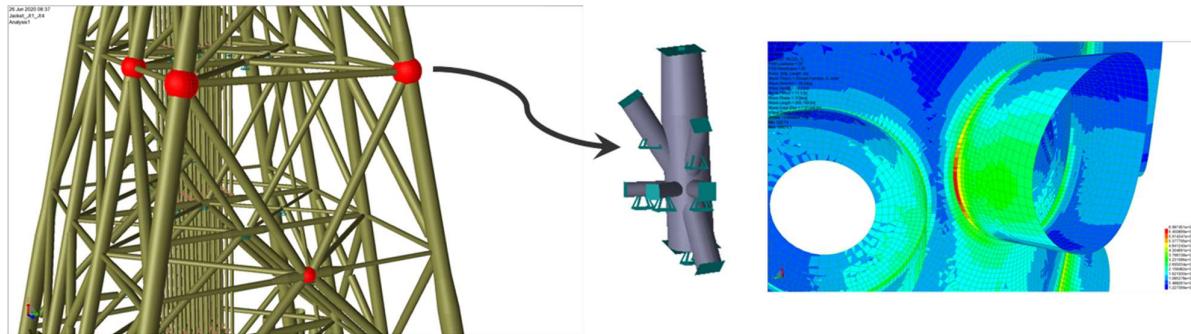


SESAM GENIE TUTORIAL – CONVERSION OF TUBULAR JOINTS



This example will show you how to convert tubular joints to shell models and control the mesh so that it matches your needs or other requirements. The focus of this tutorial is to demonstrate how to convert tubular joints without brace overlaps and to create a mesh that has the same mesh density (for stress screening) or a mesh that includes a mesh refinement zone around all brace/chord connections (for refined stress analysis). The latter is a useful approach in case you want to make a mesh supporting DNV RP-C203 Chapter 4.2 to do fatigue analysis. Note that other mesh editing functionalities in GeniE can be used, but they are not described here. Tubular joints with brace overlaps can easily be done and will be part of a separate tutorial.

The Sesam programs GeniE V8.4-06 (inclusive of the program extension "CGEO - Curved"), Wajac V7.8-00, Splice V8.0-00, Sestra V10.16-00 and Xtract V6.0-02 or later versions are required for the presented functionality.

This workshop has two input files:

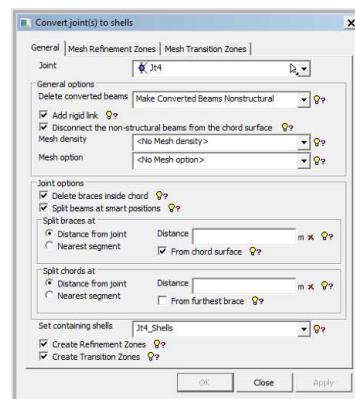
1. *Tubular_joint_initial.gnx* – A complete jacket model with pre-defined joints Jt1 – Jt2 at elevation -15.0 m. Import this model file and use it when you convert the tubular joints to shell model with automatic connection to the jacket model
2. *Tubular_joint_completed.gnx* – Import this workspace file to get a concept model including the conversion of joints Jt1 – Jt2 plus other joints at elevation -15.0 m. Note that the tutorial addresses the conversion of Jt1 – Jt2 only.

The main focus of the tutorial is to explain how the command "Convert joint(s) to shells" works.

There are two abbreviations frequently used:

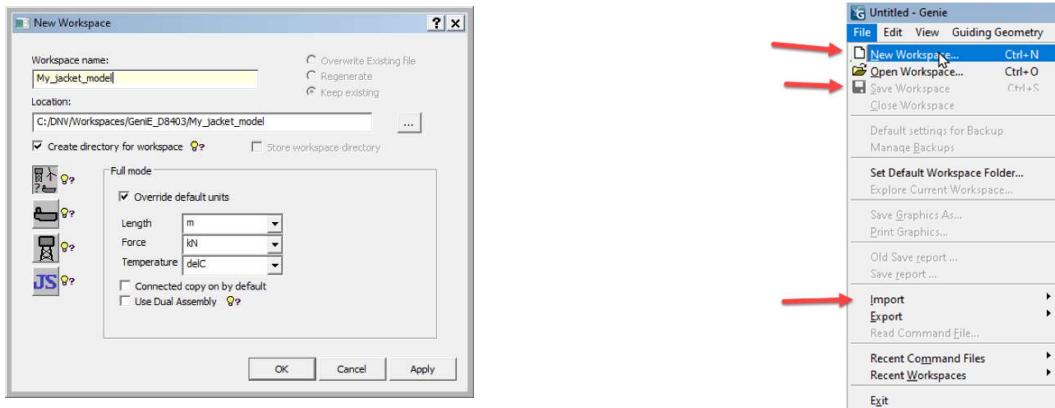
LMB: Left Mouse Button

RMB: Right Mouse Button

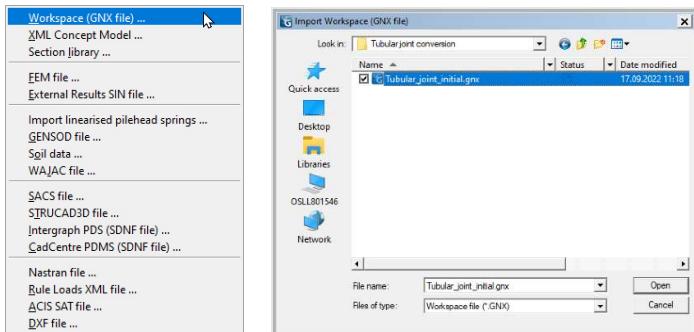


1 CREATE THE BEAM MODEL

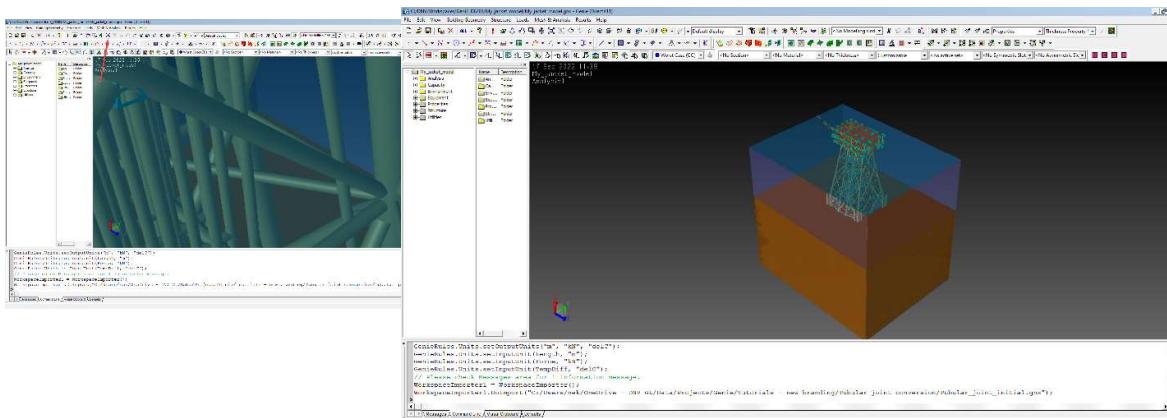
Start GeniE and **(1)** make a workspace called "My_jacket_model" followed by **(2)** "Save Workspace". Check the "Override default units" if you want to modify units like in this tutorial (or to some other alternative).



Then **(3)** import the input file *Tubular_joint_initial.gnx* by using the Import functionality "Workspace (GNX file)". Click **(4)** "Save Workspace".

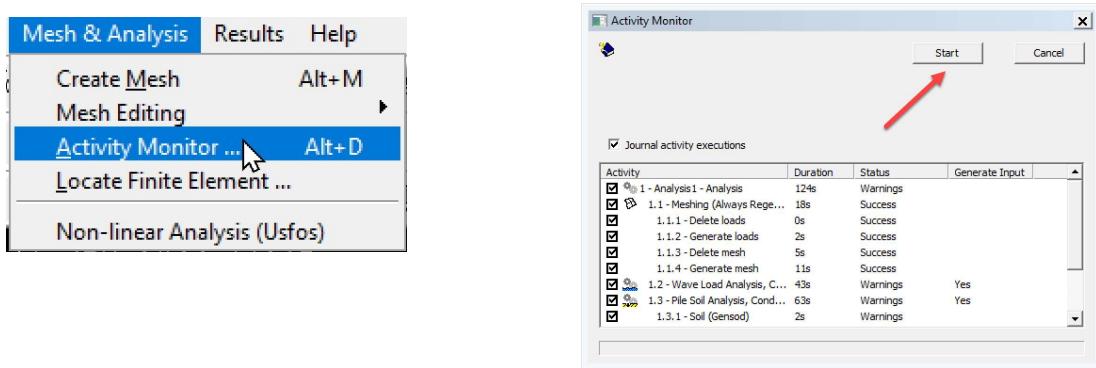


You will now see the following. **(5)** Click the "Fit to window button"  to see the whole jacket.



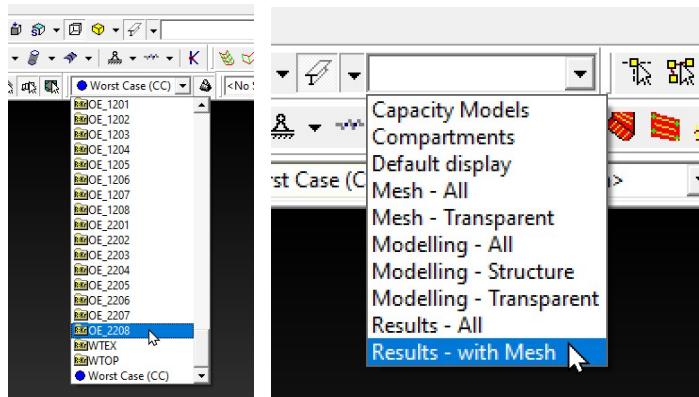
2 RUN FIRST ANALYSIS

To check that you have a sound concept model an analysis should be executed. **(1)** click "Mesh & Analysis – Activity Monitor" or ALT+D to **(2)** start analysis.

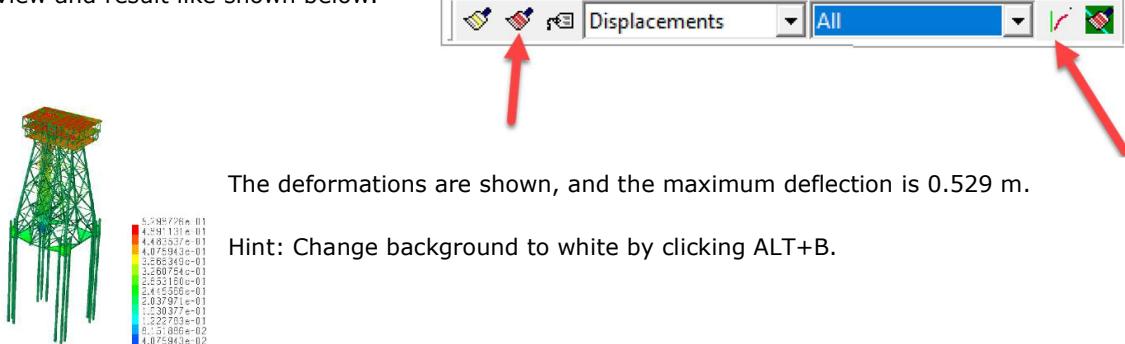


When the analysis has been completed then check the global displacements for loadcase OE2208.

(3) from the loadcase selection window, find OE2208 and select it. **(4)** from the view settings select "Results – with Mesh".

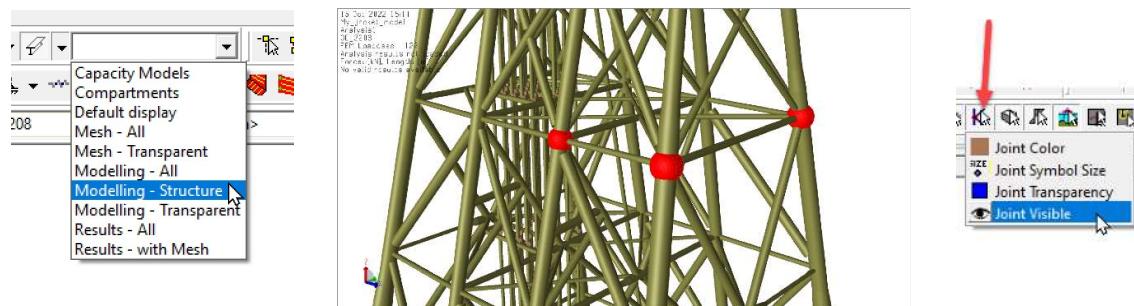


The finite element model will now be shown. **(5)** to see the results for displacements, then select "Displacements" and "All" plus hit the symbol for deformed model. The results are added to the display when you hit the paintbrush symbol "Structure color coding of all visible concepts". This will give the view and result like shown below.

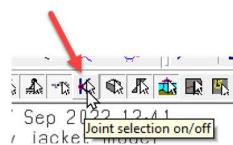


3 CONVERT JT1

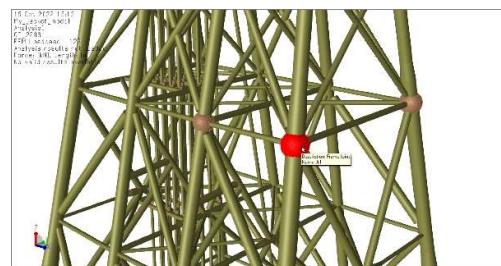
(1) Change the view setting to "Modelling – Structure". The view focus is now on the structure and neither soil nor wave are shown. To make joints visible, you need to right click joint selection on/off button to view the joints. Jt1 – Jt3 are highlighted in the view below.



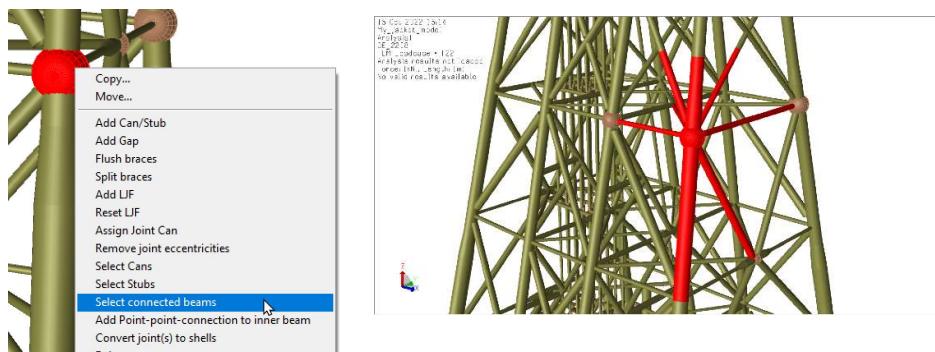
Zoom in on the joints at elevation -15 m. The pre-defined joints are shown as red balls. **(2)** To select a joint, you need to activate joint selection from



(3) Select Jt1 by clicking on it (left mouse button). It will be highlighted as shown below.

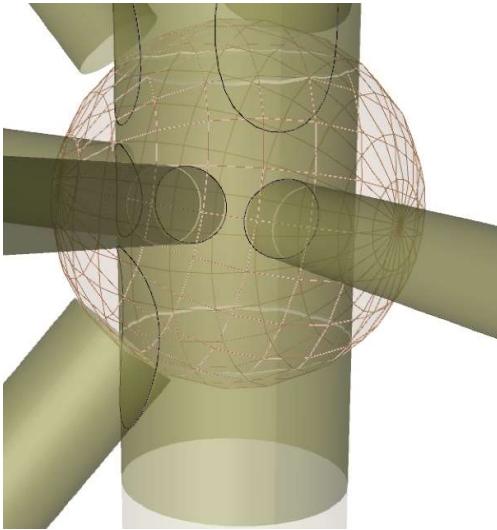


Hover the mouse symbol over Jt1 (and make sure it stays highlighted), then **(4)** select the beams connected to Jt1 by right clicking the right mouse button and use of the command "Select connected beams". **(5)** Click ALT+S (or use the browser under Files) to show only the selected beams and Jt1. Note that the view on selected beams only has been rotated with a viewpoint from inside the jacket.





(6) Check if the tubular joint has overlapping braces. Double-click Jt1 and zoom in.



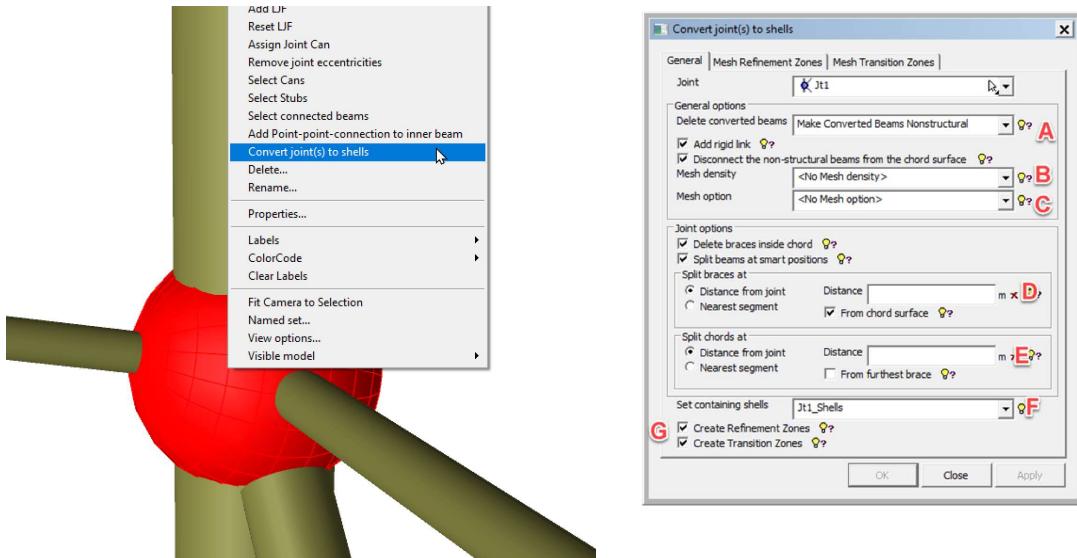
There are no overlapping braces for Jt1. This means that there should be space between all braces to make a finite element mesh.

For this example, the purpose of converting Jt1 to a shell model is to create a mesh that has same size for the whole model. This method is often used if the purpose is to e.g. do a fatigue screening where the mesh requirements are not so strict as they would be for a refined fatigue analysis (such a mesh will be created for Jt2 and Jt4, see later).

(7) Double-click Jt1 again to get back to normal view.

The diameter of the chord is 2.4 m and the diameter of the largest brace is 1.3 m. We will use 3 times these values when we decide where to divide the beams (i.e., at the outer edges of the shell model). It is your decision where to split, but literature and DNV research all recommends a factor of 3.

(8) Select Jt1, use RMB (Right Mouse Button) and select "Convert joint(s) to shells".

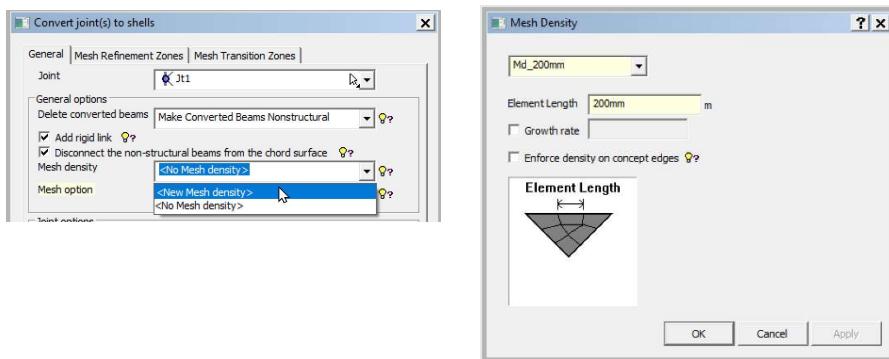


The following input parameters could be used for this exercise. More definitions may be found in the GeniE User Manual or by clicking on the light bulbs:

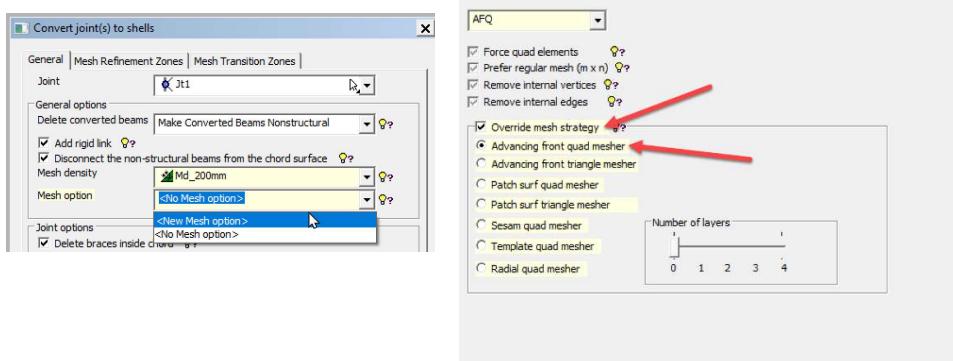
- A. "Make converted beams non-structural" is the default setting. This means that wave loads will be generated on the beams and transferred to the shell model at the connection points between beams and shells. The non-structural beams do not contribute to the stiffness of the structure. Alternatively, you may specify that the converted beams are deleted (i.e. no wave loads are calculated) or that they remain as beams (i.e. wave calculation and stiffness contribution).

- B. The mesh density for the whole shell model is now defined. The smaller the mesh is the larger the FE model becomes. If a refined model is the goal, then it is advised to use option G. This will be shown on Jt2 and Jt4. For this case, we specify the mesh density to 200mm.
- C. The selection of mesh option will instruct GeniE on which mesh algorithm to use. It is our experience that the option "Advanced Front Quad Mesher" gives the best mesh around chord/brace connections.
- D. Since the brace diameter is 1.3 m, we use the value 3.9m (i.e., 3 times the brace diameter)
- E. Similarly, we use 7.2 m for the chord (i.e., 3 times the chord diameter)
- F. The shell model is automatically included in a named set. The default name is Jt1_Shells. We will use this when we look at the mesh and results
- G. We will not include mesh refinement nor mesh transition zone for the shell model of Jt1. This will be shown when converting joint Jt2.

(9) The mesh density Md_200mm is specified as follows:

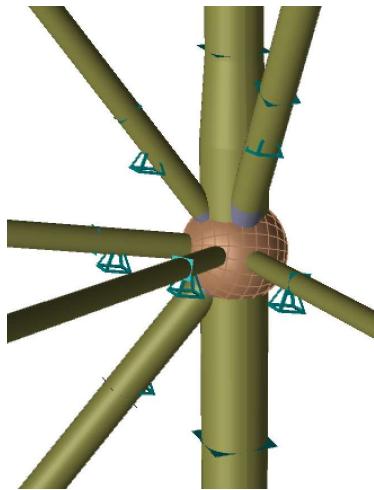
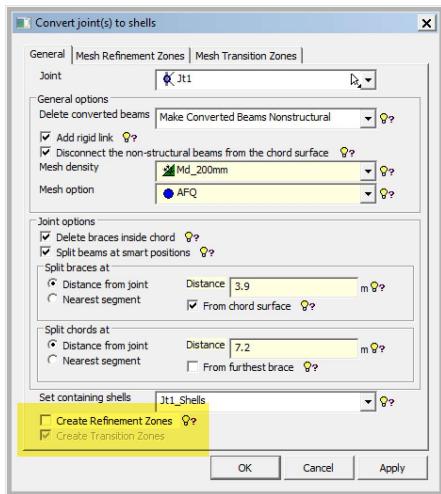


(10) The mesh option AFQ is defined like this:



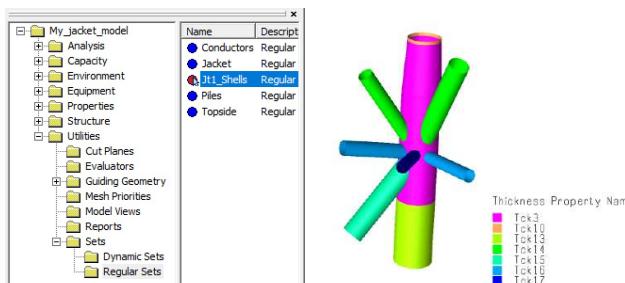


(11) Fill in the values for brace (3.9m), chord (7.2m) and deselect options for refined mesh and transition zones and click OK.



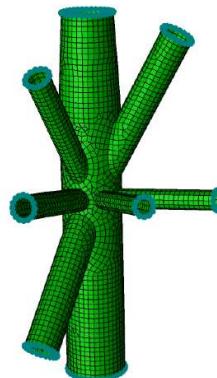
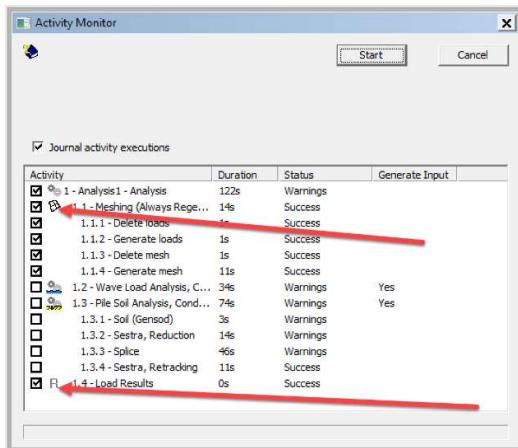
The rigid link symbols show the connection between shell and beam model.

(12) To get a better view of the shell model you could select the named set Jt1_Shell from the browser and use the short command ALT+S. To view all again use ALT+A.



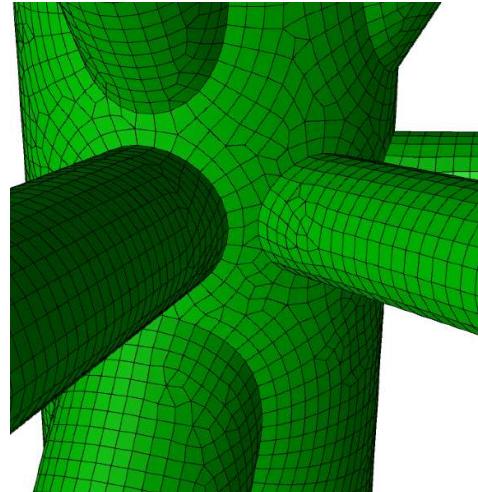
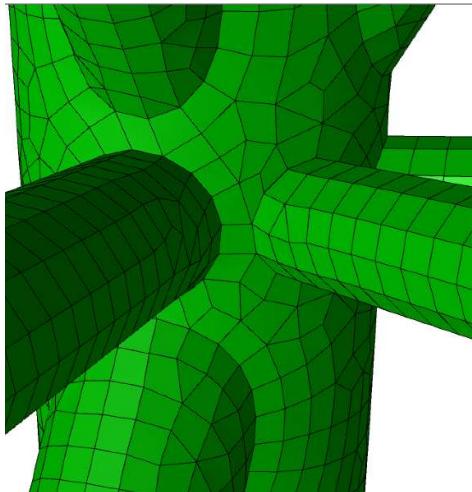
The joint has been converted and the properties for shell thickness and material are the same as for the beam model.

(13) Inspect the mesh quality by creating the mesh (either from short cut ALT+M) or from Activity Monitor

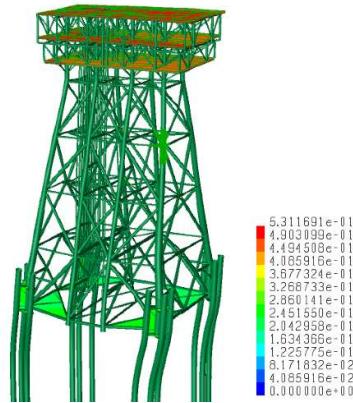
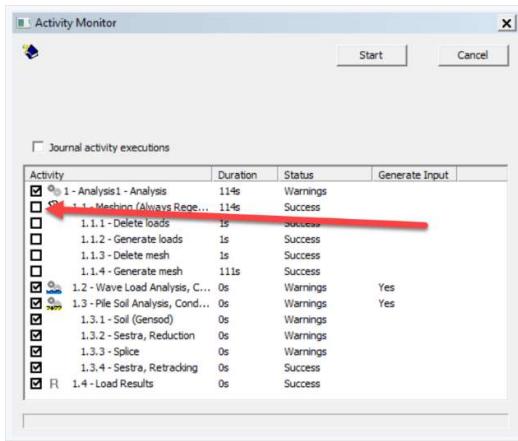




When zooming in on the joint we see that the mesh is coarse like we specified (picture to the left). The effect of changing mesh density to 100mm is shown on the picture to the right below.



To check that the model behaves like expected you can check the global displacements. **(14)** Run a full analysis. There is no need to create the mesh, so you can deselect Meshing. **(15)** After analysis change the view to "Results – With Mesh" and color code displacements for loadcase OE_2208. The maximum displacement is 0.531 m which is slightly different than the pure beam model (0.529 m). The reason is because the shell model will introduce more flexibility in the joint.



Hint: Make a new set where the non-structural members are not included and use this set during result presentations.

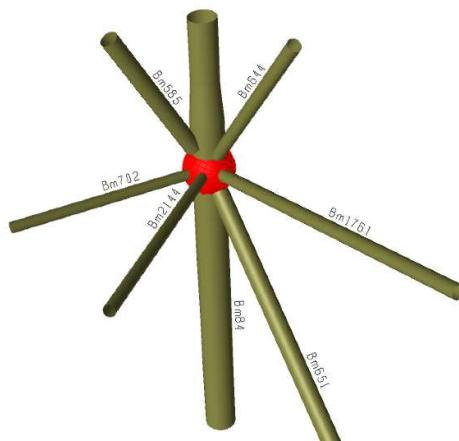
(16) Save the model before you continue.

4 CONVERT JT2

This joint is identical to Jt1, but during the conversion of Jt2 we will include mesh refinement zones and transition zones to make a

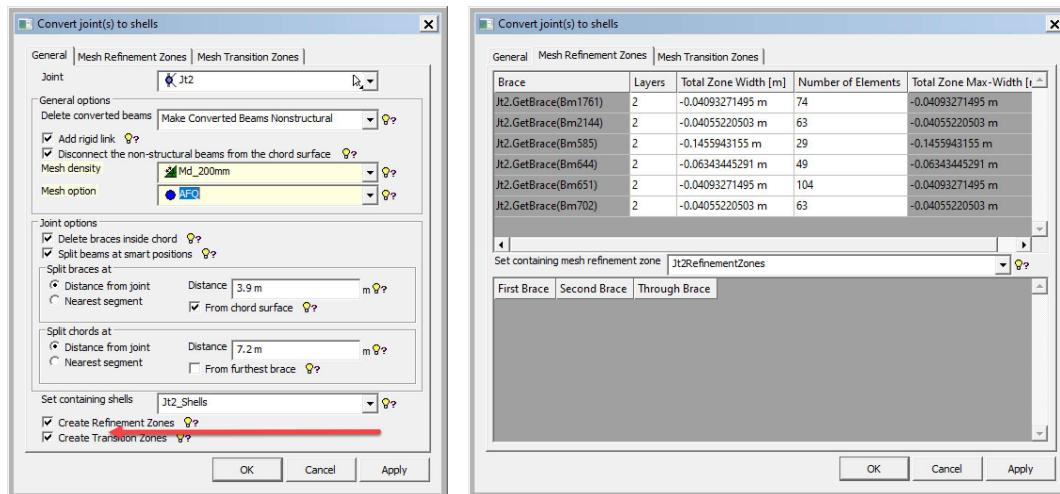
- Coarse mesh for non-critical part
- Refined mesh for areas of interest

According to DNV RP-C203 Chapter 4.2 the mesh density should be $0.1 * \text{SQRT}(\text{radius brace} * \text{thickness brace})$ and there should at least be one layer of 1st order radial square mesh. For this tutorial we will use the same mesh density and number of layers around each brace. The mesh density used is 20mm and 2 layers.



Start with **(1)** select Jt2 and RMB -> Convert joint to shell. The same input parameters as Jt1 are used, but this time we also select the "Create Refinement Zones" and "Create Transition Zones".

(2) Click the tab "Mesh Refinement Zones". This will open a new window as shown to the right. The default is 2 layers of regular mesh zones around each brace/chord connection, both on the chord surface, the brace surface and inside the brace on the chord surface (chord plug).



The following parameters are important when you define the mesh settings to support your criteria based on DNV RP-C203 or others:

- Number of Layers of the zone, two layers means two feature edges outside the chord-brace intersection, for both the chord and brace surfaces
- Total Zone Width is the width of the given number of Layers combined, it cannot exceed the distance between the braces, it is given as a negative value, but the absolute value is used in the calculations
 - o **This means that if you want 2 layers each of 20mm the input parameter is -40mm**
- Number of Elements along the feature edges, corresponding mesh properties are created and assigned to the feature edges
- Total Zone Max-Width is the highest allowable value for the Total Zone Width

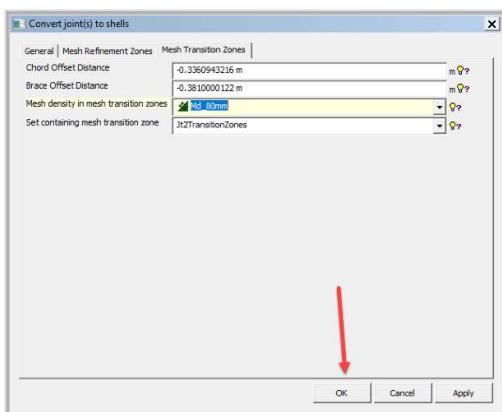
(3) Using the mesh settings and 2 layers the input should be modified to

General Mesh Refinement Zones Mesh Transition Zones				
Brace	Layers	Total Zone Width [m]	Number of Elements	Total Zone Max-Width [m]
Jt2.GetBrace(Bm1761)	2	-40mm	76	-0.04093271495 m
Jt2.GetBrace(Bm2144)	2	-40mm	64	-0.04055220503 m
Jt2.GetBrace(Bm585)	2	-40mm	99	-0.1455943155 m
Jt2.GetBrace(Bm644)	2	-40mm	76	-0.06343445291 m
Jt2.GetBrace(Bm51)	2	-40mm	106	-0.04093271495 m
Jt2.GetBrace(Bm702)	2	-40mm	64	-0.04055220503 m

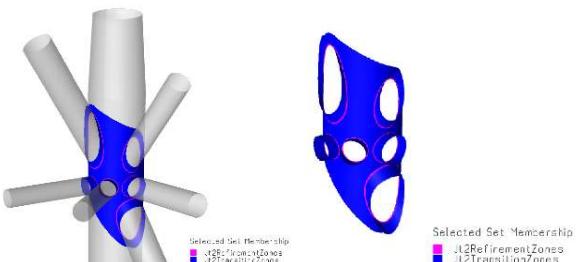
In case you input a combination of layers and total zone width that is larger than the "Total Zone Max" the error will be highlighted, and you need to modify.

Either modify to 1 layer or reduce the element size (width of layer).

(4) click on tab "Mesh Transition Zones". **(5)** Create a mesh density of 80 mm and use for the transition part. For the rest use the default options. **(6)** Finally click OK.



There are now autogenerated named sets; Jt2_Shells, Jt2RefinementZones and Jt2TransitionZones. They are colour coded on the picture to the left and showed more in detail below.

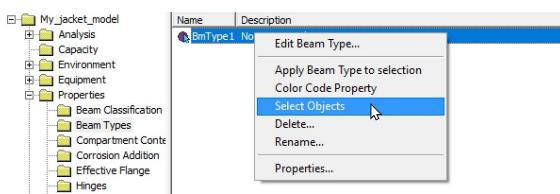


Hint: When doing a shell fatigue analysis, the total analysis time can be reduced significantly by only exporting the results for the focused part Jt2RefinementZones. Further, making the mesh first for the same will be of benefit when doing the fatigue analysis.

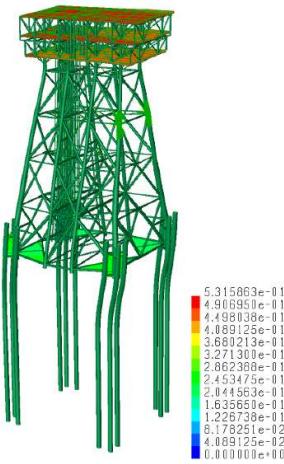
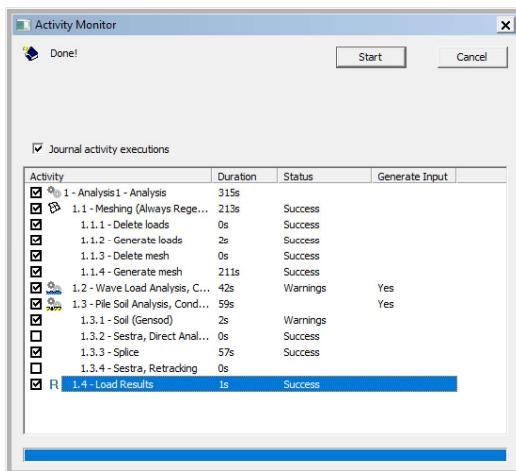


(7) make a new set where all structure is included except for the non-structural members. This will make it more realistic to view deformations.

- Open the view "Modelling Structure" and make sure all members are part of the view (ALT+A)
- Go to the browser and open Properties -> Beam Types
- Right click on BmType1 Nonstructural and click on Select Objects followed by ALT +- (ALT + "Minus"). This will remove the non-structural beams from the view
- Select all members in the graphical view, RMB -> Named set and make a new set called "View_model"



(8) make the mesh of Jt2 and run analysis by ALT+D. **(9)** check the deformations and select to show result for the set "View_model" only. Max deformation is 0.532 m which is almost identical when only Jt1 was converted to shells.



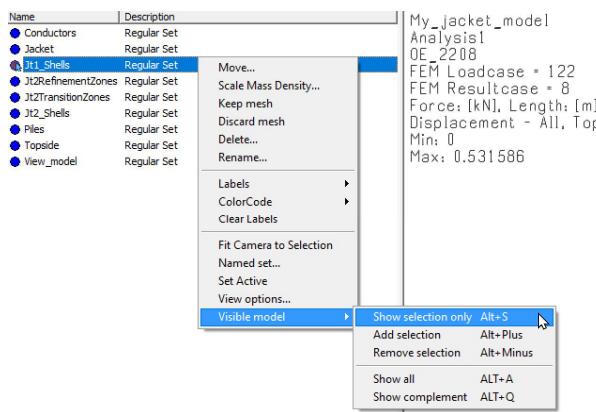
5 LOOK AT RESULTS FOR JT1 AND JT2

This chapter shows some ways of looking at the stresses from analysis. GeniE can present elementwise results while Xtract is used for more refined result evaluation.

5.1 Result viewing in GeniE

Make sure that you have selected the view "Results - with mesh". The loadcase OE2208 is the default from the model, but you can change this to one of OE2201 – OE 2208 (These were the only ones exported from the analysis run).

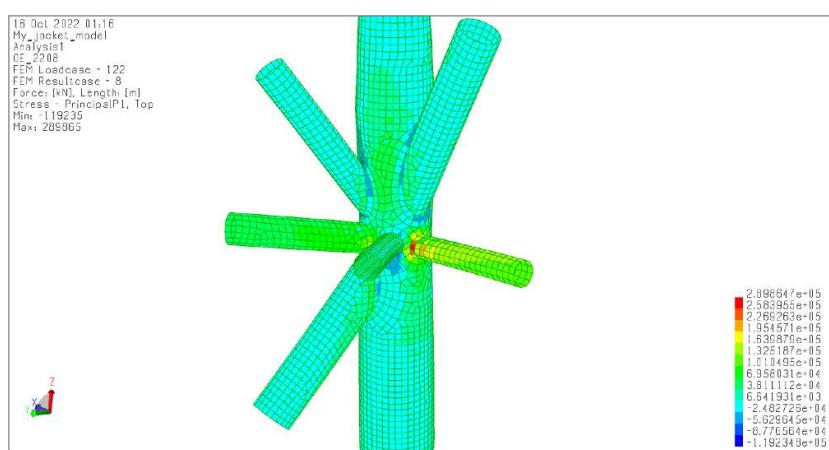
(1) from the browser "Utilities" -> "Sets" -> "Regular sets" you right click on Jt1_Shells to view this set only. You can click on the tool button "Fit" to better view the set



(2) from the result property menu, you select principal stresses P1 (or something else if you want).

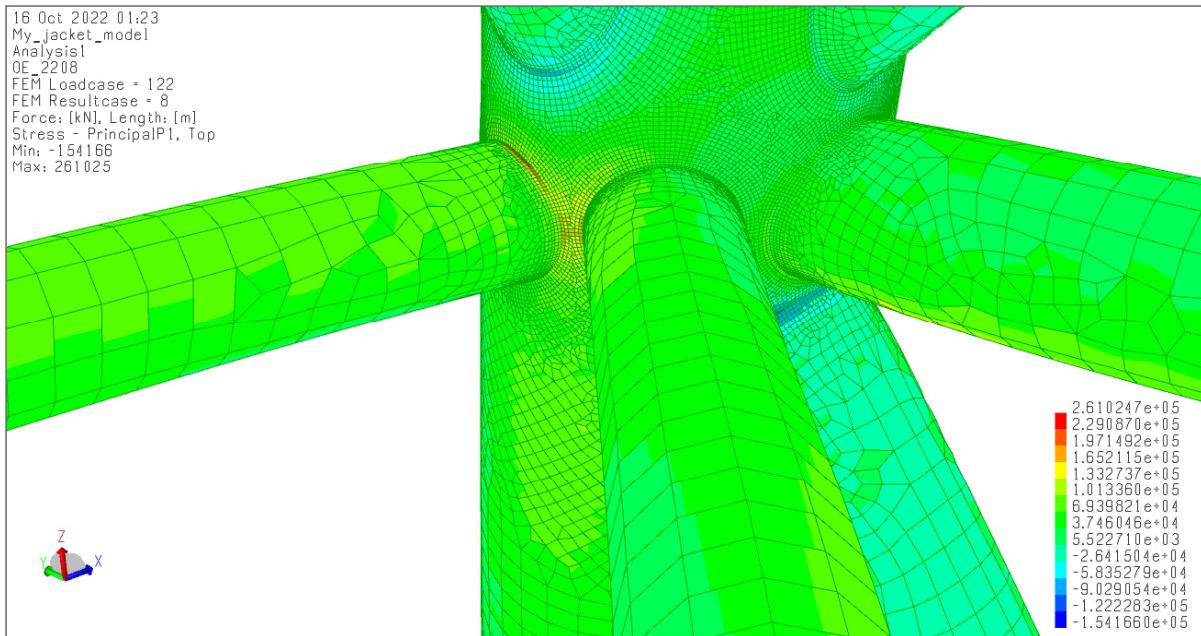


(3) you can now look at the stresses graphically, rotate and zoom in etc.



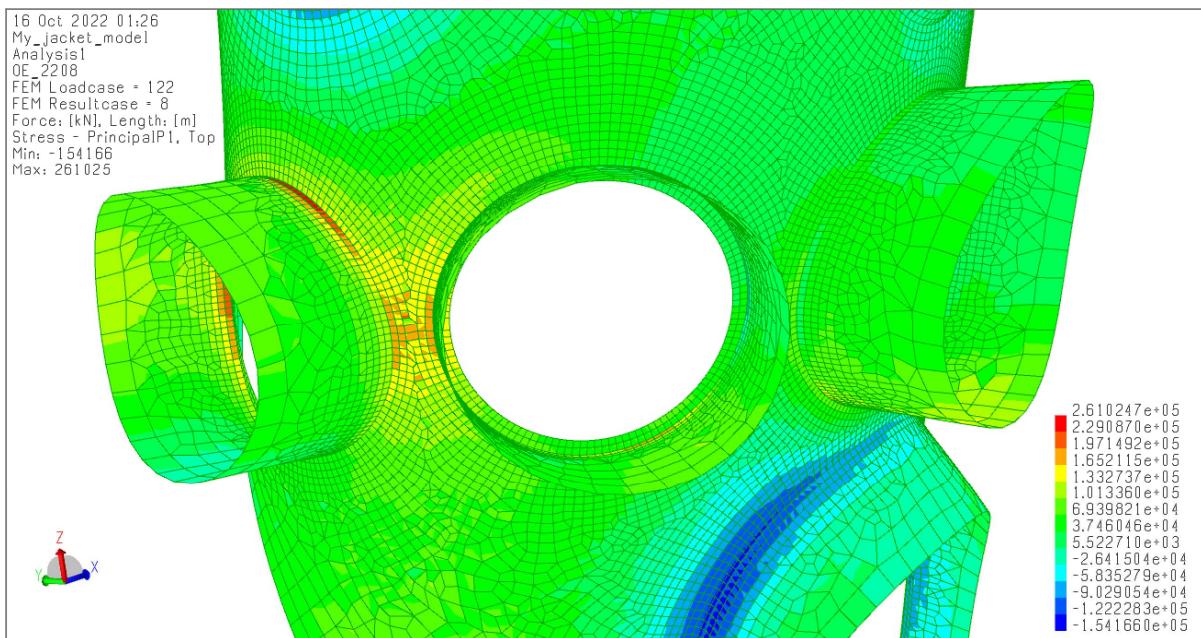


(4) select Jt2_Shells and show same results



(5) or drill down by selecting Jt2RefinementZones and Jt2TransitionZones

Name	Description
Conductors	Regular Set
Jacket	Regular Set
Jt1_Shells	Regular Set
Jt2_RefinementZones	Regular Set
Jt2_TransitionZones	Regular Set
Jt2_Shells	Regular Set
Piles	Regular Set
Topside	Regular Set
View_model	Regular Set

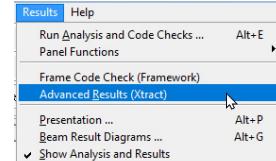




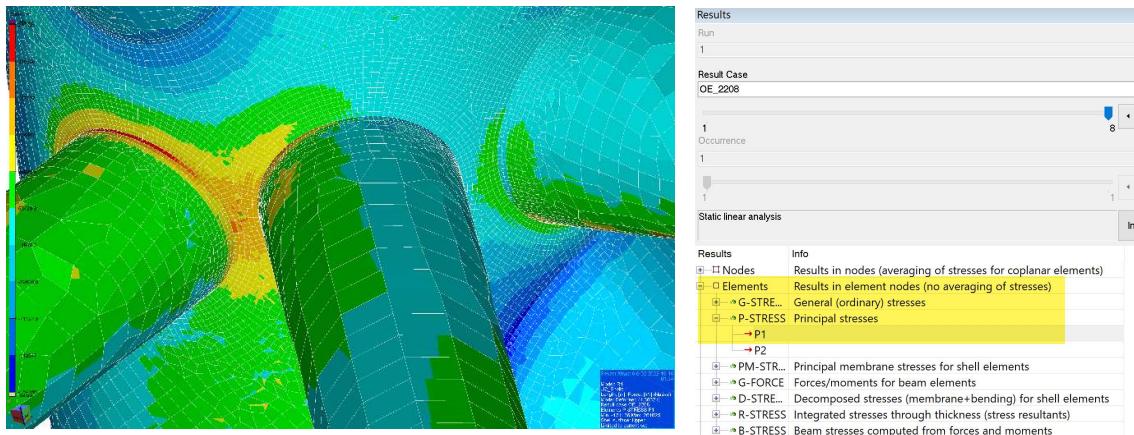
5.2 Result viewing in Xtract

Xtract will show more than elementwise results like node results.

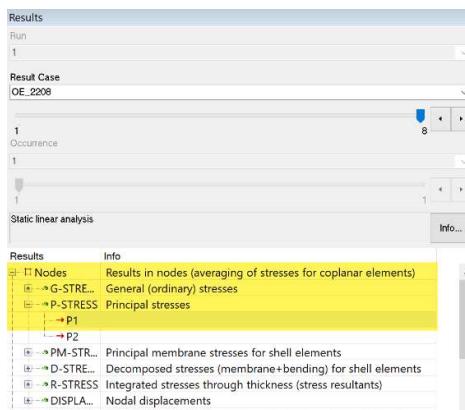
(1) from the pulldown menu "Results" you can start Xtract.

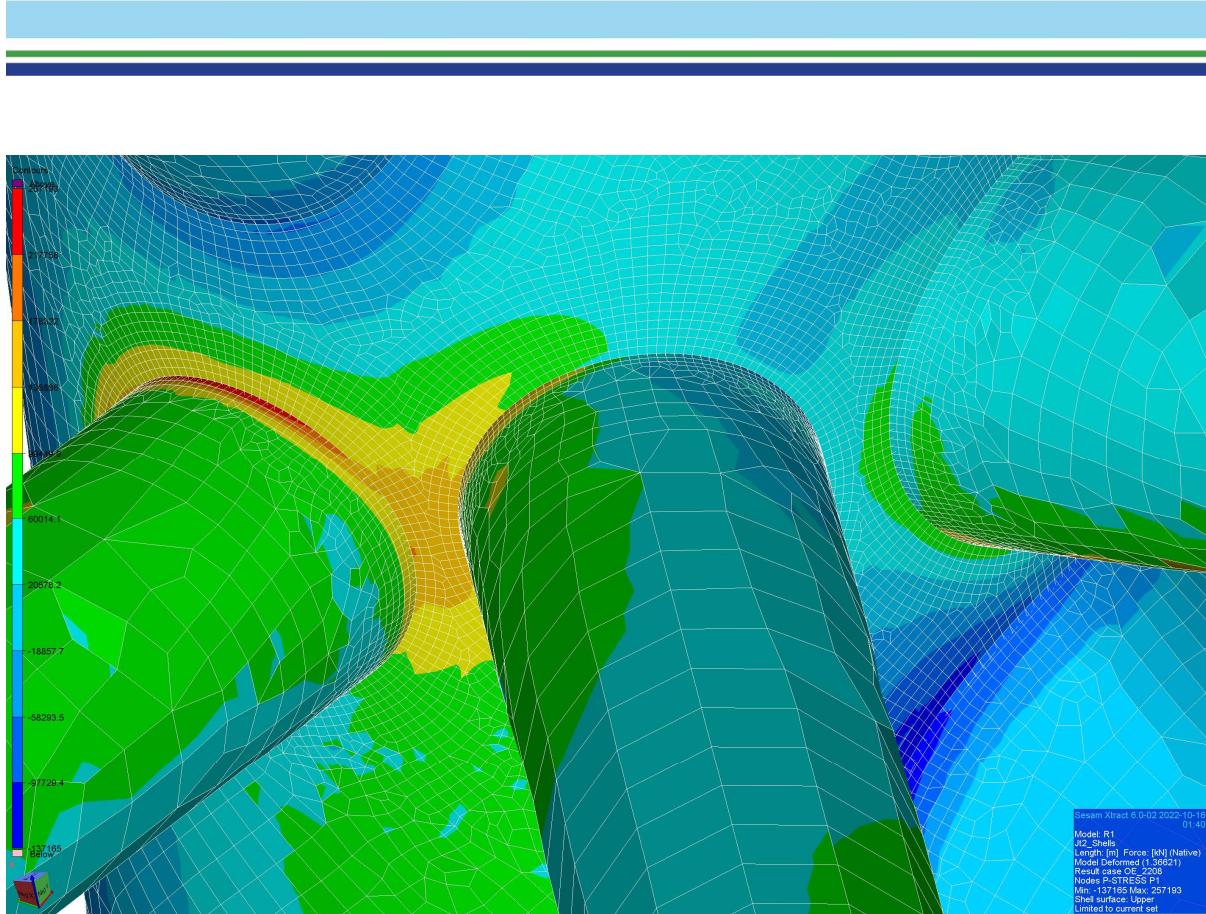


(2) from the Xtract user interface you can now select which parts, load-cases and result attributes you want to look at. In the example below Jt2_Shells and loadcase OE_2208 have been chosen. Two examples are shown, elementwise and nodal principal stresses (P1). The results for the elementwise stresses are the same as for GeniE



The picture on the next page shows nodal stresses



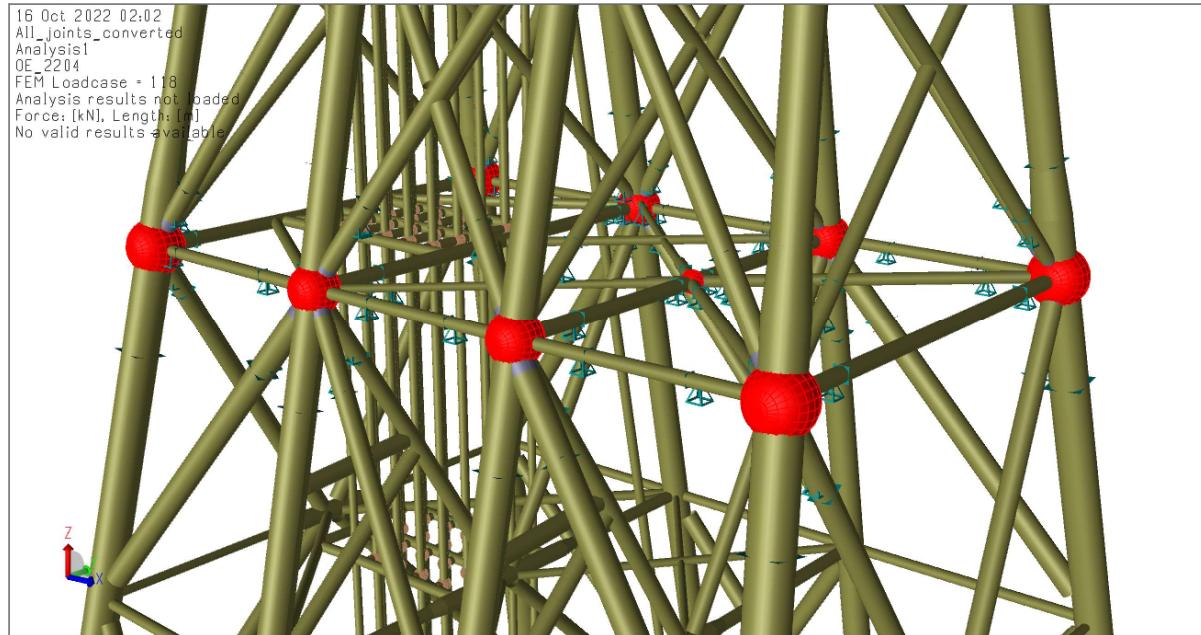


6 IMPORT MODEL WITH ALL JOINTS IN ELEVATION CONVERTED

The purpose of this step is to show the feasibility of converting more than one joint and to run analysis.

Make a new workspace in GeniE and import the file "Tubular_joint_completed.gnx". After import run a full analysis by ALT+D to generate results. All joints except Jt2 have mesh density 200mm and no refinement zones. Jt2 has refinement 2 layers each of 20mm for all braces.

The model in GeniE:



Sample results from Xtract:

