

SESAM TUTORIAL

GeniE

Tubular Joint Modelling

Valid from program version 8.2





Sesam Tutorial

GeniE – Tubular Joint Modelling

Date: September 2021

Valid from GeniE version 8.2

Prepared by: Digital Solutions at DNV

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

E-mail sales: digital@dnv.com

© DNV AS. All rights reserved

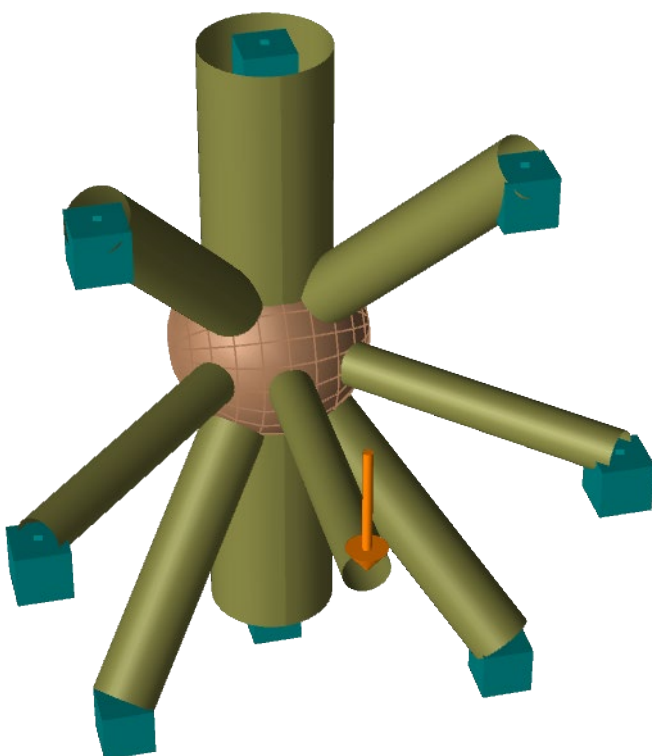
This publication or parts thereof may not be reproduced or transmitted in any form or by any means, including copying or recording, without the prior written consent of DNV AS.

TABLE OF CONTENTS

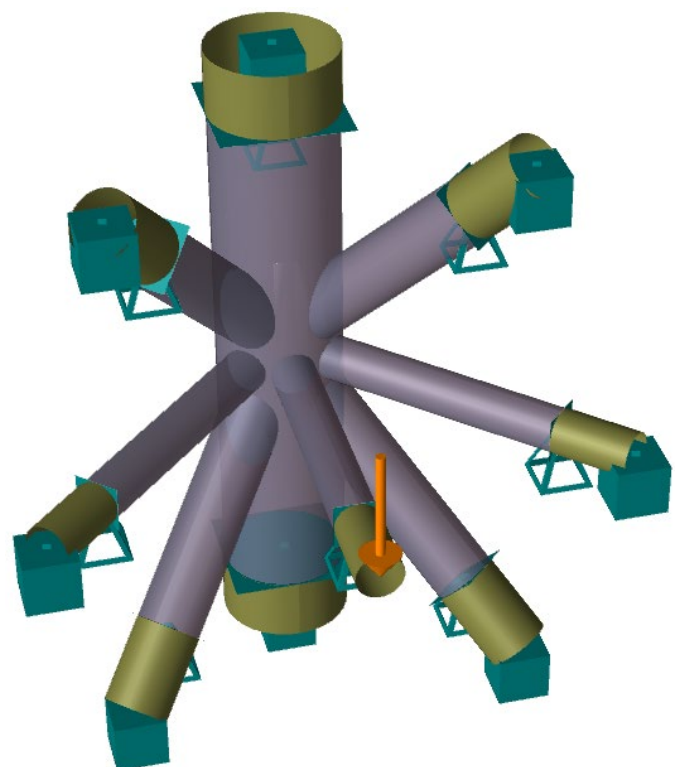
1. Introduction	Page 4
2. Open Beam Model Workspace, Create Material and Sections	Page 5
3. Model Beams	Page 7
4. Add Boundary Conditions for the Beam Model	Page 12
5. Add Loads for the Beam Model	Page 13
6. Run Structural Analysis for the Beam Model	Page 14
7. Extract Results for the Beam Model	Page 15
8. Open New Workspace for Shell Model and Import Beam Model	Page 18
9. Convert Beam Model to Shell Model	Page 19
10. Create FE Mesh for the Shell Model	Page 22
11. Analyse the Shell Model	Page 24
12. Extract Results for the Shell Model	Page 25
13. Convert Beam Model to Shell Model with Refinement Zones	Page 27
14. Create FE Mesh for the Shell Model with Refinement Zones	Page 32
15. Analyse and View Results for the Shell Model with Refinement Zones	Page 33

1 INTRODUCTION

- In this tutorial, two models of a tubular joint are created – a beam model and a 3D shell model. The latter model is created by converting the former by a highly automatic feature in GeniE. The two models are analysed and displacement and stress results are compared.
 - Comparison of stresses determines so-called stress concentration factors (SCFs).
- The following topics are covered:
 - Beam modelling
 - Load application
 - Boundary conditions
 - Automatic conversion of tubular beam model to shell model
 - Analysis and results processing of a model with rather coarse FE mesh
 - Analysis and results processing of a model with refinement and transition zones controlling the FE mesh
- Input files for the two models are provided:
 - Tubular_Joint_Beams_input.js for creating the beam model of the joint
 - Tubular_Joint_Shell_input.js for creating the shell model of the joint by converting the beam model



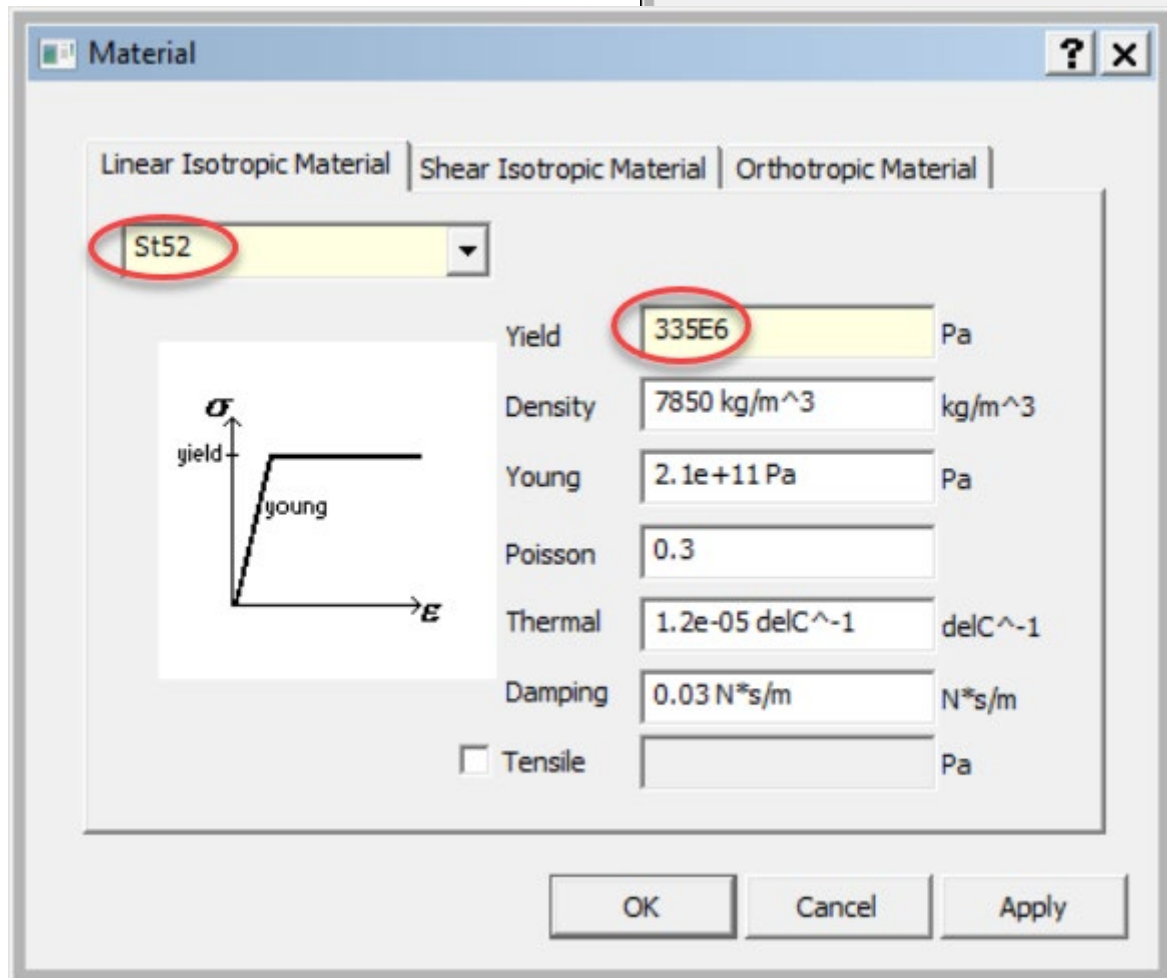
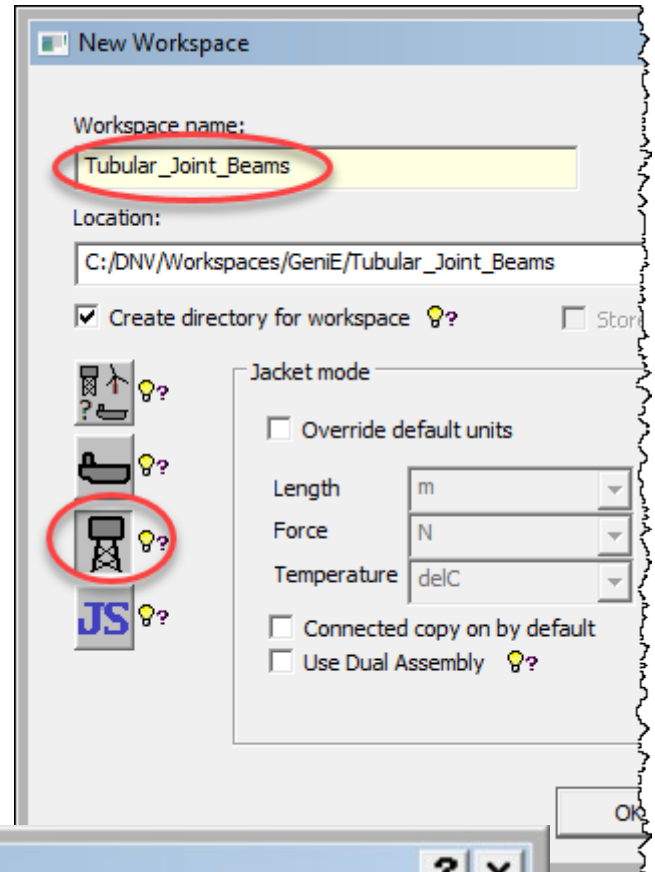
Beam model



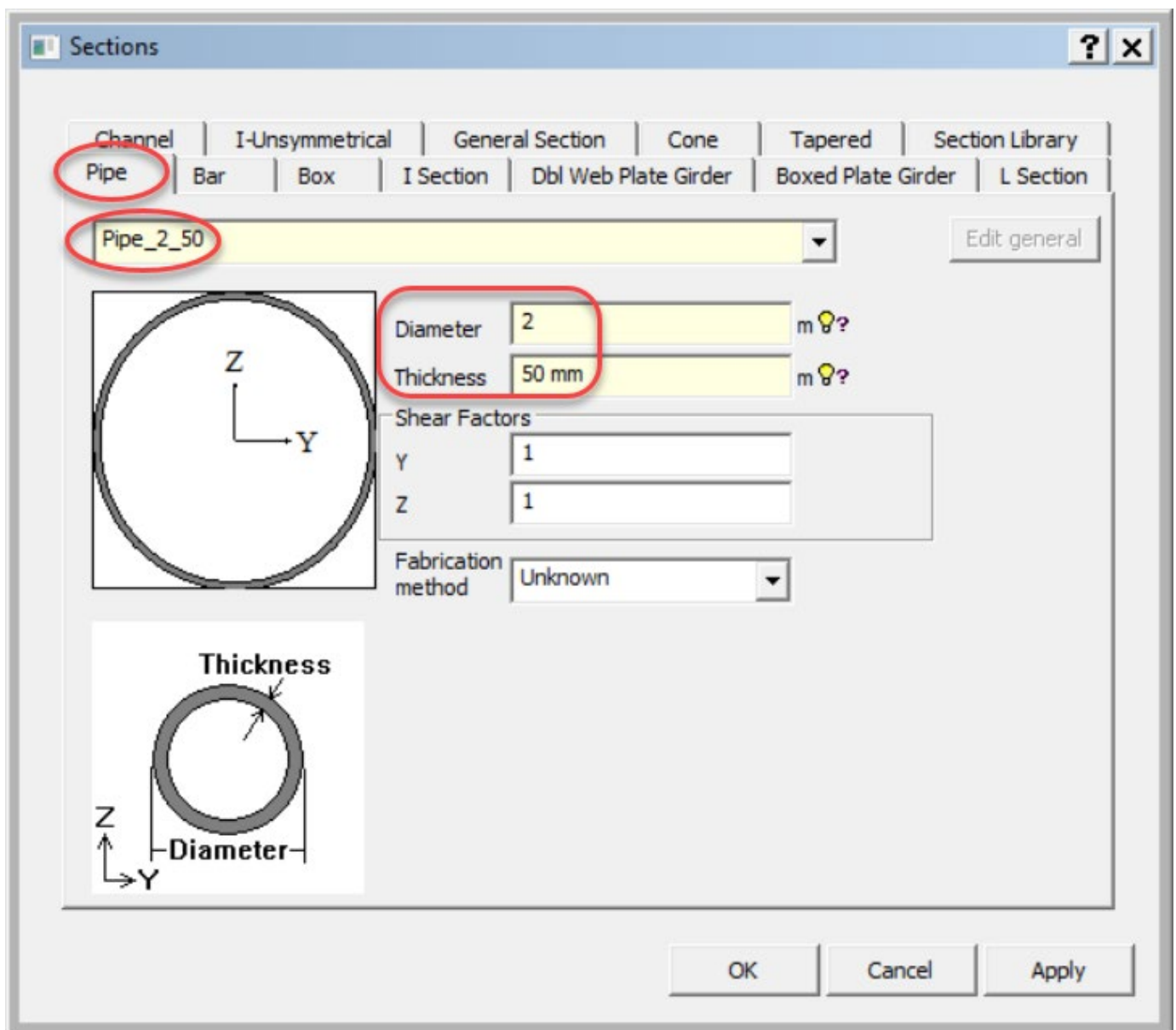
Shell model

2 OPEN BEAM MODEL WORKSPACE, CREATE MATERIAL AND SECTIONS

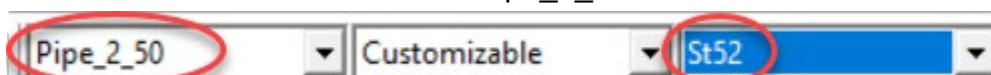
- Start GeniE and open a new workspace.
 - Give a workspace name, for example Tubular_Joint_Beams.
 - Press the *Jacket mode* button to limit menus and buttons to those relevant for beam modelling.
 - Accept default units m and N and click OK.
- Create a steel material.
 - Use *Edit | Properties* to open the *Properties* dialog.
 - In the *Material* tab click *Create/Edit Material*.
 - In the *Material* dialog give a material name and the *Yield* value 335E6. Accept default values and click OK.



- Create beam cross sections.
 - Use *Edit | Properties* to open the *Properties* dialog.
 - In the *Section* tab click *Create/Edit Section*.
 - In the *Pipe* tab of the *Sections* dialog create the following pipe sections:
 - Pipe_2_50: *Diameter* = 2 m and *Thickness* = 50 mm
 - Pipe_1_16: *Diameter* = 1 m and *Thickness* = 16 mm
 - Pipe_07_15: *Diameter* = 0.7 m and *Thickness* = 15 mm

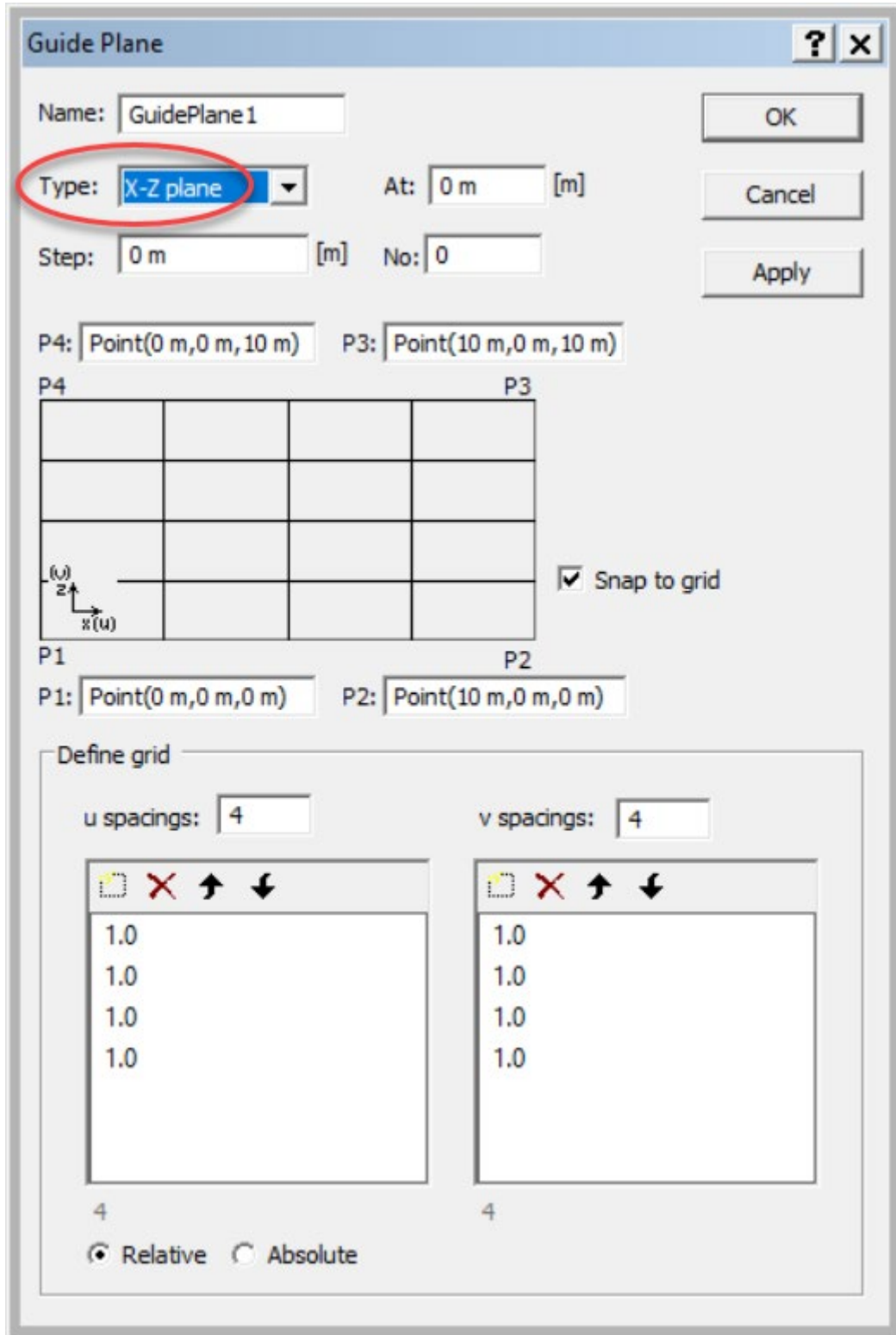


- Set material St52 and beam cross section Pipe_2_50 as default:



3 MODEL BEAMS

- Use *Guiding Geometry | Planes | Guide Plane Dialog* to create a guide plane as shown below, i.e. set *Type* to *X-Z plane*, otherwise accept all default values.
 - Press function key F7 to view the guide plane in positive Y-direction.



Guide Plane

Name:

Type: X-Z plane At: [m]

Step: [m] No:

P4: P3:

P4 P3

☒ Snap to grid

P1 P2

P1: P2:

Define grid

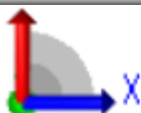
u spacings: v spacings:



1.0 1.0 1.0 1.0

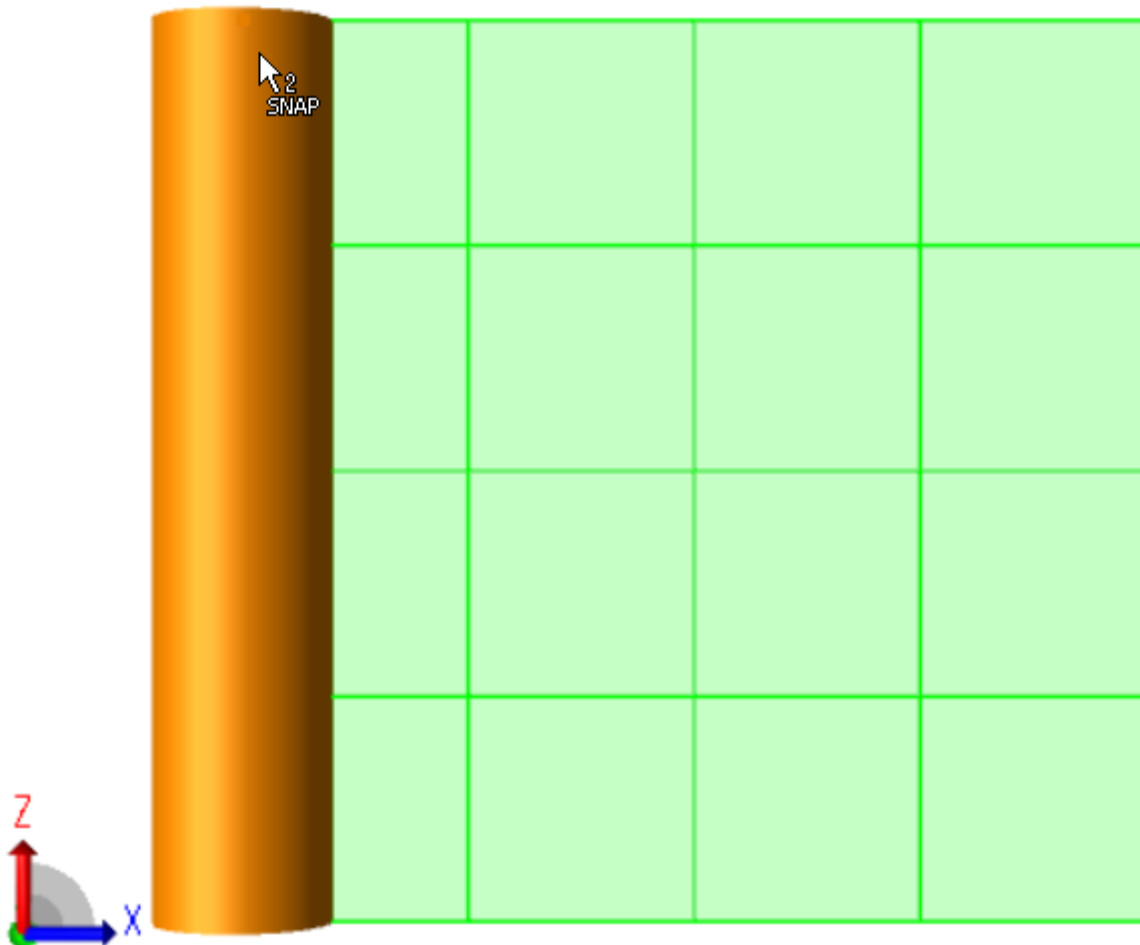
1.0 1.0 1.0 1.0

4 4

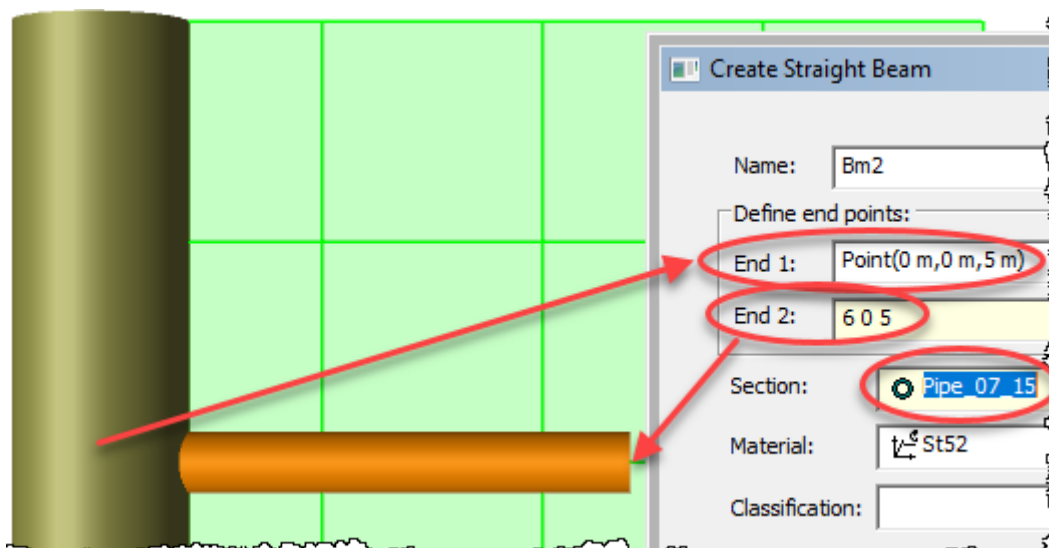
☒ Relative ☐ Absolute



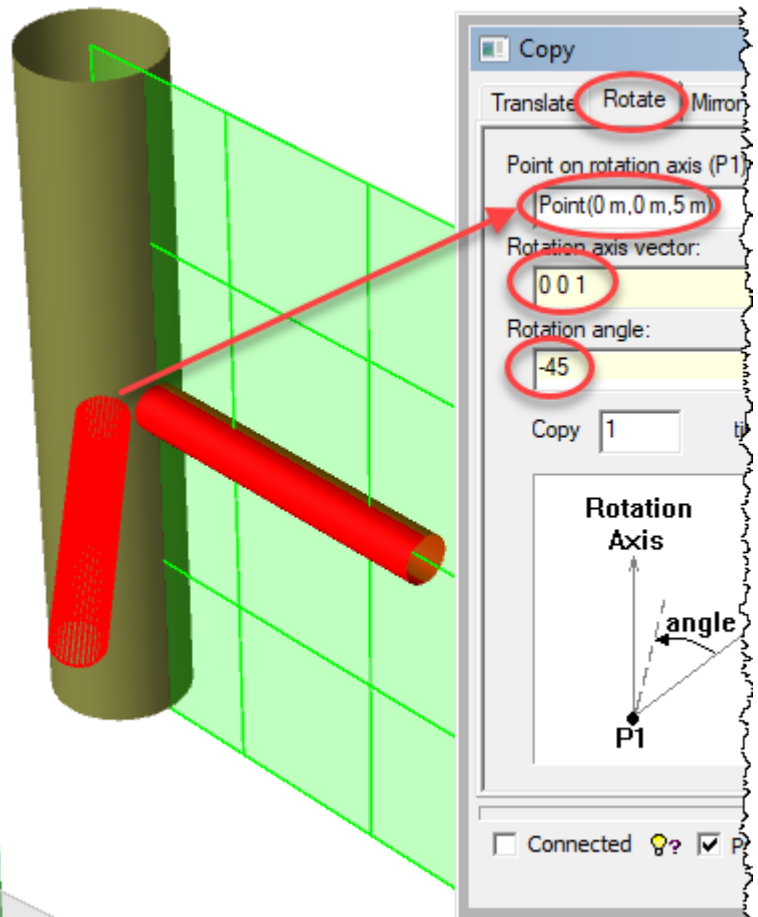
- Create the vertical chord member shown below with beam cross section Pipe_2_50 by *Structure | Beams and Piles | Straight Beam* (or press  ).



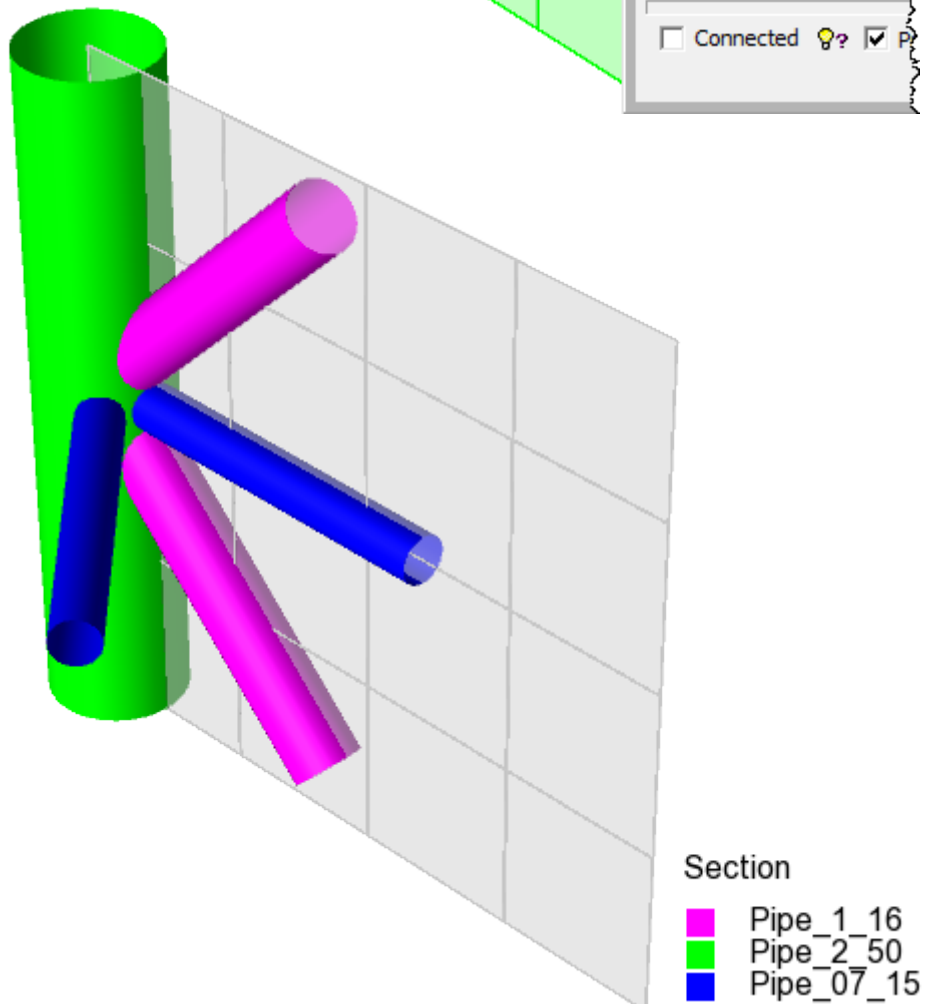
- Create a horizontal brace with beam cross section Pipe_07_15 by *Structure | Beams and Piles | Straight Beam Dialog*. Click to enter coordinates for *End 1* and fill in coordinates for *End 2* as shown.



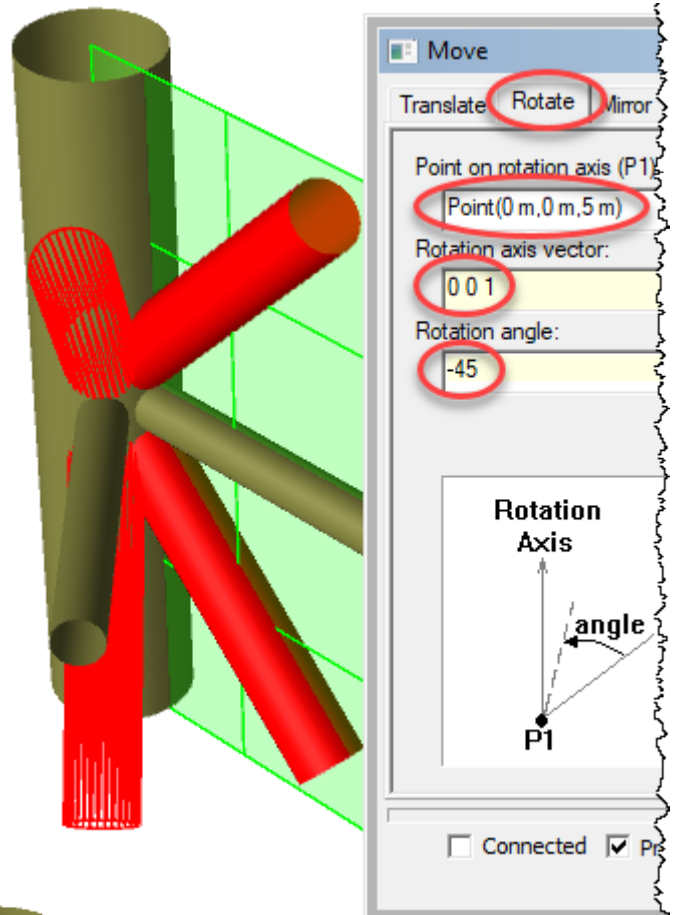
- Press function key F5 to display the model in an isometric view.
- Create another horizontal brace by rotational copying the existing one as shown to the right.



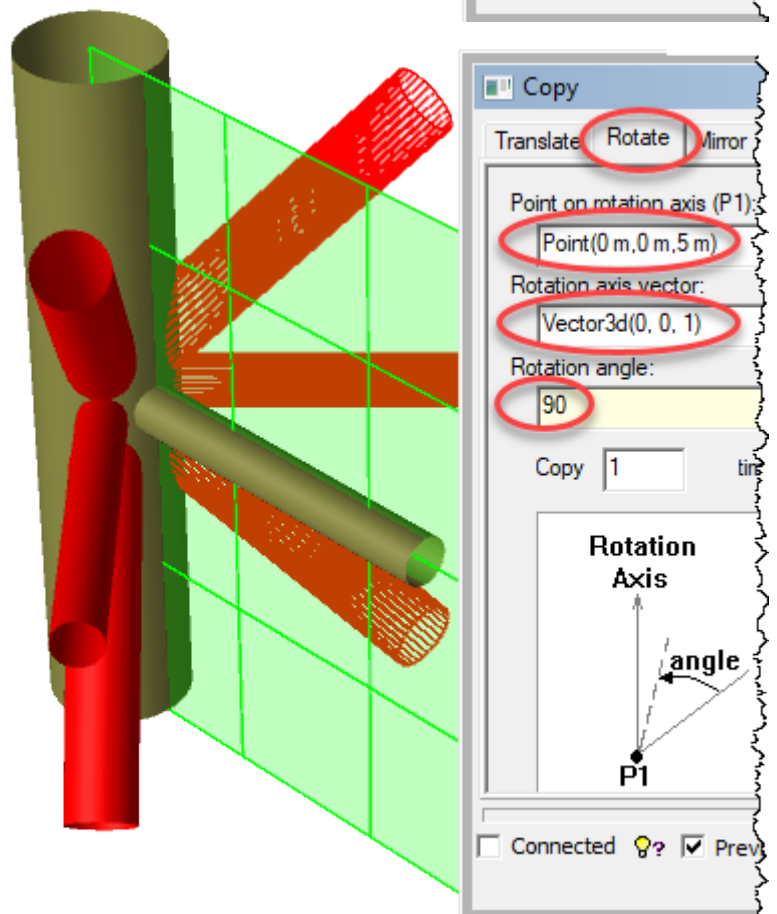
- Create two more braces by rotational copying the original brace 45 degrees up and down about an axis in Y-direction (0, 1, 0).
 - Modify their beam cross sections to Pipe_1_16.




- Move the two inclined braces by a 45 degree rotation about the vertical axis as shown to the right.

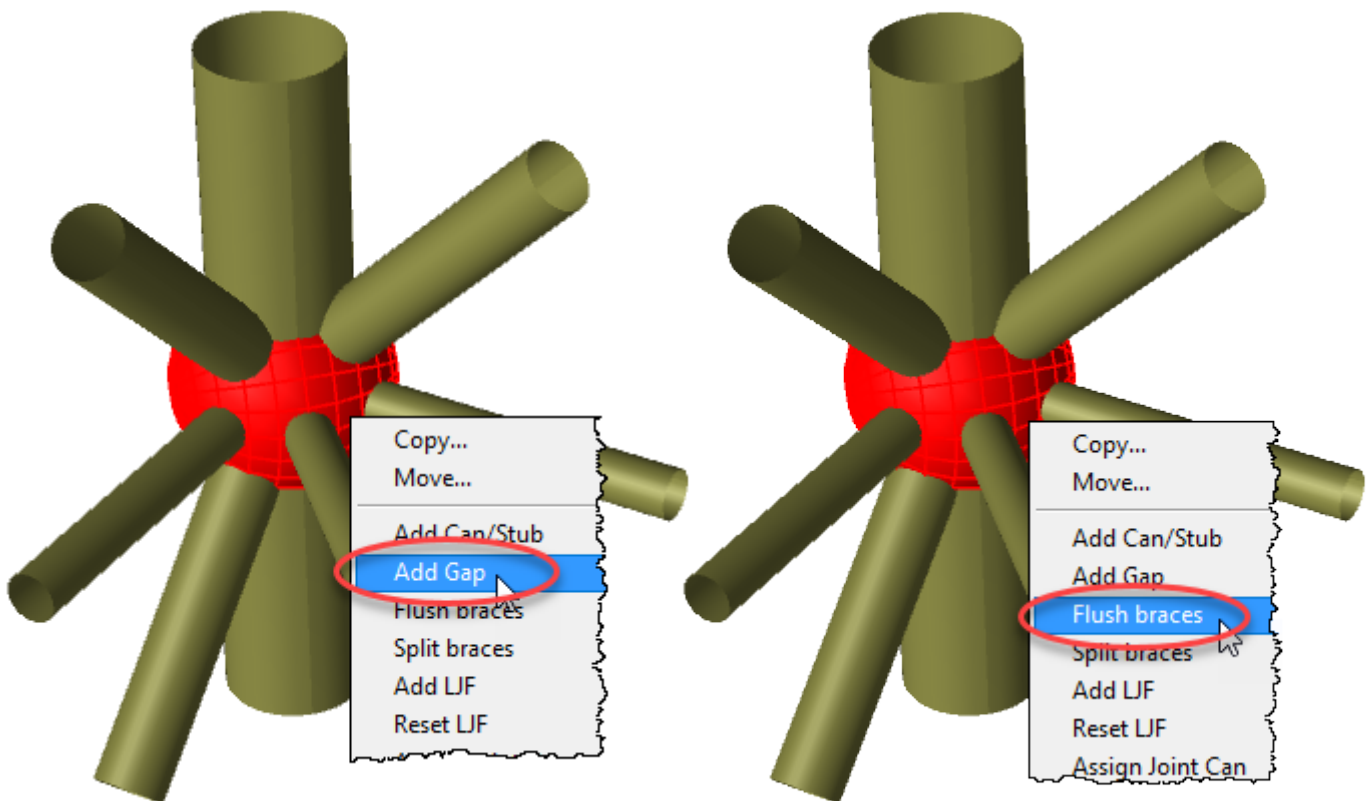
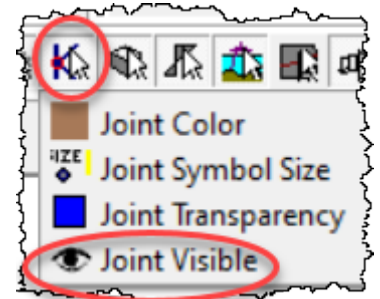


- Copy the three braces by a 90 degree rotation about the vertical axis as shown to the right.

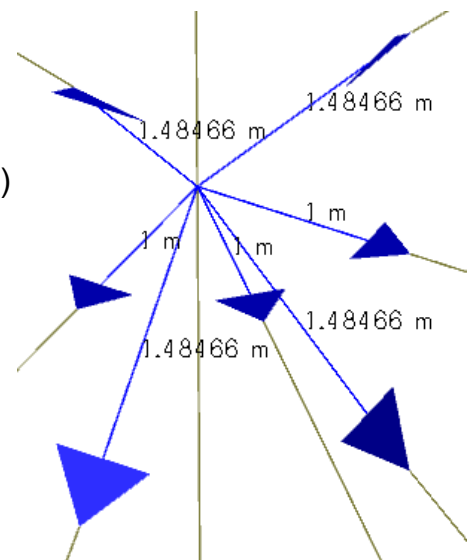


➤ Introduce gaps between brace-chord intersections and flush the braces to the chord wall, i.e. introduce eccentricities (offsets) so that their flexible parts end at the chord wall.


- To do so, create a joint at the chord brace connection by pressing the *Joint* button  and clicking the connection point.
- To see the joint, right-click the *Joint selection* button and open the eye symbol.
- Select the joint (the *Joint selection* button must be depressed), right-click and select *Add Gap*.
- Thereafter, right-click the joint and select *Flush braces*.

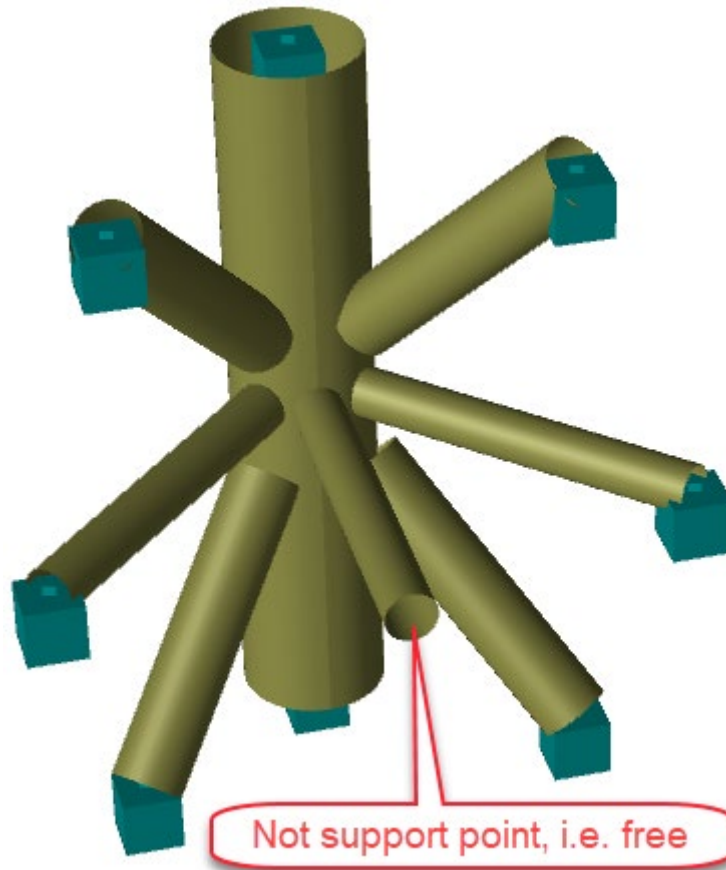


- Verify the eccentricities by switching to wireframe view, selecting all beams, right-clicking and selecting *Labels | Eccentricities*. A zoomed view is shown to the right. (The joint symbol has been hidden by selecting and Alt+Minus.)
- Remove labels by reselecting the beams, right-clicking and selecting *Clear Labels*.



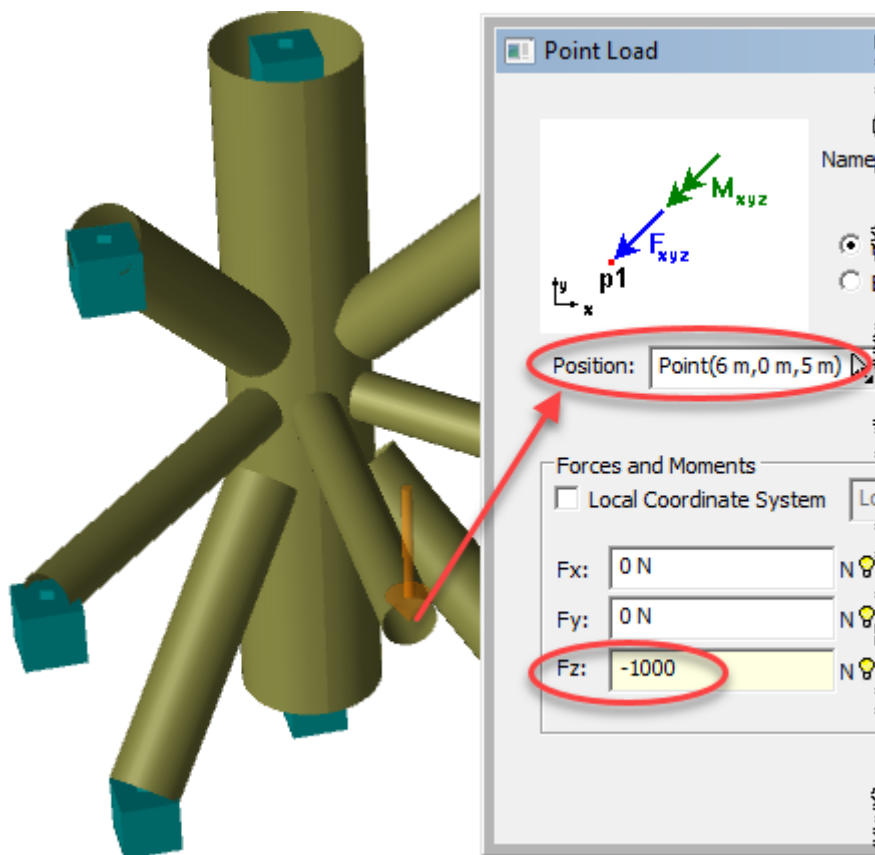
4 ADD BOUNDARY CONDITIONS FOR THE BEAM MODEL

- Add boundary conditions by *Structure* | *Support* | *Support Point* (or press ) at all free beam ends except at the horizontal brace in X-direction.
 - Let all six degrees of freedom be fixed, i.e. accept the default condition for all support points.



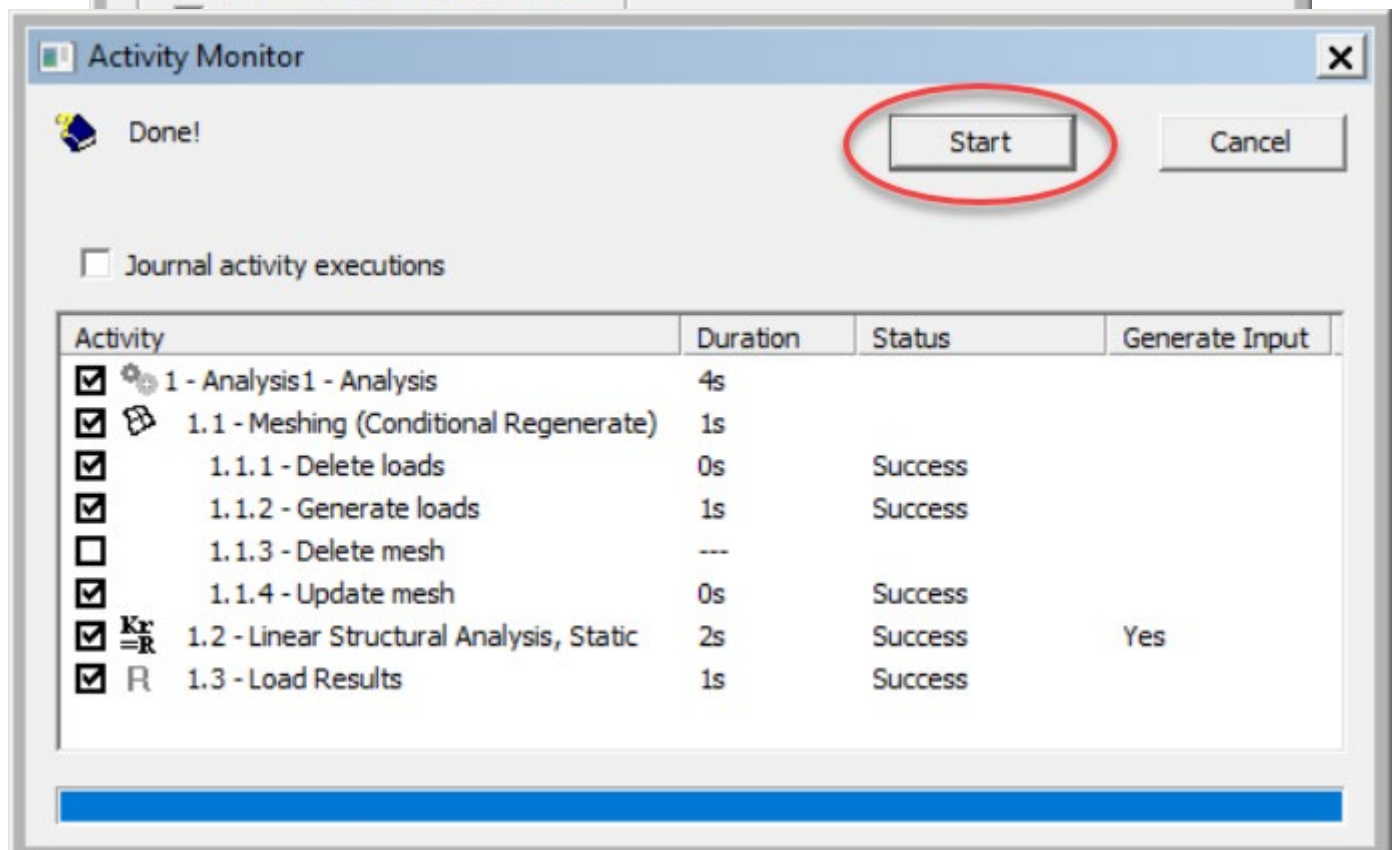
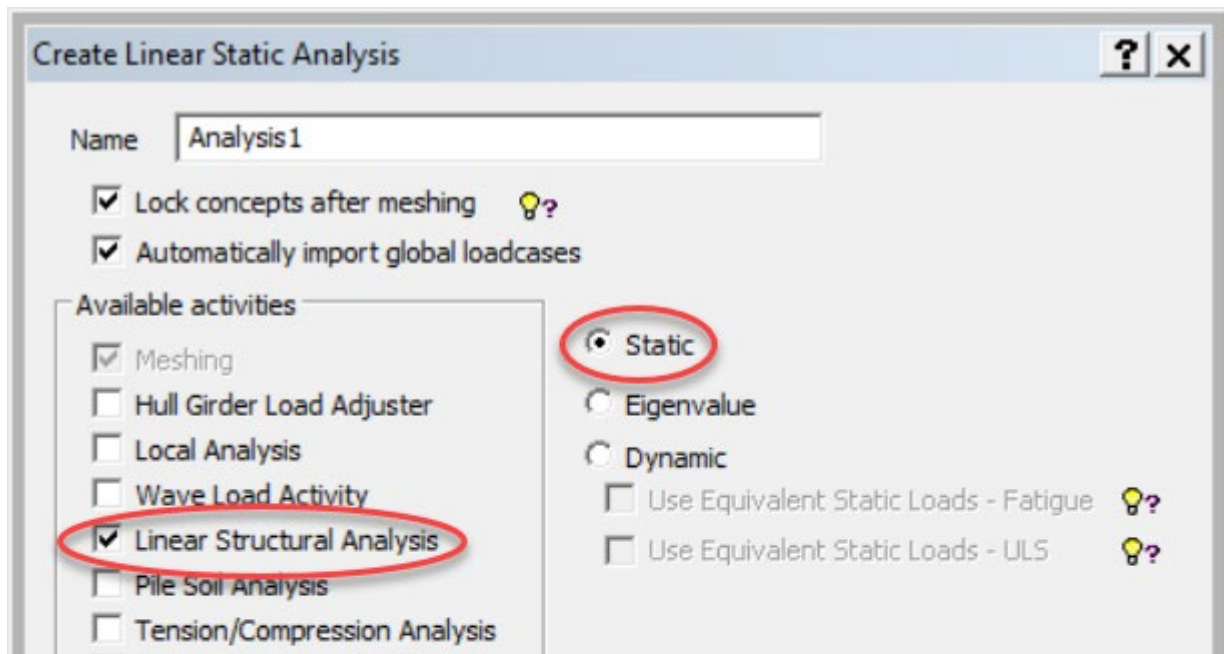
5 ADD LOADS FOR THE BEAM MODEL

- The tubular joint is subjected to three load cases:
 - LC_Vert – vertical point load of –1000 N at the free end of the horizontal brace in X-direction
 - LC_Axial – axial point load of 1000 N at the free end of the horizontal brace in X-direction
 - LC_Hori – horizontal point load of 1000 N in Y-direction at the free end of the horizontal brace in X-direction
- Use *Loads | Load Case* to create a load case.
 - Point loads are defined by *Loads | Explicit Load | Point Load* as shown below for LC_Vert.
 - Note that the proper load case must be set as current prior to defining the point load.



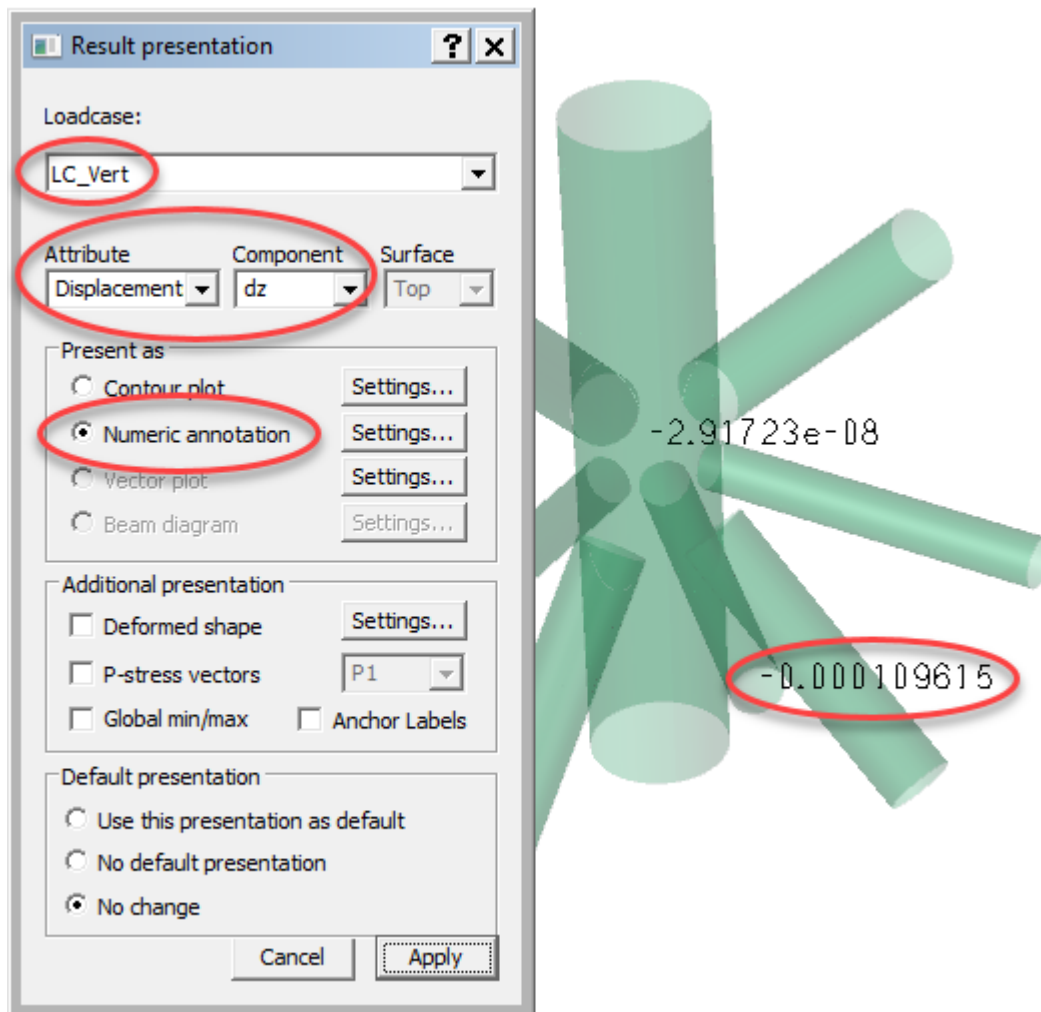
6 RUN STRUCTURAL ANALYSIS FOR THE BEAM MODEL

- Use *Mesh & Analysis | Activity Monitor* (or Alt+D) to open the *Create Linear Static Analysis* dialog. *Static* and *Linear Structural Analysis* are selected by default. Click *OK* and thereafter *Start* in the *Activity Monitor*.
- Verify the analysis by right-clicking the *Linear Structural Analysis, Static* activity in the *Activity Monitor* to open the *sestra.lis* file. In this file make sure the loads and reaction forces are correct.

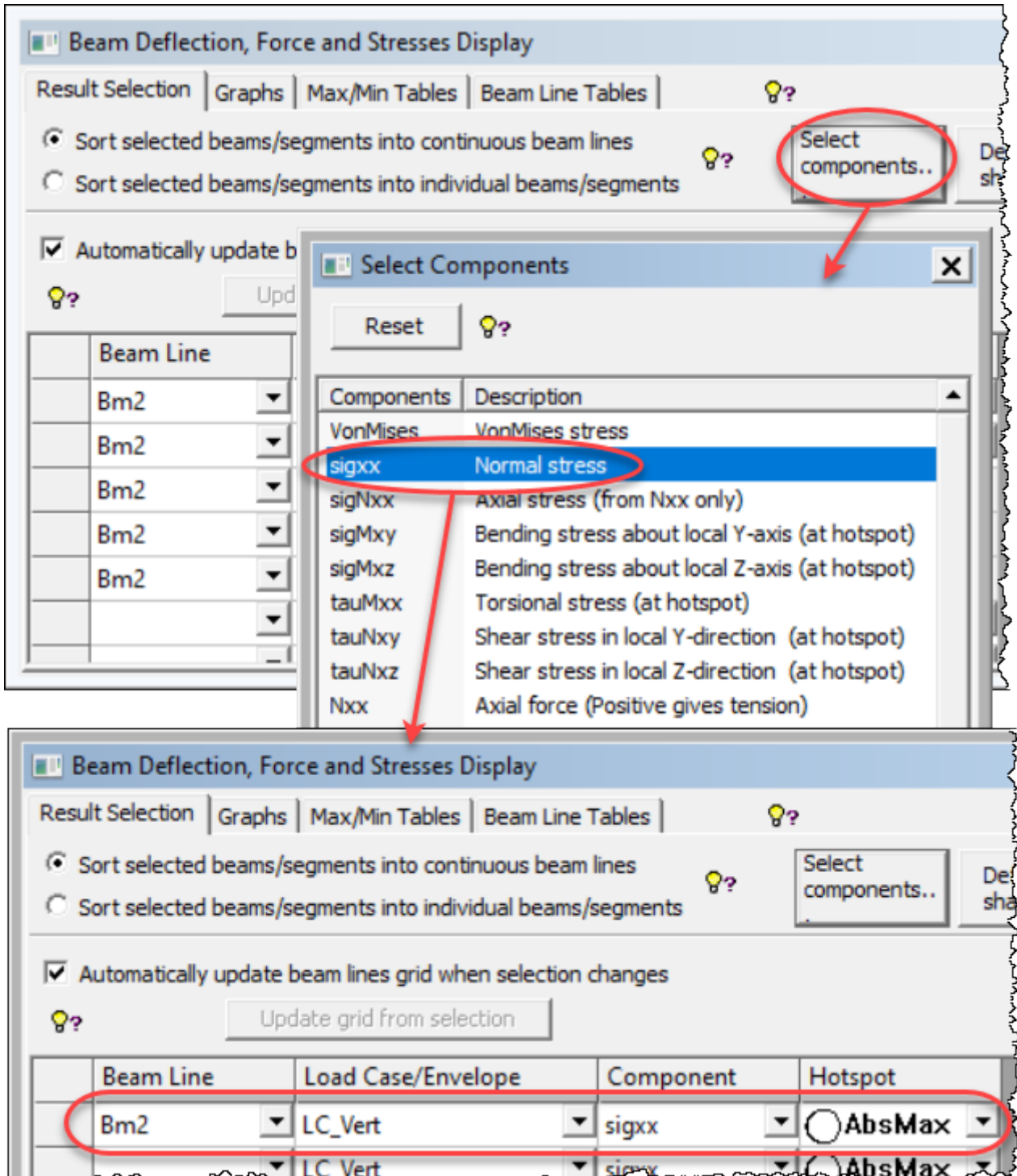


7 EXTRACT RESULTS FOR THE BEAM MODEL

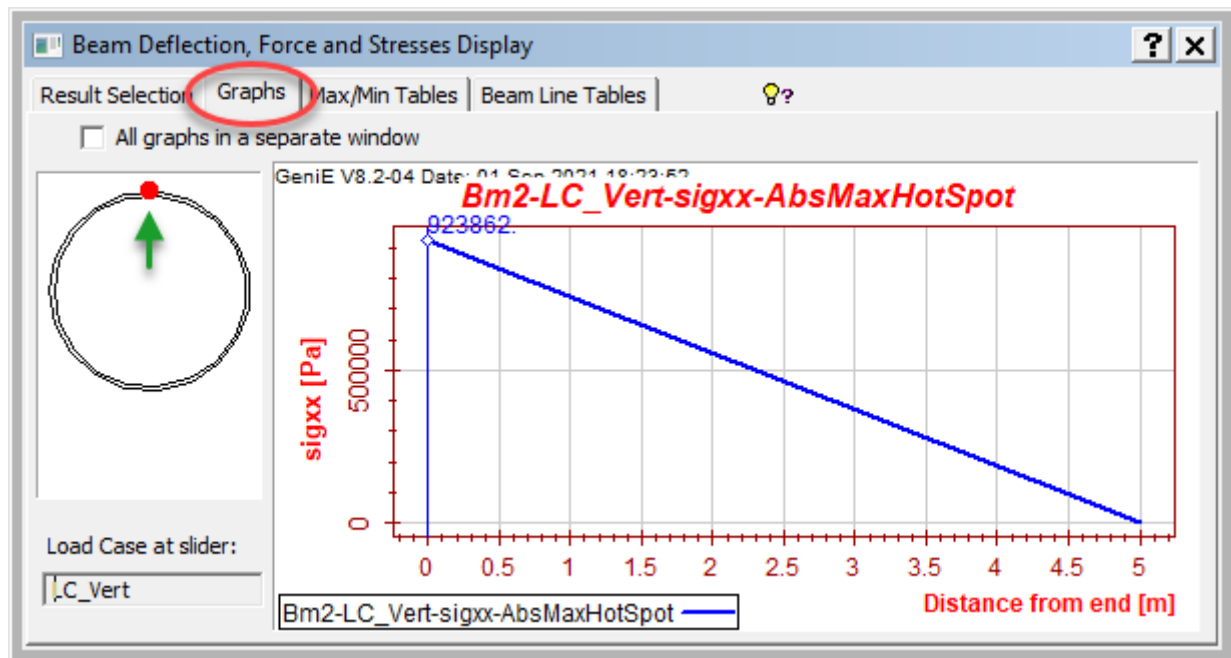
- In this tutorial comparison of results for the beam model with results for the subsequent 3D shell model is limited to the following result components for load case LC_Vert:
 - Vertical displacement at the free beam end of the horizontal brace in X-direction where the point load is applied
 - Maximum axial stress for the same brace at the end connected to the chord
- Find the vertical displacement by switching to *Results - with Mesh* display configuration. Use Alt+P to open the *Result presentation* dialog. In the dialog select *Attribute Displacement* and *Component dz* and then *Present as Numeric annotation*.
 - See that the vertical displacement is -0.000109615 .
 - Note that the only nodes with displacements are the two nodes of the horizontal brace in X-direction since all other nodes are fixed.



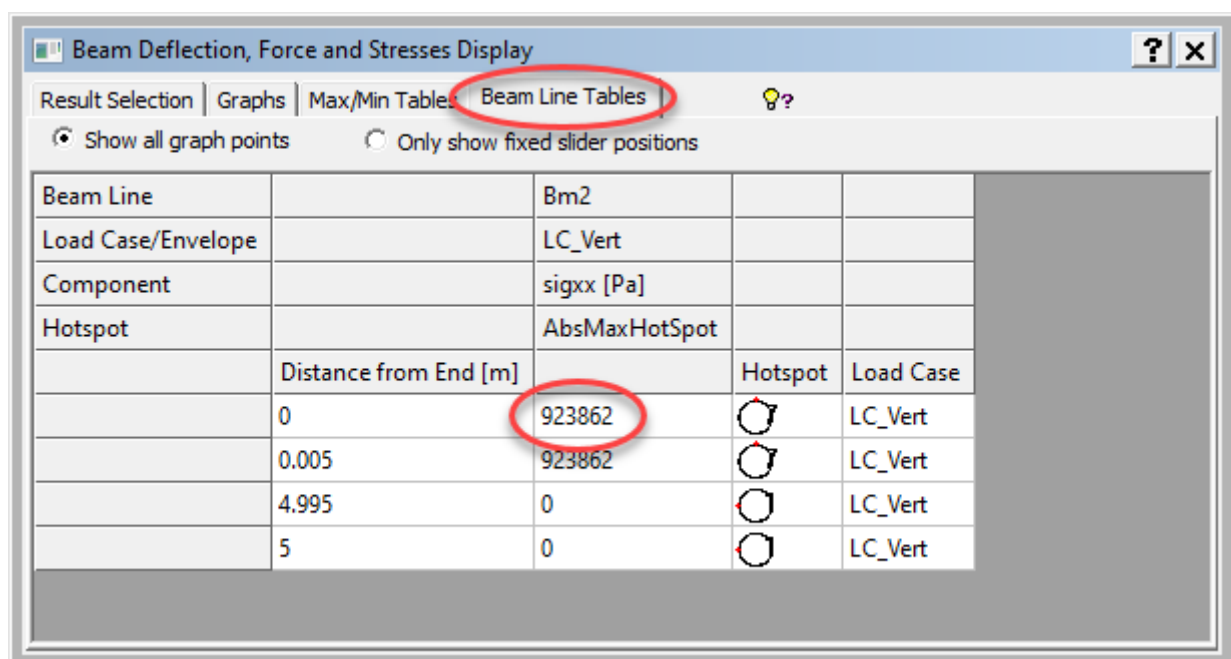
- Find the maximum axial stress at the brace end connected to the chord as follows.
 - Select the brace and use *Results | Beam Result Diagrams* to open the *Beam Deflection, Force and Stresses Display* dialog. In the dialog click the *Select components* button and in the *Select Components* dialog select *sigxx*.
 - There may be several lines in the *Beam Deflection, Force and Stresses Display* dialog but only in the first line, a beam (Bm2) is given, the other lines are therefore irrelevant.







- Go to the *Graphs* tab to see a graph of the stress component along the beam. Right-click the graph and select *Add slider at max* to see a vertical line and value 923862 at maximum.
 - Notice the red dot (pointed at by added green arrow) marking where in the pipe section the maximum stress component occurs, i.e. at top of the section. The dot relates initially to position 0 m along the beam. When hovering the mouse over the graph and moving it horizontally the red dot may shift to another point in the pipe section.



- Go to the *Beam Line Tables* tab to see the stress values along the beam. As seen, the axial stress at position 0 m (where the brace is connected to the chord) is 923862 Pa.

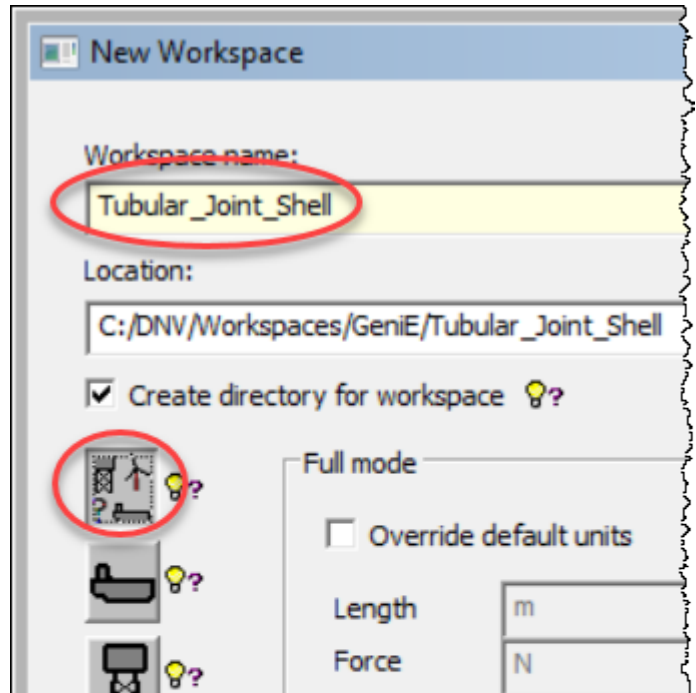


Beam Line	Distance from End [m]	sigxx [Pa]	AbsMaxHotSpot	Hotspot	Load Case
Bm2	0	923862			LC_Vert
	0.005	923862			LC_Vert
	4.995	0			LC_Vert
	5	0			LC_Vert

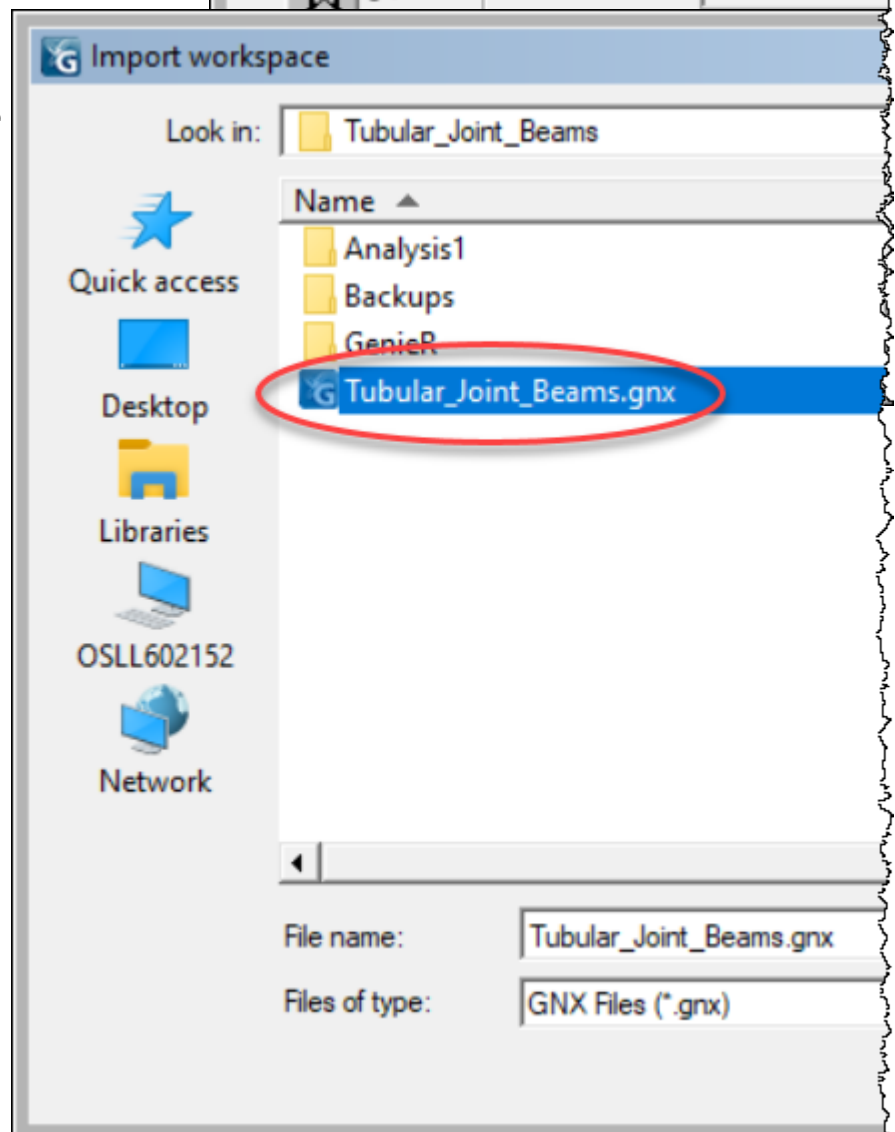
8 OPEN NEW WORKSPACE FOR SHELL MODEL AND IMPORT BEAM MODEL

➤ Rather than converting the beam model in the current workspace open a new workspace, import the beam model and then do the conversion.

➤ Save the current workspace and without closing it click *File | New Workspace* (or Ctrl+N). Give a *Workspace name* and press the *Full mode* button to access curved shell modelling features.



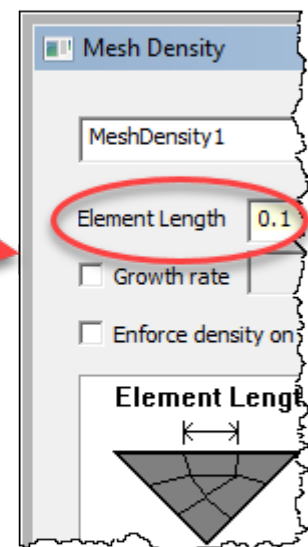
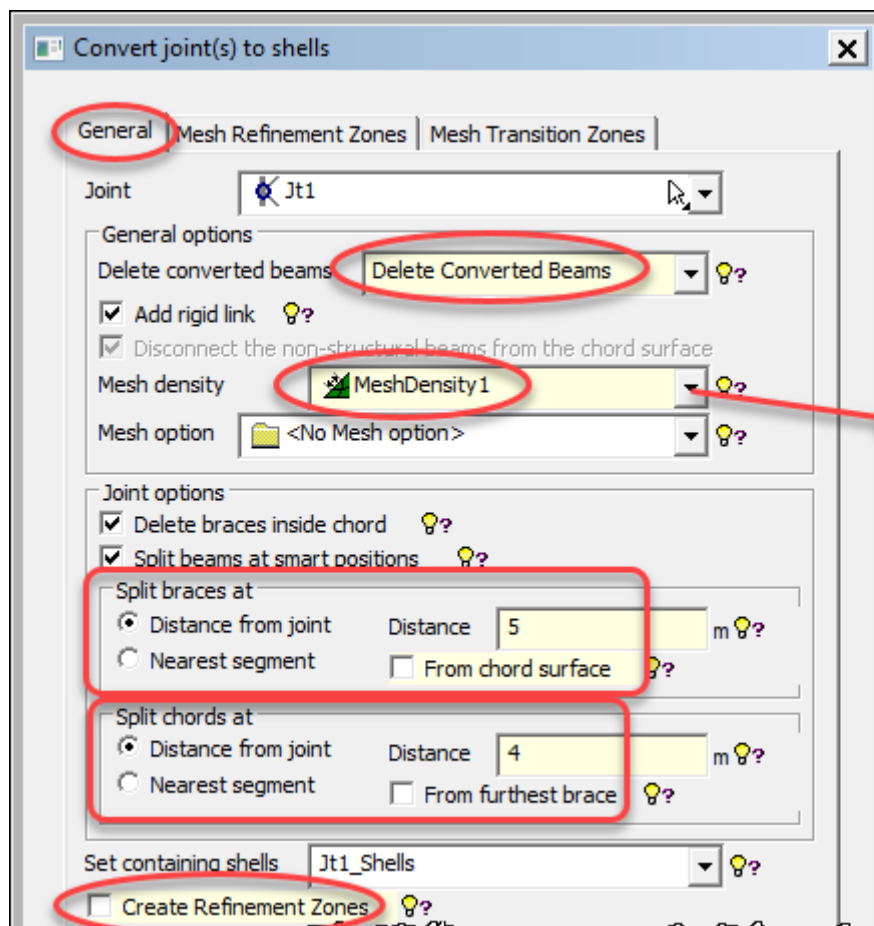
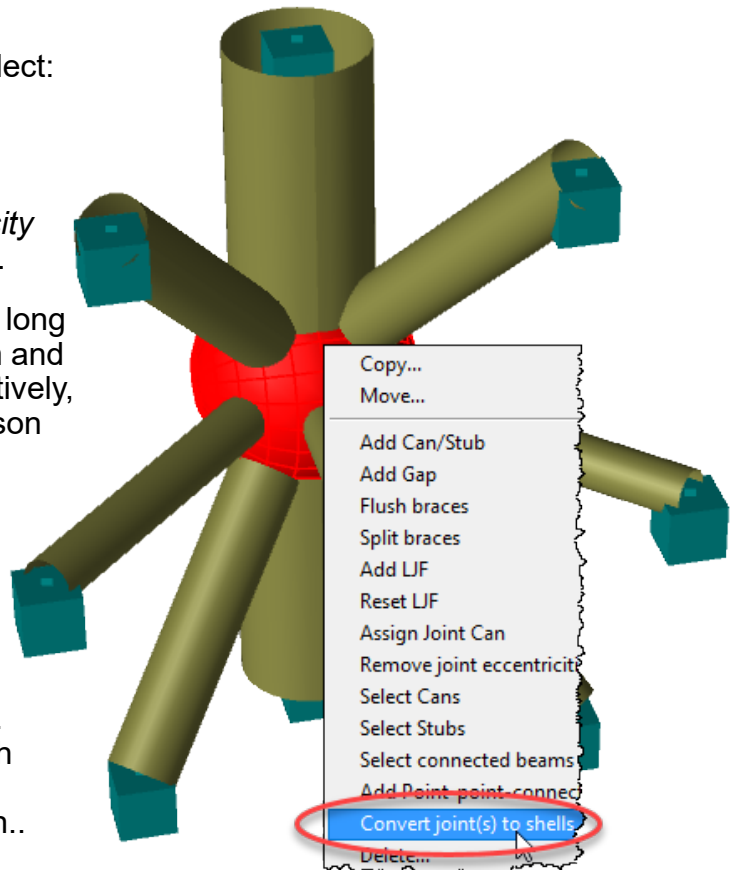
➤ Use *File | Import | Workspace (GNX file)* and browse to find the workspace of the beam model.



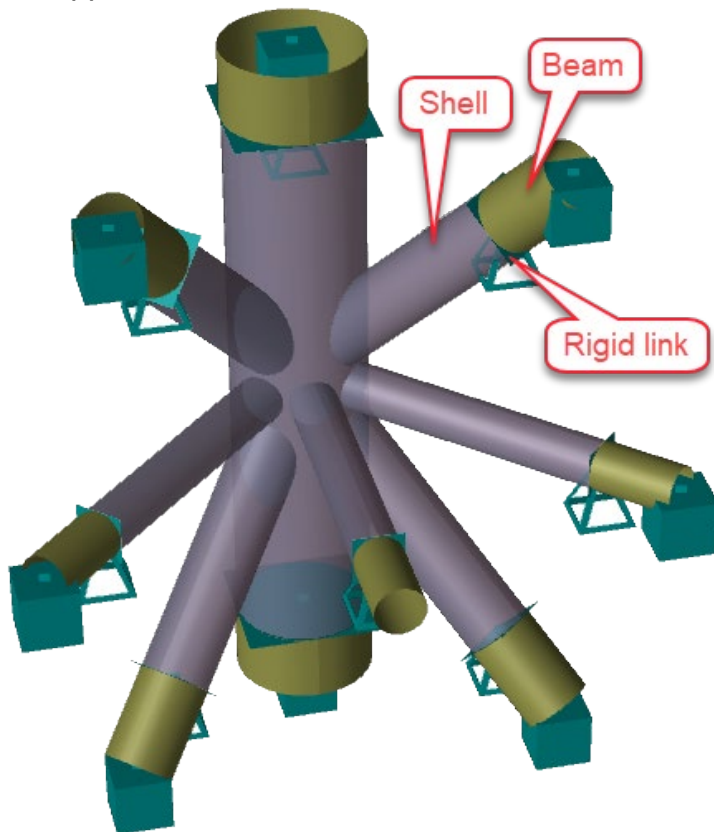
9 CONVERT BEAM MODEL TO SHELL MODEL

➤ Select the joint, right-click and select *Convert joint(s) to shells*. In the dialog select:

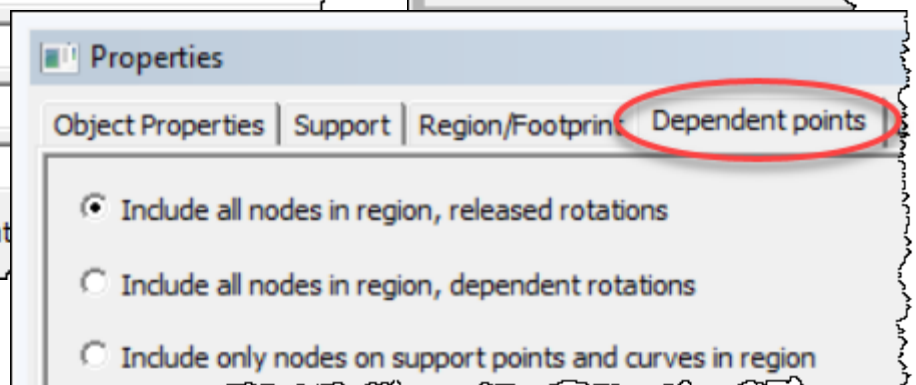
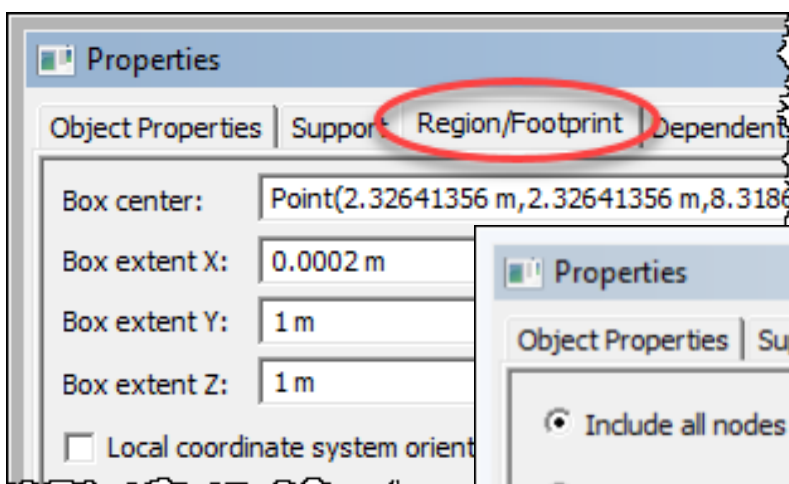
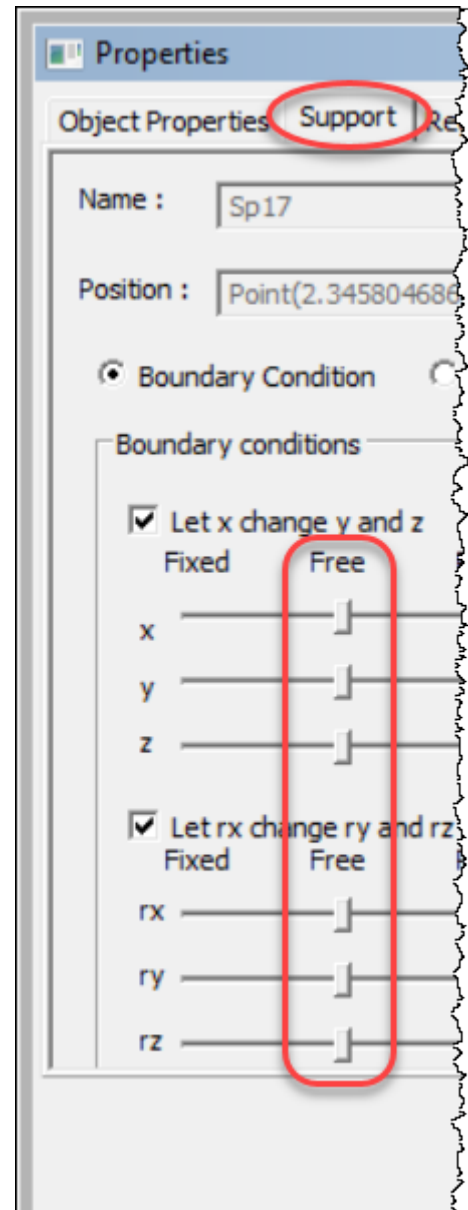
- *Delete Converted Beams* (the beams inside the shells)
- Use the pulldown menu for *Mesh density* to create a new mesh density of 0.1 m.
- All braces are 6 m and the chords 5 m long from the joint centre point. Specify 5 m and 4 m for the braces and chords, respectively, leaving 1 m long beam stubs. The reason for this is explained later.
 - Note that the *From chord surface* and *From furthest brace* must be unchecked.
- The *Create Refinement Zones* option is for controlling the mesh next to the welds at the brace-chord intersections. If you want detailed control of the mesh in the weld zones, go to section 13 on page 27, otherwise uncheck this option..




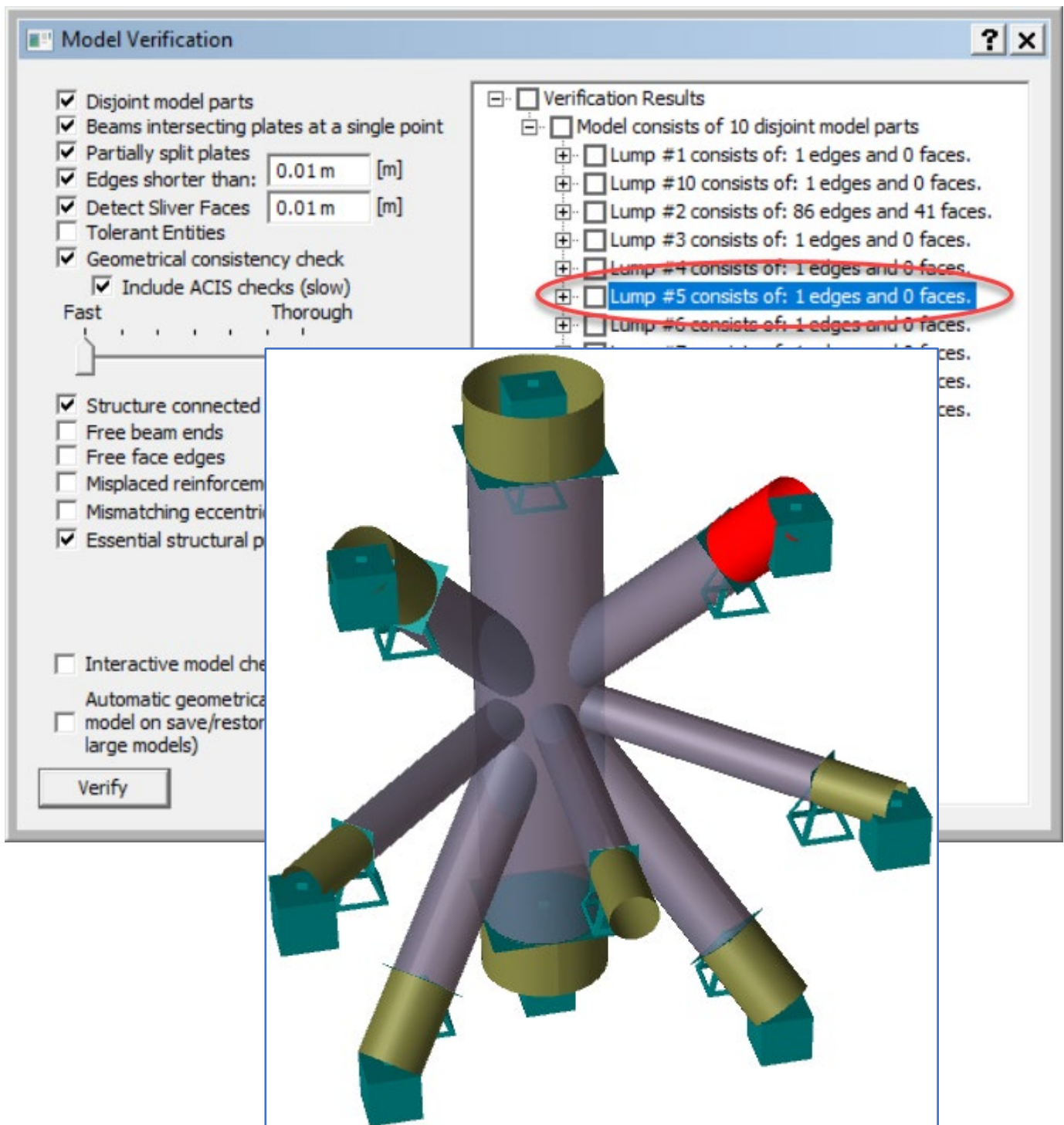
- The appearance of the model should now be:



- The rigid links (look up the term 'support rigid link' in the user documentation) couple the circles of nodes at the shell cylinder ends linearly to the beam nodes of the beam stubs. Thus forces and moments are properly transferred between the shell and beam parts.
- Open the *Properties* dialog for a rigid link to inspect the automatically created shell to beam couplings.

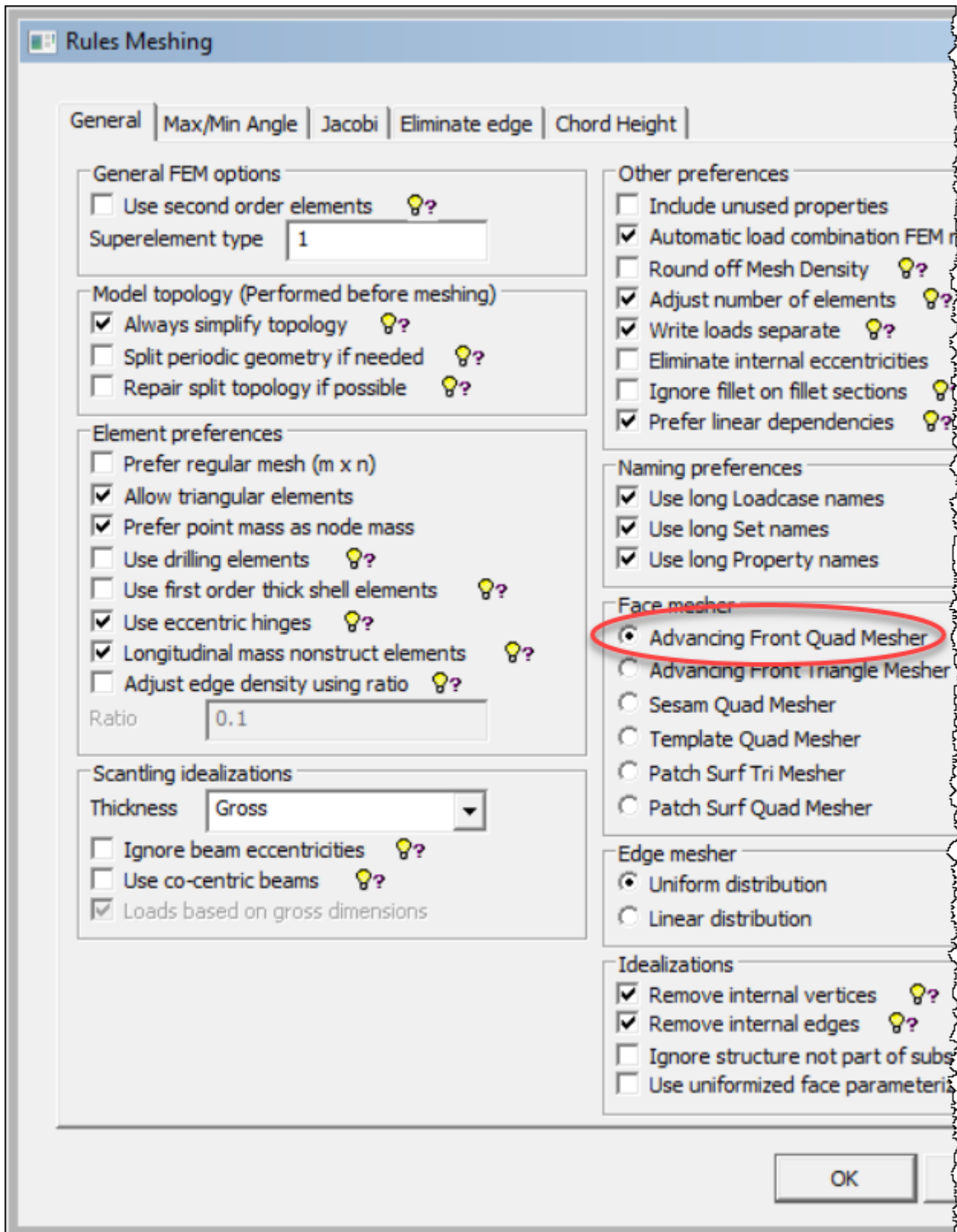


- Do a *Structure | Topology | Simplify Topology* to clean up the model.
- Verify the consistency of the model by *Structure | Topology | Verify Model* (or ) and see that there are ten disjoint parts. The reason for this is that the shell part is not geometrically coupled to the beam stubs, rather they are connected by linear coupling between nodes. This is therefore OK.
 - Select a 'lump' and see that one of the lumps is the shell part and the other nine lumps are the beam stubs.



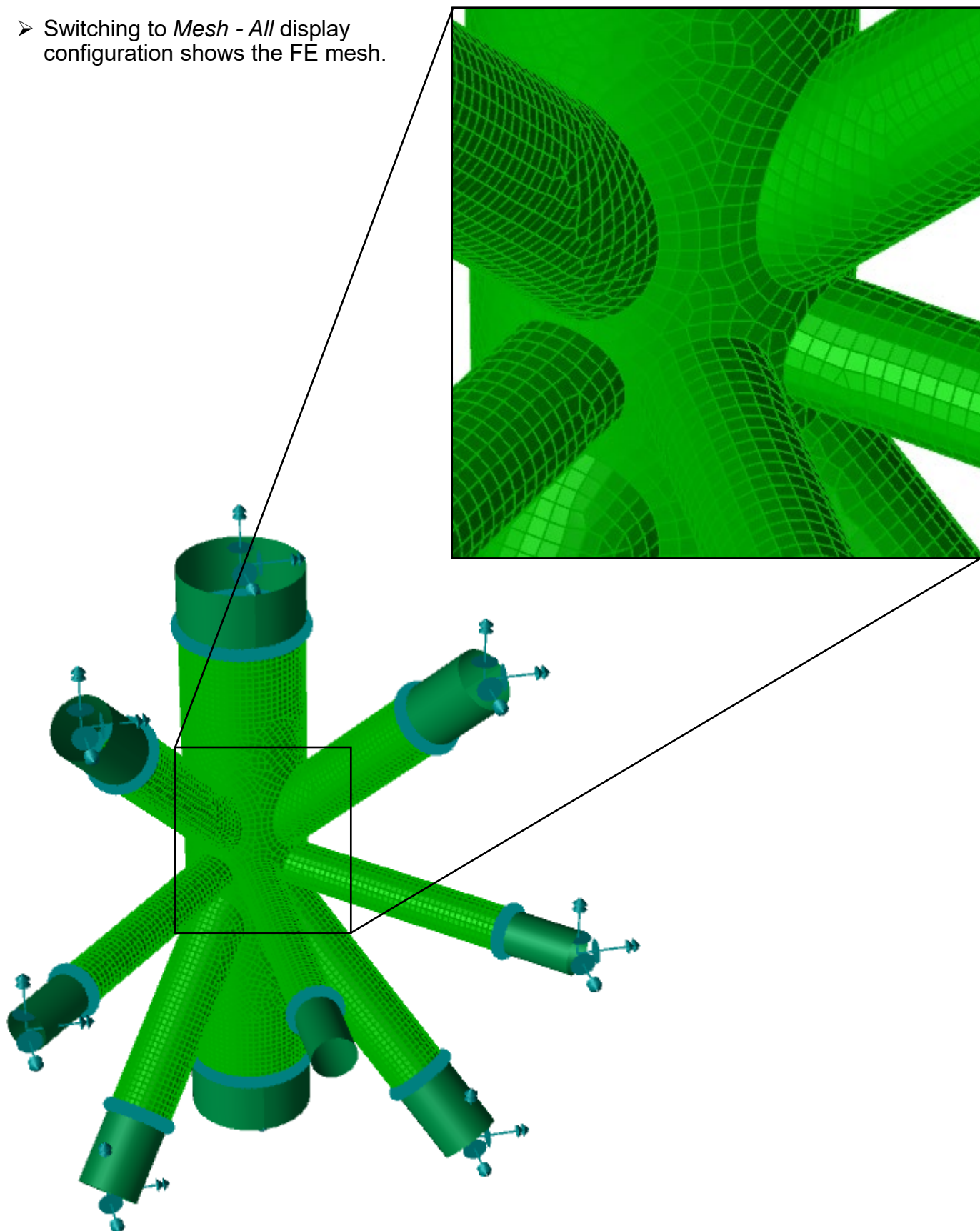
10 CREATE FE MESH FOR THE SHELL MODEL

- The shell model is now ready for analysis but prior to this, select a meshing algorithm more suitable for tubular joints. Use *Edit | Rules | Meshing Rules* and select *Advancing Front Quad Mesher*.



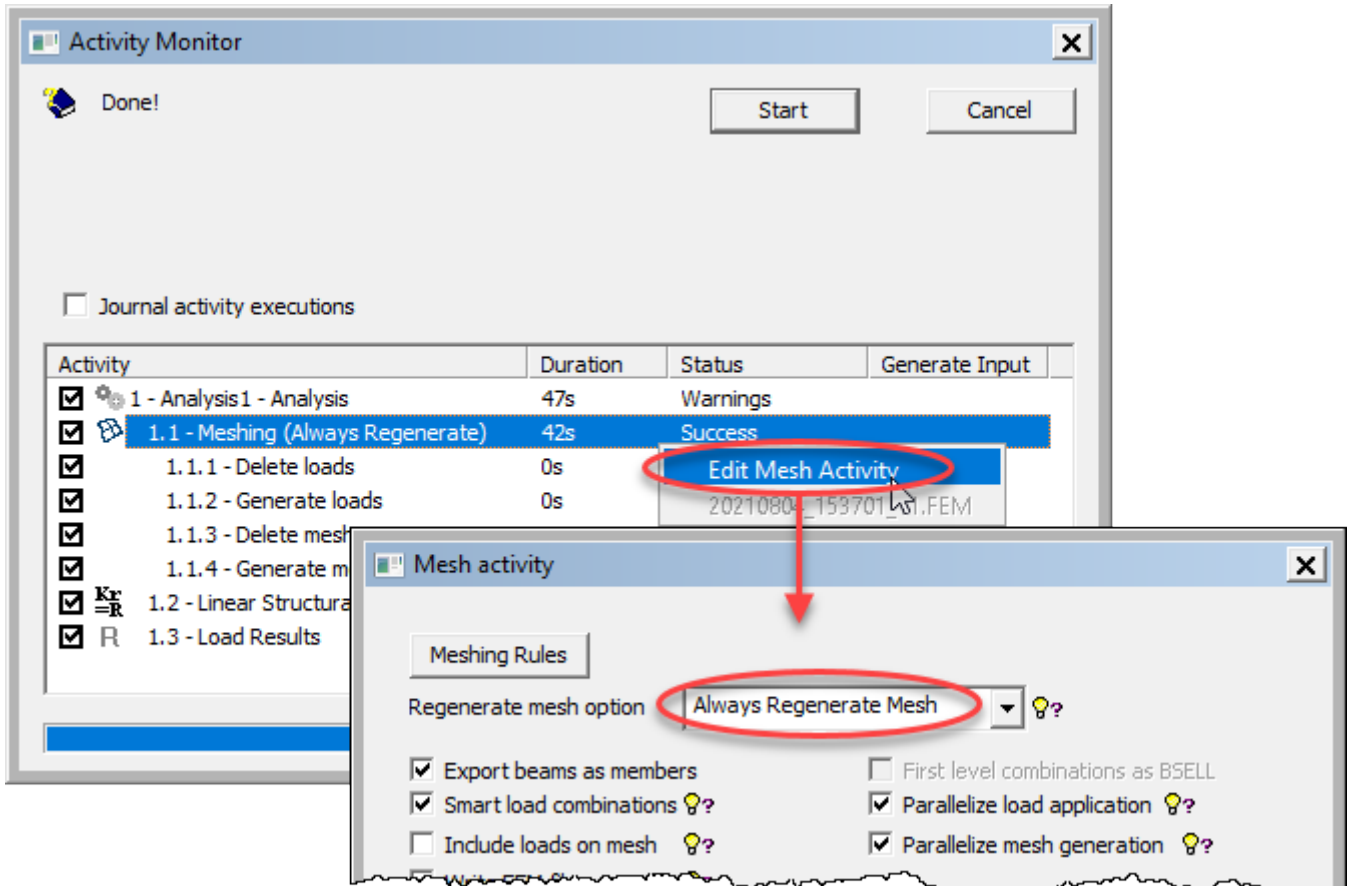
➤ Use *Mesh & Analysis* | *Create Mesh* (or Alt+M) to create the FE mesh.

➤ Switching to *Mesh - All* display configuration shows the FE mesh.



11 ANALYSE THE SHELL MODEL

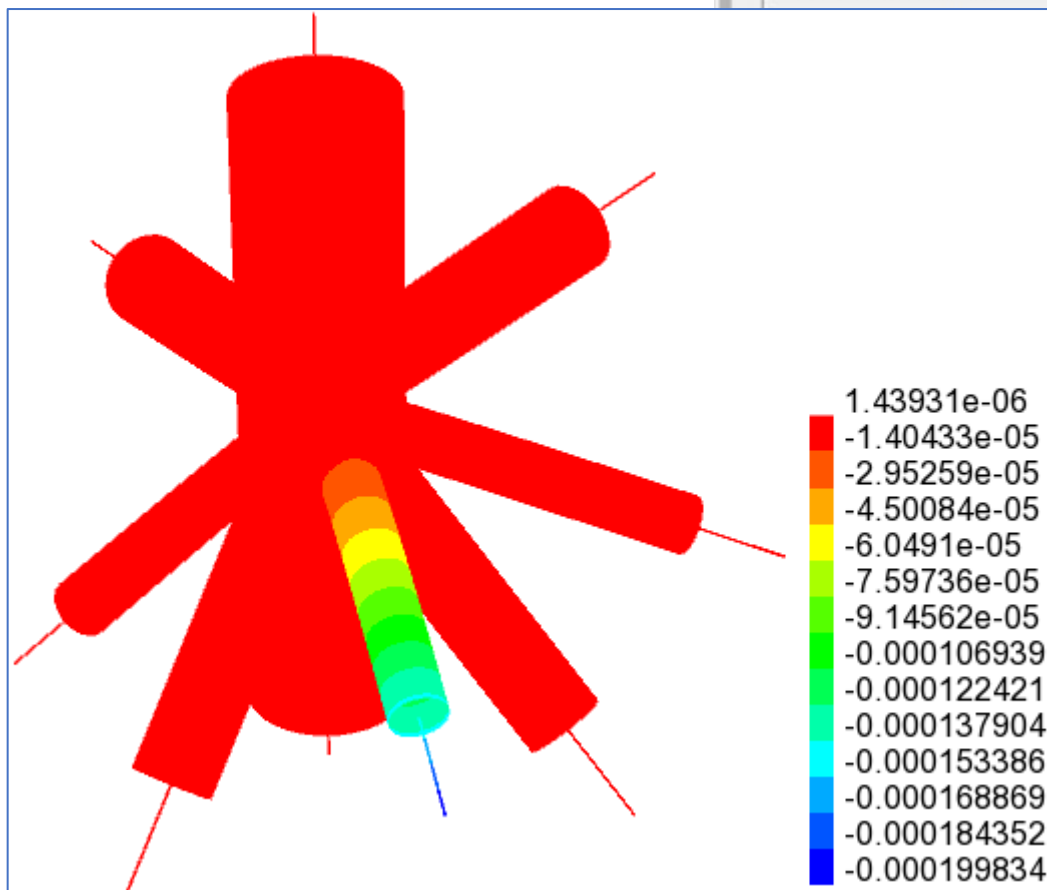
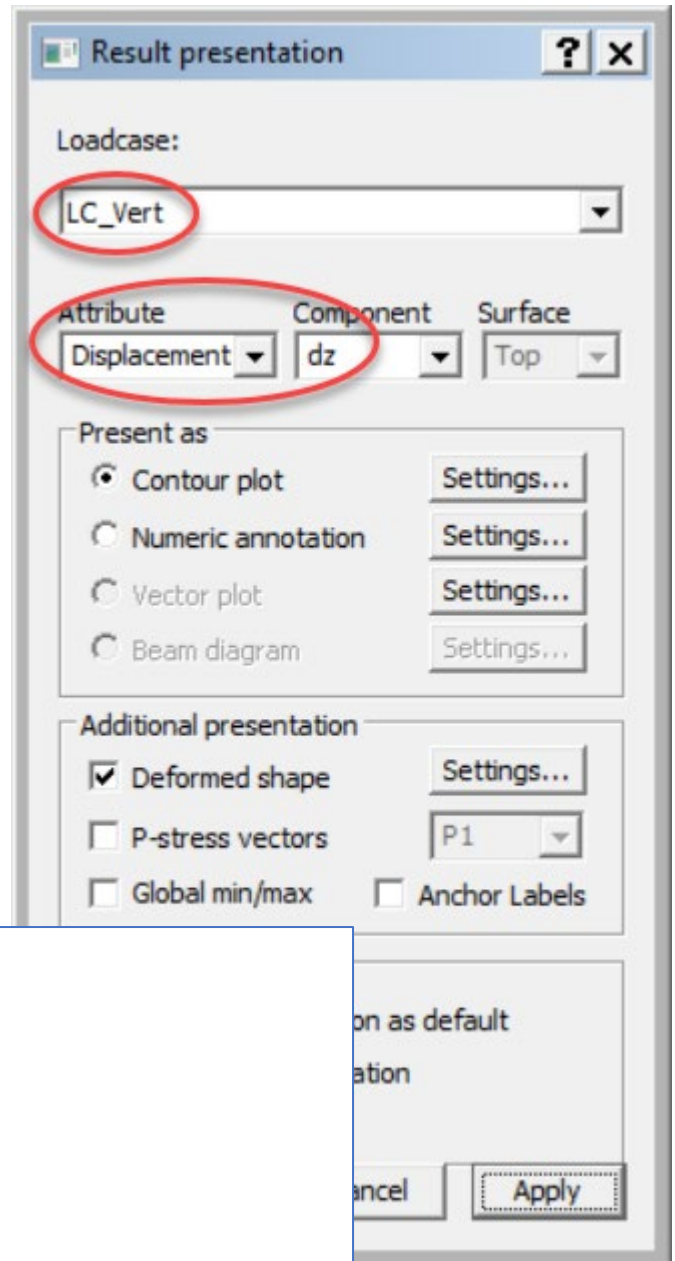
- Use *Mesh & Analysis | Activity Monitor* (or Alt+D) to open the *Activity Monitor*.
 - Right-click the Meshing activity to set *Regenerate mesh option* to *Always Regenerate mesh*. This action may be superfluous but since a FE mesh has already been created, possibly without loads, this is a precaution ensuring that loads are created.



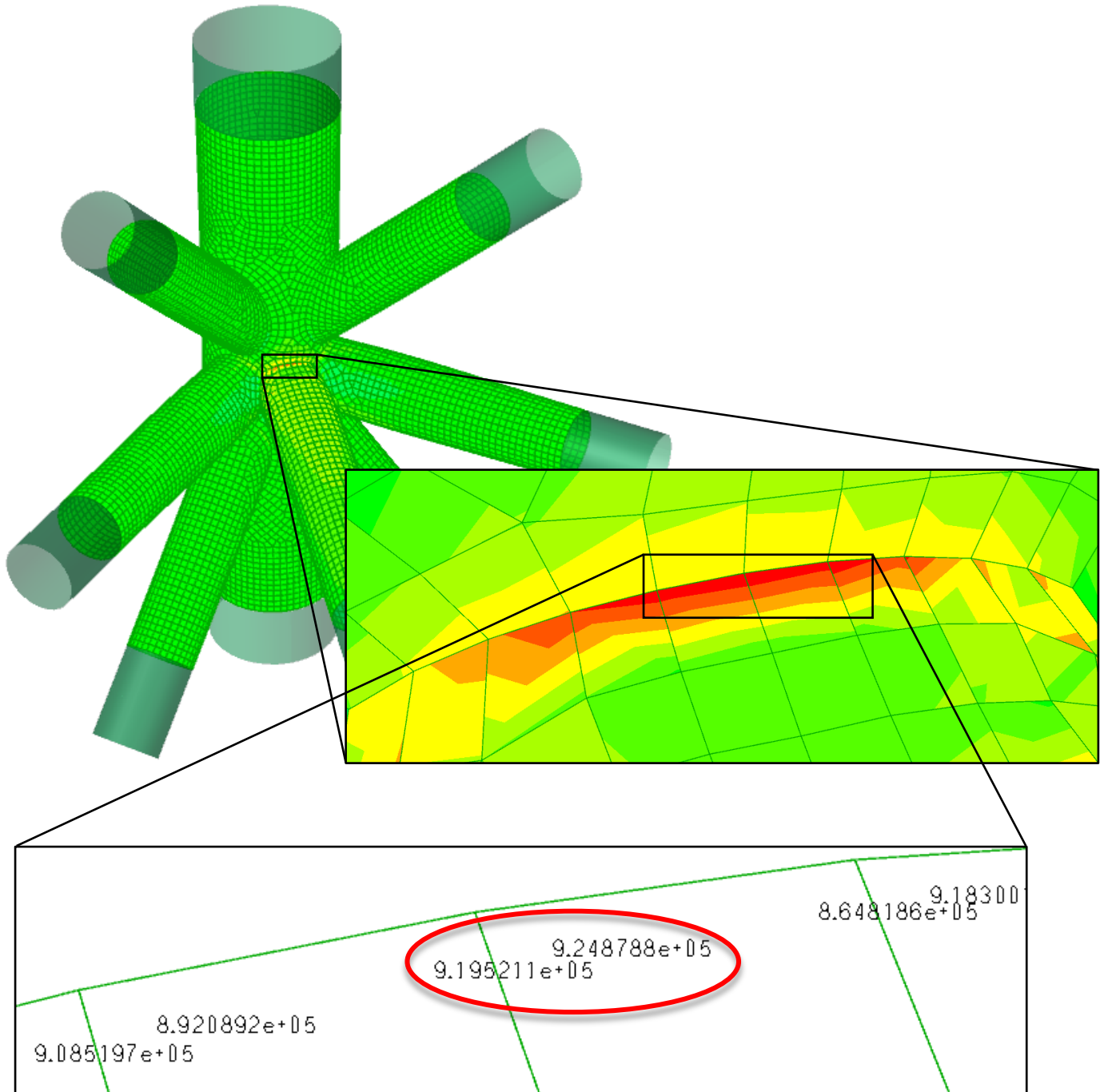
- Again, verify the analysis by right-clicking the *Linear Structural Analysis, Static* activity in the *Activity Monitor* to open the *sestra.lis* file. In this file make sure the loads and reaction forces are correct.

12 EXTRACT RESULTS FOR THE SHELL MODEL

- Switch to *Results - All* display configuration to find the maximum vertical displacement at the free end of the horizontal brace in X-direction.
- Open the *Result Presentation* dialog by *Results | Presentation* (or Alt+P).
 - Select *Loadcase* LC_Vert.
 - Select *Attribute Displacement* and *Component dz*.
 - See that the only displacement of significance is, as expected, the horizontal brace bending down. The maximum value of the legend -0.000199834 corresponds to the free tip of the beam stub.
- The corresponding value from the beam model analysis is -0.000109615 .
 - The factor between the shell and beam model displacements is 1.82.



- Switch to *Results – with Mesh* display configuration to find the maximum axial stress at the brace end connected to the chord.
- Use Alt+P to open the *Result presentation* dialog. *G-stress sigxx* in the *Top* surface is desired stress component. (Xtract can be used to verify that sigxx is the axial stress.)

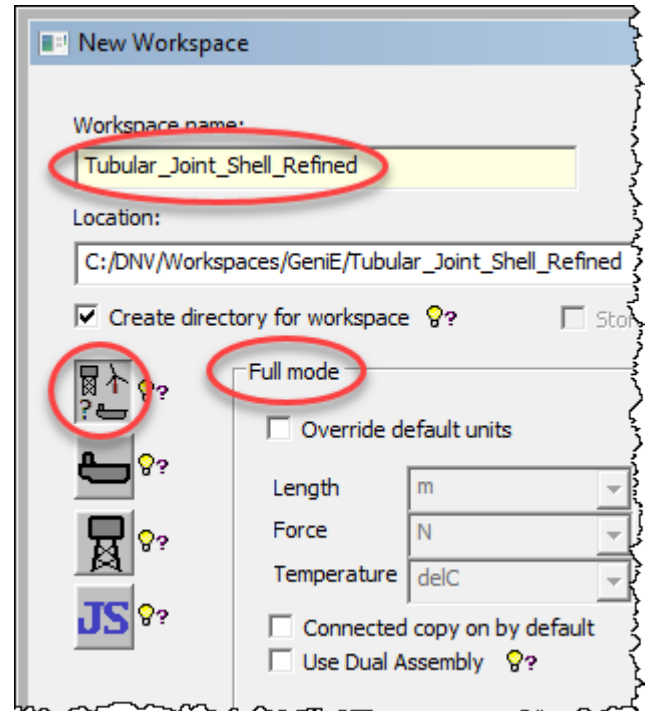
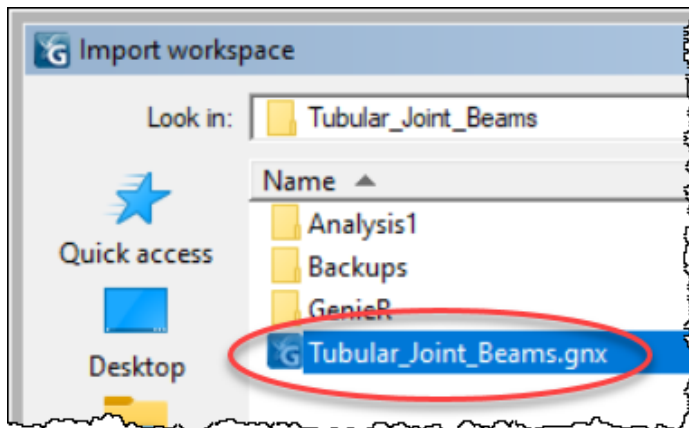


- The stress is $2.47913E6$ and $2.46157E6$ in the two element corners closest to the brace top and weld. The average of these two is: $(2.47913E6 + 2.46157E6)/2 = 2.47035E6$
- The corresponding value from the beam model analysis is 923862.
 - The factor between the shell and beam model stresses is 2.67.

13 CONVERT BEAM MODEL TO SHELL MODEL WITH REFINEMENT ZONES

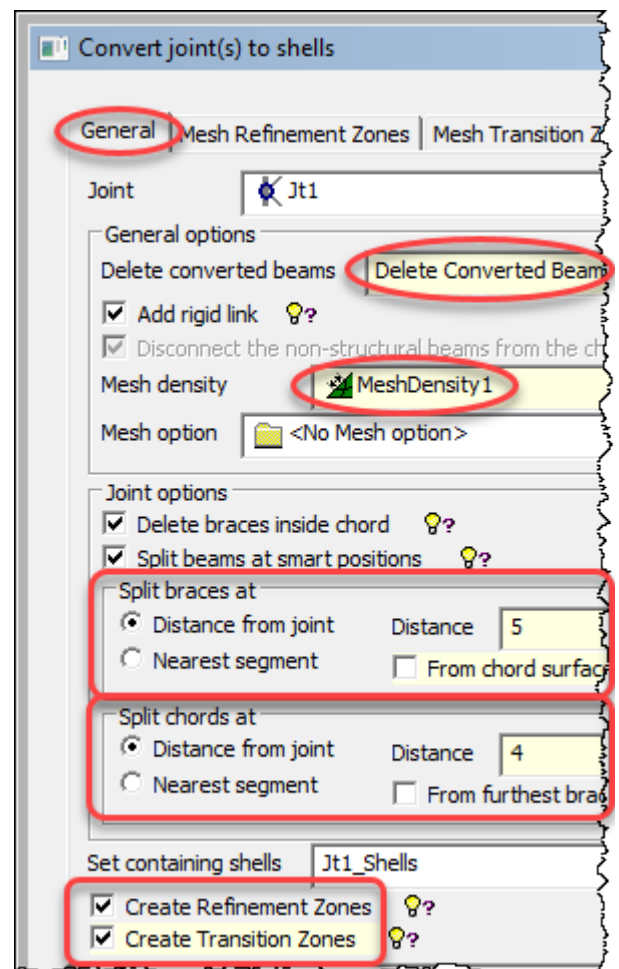
➤ If you followed the tutorial as explained in sections 8 through 12, and now want to try the *Create Refinement Zones* option, then open a new workspace.

➤ Use *File | Import | Workspace (GNX file)* and browse to find the workspace of the beam model.

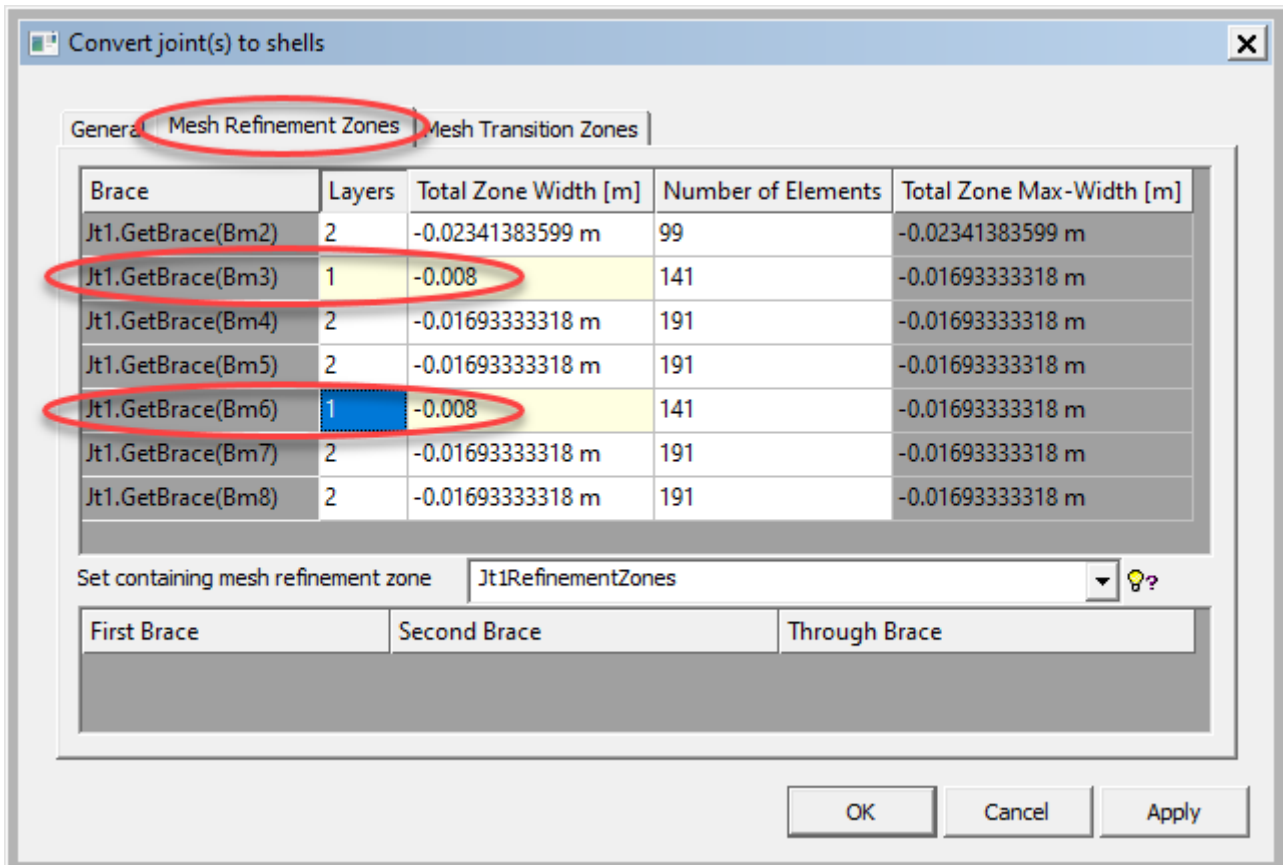


➤ Select the joint, right-click and select *Convert joint(s) to shells*. In the dialog select:

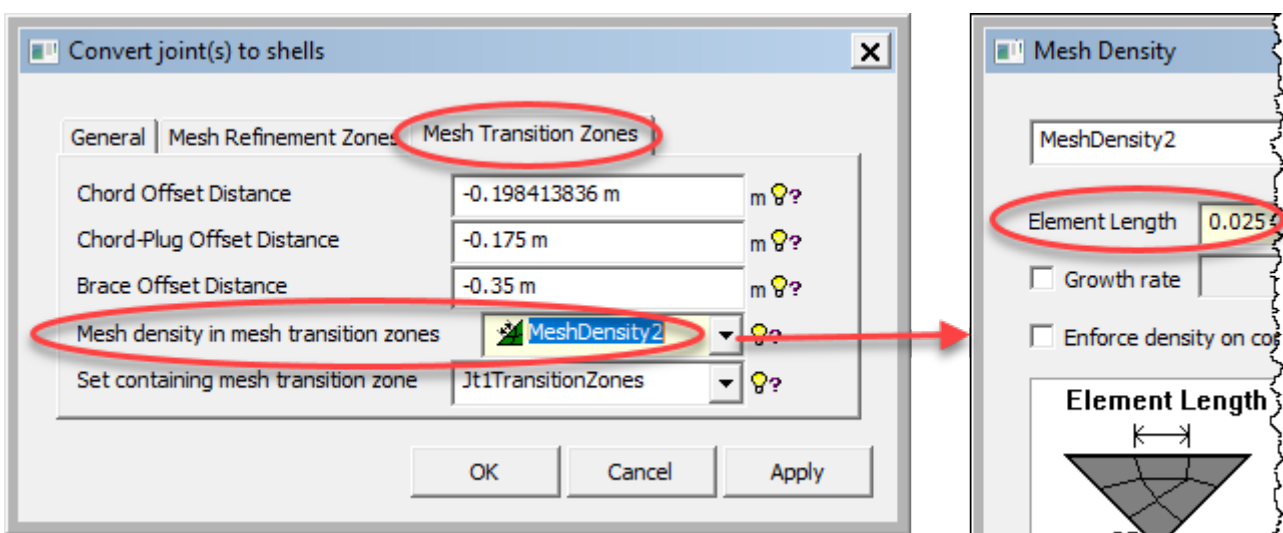
- *Delete Converted Beams* (the beams inside the shells)
- Use the pulldown menu for *Mesh density* to create a new mesh density of 0.1 m.
- All braces are 6 m and the chords 5 m long from the joint centre point. Specify 5 m and 4 m for the braces and chords, respectively, leaving 1 m long beam stubs. The reason for this is explained later.
 - Note that the *From chord surface* and *From furthest brace* must be unchecked.
- Check the *Create Refinement Zones* option for controlling the mesh next to the welds at the brace-chord intersections. This involves that feature edges are created as specified in the *Mesh Refinement Zones* tab, see below.
- Check the *Create Transition Zones* option for controlling the transition from fine to coarse mesh as specified in the *Mesh Transition Zones* tab, see below.



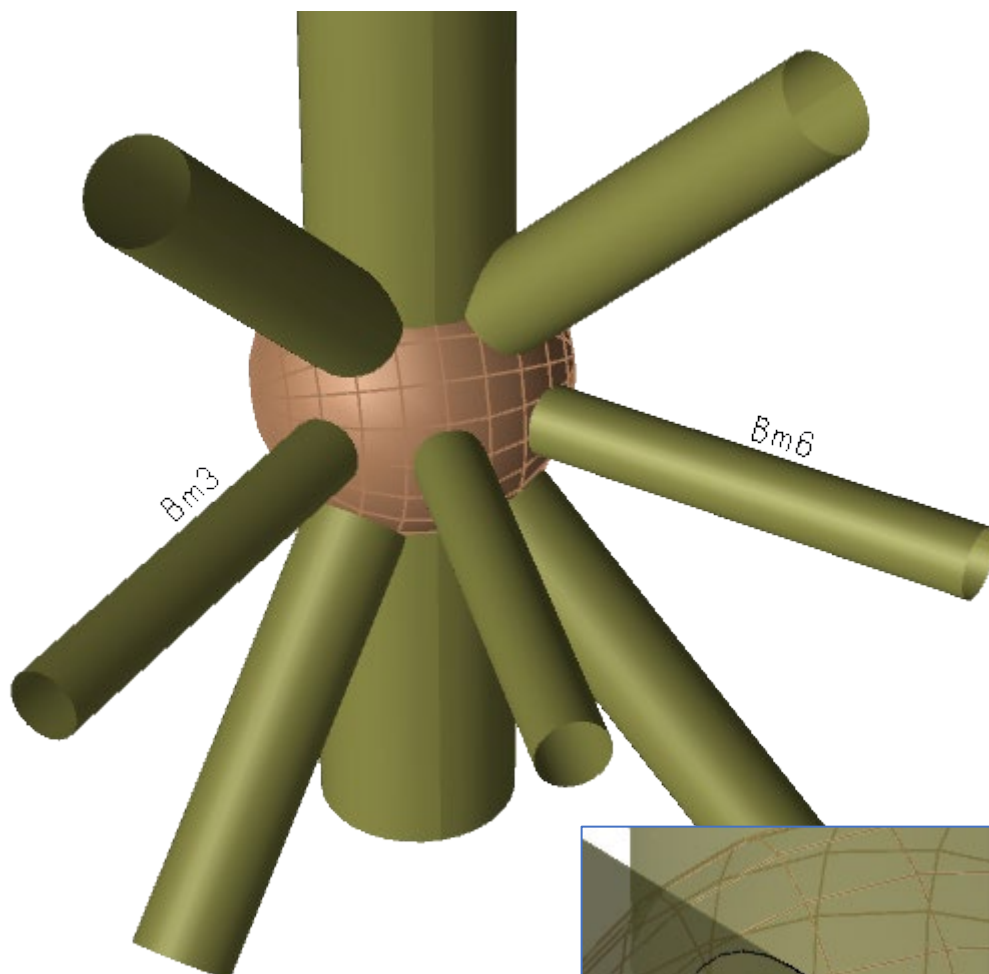
- The *Mesh Refinement Zones* tab allows control of the number of layers and widths of the refinement zones next to the welds at the brace-chord intersections. The program computes the widths of the zones based on the prevailing gaps between the brace-chord intersections.
 - Reduce the zone widths and number of layers for the two horizontal braces in-plane with the diagonal braces. These horizontal braces are named Bm3 and Bm6, see next page.



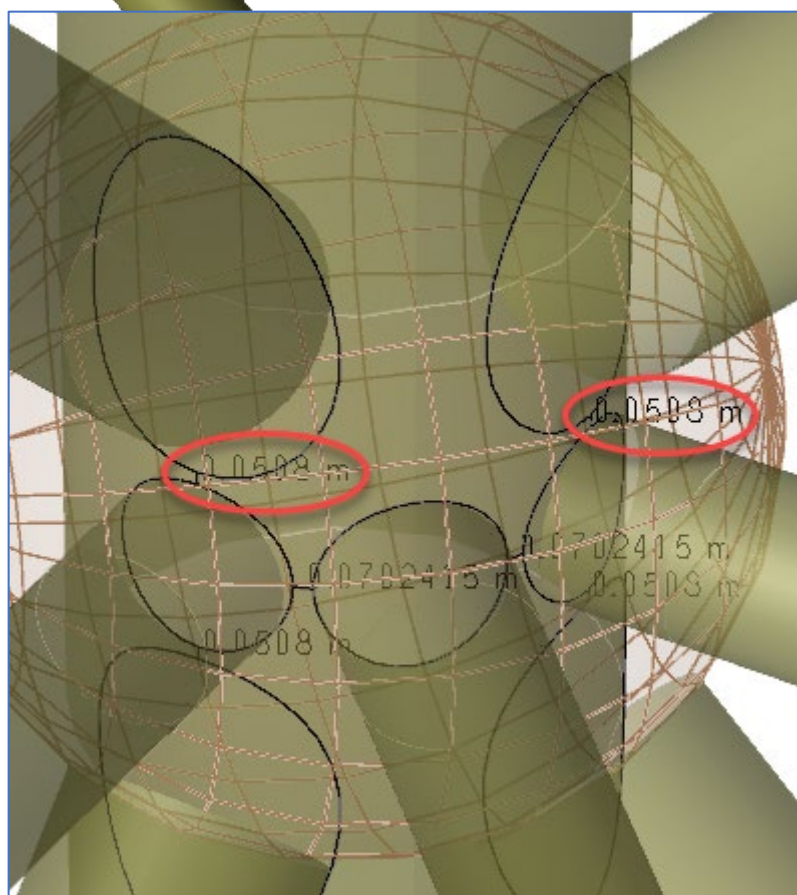
- The *Mesh Transition Zones* tab allows control of the transition from fine mesh at the weld zones (brace-chord intersection) to the coarser mesh for the cylindrical shell parts away from the weld zones. Create a new mesh density (0.25 m) for these zones.



- The refinement zones for the two horizontal braces in-plane with the diagonal braces, Bm3 and Bm6, shall be adjusted.

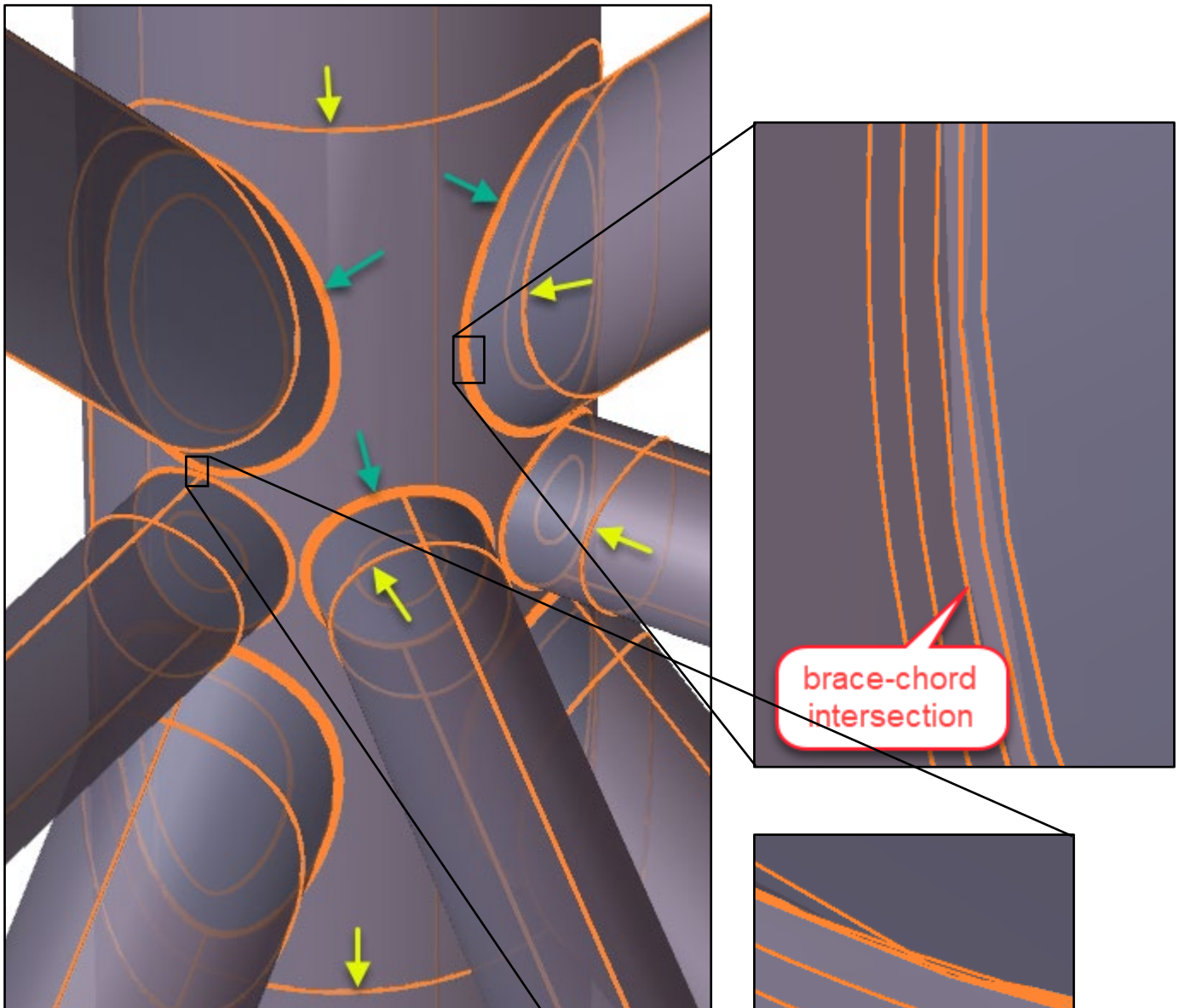
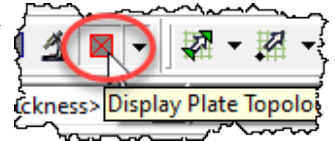


- Double-clicking the joint displays the brace-chord intersection curves as well as the gaps between them. The encircled values to the right are 0.0508 m.

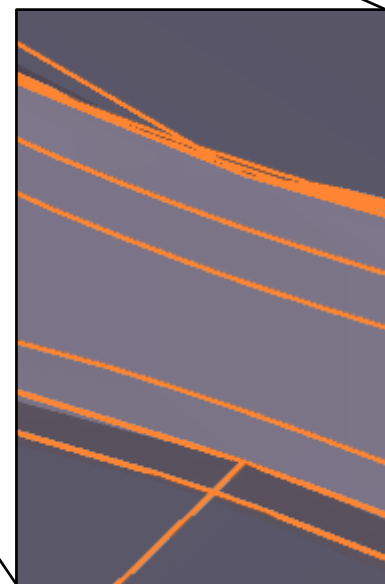


➤ Click OK in the *Convert joint(s) to shells* dialog to perform the conversion.

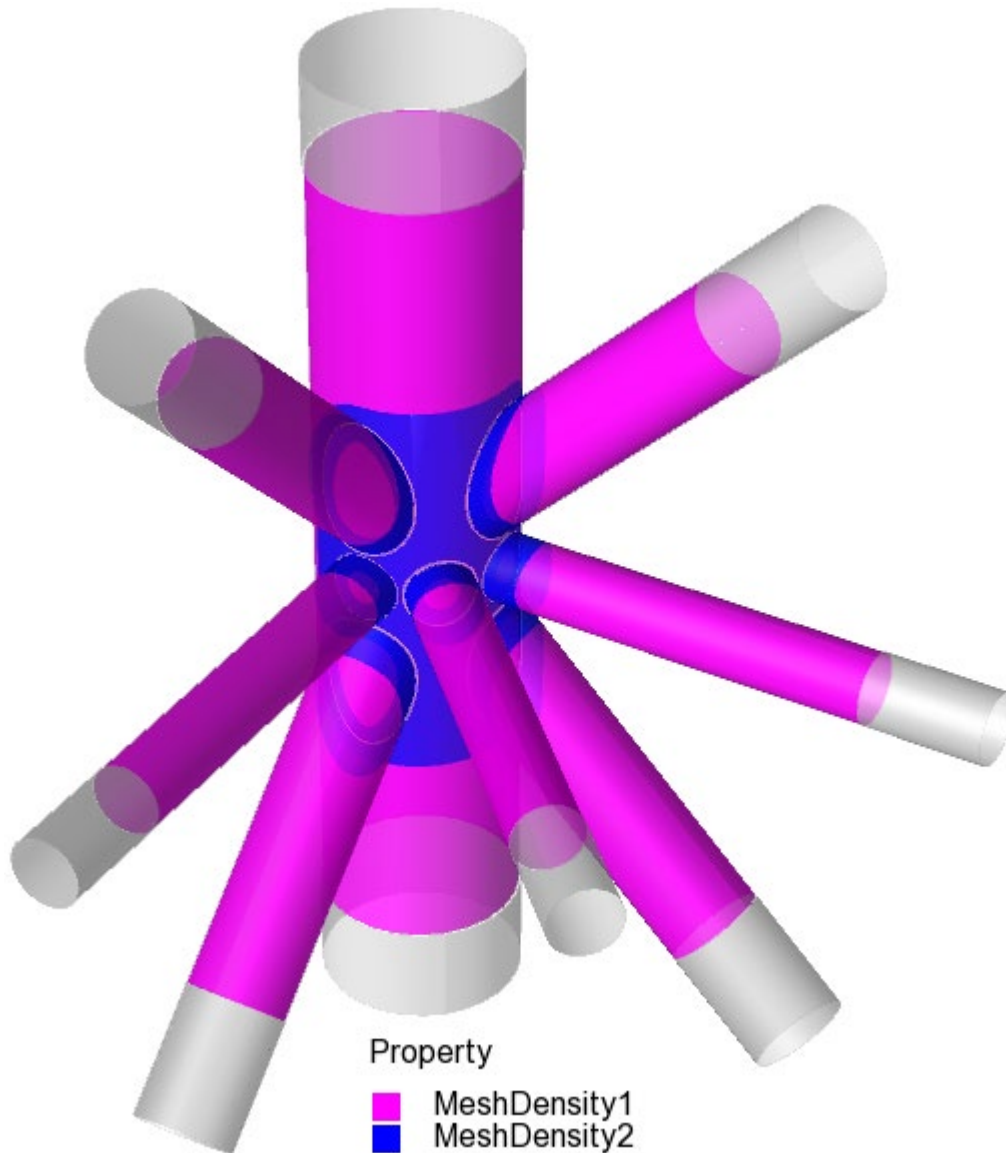
➤ The model will now appear as shown below after pressing the *Display Plate Topology* button. The yellow arrows point to the transition zone boundaries. The green arrows point to the refinement zones.



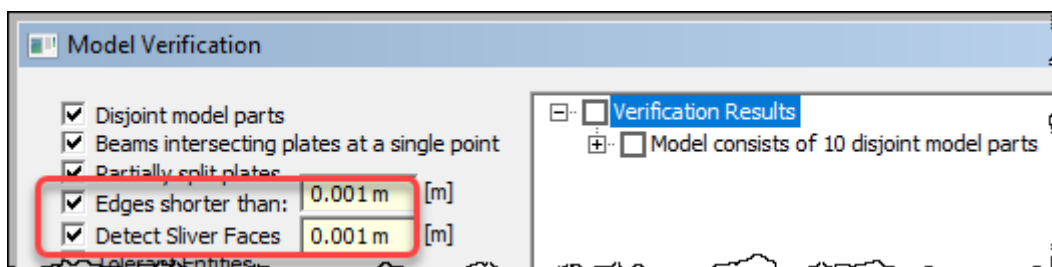
➤ The zoomed view to the right shows the diagonal brace with a transition zone with two layers, and the horizontal brace with a transition zone with a single layer.



- Colour coding mesh densities shows that the transition zones have a different (finer) density than the outer parts of the shell.

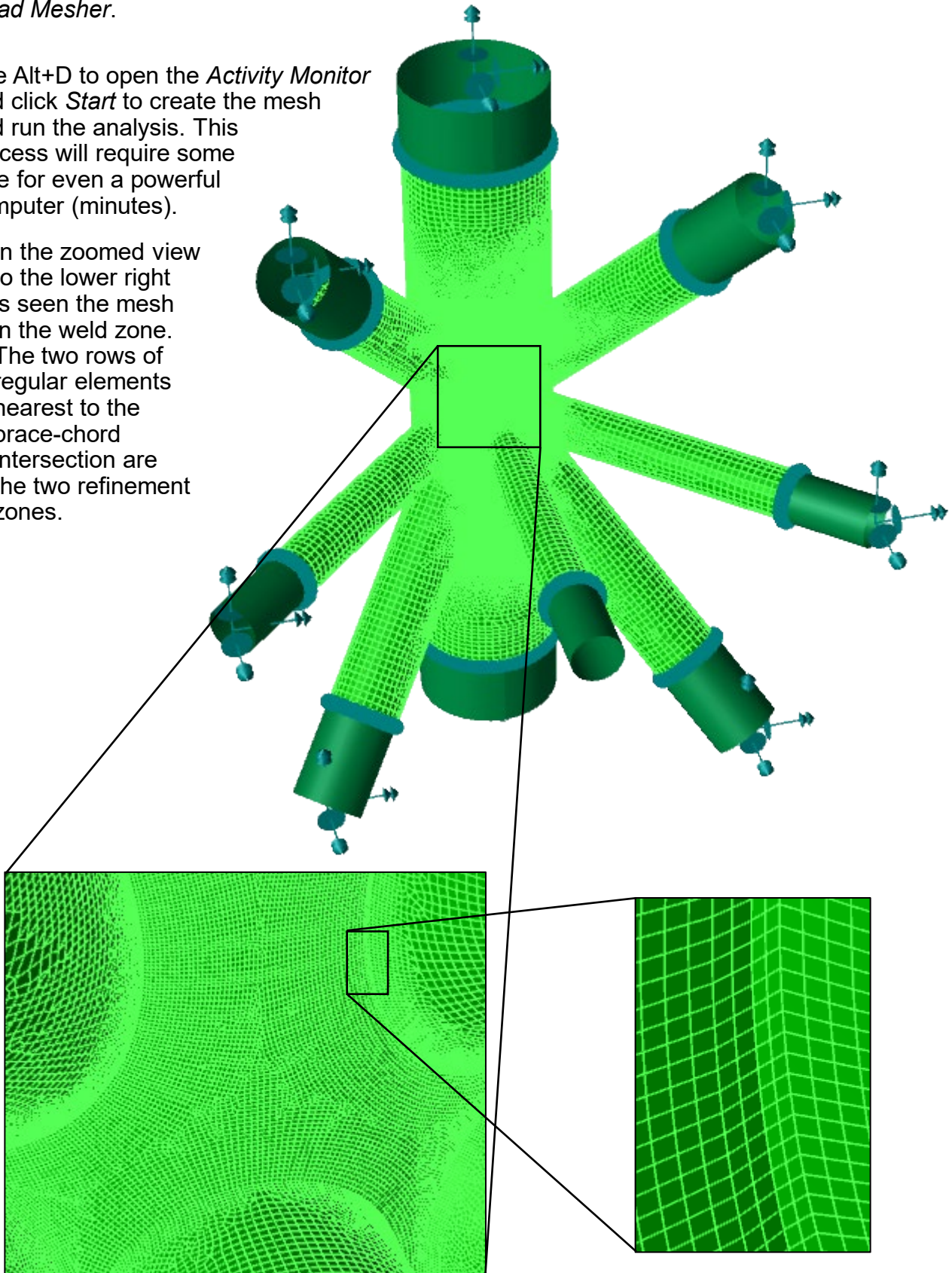


- Do a *Structure | Topology | Simplify Topology* to clean up the model.
- When verifying the consistency of the model by *Structure | Topology | Verify Model*, the *Edges shorter than* and *Detect Sliver Faces* should be decreased to avoid the verification reacting to the very narrow refinement zones. The model will still consist of ten disjoint parts since the shell part is not geometrically coupled to the beam stubs, rather they are connected by linear coupling between nodes. This is therefore OK.



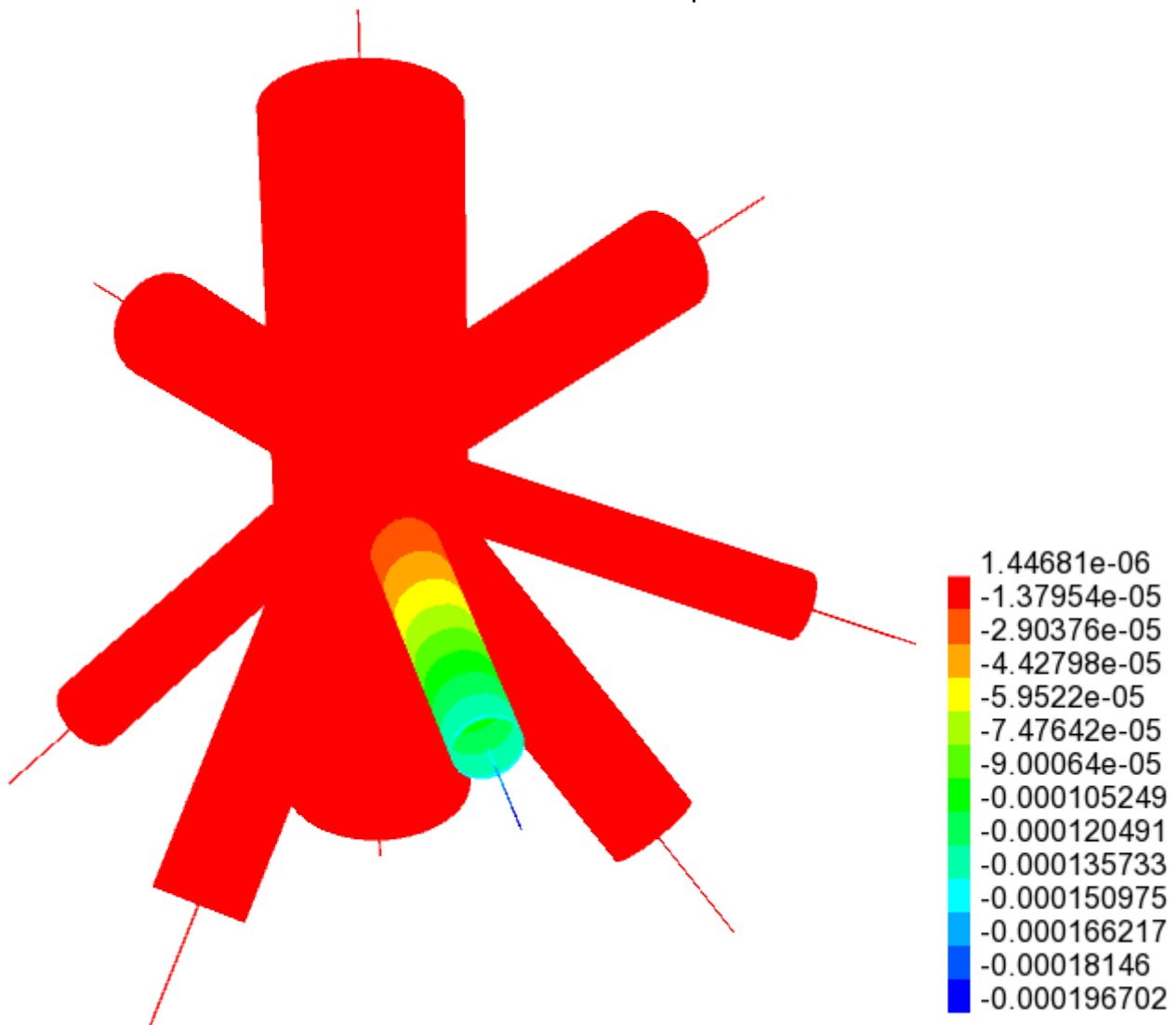
14 CREATE FE MESH FOR THE SHELL MODEL WITH REFINEMENT ZONES

- The shell model is now ready for analysis but prior to this, select a meshing algorithm more suitable for tubular joints. Use *Edit | Rules | Meshing Rules* and select *Advancing Front Quad Mesher*.
- Use Alt+D to open the *Activity Monitor* and click *Start* to create the mesh and run the analysis. This process will require some time for even a powerful computer (minutes).
 - In the zoomed view to the lower right is seen the mesh in the weld zone. The two rows of regular elements nearest to the brace-chord intersection are the two refinement zones.

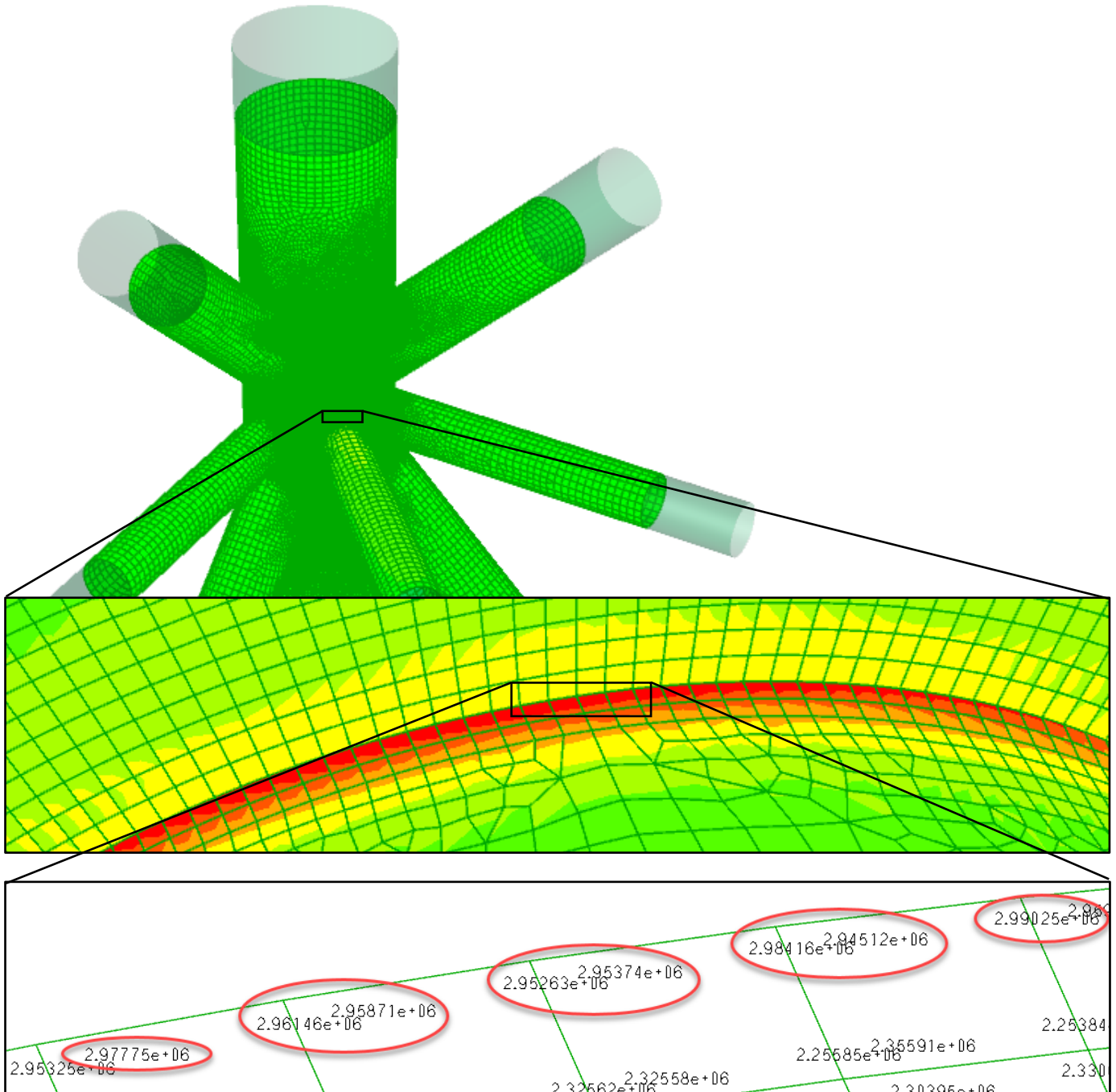


15 ANALYSE AND VIEW RESULTS FOR THE SHELL MODEL WITH REFINEMENT ZONES

- Use *Mesh & Analysis | Activity Monitor* (or Alt+D) to open the *Activity Monitor* and click *Start* to run the analysis.
- Switch to *Results - All* display configuration to find the maximum vertical displacement at the free end of the horizontal brace in X-direction.
- Open the *Result Presentation* dialog by *Results | Presentation* (or Alt+P).
 - Select *Loadcase LC_Vert*.
 - Select *Attribute Displacement* and *Component dz*.
 - The maximum value of the legend -0.000196702 corresponds to the free brace tip.
- The corresponding value from the beam model analysis is -0.000109615 .
 - The factor between the shell and beam model displacements is 1.79.



- Switch to *Results – with Mesh* display configuration to find the maximum axial stress at the brace end connected to the chord. Use Alt+P to open the *Result presentation* dialog. *G-stress sigxx* in the *Top* surface is desired stress component.



- With such a fine mesh it is a question of how many of the elements on each side of the top point of the brace to take into account to determine the stress. In this case two elements on each side are used. The average of the encircled values above is 2.965478E6
- The corresponding value from the beam model analysis is 923862.
 - The factor between the shell and beam model stresses is 3.21.



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.