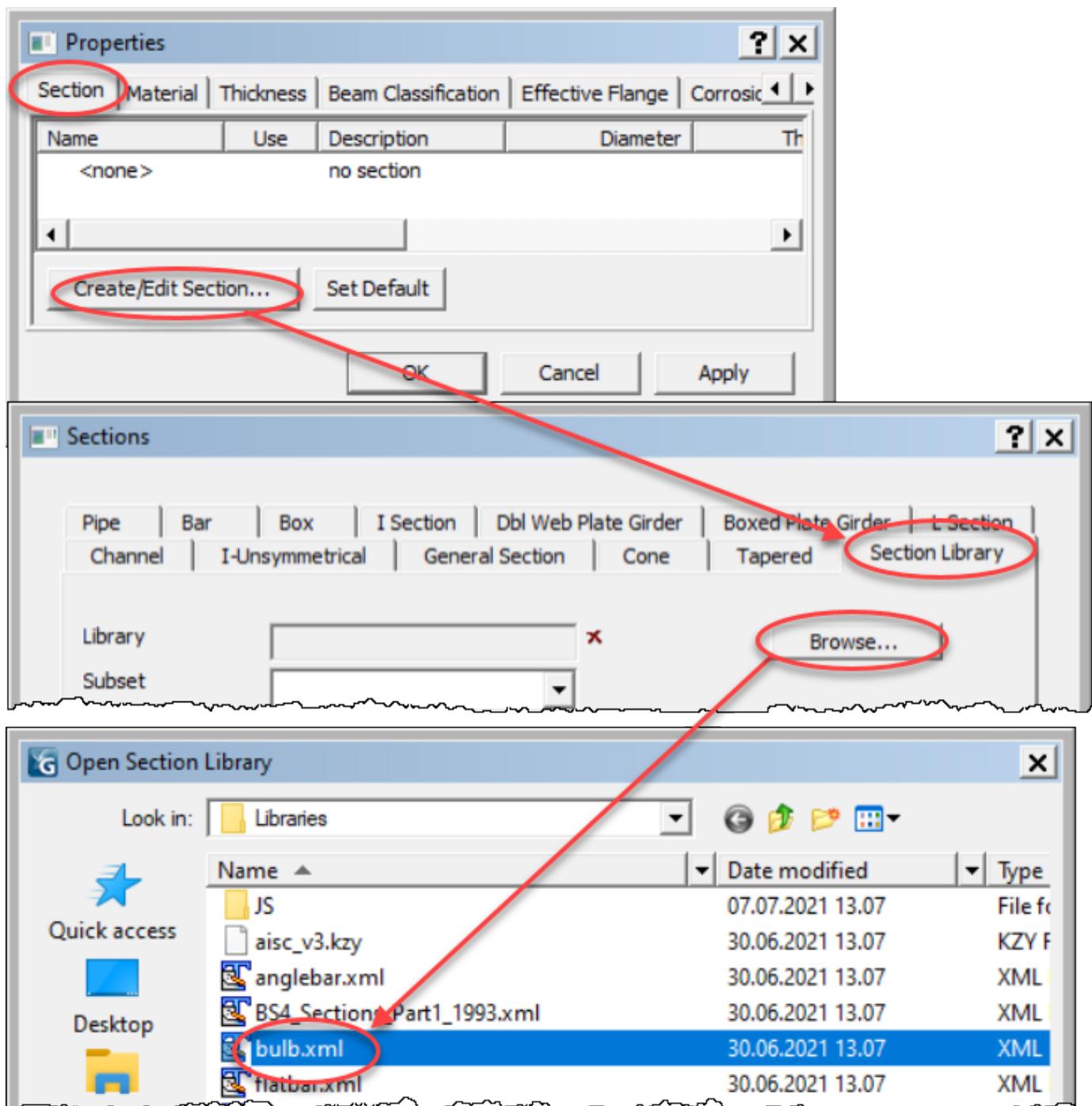


2 UNITS, MATERIAL, CROSS SECTIONS, PLATE THICKNESSES

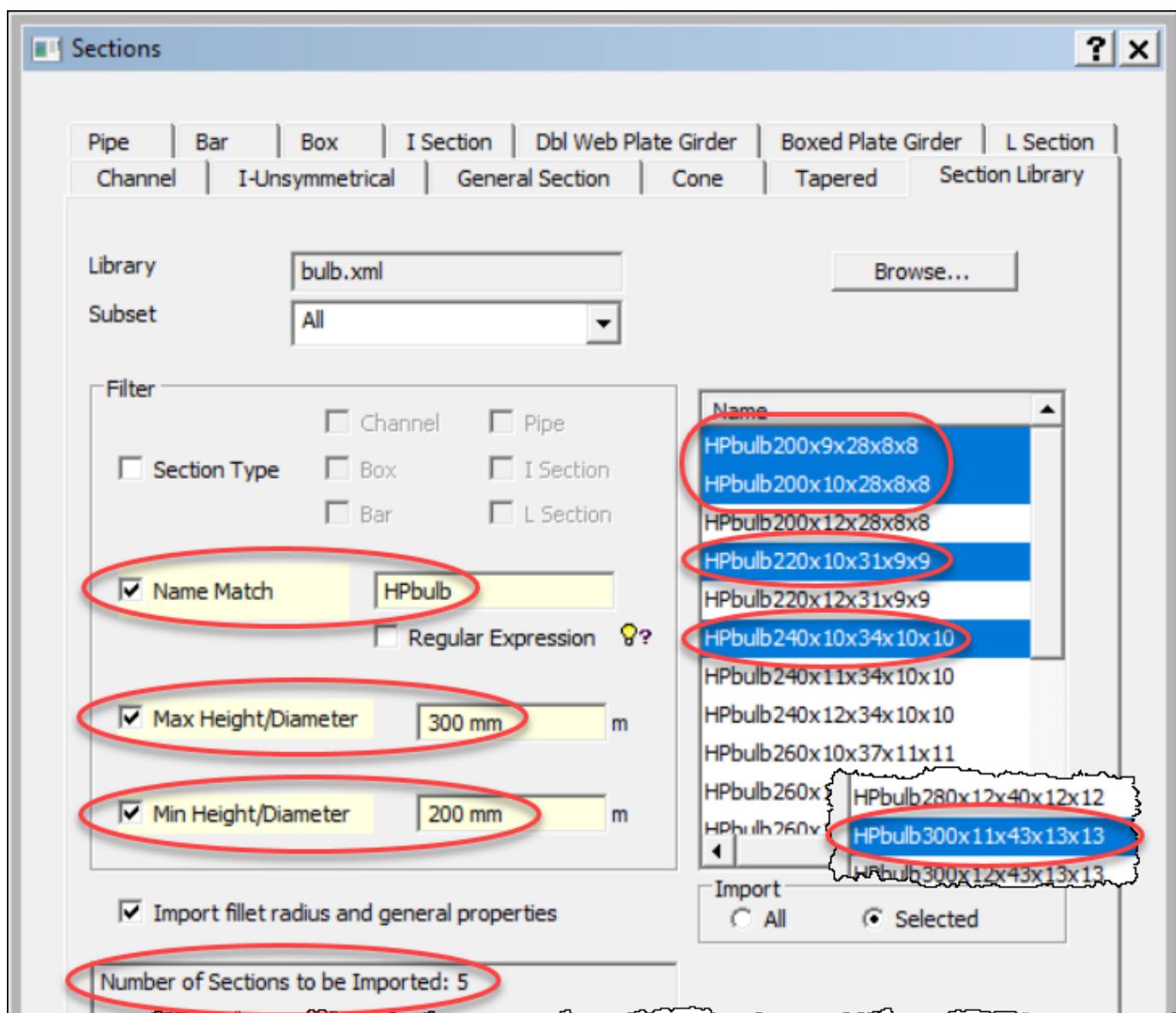
- Start GeniE and open a new workspace.
 - Give a *Workspace name*.
 - Accept default units m and N and click **OK**.
 - Unless otherwise specified, all values in this tutorial are in these units.
 - Make sure *Full mode* is selected as this tutorial involves curved modelling.
- Define steel material.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



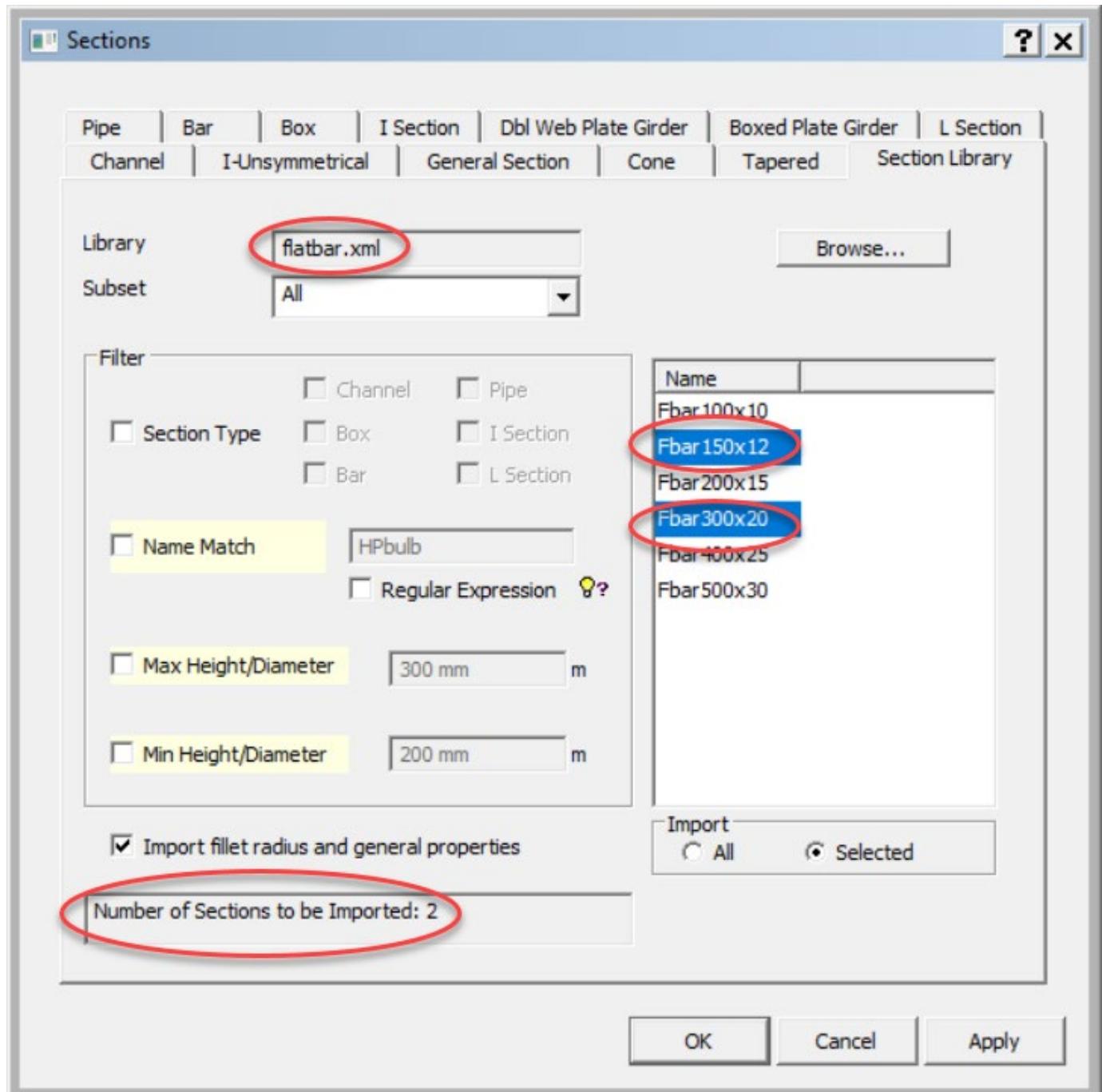
- Define beam cross section properties.
 - All sections except the rails are imported from a library.
 - Use *Edit | Properties* and go to the *Section* tab. Click *Create/Edit Section* to open the *Sections* dialog.
 - In the *Sections* dialog go to the *Section Library* tab and click *Browse* and find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



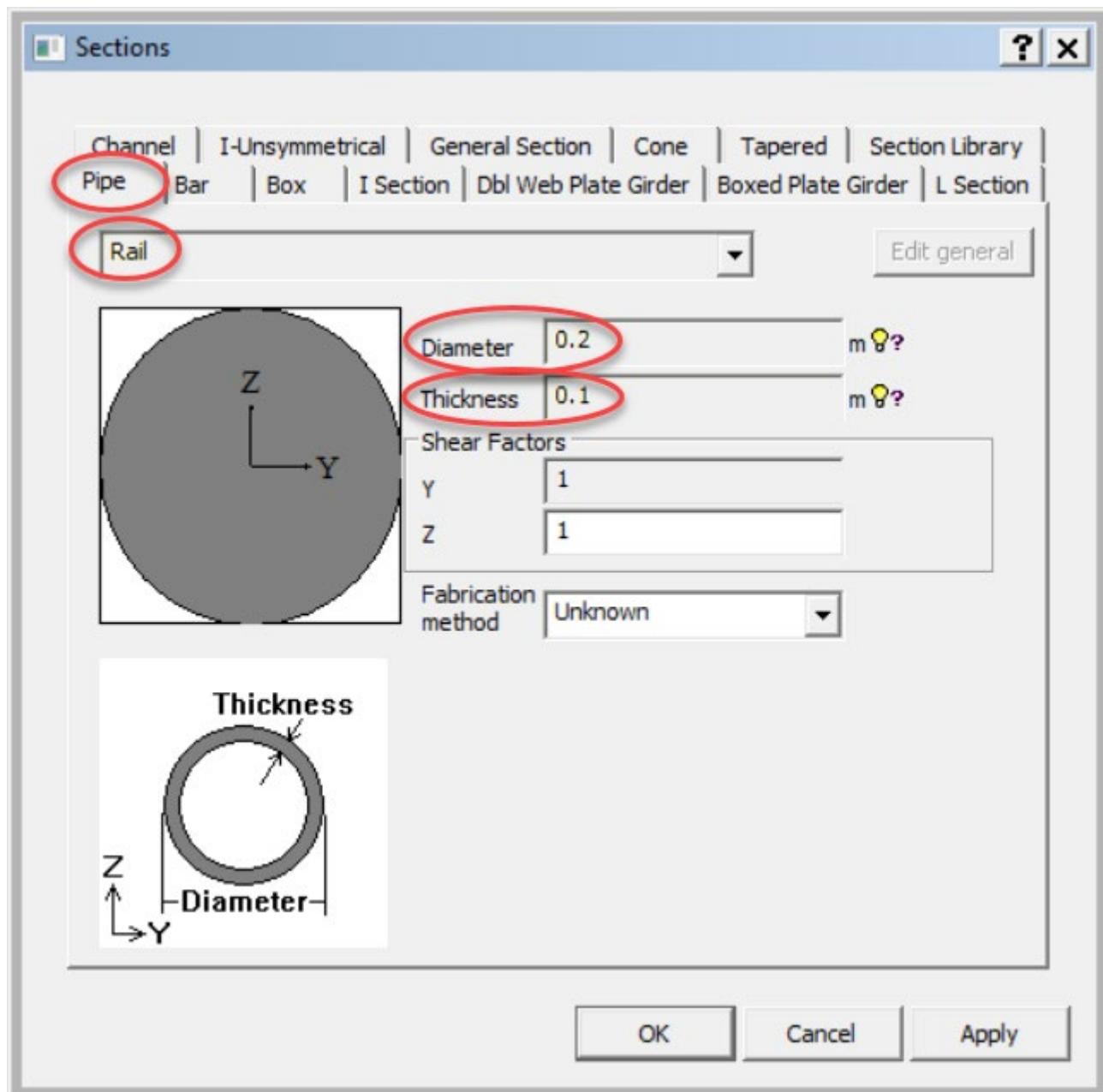
- Import the following five bulb sections into the workspace:
 - HPbulb200x10x28x8x8
 - HPbulb200x9x28x8x8
 - HPbulb220x10x31x9x9
 - HPbulb240x10x34x10x10
 - HPbulb300x11x43x13x13
- To ease the selection check *Name Match* and enter HPbulb and see that only sections with name containing the string HPbulb are listed.
- Also limit the list to sections within the height range 200 mm to 300 mm.
- Having made the selection make sure the *Import* radio button *Selected* is chosen and see the text *Number of Sections to be Imported: 5* appears.



- Open the section library *flatbar.xml* to import the following sections:
 - Fbar150x12
 - Fbar300x20



- Manually create the section Rail as a pipe section as shown below. A thickness of half the diameter means that the section is a solid rod.

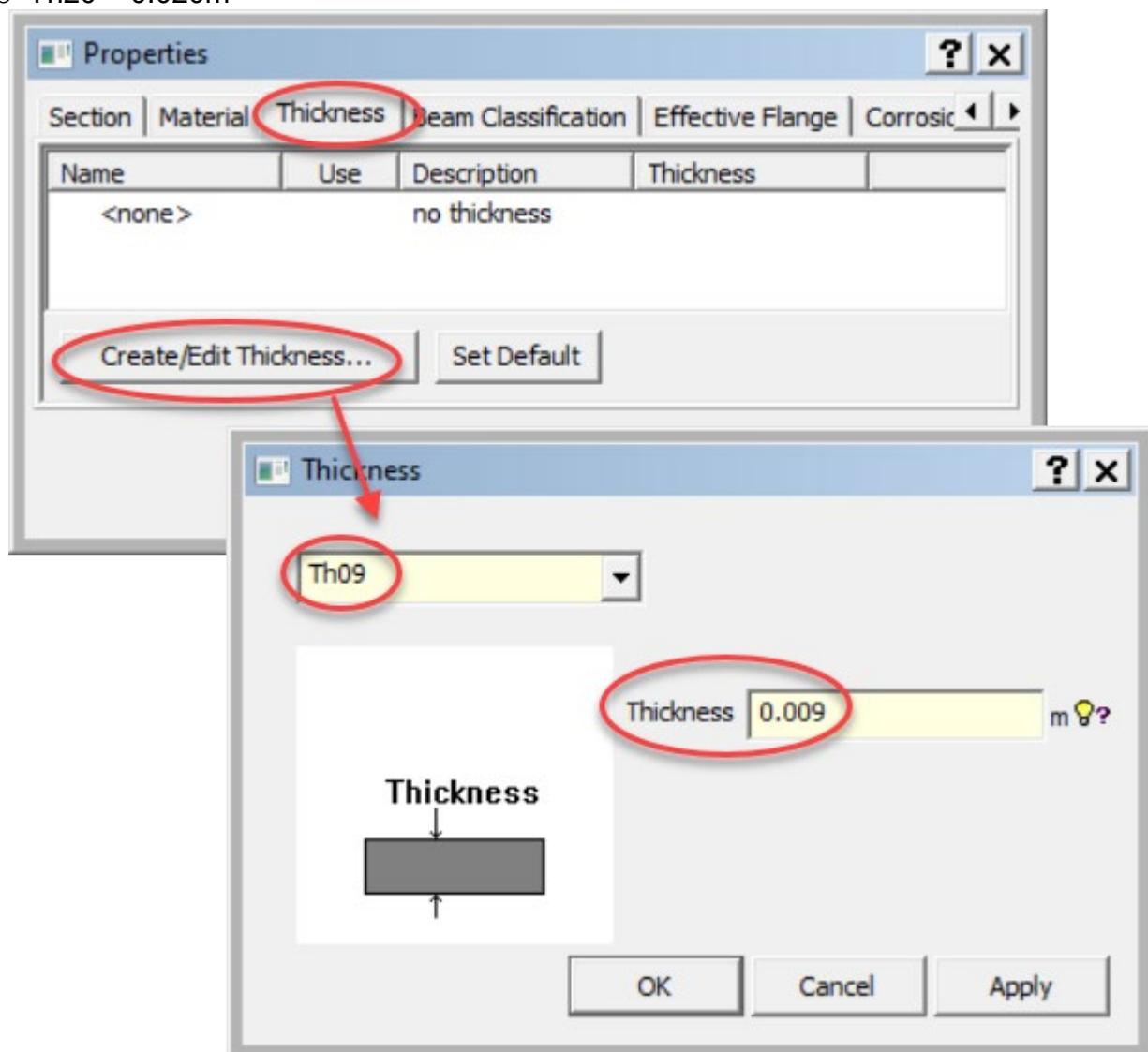


- See that all eight sections appear in the Section tab of the *Properties* dialog.

HPbulb200x9x28x8x8	0	L Section
HPbulb200x10x28x8x8	0	L Section
HPbulb220x10x31x9x9	0	L Section
HPbulb240x10x34x10x10	0	L Section
HPbulb300x11x43x13x13	0	L Section
Fbar150x12	0	Bar Section
Fbar300x20	0	Bar Section
Rail	0	Pipe Section

➤ Define the plate thicknesses:

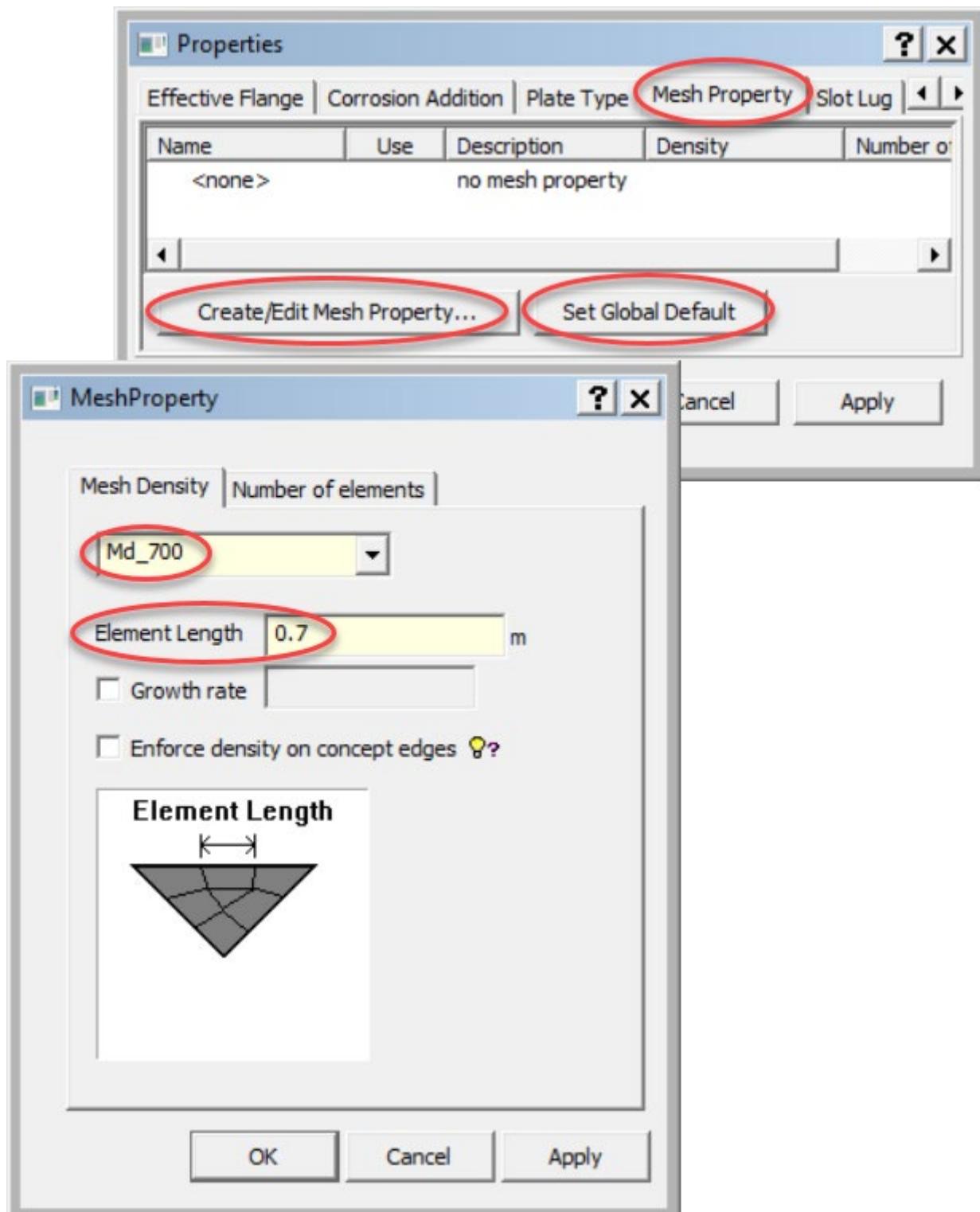
- Th08 = 0.008 m
- Th09 = 0.009 m
- Th10 = 0.010 m
- Th11 = 0.011 m
- Th12 = 0.012 m
- Th13 = 0.013 m
- Th15 = 0.015 m
- Th20 = 0.020m



➤ Set default properties: section HPbulb200x10x28x8x8, material St52 and thickness Th08.

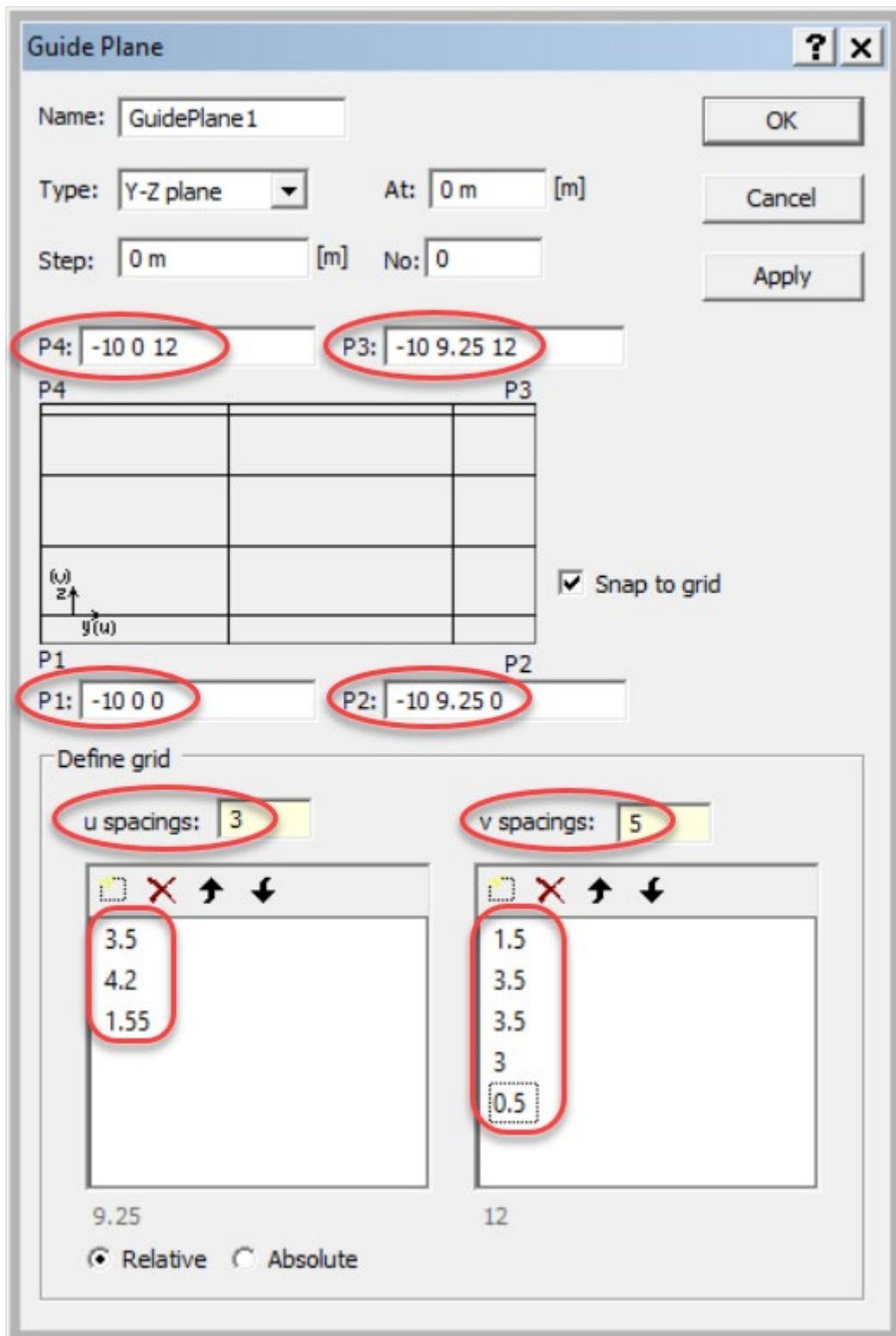


- Use *Edit | Properties | Mesh Property* to define a default mesh density.
- Click *Create/Edit Mesh Property* and give a property name and *Element Length* in the *Mesh Property* dialog.
 - Back in the *Properties* dialog click *Set Global Default* to make this mesh property valid for all beams and surfaces for which no other mesh property have been assigned.

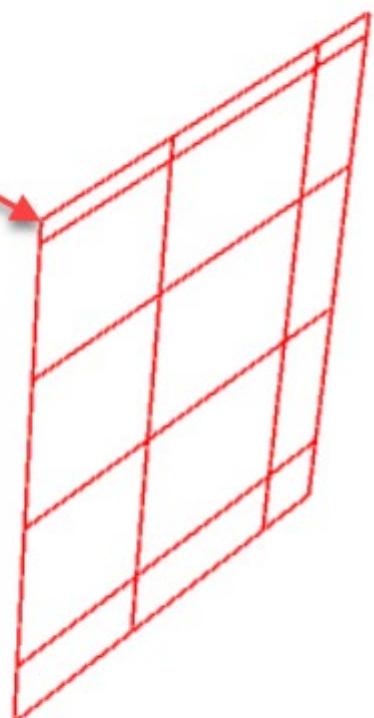
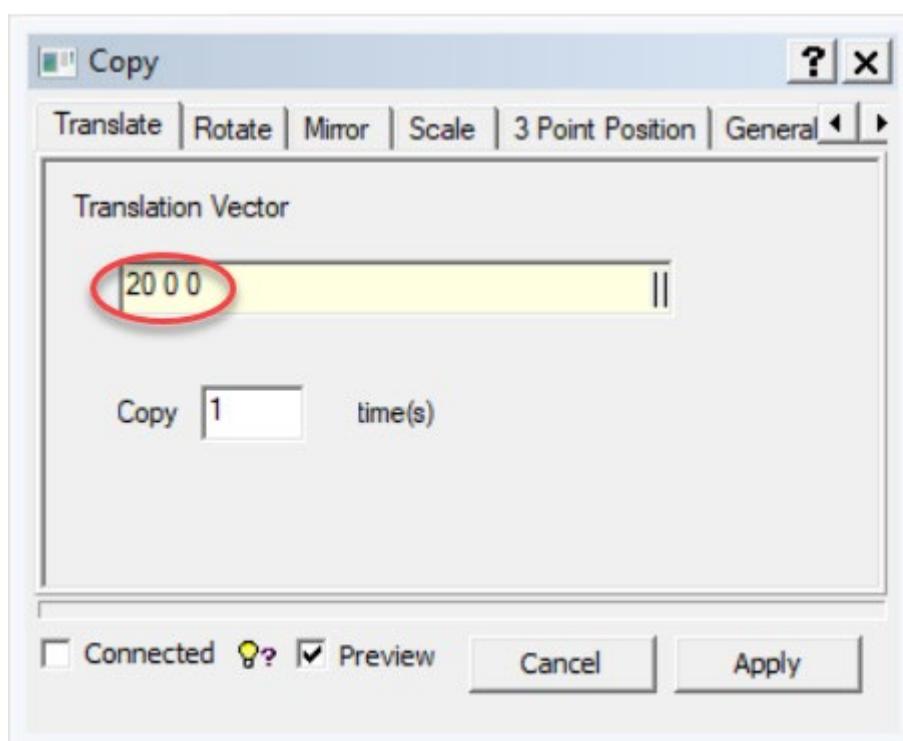
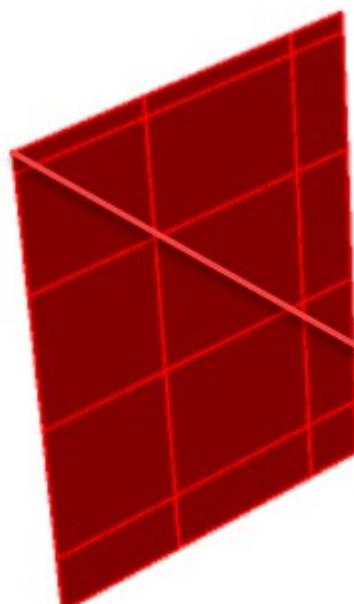
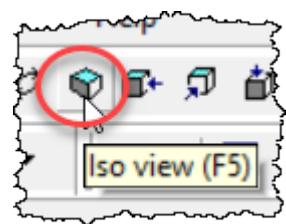


3 CREATE GUIDING GEOMETRY

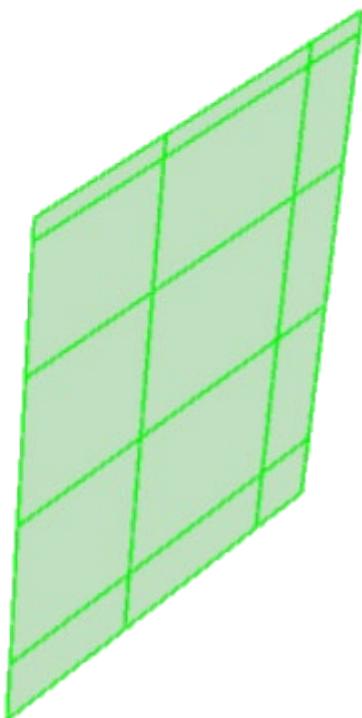
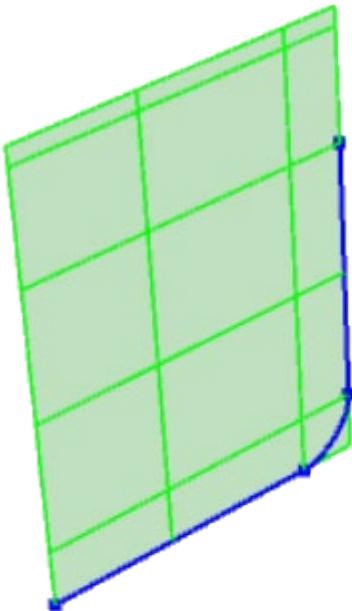
- Use *Guiding Geometry | Planes | Guide Plane Dialog* to create a guide plane with data as shown below.
- After entering the last *u spacing* value (1.55) click another *u spacing* value (3.5 or 4.2) to see the guide plane sketch in the dialog updated.
 - Do the same after entering the last *v spacing* value (0.5).



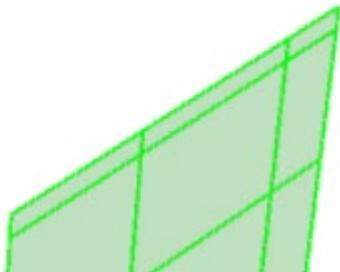
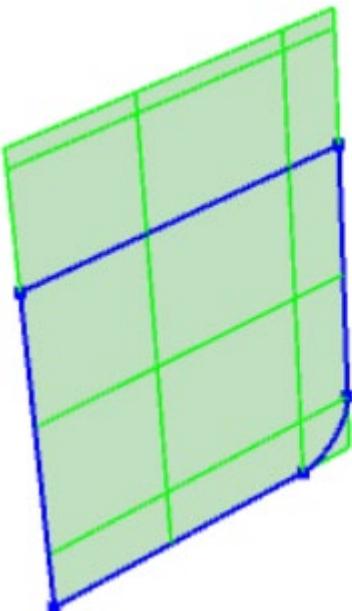
- Copy the guide plane 20 m in the X-direction.
- The view below is after clicking the *Iso view* button (or pressing the function key F5). Unless otherwise stated or shown, all views in this tutorial are using this viewpoint, or approximately this viewpoint.



- Use *Guiding Geometry | Free-Form Curves | Poly-curve* to create the curve shown below.

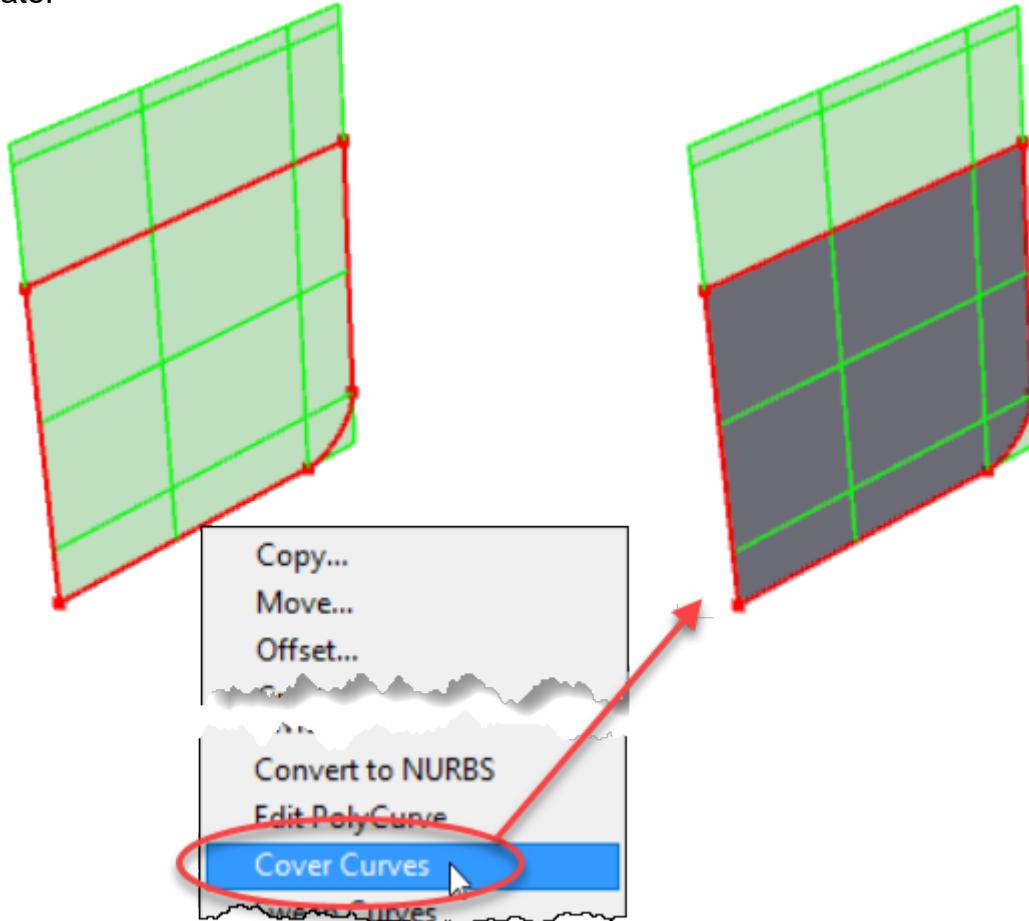


- Use *Guiding Geometry | Lines | From Two Points* to create the two additional straight lines below.

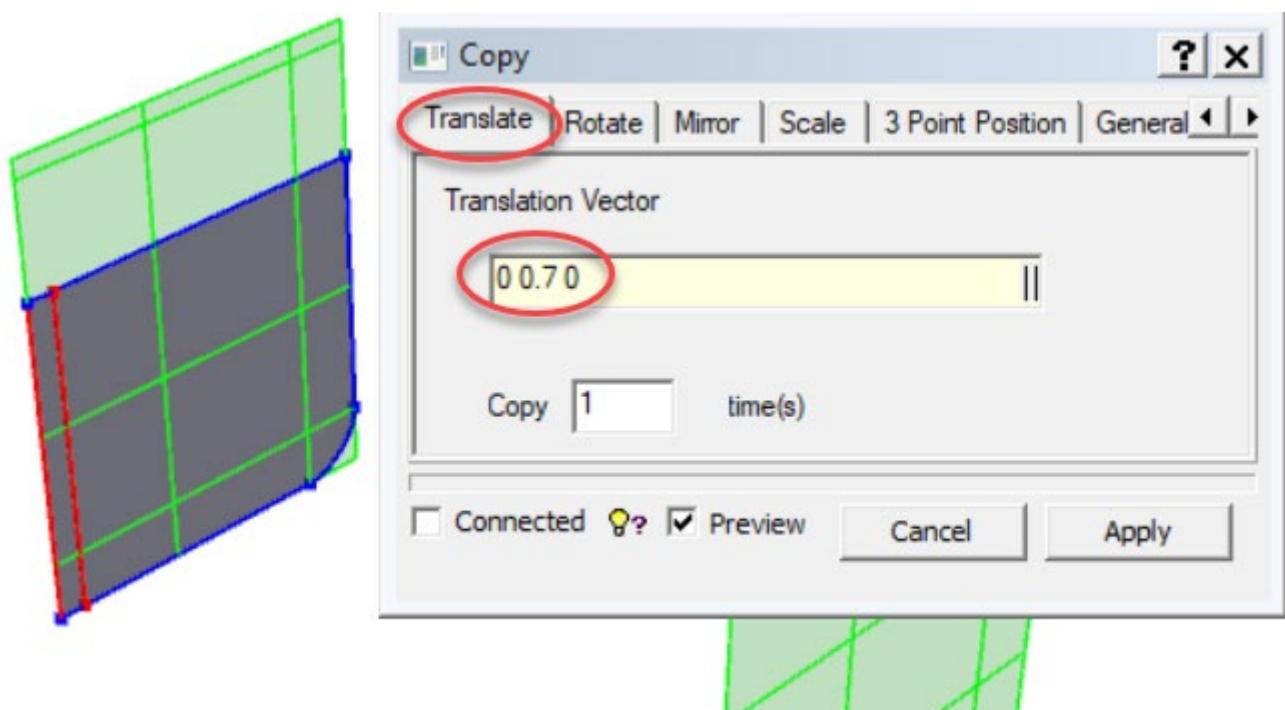


4 CREATE PLATE WITH STIFFENERS AT X = -10 M

- Select the poly-curve and the two lines and right-click to *Cover Curves* thereby creating a plate.

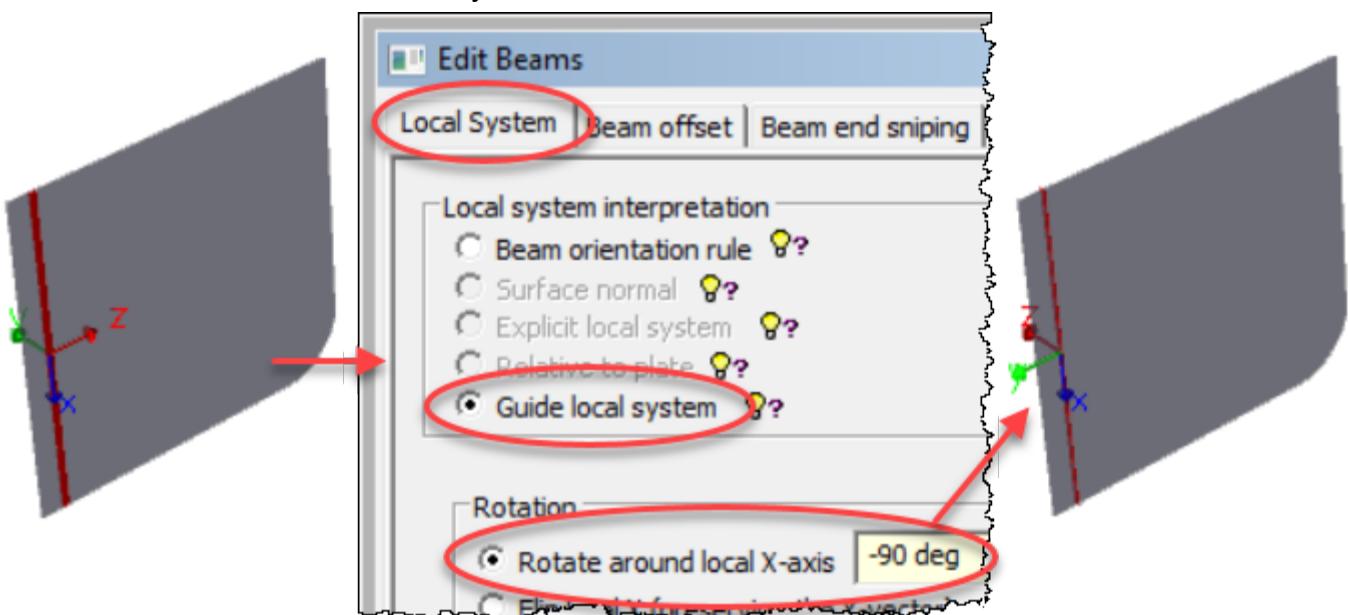


- Copy the vertical guide line 0.7 m in Y-direction.

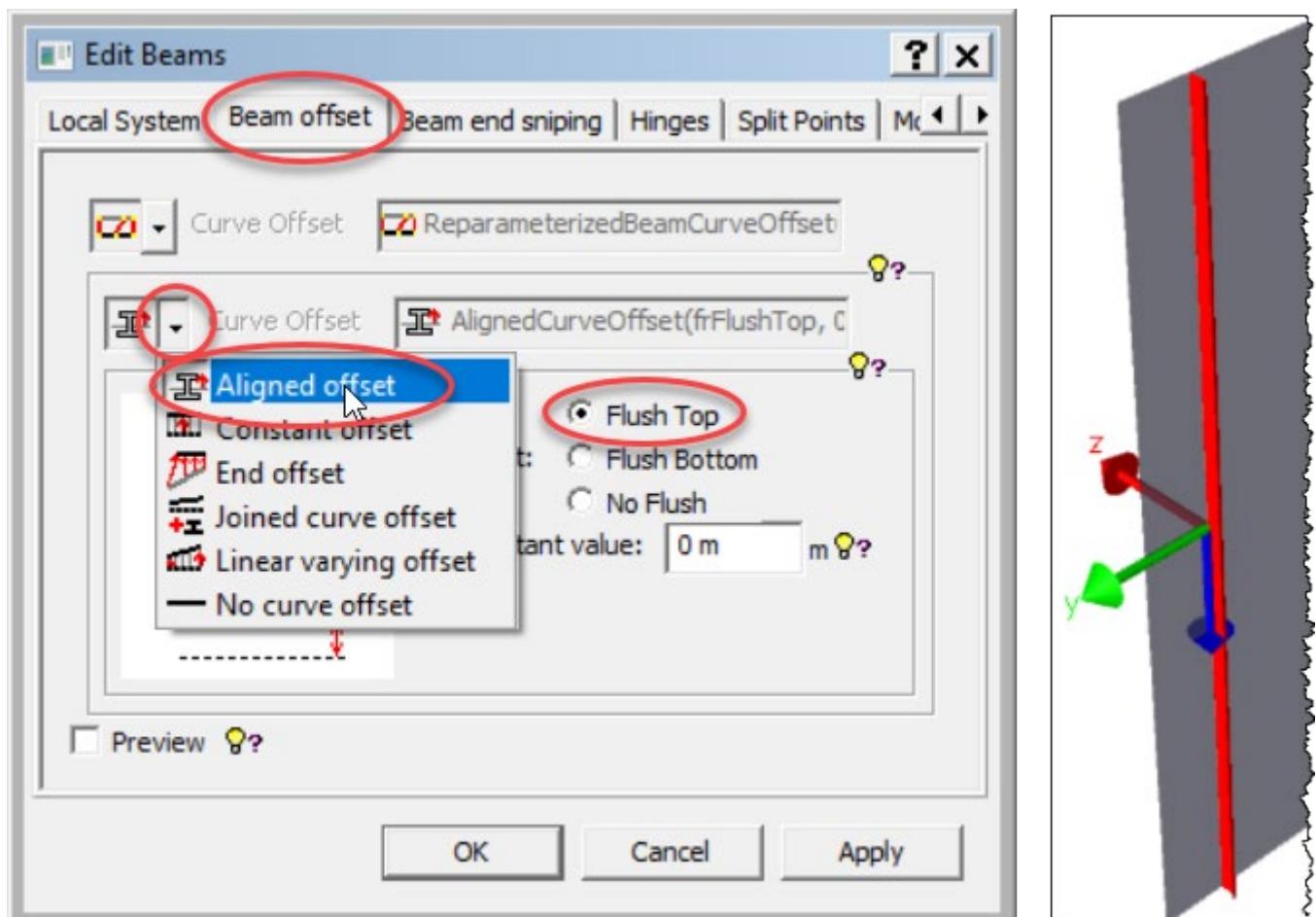


- Create a beam with cross section HPbulb200x10x28x8x8 along the new guide line by right-clicking it and selecting *Create Beam*.

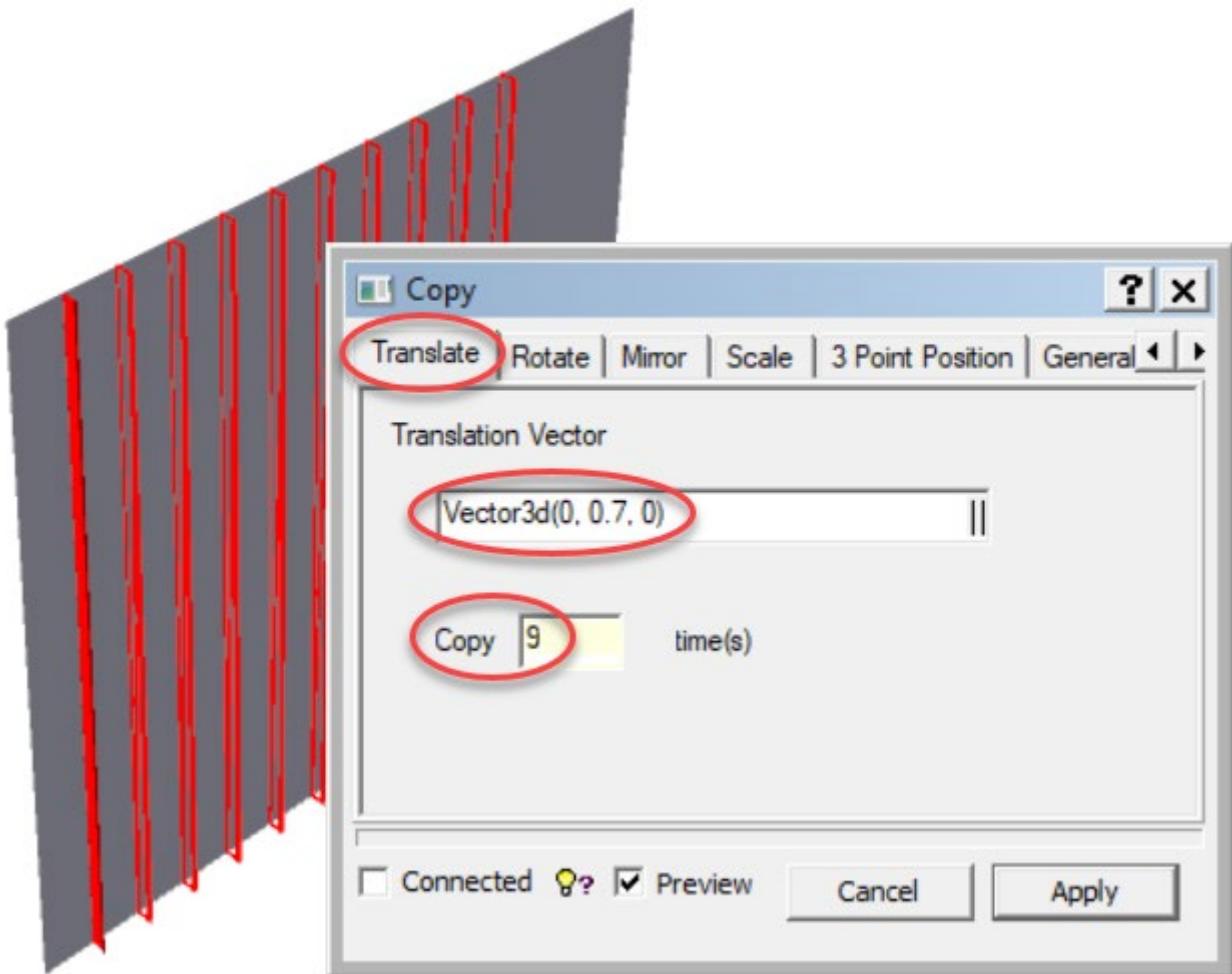
- Orientate the beam correctly as shown below.



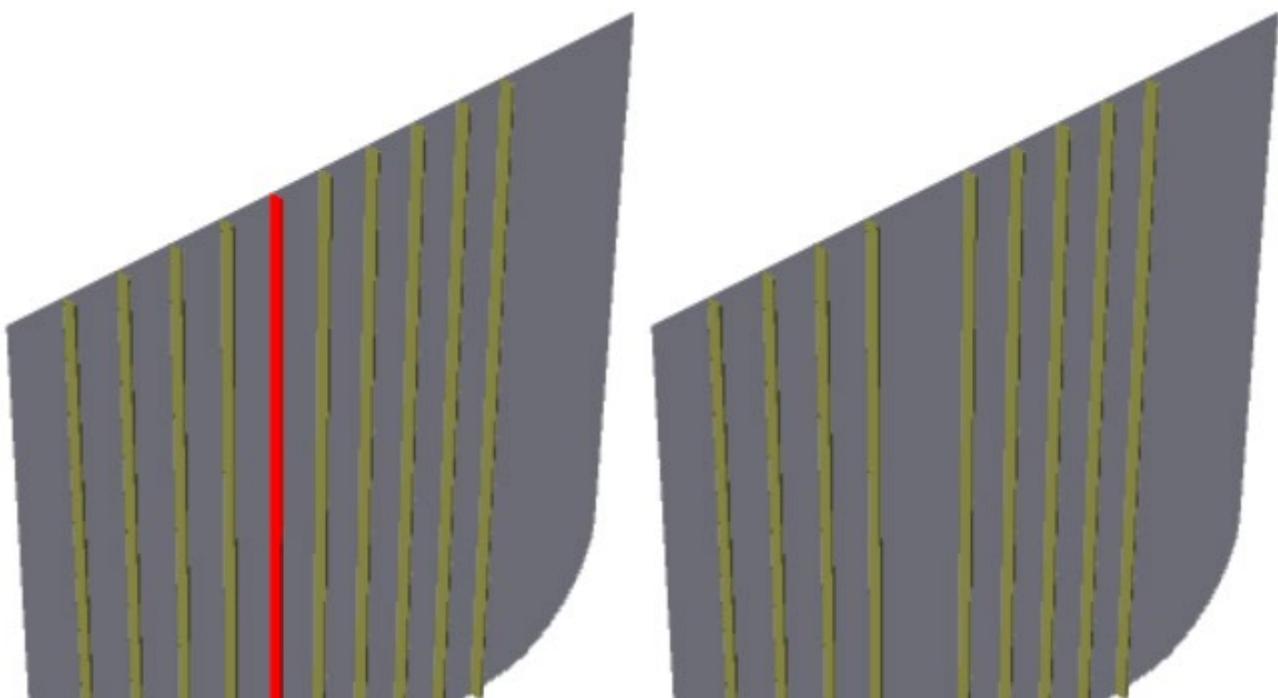
- Flush the beam with the plate as shown below so as to become a vertical plate stiffener.



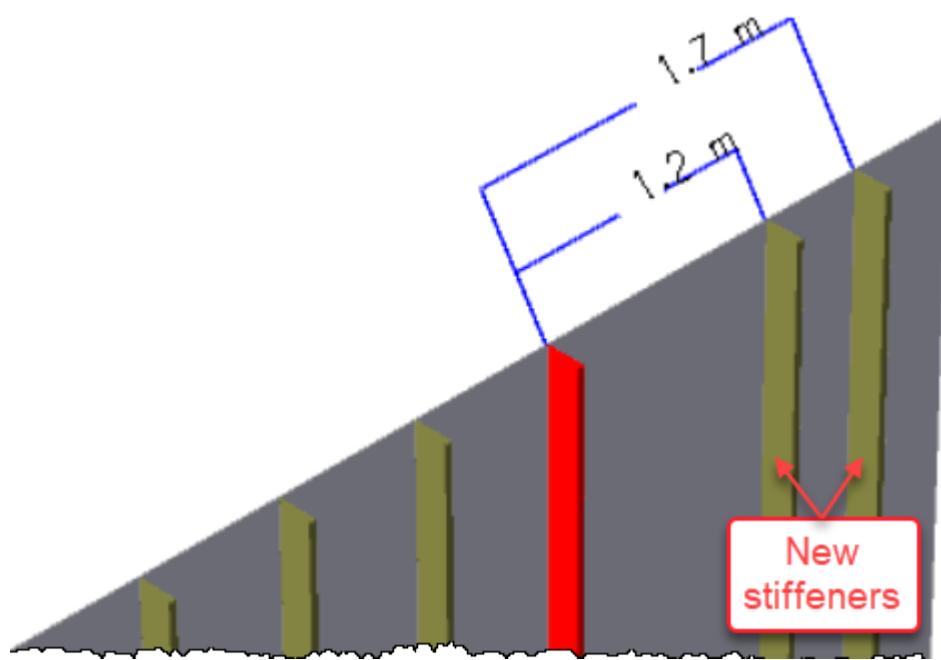
- Copy the vertical stiffener nine times a distance of 0.7 m in Y-direction.



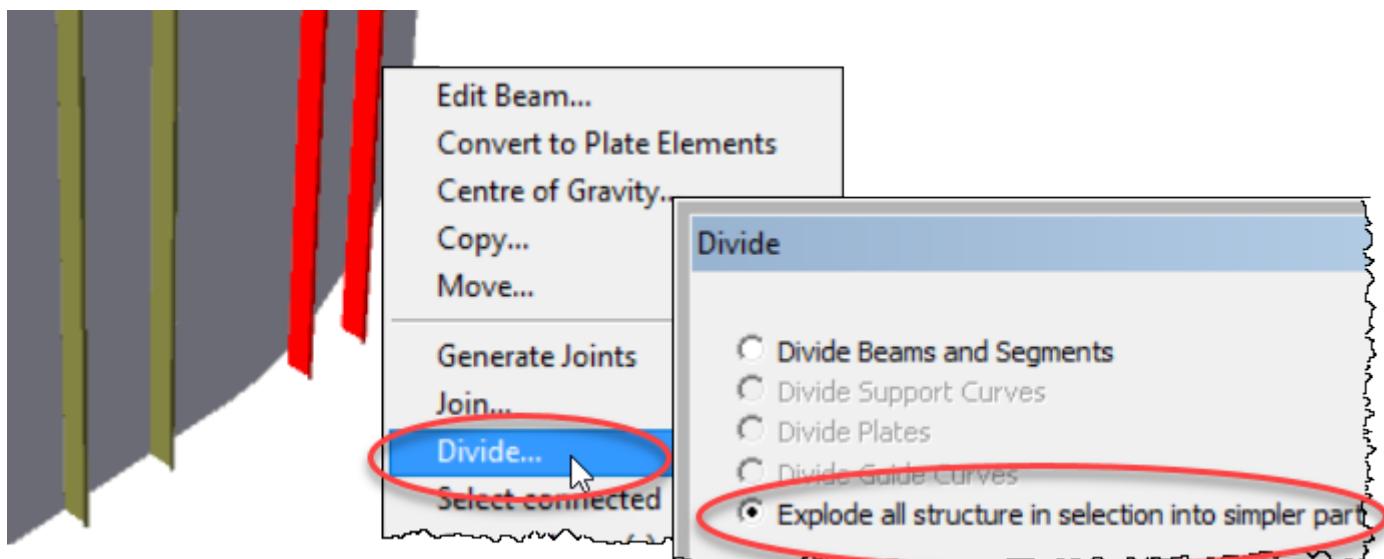
- Delete the fifth of the ten vertical stiffeners.



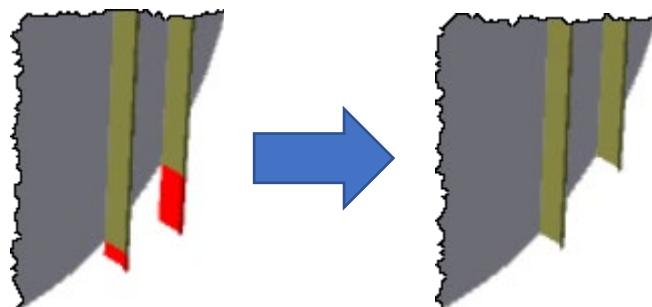
- Copy the vertical stiffener with highest Y-coordinate to the two new positions as shown below.



- Select the two new stiffeners that in their lower ends extend outside the plate and divide them as shown.

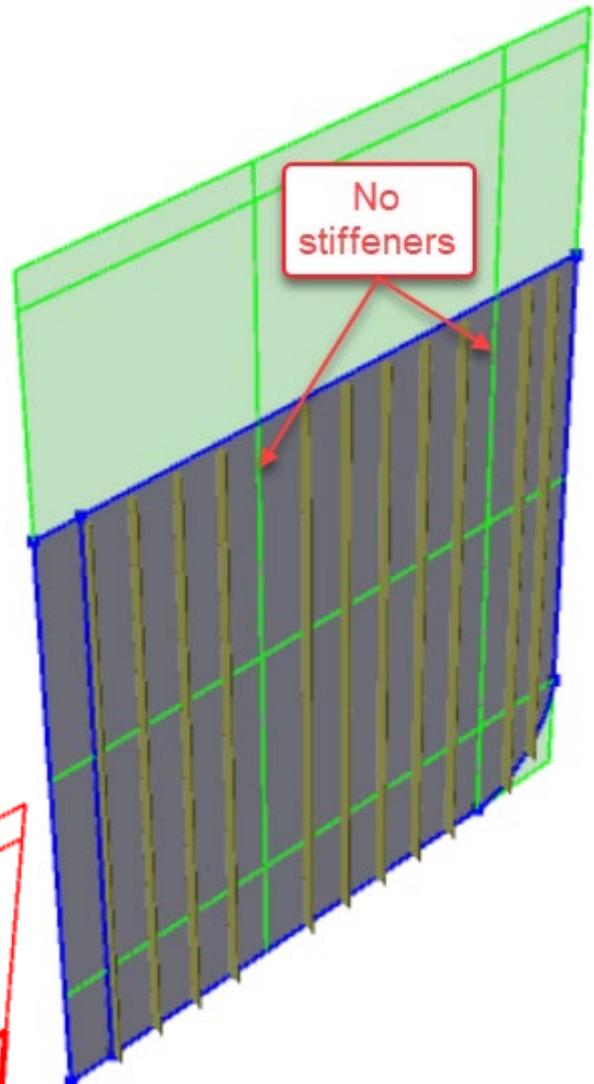


- Delete the protruding parts.

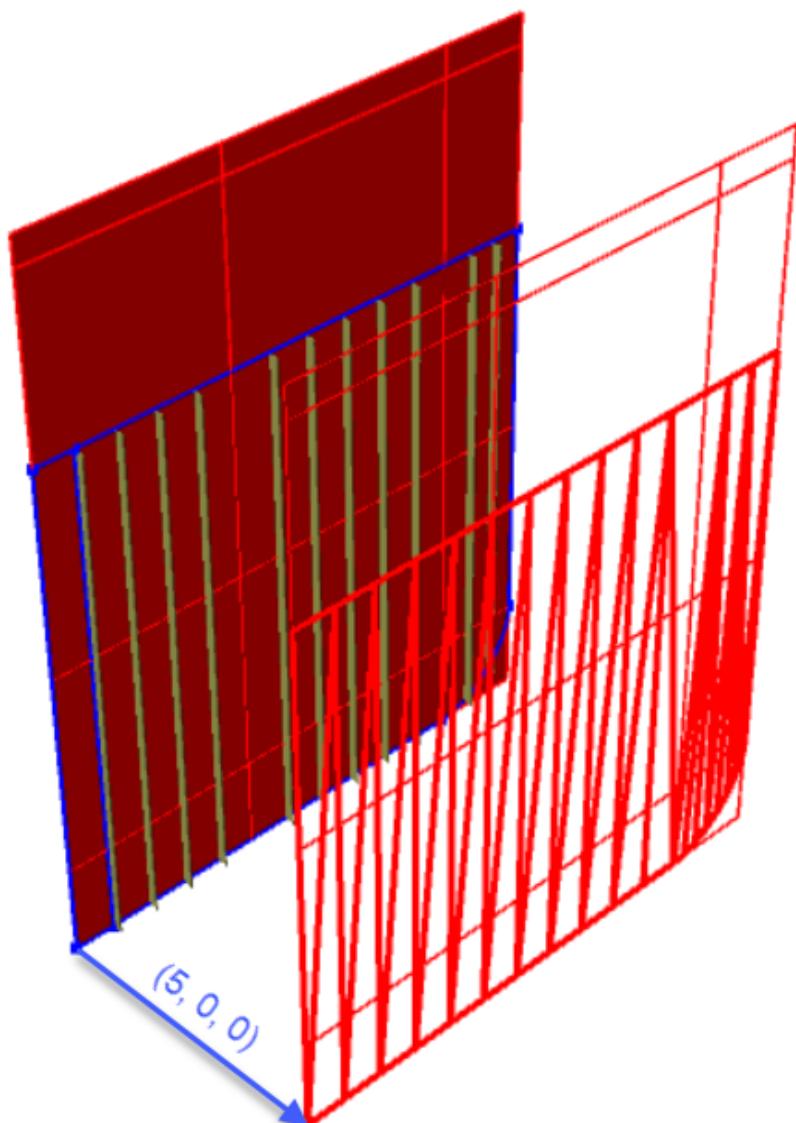


5 CREATE WEB FRAME WITH STIFFENERS AT X = -5 M

- The model should now appear as shown to the right together with the guide plane. Note that there should be no stiffeners at the guide plane spacing lines.



- Copy the guide plane and plate (not stiffeners) at X = -10 m a distance of 5 m in X-direction as shown below.



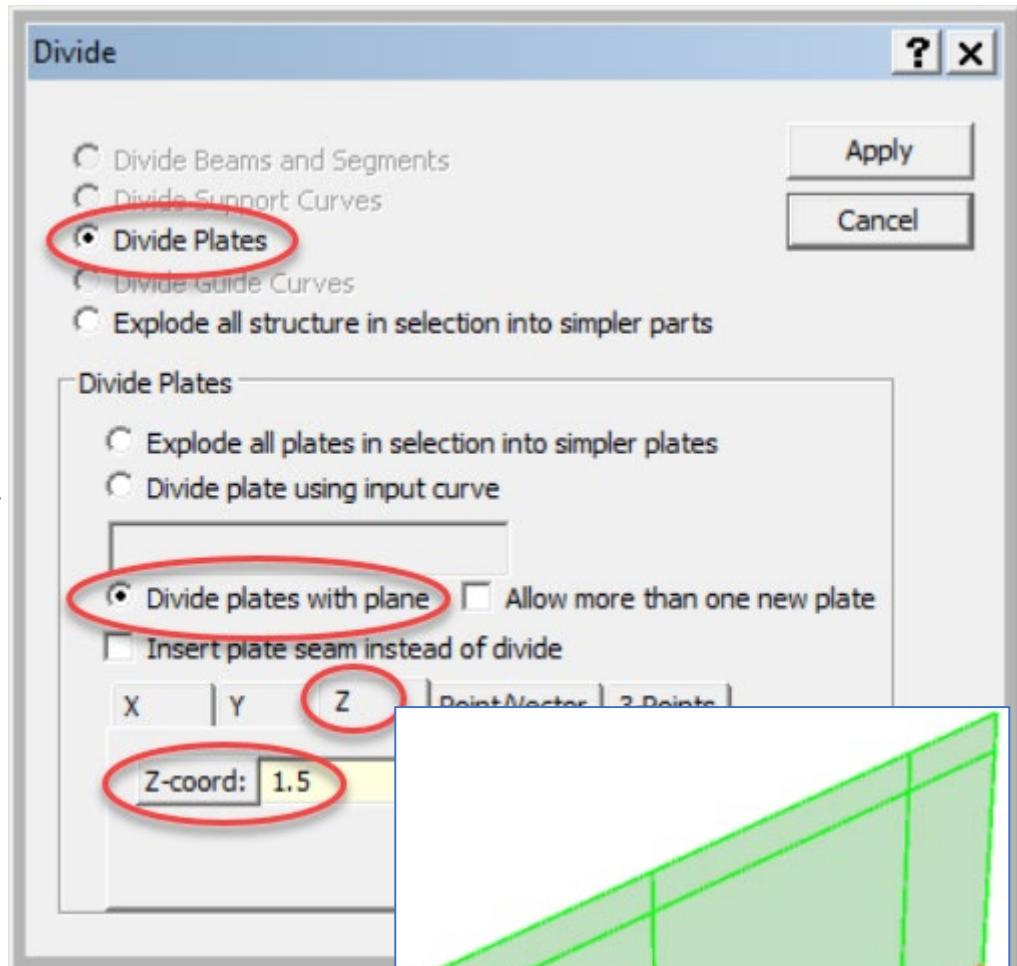
- Select the new plate at $X = -5\text{ m}$,

right-click it and divide it with the following planes:

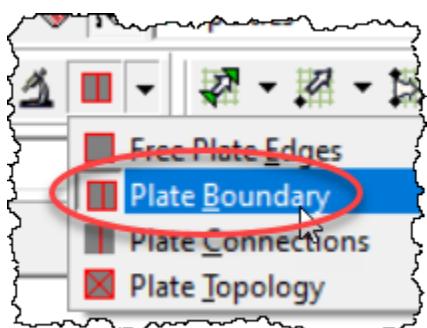
- $Z = 1.5\text{ m}$
- $Z = 7.8\text{ m}$
- $Y = 3.5\text{ m}$
- $Y = 3.1\text{ m}$

- The *Divide* dialog for the dividing process

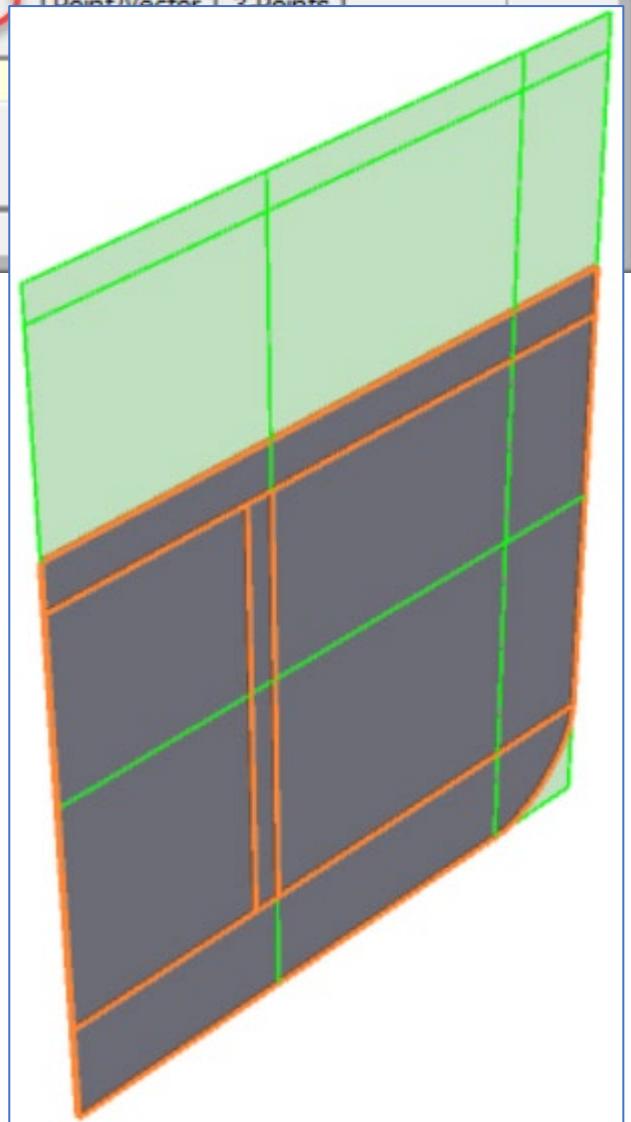
at $Z = 1.5\text{ m}$ is shown to the right.



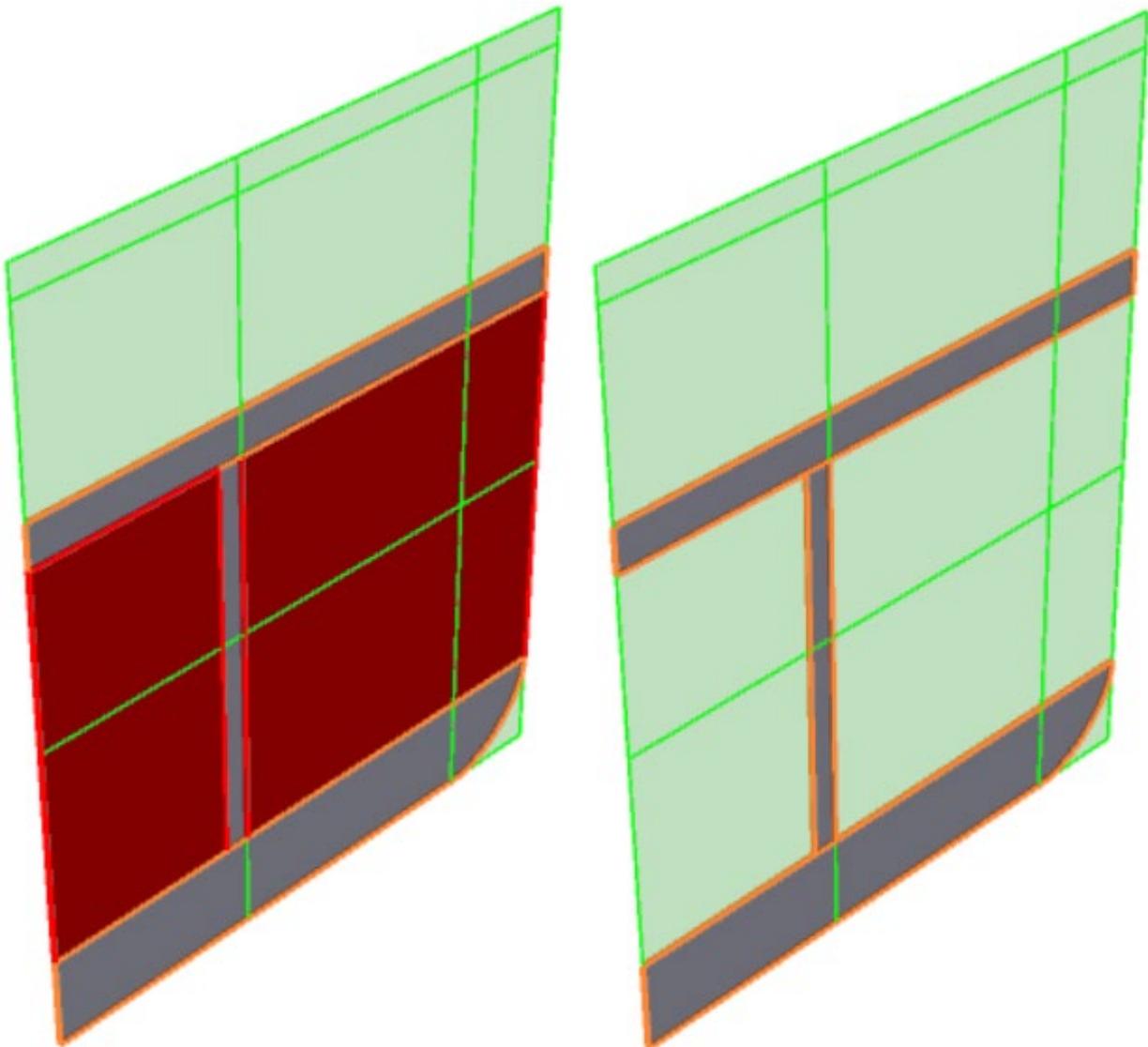
- Add plate boundaries by *Structure | Topology | Display | Plate Boundary* or click the button:



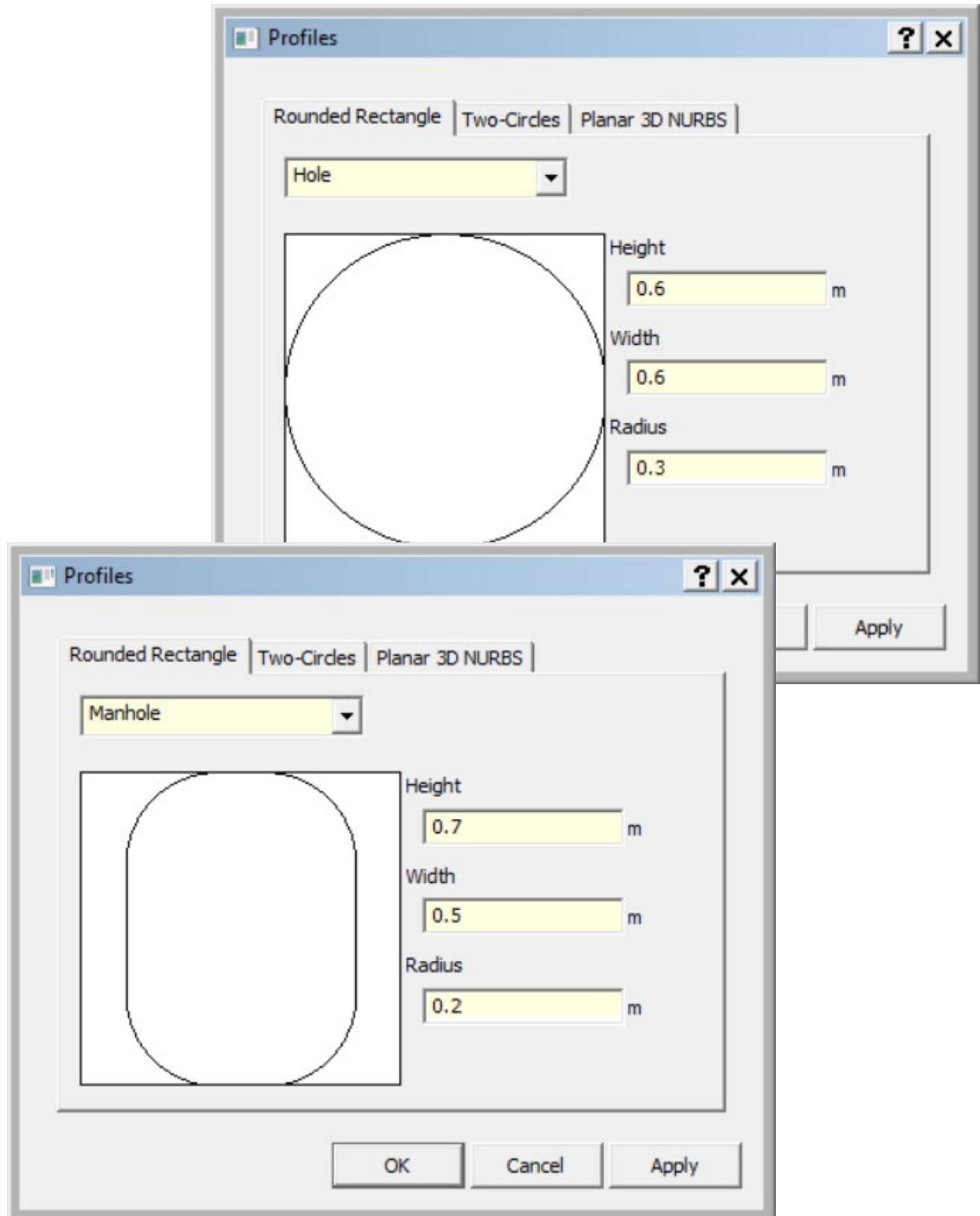
- The plate should appear as shown to the right.



- Delete two plates as shown below.

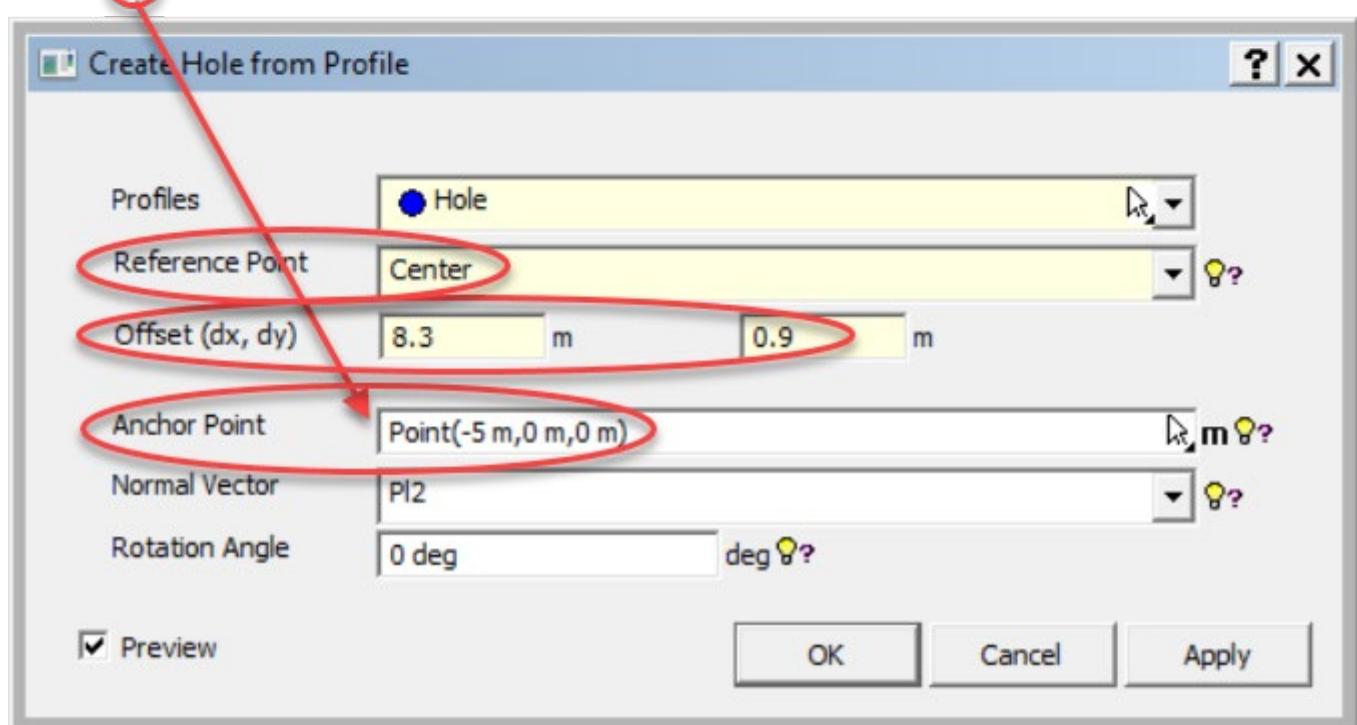
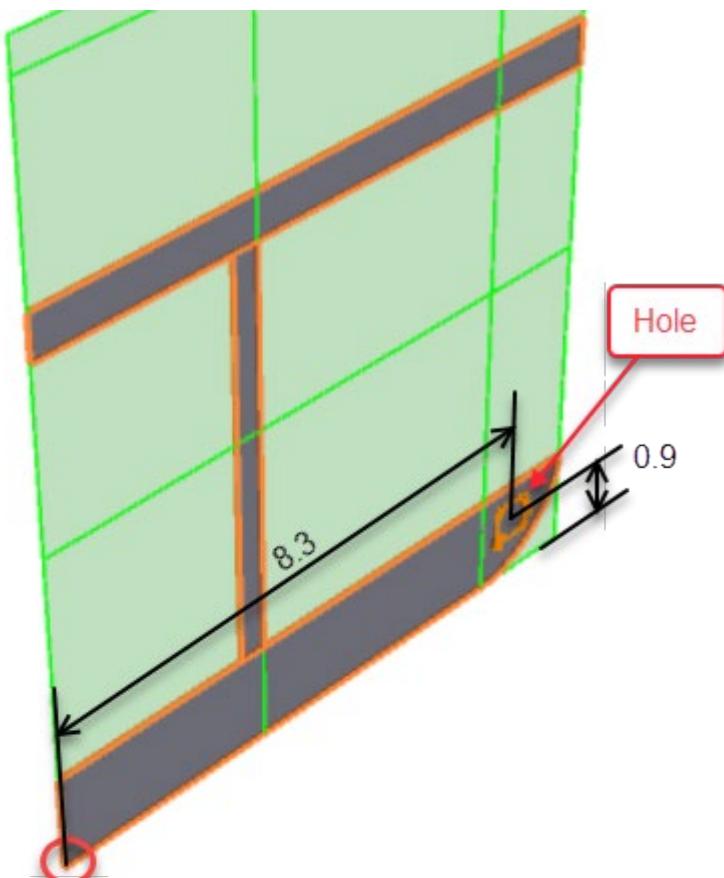


- Define two profiles by *Guiding Geometry | Profiles | Profile Dialog* as shown. These will be used for punching holes in the web frame.



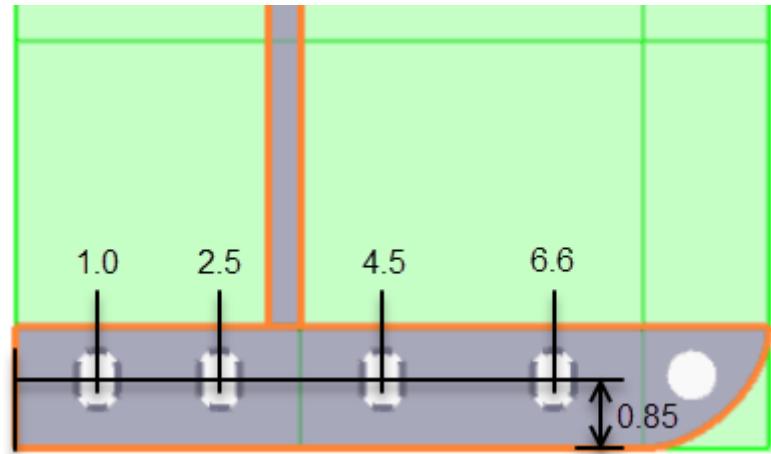
- Punch a hole using the a profile by *Structure | Features | Hole from Profile*.

- In the *Create Hole from Profile* dialog, select the profile **Hole**, **Reference Point Center** (of the hole), **Anchor Point** by clicking the geometry as shown, and give **Offset** as shown. The offset *dx* and *dy* are in a local profile coordinate system rather than in model coordinates.



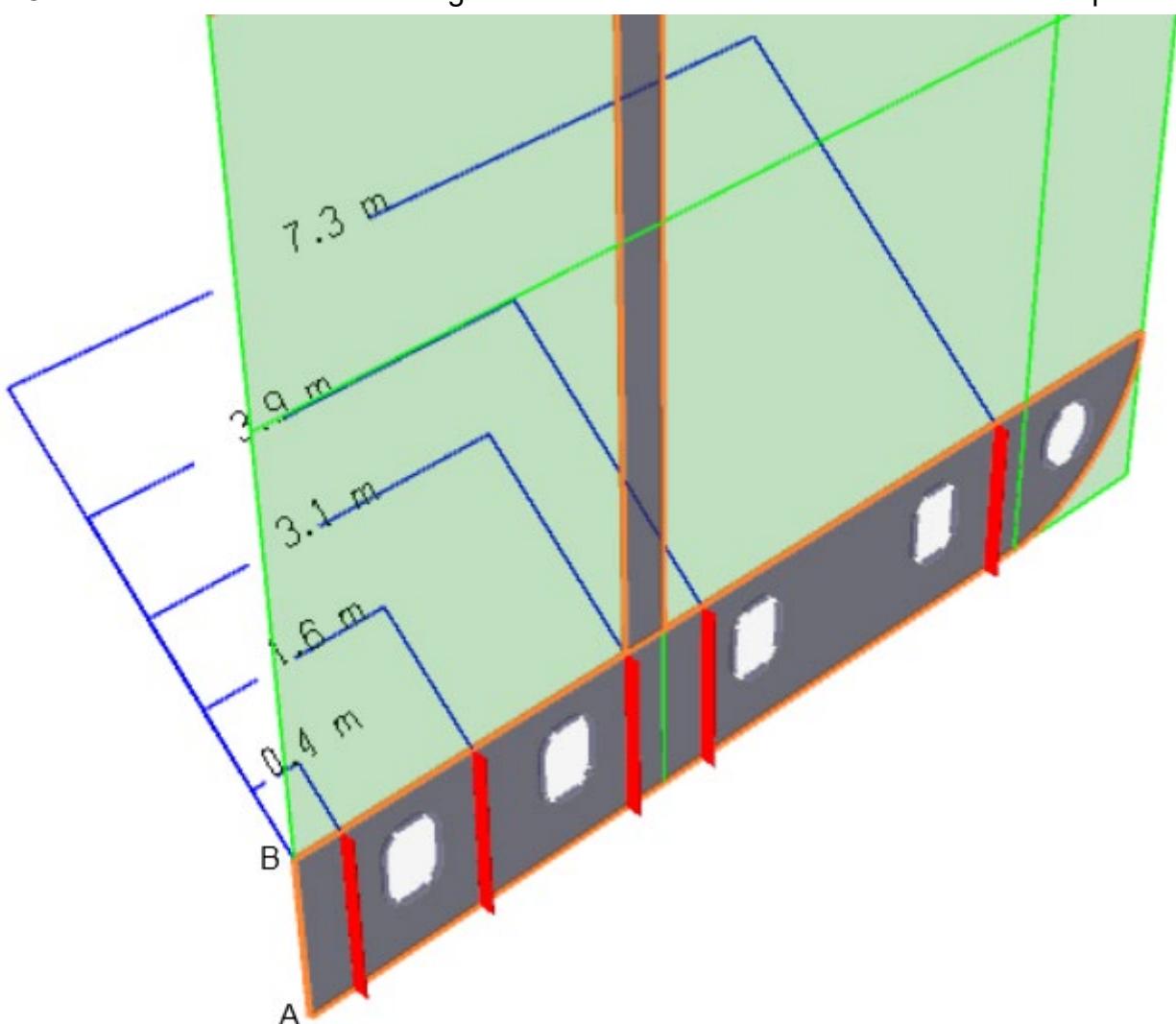
- Punch four holes using the Manhole profile with the following offsets dx and dy to the profile centre point from the same *Anchor Point* as above:

- $dx = 6.6 \text{ m}$, $dy = 0.85 \text{ m}$
- $dx = 4.5 \text{ m}$, $dy = 0.85 \text{ m}$
- $dx = 2.5 \text{ m}$, $dy = 0.85 \text{ m}$
- $dx = 1.0 \text{ m}$, $dy = 0.85 \text{ m}$

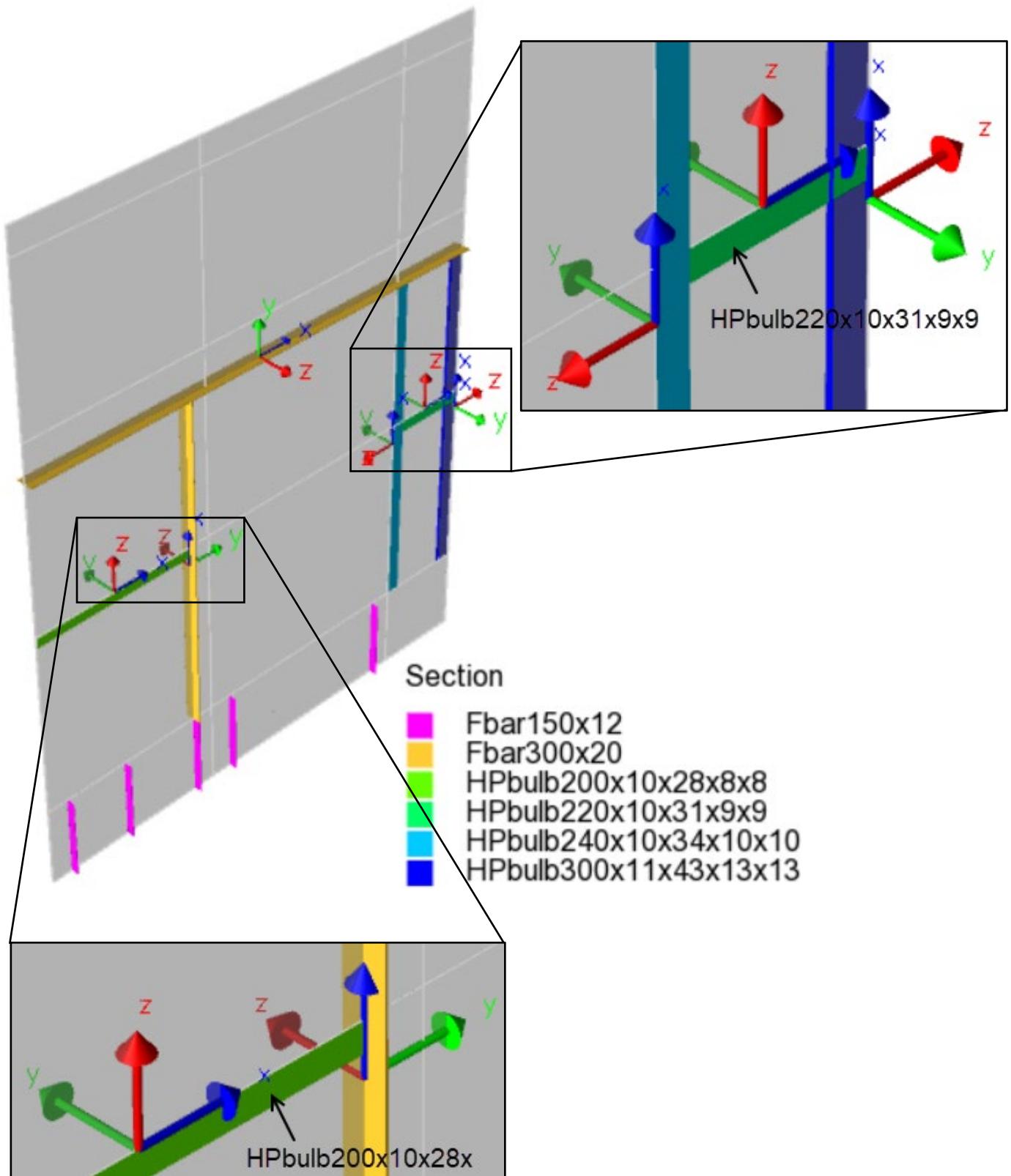


- Create the Fbar150x12 stiffener beams shown below by *Structure | Beams and Piles | Straight Beam Dialog*.

- Fetch coordinates from points A and B in the figure and edit the Y-coordinates before clicking *Apply*.
- Orientate the beam z-axis in negative X-direction and flush the beams with the plate.

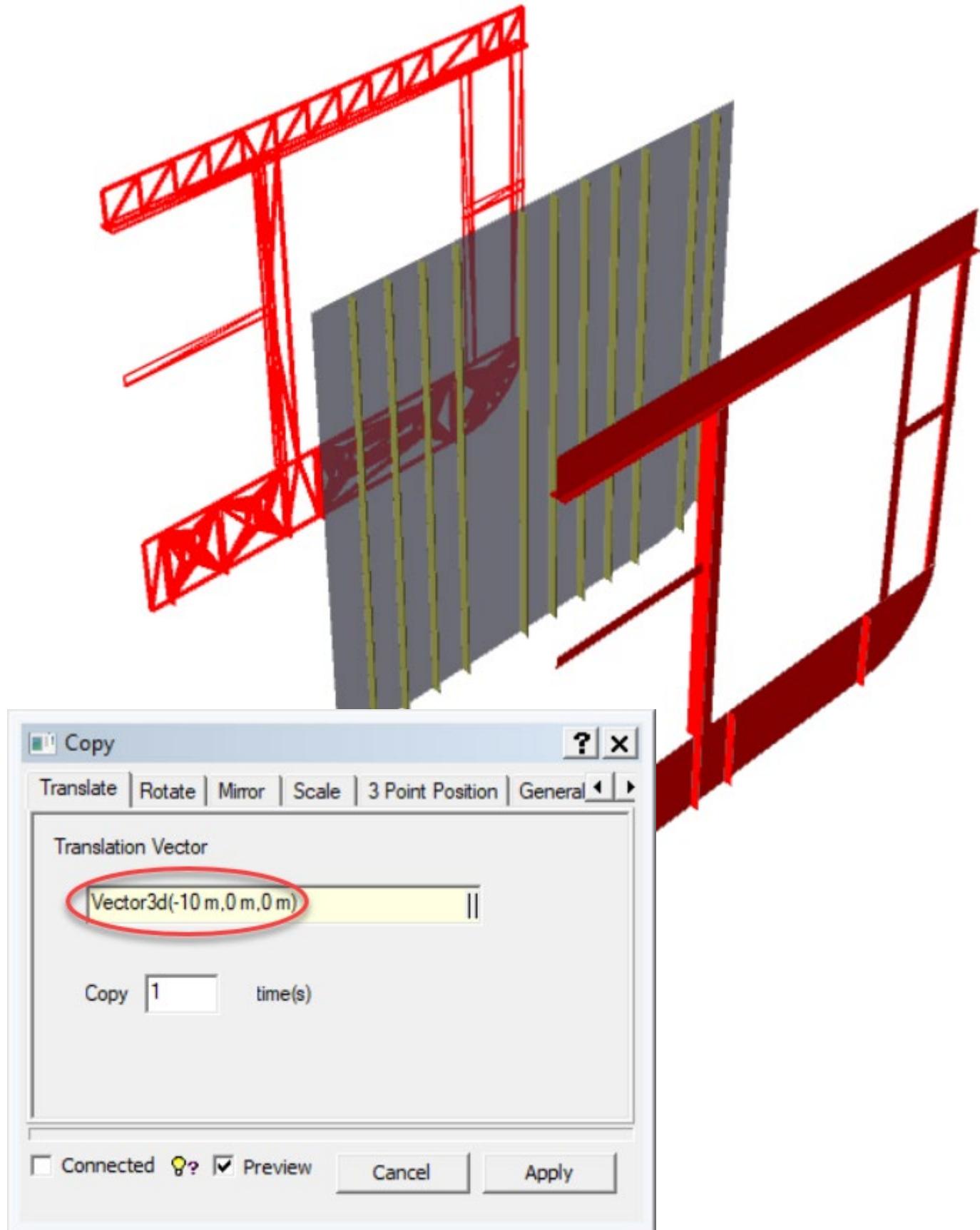


- Add beams and stiffeners as shown below (only the Fbar150x12 beams exist already).
- Rotate their axis systems to the shown orientation.
- Flush all beams. Note that flushing a beam can be done even when there is no plate to flush with. Flushing will be done to the system line.

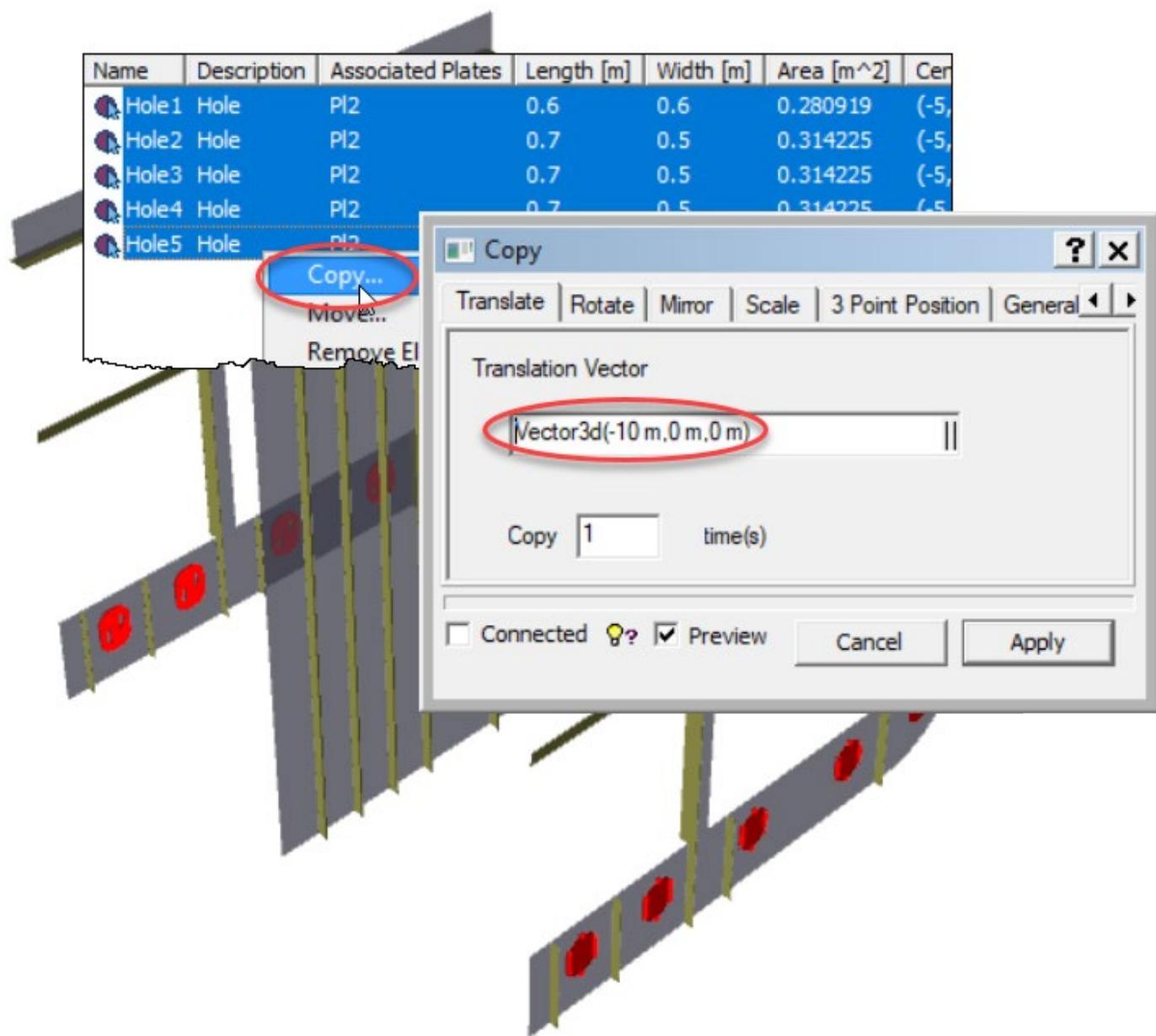


6 COPY WEB FRAME WITH STIFFENERS

- Copy the web frame at position $X = -5 \text{ m}$ to the other side of the stiffened plate at position $X = -10 \text{ m}$, i.e. a distance of -10 m in X-direction to the new position $X = -15 \text{ m}$.



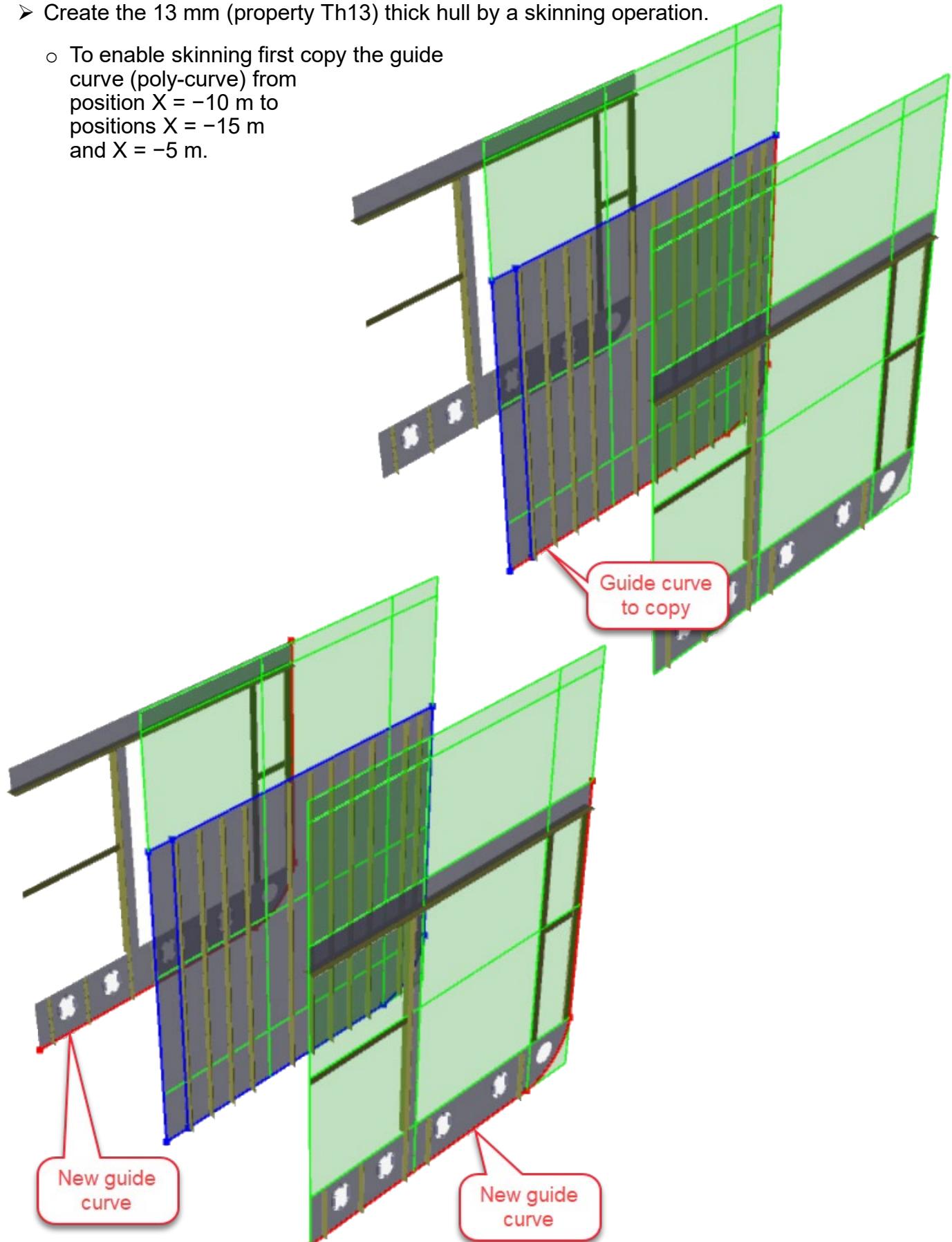
- The five holes in the original web frame created by punching it with the profiles Hole and Manhole are so-called hole concepts. This means that without these, the web frame is without holes. Consequently, these hole concepts must also be copied or else there will be no holes in the web frame just copied.
- Find and select the hole concepts in the browser folder *Structure | Holes*. Right-click, select *Copy* and in the *Copy* dialog give the vector $(-10, 0, 0)$.



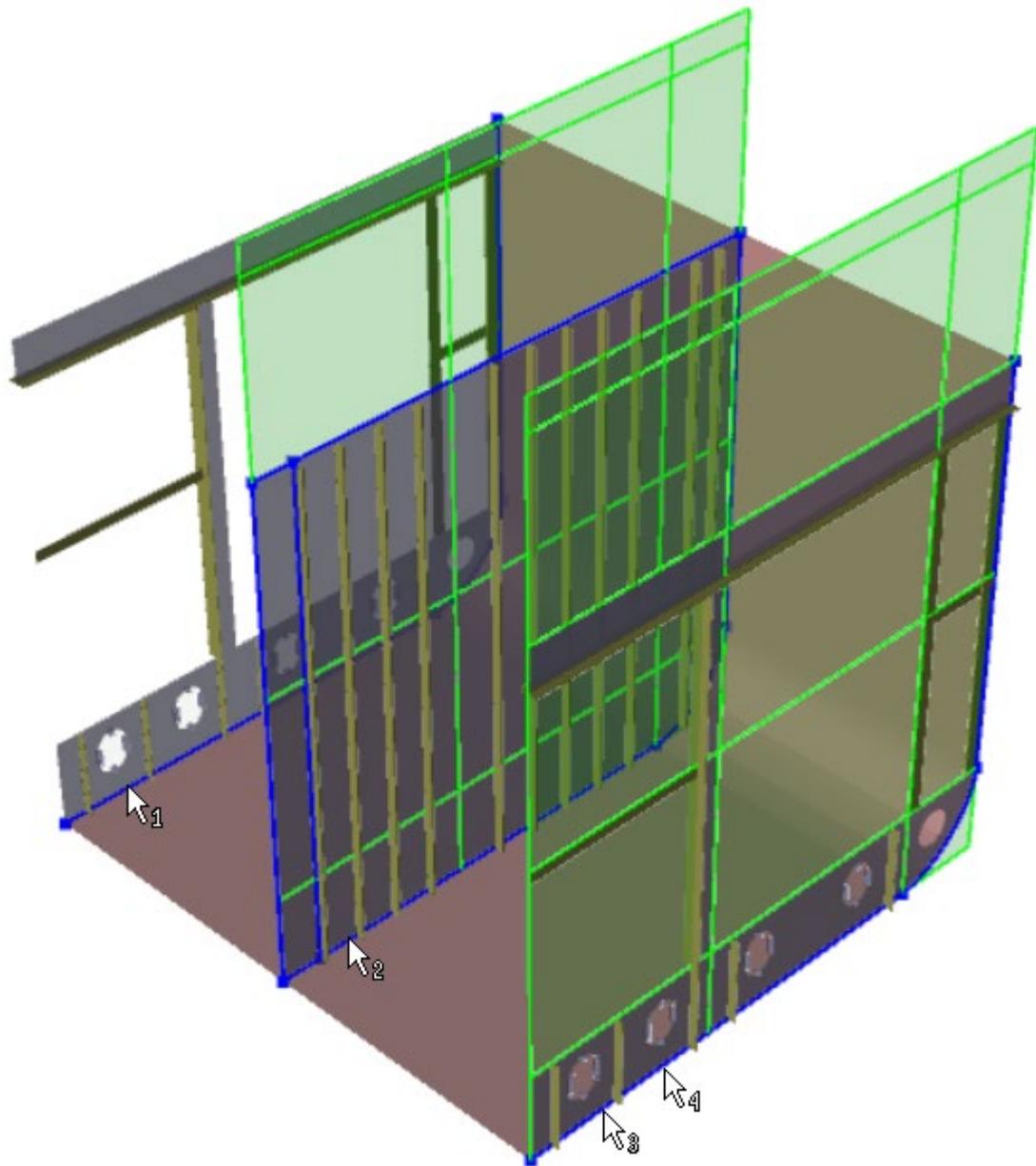
7 CREATE HULL

- Create the 13 mm (property Th13) thick hull by a skinning operation.

- To enable skinning first copy the guide curve (poly-curve) from position X = -10 m to positions X = -15 m and X = -5 m.

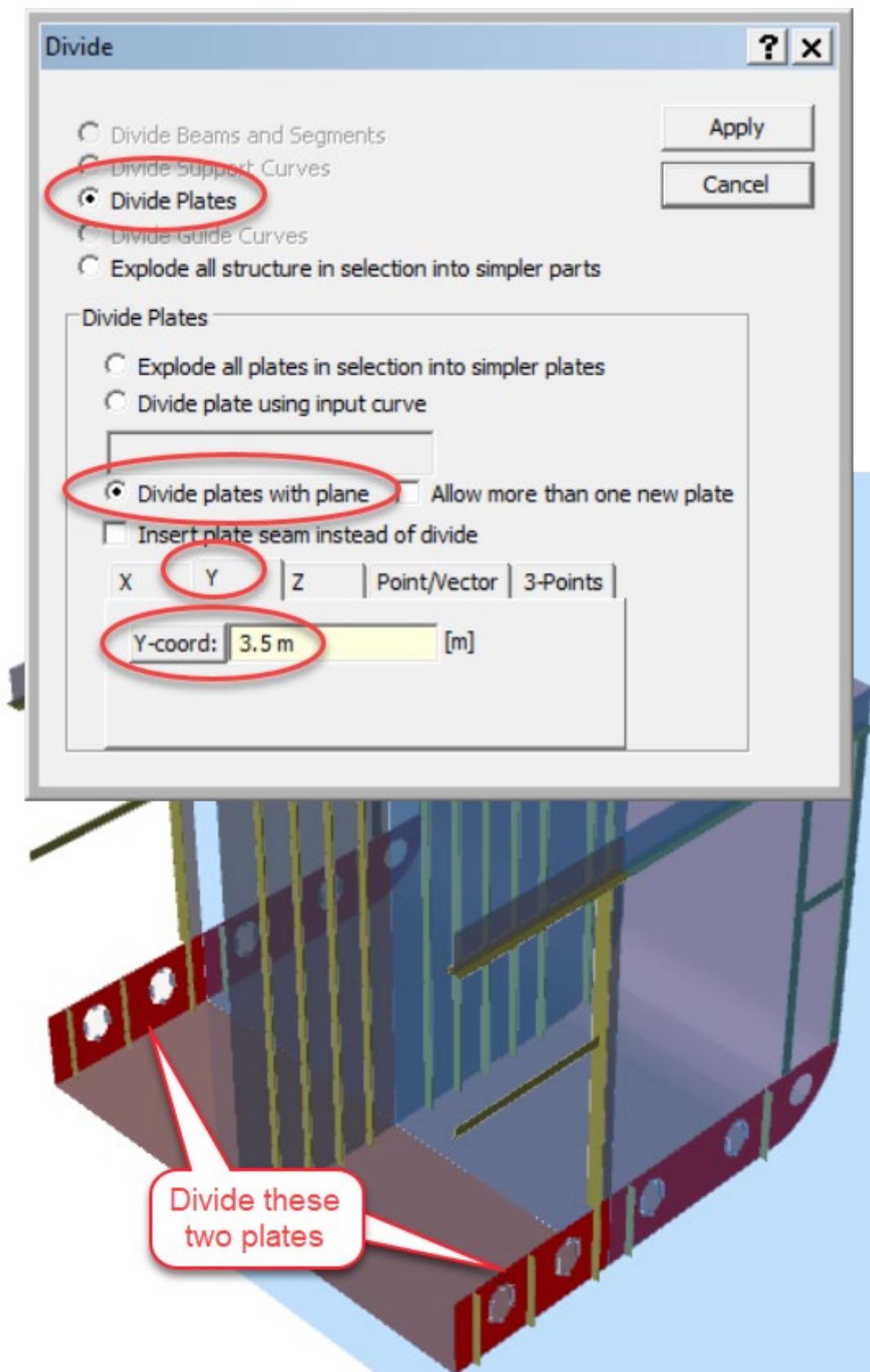


- Use *Structure | Free Form Shells | Skin/Loft Curves* to create the hull. Click the three guide curves as shown, the last curve is clicked twice to complete the operation.

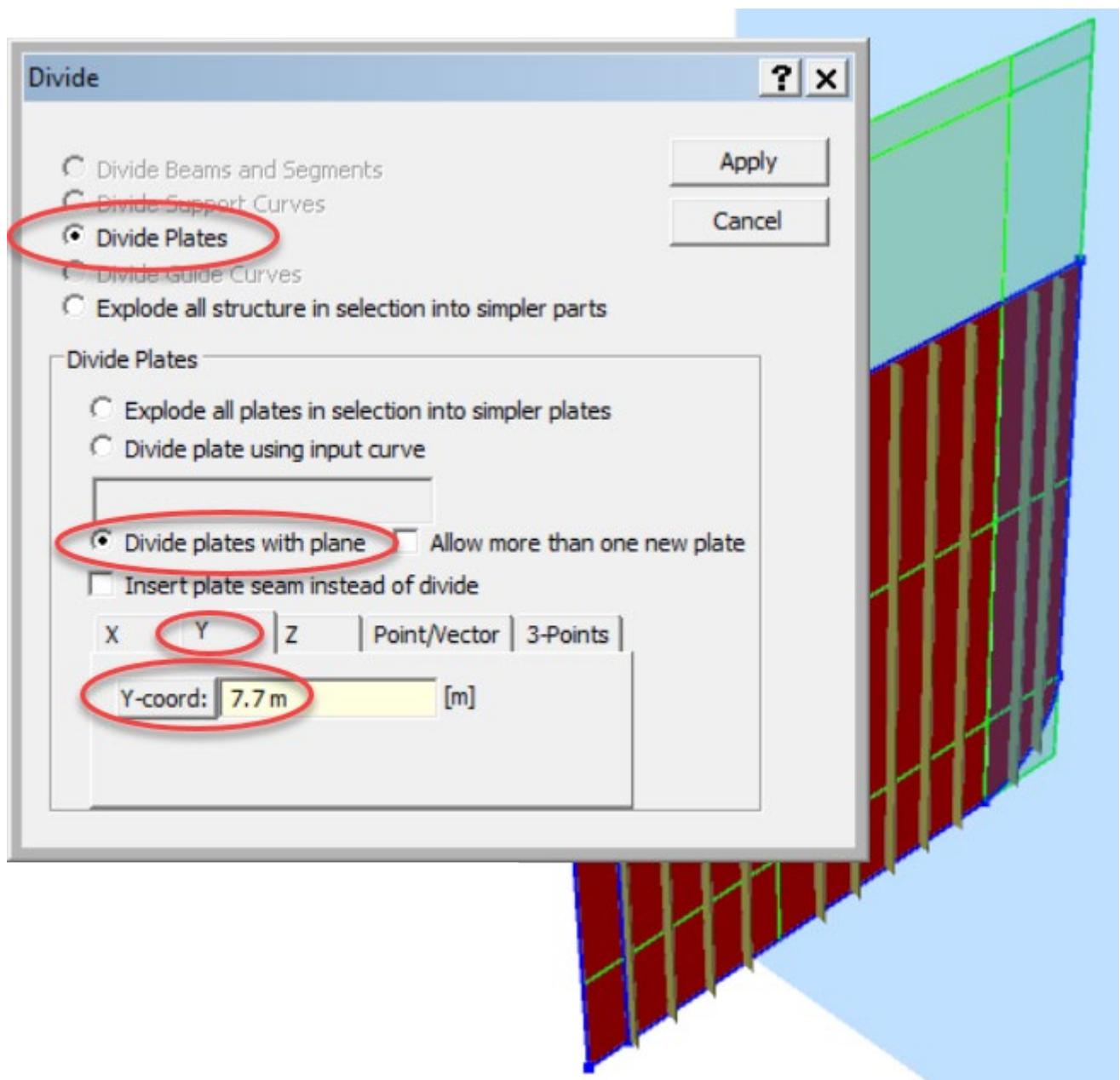


8 MODIFY THICKNESSES

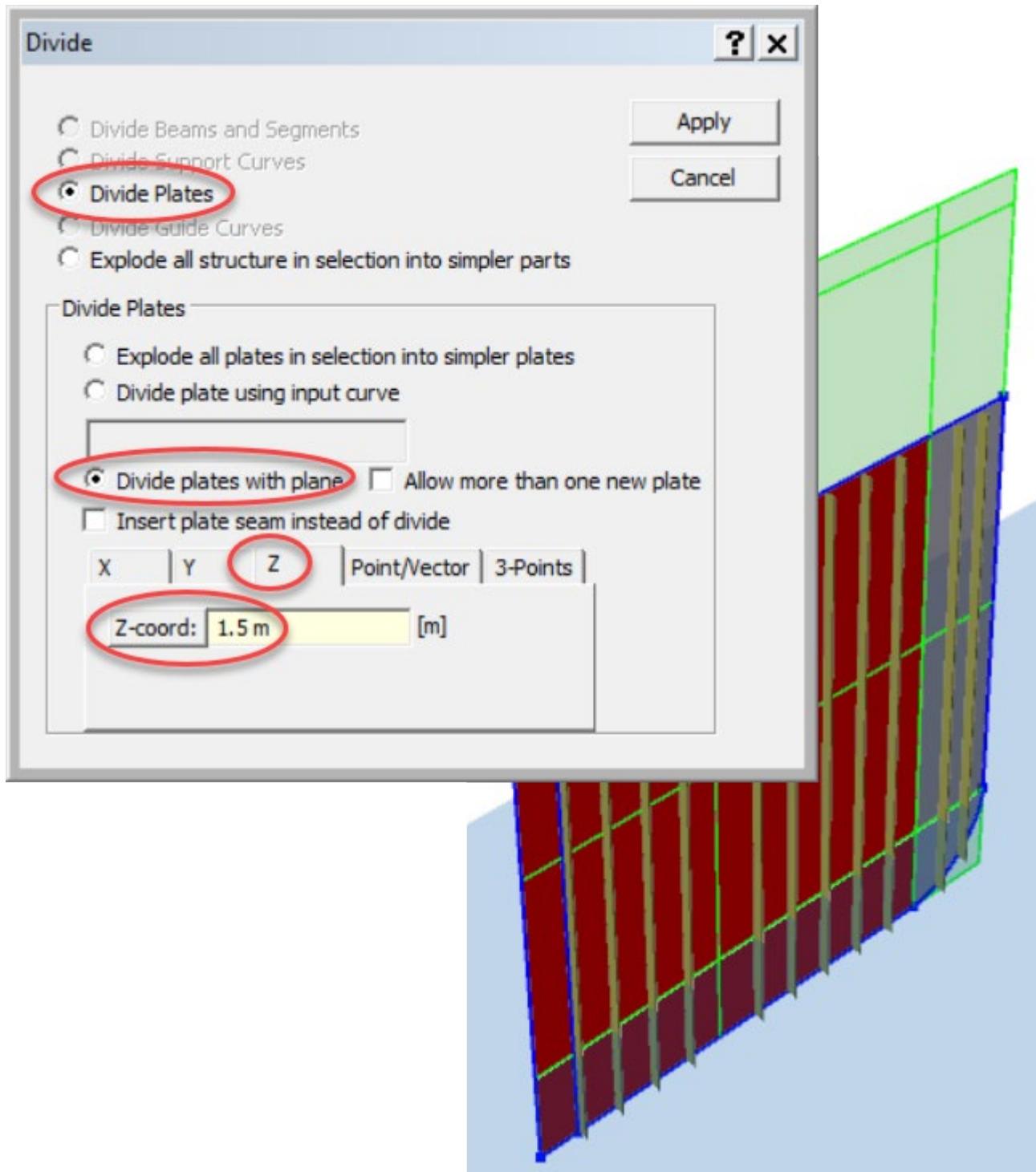
- To modify plate thicknesses, first divide the plate and web frames as explained.
 - First divide the two web frames.



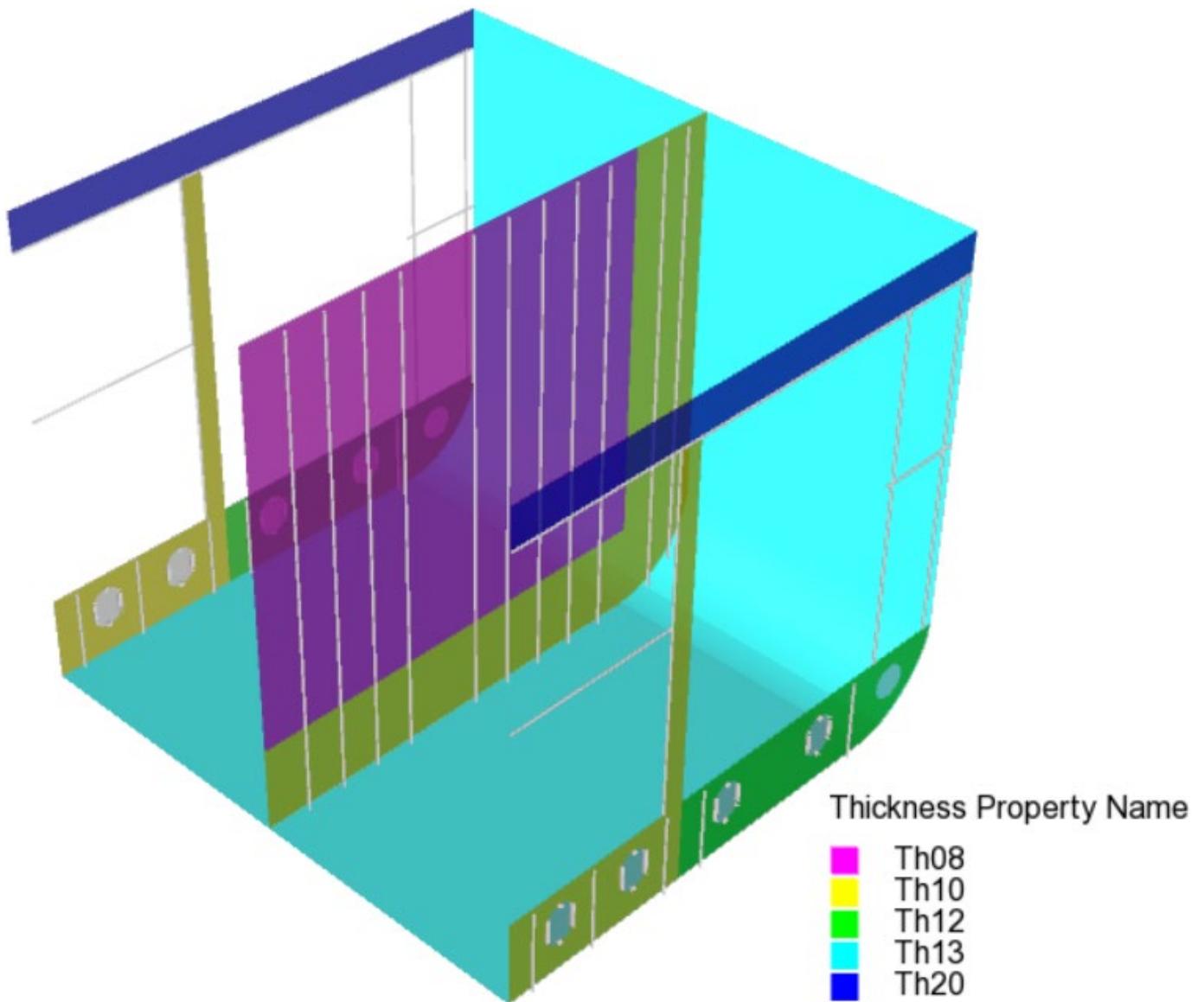
- Then divide the plate at position $X = -10 \text{ m}$ vertically at $Y = 7.7 \text{ m}$, i.e. at a guide plane line.



- Finally, divide the plate at position $X = -10 \text{ m}$ horizontally at $Z = 1.5 \text{ m}$, i.e. at a guide plane line.

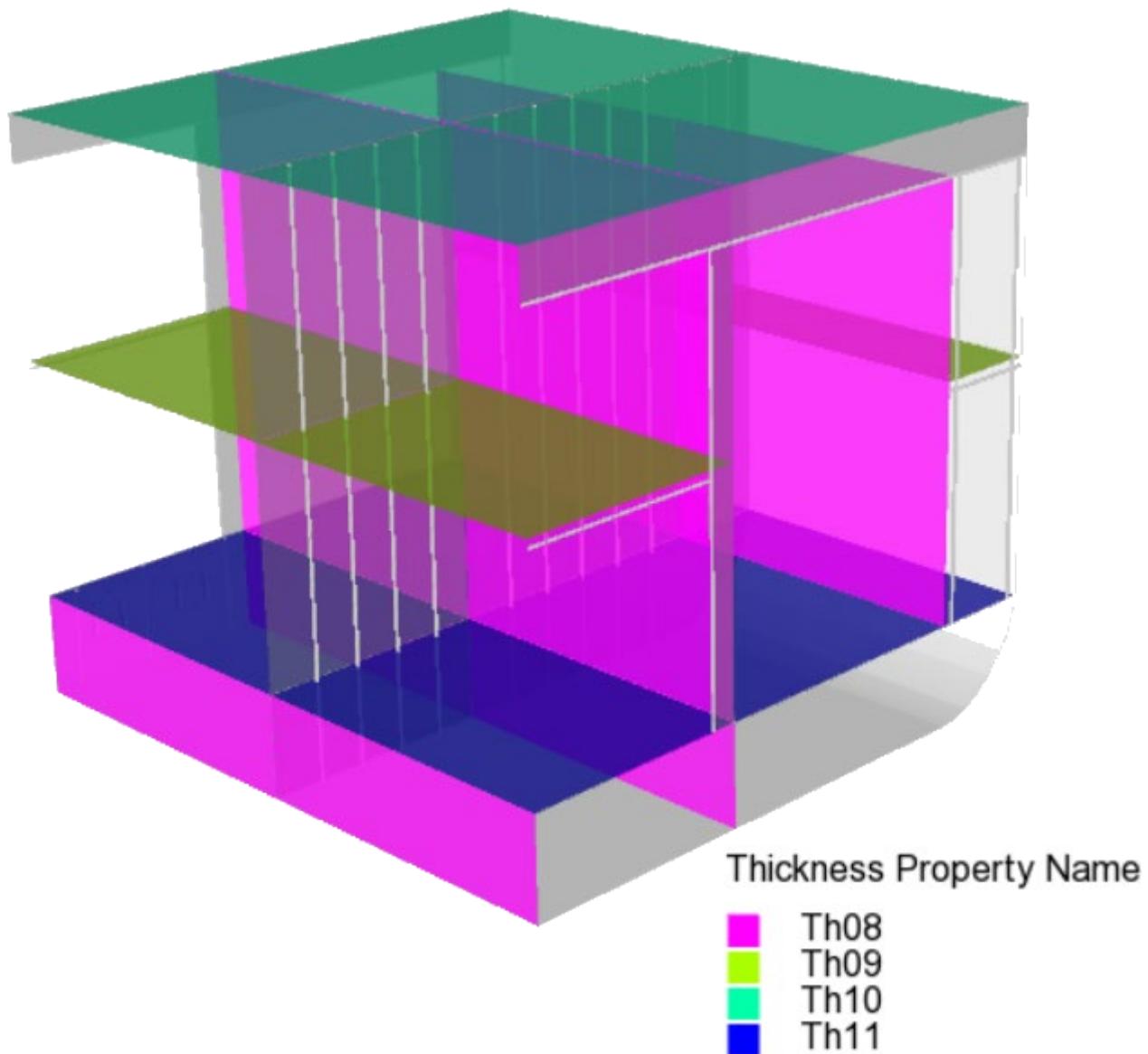


- Modify the plate and shell thicknesses to match the colour coding below.
 - Beams are displayed in wireframe mode.

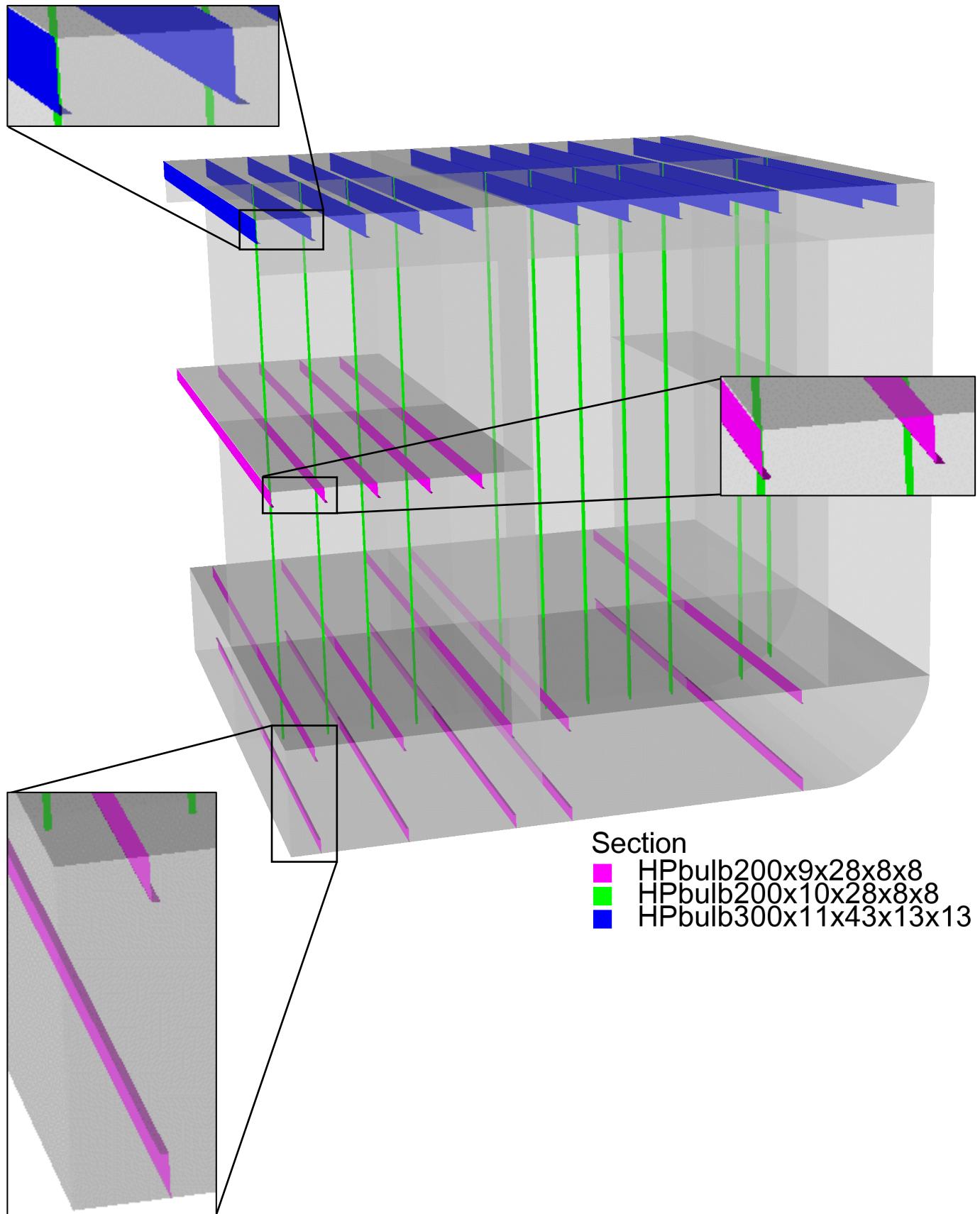


9 CREATE DECKS WITH STIFFENERS

- Use *Structure | Flat Plates | Flat Plate* to create the decks coloured with thickness properties below.
- Also create three plates in the XZ-plane, all with thickness property Th08.



- Add stiffeners to decks. Remember to flush them with the decks.

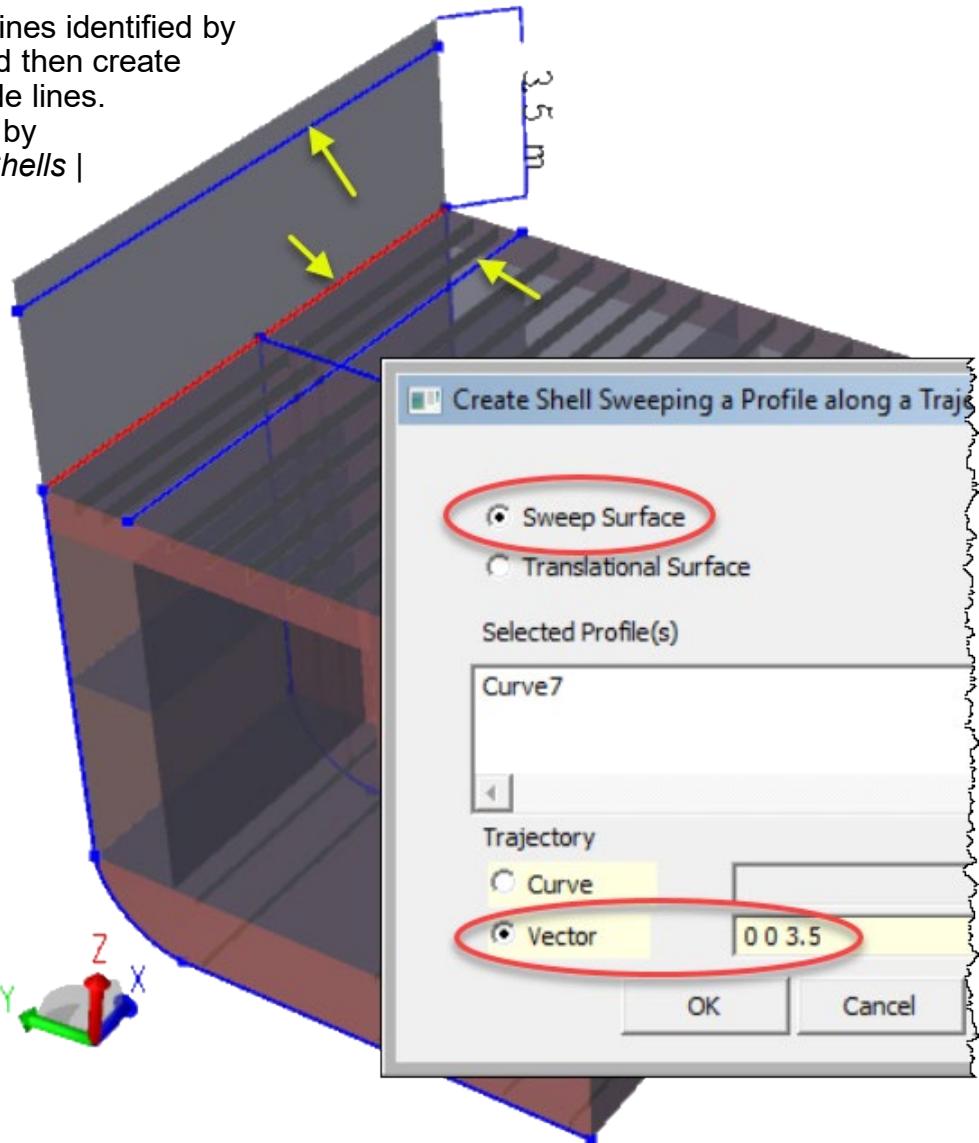


10 CREATE THE CARGO RAIL

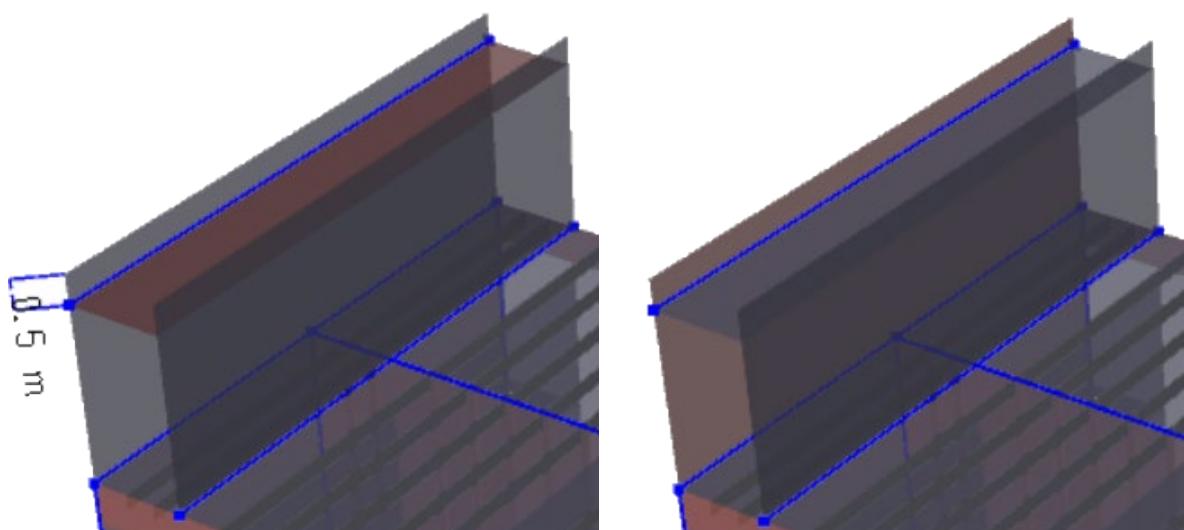
- Create the three guide lines identified by yellow arrows below and then create plates by sweeping guide lines.

Sweeping is accessible by
Structure | Free Form Shells | Sweep Curves Dialog.

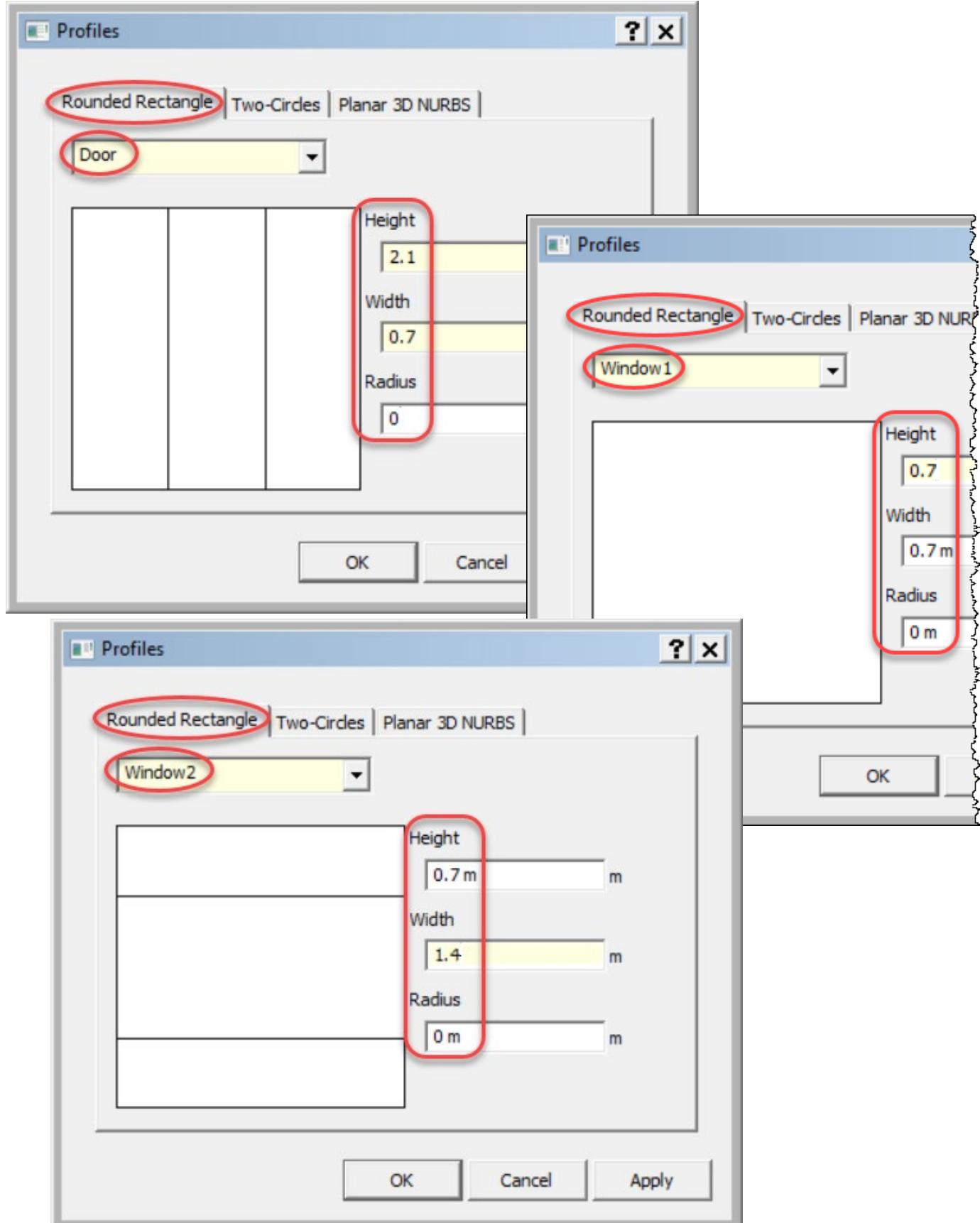
- Sweeping is also accessible by selecting a guide line, right-clicking and selecting *Sweep Curves*.



- Having created the three new plates shown to the left below, ensure that their plate normals are consistent with the other plates as shown to the right below. (Bluish colour on the positive surface side and reddish on the negative side.)

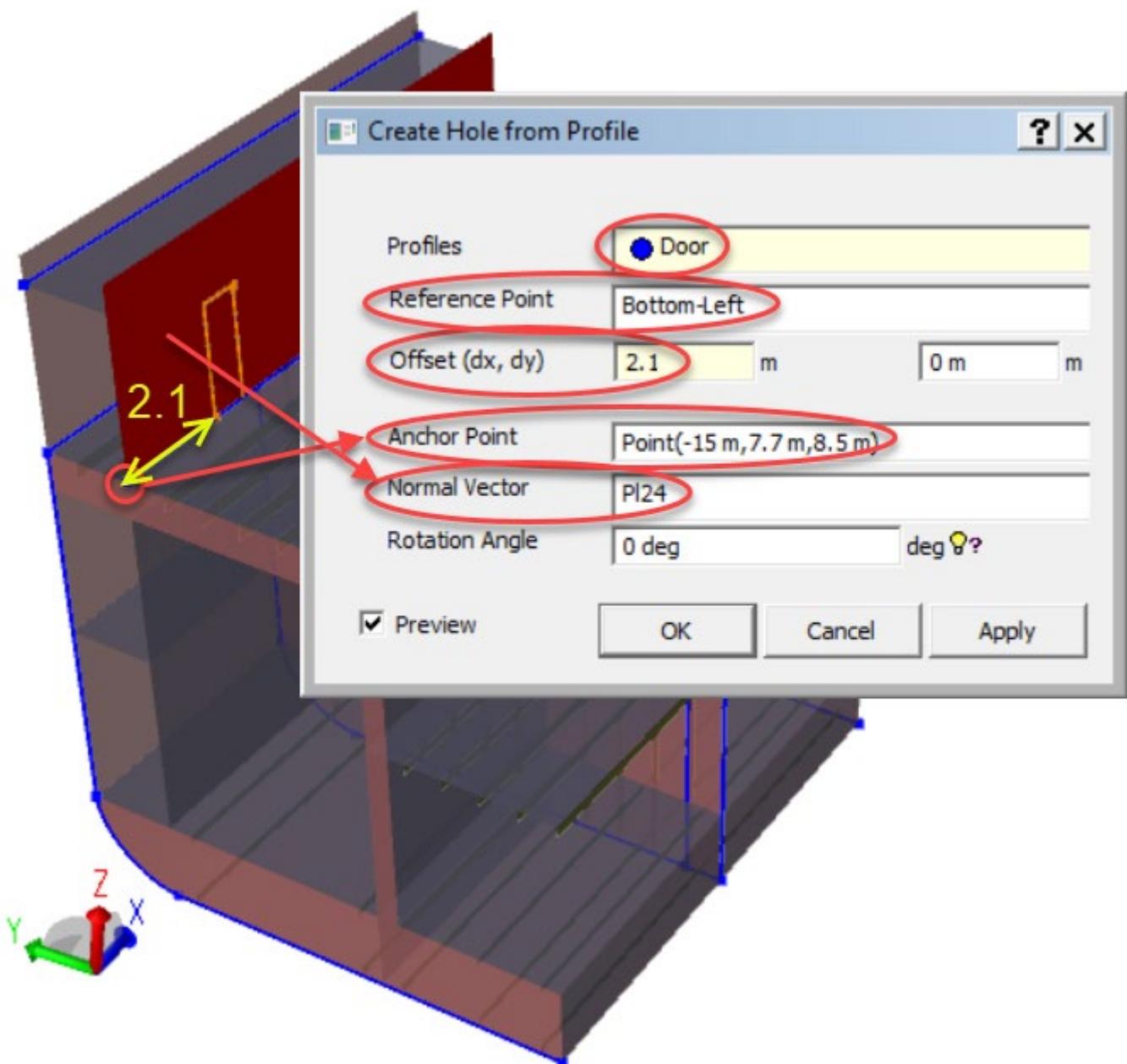


- The cargo rail has openings, so it must be punched. Define the three profiles Door, Window1 and Window2 below by *Guiding Geometry | Profiles | Profile Dialog*.

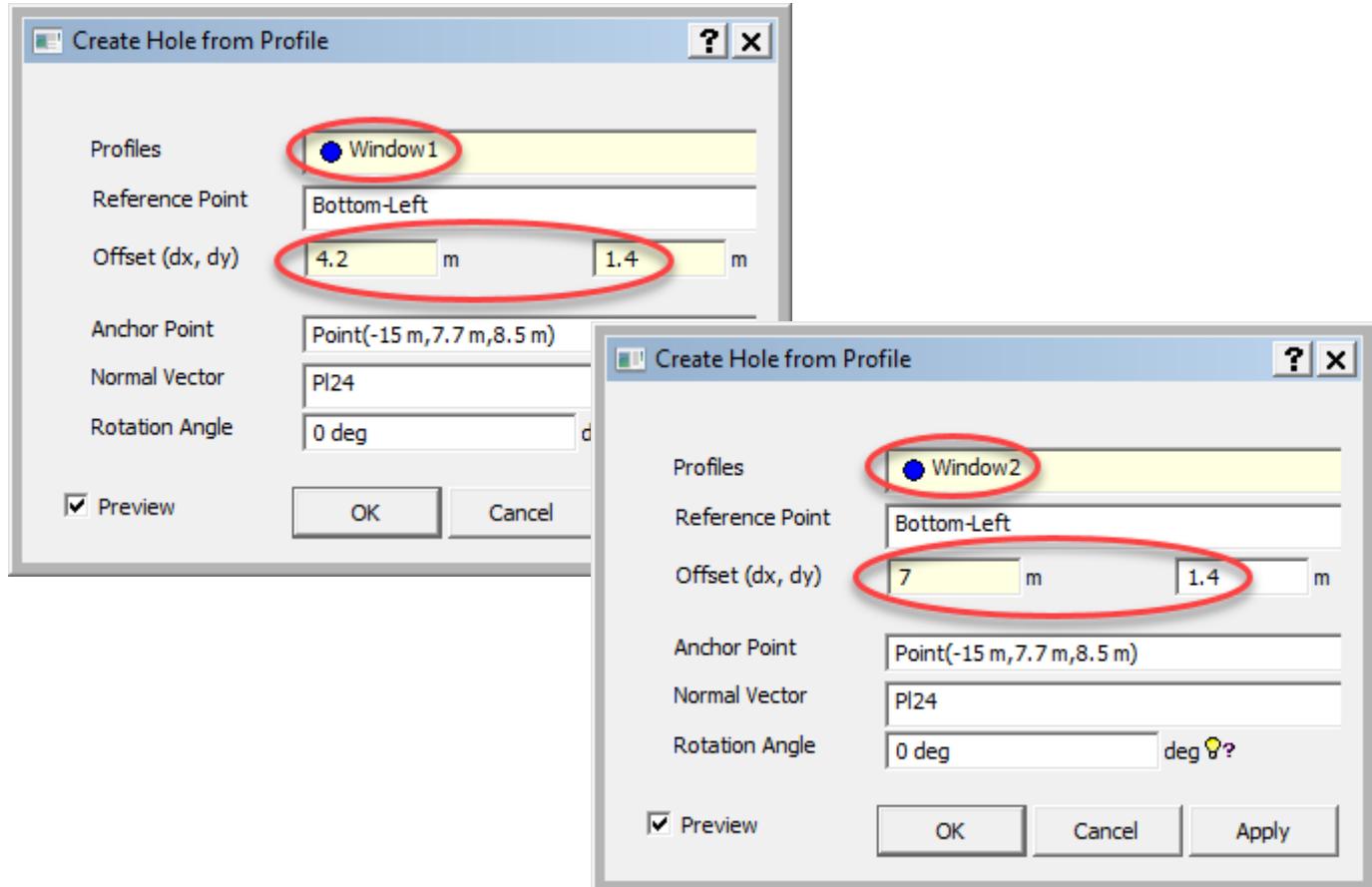


- Punch a hole using the a profile by *Structure | Features | Hole from Profile*.

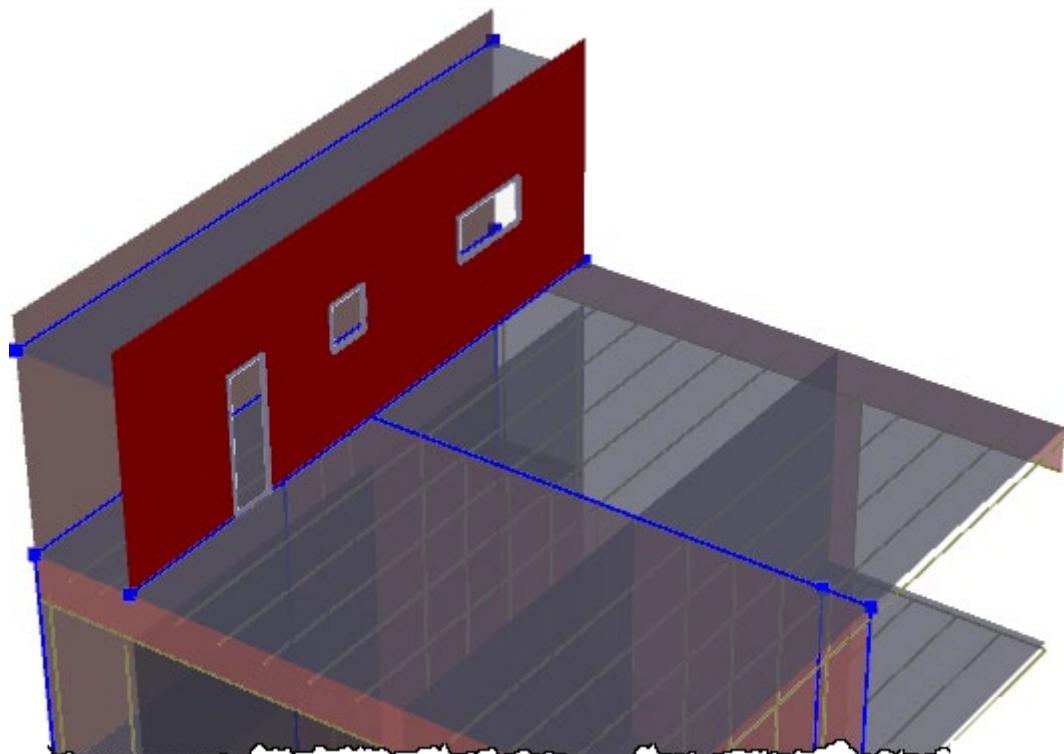
- In the *Create Hole from Profile* dialog, select the profile *Door*, *Reference Point Bottom-Left* (of the Door), *Anchor Point* by clicking the geometry point shown, and give *Offset* as distance from *Anchor Point* to *Reference Point*. The offset dx and dy are in a local profile coordinate system rather than in model coordinates. *Normal Vector* is specified by clicking the appropriate plate.



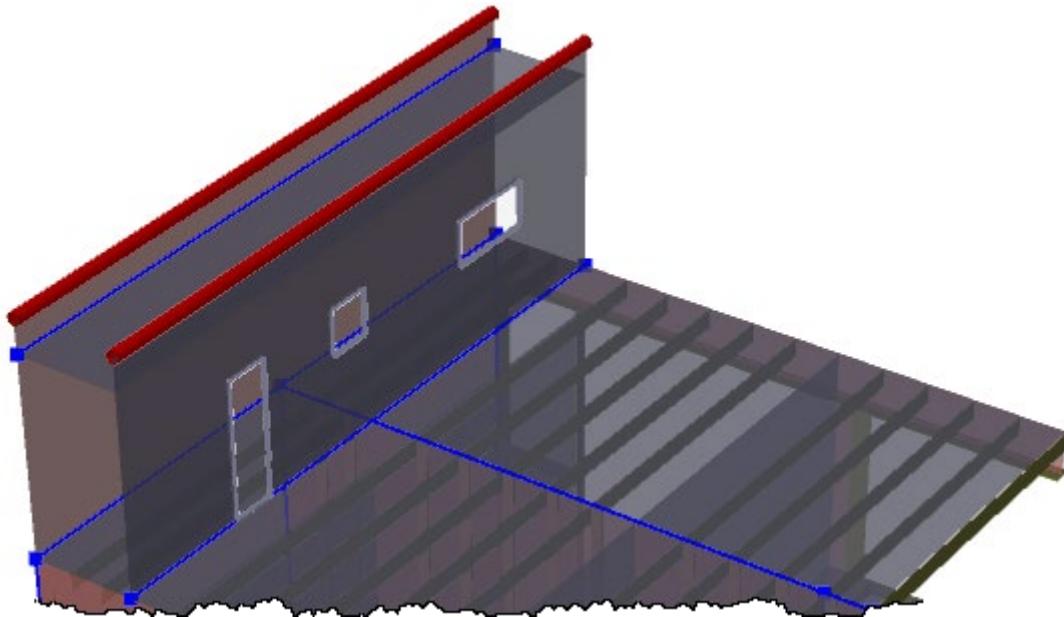
- Repeat the process by punching holes using the Window1 and Window2 profiles. The *Anchor Point* and *Normal Vector* are the same for all.



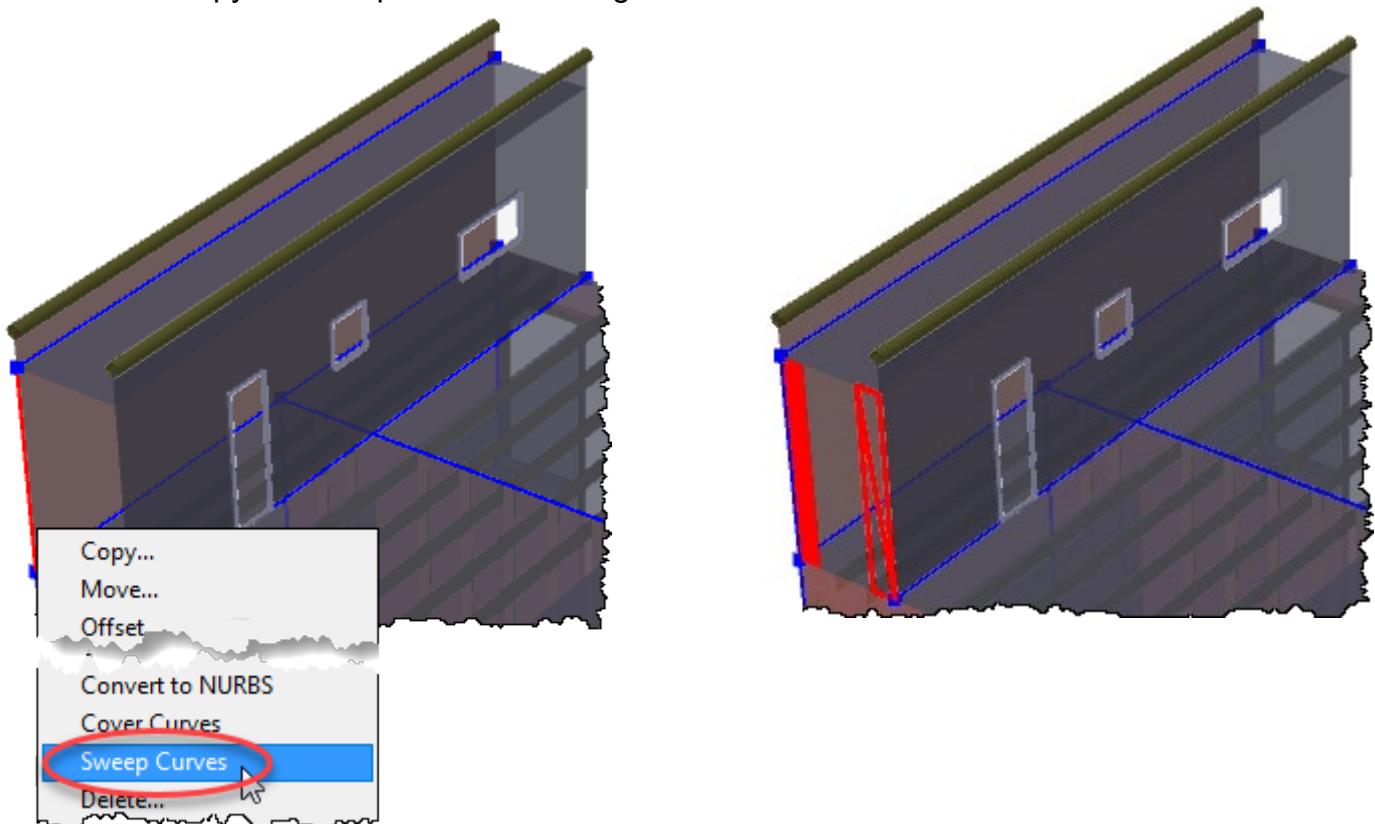
- The result is shown below.



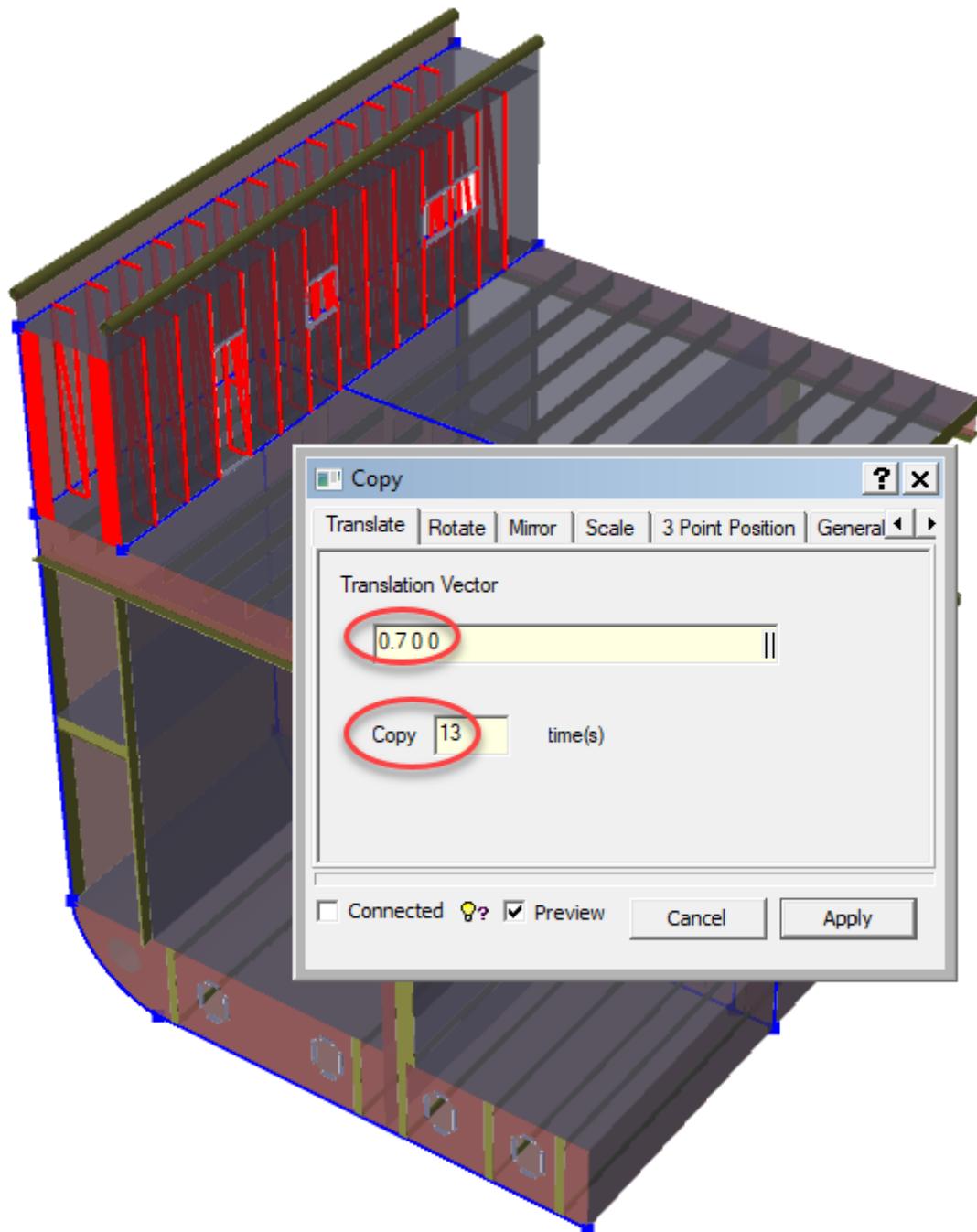
- Create cargo rail beams for a crane.
 - Set Rail as default beam cross section.
 - Click in the model to insert the two beams highlighted below.



- Create the two 0.35 m wide and 8 mm thick stiffener plates highlighted to the right below.
 - First, create a new vertical guide line and sweep it in 0.35 m in negative Y-direction.
 - Then copy the new plate 1.2 m in negative Y-direction.



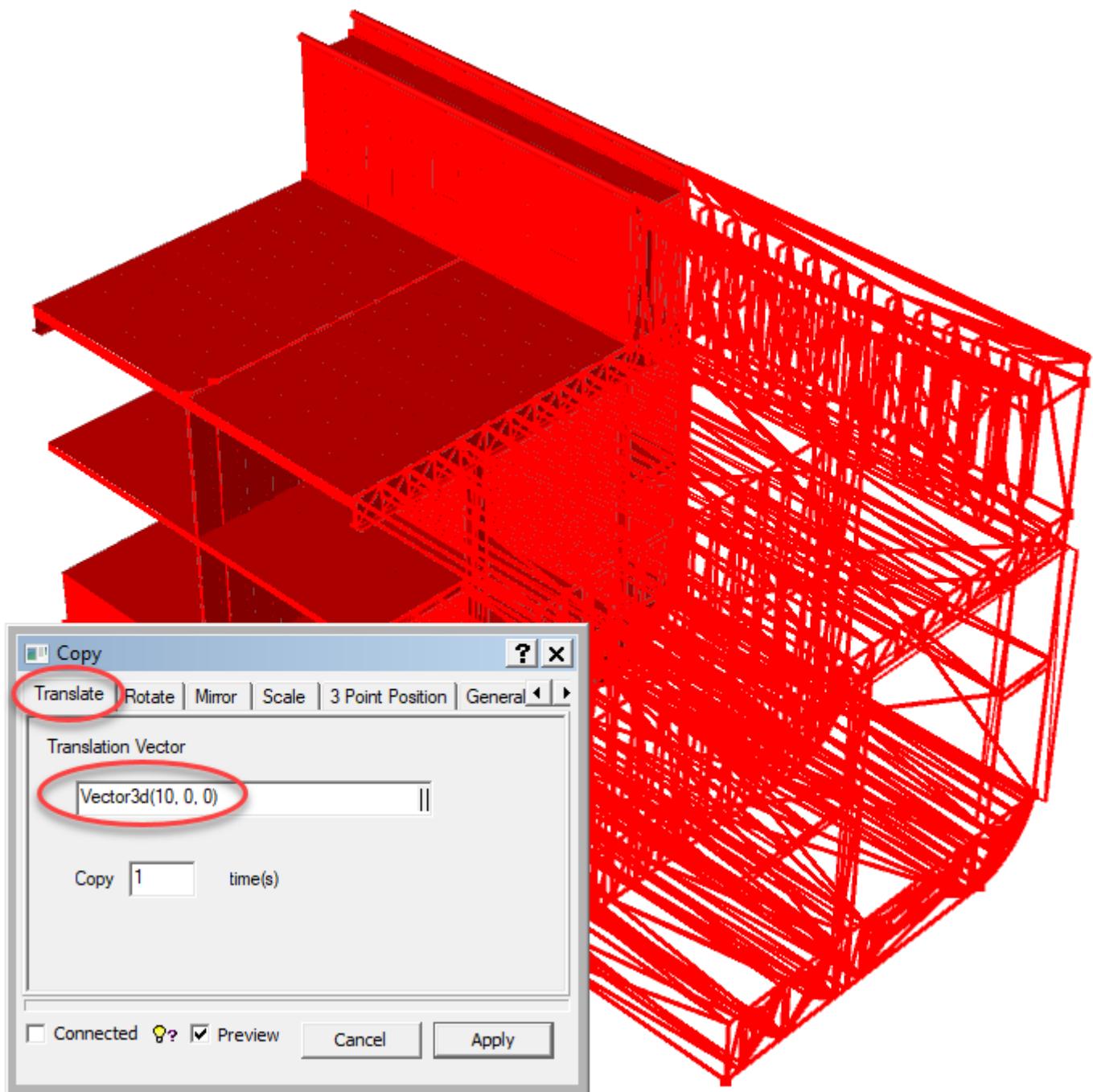
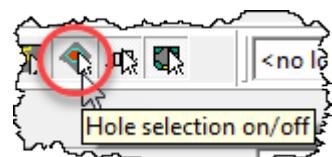
- Copy the two newly created stiffener plates with a spacing of 0.7 m in X-direction 13 times.



11 COPY THE MODEL

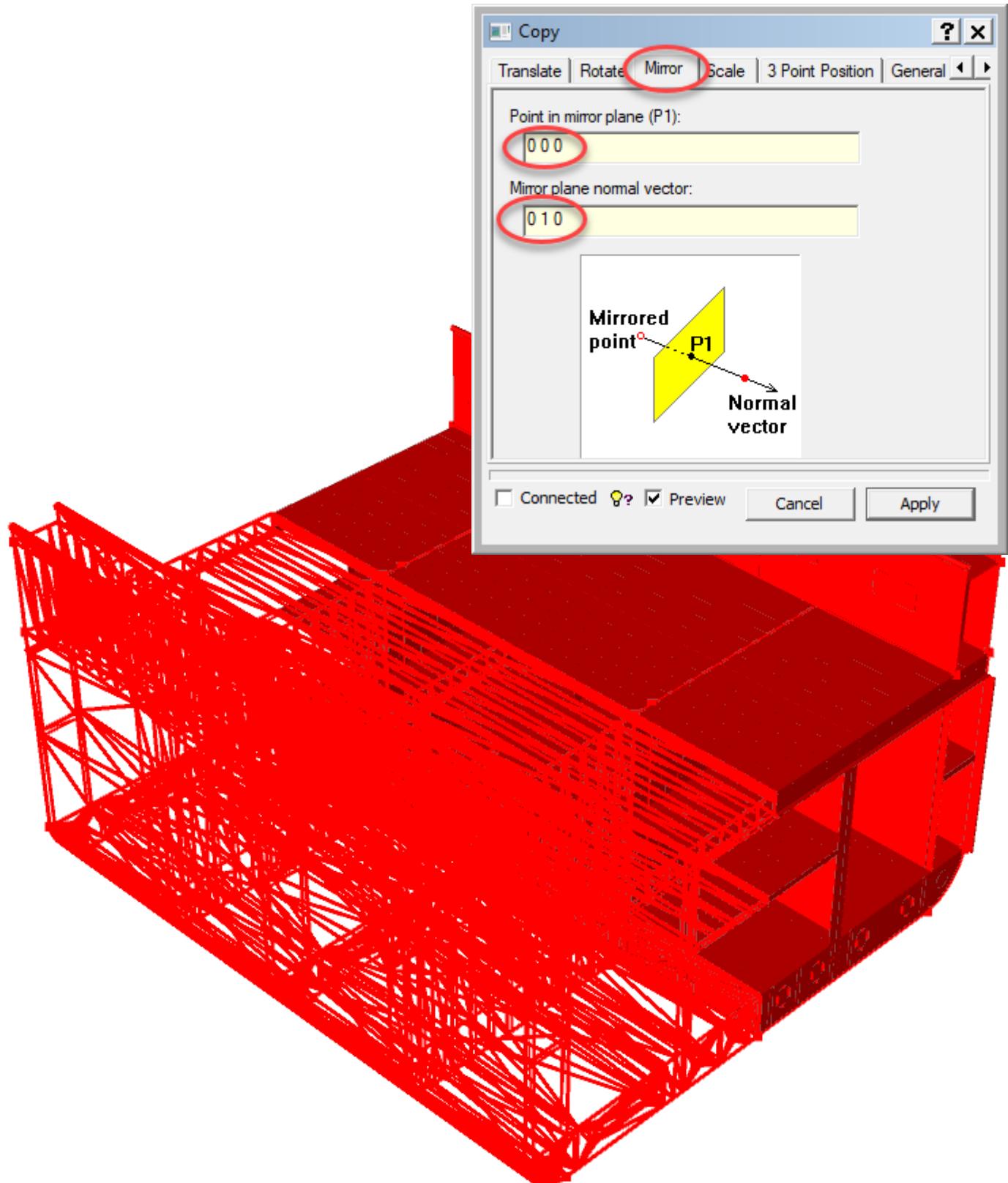
➤ Copy the model in X-direction.

- When dragging a rubberband to select the whole model, ensure that all parts of it are selected by having all selection buttons pressed down including, for example, the *Hole selection* button.
- Include guide lines/curves but not guide planes in the copy operation.



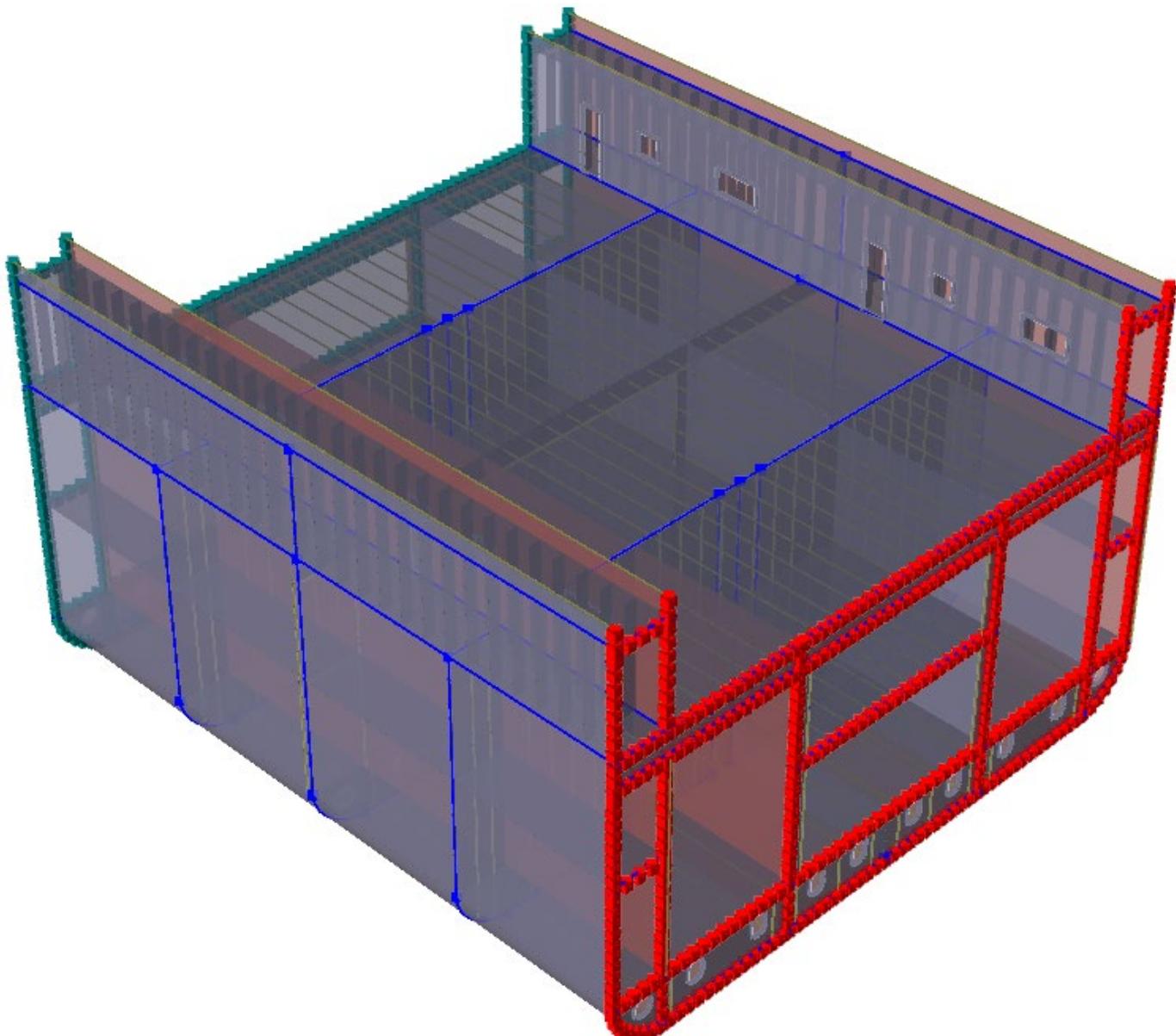
➤ Note that beams, plates and profiles in the $X = -15$ m plane will not be copied as that would cause double geometry at $X = -5$ m.

- Copy the complete model by a mirror operation to create the starboard side of the ship section.
- Again, geometry in the mirroring plane will not be copied to avoid double geometry.



12 INTRODUCE BOUNDARY CONDITIONS

- Introduce fixed boundary conditions at all edges in the planes $X = -15\text{ m}$ and $X = 5\text{ m}$.
- Where there are guide lines/curves, select these, right-click and select *Create Support Curve*.
 - Where this is not the case, there are two options:
 - Double-click the surfaces to show their edges. Select the appropriate edges, right-click and select *Create Support Curve*.
 - Create a guide line along the edge, select and right-click this to *Create Support Curve*.
 - Having created all support curves in one of the planes, copy these to the other plane.



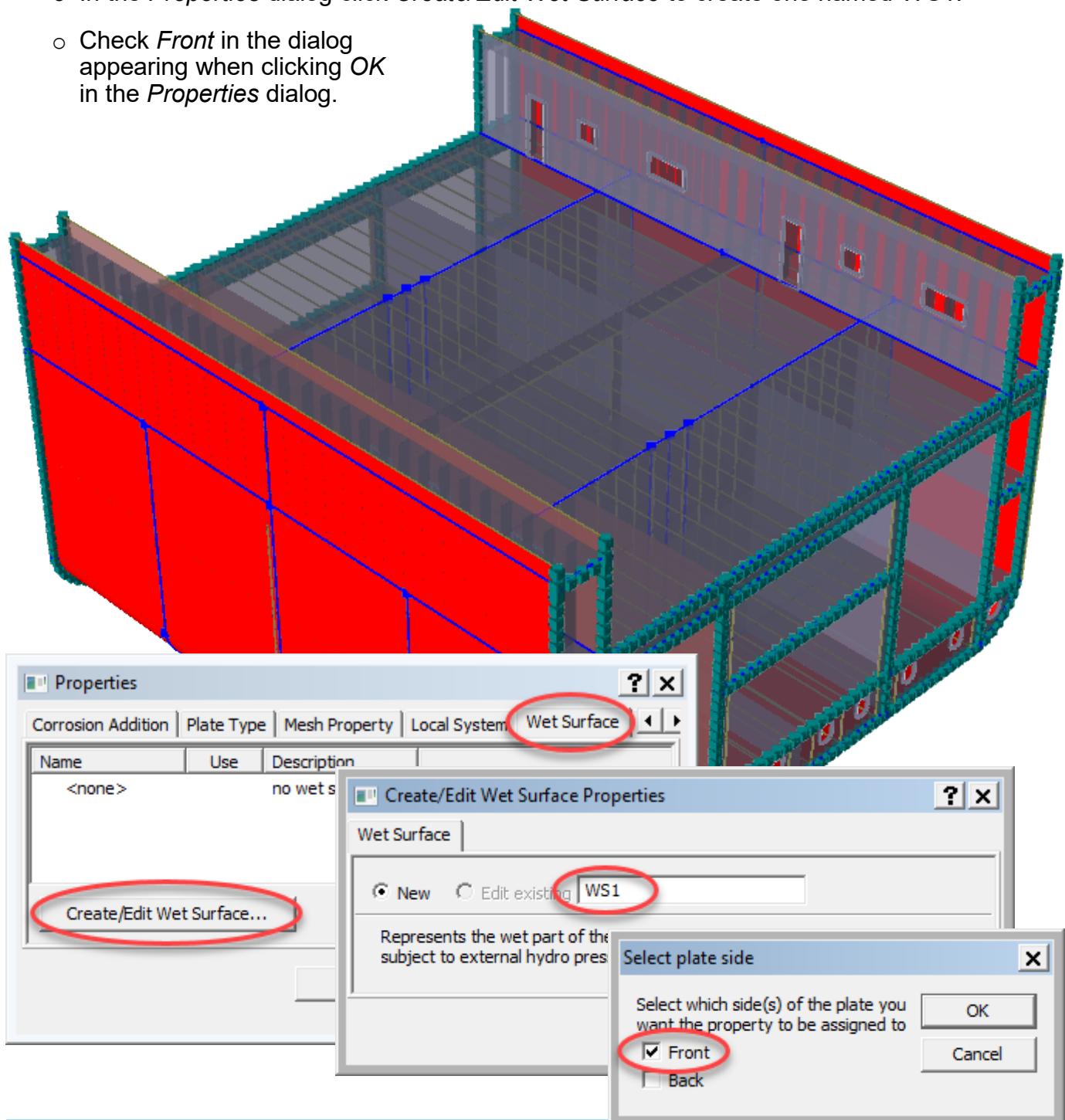
13 CREATE LOADS

- Use *Loads | Load Case* to create a load case named Gravity.

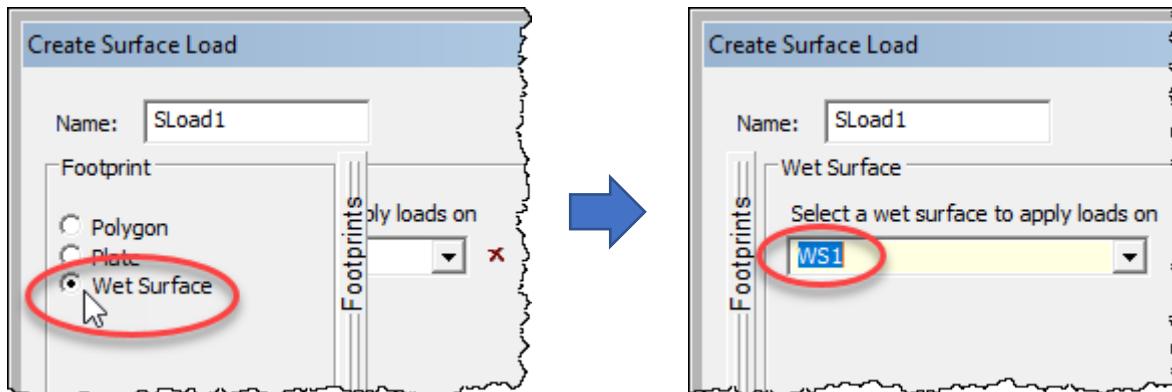
 - Right-click it in the *Analysis | Load Cases* folder to open the *Properties* dialog. In this dialog check *Include structure self-weight in structural analysis*.

- Create another load case named SeaPressure being water pressure on the outer hull.

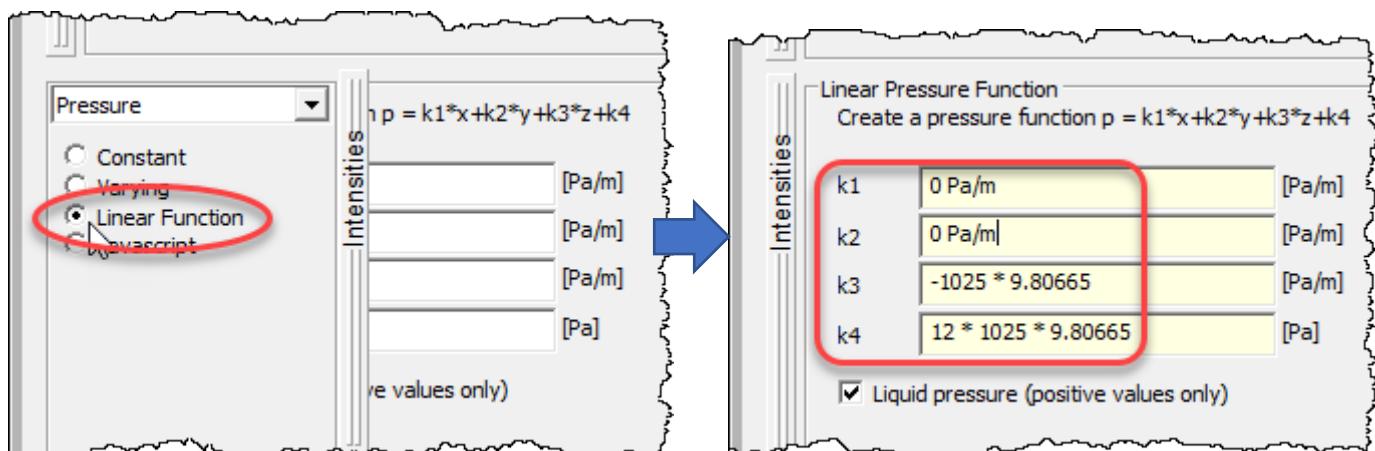
 - First create a so-called wet surface property for the hull by selecting the hull, right-clicking it and opening the *Properties* dialog.
 - In the *Properties* dialog click *Create/Edit Wet Surface* to create one named WS1.
 - Check *Front* in the dialog appearing when clicking *OK* in the *Properties* dialog.



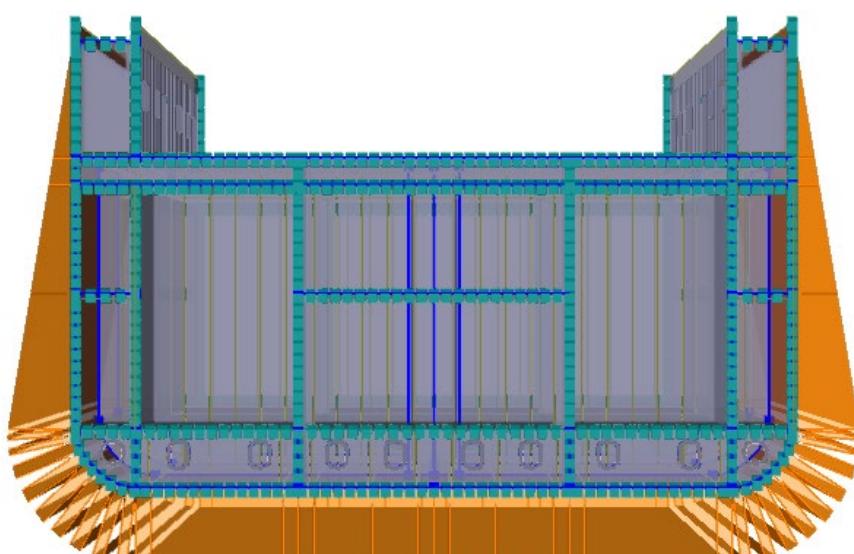
- Use Loads | Explicit Load | Surface Load. In the dialog opening up, hover the pointer over the *Footprint* 'curtain' to select the *Wet Surface* and then WS1.



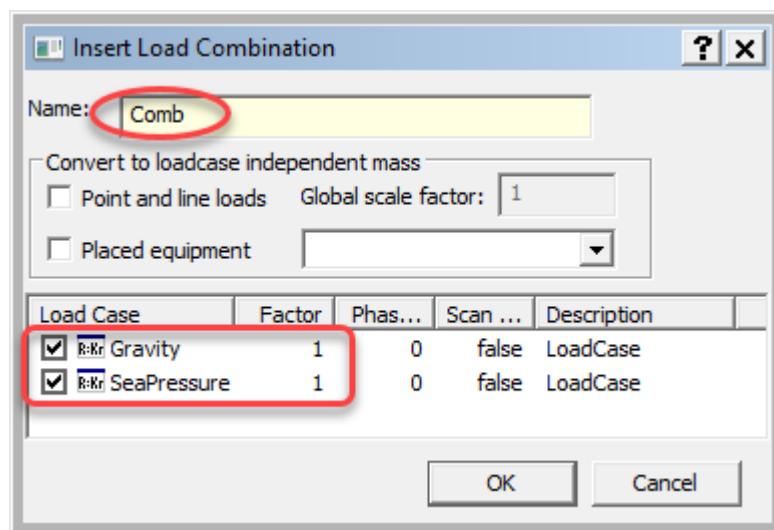
- In the same dialog, hover over the *Intensities* 'curtain' to select *Linear Function* and give values for the linear function as shown.



- Notice that for Z = 12 m (at top of the model) the value of the function $p = k1*x + k2*y + k3*z + k4$ is 0, and for Z = 0 m the value is 110,570 Pa.
- The water pressure load is displayed below, seen in negative X-direction.

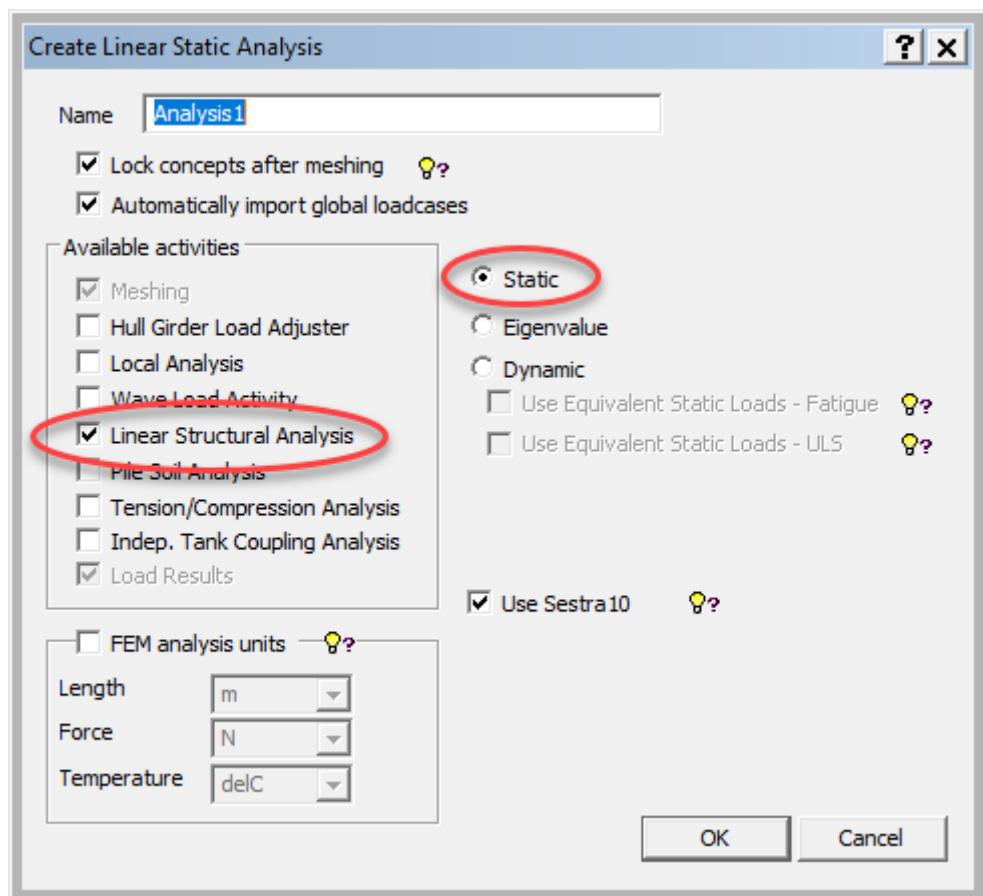


- Use *Loads | Load Combination* to create a combination named Comb of the two load cases.
 - Use factor 1.0 for both load cases.

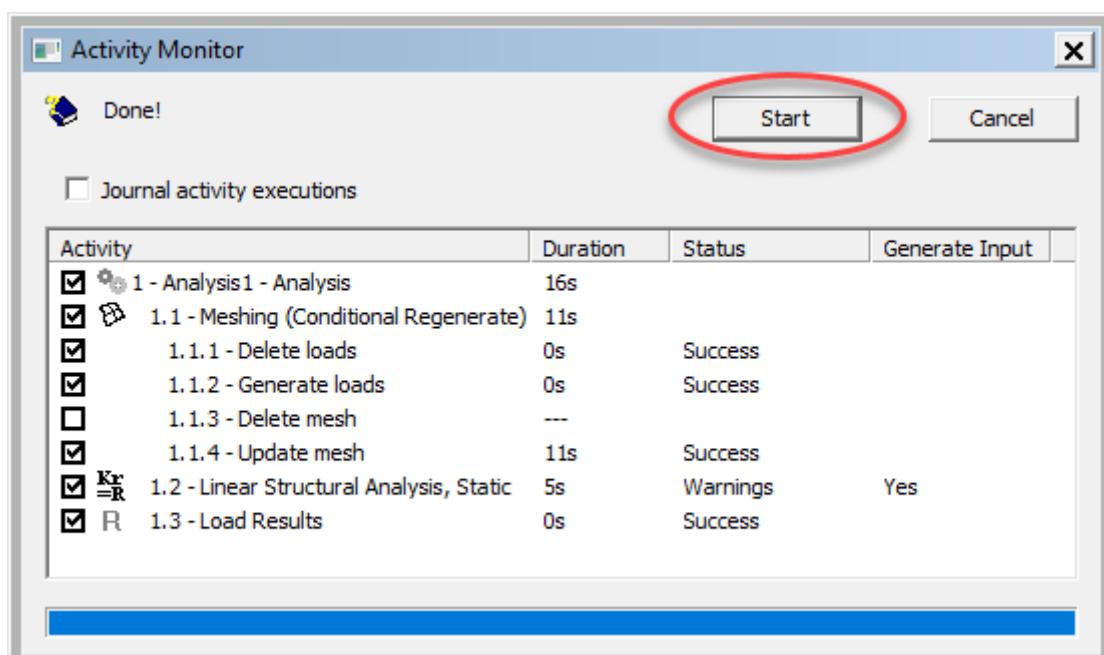


14 RUN THE ANALYSIS

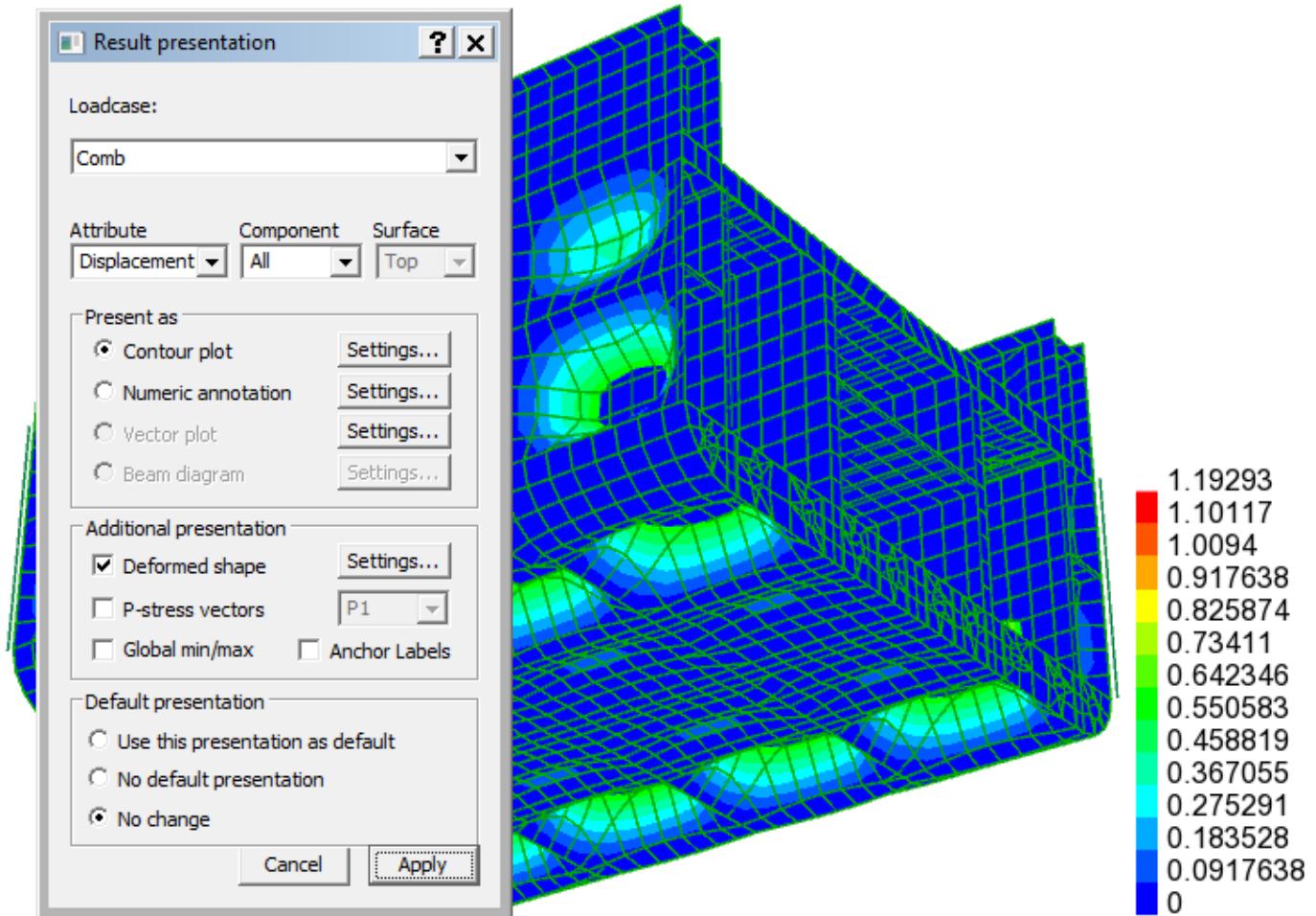
- Use Alt+D to create a *Static Linear Structural Analysis*, i.e. running Sestra.



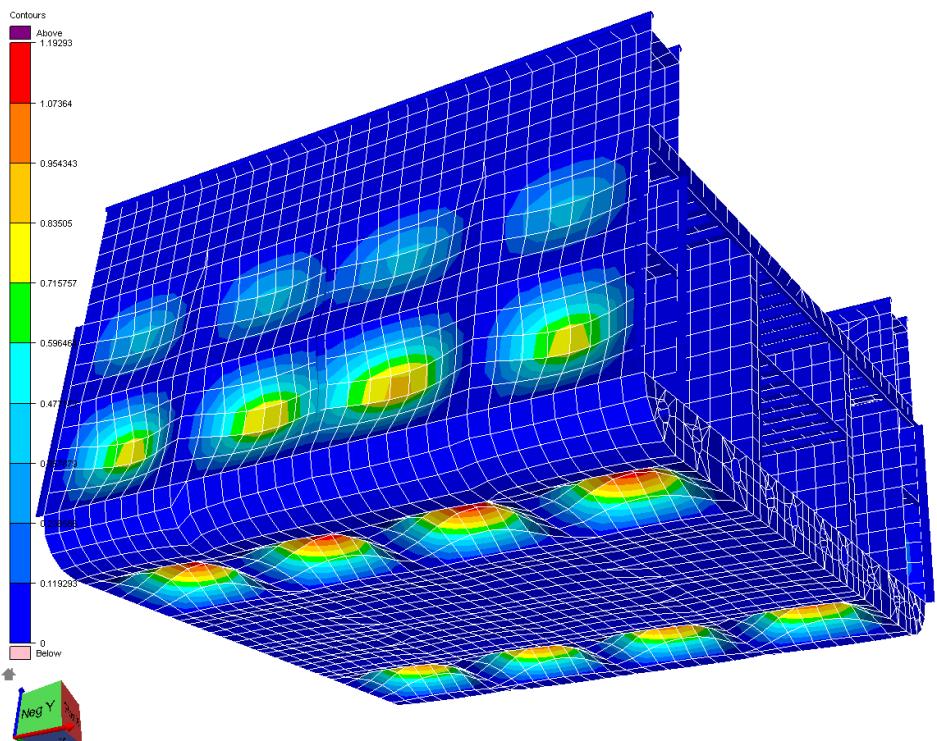
- Click *Start* in the *Activity Monitor* to run the analysis. Warnings in the Sestra.mlg file about bad element shapes are likely to occur since no mesh properties have been assigned to the model. In fact, the FE mesh is determined solely by the geometry of surfaces and beams.



- Switch to *Results - All* display configuration and use Alt+P to open the *Result Presentation* dialog to select some result to show.



- The results may also be viewed in Xtract by **Results | Advanced Results (Xtract)**.



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.