WHEN TRUST MATTERS

Sesam^{TM} – Shell fatigue analysis of tubular joints

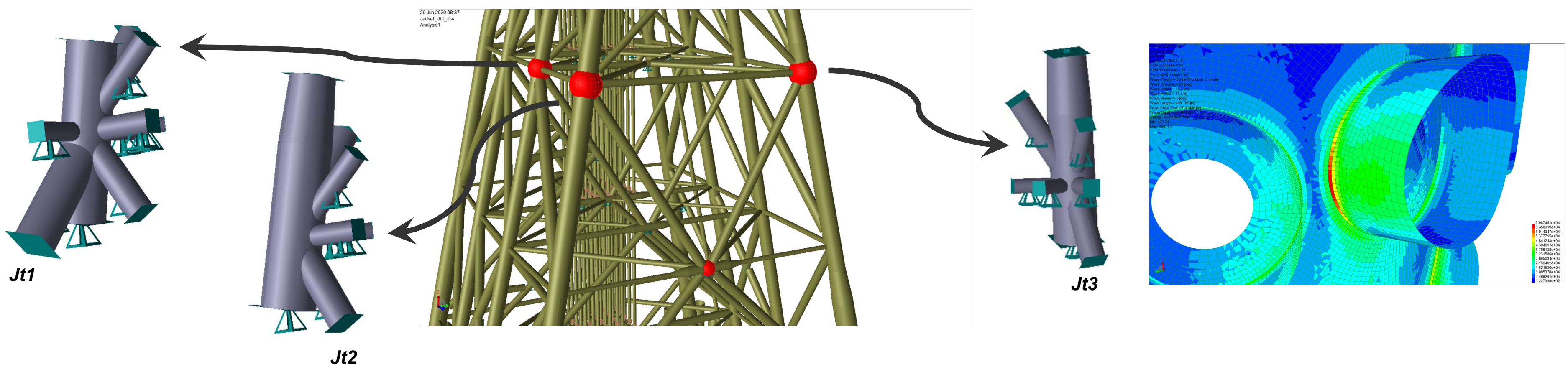
How to convert tubular joints to shell models for use in shell fatigue analysis

20 April 2021

This presentation focuses the conversion of tubular joints and creation of mesh satisfying the requirements of DNV RP-C203. The main goal of recent Sesam developments has been simplification, minimum of user input, performance in dynamic analysis and the subsequent shell fatigue analysis

Content

- Background
- How to convert different tubular joint types – with and without overlapping braces
  - How to make a quadratic regular mesh for use in shell fatigue and when calculating SCF’s
  - Using a one click command to convert and automatically make a mesh satisfying DNV RP-C203
- Case study on fatigue calculations of joint Jt3
  - Beam fatigue using empirical SCF
  - Beam fatigue using calculated SCF
  - Shell fatigue



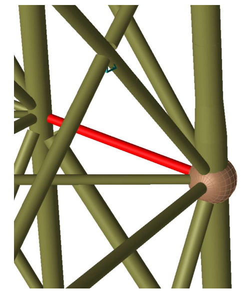
Background and requirements when converting a beam tubular joint to a shell model for use in shell fatigue

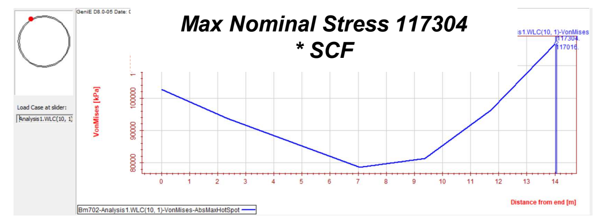
Re-analysis of an asset after many years in service may show that fatigue life has been passed or is close to end of life. This could be a showstopper for requalification as it may be too expensive to strengthen the joints. Fatigue analysis using shell elements may give less conservative fatigue life

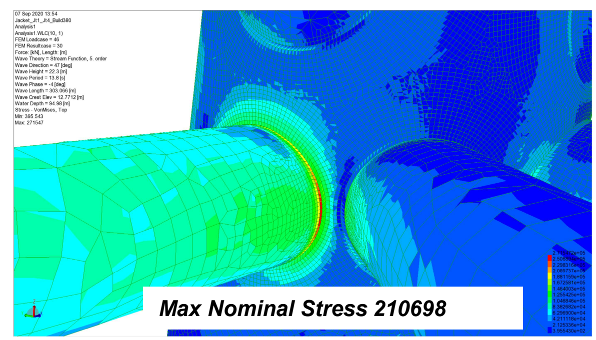
Fatigue of tubular joints

- Lately the OWT and O&G industry has considered shell fatigue analysis in order to demonstrate improved fatigue life of new and ageing assets based on DNV RP-C203 or similar. DNV has been asked to help the industry become more efficient in doing such analyses
- Fatigue analysis methods
  - Beam models and parametric empirical SCF
    - This has been the industry standard for many years
  - Beam models with user defined SCF
    - SCF derived from FE models may give more realistic SCF
  - Integrated shell models and using stochastic shell fatigue
    - Stiffness and loads integrated, better prediction of dynamic stresses
    - Allows to include local stiffeners to better represent local stiffness
  - Shell fatigue by use of local sub-models
    - Separate analyses of joints would allow bigger models and may reduce computation time
    - Difficult to automate the modelling process
- The industry needs to select which method to use – Sesam supports all

This is the focus of this presentation



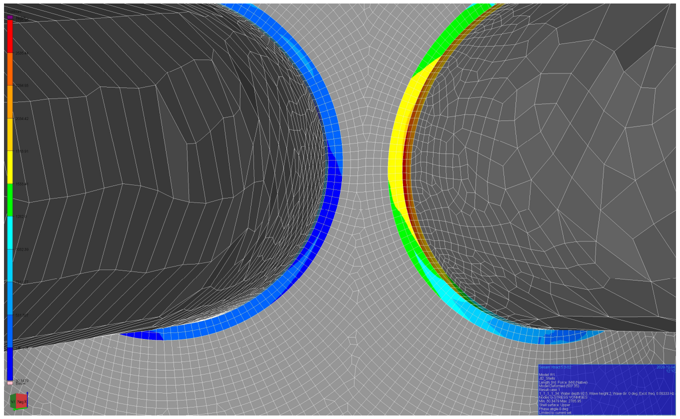




It is much more than just saving time in dynamic analysis. Much more can be saved with smartness in conversion of the tubular joints and automatically making a quality mesh

Background

- Sesam has always supported tubular joints modelled with shells and integrating the shell model with the jacket (stiffness and load transfer)
  - A manual definition of the shell model had to be done. This was partly improved by using scripts doing parts of the conversion
- The GeniE 7.13 release helped a lot related to the conversion of a “beam” joint to a “shell” joint and many expressed that this was a huge time saver
  - The beams are automatically converted to shell and connected to structure & automatic trim of the chord (i.e. chord is clean inside)
- The GeniE 8.0 release focused better auto created mesh & non-structural members
  - Auto creation of curves used for splitting structure
  - Easy definition of feature edges to control the quality of auto generated mesh
- The GeniE 8.1 release can automatically make a quality mesh for fatigue around the connection between the chord and brace
  - There is a refined mesh zone where user decides number of element layers and mesh size
  - There is a transition part between refined and coarse mesh to better satisfy DNV RP-C203
  - The through brace is trimmed when there are overlapping braces
  - Fatigue analysis can be carried out on the refined mesh zone to significantly save analysis time

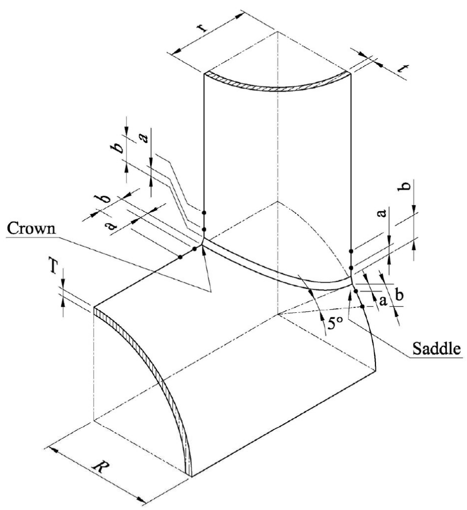


DNV

The industry is using both Chapters 4.2 and 4.3 for fatigue of tubular joints

DNV GL RP-C203 – shell fatigue of tubular joints

- Chapter 4.2 is made for tubular joints
  - One to two layers of 1st order radial square mesh density = 0.1 * SQRT(radius brace * thickness brace). Mesh outside the layer to have sufficient good quality. **This method is made for tubular joints**
  - **The stresses can be used as is in fatigue calculations**



This option is used in this presentation

Figure 4-1 Points for read out of stresses for derivation of hot spot stress in tubular joints

An alternative analysis approach is to use the stresses at the Gaussian points if these are placed $0.1\sqrt{rt}$ from the weld toe (r = radius of considered tubular and t = thickness). The stress at this point may be used directly in the fatigue assessment.

- Chapter 4.3 is made for flat plates
  - Two layers of 1st order square quad mesh with mesh density same as thickness. Mesh outside the layer to have sufficient good quality. **This method is made for flat plated structures**. It contains two methods for extracting hot spot stresses, "Method A" and "Method "B"
  - **Both options require that effective hot spot stresses are derived**

Method A

For modelling with shell elements without any weld included in the model a linear extrapolation of the stresses to the intersection line from the read out points at 0.5t and 1.5t from the intersection line can be performed to derive hot spot stress.

Method B

For modelling with shell elements without any weld included in the model the hot spot stress is taken as the stress at the read out point 0.5t away from the intersection line.

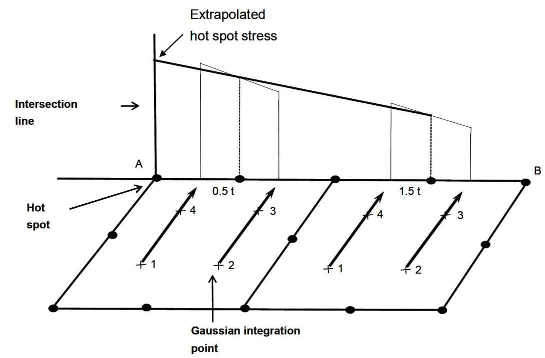
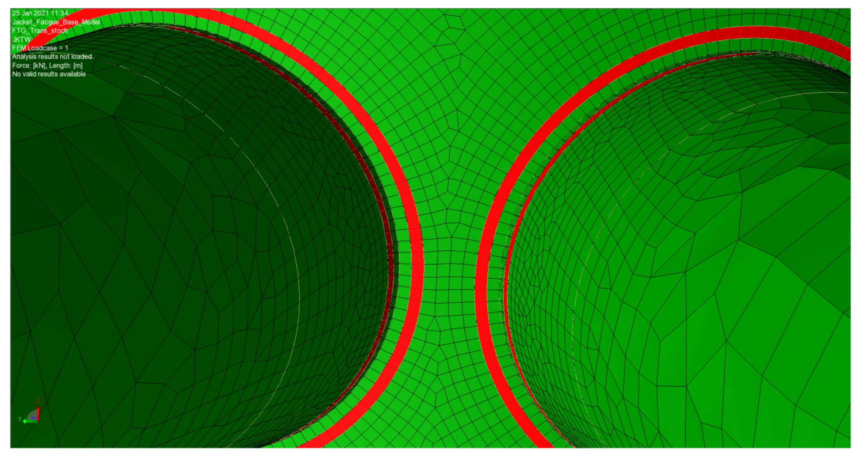


Figure 4-3 Example of derivation of hot spot stress

Mesh requirements - it all depends on which fatigue method

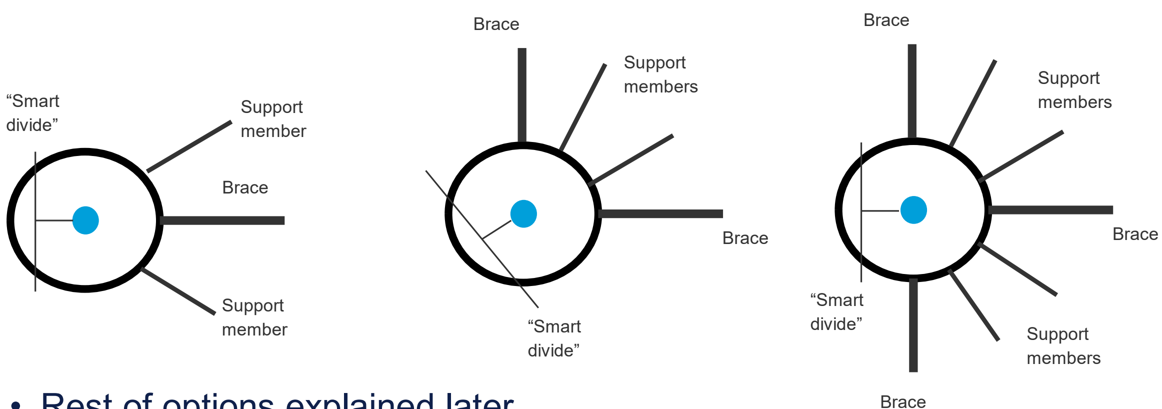
- It all depends on the ratio between the gap and thicknesses
  - Chapter 4.2: There must be at least 3 finite elements on the chord between the braces. One layer of 1st order radial square mesh with density = 0.1 * SQRT(radius brace * thickness brace). Mesh outside the layer to have sufficient good quality.
  - Chapter 4.3 Method A: There must be at least 5 finite elements over the gap width: Two layers of 1st order square quad mesh with mesh density same as brace thickness. Mesh outside the layer to have sufficient good quality
  - Chapter 4.3 Method B: There must be at least 3 finite elements on the chord between the braces: One layer of 1st order radial square mesh with density = 0.5 * brace thickness. Mesh outside the layer to have sufficient good quality
- Normal size of brace thickness often 15 - 40 mm and gap around 75 mm (3 in); gap/brace range 1 - 3
- Recommendation
  - Gap/brace thickness < 5: One layer of mesh around each brace is doable. Supports Chapter 4.2 and Chapter 4.3 Method B
  - Gap/brace thickness > 5: Two layers of mesh around each brace is doable. Supports Chapter 4.2 and Chapter 4.3 Method A & B
  - When overlap between braces: Not possible to automatically make a mesh layer. Mesh layers can be made for braces with gap in between
  - ***Stofat uses all nodes of an element during fatigue calculation. One should aim for at least two layers and use layer number 2 when assessing the fatigue results. Then one will easily find the fatigue life at the positions specified by DNV RP C203 Chapter 4.2***



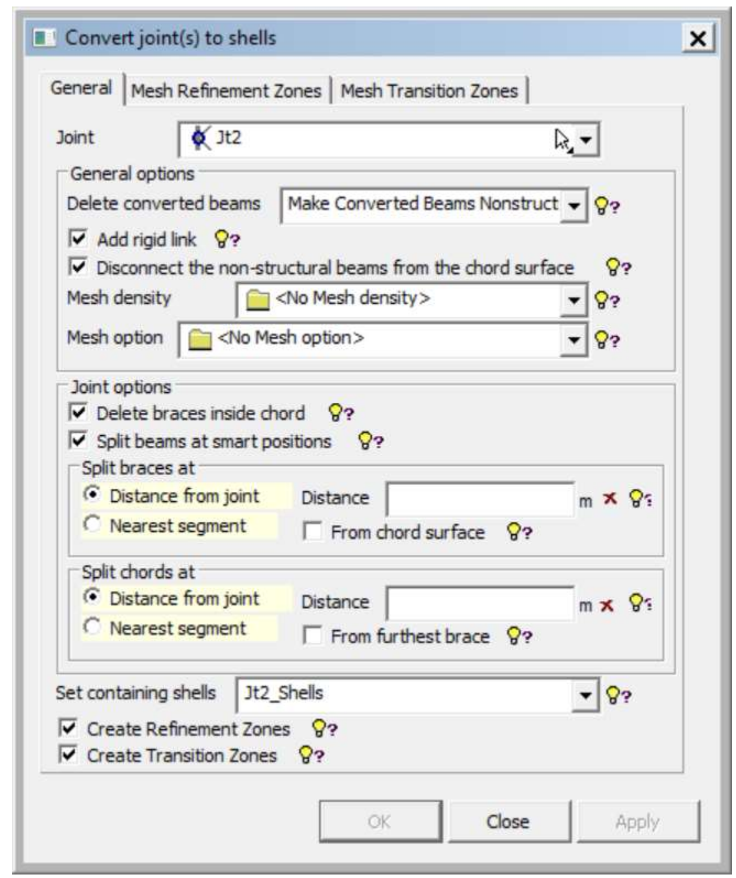
It is also possible to do all manually by use of other commands in GeniE – this requires more editing and time, but still fast!

The command “Convert joint(s) to shells”

- General options, program defaults
  - Delete the converted beams and make them non-structural
  - Connect the converted beams to structure by use of rigid links (aka. master-slave)
  - Disconnect the non-structural beams from the chord surface (in the plugs)
- Joint options, program defaults
  - Trim the inside surface of the chord (delete braces inside chord)
  - “Smart divide” of chord and braces **not influencing the mesh quality**



- Rest of options explained later



Fatigue analysis is based on the dynamic loads – not the static loads. It is therefore assumed satisfactory to include hydrodynamic loads from the beams and not the shell model. It is, however important to represent the converted shell model with non-structural members to capture the wave loads

Handling dynamic wave loads

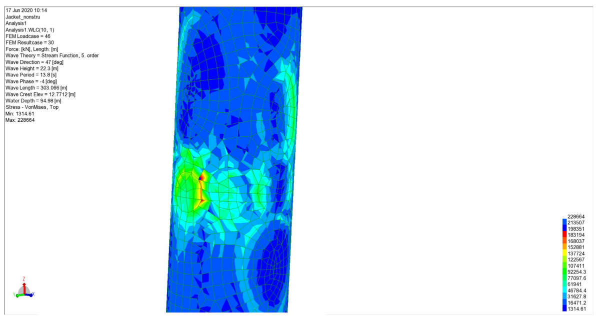
- The wave loads of the converted joint is calculated by use of non-structural members. They are connected to the converted joint at the ends of the chord and braces
  - The use of non-structural members ensures same base shear and overturning moment
  - The non-structural members are disconnected from the chord plugs

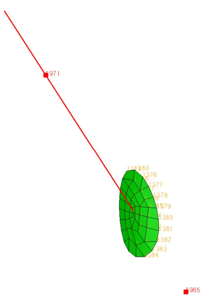
Results from wave load analysis	Beams only	With shell model & non-structural	Difference (%)	With shell model & no non-structural	Difference (%)
Max base shear sea state 16	6.6852E04	6.6797E04	0.08 %	6.5808E04	1.56 %
Max overturning moment sea state 16	4.3844E06	4.3799E06	0.10 %	4.3022E06	1.87 %
Total dry weight	1.1723E05	1.1778E05	-0.47 %	1.1703E05	0.17 %
Vertical COG (excl. marine growth)	-7.8228E01	-7.7891E01	0.43 %	-7.8233E01	-0.01 %
Vertical COG (incl. marine growth)	-7.4278E01	-7.4000E01	0.37 %	-7.4386E01	-0.15 %
Total buoyancy	9.1248E04	9.1232E04	0.02 %	9.0810E04	0.48 %
Vertical COB (excl. marine growth)	-5.9028E01	-5.9032E01	-0.01 %	-5.9187E01	-0.27 %
Vertical COB (incl. marine growth)	-5.7098E01	-5.7107E01	-0.02 %	-5.7307E01	-0.37 %

You may also print out a list to document that all non-structural members have been disconnected from the chord plugs

Disconnecting non-structural members from the chord plugs

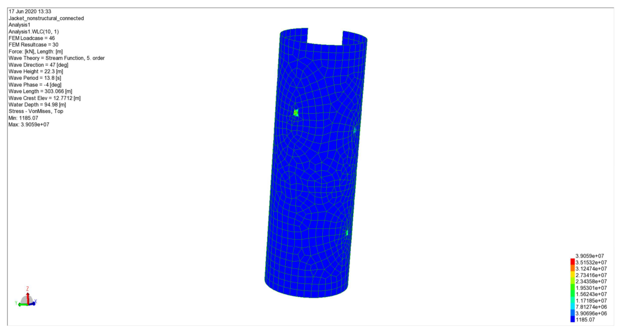
- Non-structural members disconnected from chord plugs
  - Max 228664

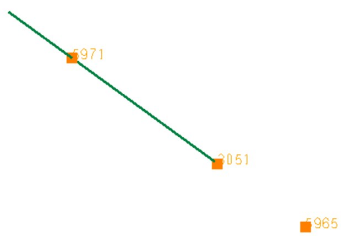




This is the default behaviour when converting a tubular joint

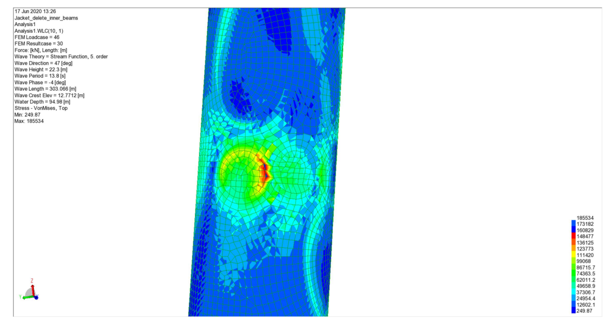
- Non-structural members connected to chord plugs
  - Max 39059000





Extreme peaks in chord plug when non-structural not disconnected (for this case)

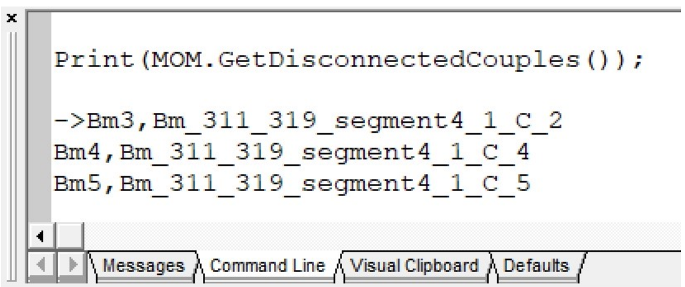
- No non-structural members
  - Max 185534



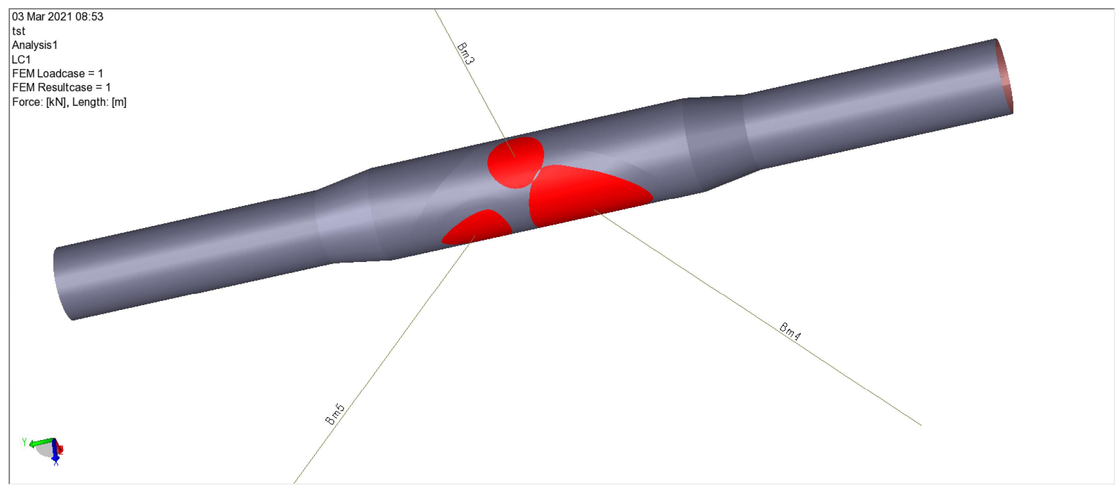
VonMises chord stress 19% under estimated for hydrodynamic loads (for this case)

Disconnecting non-structural members from chord plugs

- If you want to check the disconnection between non-structural members and the chord surface
  1. Type in the command line window:
     MOM = ManualOverlapManager();
     MOM.SetActive();
  2. Then do the conversion of the joint
  3. Type in the command line window:
     Print(MOM.GetDisconnectedCouples());



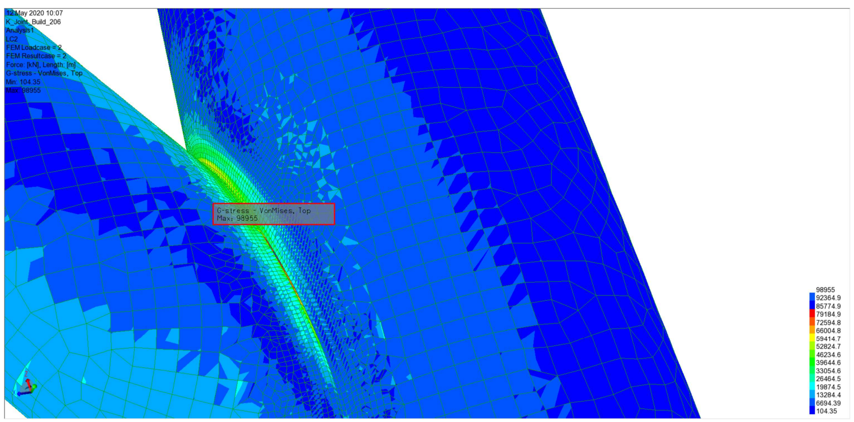
- This particular joint has 4 non-structural members
  - Bm1 (the chord)
  - Bm2, Bm4, Bm5 (the braces)



Sesam supports 1st order thick shell elements more applicable for thicknesses used in the design of jackets. Note that 2nd order thick shell elements are not supported in hydrodynamic analysis hence 1st order elements must be used

Selecting the finite element type

- With 1st order thin shell element
  - Max vonMises (top surface) = 0.98955E06
  - Max vonMises (middle surface)= 0.25962E06
  - Max vonMises (lower surface)= 0.97284E06
  - DOF = 204084
  - Mesh time 99 sec
  - Sestra analysis time 9 sec



- With 1st order thick shell element
  - Max vonMises (top surface) = 0.77709E06
  - Max vonMises (middle surface)= 0.26353E06
  - Max vonMises (lower surface)= 0.57132E06
  - DOF = 204084
  - Mesh time 102 sec
  - Sestra analysis time 9 sec

DNV RP-C203 supports the use of 1st order thick shell elements: "8-noded elements are recommended in case of steep gradients. Use of 4-noded elements with improved in-plane bending modes is a good alternative"

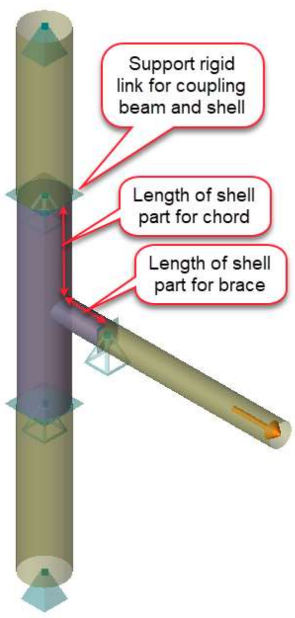
- With 2nd order thick shell element
  - Max vonMises (top surface) = 0.63289E06
  - Max vonMises (middle surface)= 0.24895E06
  - Max vonMises (lower surface)= 0.47842E06
  - DOF = 617748
  - Mesh time 108 sec
  - Sestra analysis time 36 sec

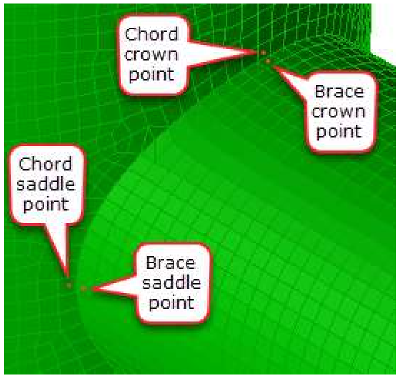
DNV

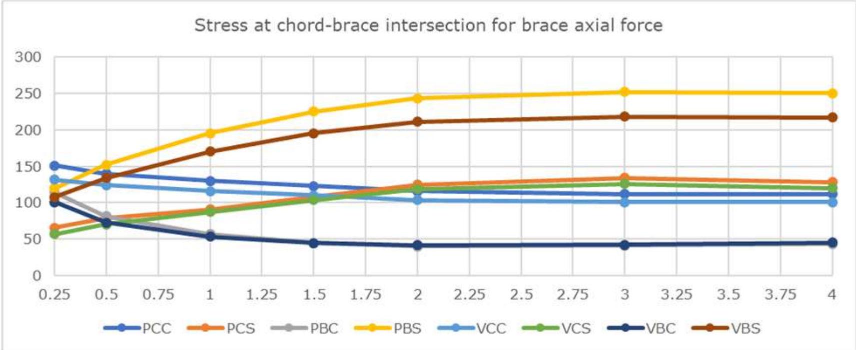
This is also the conclusion of multiple studies and public papers

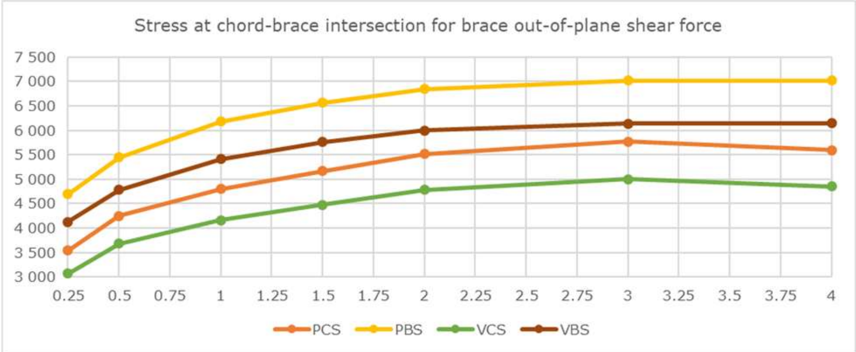
Decide where to split the chord and braces

- Our studies show that the required length of the shell part when converting a tubular joint to a shell model should be 3 times the chord and brace diameters, respectively
  - Checking axial force and in/out of plane shear forces







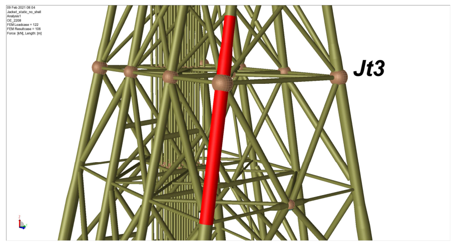


DNV

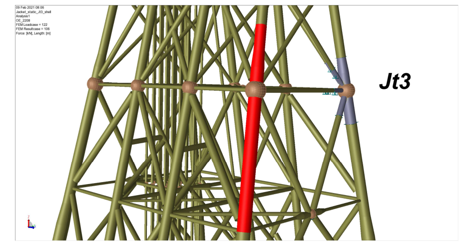
A shell model may introduce more flexibility in a joint. The force redistribution should be checked in adjacent members to ensure that the shell fatigue is not based on underestimated forces

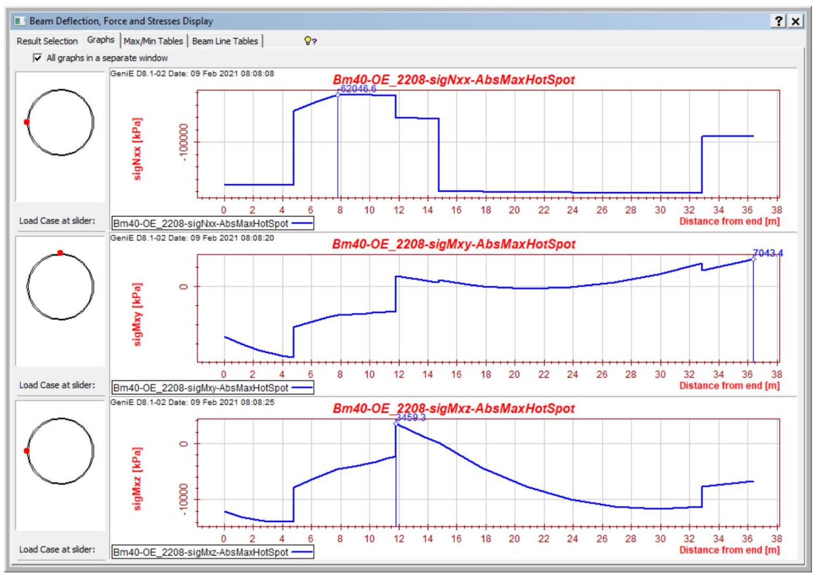
Redistribution of the forces based on stiffness and displacements

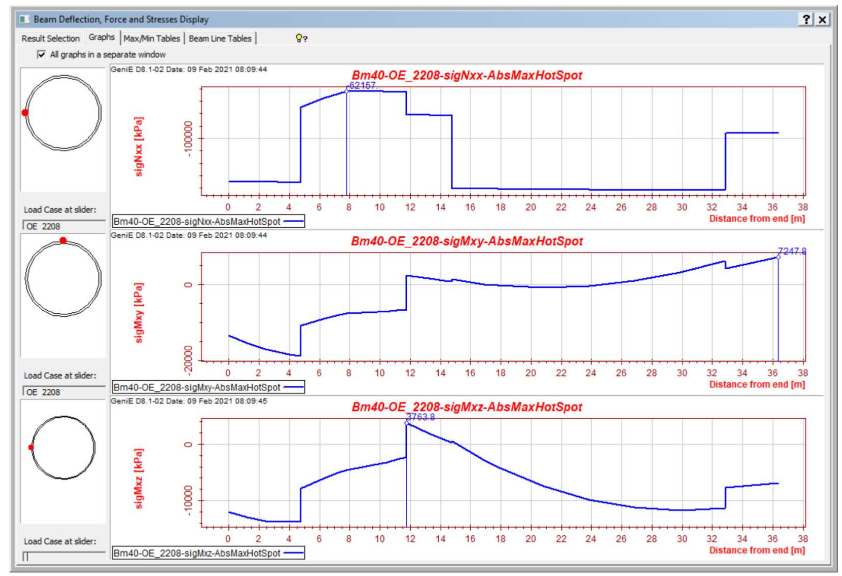
- Beams only, Bm40
  - SigNxx = 62047
  - SigMxy = 7043
  - SigMxx = 3459



- Jt3 as shell, Bm40
  - SigNxx = 62157 (0.2% diff.)
  - SigMxy = 7248 (2.8% diff.)
  - SigMxx = 3764 (8.1% diff.)





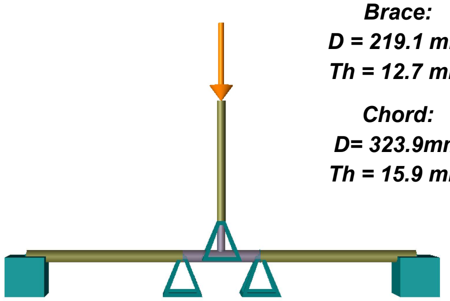


DNV

"Evaluation of SCFs for Stiffened Tubular Joints" (2015) - Anders Lysø Kleven (Master Thesis)

Comparing SCF calculations

- Comparing SCF hand calculations in Sesam with papers
  - The Kleven report is based on Abaqus, 2nd order elements and mesh density of 7.46 mm
    - DNV RP-C203 Chapter 4.2 specifies 5.1 mm mesh density. This leads to slightly better compliance with Efthymiou
- Sesam and Abaqus give similar FE results
  - Sesam, Nastran and Ansys also give similar FE results



Mesh density 7.46 mm		Sesam SCF from FE		Abaqus SCF from FE		Difference		SCF Efthymiou	Difference Sesam SCF & Efthymiou	Difference Abaqus SCF & Efthymiou
		P1 **(SCF)**	P2 **(SCF)**	P1 **(SCF)**	P2 **(SCF)**	P1	P2			
Chord crown	Axial	3.34	**11.79**	3.34	**11.82**	0%	0%	10.1	-17%	-17%
Brace crown	Axial	1.96	**4.07**	1.91	**4.08**	-3%	0%	5.2	21%	21%
Chord saddle	Axial	5.98	**6.91**	5.84	**6.78**	-2%	-2%	8.2	16%	18%
Brace saddle	Axial	5.64	**8.10**	5.96	**8.31**	5%	3%	7.0	-16%	-19%
Chord crown	In-plane	1.76	**2.65**	1.76	**2.63**	0%	-1%	2.8	7%	7%
Brace crown	In-plane	1.16	**1.96**	1.16	**1.93**	0%	-2%	2.5	22%	23%

DNV

Known limitations in automatic conversion of tubular joints

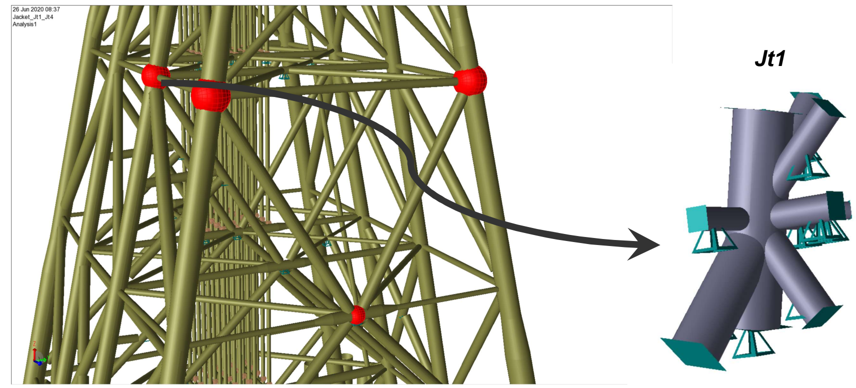
- Chords should be continuous through the joint to avoid a mesh line at this position
- Chords and braces cannot have changes in section properties unless there is a conical transition
- Grouted chord (i.e. outer leg, grout and inner pile)
  - Workaround: The inner pile inside the joint needs to be removed before conversion. For the converted shell part you need to use an equivalent thickness of the steel. The industry practice is to use SQRT($\text{Leg}_{\text{thickness}}^2$ + $\text{Pile}_{\text{thickness}}^2$). After conversion make sure that the inner pile outside the joint is connected to the corresponding outer legs
- Non-grouted legs (i.e. outer leg and inner pile connected at designated positions)
  - Workaround: The inner pile needs to be removed before conversion. After conversion the chord non-structural member needs to be made inner pile and connected by use of rigid links to the shell model at designated positions except for top/bottom of shell model
- X-joints where through brace and brace have same diameter
  - Workaround: Select individual braces, divide at designated positions, convert by use of command “Convert beam(s) to shells” prior to meshing
- Variations in hydrodynamic pressure (surface loads) on shells is not covered
  - With joints in the splash zone and large waves one should be careful as this may have an impact of the fatigue lives
- In some cases the refined mesh zones cannot be created due to numerical errors
  - Workaround: Try a slightly different width of the layers; e.g. example reduce from 30mm to 29mm

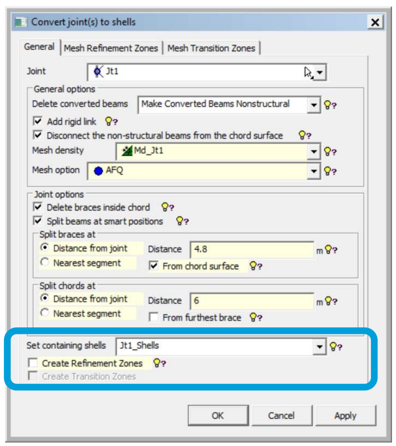
Converting different tubular joint types with one command – the choice is yours depending on type of joint and type of fatigue

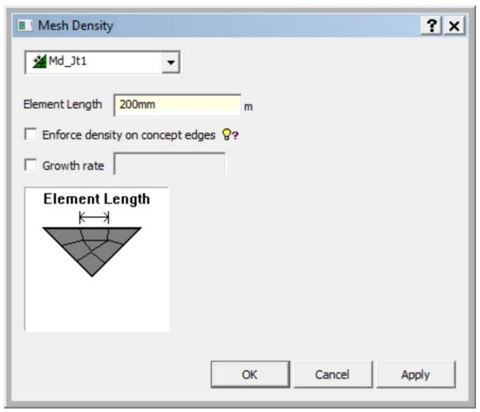
Total editing work: Less than 1 minute work

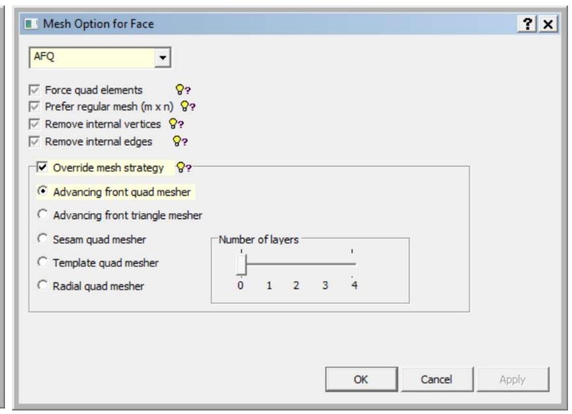
Ex. 1: Fatigue screening

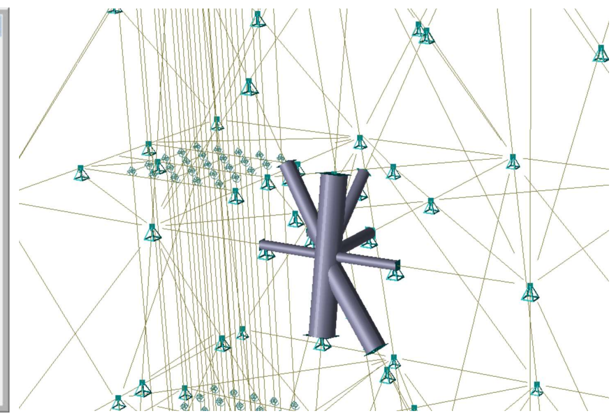
- Specify where to split chord and braces
  - Our studies and public papers show that the required length of the shell part should be 3 times the chord and brace diameters, respectively
- Specify mesh density to be used for whole joint shell model
  - No particular focus on mesh refinement except what's build into the mesh algorithms
- Specify which mesh option to be used for whole joint
  - Our testing shows that the Advancing front quad mesher gives the best mesh for tubular joints







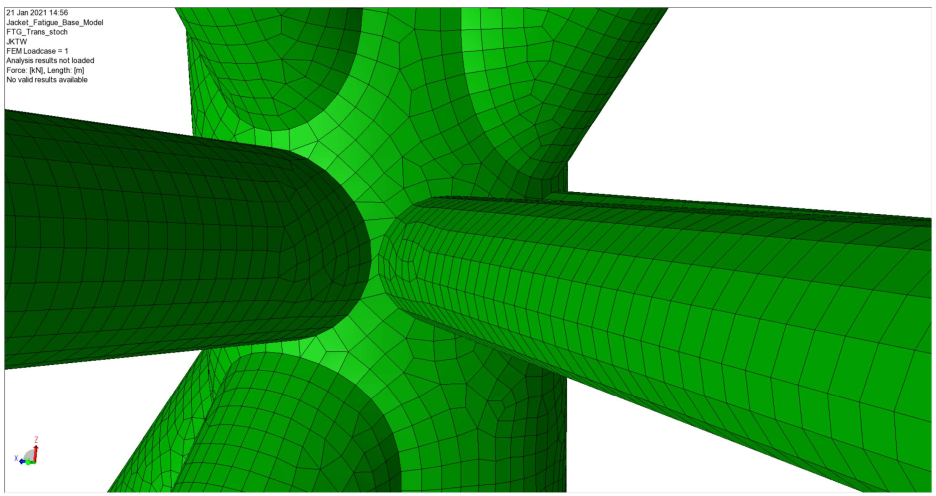




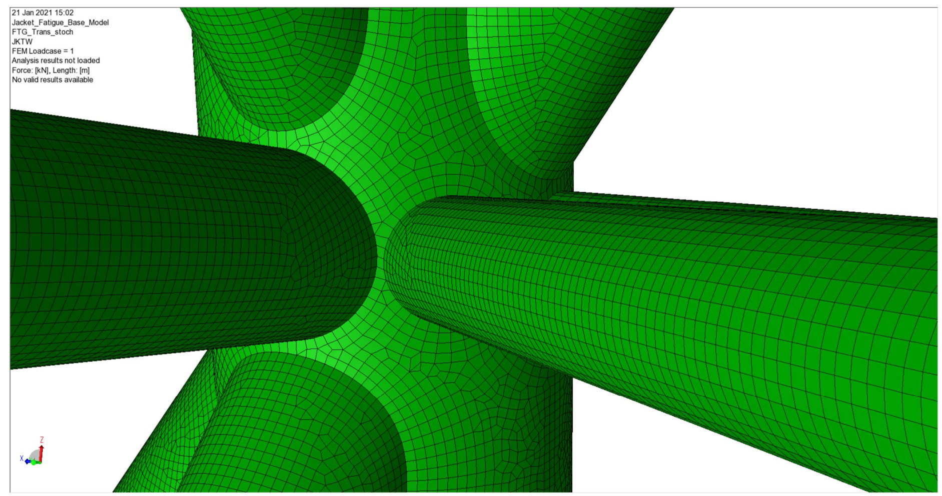
DNV

Ex. 1: Fatigue screening

- The mesh is automatically made
  - Two examples shown here with mesh density 100mm and 50mm



Mesh density 100 mm

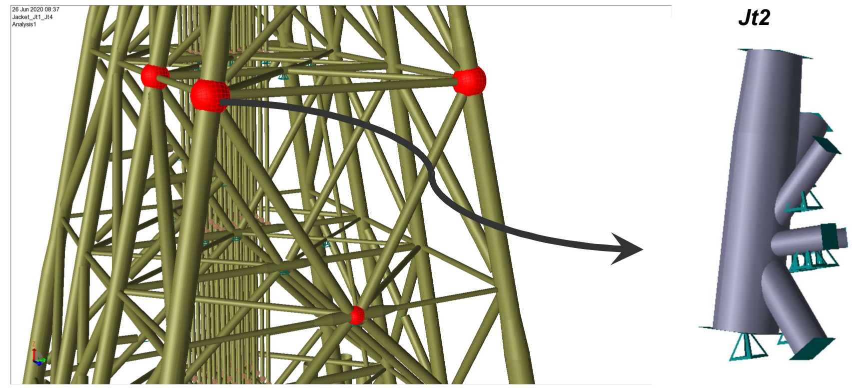


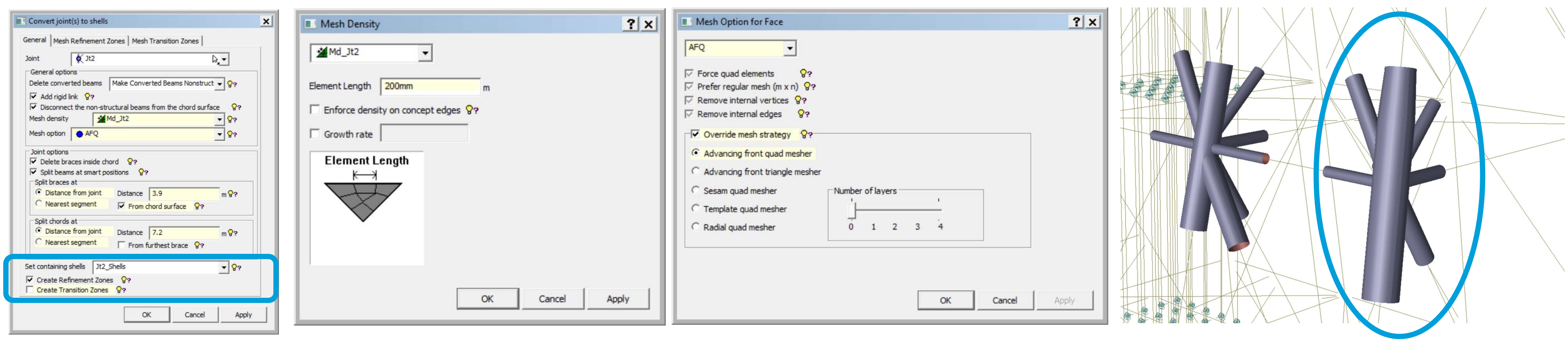
Mesh density 50 mm

Total editing work: Less than 1 minute work.
Main difference from previous model is that the mesh is refined at the brace/chord connections

Ex. 2: Fatigue of whole joint with fine mesh

- Specify where to split chord and braces
- Specify mesh density and mesh option to be used for whole joint shell model
- Make a refined mesh for the brace/chord connections
  - See next slide



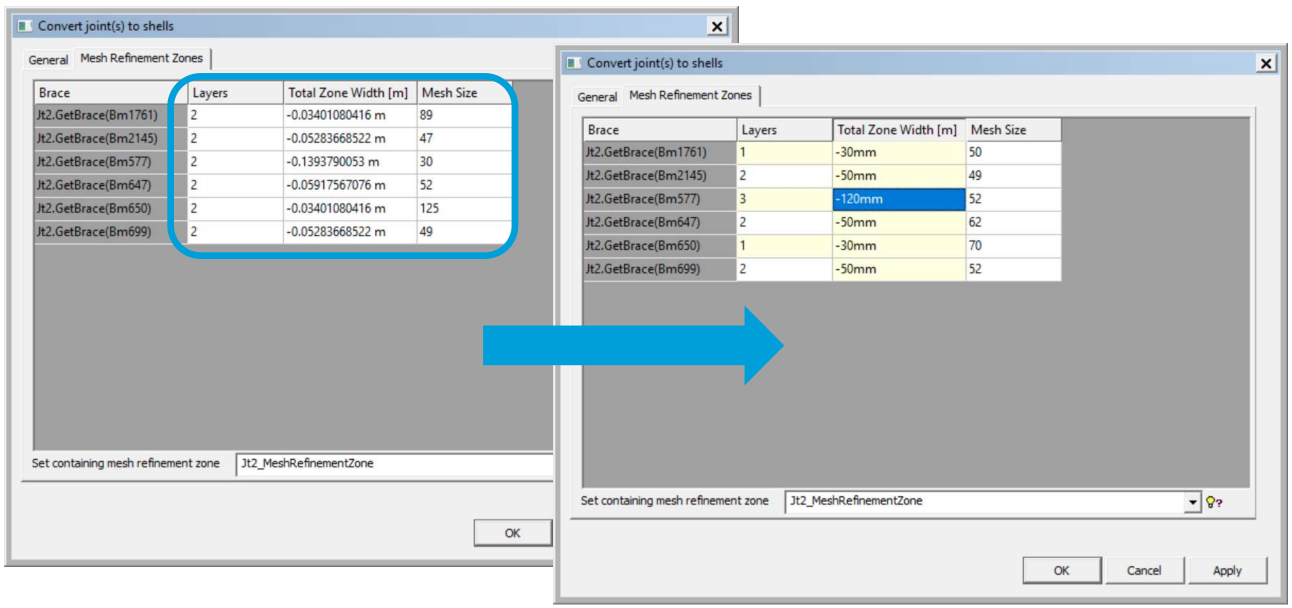


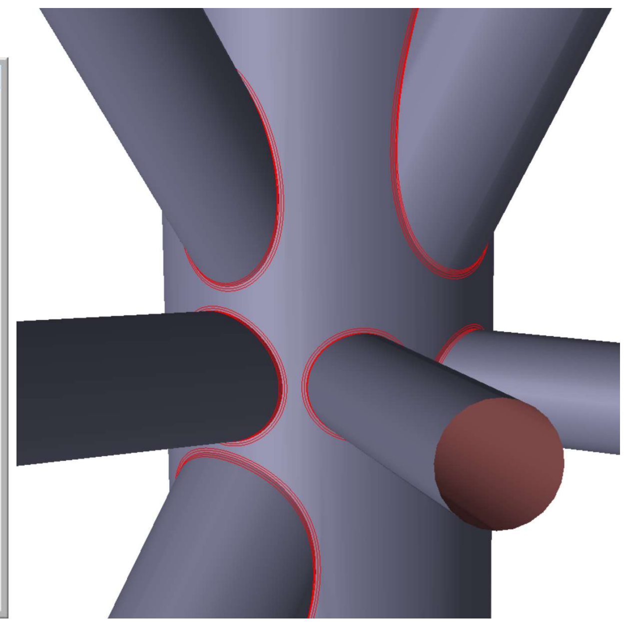
DNV

The set for mesh refinement should be used when defining a mesh priority. This will force the meshing to start on the refined part

Ex. 2: Fatigue of whole joint with fine mesh

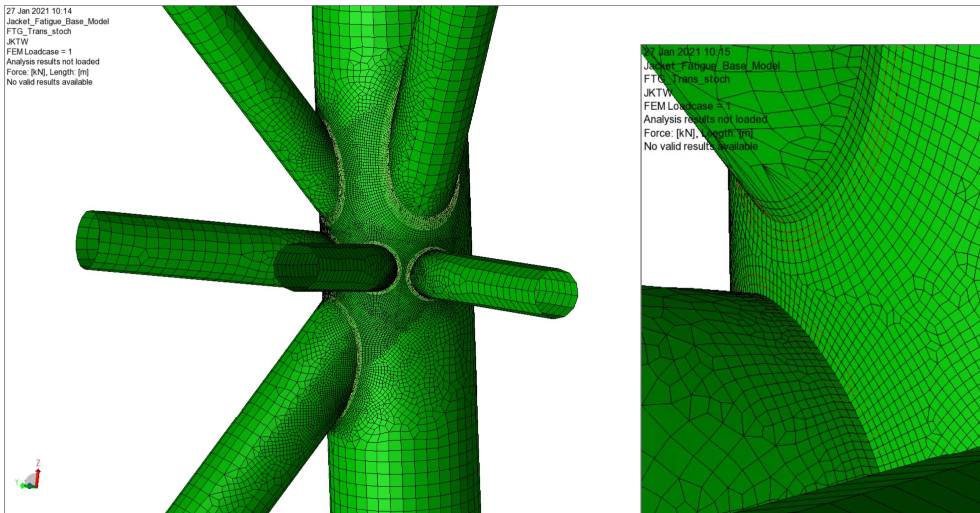
- Setting the mesh densities for the refined mesh zone
  - The mesh density will be the total width zone divided by number of layers
  - The default is 2 layers
  - GeniE reports the maximum allowable total zone width
- The below selection of layers and width is to show you flexibility
- In real work probably 1-3 layers will be used if possible
  - Depends on the gap between braces

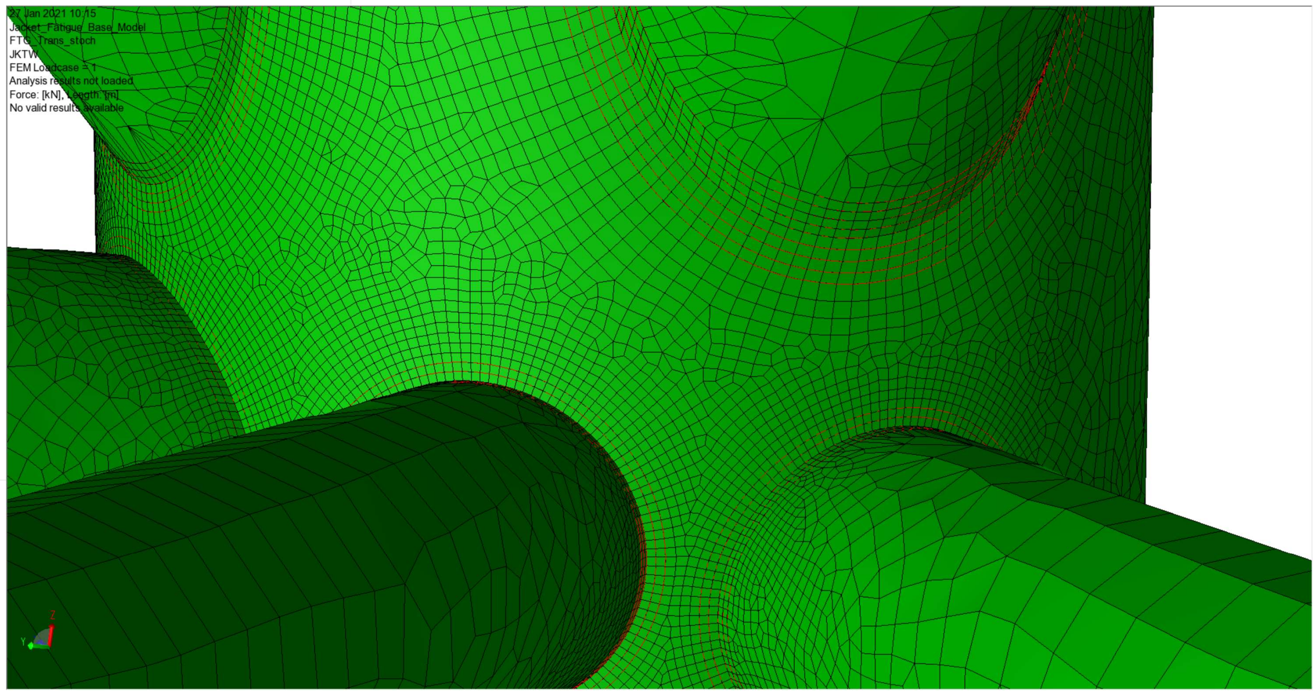




Ex. 2: Fatigue of whole joint with fine mesh

- The mesh is automatically made

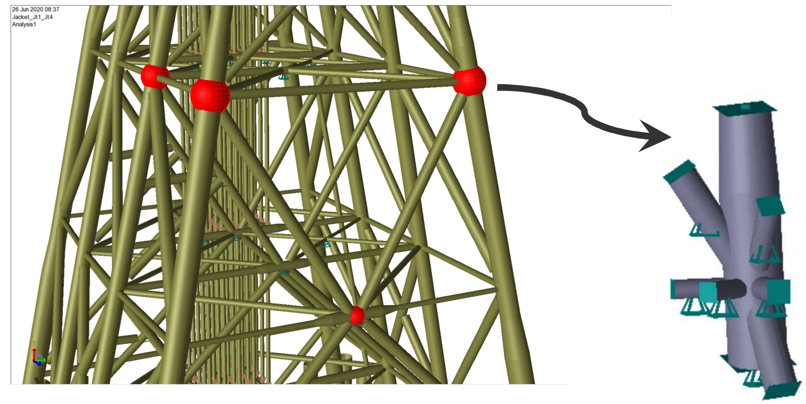


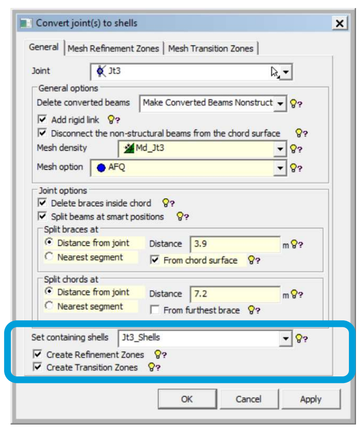


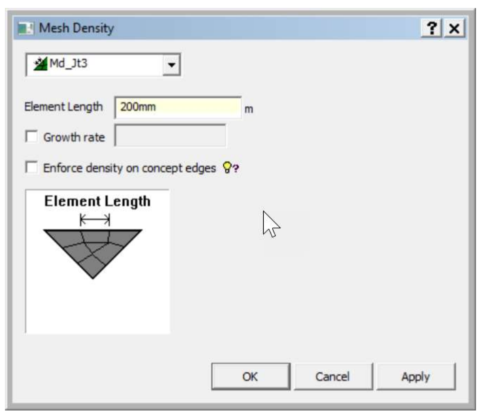
Total editing work: Less than 2 minutes work.
Main difference from previous model is that total number of finite elements is lower and better close to the refined mesh

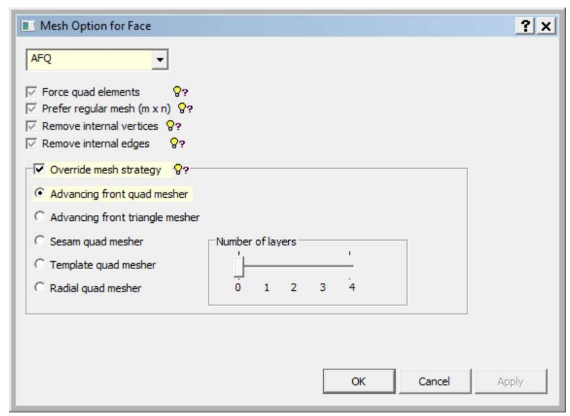
Ex. 3: Fatigue of selected parts with fine mesh

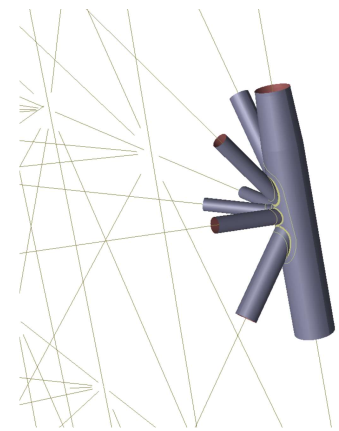
- Specify where to split chord and braces
- Specify mesh density and mesh option to be used for whole joint shell model
- Make a refined mesh for the brace/chord connections
  - See next slide
- Make a mesh transition zone
  - See next slide





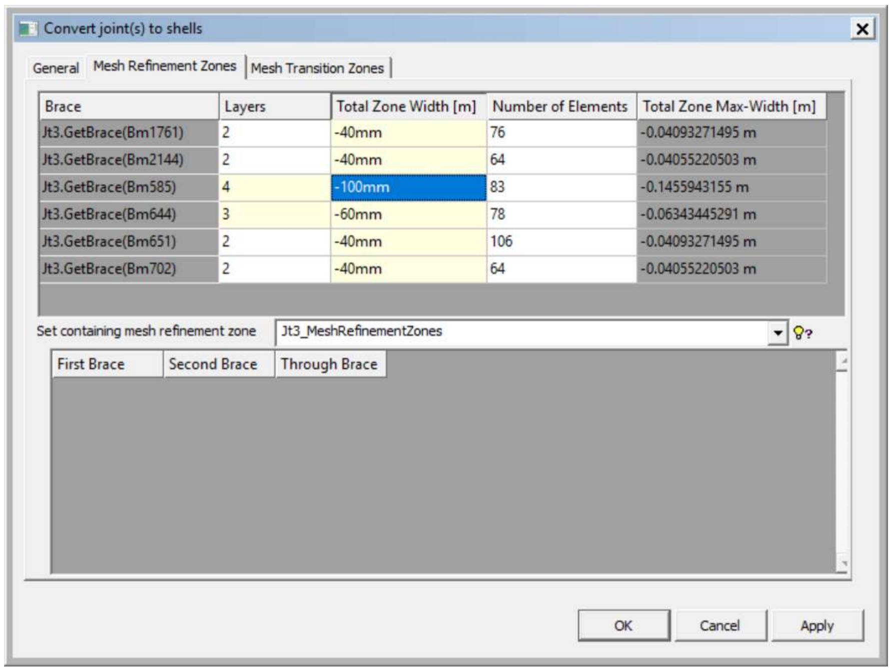


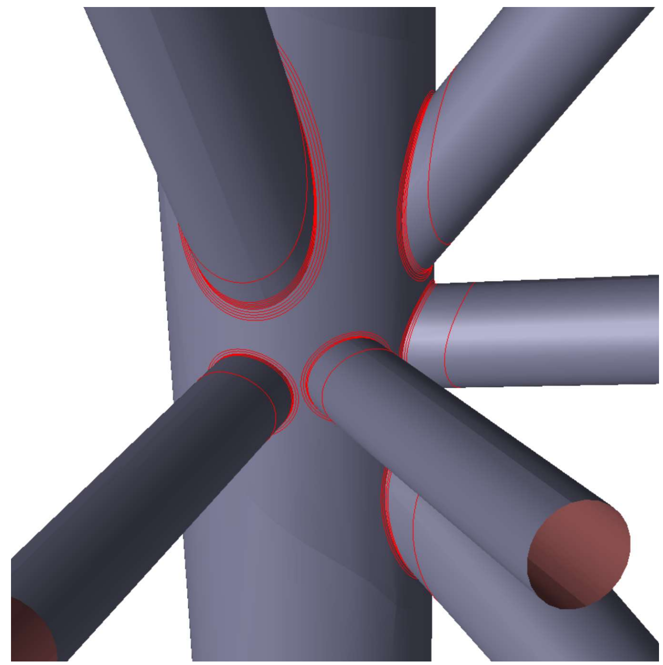




Ex. 3: Fatigue of selected parts with fine mesh

- Setting the mesh densities for the refined mesh zone
  - This will give 20mm and 25mm mesh density around all chord/brace connections

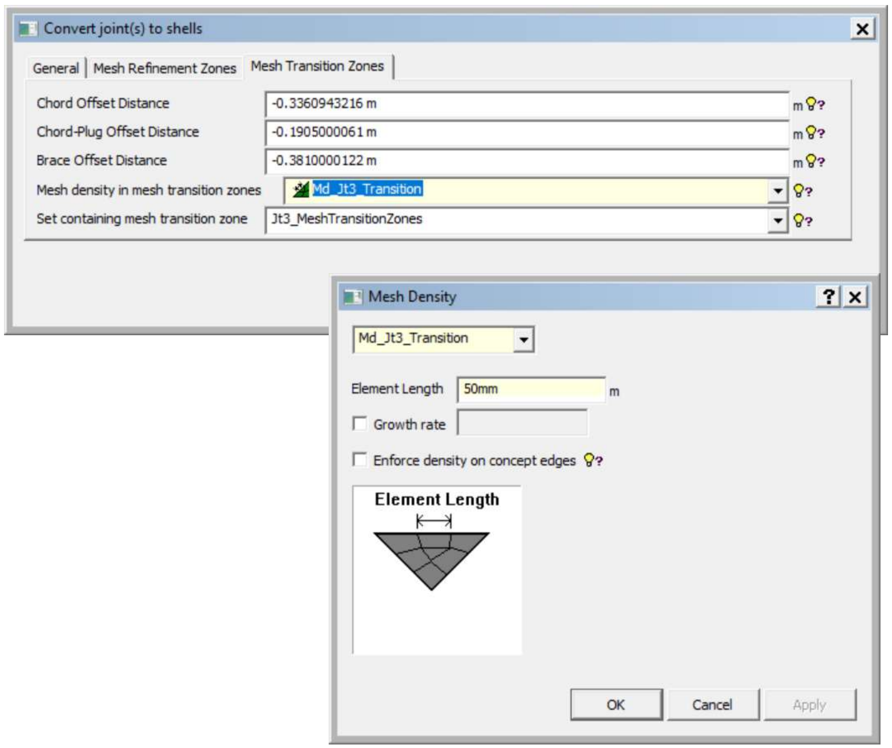


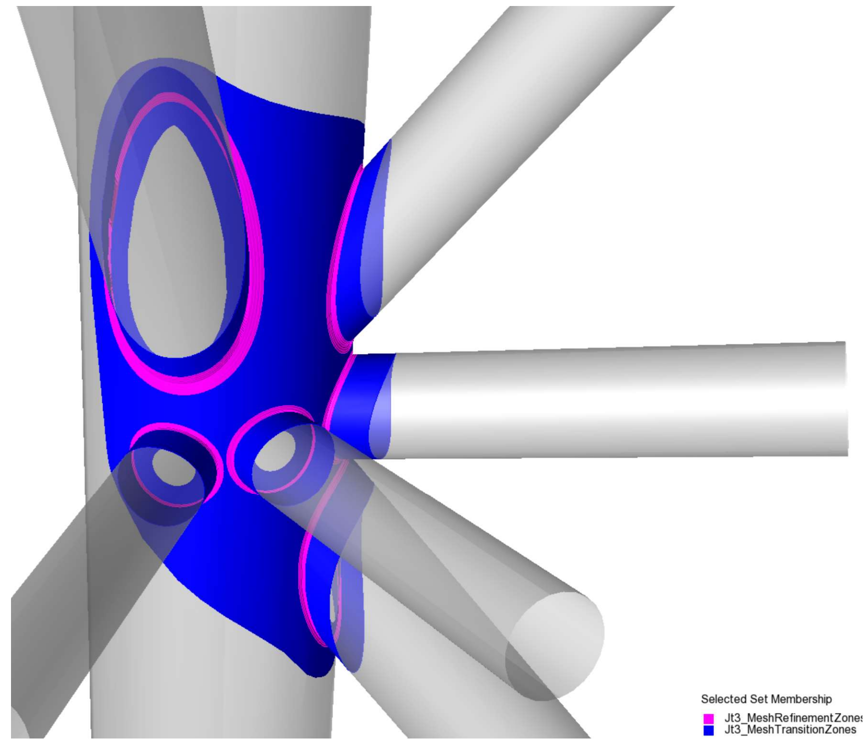


The default values for offset distances are based on the maximum allowable width of the refined zone. They can be edited by the user

Ex. 3: Fatigue of selected parts with fine mesh

- Make the mesh transition zone
  - Main purpose is to reduce the size of the FE model and to increase the quality of auto generated mesh close to the mesh refinement zone



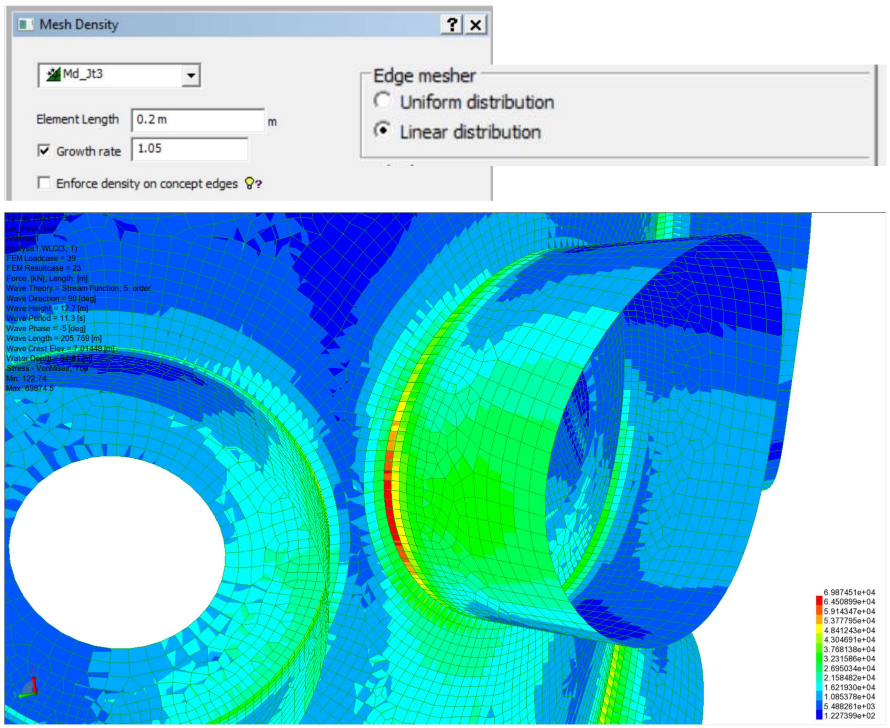


DNV

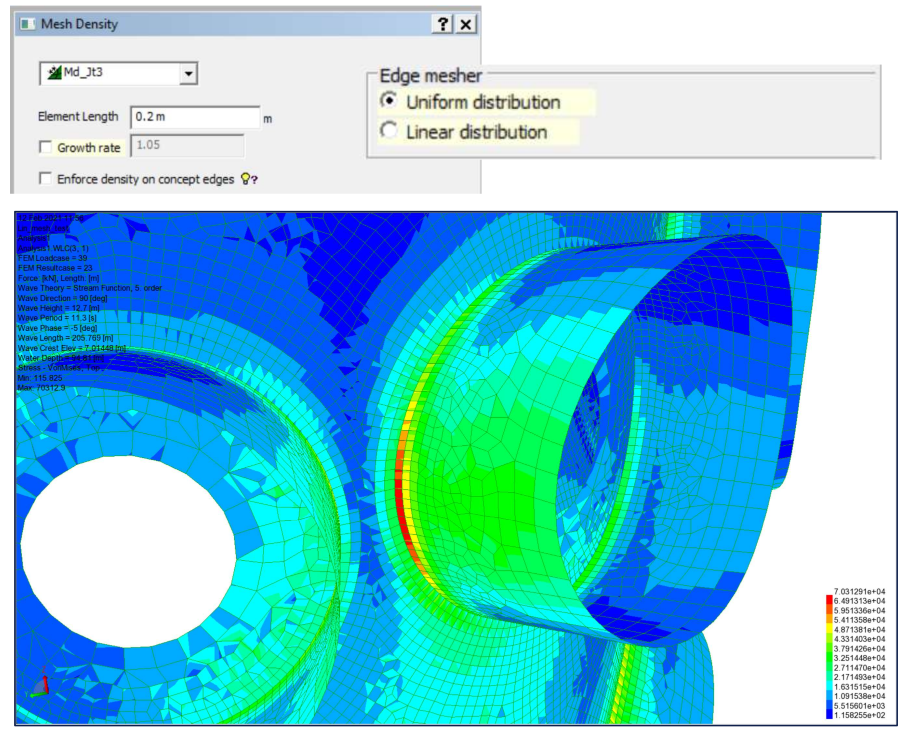
The transition mesh between fine and coarse can be adjusted to a slower growth by use of mesh option “Linear edge meshing” and “Mesh growth factor”

Ex. 3: Fatigue of selected parts with fine mesh

- Using Linear Edge Meshing and Mesh Growth Factor 1.05



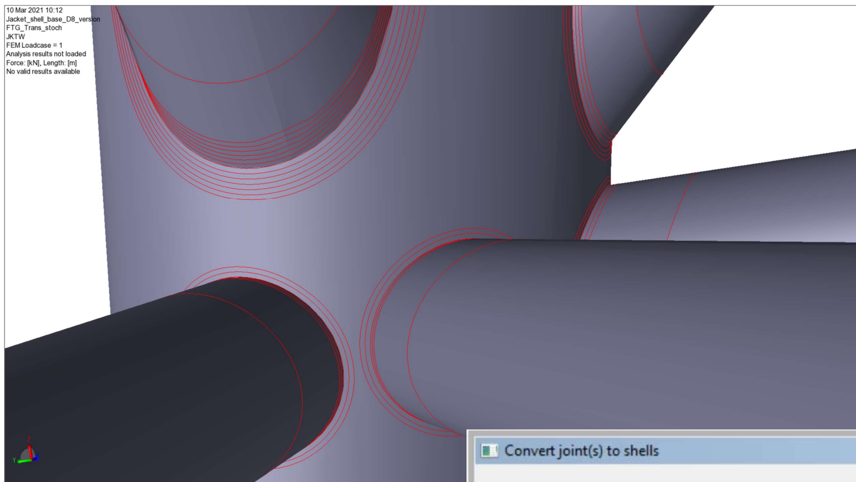
- Using Uniform Edge Meshing



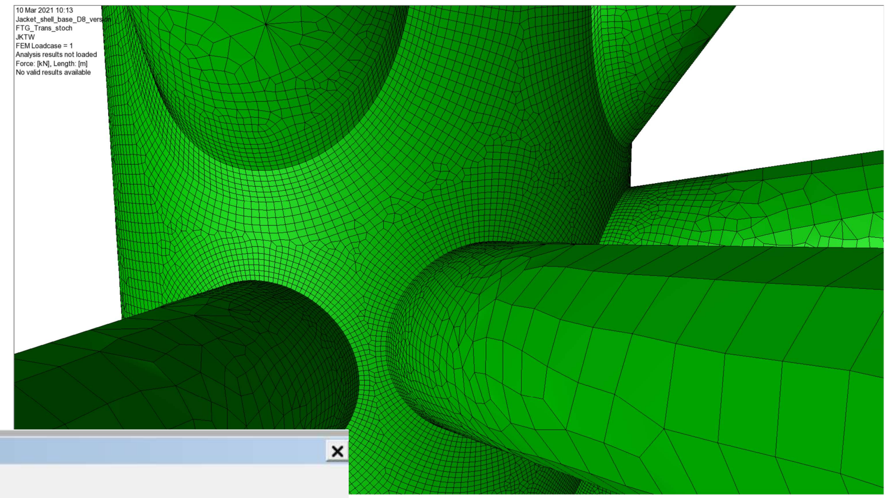
The refined mesh size can be different for each brace

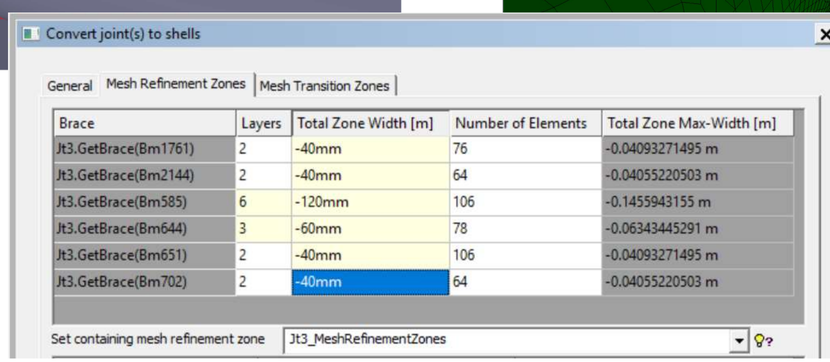
Ex. 3: Fatigue of selected parts with fine mesh

- Use the maximum allowable refined zones and specify 20mm mesh for all



- The mesh



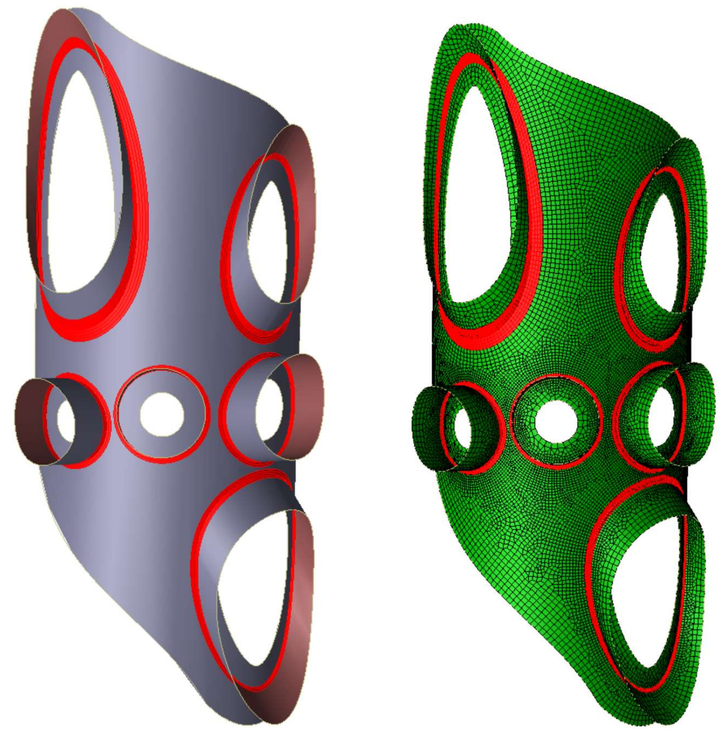


Brace	Layers	Total Zone Width [m]	Number of Elements	Total Zone Max-Width [m]
Jt3.GetBrace(Bm1761)	2	-40mm	76	-0.04093271495 m
Jt3.GetBrace(Bm2144)	2	-40mm	64	-0.04055220503 m
Jt3.GetBrace(Bm585)	6	-120mm	106	-0.1455943155 m
Jt3.GetBrace(Bm644)	3	-60mm	78	-0.06343445291 m
Jt3.GetBrace(Bm651)	2	-40mm	106	-0.04093271495 m
Jt3.GetBrace(Bm702)	2	-40mm	64	-0.04055220503 m

DNV

Ex. 3: Fatigue of selected parts with fine mesh

- Model strategies 1 and 2
  - Pro’s: Very limited data editing to create the finite element models
  - Con’s: Could lead to large finite element models demanding much time for doing analysis and fatigue calculations
- Model strategy 3
  - Pro’s: Very limited data editing to create the finite element models
  - Pro’s: The shell model is divided into mesh zones to reduce the size of the finite element model
  - Pro’s: Reduce the analysis time and speed up the fatigue calculation
  - Con’s: Could lead to slightly longer time to generate the mesh

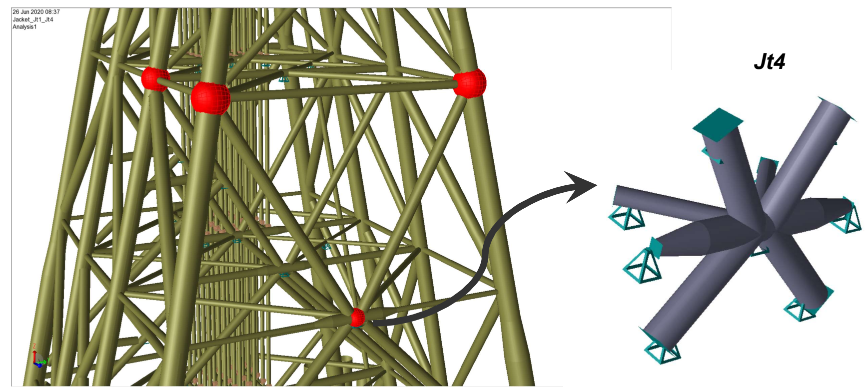


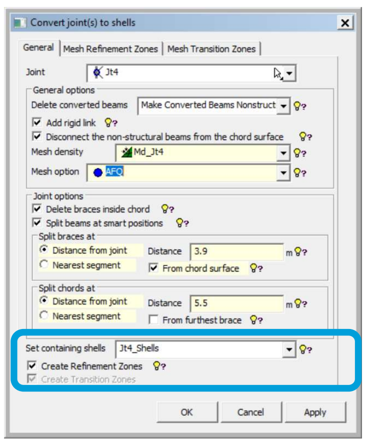
The transition zone for the chord and braces will lead to fewer finite elements, but keeping the mesh quality where needed

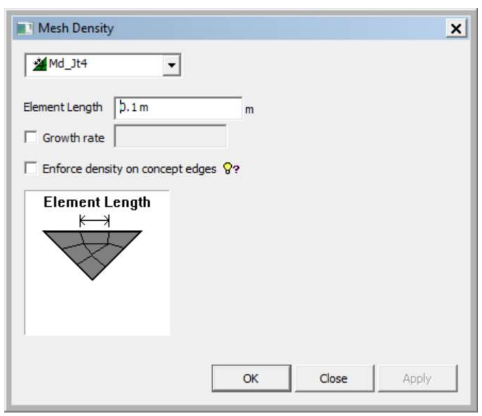
Total editing work: Less than 1 minute work

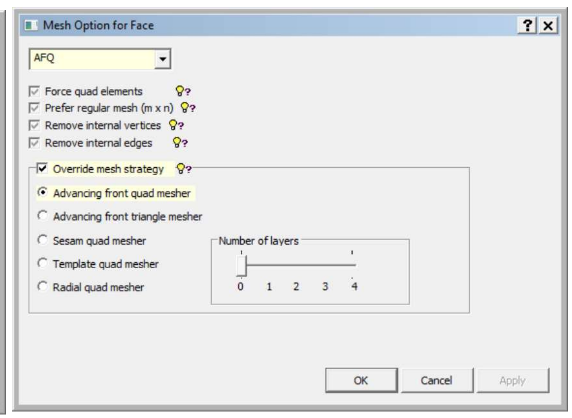
Ex. 4: Fatigue screening overlapping braces

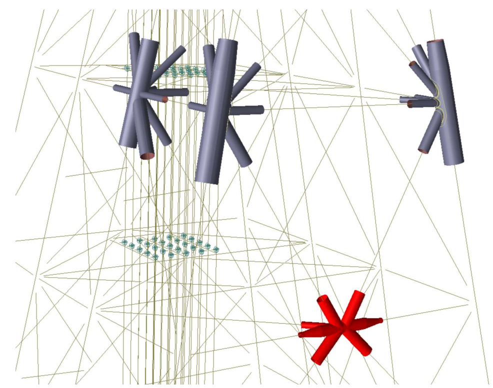
- Divide chord length at 5.5 m (diameter is 1.3 m)
  - Including the cones
- Divide brace length at 3.9 m (largest diameter is 1.3 m)
  - From chord surface
- Mesh density for whole joint is 100 mm
- Mesh option for whole joint is Advancing front quad mesher
- All braces have overlaps (see next slide)





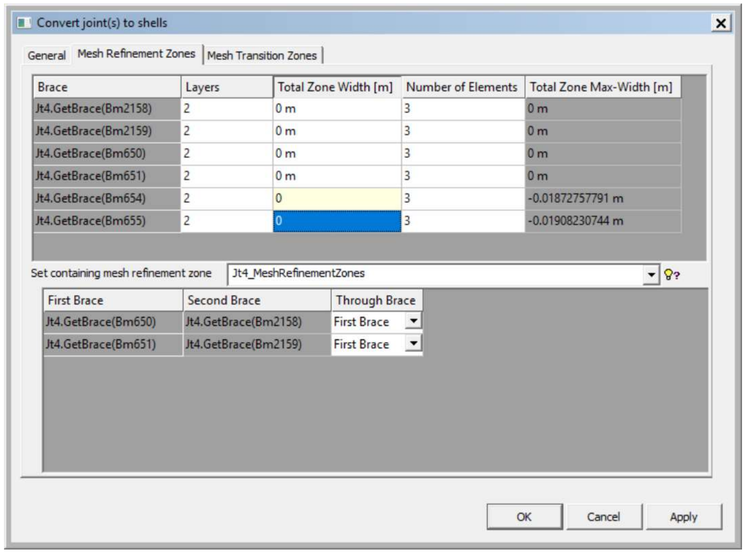






Ex. 4: Fatigue screening overlapping braces

- For this joint - using defaults for through braces and no mesh refinement zone
  - Priority 1: Diameter
  - Priority 2: Thickness
  - Users can change through braces



Convert joint(s) to shells

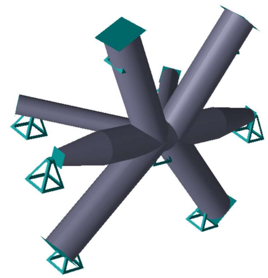
General | Mesh Refinement Zones | Mesh Transition Zones

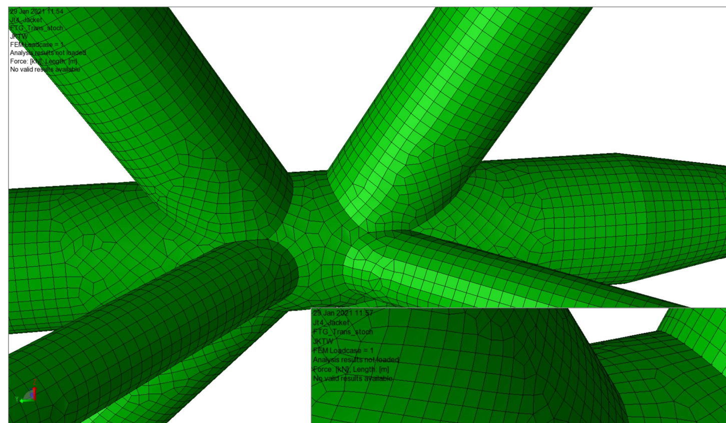
Brace	Layers	Total Zone Width [m]	Number of Elements	Total Zone Max-Width [m]
Jt4.GetBrace(Bm2158)	2	0 m	3	0 m
Jt4.GetBrace(Bm2159)	2	0 m	3	0 m
Jt4.GetBrace(Bm650)	2	0 m	3	0 m
Jt4.GetBrace(Bm651)	2	0 m	3	0 m
Jt4.GetBrace(Bm654)	2	0	3	-0.01872757791 m
Jt4.GetBrace(Bm655)	2	0	3	-0.01908230744 m

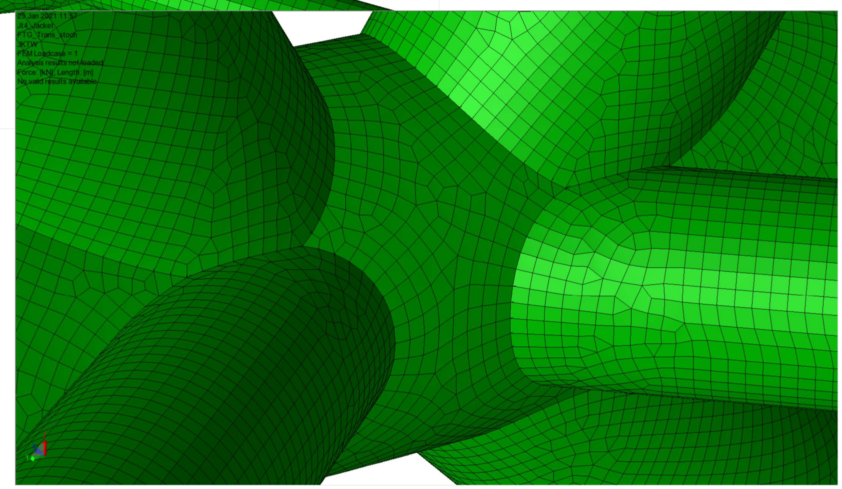
Set containing mesh refinement zone: Jt4_MeshRefinementZones

First Brace	Second Brace	Through Brace
Jt4.GetBrace(Bm650)	Jt4.GetBrace(Bm2158)	First Brace
Jt4.GetBrace(Bm651)	Jt4.GetBrace(Bm2159)	First Brace

OK | Cancel | Apply



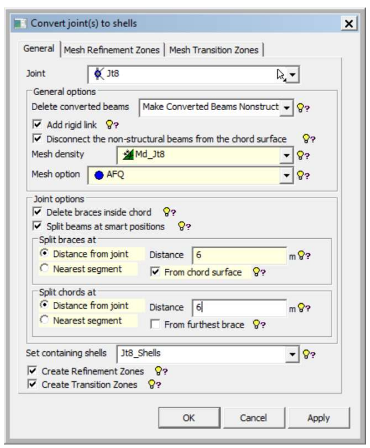


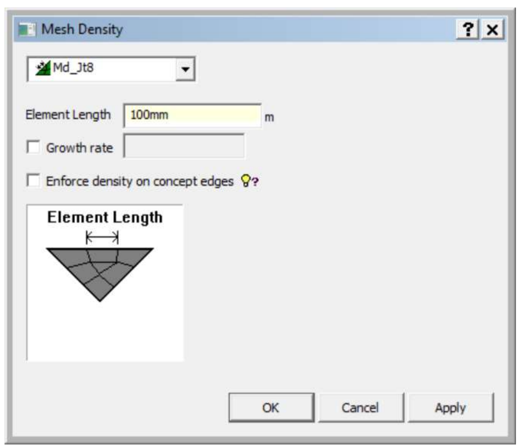


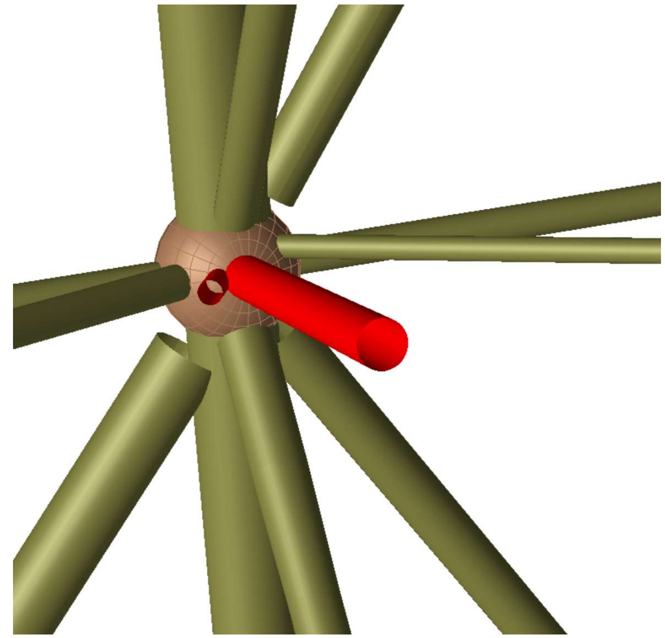
Total editing work: Less than 1 minute work

Ex. 5: Detailed fatigue when overlapping braces

- When there is a mix of overlapping braces and braces with sufficient gaps, the refined mesh zone can be made
- This example has 11 braces of which 3 are overlapping
- Split position is 6 m for both braces and chord
- Global mesh density is 100 mm and mesh option is Advancing front quad mesher



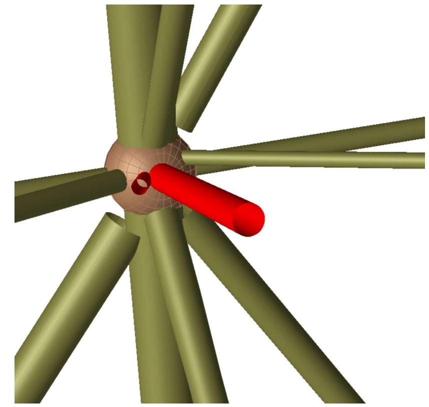


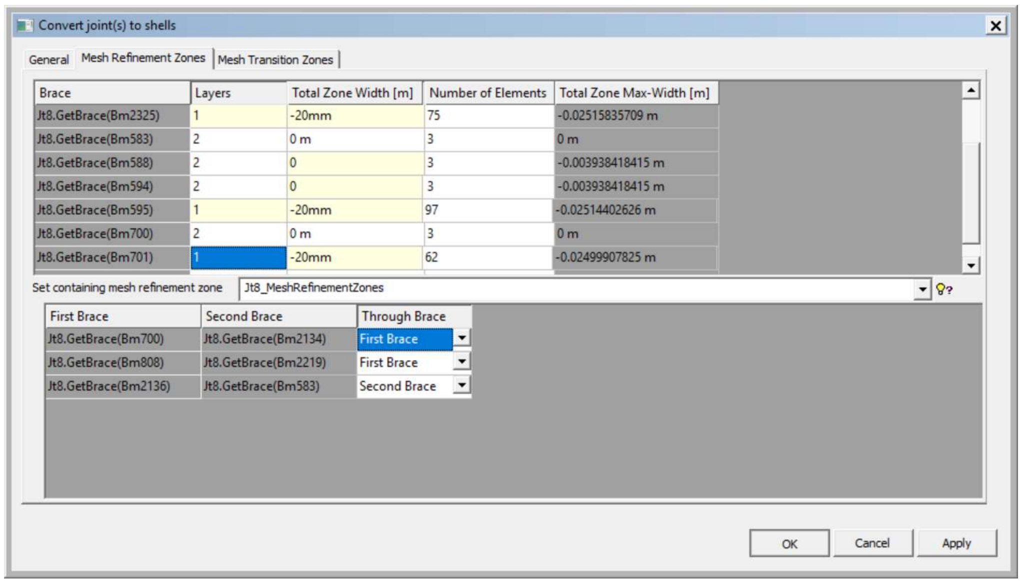


DNV

Ex. 5: Detailed fatigue when overlapping braces

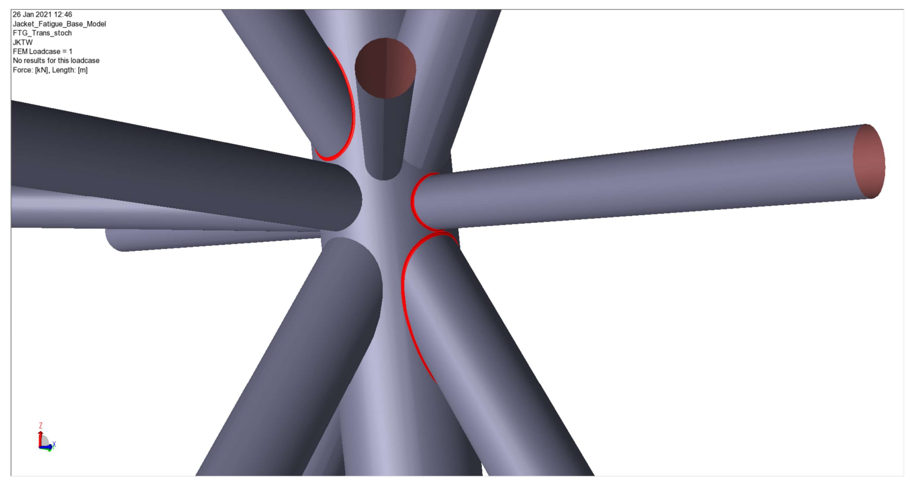
- The total max zone width varies from 0 (for the overlapping braces) through 4mm to 25mm
  - For the 4mm zones this is edited to 0
- Editing the through braces in this case
  - This joint has 11 braces of which 3 are overlapping





Brace	Layers	Total Zone Width [m]	Number of Elements	Total Zone Max-Width [m]
Jt8.GetBrace(Bm2325)	1	-20mm	75	-0.02515835709 m
Jt8.GetBrace(Bm583)	2	0 m	3	0 m
Jt8.GetBrace(Bm588)	2	0	3	-0.003938418415 m
Jt8.GetBrace(Bm594)	2	0	3	-0.003938418415 m
Jt8.GetBrace(Bm595)	1	-20mm	97	-0.02514402626 m
Jt8.GetBrace(Bm700)	2	0 m	3	0 m
Jt8.GetBrace(Bm701)	1	-20mm	62	-0.02499907825 m

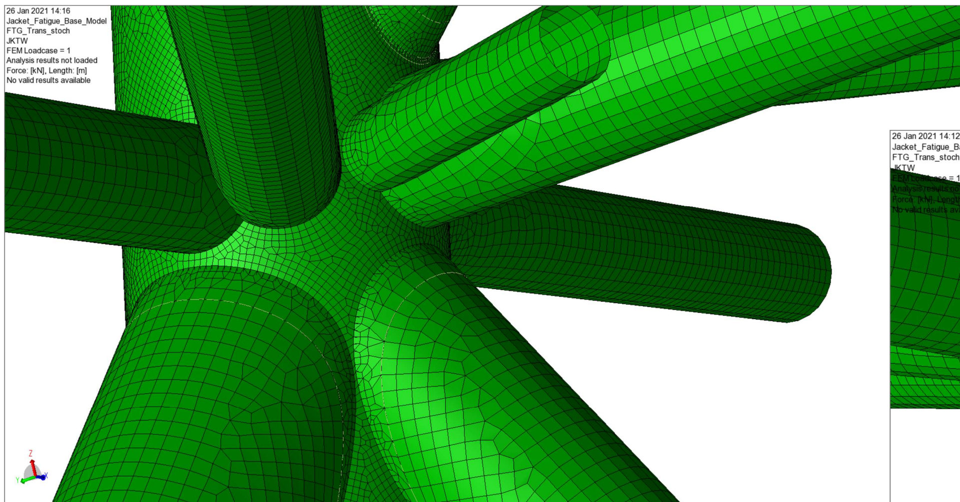
First Brace	Second Brace	Through Brace
Jt8.GetBrace(Bm700)	Jt8.GetBrace(Bm2134)	First Brace
Jt8.GetBrace(Bm808)	Jt8.GetBrace(Bm2219)	First Brace
Jt8.GetBrace(Bm2136)	Jt8.GetBrace(Bm583)	Second Brace

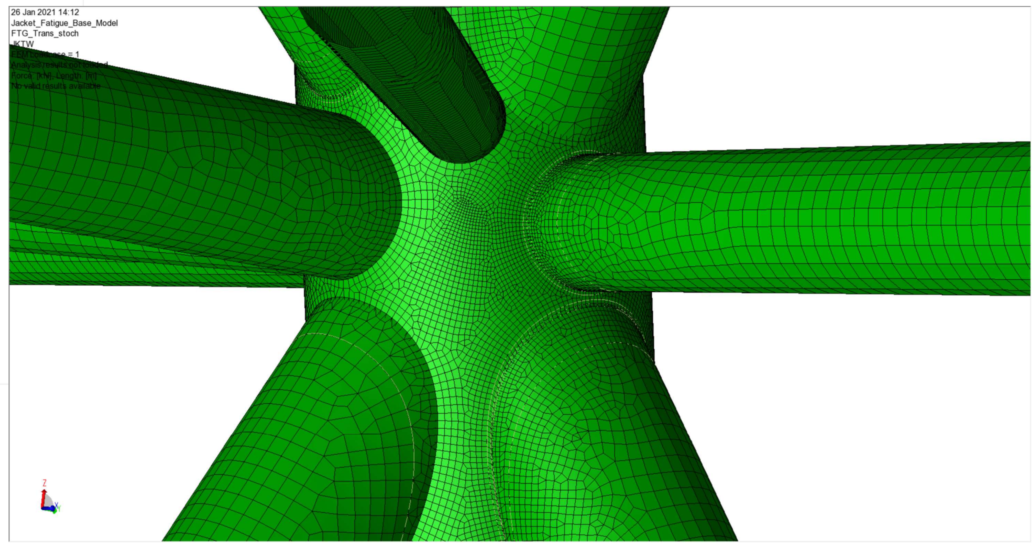


With one command the beam tubular joint is converted to a finite element model that is integrated with the jacket maintaining the wave loads. It is ready for stress and fatigue analysis

Ex. 5: Detailed fatigue when overlapping braces

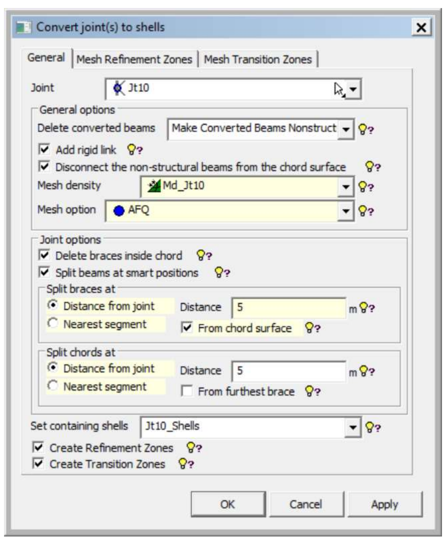
- Automatic conversion and meshing of joint with 11 braces – of them 3 pairs of overlapping braces
  - Using defaults for mesh transition zone and mesh density of 40mm

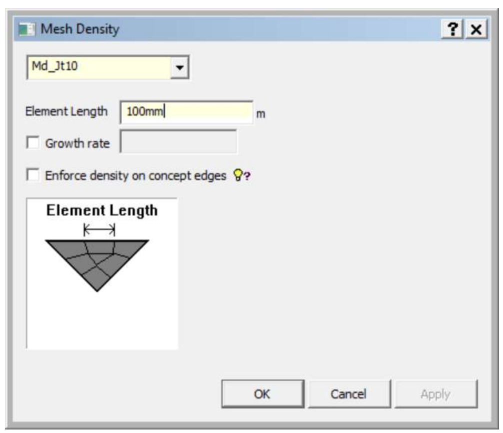


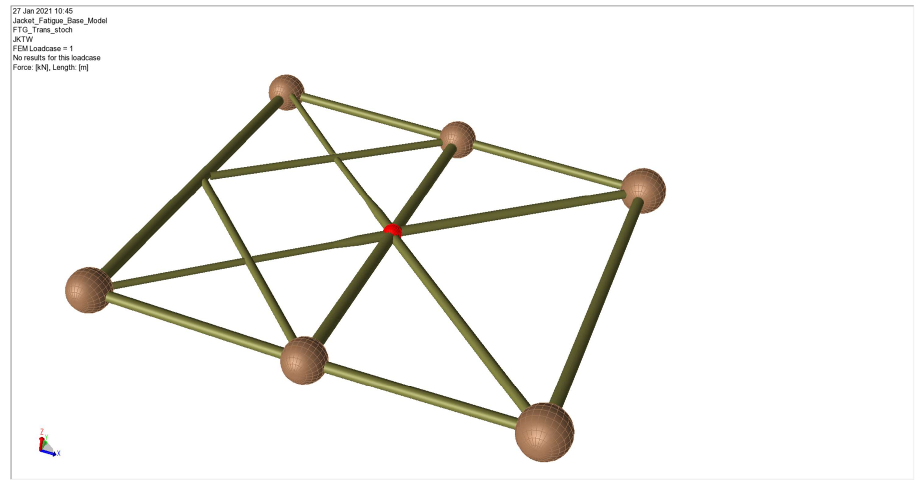


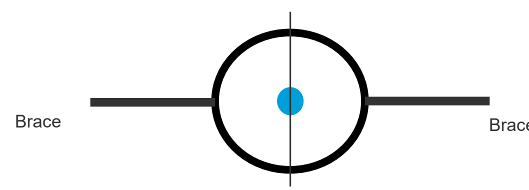
Ex. 6: Detailed fatigue when the joint has braces on both sides

- This is a special case where the angle between braces is 180 degrees
- Split position is set to 5 m for both braces and chord
- Global mesh density is 100 mm and mesh option is Advancing front quad mesher





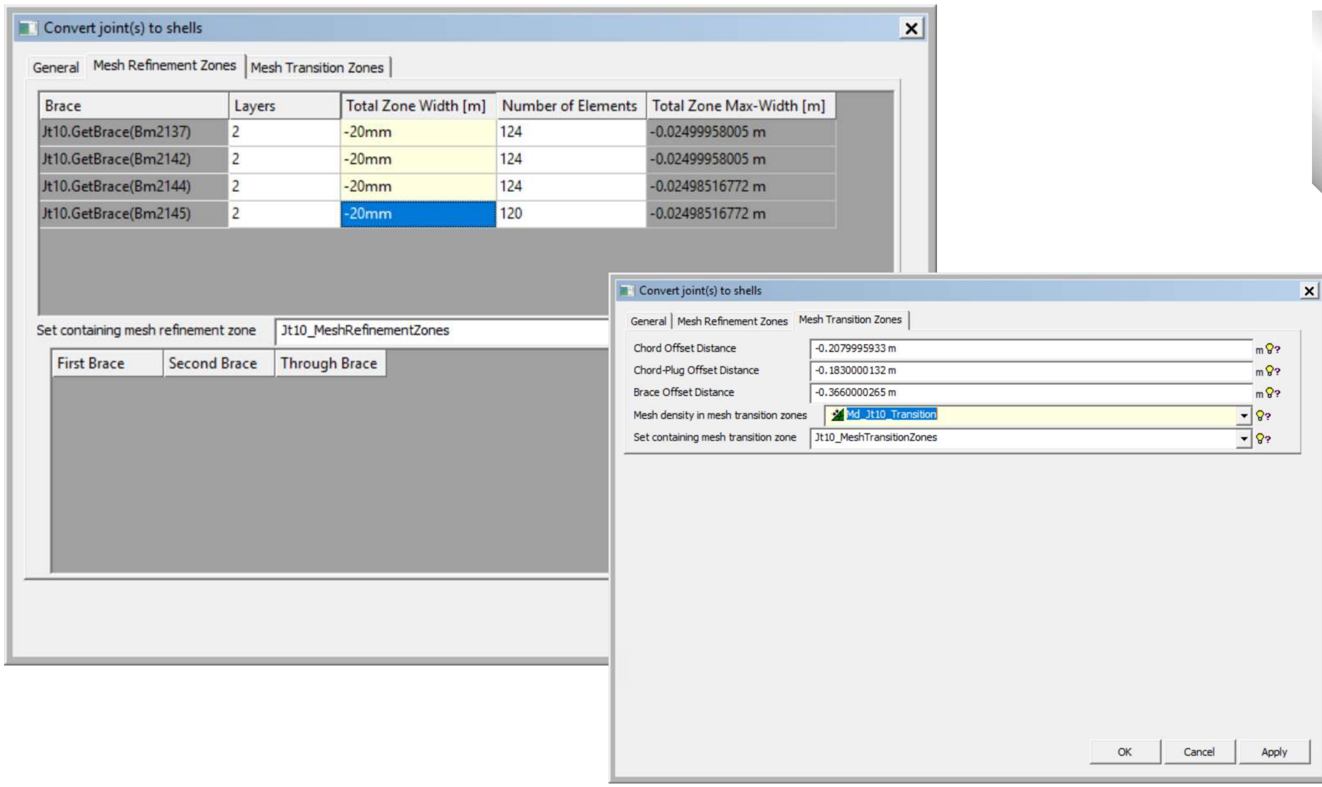


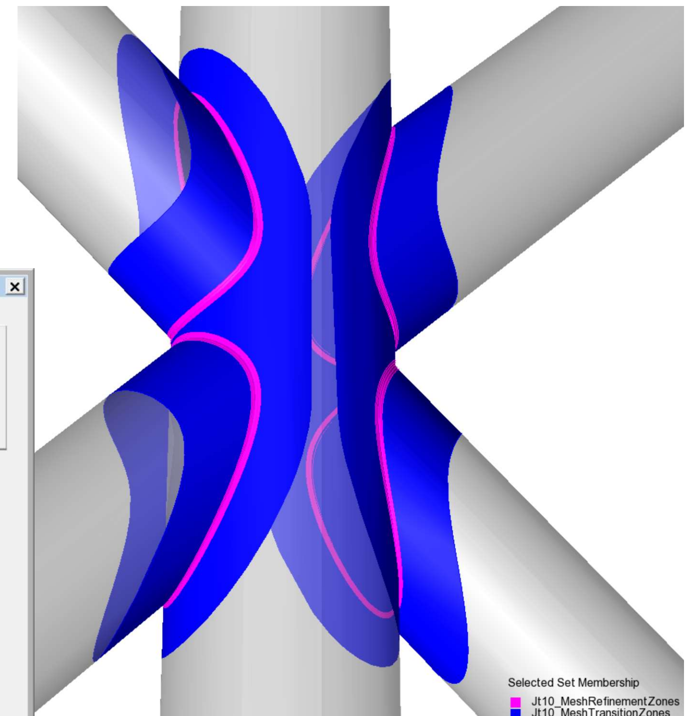


The "smart divide" of the chord will split the chord as shown above to ensure best possible mesh

Ex. 6: Detailed fatigue when the joint has braces on both sides

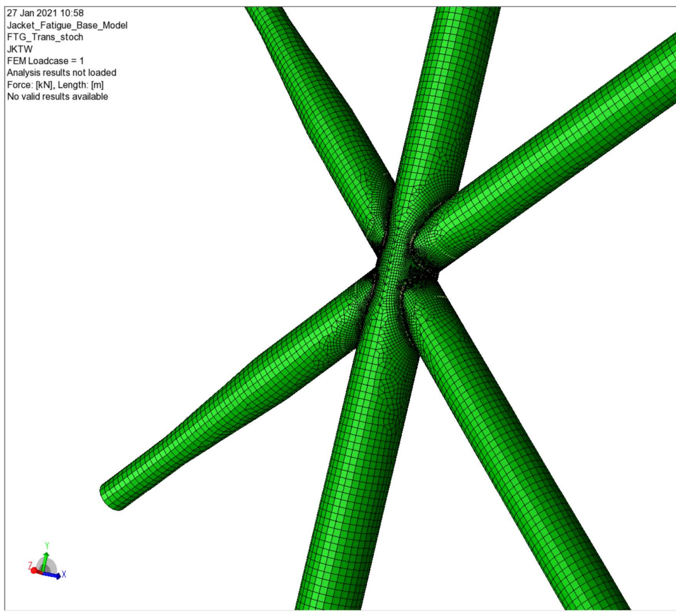
- Specifying 10mm mesh for the refined part and 40mm mesh in the transition part

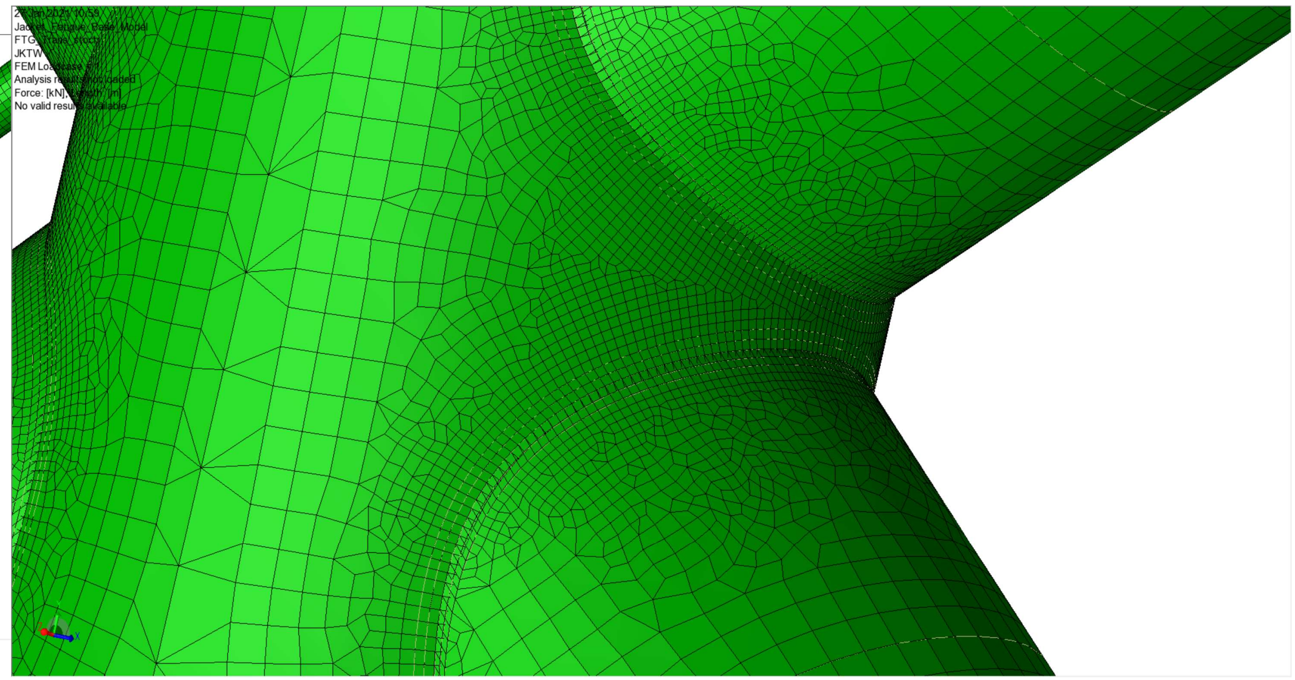




Ex. 6: Detailed fatigue when the joint has braces on both sides

- Automatic meshing of the joint with 11 braces
  - Using defaults for mesh transition zone and mesh density of 40mm

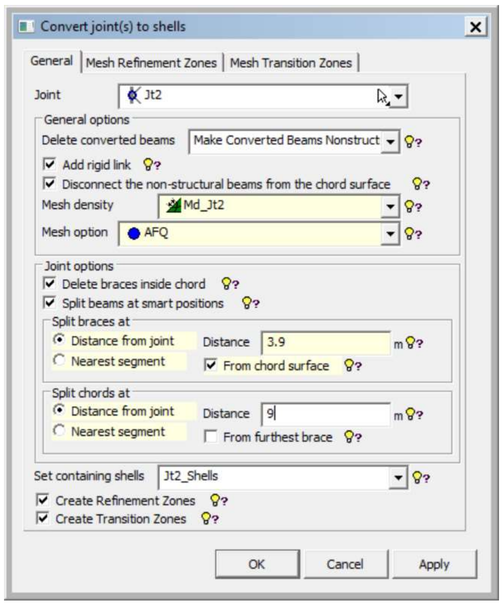


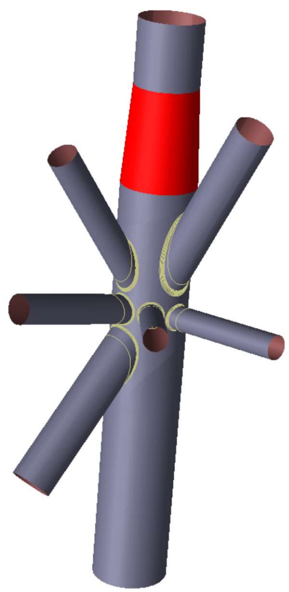


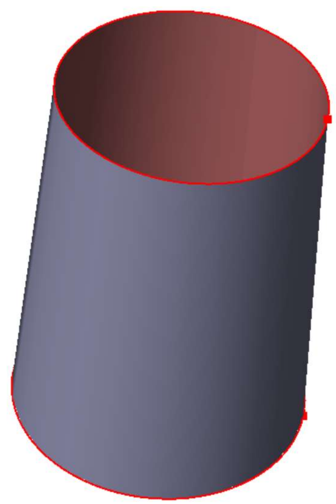
Total editing work: Less than 5 minutes work

Ex. 7: Adding a ring stiffener to Jt2

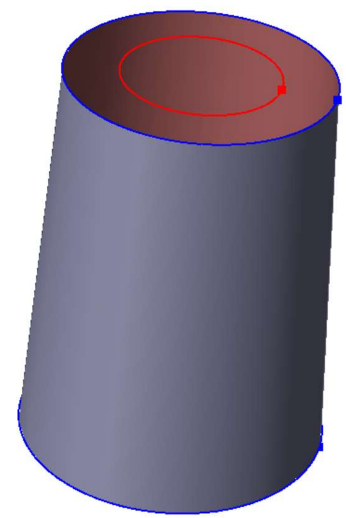
- This example shows how to convert a joint and insert ring stiffeners at top and bottom of a conical transition
  - Width 400mm and flange height 150mm







1st step is to create model curves for use when making the ring stiffener

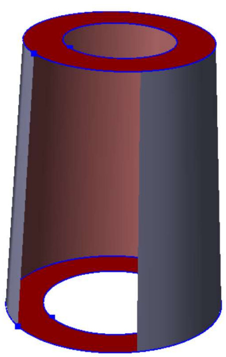


2nd step is to offset the model curves 400mm

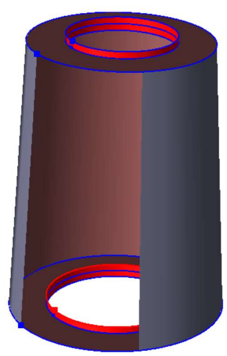
Parts of the chord have been hidden to view the ring stiffeners

Ex. 7: Adding a ring stiffener to Jt2

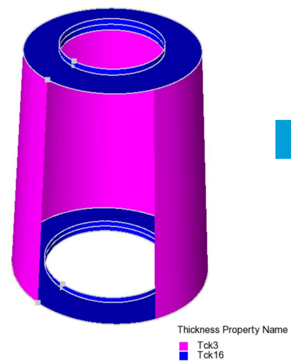
- Make the ring stiffeners based on curves and make the mesh



3rd step is to cover the curves to make the ring stiffener

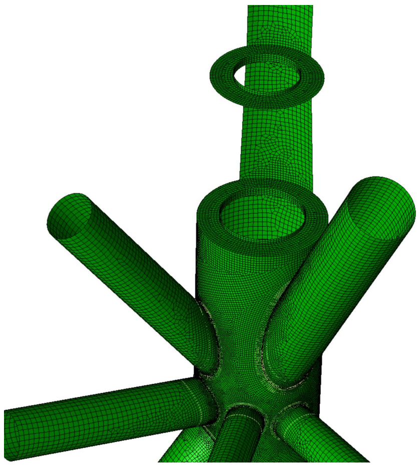


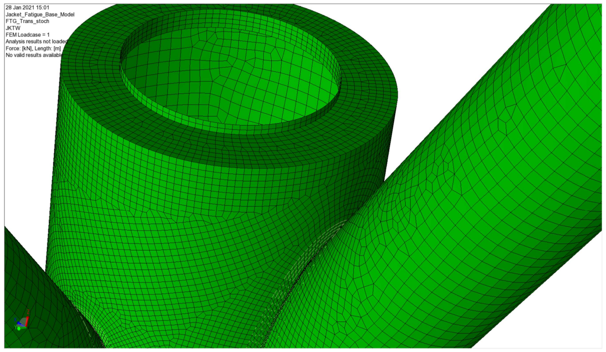
4th step is to copy the curves +/- 75mm to make the flanges



Add thickness, material and mesh properties

Make mesh

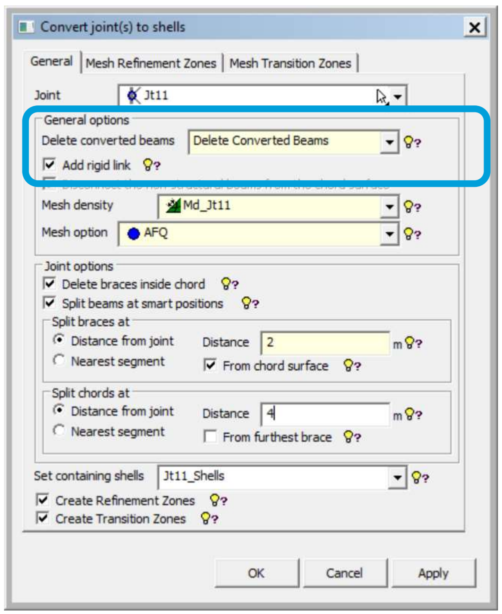


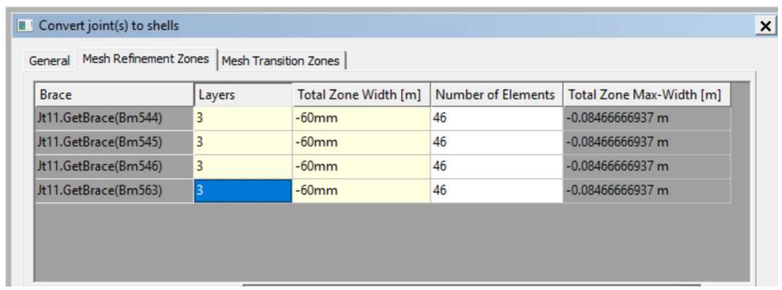


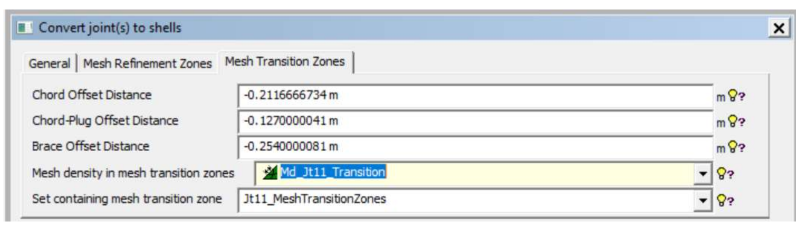
Total editing work: Less than 1 minute work

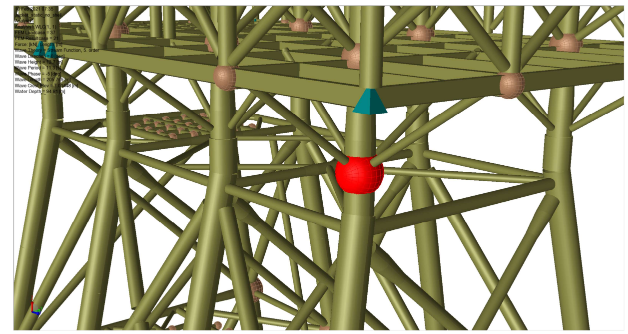
Ex. 8: Convert a joint not exposed to wave loads

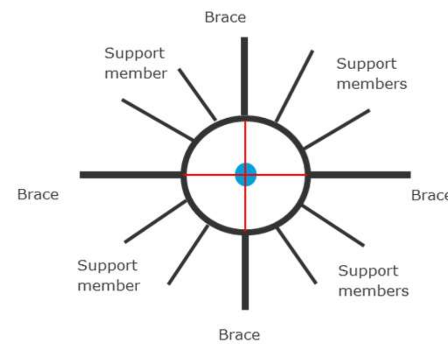
- Difference from other examples is to delete the converted beams (i.e. not make them non-structural)







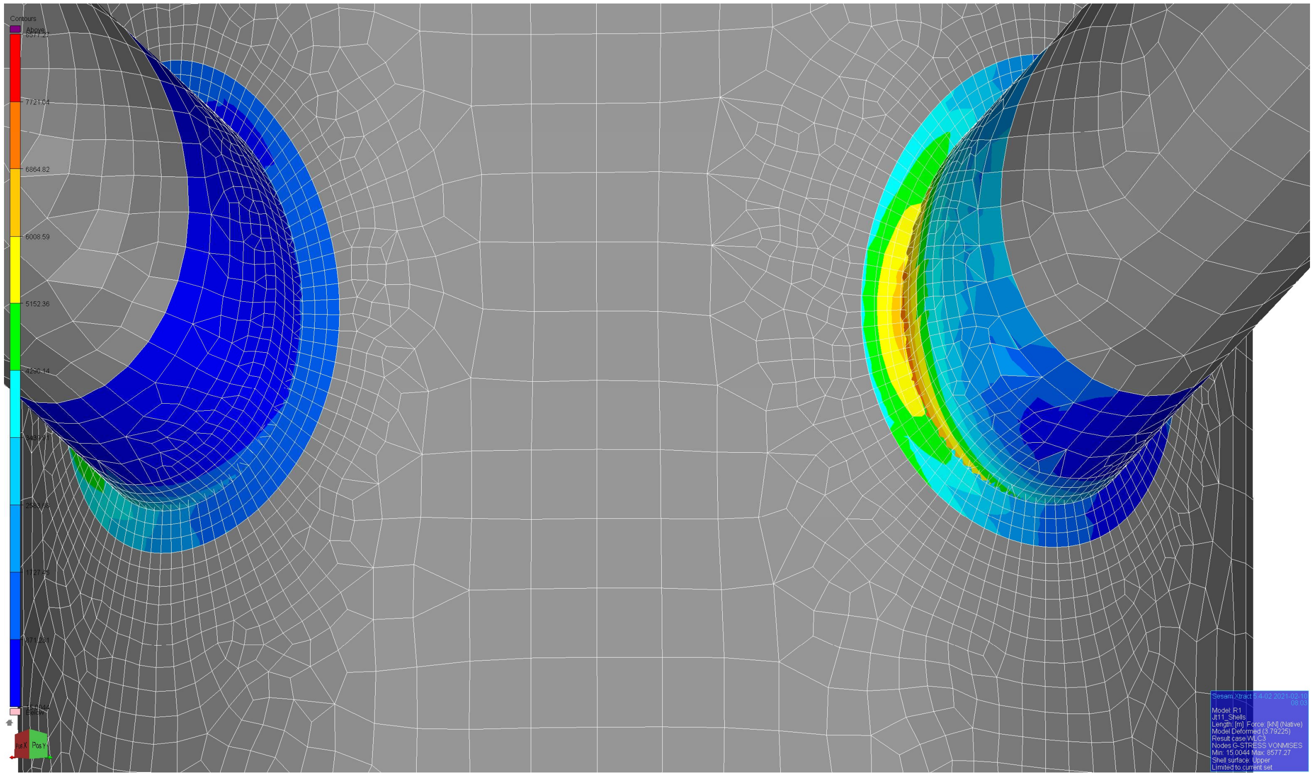




The "smart divide" of the chord will split the chord as shown above to ensure best possible mesh

Ex. 8: Convert a joint not exposed to wave loads

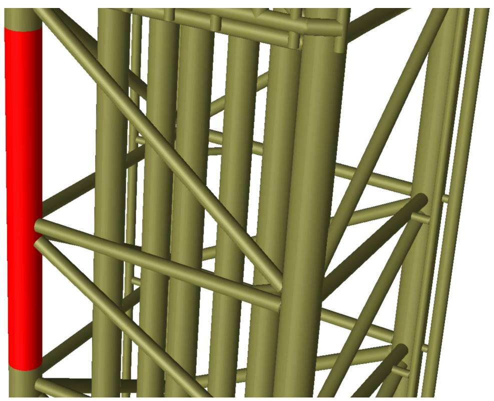
- The mesh and example of viewing vonMises stresses
  - Refined zone: 3 layers each 20mm mesh
  - Transition zone: 50mm mesh



Total editing work: Less than 2 minute work

Ex. 9: Convert a joint with grouted chord

- First step is to delete the inner beam and to modify the thickness of the chord
  - Divide the beams at split positions first
  - Equivalent leg thickness = $\sqrt{\text{Leg thickness}^2 + \text{Pile thickness}^2}$
  - Maintain the mass (structure and grout) by use of equivalent material density of the outer leg
  - Delete the inner beam

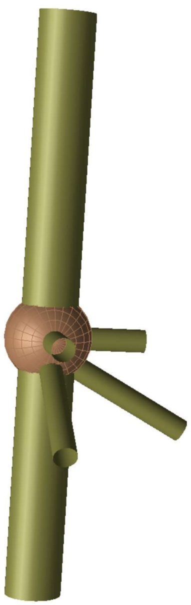


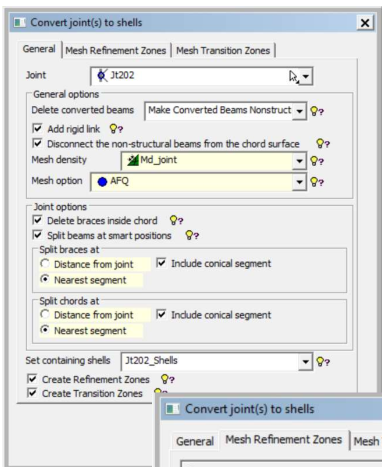
Outer leg thickness 13 mm
Inner leg thickness 10 mm

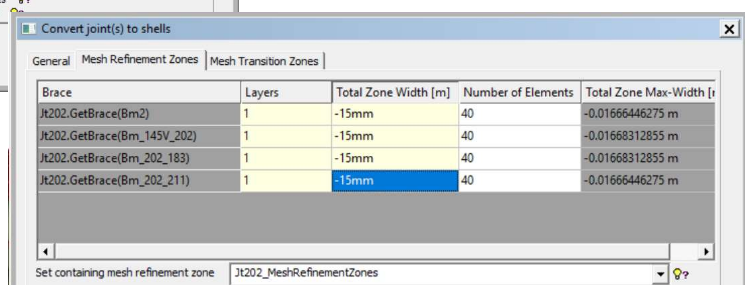
Equivalent leg thickness 16 mm

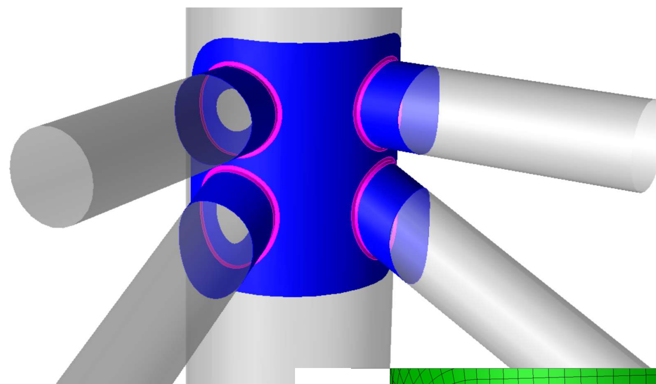
Ex. 9: Convert a joint with grouted chord

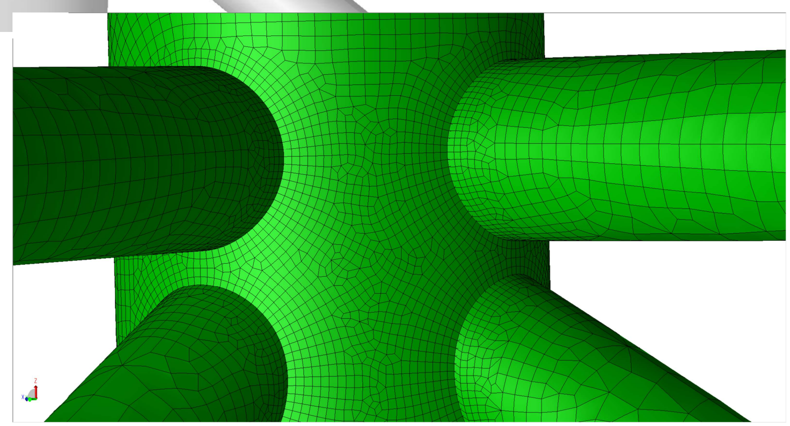
- From now on similar as other joint types
  - Example shown below using refined and transition zone
  - One layer as the gap is less than 50mm









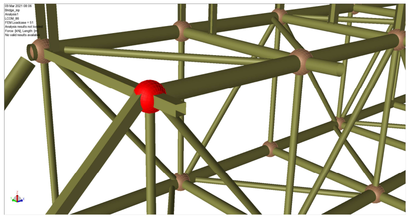


DNV

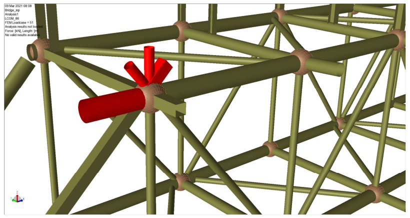
This example is based on a manual conversion using the command "Convert beam to shell". The use of this command will lead to significant savings compared to a manual creation of the shell model

Ex. 10: Convert a non-tubular joint

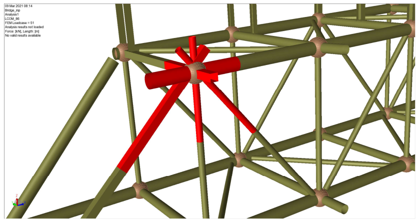
- A typical bridge is used to demonstrate the conversion process
  - This joint has 3 tubular beams and 2 box beams



- 1st step is to ensure that all members are passing through the joint
  - The superfluous parts must be deleted after the conversion

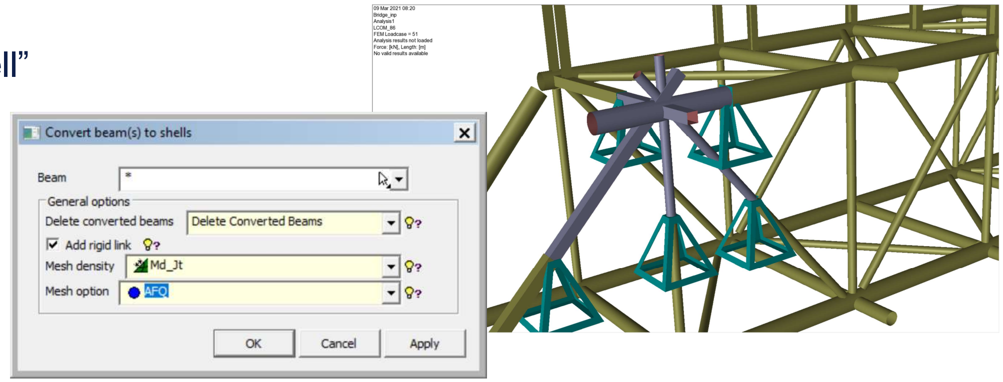


- 2nd step is to divide the beams where the shell model shall start

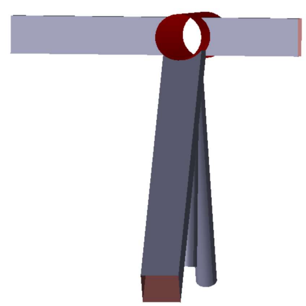


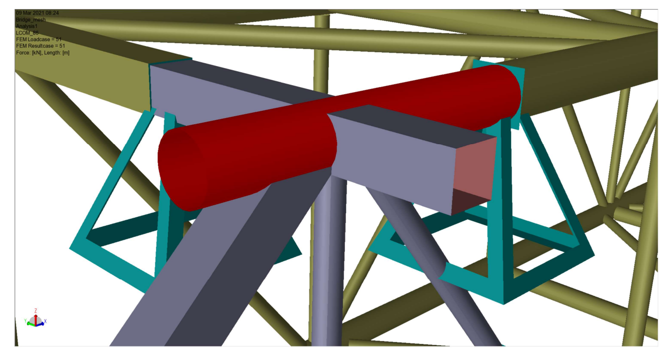
Ex. 10: Convert a non-tubular joint

- Each beam is converted to shell by “Convert beam(s) to shell”
  - This can be done by selecting one by one – or by selecting all beams
  - If member wind load analysis, then include non-structural members



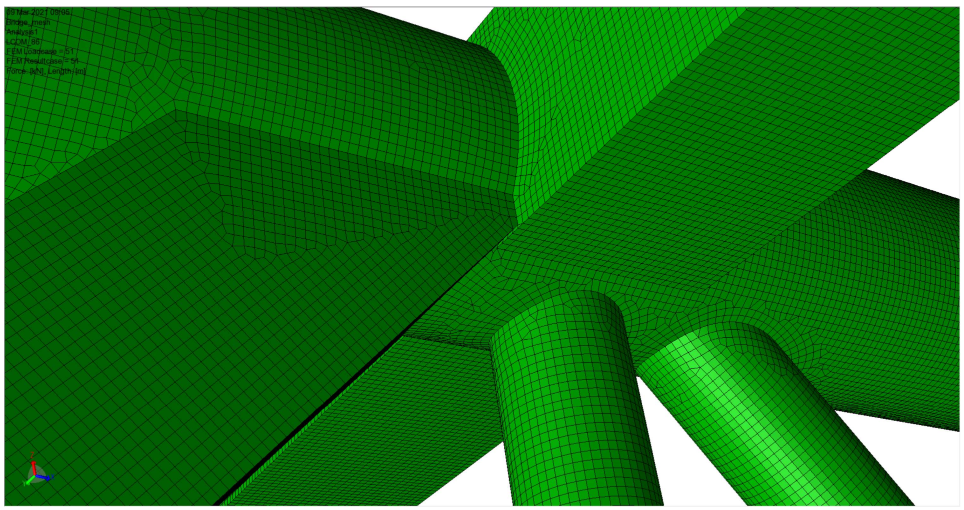
- Decide which member is the through member and trim manually
  - Divide each plate (use “explode”) and delete superfluous parts



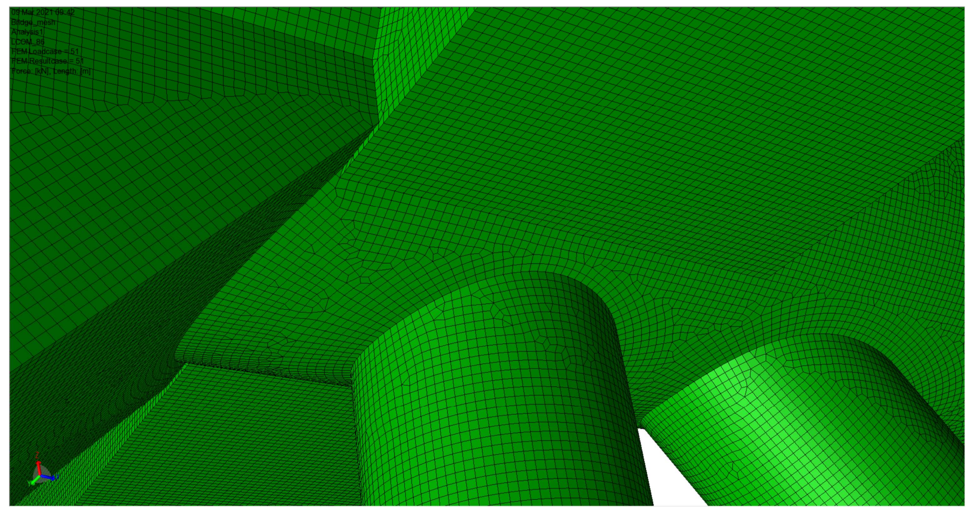


Ex. 10: Convert a non-tubular joint

- Make mesh and run analysis
  - Notice that such conversion does not include “smart divide” nor refined mesh zones so it may be necessary to manually control mesh generation or edit the mesh



Mesh density 10mm, linear edge meshing, Advanced Front Quad Mesher

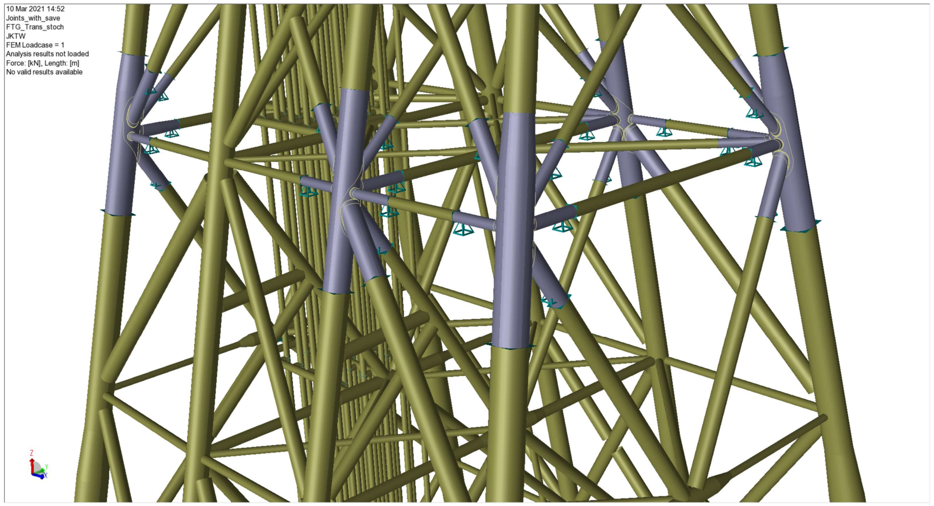


Mesh density 5mm, uniform edge meshing, Advanced Front Quad Mesher

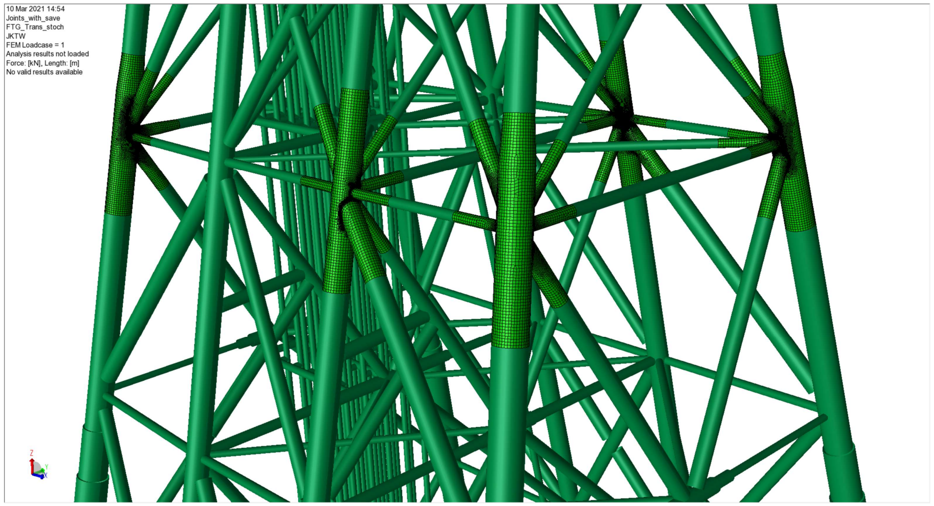
One may use “Conditional meshing” in case a joint is converted and meshed before the next joint. Then the previous mesh is kept

Ex. 11 Converting multiple joints

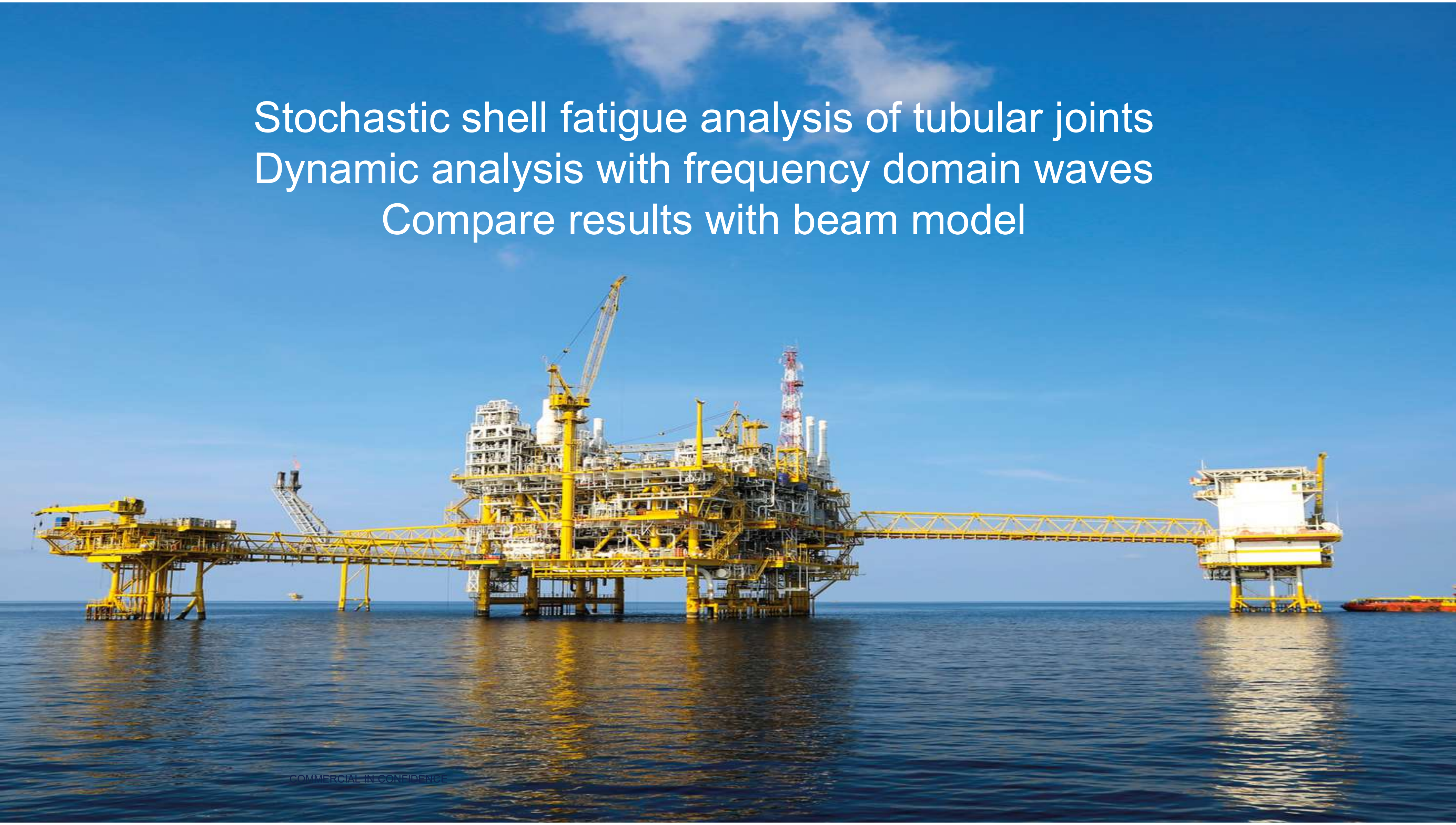
- 5 joints converted



- The mesh



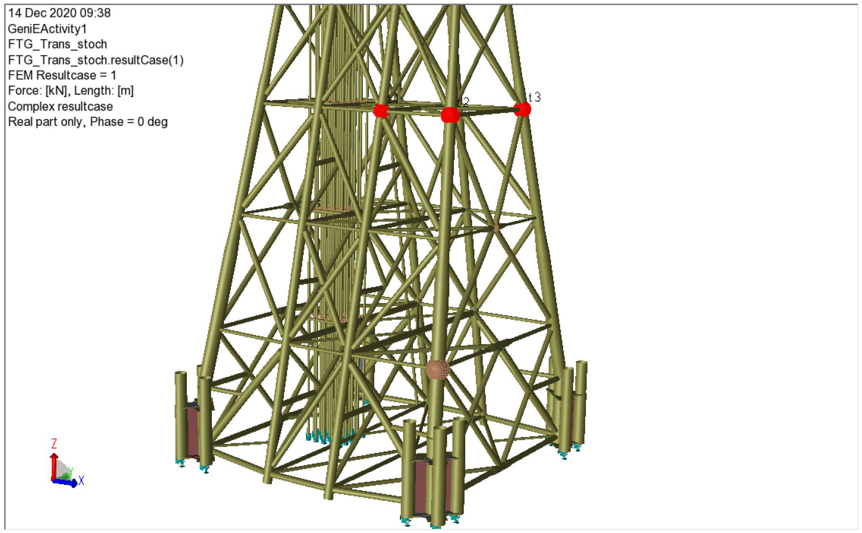
Stochastic shell fatigue analysis of tubular joints
Dynamic analysis with frequency domain waves
Compare results with beam model



Disclaimer: The results presented herein cannot be used to make general conclusions. However, several research projects show that empirical SCF may be on the conservative side

The jacket model

- Dynamic analysis
  - Linearized piles
  - Modal superposition and master slave
  - Eigenvalue analysis
  - 3 wave directions (0, 45 and 90 degrees) used in stochastic dynamic analysis
    - 34 wave periods
    - PM spectrum, Hs=3.67 m, Tz=5.953 sec
- Eigenvalues
  - T1 = 2.613 sec
  - T2 = 2.514 sec
  - T3 = 1.659 sec
- Fatigue life (20 year life time) based on empirical SCF
  - Jt1 = 5.76 (failure)
  - Jt2 = 6.91 (failure)
  - Jt3 = 7.74 (failure)

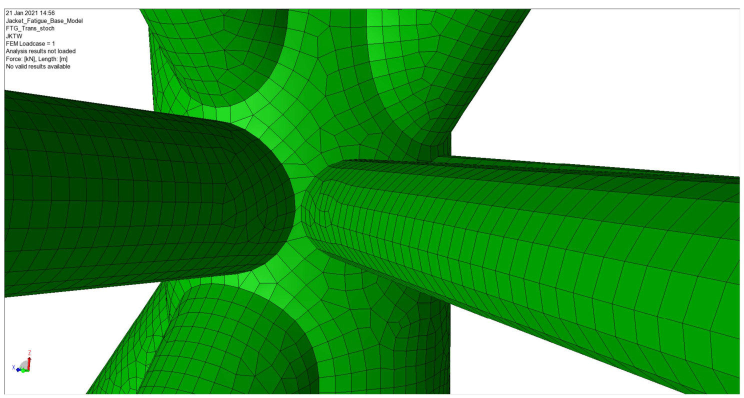


FRQ	34.						
C	T1	T2	T3	T4	T5	T6	T7
FRQ	1.4	1.6	1.68	1.74	1.85	1.9	2.07
	2.10	2.15	2.20	2.25	2.30	2.348	2.379
	2.45	2.5	2.55	2.60	2.65	2.70	2.80
	2.90	3.0	3.1	3.20	3.50	4.0	4.50
	5.00	6.0	7.0	8.0	10.0	12.00	

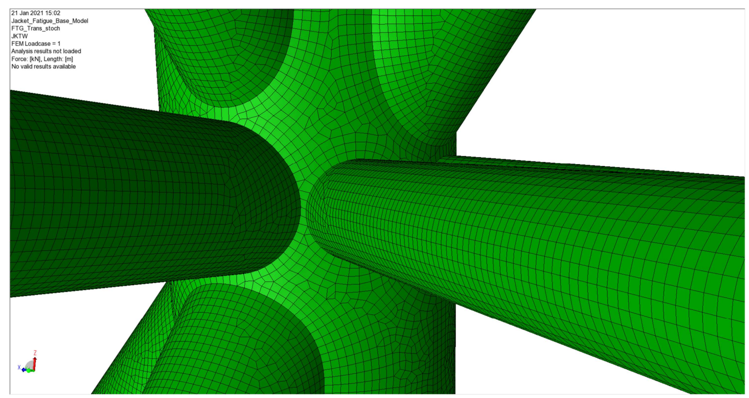
Total editing time: Less than 3 minutes

Fatigue analysis of Jt1

- The basis for this dynamic analysis is a beam model using **modal superposition method and master-slave**
  - The first 10 eigenmodes have been used in the analysis
- Prepare the combined beam and shell model for dynamic analysis
  - If there is a supernode (master-slave) in the middle of the joint Jt1 it must be removed. Non-structural members do not support supernodes and analysis will fail
  - Modify all support rigid links to Super (1, 1, 1, 0, 0, 0) to ensure that the mass of the shells are counted for in the dynamic analysis
  - Specify the use of 1^{st} order thick shell elements
  - Make a mesh priority Mpri1 – and include set Jt1_shell. Modify the mesh activity to include Mpri1
  - Make a set “Visualize_dynamics” – include all structure except non-structural. This is strictly not needed, but handy to have when looking at eigenvalues and deformations in Xtract



Mesh density 100 mm



Mesh density 50 mm

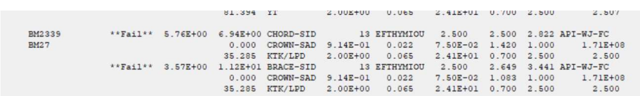
DNV

For this particular model a mesh density of 100 mm gives convergence of fatigue life

Fatigue analysis of Jt1

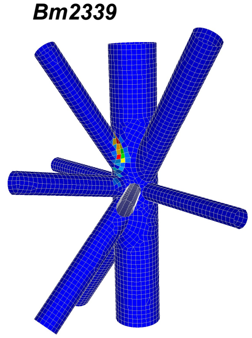
- Fatigue life – and compared with beam model
  - Life time is set to 20 years
  - This case shows the importance of using 1st order thick shell as compared to 1st order thin shell

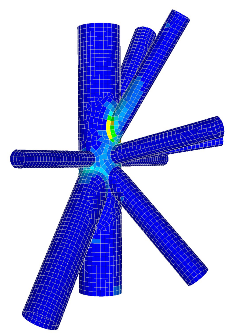
	Jt1 – Beam model	Jt1 – Shell model, 200 mm thin shell	Jt1 – Shell model, 200 mm thick shell	Jt1 – Shell model, 100 mm thick shell	Jt1 – Shell model, 50 mm thick shell
Max Uf	5.76	5.61	0.37	0.59	0.58
	Bm2339			Thin shell: 1.80	

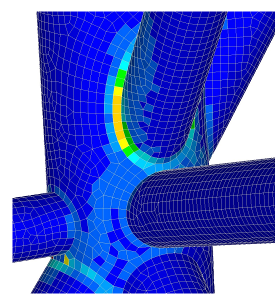


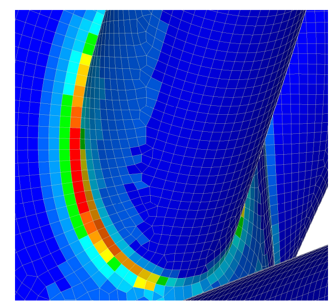
```
                                81.394  YT        2.00E+00   0.065     2.41E+01  0.700  2.500        2.507

   BM2339         **Fail**  5.76E+00  6.94E+00 CHORD-SID          13 EFTHYMIOU   2.500     2.500  2.822 API-WJ-FC
   BM27                                  0.000 CROWN-SAD  9.14E-01   0.022     7.50E-02  1.420  1.000     1.71E+08
                                        35.285 KTK/LPD    2.00E+00   0.065     2.41E+01  0.700  2.500        2.500
                  **Fail**  3.57E+00  1.12E+01 BRACE-SID          13 EFTHYMIOU   2.500     2.649  3.441 API-WJ-FC
                                         0.000 CROWN-SAD  9.14E-01   0.022     7.50E-02  1.083  1.000     1.71E+08
                                        35.285 KTK/LPD    2.00E+00   0.065     2.41E+01  0.700  2.500        2.500
```

Bm2339





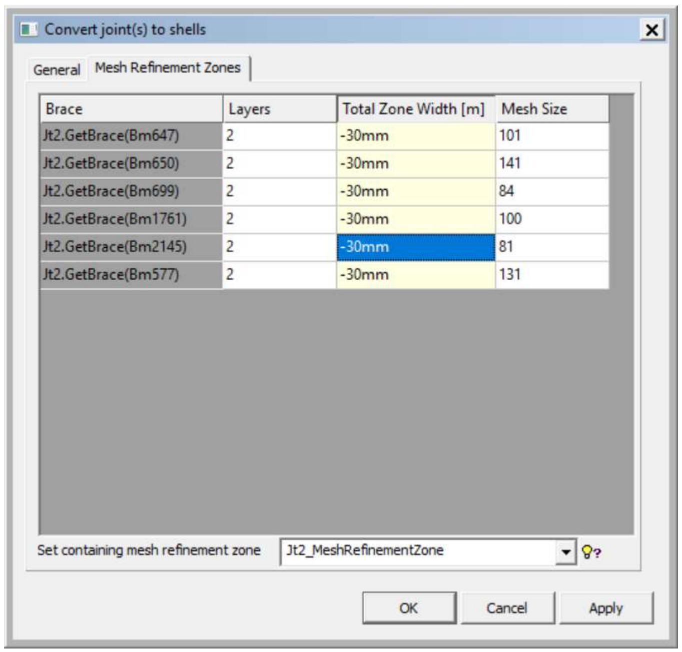


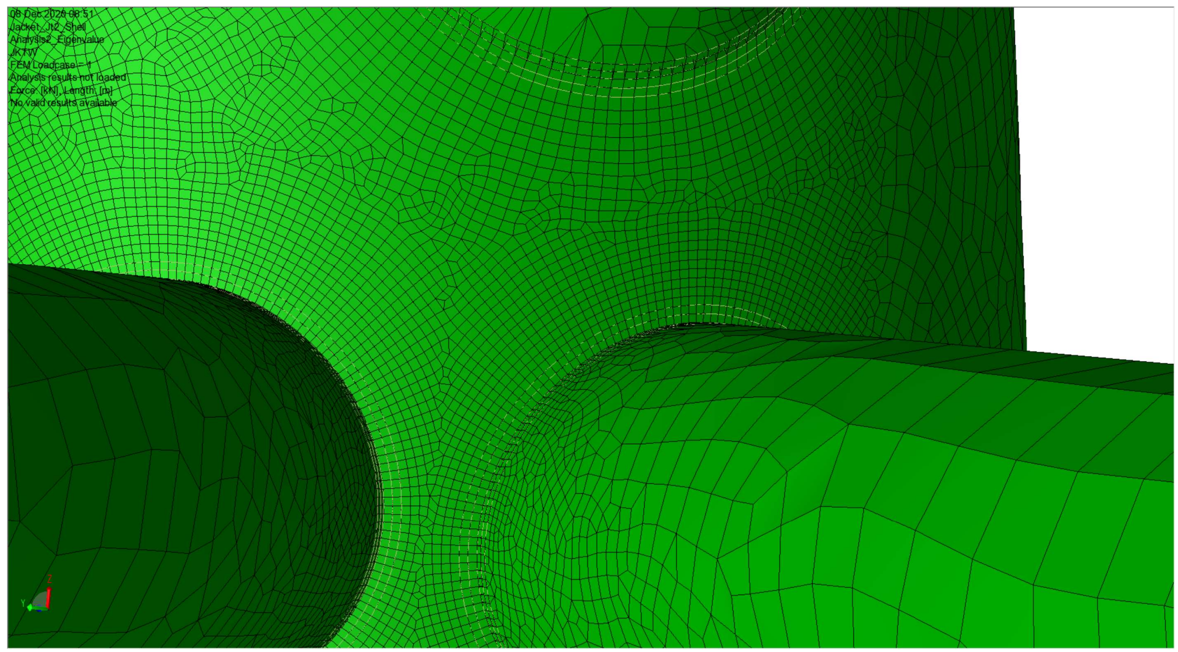


DNV

Fatigue analysis of Jt2

- A refined mesh is generated with total zone 30 mm and 2 layers -> mesh size 15mm
- The set Jt2_MeshRefinementZone is automatically made



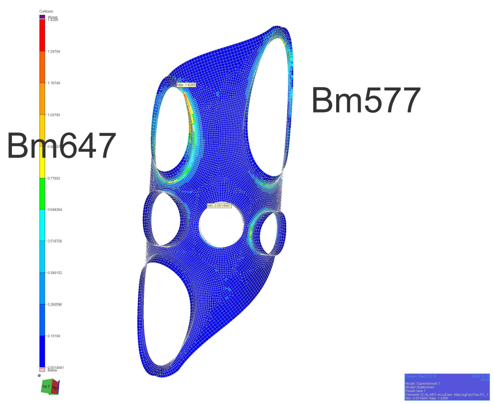


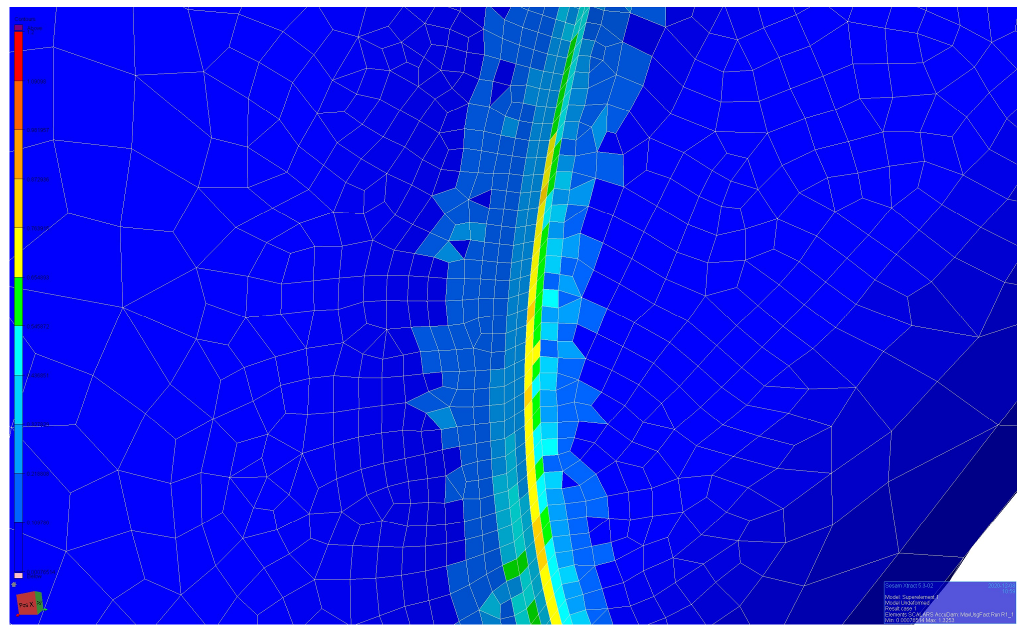
DNV

Fatigue analysis of Jt2

- Fatigue life – and compared with beam model
  - Life time is set to 20 years
  - Fatigue life found from mesh layer no. 2

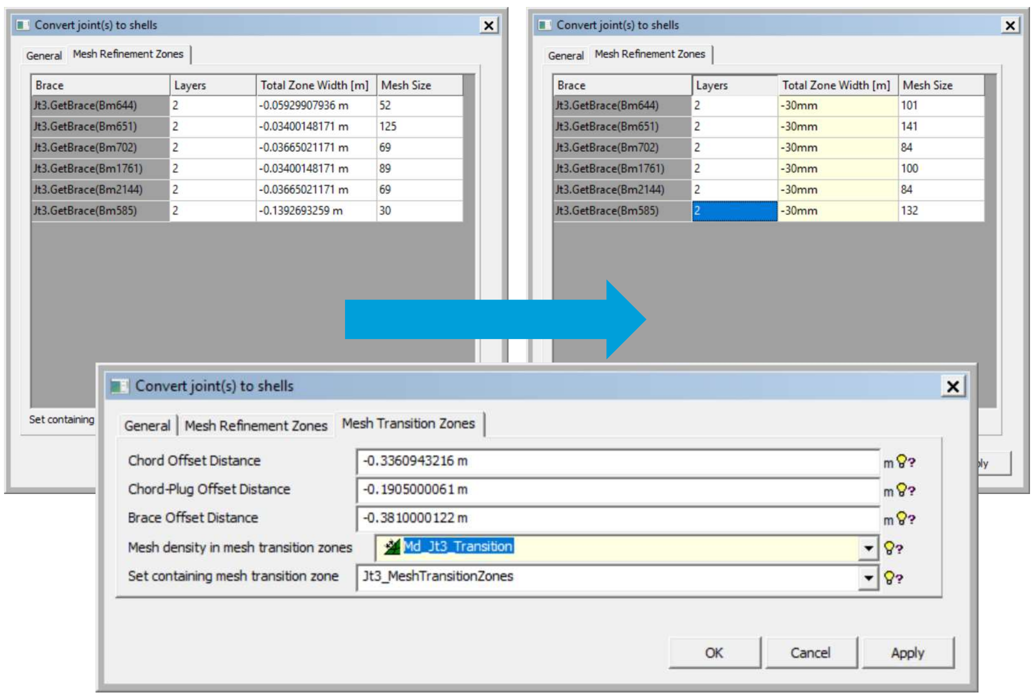
	Jt2 – Beam model	Jt2 – thick shell
Max Uf	6.91 (Bm577)	0.798
	5.02 (Bm647)	0.777





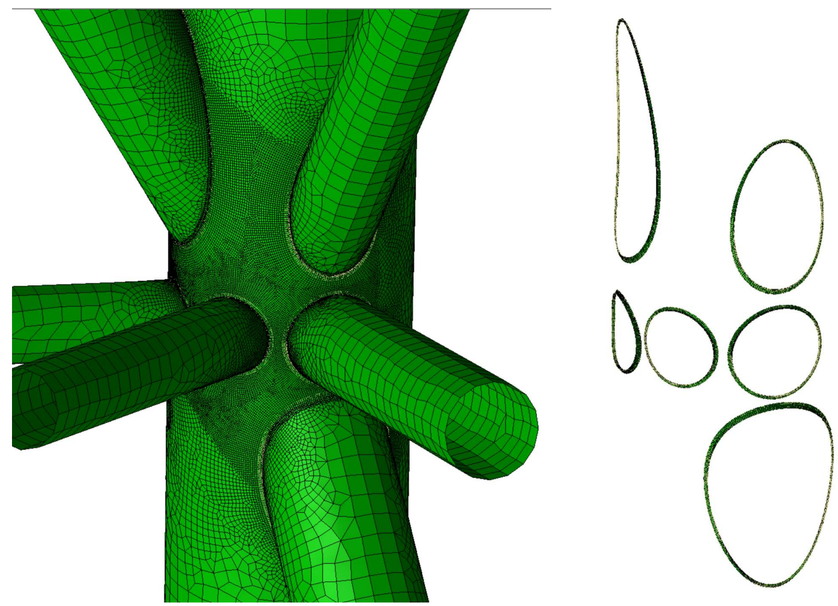
Fatigue analysis of Jt3

- A refined mesh is generated with total zone 30 mm and 2 layers -> mesh size 15mm
- The set Jt3_MeshRefinementZone is automatically made



Brace	Layers	Total Zone Width [m]	Mesh Size
Jt3.GetBrace(Bm644)	2	-0.05929907936 m	52
Jt3.GetBrace(Bm651)	2	-0.03400148171 m	125
Jt3.GetBrace(Bm702)	2	-0.03665021171 m	69
Jt3.GetBrace(Bm1761)	2	-0.03400148171 m	89
Jt3.GetBrace(Bm2144)	2	-0.03665021171 m	69
Jt3.GetBrace(Bm585)	2	-0.1392693259 m	30

Brace	Layers	Total Zone Width [m]	Mesh Size
Jt3.GetBrace(Bm644)	2	-30mm	101
Jt3.GetBrace(Bm651)	2	-30mm	141
Jt3.GetBrace(Bm702)	2	-30mm	84
Jt3.GetBrace(Bm1761)	2	-30mm	100
Jt3.GetBrace(Bm2144)	2	-30mm	84
Jt3.GetBrace(Bm585)	2	-30mm	132



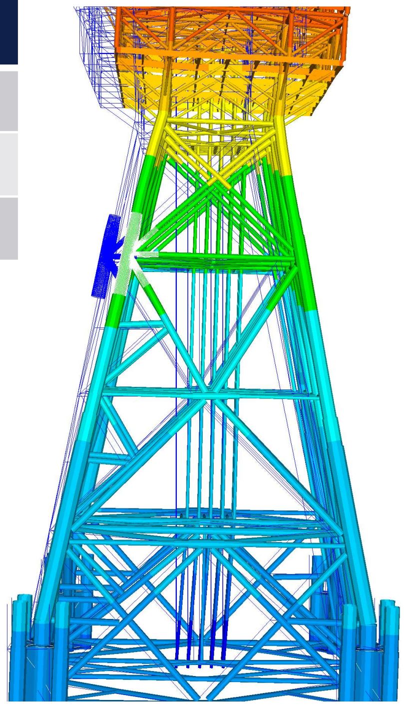
The set Jt3_MeshRefinementZone is meshed first

Fatigue analysis of Jt3

- Eigenvalues – and compared with the original beam model and the models with Jt1 & Jt2 converted to shell

Mode	Beam model	Jt1 – 100 mm 1st order thick shell	Jt2 1st order thick shell	Jt3 1st order thick shell
T1	2.616 sec	2.604 sec	2.598 sec	2.598 sec
T2	2.517 sec	2.506 sec	2.500 sec	2.498 sec
T3	1.658 sec	1.652 sec	1.647 sec	1.648 sec

- Meshing time 85 sec
  - 33107 nodes
  - 35599 elements
  - 198690 DOF
- Analysis time 71 sec
  - Sestra v10.13
- Result file size 0.092 GB



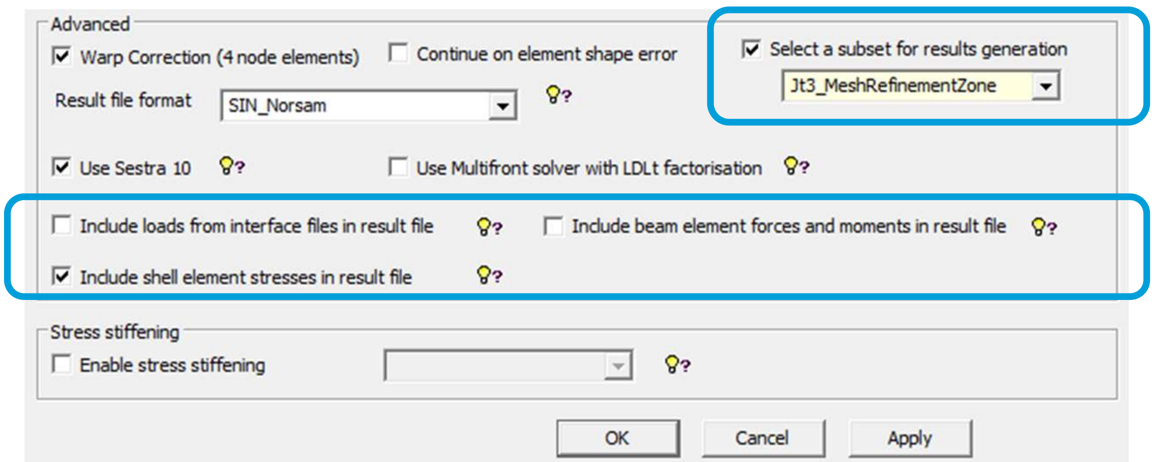
1st eigenmode

DNV

The smaller result file the faster fatigue analysis will be!

Fatigue analysis of Jt3

- Limit the data to be used in fatigue calculation to the refined mesh zone
  - In analysis set-up specify that Jt3_MeshRefinementZone shall be the only subset for results generation ("the SIN file")
  - Reducing SIN file size from typically 2.38 GB to 0.77 GB
- Reduce the amount of data exported to the result file
  - There is no need to include loads from wave load analysis nor beam element forces/moments when performing shell fatigue
  - This leads to a further reduction of the SIN file size from typically 0.77 GB to 0.42 GB

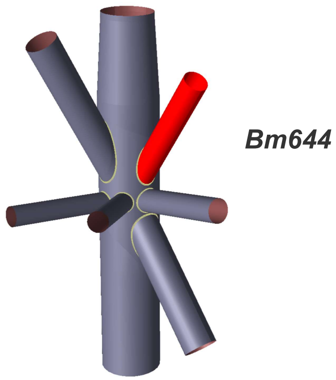


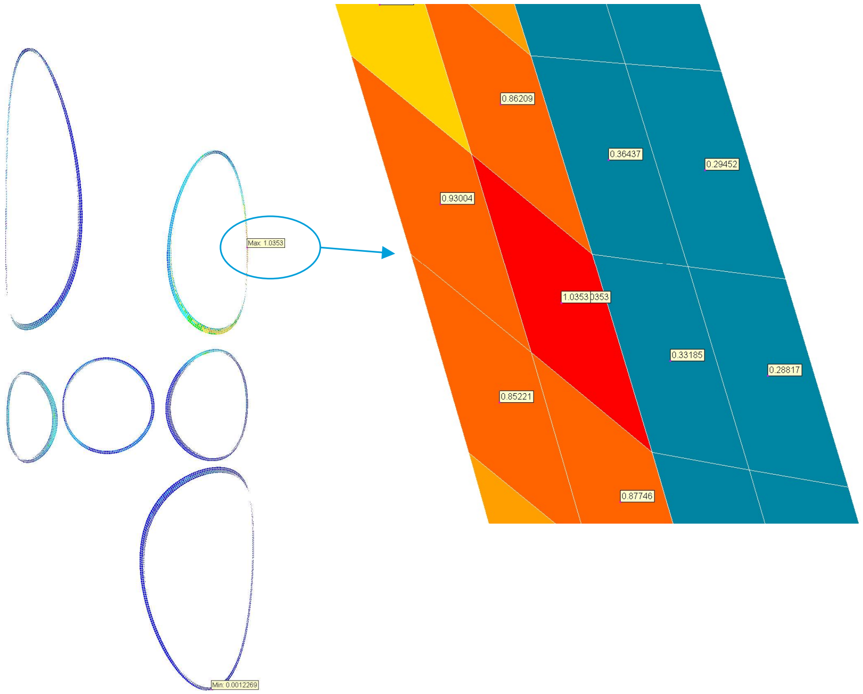
A further reduction of the result file size can be achieved by not exporting nodal displacements (NOPT card on Sestra input file)

Fatigue analysis of Jt3

- Fatigue life – and compared with beam model
  - Life time is set to 20 years
  - Fatigue damage is found from layer no. 2
  - Dynamic analysis time 431 sec
  - Fatigue calculation time 27 sec

	Jt3 – Beam model	Jt3 – thick shell
Max Uf	7.74 (Bm644)	0.93





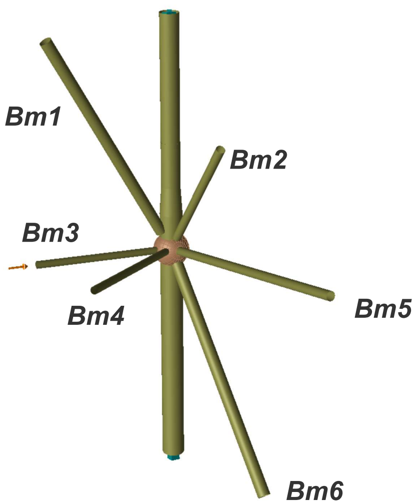
This example shows how easy it is to calculate SCF's using the "Convert joint to shell command"

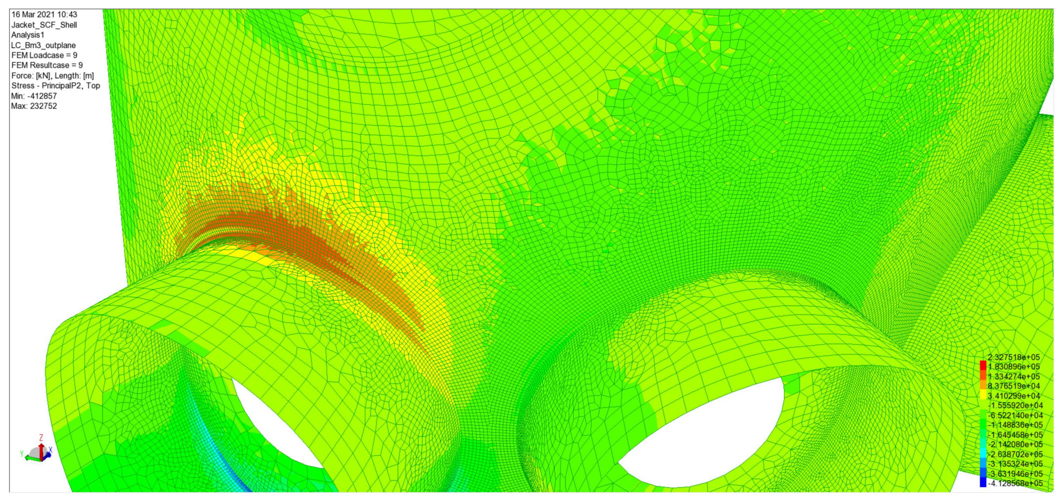
Fatigue analysis of Jt3 based on calculated SCF

- Calculating the SCF is based on axial force, in-plane & out-of-plane moments
  - Find the ratio between nominal (principal) stresses in shell model and beam model
  - Use a local model and fix the chords
- The SCF is highly dependent on the selection of mesh density
  - This example uses the recommendations from DNV RP-C203 Chapter 4.2: Mesh density = 0.1 * SQRT(radius brace * thickness brace)
  - There are probably other ways of doing this in the industry

General | Mesh Refinement Zones | Mesh Transition Zones

Brace	Layers	Total Zone Width [m]	Number of Elements	Total Zone Max-Width [m]
Jt3.GetBrace(Bm1)	2	-27.8mm	142	-0.1455943155 m
Jt3.GetBrace(Bm2)	2	-21.4mm	140	-0.06343445291 m
Jt3.GetBrace(Bm3)	2	-15.6mm	159	-0.04055220503 m
Jt3.GetBrace(Bm4)	2	-14.0mm	177	-0.04055220503 m
Jt3.GetBrace(Bm5)	2	-17.2mm	173	-0.04093271495 m
Jt3.GetBrace(Bm6)	2	-24.0mm	176	-0.04093271495 m





Fatigue analysis of Jt3 – beams and calculated SCF

- SCF calculation for this particular joint
  - This example is based on finding max SCF around the circumferential at position inner side of the second mesh layer
  - When using these SCF on Jt3 we see that fatigue life improves from 7.74 to 6.01

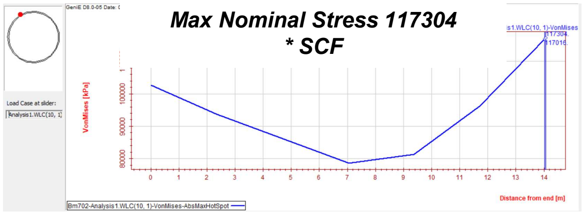
	Load type	Max nominal stress Beam	Diameter (mm)	Thickness (mm)	Mesh size (mm)	Max nominal stress shell (P1) Brace	Max nominal stress shell (P2) Brace	Max nominal stress shell (P1) Chord	Max nominal stress shell (P2) Chord	SCF Brace (P1)	SCF Brace (P2)	SCF Chord (P1)	SCF Chord (P2)	Max SCF Brace	Max SCF Chord
Bm1	Axial	8522	1200	32	13.9	16840	30757	16589	32573	1.98	3.61	1.95	3.82	**3.61**	**3.82**
	In-plane	29940				66573	44303	65095	46189	2.22	1.48	2.17	1.54	**2.22**	**2.17**
	Out-plane	29940				47676	48235	57236	57678	1.59	1.61	1.91	1.93	**1.61**	**1.93**
Bm2	Axial	14330	914	25	10.7	54696	30089	41746	25596	3.82	2.10	2.91	1.79	**3.82**	**2.91**
	In-plane	66200				123180	144384	93379	97808	1.86	2.18	1.41	1.48	**2.18**	**1.48**
	Out-plane	66200				139339	104751	118139	119202	2.10	1.58	1.78	1.80	**2.10**	**1.80**
Bm3	Axial	26670	762	16	7.8	77377	161250	91973	101186	2.90	6.05	3.45	3.79	**6.05**	**3.79**
	In-plane	146000				510886	517042	395556	391058	3.50	3.54	2.71	2.68	**3.54**	**2.71**
	Out-plane	146000				363428	367948	236091	212699	2.49	2.52	1.62	1.46	**2.52**	**1.62**
Bm4	Axial	33610	762	13	7.0	76007	160125	80799	92473	2.26	4.76	2.40	2.75	**4.76**	**2.75**
	In-plane	177600				210590	516985	373107	363216	1.19	2.91	2.10	2.05	**2.91**	**2.10**
	Out-plane	177600				391178	392212	204745	211960	2.20	2.21	1.15	1.19	**2.21**	**1.19**
Bm5	Axial	22150	914	16	8.6	127360	64395	78364	72198	5.75	2.91	3.54	3.26	**5.75**	**3.54**
	In-plane	100400				368716	360668	269306	267179	3.67	3.59	2.68	2.66	**3.67**	**2.68**
	Out-plane	100400				242273	239687	134147	137856	2.41	2.39	1.34	1.37	**2.41**	**1.37**
Bm6	Axial	12100	1300	22	12.0	52354	30081	32767	24780	4.33	2.49	2.71	2.05	**4.33**	**2.71**
	In-plane	36030				73075	65129	44103	49886	2.03	1.81	1.22	1.38	**2.03**	**1.38**
	Out-plane	36030				112658	95414	84414	84591	3.13	2.65	2.34	2.35	**3.13**	**2.35**

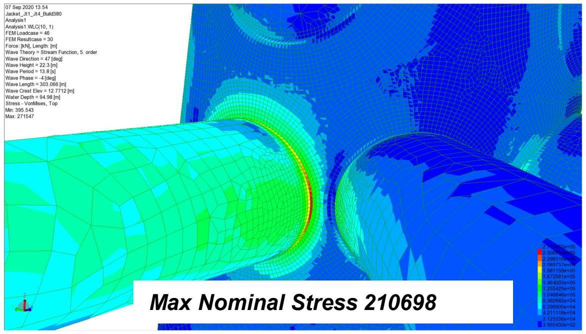
DNV

Summarizing the results for Jt3

Disclaimer: The results presented herein cannot be used to make general conclusions. However, several research projects show that empirical SCF may be on the conservative side

- All fatigue computations based on same wave loads and stochastic dynamic analysis
- This case study is based on one jacket and one joint only and cannot be used to make a conclusion that the calculated SCF or shell fatigue will always give better fatigue life than the use of empirical SCF
  - The load re-distribution is highly dependent on variations in stiffness and vary for each jacket
- Stresses used in shell fatigue or when deriving SCF are highly dependent on the mesh sizes
  - Calculating fatigue is sensitive to variation in stress. Typically, a stress increase of 20% or 50% leads to an increase of fatigue damage of 2.5 and 7.5 respectively
  - The recommendations of DNV RP-C203 Chapter 4.2 has been used in this study
- The fatigue life of Jt3 from the various analysis performed in this study (lifetime set to 20 years)
  - Beam fatigue and the use of empirical SCF: 7.74 (failure)
  - Beam fatigue and the use of calculated SCF: 6.01 (failure)
  - Shell fatigue: 0.93 (pass)

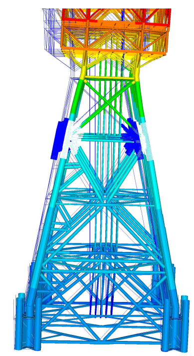


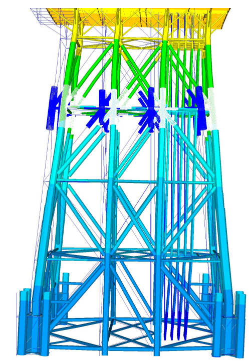


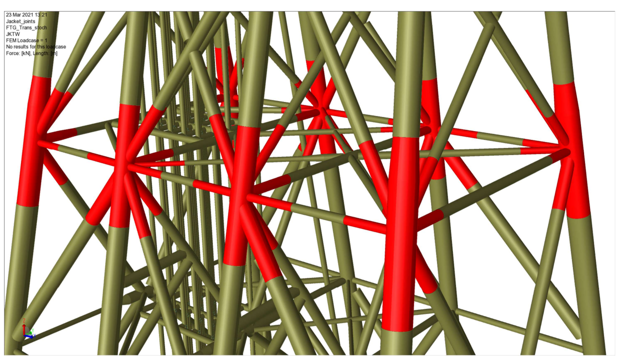
In many cases one needs to convert more than one joint to ensure symmetry and proper load re-distribution

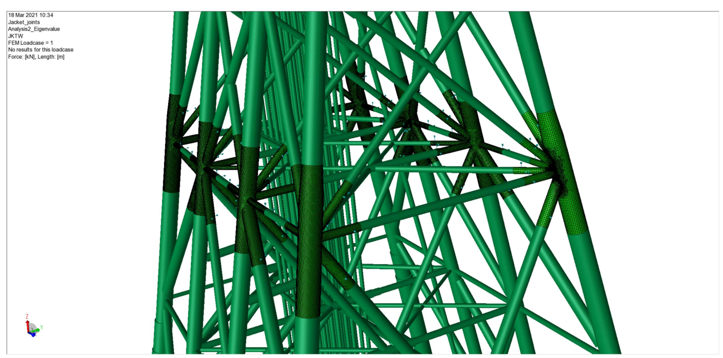
Dynamic analysis with 8 joints converted

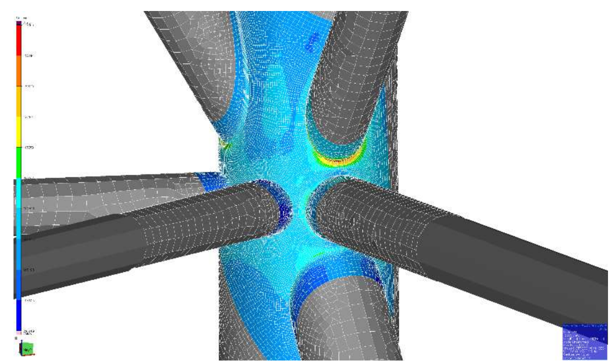
- Jt3 – detailed, rest of joints with 100mm mesh density and no refined mesh zones
  - 184 464 elements
  - 181 459 nodes
  - **1 089 204 DOF (close to 1.1 million DOF)**
  - Eigenvalue analysis 1370 sec
  - Eigenvalues consistent with beam eigenvalue analysis
  - Dynamic analysis time 5477 sec
  - Fatigue result of Jt3 consistent with analysis with one joint converted









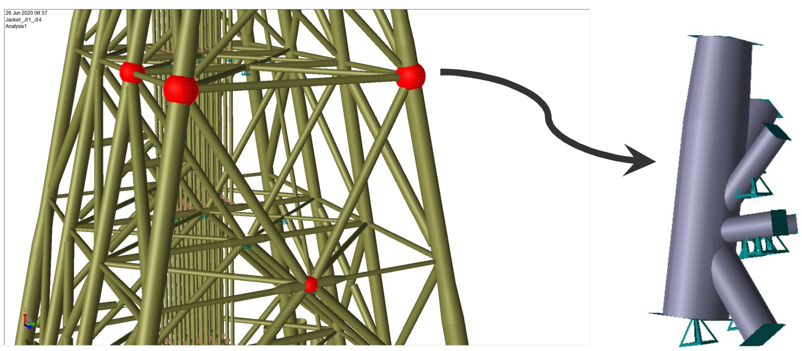


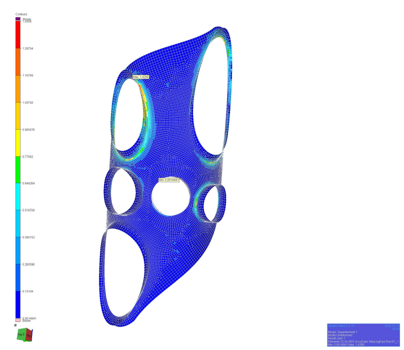

Summary

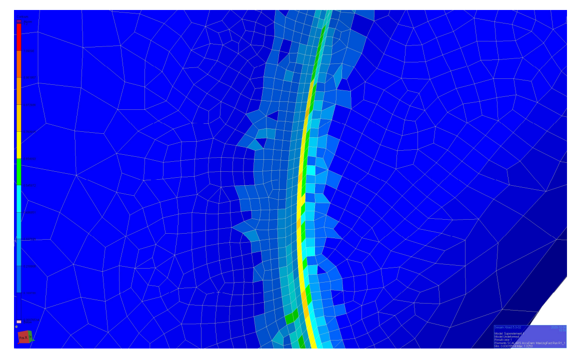
The new functionalities in Sesam have resulted in reducing
- editing work from days to a few minutes when converting a tubular joint
- dynamic analysis time in the order of 50%
- stochastic fatigue analysis time down from typically 30 min to 30 sec

Summary

- Tubular joints can be converted with one command satisfying the mesh requirements of DNV RP-C203 to do fatigue analysis of shell or to calculate SCF's
- The shell model is automatically connected to the jacket by use of rigid links (master slave)
- The hydrodynamic loading of converted joints is covered by use of non-structural members
- The dynamic analysis is faster because of a new version of Sestra and exporting only relevant structure and result components
- Stochastic fatigue is significantly faster because fatigue is performed on selected parts only
- This use case shows an increase of fatigue lives using shell fatigue or calculated SCF's as compared to beam fatigue using empirical SCF







WHEN TRUST MATTERS

Sesam™ – Shell fatigue analysis of tubular joints

How to convert tubular joints to shell models for use in shell fatigue analysis

www.dnv.com

DNV