WHEN TRUST MATTERS

SESAM EXAMPLE
Shell fatigue of tubular joints

WORKSHOP

Shell fatigue of tubular joints

Training Workshop

Date: 6 December 2024

Prepared by DNV – Digital Solutions

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

E-mail sales: software@dnv.com

Table of Contents

1 INTRODUCTION 1

2 METHODOLOGY AND DISCUSSIONS 2
2.1 Background 2
2.2 Fatigue analysis by use of calculated SCF's 3
2.3 Fatigue analysis by use of shell theory 3

3 MODEL INFORMATION 5

4 CREATE THE PROJECT 6

5 STEP 1: BEAM EIGENVALUE AND DYNAMIC ANALYSIS 7

6 STEP 2: BEAM FATIGUE ANALYSIS 12

7 STEP 3: CONVERT JT3 TO SHELL MODEL, EIGENVALUE AND DYNAMIC ANALYSIS 13
7.1 Eigenvalue analysis 25
7.2 Stochastic dynamic analysis 28

8 STEP 4: SHELL FATIGUE ANALYSIS 31

9 EXAMPLE DATA FILES 34

10 DISCLAIMER 34

1 INTRODUCTION

This workshop addresses how to use shell fatigue as compared to beam fatigue to achieve more accurate calculated fatigue life by use of FE shell methodology. It is common understanding that shell elements provide more accurate dynamic stress ranges for fatigue calculations than stresses calculated by beam theory and multiplied with parametric SCF. The main limitations lie in

- Parametric SCF represent planar parts of the tubular joint, and they normally don't reflect complex tubular joints
- The stress calculation around a tubular cross section is limited to 8 positions, and these may not capture the worst occurrence of stress
- A shell model allows for flexibility between braces and chord. In a beam model, this can only partially be handled by use of hinges
- The shell model allows for ovalisation of the braces
- A shell model easily allows for the inclusion of stiffeners represented by beams or shell elements

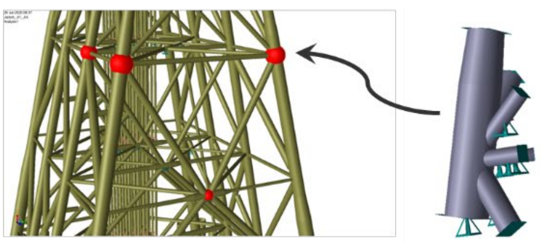
The workshop contains a structural global model for dynamic analysis and beam fatigue analysis. A part of the model is selected for calculating fatigue, and it has been found that the most critical joint is Jt3 with a fatigue damage of 6.1. This means that the calculated lifetime is just above 3 years (with design lifetime of 20 years).

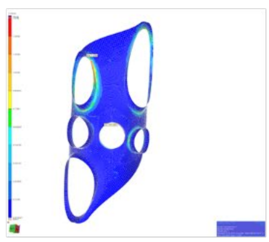
The second part of the workshop shows how to convert the tubular Jt3 to a shell model, how it is integrated in the global beam model, how to ensure that the eigenmodes are the same as for the pure beam model, and how to calculate fatigue life for the shell model. This workshop assumes a coarser mesh density for the shell model than required by DNV-RP-C203 Chapter 4.2. The workshop shell model will result in calculated fatigue life of 27 years while a shell model according to DNV-RP-C203 will yield a calculated fatigue life of 15 years (5 times better than the global beam model).

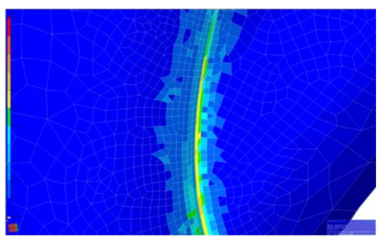
The Sesam modules Sesam Manager (v6.7-00), GeniE (v8.10-01 with extension CGEO), Wajac (v7.12-00), Sestra (v10-18.00), Framework (v4.4-00), Stofat (v4.1-02) and Xtract (v6.2-03) are used to create the workshop material. **See additional notes under Chapter 9**.

This workshop assumes knowledge in how to use all the above Sesam modules. The focus is to show the workflow, how DNV-RP-C203 fits into it and not modelling details.

It is also proven by this workshop that the amount of time spent is reduced to hours and not weeks as in legacy versions of Sesam or other tools.







2 METHODOLOGY AND DISCUSSIONS

This workshop highlights the significance of advanced methodologies, such as shell fatigue analysis, in improving the accuracy of fatigue life predictions. These methods can lead to more efficient designs, potentially reducing material usage and inspection frequencies, as evidenced by discrepancies between actual and predicted fatigue lives in historical analyses.

The use of DNV-RP-C203 recommendations, widely recognized in the industry, underscores the shift towards advanced fatigue analysis methods. This approach aims to enhance the fatigue life of critical tubular joints, impacting crucial decisions in design, operation, and life extension phases.

The methodology behind the fast and efficient conversion of tubular joints to shell models is described in

- DNV-RP-C203 Fatigue design of offshore steel structures, Sep 2021
- Adipec 2022, paper SPE-211257-MS: "A Study on Reliability-Based Assessment of Ageing Fixed Offshore Platforms Located in the Arabian Gulf", Abin P. Thomas and Ole Jan Nekstad, DNV
- Adipec 2023, paper SPE-216878-MS: "A Novel Methodology for Fatigue Analysis on Welded Non Tubular Connections on Offshore Platform Decks Using Shell Fatigue Method", Abin P. Thomas and Ole Jan Nekstad, DNV
- OTC 2024, paper OTC-35157-MS: "Enhancing the Precision of Fatigue Analysis for Crucial Jacket Tubular Joints", Ole Jan Nekstad and Abin P. Thomas, DNV
- GeniE tutorial A16 "Conversion of Tubular Joints", DNV

The below is an extract from the above references.

2.1 Background

Historically, fatigue analysis of tubular joints has evolved from simplistic beam theory and stress concentration factors (SCF) to more sophisticated three-dimensional finite element (FE) methods. The latter approach, less conservative by nature, plays a pivotal role in crucial decision-making processes, whether in new design ventures or in the reinforcement of existing structures during operation or life-extension.

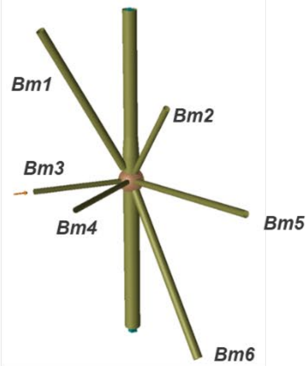
Interestingly, post-decommissioning investigations of jacket structures have revealed a discrepancy between predicted and actual fatigue failures. Joints forecasted to fail based on parametric SCF calculations were often found devoid of fatigue cracks, suggesting potential over-conservatism in existing methodologies. This observation raises a pertinent question: can advanced methodologies, such as 3D FE analysis, replace traditional parametric SCF calculations for critical joint assessments? While parametric SCF offers simplicity and efficiency, the creation and analysis of 3D FE models for shell fatigue are known for their complexity, often extending over weeks per tubular joint.

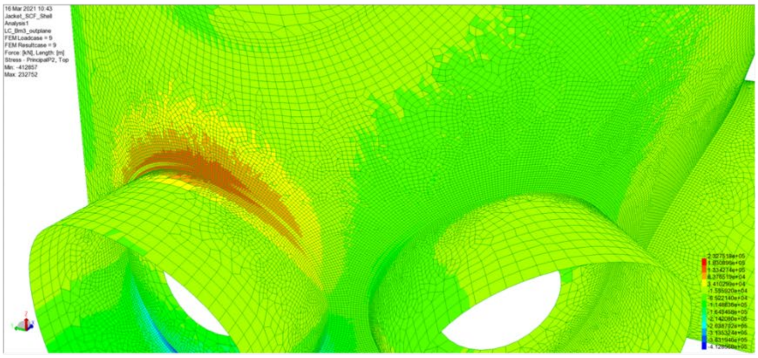
Addressing this challenge, the methodology presents a novel software functionality developed by DNV, in collaboration with major industry operators and design houses. This tool revolutionizes shell fatigue analysis by streamlining the creation of 3D FE models for tubular joints – a process that traditionally took weeks now condensed into mere hours. This advancement not only aligns with regulatory and recommended practices, such as DNV-RP-C203, but also significantly impacts fatigue analysis in terms of accuracy and efficiency.

2.2 Fatigue analysis by use of calculated SCF's

Calculating and using SCF (Stress Concentration Factor) instead of parametric SCF has been an industry practice for several years. Normally, such approach will improve the calculated fatigue life, but limited. There are case studies showing improvements in calculated fatigue life of 1.3.

The derivation of SCF for a tubular joint is based on the comparison of nominal stresses as a result from axial, in-plane and out of planes from a beam model and a 3D finite element model. This results in SCF factors for each brace and chord connections for axial, in-plane and out of plane loading situation. The SCF factors are normally the largest one from one of the eight hotspots around the circumference of a beam. The main difference from a parametric SCF calculation is that it will be based on finding SCF for the crown and saddle only – and not always reflecting the geometry stiffness and the acting stresses (axial and moment).





The SCF's are found from the ratio between principal stresses in shell and beam model.

This workshop does not address such methodologies, please contact software.support@dnv.com if you need workshop materials for this.

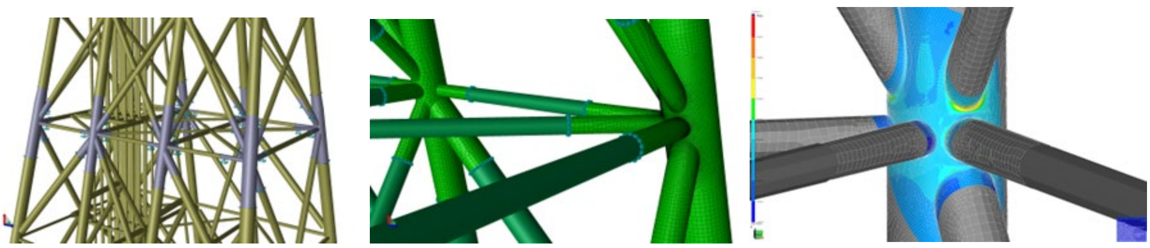
2.3 Fatigue analysis by use of shell theory

Fatigue analysis by use of shell theory requires that a tubular joint is converted to a finite element model as represented in the figure below. The methodology in this workshop is based on Chapter 4.2 of DNV-RP-C203. The requirements for the finite element mesh are as listed below.

1. The mesh shall be regular and quadratic to ensure there are no triangular elements present
2. The specification of mesh size will ensure that the location of the stress points around the circumferential are defined, and this will be different for the various brace/chord connections. The stress points for fatigue calculation shall be located at 0.1 x √(r x t) where r is radius of brace and t is thickness of brace. Note that stresses used in shell fatigue or when deriving SCF are highly dependent on the mesh sizes (or location of stress point). Typically, a stress increase of 20% or 50% leads to an increase of fatigue damage of 2.5 or 7.5 respectively (with the assumption single slope SN curve, m = 5).

3. When using shell elements, though 2nd order thick shell is recommended, 1st order thick shell theory is generally accepted for use in industry and followed in this study. 1st order thin shell formulations should be used with care as the shear is considered constant over the thickness.
4. There shall be minimum 3 finite elements between braces (on the chord) and the quality of the mesh on the chord inside a brace must have same quality as on the outside. This is not always possible when there is not enough space between braces (because of minimum 3 finite elements and requirement on mesh size) or there are overlapping braces. It is recommended to have at least 2 layers of regular mesh around each brace/chord connection.
5. The finite element model should have boundary conditions representing the stiffness of the connected geometry or be an integrated part of the global model enforcing a correct stiffness and load transfer.
6. A shell model of a tubular joint has more local flexibility and allows for ovalisation as compared to a pure beam model. It is therefore important to compare the eigenmodes between a beam model and the combined beam/shell model to ensure consistency between the modelling approaches. It may be necessary to convert all chord joints in an elevation to get comparable eigenmodes.
7. The wave base shear and overturning moments should be the same for a global beam model and a combined model including tubular joint(s) as FE model. Previous studies by the authors have shown that the stresses in a shell model could be underestimated with 19% if non-structural beams are not used to calculate wave forces for the converted joint. The dynamic wave forces from the non-structural beams are transferred to the shell model at their edges.
8. The split points for the shell model (where to split braces and chord) should be at least 3 times the diameter of chord and braces. This is also recommended by other researchers. At these positions rigid support points (also known as master-slave connections) should be inserted to ensure a consistent interface between the global beam model and the combined beam/shell model.

Sesam satisfies all the above requirements automatically. The conversion from a beam model to a shell model is made repetitive and fast by using a special purpose developed software algorithm part of the Sesam module GeniE.



An example where all tubular joints at one elevation has been converted to shell elements. In this case, one joint with refined mesh for fatigue assessments while the others have a coarser FE mesh density.

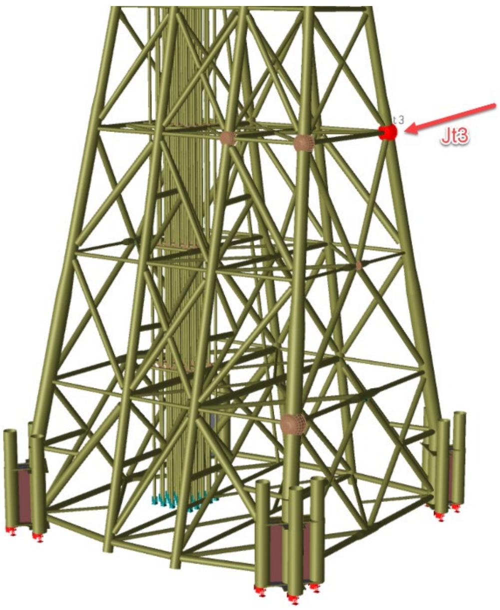
Note that for legs with inner piles and/or concrete grout, some more work is needed to make sure proper connection between shell model and inner pile (if relevant) and the use of equivalent thickness for the shell model (see DNV-RP-C203 for guidance).

3 MODEL INFORMATION

The jacket used in this workshop is a modified and simplified eight-legged jacket. It has linearized springs at ground elevation and there are 3 wave load conditions in frequency domain. Normally there would be wave directions for e.g. each 15 degrees. Note also that the wave load conditions have been set to result in fatigue failures.

The workshop is based on the following

- Dynamic analysis
- Linearized piles
- Modal superposition and master slave
- Eigenvalue analysis
- 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 (beam model)
    - T1 = 2.613 sec
    - T2 = 2.514 sec
    - T3 = 1.659 sec



- Fatigue damage (20 year life time) based on parametric SCF (Efthymiou) and dynamic beam analysis
    - Jt1 = 3.70 (failure)
    - Jt2 = 3.64 (failure)
    - Jt3 = 6.07 (failure)

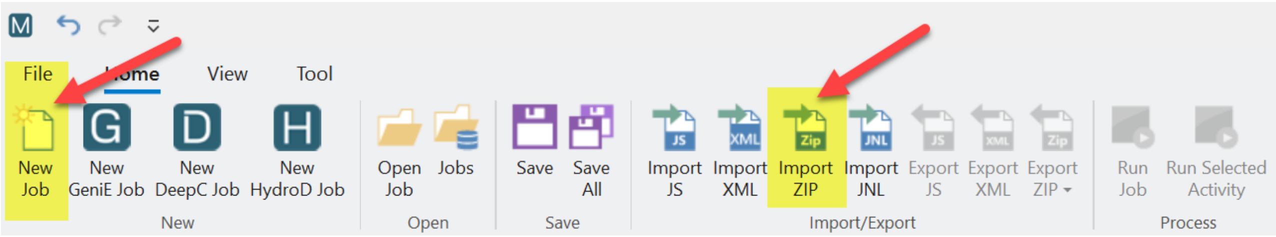
The objective of this workshop is to convert Jt3 to shell model and to perform a shell fatigue analysis. The steps are explained in the following. The same methodology can be used to calculate shell fatigue of other critical tubular joints too.

The input parameters for fatigue are examples and should be carefully examined if you want to use this example as basis for a real project. The beam fatigue input has no safety factors included, while the shell fatigue example has a weld stress concentration factor of 1.2 (see DNV-RP-C203 Chapter 4.2.1).

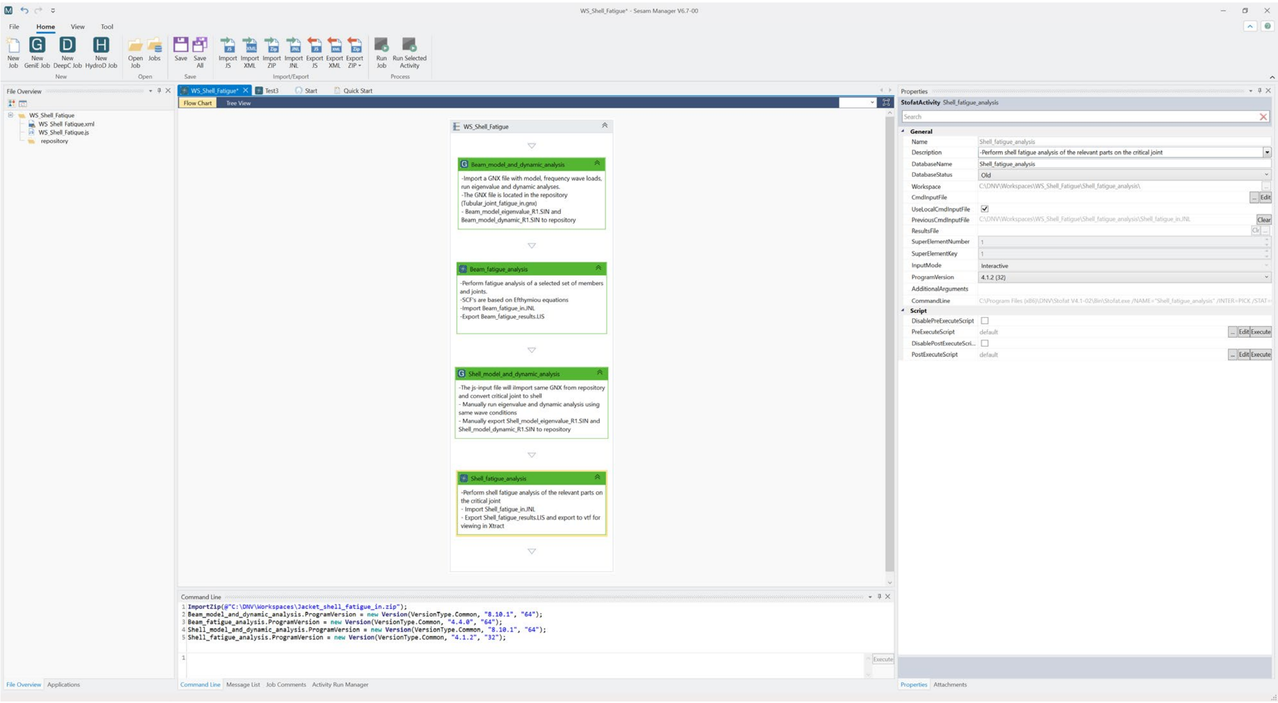
4 CREATE THE PROJECT

We start by using Sesam Manager to set up the workflow.

1. Open Sesam Manager, click on New Job and give it a name and specify a location if you want it to be somewhere else than the default location.



2. Click on Import ZIP and select the xml file that is part of this workshop, Jacket_shell_fatigue_in.zip.
3. You will now see the following in Sesam Manager:

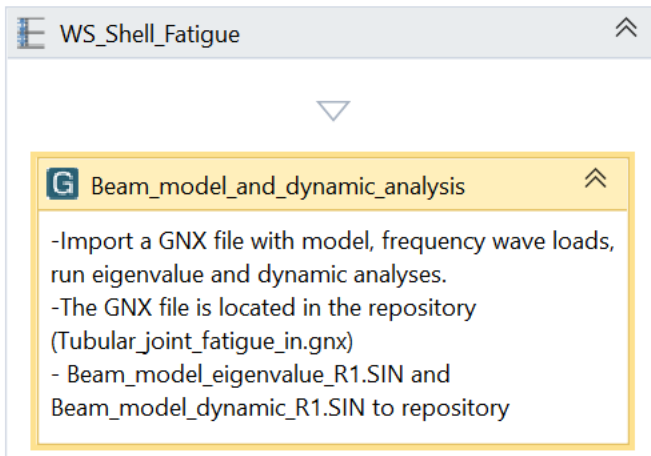


The workflow contains the following steps

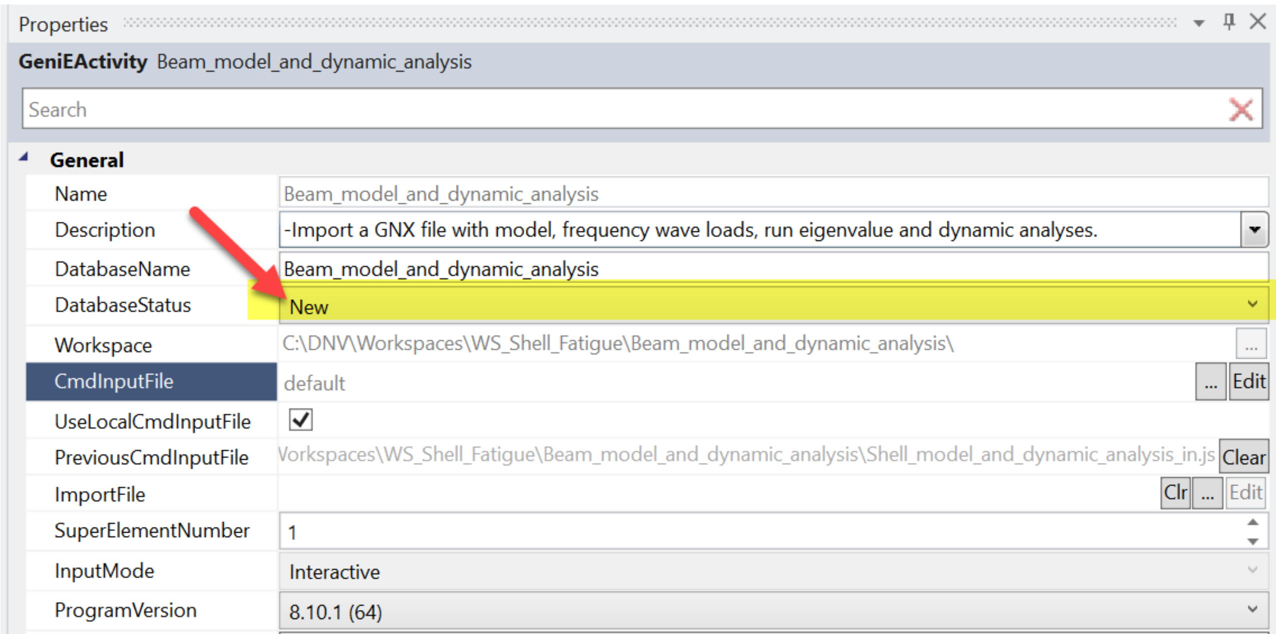
- Import a pre-made jacket model and perform eigenvalue and stochastic dynamic analysis
- Perform a beam fatigue analysis (based on results from beam analysis and parametric SCF)
- Convert Jt3 to shell model, perform eigenvalue analysis and compare with beam analysis, perform stochastic dynamic analysis
- Perform shell fatigue analysis of selected part of Jt3

5 STEP 1: BEAM EIGENVALUE AND DYNAMIC ANALYSIS

1. Click on the box

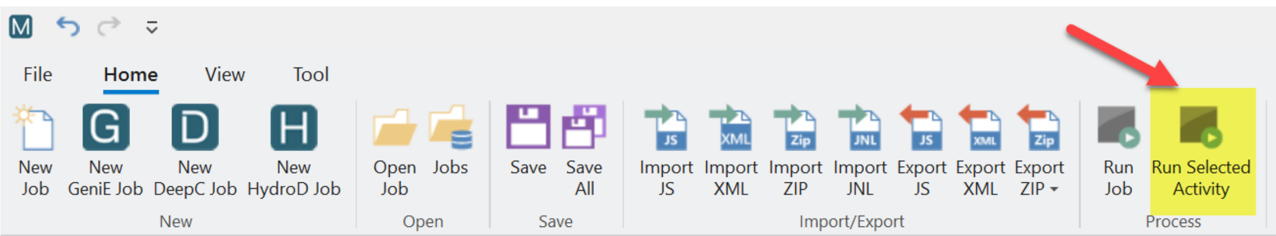


2. Edit the input data as follows:

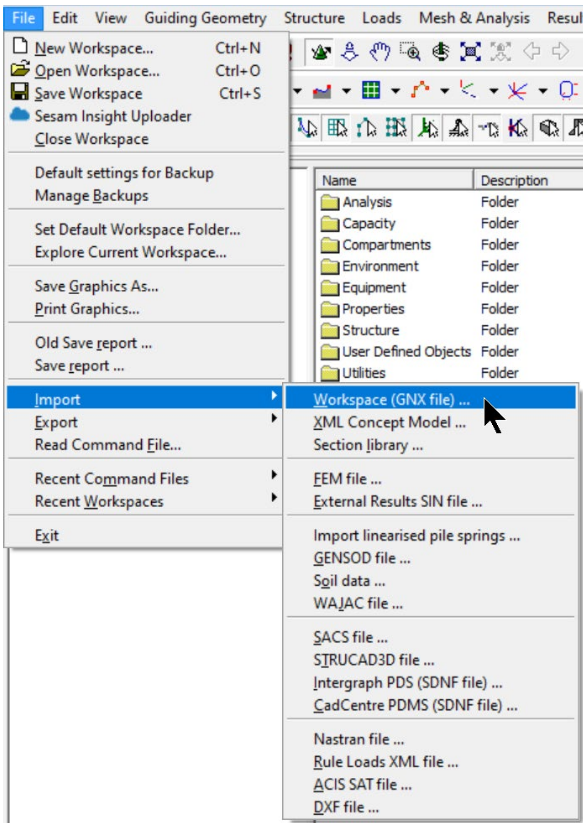


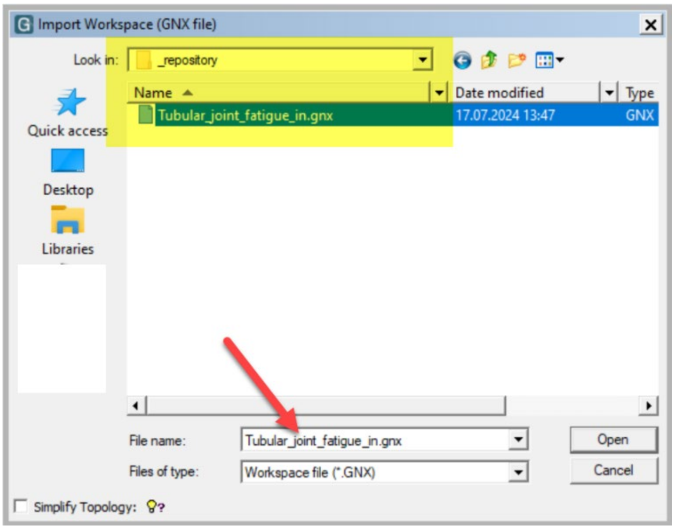
If you have later versions of GeniE, you may want to use it instead by referring to it.

3. Click on Run Selected Activity (alternatively right click the box and start run from the menu selection):

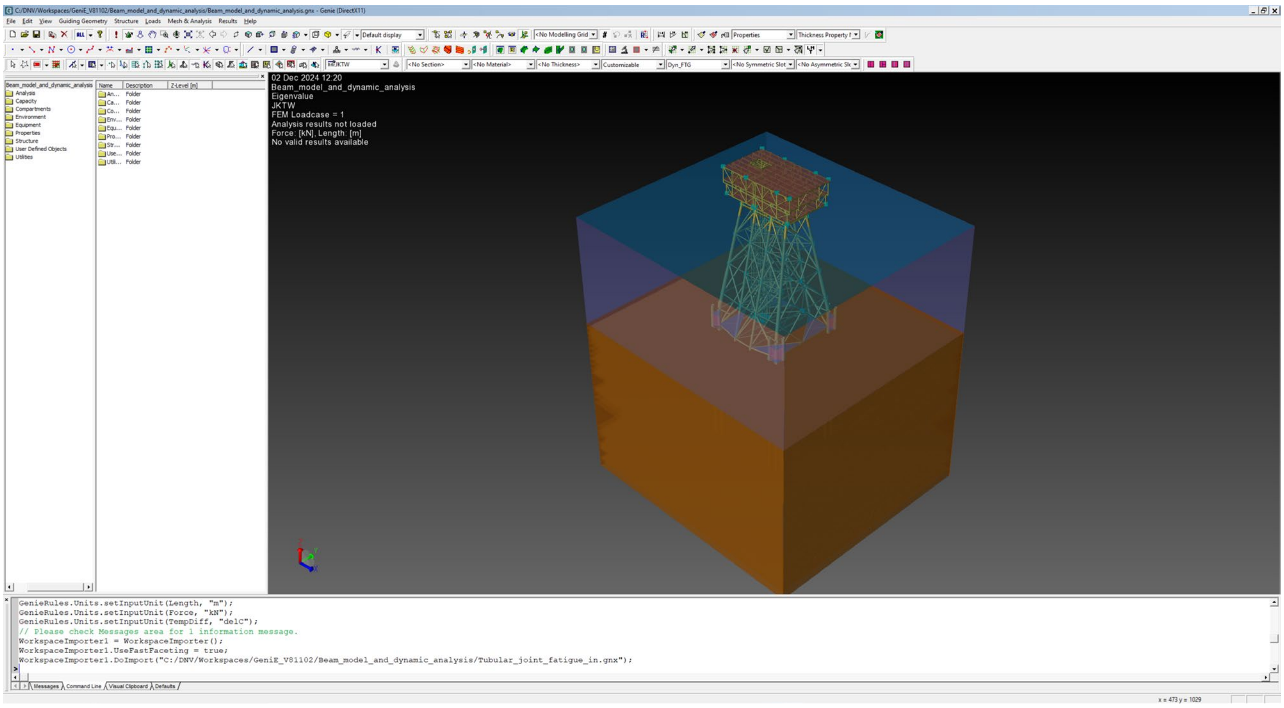


4. When GeniE opens, then import the gnx file from the repository

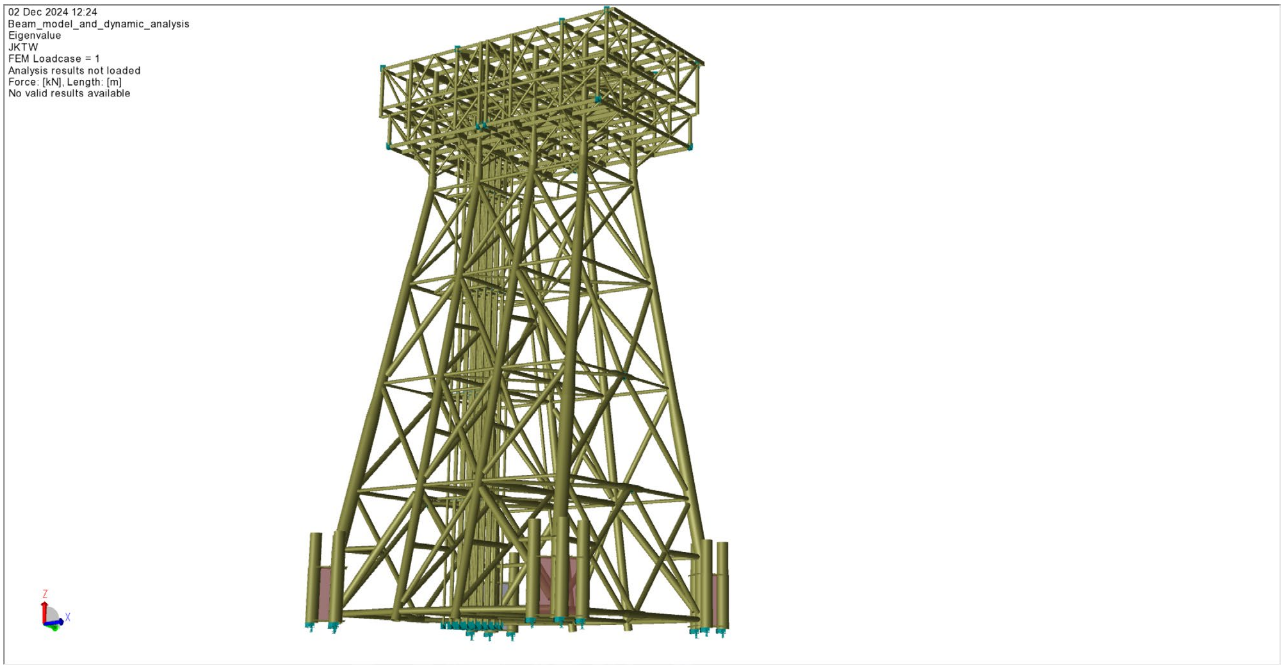




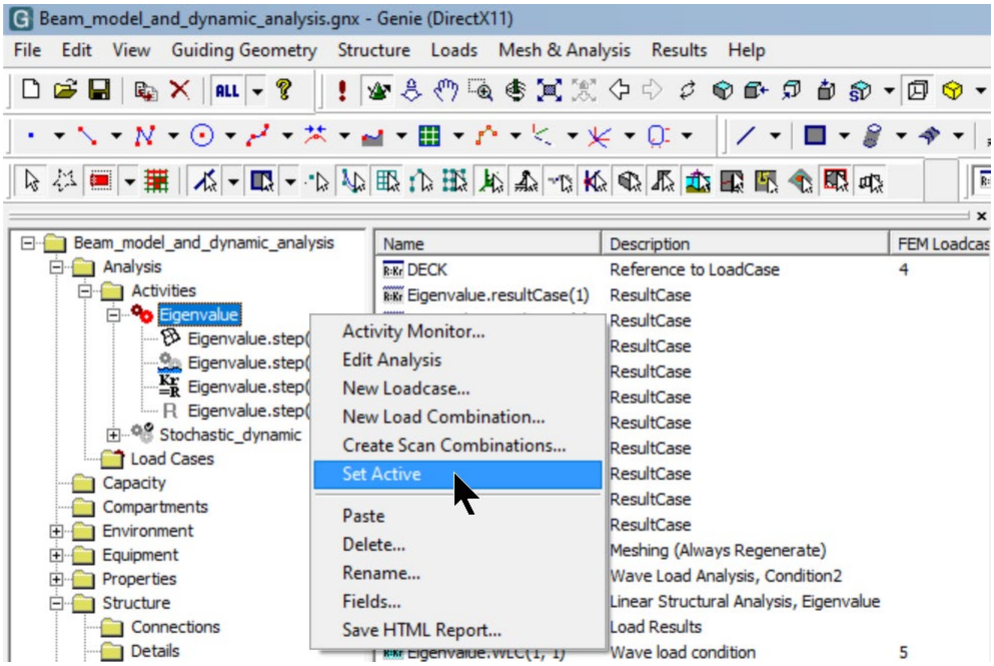
You will now see the model



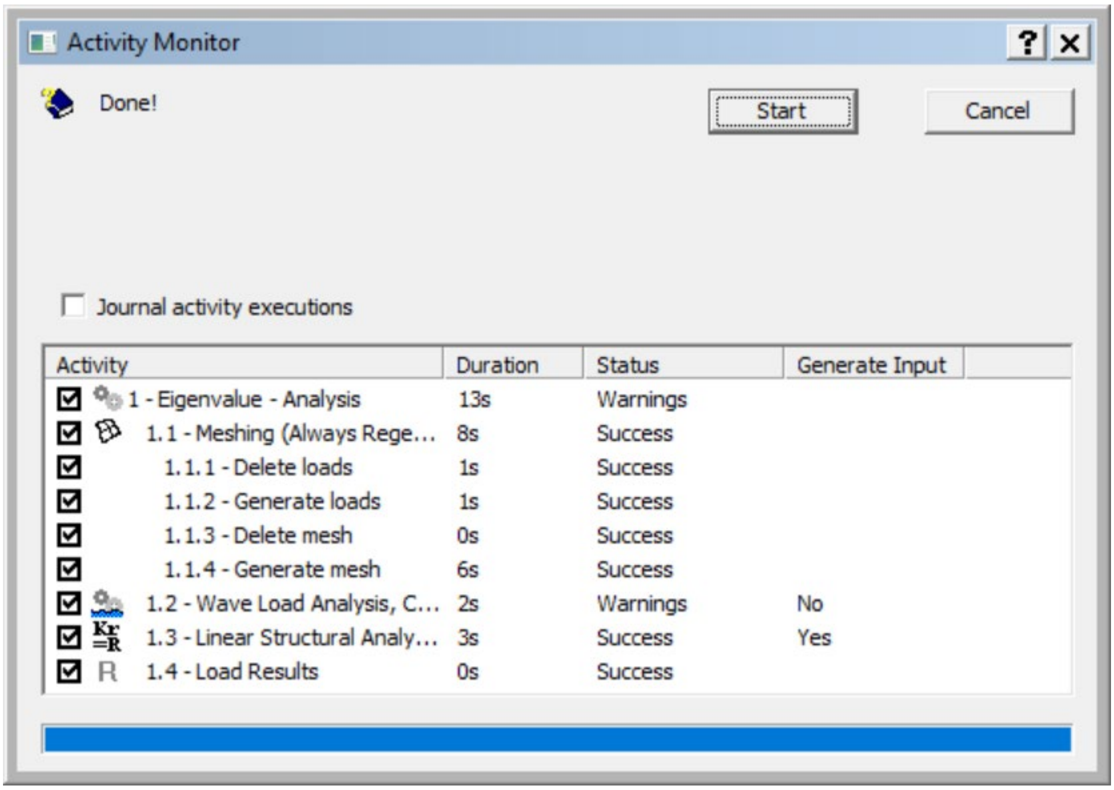
5. Select the named set Dyn_FTG and show only that selection. This set does not include the piles, but includes the linearized springs at the bottom of legs (the picture below is shown with white background).



6. Set the activity Eigenvalue to active.



7. Run the eigenvalue analysis from ALT+D or from pulldown menu Mesh & Analysis – Activity Monitor. This will start the eigenvalue analysis (10 mode shapes).

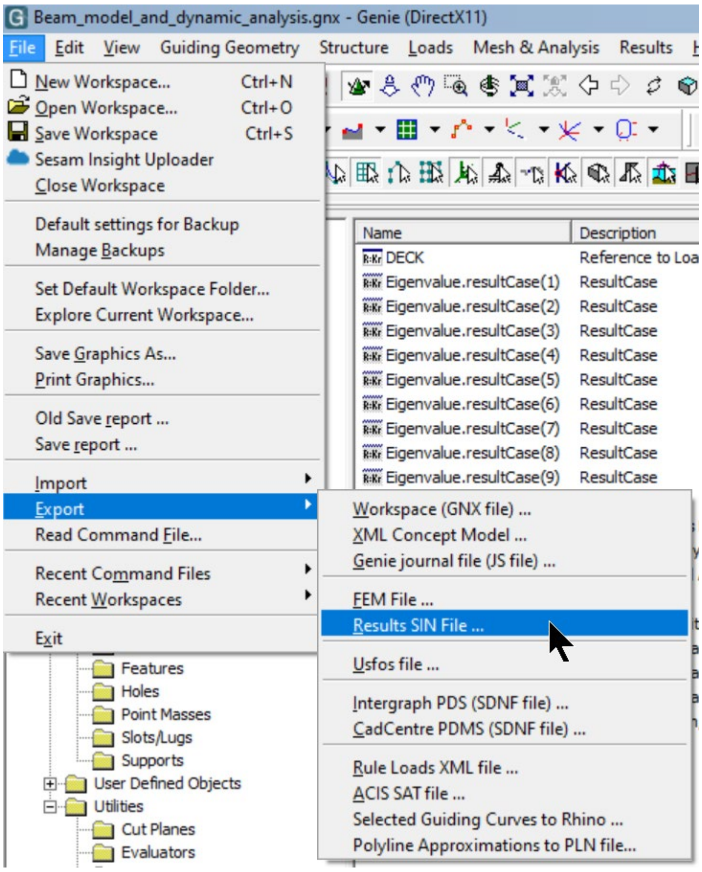


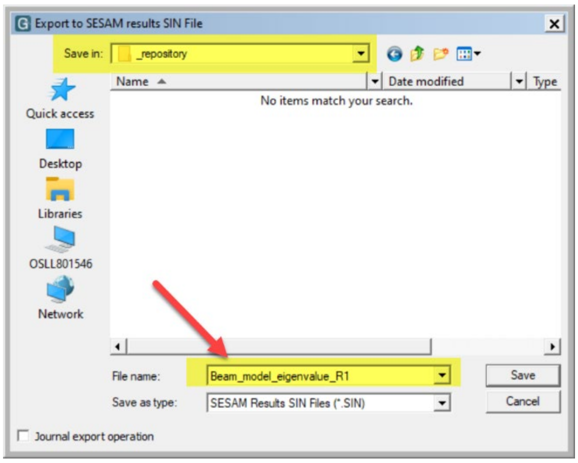
Investigate the results from the eigenvalue analysis from the Sestra listing file, GeniE results view or from using the advanced post-processor Xtract. These eigenvalues should be matched by the results from the model with joint converted to shell elements. See Step 7.

```
Print of eigenvalues.
Eigenvalues have unit sec^-2; frequency = sqrt(eigenvalue) / (2 * pi); period = 1 / frequency.
        Number;     Eigenvalue;      Frequency;         Period
             1;  5.781788e+00;  3.826936e-01;  2.613056e+00
             2;  6.249850e+00;  3.978826e-01;  2.513304e+00
             3;  1.433589e+01;  6.026046e-01;  1.659463e+00
             4;  3.936200e+01;  9.985245e-01;  1.001478e+00
             5;  4.229004e+01;  1.034997e+00;  9.661861e-01
             6;  5.482421e+01;  1.178437e+00;  8.485817e-01
             7;  8.764942e+01;  1.490029e+00;  6.711280e-01
             8;  1.218744e+02;  1.757019e+00;  5.691459e-01
             9;  1.260375e+02;  1.786775e+00;  5.596674e-01
            10;  1.874032e+02;  2.178756e+00;  4.589775e-01
```

- T1 = 2.613 sec
- T2 = 2.513 sec
- T3 = 1.659 sec

8. Export the result fil to the repository for later use. Use the name e.g. like shown below.





9. Set the other analysis activity Stochastic_dynamic to active and run analysis. It will do a dynamic analysis including 3 wave directions (0. 45 and 90 degrees) with the following wave frequencies for all directions. Follow the description of 6 and 7 above.

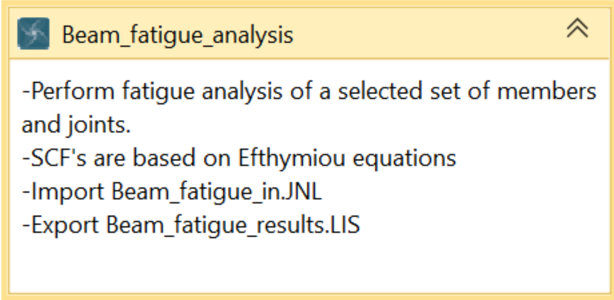
```
FRQ             34.
C                T1        T2        T3        T4        T5        T6        T7
FRQ             1.4       1.6      1.68      1.74      1.85       1.9      2.07
               2.10      2.15      2.20      2.25      2.30     2.348     2.379
               2.45       2.5      2.55      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
-
```

10. Upon completion of the analysis, export the result SIN file to repository. Follow the description of 8. **You must use the name Beam_model_dynamic_R1.SIN (this is referenced in the script file in the next Step).**

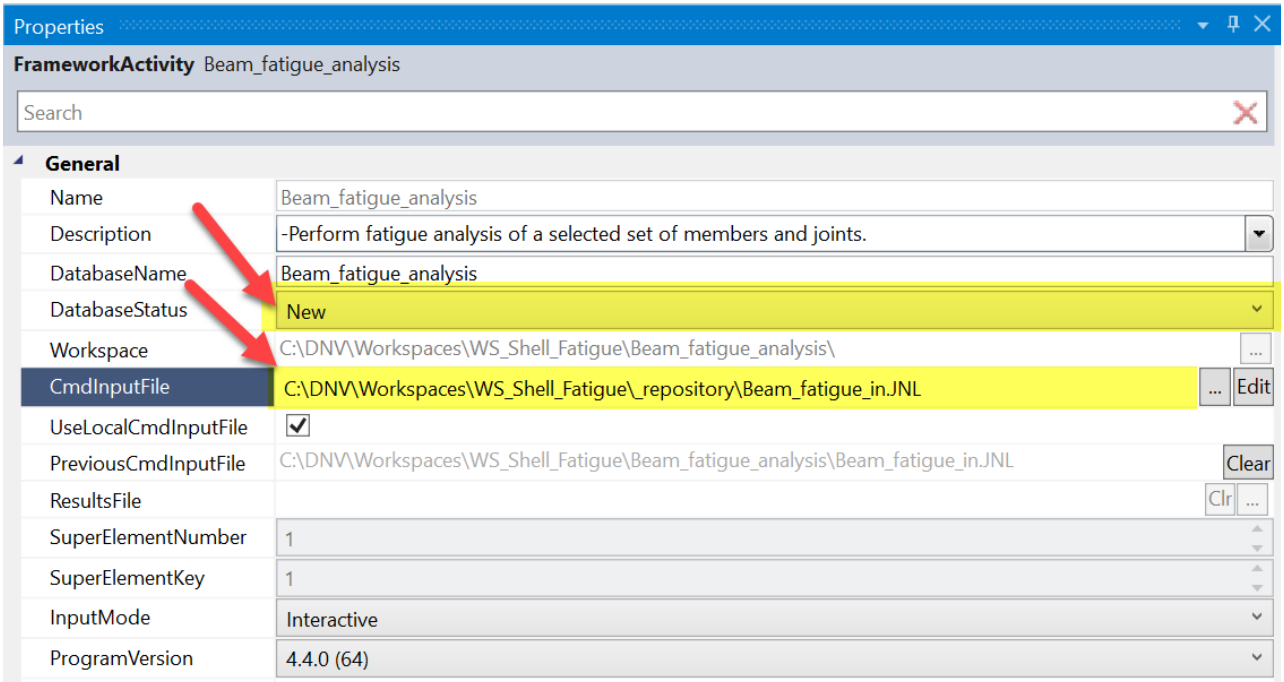
11. Exit GeniE (click ”Yes” to save the data)

6 STEP 2: BEAM FATIGUE ANALYSIS

1. Click on the box Click on the box



2. Edit the input data as follows. The database status must be set to "new" and the pre-defined input file for beam fatigue must be referenced to location "_repository".



The input file can be viewed and edited by you. The file is just an example of how input could be, and not necessarily reflecting a real project.

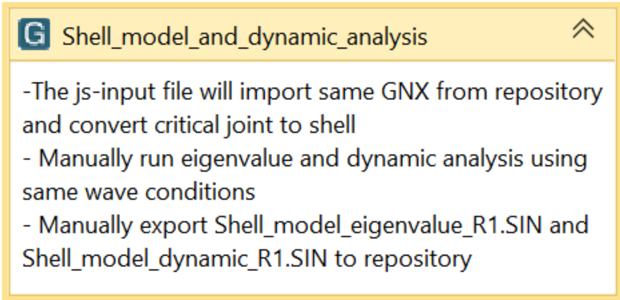
3. Click Run activity. When Framework has completed the fatigue calculations, exit the program and inspect the listing file Beam_fatigue_results.LIS. From the selection made for this project we see that the fatigue damages are

- Jt1 = 3.70 (failure)
- Jt2 = 3.64 (failure)
- Jt3 = 6.07 (failure)

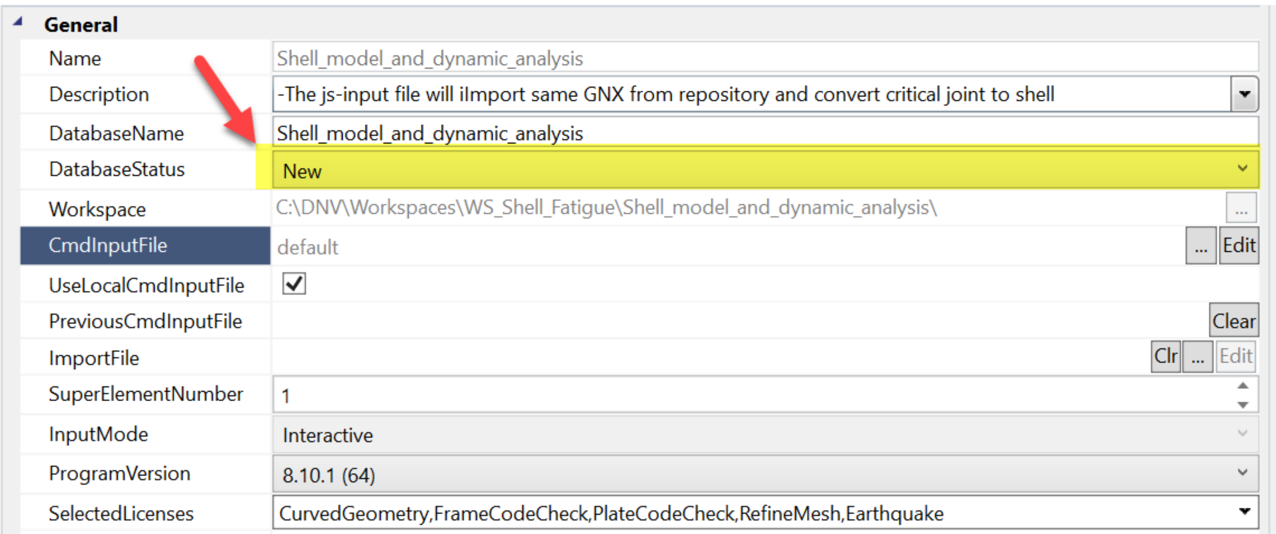
```
BM644     PIPE      JT3      **Fail**   6.07E+00
          LQT

BM563     PIPE      JT5      **Fail**   4.66E+00
          G_1_DB3

BM2339    PIPE      JT1      **Fail**   3.70E+00
          G_1_X16

BM647     PIPE      JT2      **Fail**   3.64E+00
          LQT
```

7 STEP 3: CONVERT JT3 TO SHELL MODEL, EIGENVALUE AND DYNAMIC ANALYSIS

1. Click on the box Click on the box

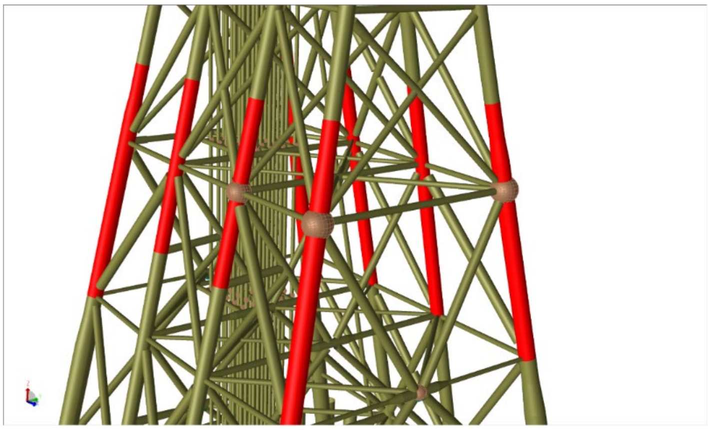


2. The input data for this task is the same as the previous GeniE session. This means that we set the database status to “new”.
    a. Notice that if you don’t want to do the manual steps to convert the tubular joint Jt3 to shell elements, you could edit the below and reference the file Shell_model_and_dynamic_analysis_in.js located in the _repository. This file will do most of the steps explained in this Chapter 7, but there are some that needs to be done by the user. Read the script file to see more.



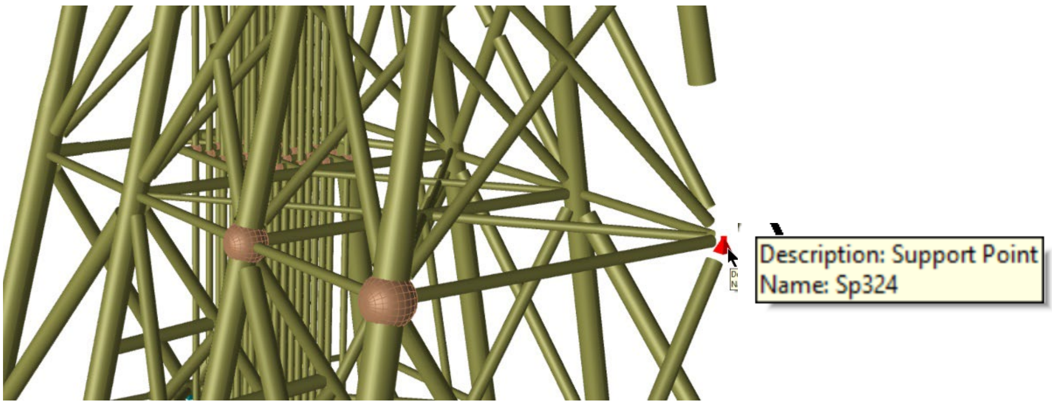
3. When GeniE opens, then import the GNX file from _repository “Tubular_joint_fatigue_in.gnx”

4. Before we start the conversion of a joint to shell model: make sure that the chords on both sides of a joint is joined together to one continuous member (it can be segmented). The reason for this is that this make sure there is no constraints when the finite element mesh is generated. For all the main joints (at each of the eight legs) you should select the chord member below and above and click join.
    a. Notice that the filter for active set is on and is referring to the set Dyn_FTG. This means that all changes to the model will be captured in the set Dyn_FTG. This set is used as reference in the meshing activity and will determine which structural parts are part of the analysis model.



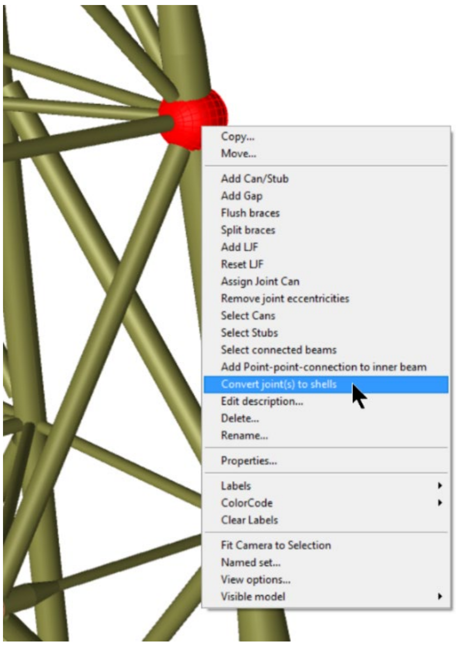
Select the chords shown above and join them. This workshop will cover conversion of one joint (Jt3) with detailed mesh, but in case you want to convert other joints with coarse mesh to improve mode shape behaviour you need to join the chords for these as well.

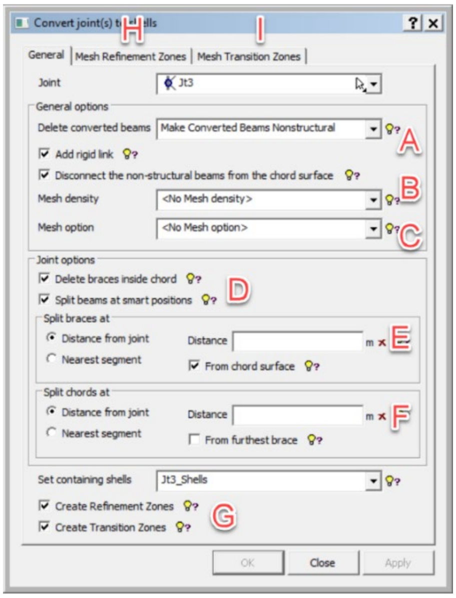
This model contains several supernodes for use in dynamic analysis. For the joints you convert, they must be deleted. This means that we need to delete the support point Sp324 (supernode) at location Jt3.



Notice that some members have been hidden to show the actual support point.

5. Before conversion of the joint can commence, you need to calculate the mesh densities around the brace (on chord side and along brace and on the plug inside the brace).
    a. According to DNV-RP-C203 Chapter 4.2, the stresses located at position 0.1 x √(r x t) where r is radius of brace and t is thickness of brace can be used in the fatigue calculations. For Jt3 this would mean a variety of mesh densities from 8mm to 18mm. Since this is a workshop where there is a need for fast calculation performance, 20mm mesh density is used for the refined mesh part.
        i. This will impact the calculated fatigue life. For 20 mm mesh density the calculated fatigue damage is 0.75, while it is 1.35 for the shell model with mesh according to DNV-RP-C203 Chapter 4.2. Similarly, it is 1.004 for 15 mm mesh density.
        ii. Analysis time for dynamic analysis 2225 sec and 700 sec for the two models in question. In other words, it is fast to do analysis of detailed model too.
    b. Notice also that Jt3 has no overlapping braces. If you want to learn how to convert tubular joints with overlapping braces, please see tutorial A16 inside GeniE.
6. Select joint Jt3, right click and use command “Convert join(s) to shells”. Below you will see the dialogue box that opens and the input to be provided is explained in the following





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 used in this workshop. For more details on how to just convert a joint for coarse finite element modelling, please see tutorial A16 inside GeniE. For this case, we specify the mesh density to 200mm. Select “New Mesh Density” to define the mesh size. Notice that we will be using linearized meshing for better mesh in transition zone. This example uses a factor of 1.05.

C. The selection of mesh option will instruct GeniE on which mesh algorithm to use. We recommend the use of “Patch Surf Quad Mesher” for the best mesh around chord/brace connections. This will give a rectangular mesh with low Jacobi determinant. Select “New Mesh Option” and tick off for “Patch Surf Quad mesher”. Remember to activate for linearized meshing as shown on the picture below.

D. The defaults will delete the beams inside the chord and braces that are converted. The non-structural beams are kept. Further, GeniE will make a geometry split of the circular shells so that they don’t interfere with the mesh generation. Normally, this will be on the back side of the chord from where the brace connections are.

E. Since the brace diameter is 1.3 m, we use the value 3.9 m (i.e., 3 times the brace diameter).

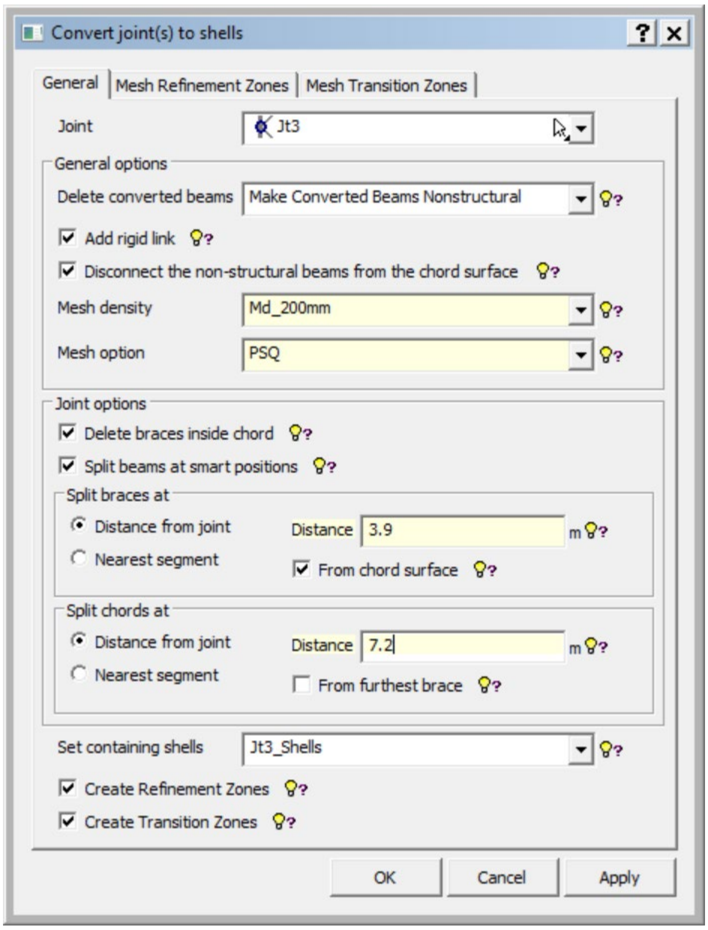
F. Similarly, we use 7.2 m for the chord (i.e., 3 times the chord diameter).

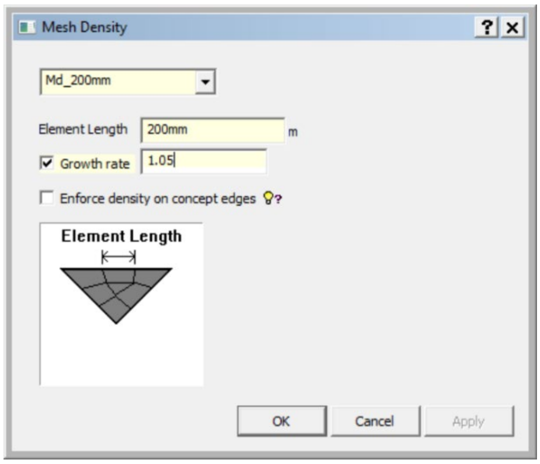
G. The shell model is automatically included in a named set. The default name is Jt3_Shells for Jt3. We will use this when we look at the mesh and results. To activate H and I you select the options “Create Refinement Zone” and “Create Transition Zone” (this is also the default setting).

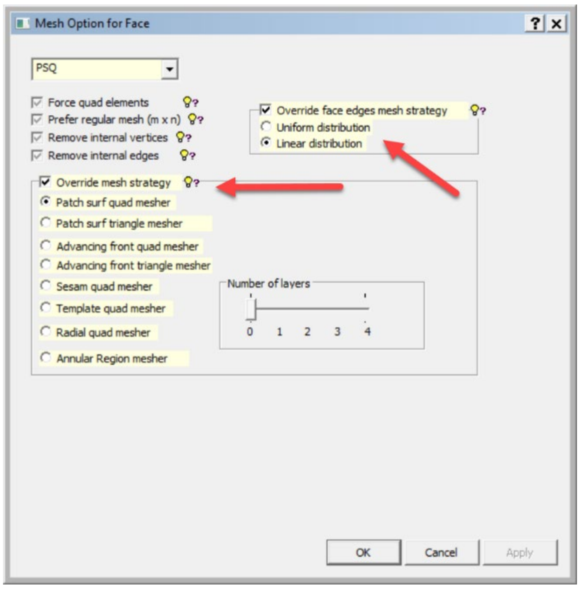
H. In case you don’t want a refined mesh around brace/chord connection you deselect these options. It could be that you want to do fatigue screening based on a coarser mesh or you want to include additional joints represented as shell finite elements to make eigenmodes matching the beam model. **For this workshop, parts of the refined mesh is used for fatigue calculation** – this will speed up the calculation time.

I. In the Mesh Refinement Zone you give number of layers and the mesh size per brace. This will enforce same mesh density for the brace connection, along the brace and inside the brace (on the chord plug). E.g. 2 layers with Total Zone Width -30mm will give 2 layers of mesh with size 15mm. In case you specify too many layers or too large Zone Width, you will get a warning that you exceed the allowed width.

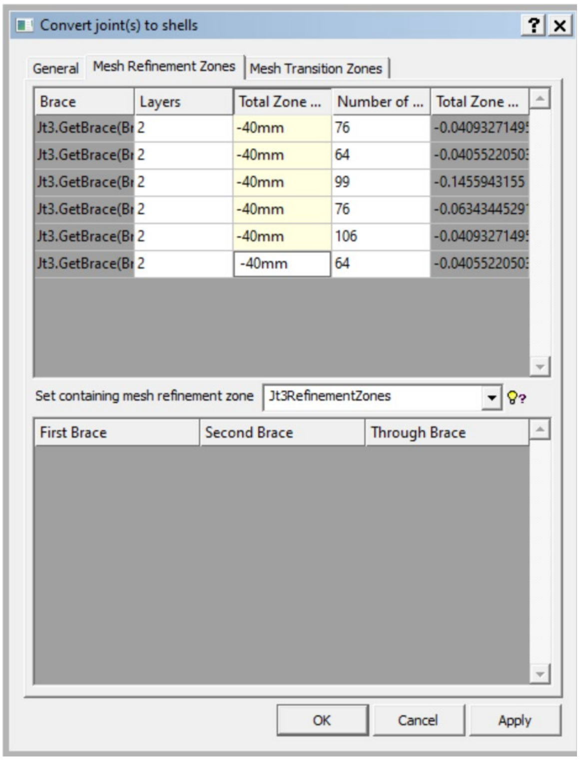
The input will then be:



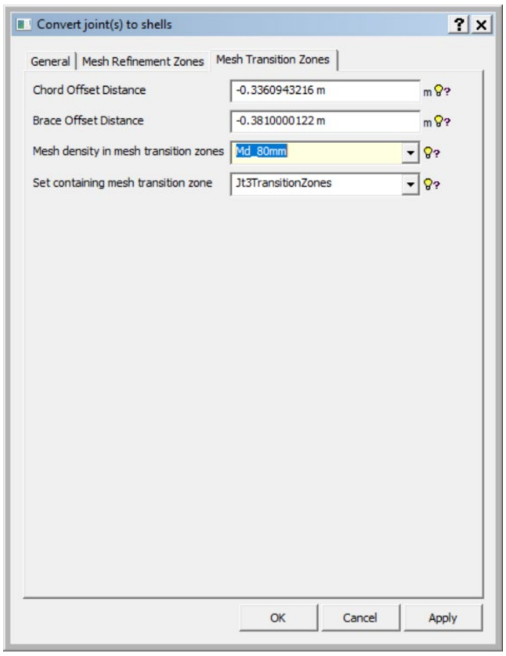


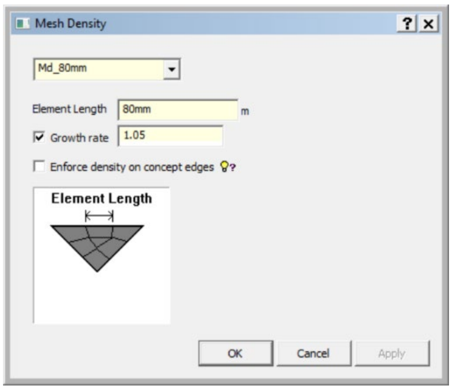


(7) For the mesh refinement zones we use a mesh of 20mm for all braces and 2 layers of regular mesh. This means that the input will be as shown below. A named set Jt3RefinementZones will be created for this part.



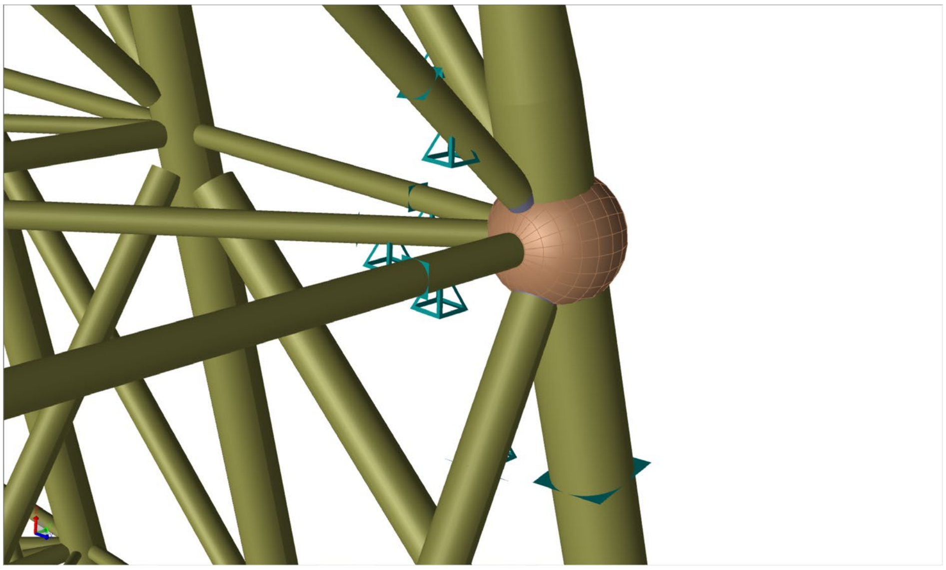
(8) The final step will be to specify the details for the mesh transition zone between fine and coarse mesh. In this case we are using four times the fine mesh, i.e. 80 mm mesh. Remember to specify the linearized mesh option. A named set Jt3TransitionZones will be created for this part. The offset values are pre-calculated, but you may modify them.





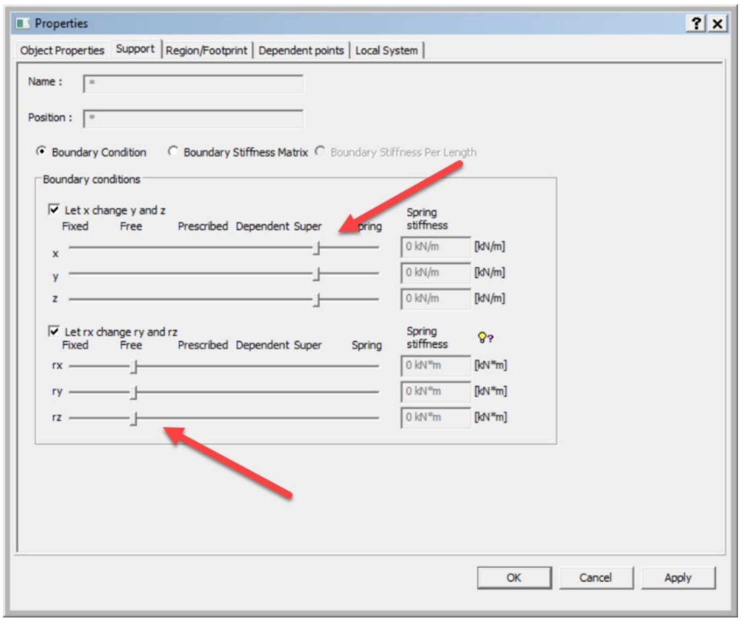
(9) After you click “OK” then GeniE will process the data and convert the joint. The performance depends on complexity of the joint (e.g. how many braces, how many overlaps and how many layers of regular mesh). Normally this is a fast operation. If you don’t use the option for refined mesh the operation is instantaneous.

You will now see the following (zoom in for details)



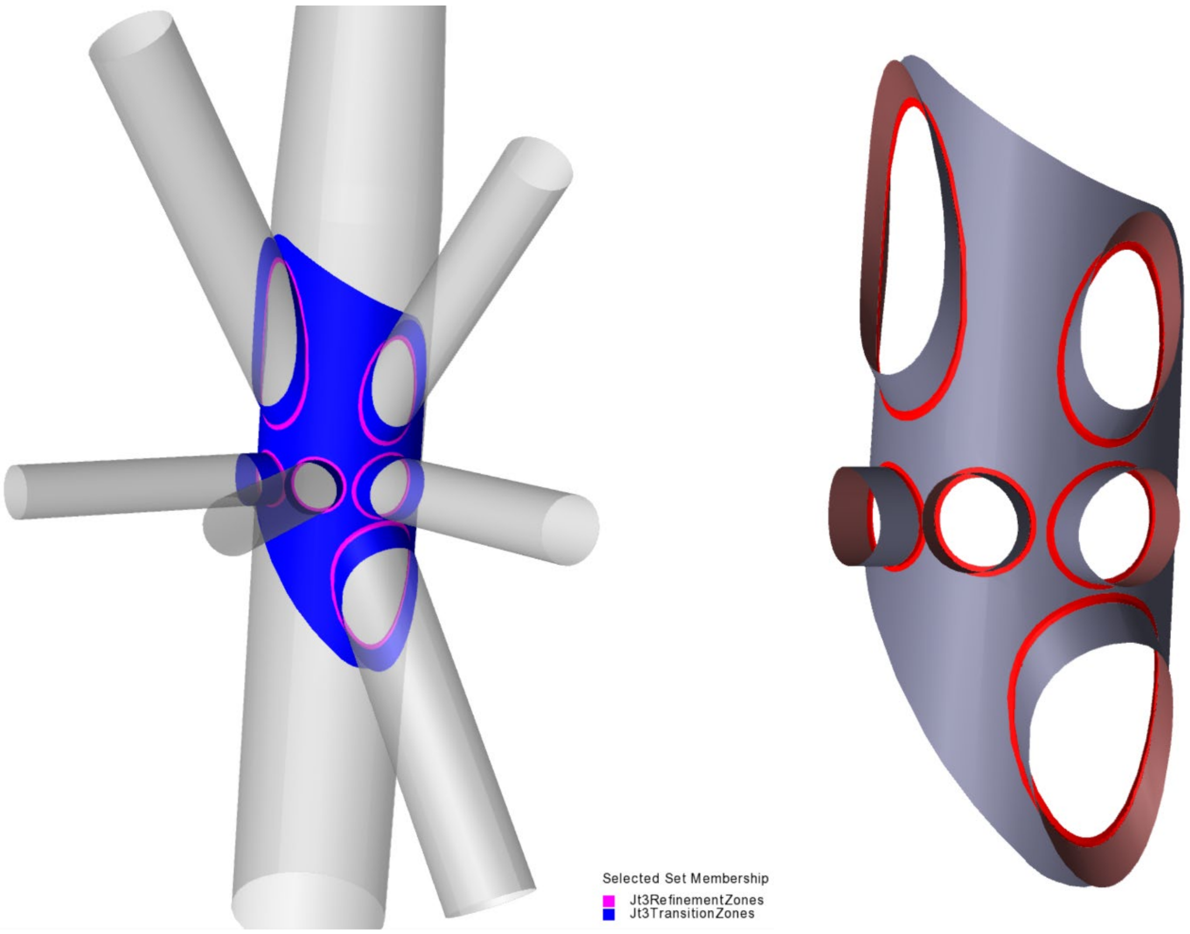
This picture shows both the non-structural beams and support rigid links (master slave connections between beams and shell model to ensure consistency).

We modify all the support rigid links to super degree of freedom for dynamic analysis by modal-superposition method. Select all of them and modify to



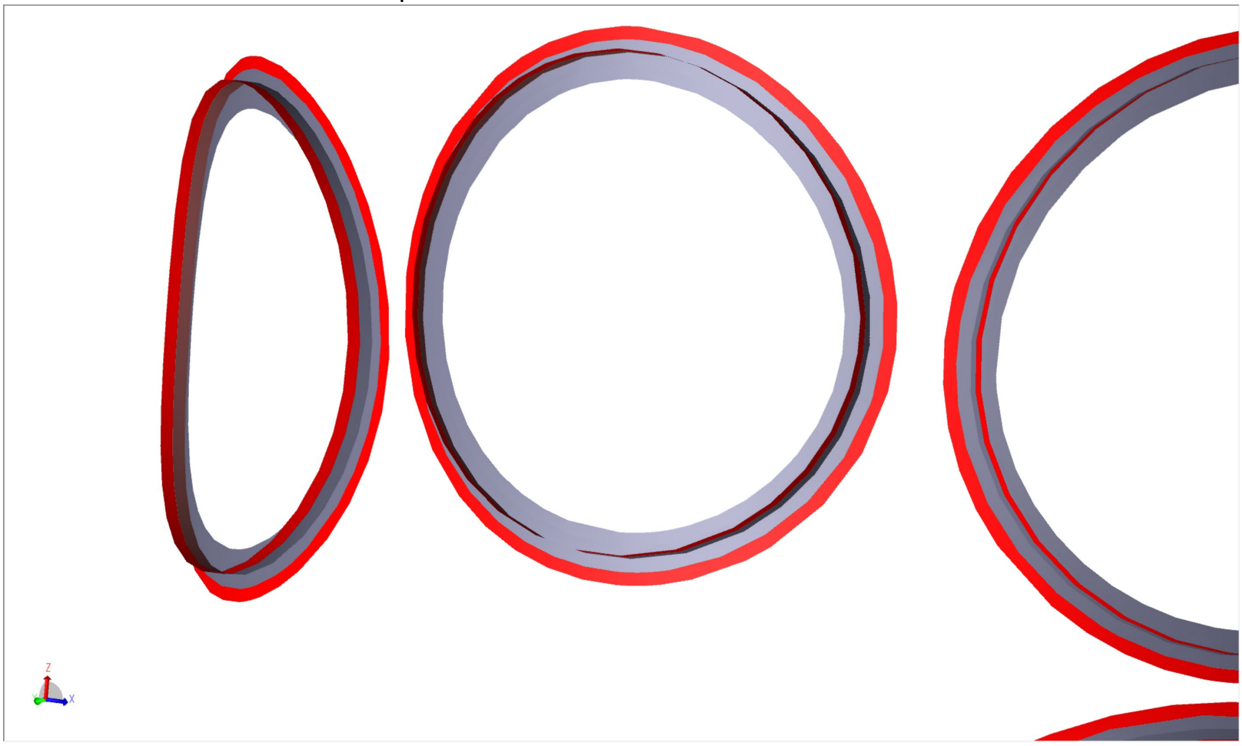
Also, the joint Jt3 can now be deleted as it no longer has a purpose.

(10) Select the named set Jt3Shells to investigate the details of the shell model. The picture below shows to the left the colour coding of the refined and transitions set. To the right we have focused the refined and transition parts only (and highlighting the refined part).



(11) For the purpose of very fast fatigue calculations, we will make a set of the part that contains the results for the stress locations 0.1 x √(r x t). We therefore select all the surface parts that are inside refinement layer number 2. The named set we call Outer_layer.

Below is shown which surface parts to select.

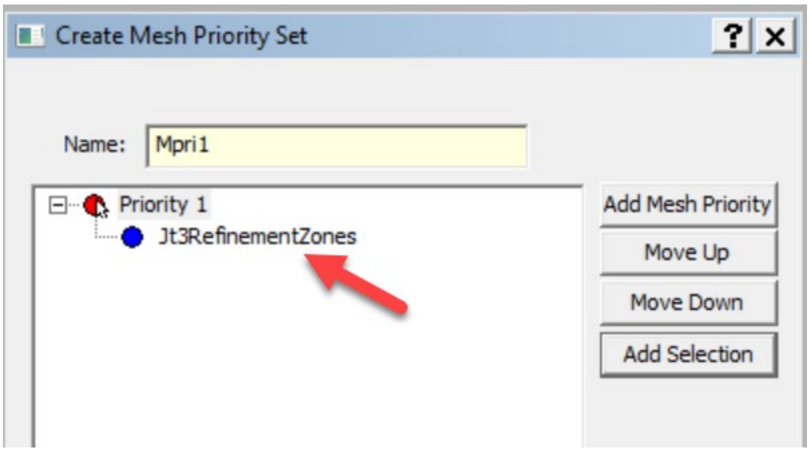


You don't have to do this, but it will significantly speed up the fatigue calculations and ease the evaluation of fatigue results.

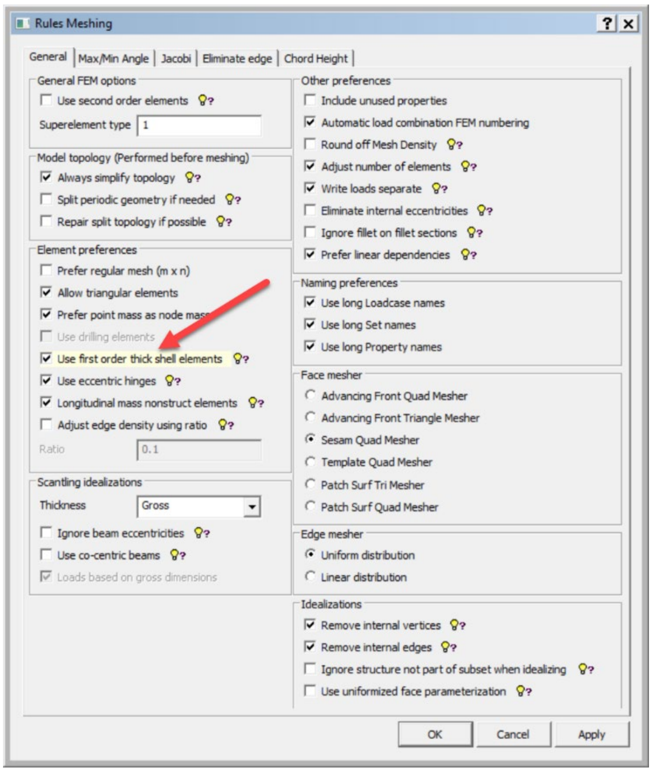
(12) Before we do the meshing, we create a named set where the non-structural beams have been deselected. This will make it easier to look at dynamic behaviour of the combined structural beam model and the tubular joint model.

Select the set Dyn_FTG and make all non-structural beams disappear from the view. One easy way is to select all "non-structure" property and make them invisible. One way to do it is by go to property "Properties|Beam Types", select BmType1, right click on it, then select objects and click ALT Minus. Then select all in the graphics window and give the new named set the name Visualize_dynamics.

(13) The fatigue calculations will benefit from structural results data where the finite element numbers are as low as possible. Therefore, we define a mesh priority (Mpri1) where we specify that the mesh numbering sequence shall start with the named set Jt3RefinementZones.

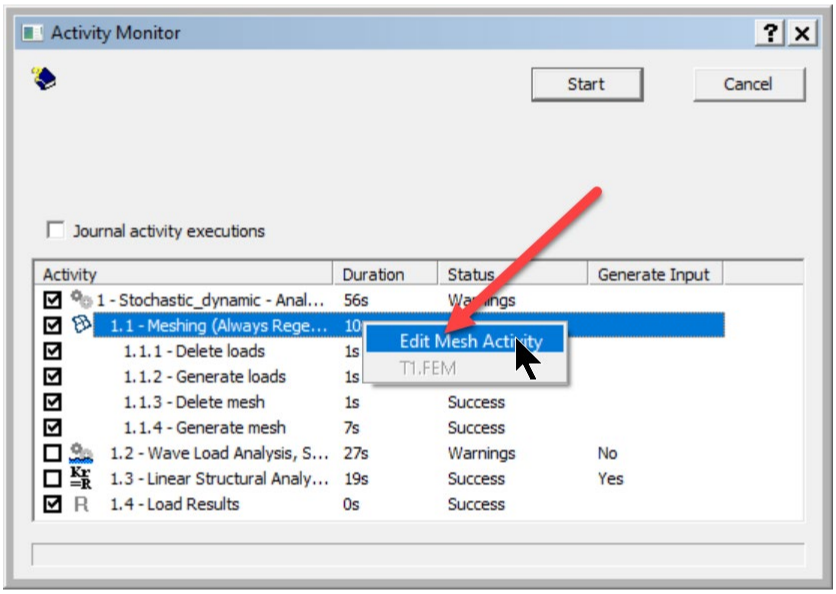


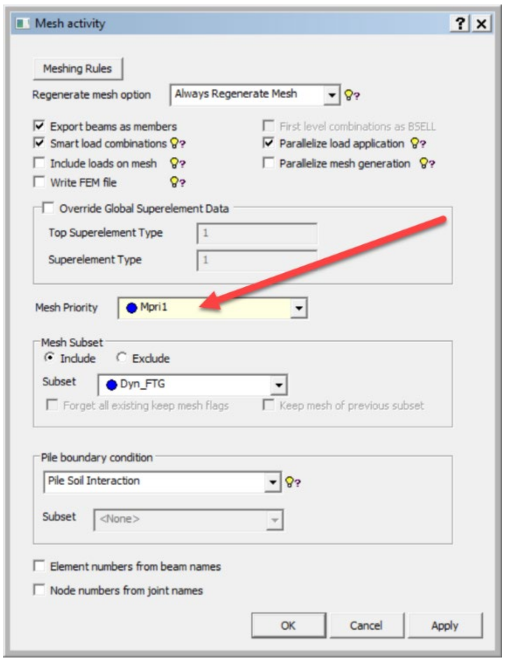
(14) Based on the recommendations from RP-C203 we will use 1st order thick shell elements for the calculations. The results will differ quite much from using 1st order thin shell elements (which is the default setting in GeniE)



(15) Check the quality of the finite element mesh.

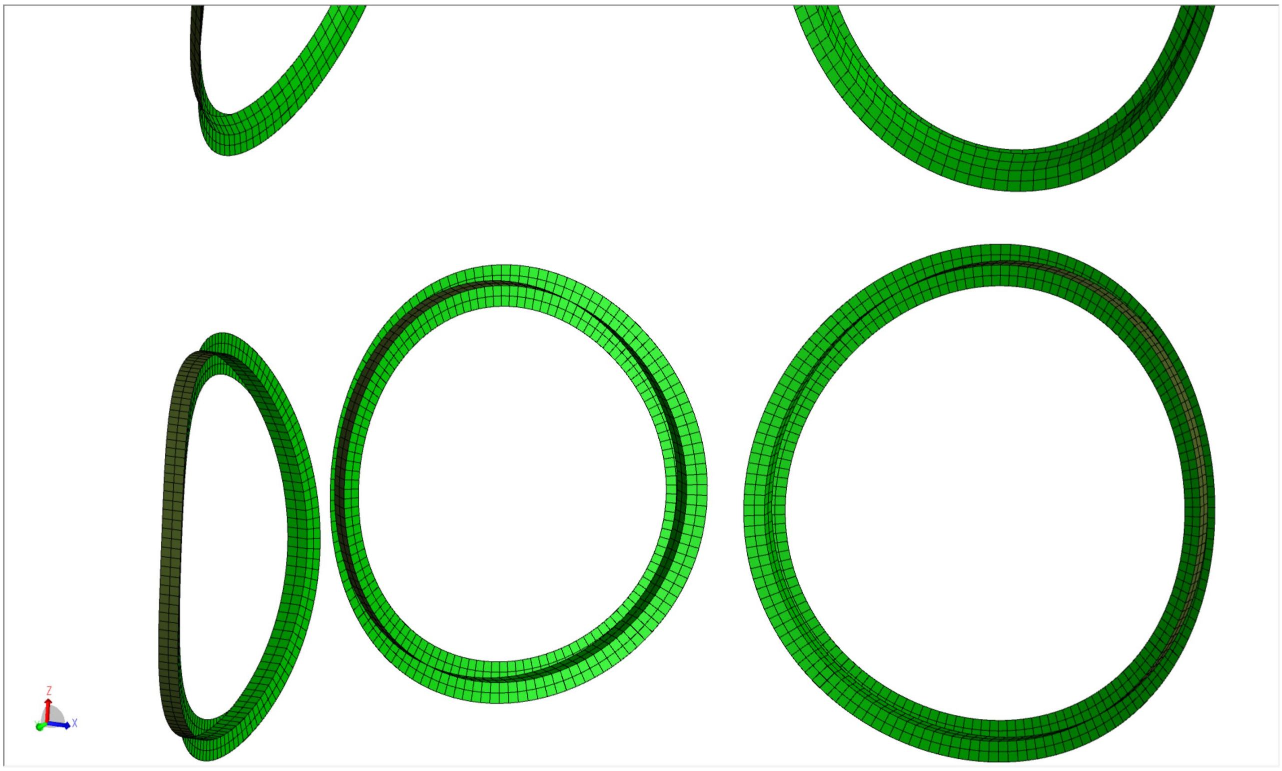
Click ALT+M and edit the mesh activity to specify that Mesh Priority shall be used.



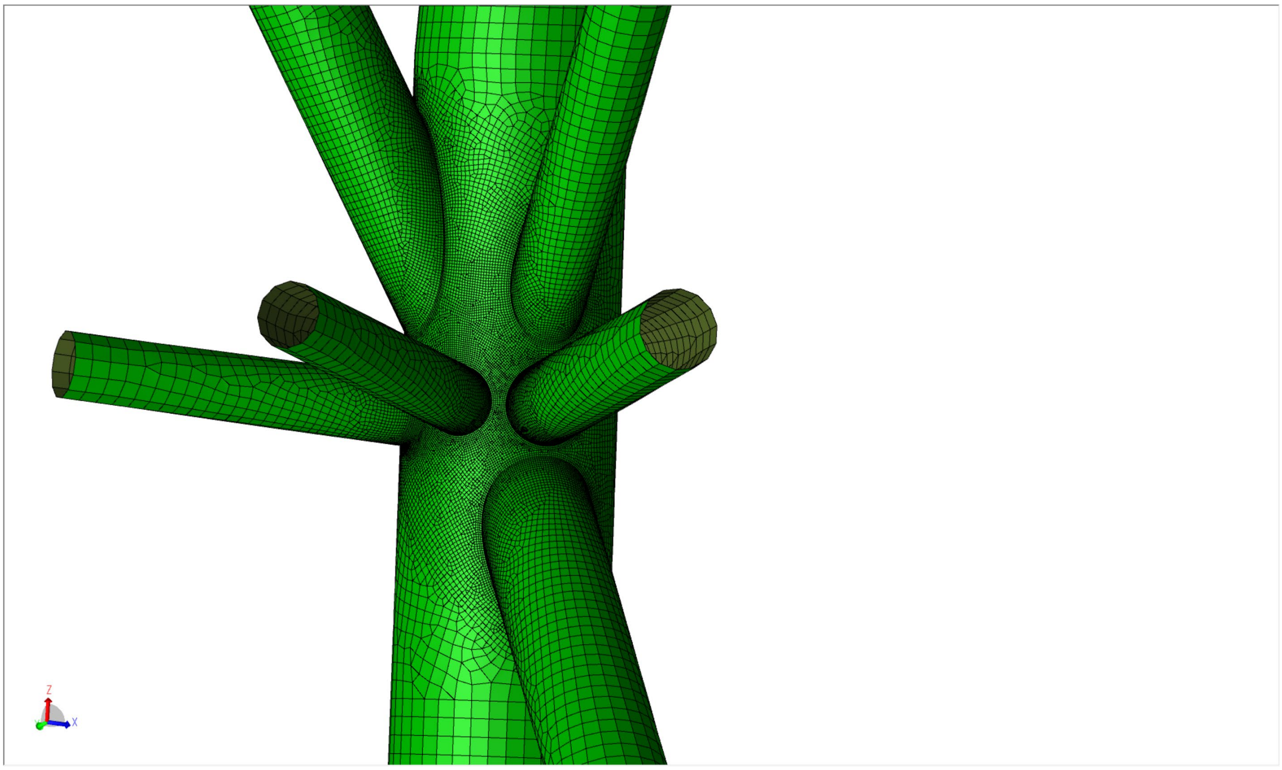


Finally click “Start” to initiate the meshing.

The mesh for the refined part looks like this

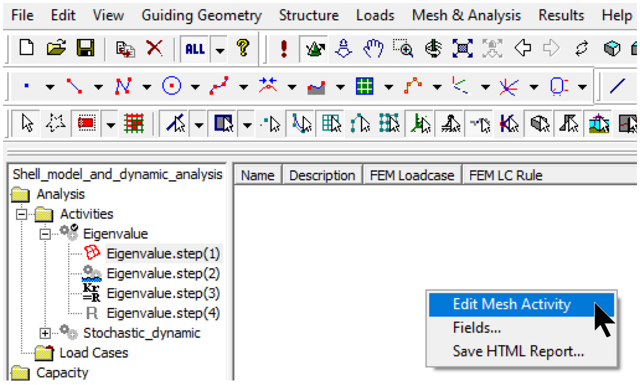


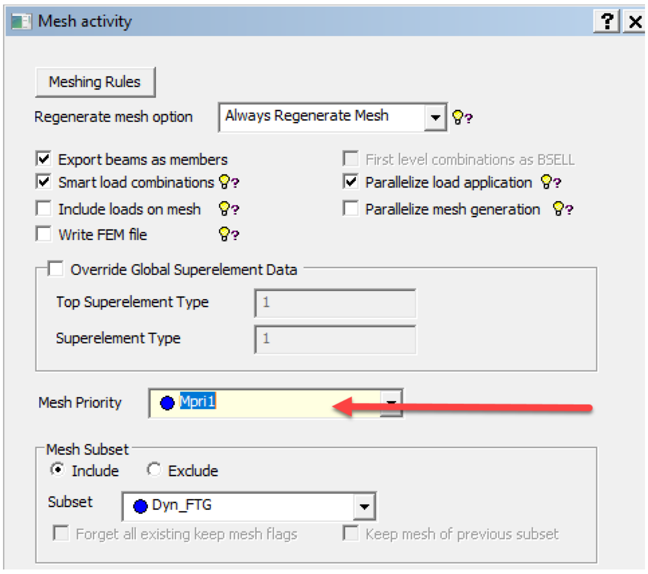
Similarly, the mesh for the whole joint Jt3 is shown below. It will satisfy requirements on regular and quadratic mesh, but if you want to adjust you can modify the mesh input details or even edit the finite element mesh (edit mesh requires GeniE extension REFM).



7.1 Eigenvalue analysis

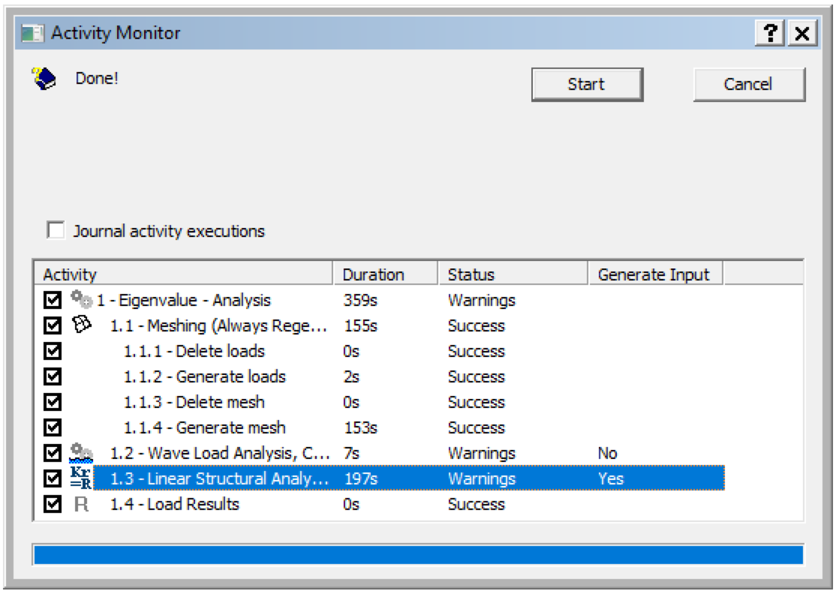
(1) Set the activity Eigenvalue to active and modify the Mesh Activity to include the mesh priority Mpri1. Make sure that the “Mesh Subset” includes Dyn_FTG.

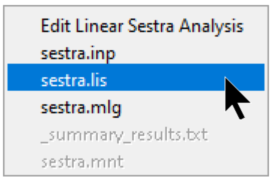




(2) Then run the eigenvalue analysis from ALT+D

(3) When analysis has been completed, then investigate the Sestra listing (Sestra.lis) file





At the end of the listing file you will find the eigenvalues for the model containing Jt3 modelled with shell finite elements and connected to the global beam model with rigid link supports.

From combined shell and beam analysis:

```
Print of eigenvalues.
Eigenvalues have unit sec^-2; frequency = sqrt(eigenvalue) / (2 * pi); period = 1 / frequency.
       Number;    Eigenvalue;     Frequency;        Period
           1;  5.833778e+00;  3.844104e-01;  2.601387e+00
           2;  6.311906e+00;  3.998530e-01;  2.500919e+00
           3;  1.441769e+01;  6.043214e-01;  1.654749e+00
           4;  3.995507e+01;  1.006019e+00;  9.940172e-01
           5;  4.319483e+01;  1.046010e+00;  9.560135e-01
           6;  5.573754e+01;  1.188212e+00;  8.416005e-01
           7;  8.753026e+01;  1.489016e+00;  6.715847e-01
           8;  1.218787e+02;  1.757050e+00;  5.691359e-01
           9;  1.263373e+02;  1.788899e+00;  5.590029e-01
          10;  1.879475e+02;  2.181918e+00;  4.583124e-01
```

From beam analysis:

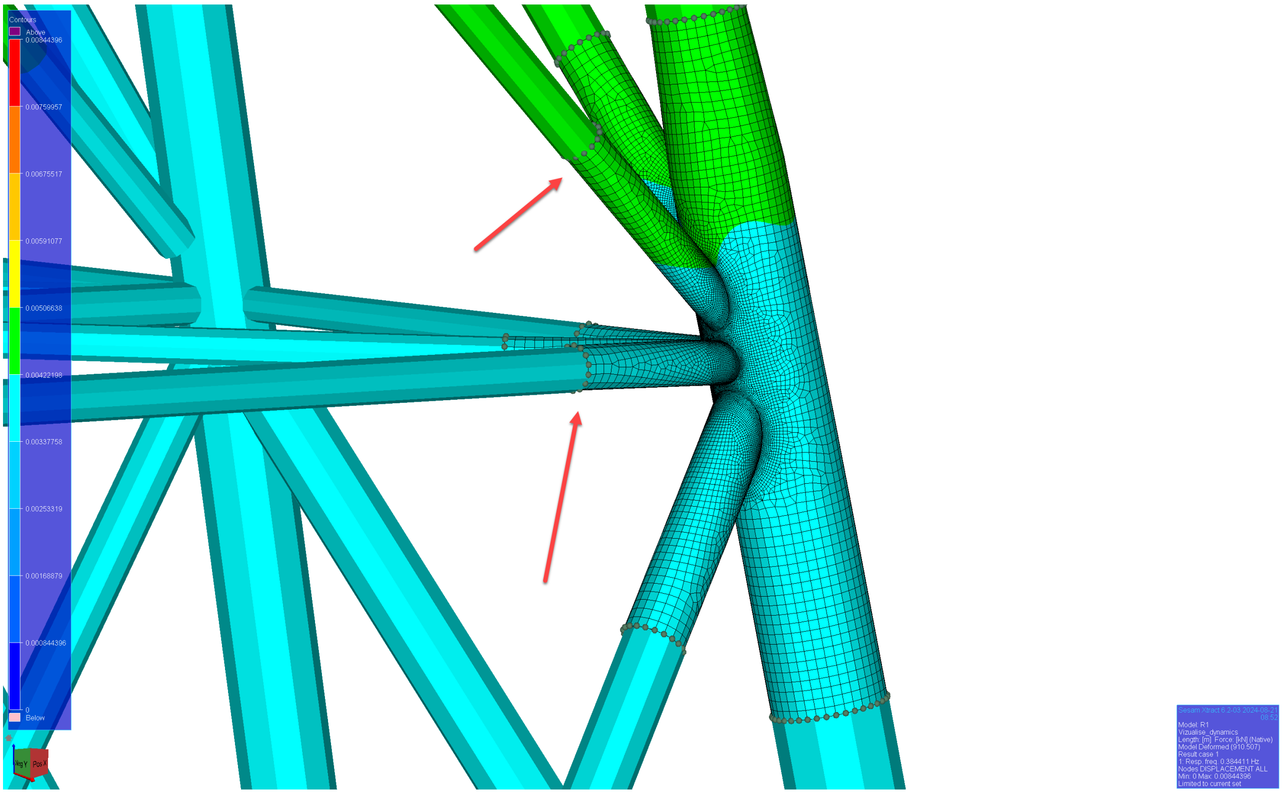
```
Print of eigenvalues.
Eigenvalues have unit sec^-2; frequency = sqrt(eigenvalue) / (2 * pi); period = 1 / frequency.
       Number;    Eigenvalue;     Frequency;        Period
           1;  5.781788e+00;  3.826936e-01;  2.613056e+00
           2;  6.249850e+00;  3.978826e-01;  2.513304e+00
           3;  1.433589e+01;  6.026046e-01;  1.659463e+00
           4;  3.936200e+01;  9.985245e-01;  1.001478e+00
           5;  4.229004e+01;  1.034997e+00;  9.661861e-01
           6;  5.482421e+01;  1.178437e+00;  8.485817e-01
           7;  8.764942e+01;  1.490029e+00;  6.711280e-01
           8;  1.218744e+02;  1.757019e+00;  5.691459e-01
           9;  1.260375e+02;  1.786775e+00;  5.596674e-01
          10;  1.874032e+02;  2.178756e+00;  4.589775e-01
```

For this case, the eigen periods match very well. In case they don’t, you may need to convert additional tubular joints at the same elevation as Jt3 to insert same flexibility in these joints as for Jt3 because shell elements are used. For the additional joints, you may not need to use refined mesh zones.

(4) Investigate the mode shapes

Open the advanced post-processor Xtract from the pull-down menu Results -> Advanced Results (Xtract).

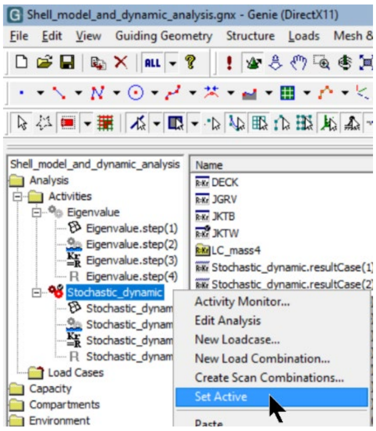
Then select the set Visualize_Dynamics and make a mode shape animation including deflected shape. Make sure that the ends of the beams and the edges of the shell model have the same deflections (i.e. they are connected during animation).

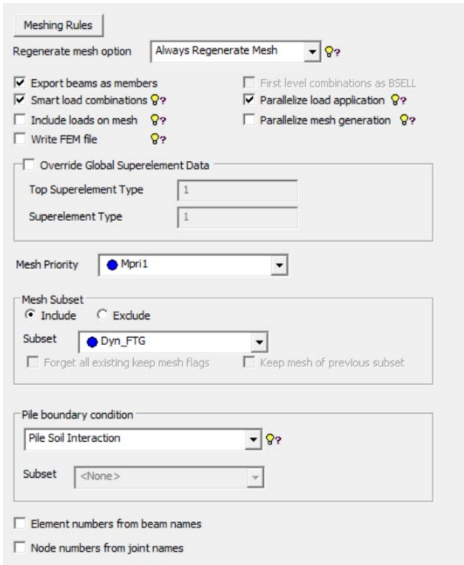


(5) Export the result SIN file to repository. You could use the name Shell_model_eigenvalue_R1.SIN.

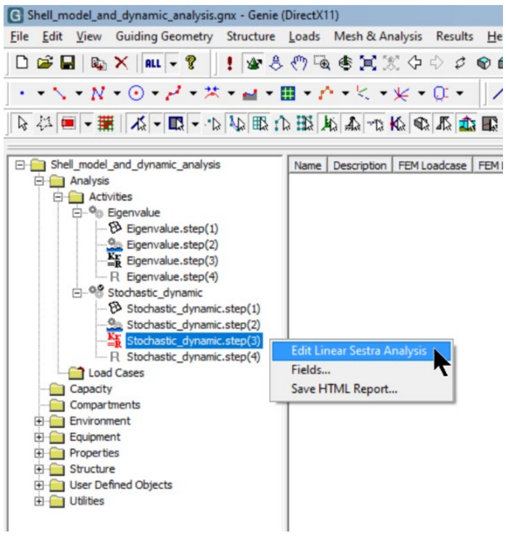
7.2 Stochastic dynamic analysis

(1) Set the activity Stochastic_dynamic to active and check that the Mesh Activity includes the mesh priority Mpri1 like shown in the previous Chapter 7.1 (step 1). Also make sure that the Mesh Subset includes Dyn_FTG.



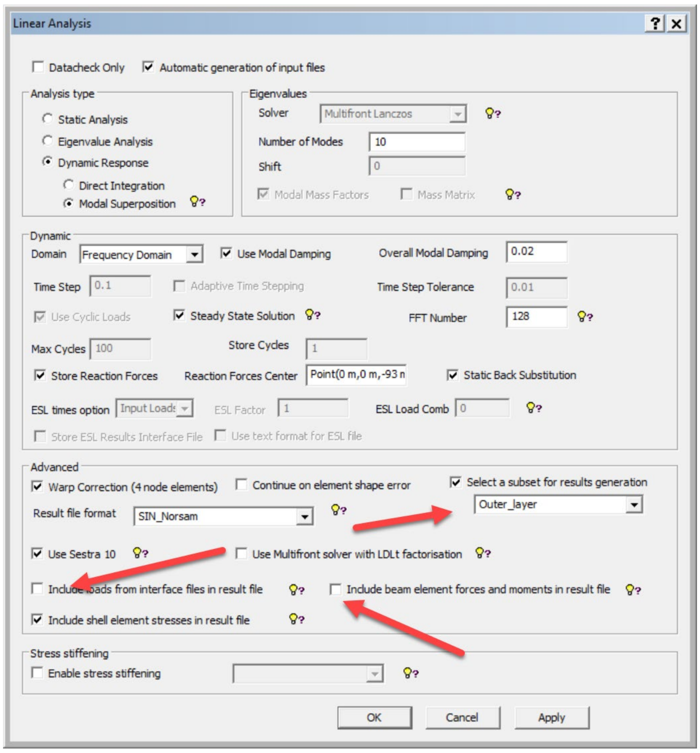


(2) From the browser, edit the analysis activity and specify a) for which parts of the shell model to export results and b) what type of results to be exported. This is strictly not needed to do, but it will significantly speed up the analysis time, the time for doing fatigue calculations (next Chapter) and data storage. When the aim is to calculate the fatigue according to DNV-RP-C203 it is sufficient to concentrate on the refined mesh only.

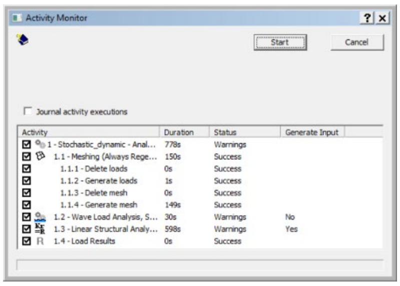


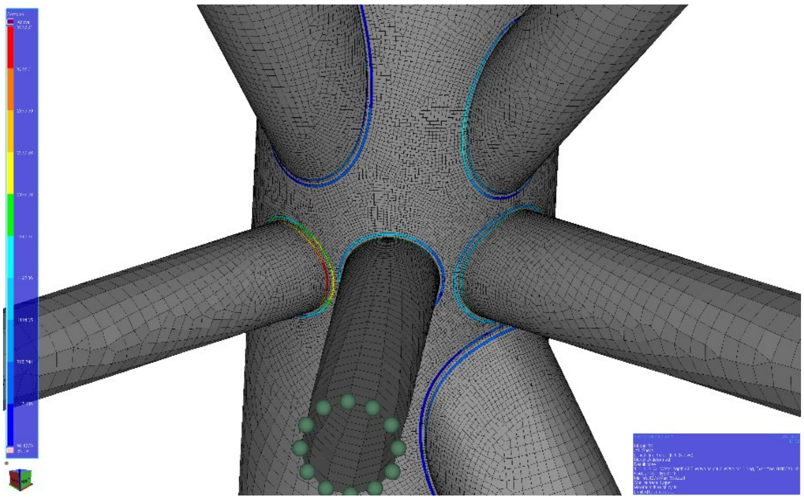
Because the stresses from the first mesh line around the braces on the chord and the brace shall be used in fatigue calculations we can select to work with the second layer of elements and checking for the worst positions. The second layer of elements are part of the named set Outer_layer. Modify the input for Sestra analysis as shown on next page.

Further, fatigue calculations are based on stresses which means that other result attributes are not needed. This means we can de-select the loads and the beam forces, also shown on the next page.



(3) Run analysis form e.g. ALT+D. When analysis is done, you should check the results from the listing files and look at the stresses in Xtract to see that they are as expected. Notice that the results are in frequency domain, so you need to select the angle for the imaginary part before looking at the stresses.





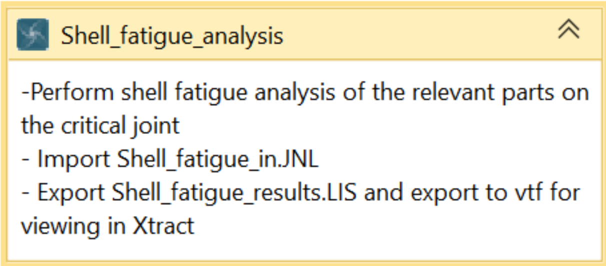
If you want to see the stresses for the whole shell model, then re-run without selecting a set for results generation.

(4) When you are done, then export the result SIN file to repository. **You must use the name Shell_model_dynamic_R1.SIN (this is referenced in the script file in the next Step).**
(5) Click Exit and confirm saving data if you are asked.

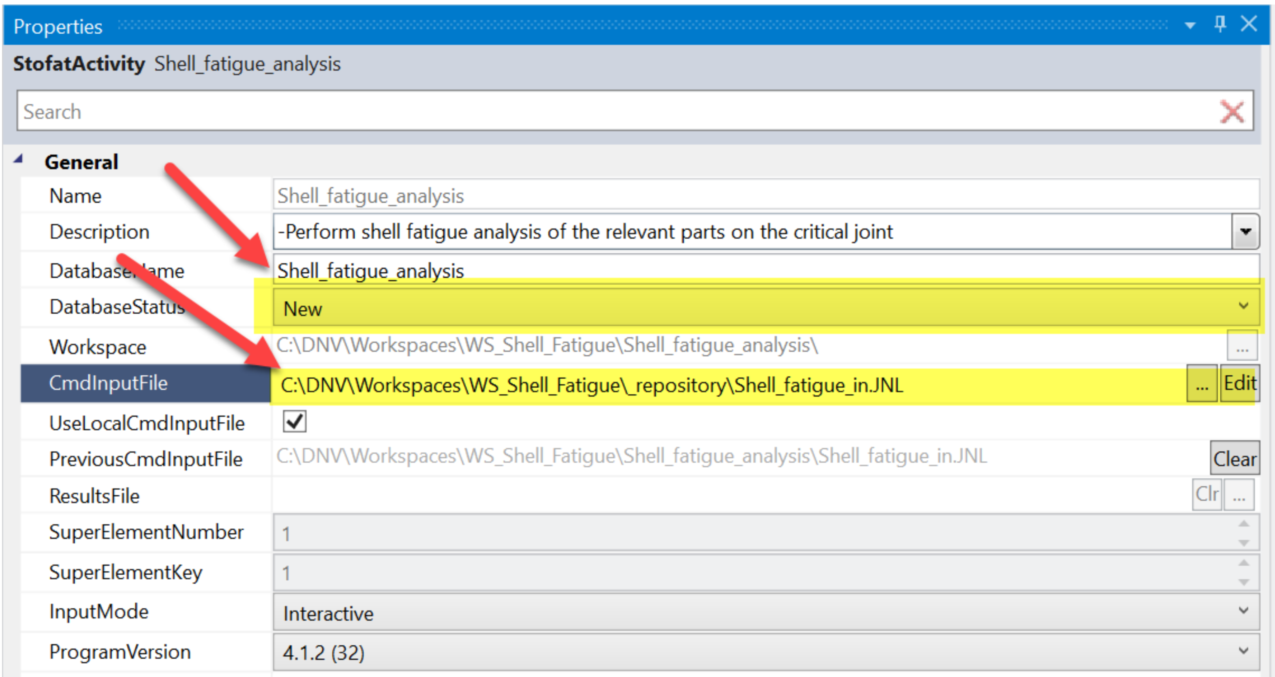
8 STEP 4: SHELL FATIGUE ANALYSIS

The final step will be to perform the fatigue calculations using the Sesam tool Stofat.

(1) From Sesam Manager, click on the box.

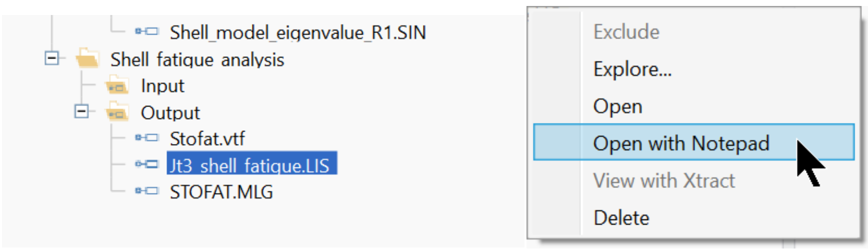


(2) Edit the input data as follows. The database must be set to “new” and the pre-defined input file for shell fatigue must be referenced to location “repository”.



Like for the beam fatigue, the input file can be viewed and edited by you. Remember this is just an example of how input could be.

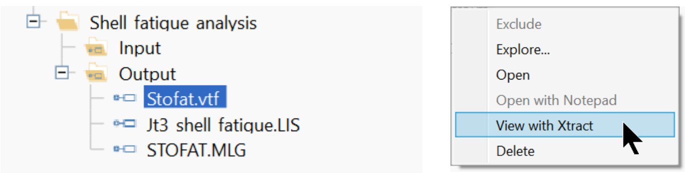
(3) Click Run Activity. When Stofat has completed the program exit the program and inspect the listing file (showing fatigue damage above 0.5).



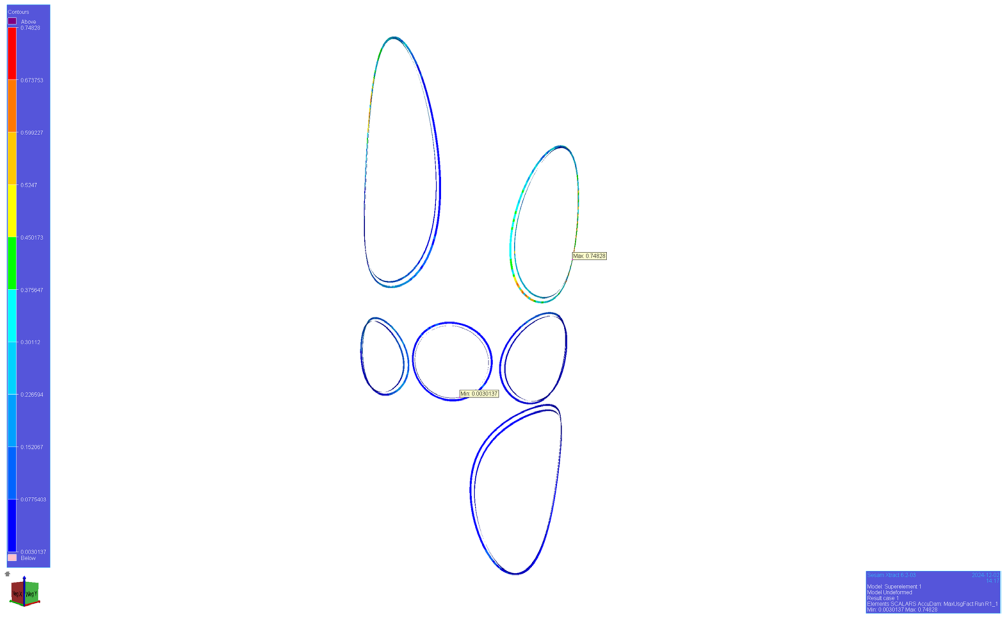
```
DATE: 02-DEC-2024 TIME: 14:03:02   PROGRAM: SESAM    STOFAT    4.1-02  27-SEP-2022                                       PAGE:     3

                                   STOCHASTIC     ELEMENT fatigue check results
                                   Run: R1        Superelement JT3_MESH
                                   Priority.....: Selected Elements
                                   Usage factor : Above  0.50
                                   Design Fatigue Life: 20.0    Years                                                    SUB PAGE:     3

Element  Status UsageFactor ChkPnt   ChkPlc  SNCurve          FatigLife  StrsCycle  X-coord.   Y-coord.   Z-coord.   StrsRange
                                     ElType  ElThick          ThckCor    AxialScf   BendngScf  ShearScf   WeibScale  WeibShape
---------------------------------------------------------------------------------------------------------------------------------
4702      PASS  6.2689E-01  9 (+z)   StrsPnt API-WJ-FC        3.190E+01  9.124E+07  2.388E+01  1.362E+01 -1.428E+01  1.173E+05
                                     FQUS24  0.07             1.4463     1.2        1.2        1.2        1.514E+04  1.399E+00
4704      PASS  6.6095E-01  9 (+z)   StrsPnt API-WJ-FC        3.026E+01  9.195E+07  2.392E+01  1.362E+01 -1.432E+01  1.242E+05
                                     FQUS24  0.07             1.4463     1.2        1.2        1.2        1.550E+04  1.375E+00
4705      PASS  5.9378E-01  9 (+z)   StrsPnt API-WJ-FC        3.368E+01  9.319E+07  2.395E+01  1.362E+01 -1.433E+01  1.253E+05
                                     FQUS24  0.07             1.4463     1.2        1.2        1.2        1.510E+04  1.350E+00
4707      PASS  5.9174E-01  9 (+z)   StrsPnt API-WJ-FC        3.380E+01  9.397E+07  2.399E+01  1.361E+01 -1.436E+01  1.289E+05
                                     FQUS24  0.07             1.4463     1.2        1.2        1.2        1.522E+04  1.336E+00
4708      PASS  5.4634E-01  9 (+z)   StrsPnt API-WJ-FC        3.661E+01  9.500E+07  2.402E+01  1.361E+01 -1.437E+01  1.290E+05
                                     FQUS24  0.07             1.4463     1.2        1.2        1.2        1.493E+04  1.323E+00
4729      PASS  6.1076E-01  10(+z)   StrsPnt API-WJ-FC        3.275E+01  9.188E+07  2.453E+01  1.362E+01 -1.419E+01  1.175E+05
                                     FQUS24  0.07             1.4463     1.2        1.2        1.2        1.521E+04  1.401E+00
4732      PASS  5.3407E-01  10(+z)   StrsPnt API-WJ-FC        3.745E+01  9.029E+07  2.457E+01  1.362E+01 -1.412E+01  1.042E+05
                                     FQUS24  0.07             1.4463     1.2        1.2        1.2        1.474E+04  1.467E+00
4736      PASS  6.0503E-01  10(+z)   StrsPnt API-WJ-FC        3.306E+01  8.909E+07  2.460E+01  1.362E+01 -1.402E+01  1.028E+05
                                     FQUS24  0.07             1.4463     1.2        1.2        1.2        1.550E+04  1.516E+00
4737      PASS  6.7129E-01  10(+z)   StrsPnt API-WJ-FC        2.979E+01  8.909E+07  2.461E+01  1.362E+01 -1.399E+01  1.060E+05
                                     FQUS24  0.07             1.4463     1.2        1.2        1.2        1.603E+04  1.518E+00
4738      PASS  7.4828E-01  10(+z)   StrsPnt API-WJ-FC        2.673E+01  8.913E+07  2.461E+01  1.361E+01 -1.397E+01  1.098E+05
                                     FQUS24  0.07             1.4463     1.2        1.2        1.2        1.661E+04  1.518E+00
4739      PASS  7.3113E-01  10(+z)   StrsPnt API-WJ-FC        2.735E+01  8.889E+07  2.462E+01  1.361E+01 -1.394E+01  1.100E+05
                                     FQUS24  0.07             1.4463     1.2        1.2        1.2        1.635E+04  1.507E+00
4740      PASS  6.7041E-01  10(+z)   StrsPnt API-WJ-FC        2.983E+01  8.875E+07  2.462E+01  1.361E+01 -1.392E+01  1.073E+05
                                     FQUS24  0.07             1.4463     1.2        1.2        1.2        1.581E+04  1.501E+00
4741      PASS  6.5988E-01  10(+z)   StrsPnt API-WJ-FC        3.031E+01  8.861E+07  2.463E+01  1.361E+01 -1.389E+01  1.073E+05
                                     FQUS24  0.07             1.4463     1.2        1.2        1.2        1.565E+04  1.493E+00
4742      PASS  6.4701E-01  10(+z)   StrsPnt API-WJ-FC        3.091E+01  8.852E+07  2.463E+01  1.360E+01 -1.386E+01  1.070E+05
                                     FQUS24  0.07             1.4463     1.2        1.2        1.2        1.549E+04  1.487E+00
4743      PASS  5.9756E-01  10(+z)   StrsPnt API-WJ-FC        3.347E+01  8.844E+07  2.463E+01  1.360E+01 -1.384E+01  1.043E+05
                                     FQUS24  0.07             1.4463     1.2        1.2        1.2        1.506E+04  1.486E+00
4744      PASS  5.8245E-01  10(+z)   StrsPnt API-WJ-FC        3.434E+01  8.836E+07  2.463E+01  1.360E+01 -1.381E+01  1.036E+05
                                     FQUS24  0.07             1.4463     1.2        1.2        1.2        1.491E+04  1.483E+00
4745      PASS  5.9168E-01  10(+z)   StrsPnt API-WJ-FC        3.380E+01  8.828E+07  2.463E+01  1.359E+01 -1.379E+01  1.045E+05
                                     FQUS24  0.07             1.4463     1.2        1.2        1.2        1.495E+04  1.479E+00
4746      PASS  5.9364E-01  10(+z)   StrsPnt API-WJ-FC        3.369E+01  8.821E+07  2.463E+01  1.359E+01 -1.376E+01  1.047E+05
                                     FQUS24  0.07             1.4463     1.2        1.2        1.2        1.495E+04  1.477E+00
4747      PASS  6.0749E-01  10(+z)   StrsPnt API-WJ-FC        3.292E+01  8.815E+07  2.463E+01  1.358E+01 -1.373E+01  1.056E+05
                                     FQUS24  0.07             1.4463     1.2        1.2        1.2        1.505E+04  1.475E+00
4748      PASS  6.1329E-01  6 (+z)   StrsPnt API-WJ-FC        3.261E+01  8.813E+07  2.465E+01  1.359E+01 -1.374E+01  1.061E+05
                                     FQUS24  0.07             1.4463     1.2        1.2        1.2        1.508E+04  1.473E+00
4749      PASS  6.0049E-01  6 (+z)   StrsPnt API-WJ-FC        3.331E+01  8.809E+07  2.464E+01  1.359E+01 -1.371E+01  1.053E+05
                                     FQUS24  0.07             1.4463     1.2        1.2        1.2        1.497E+04  1.473E+00
4750      PASS  6.1651E-01  7 (+z)   StrsPnt API-WJ-FC        3.244E+01  8.804E+07  2.464E+01  1.358E+01 -1.366E+01  1.058E+05
                                     FQUS24  0.07             1.4463     1.2        1.2        1.2        1.509E+04  1.476E+00
4751      PASS  6.1743E-01  10(+z)   StrsPnt API-WJ-FC        3.239E+01  8.802E+07  2.462E+01  1.356E+01 -1.363E+01  1.054E+05
                                     FQUS24  0.07             1.4463     1.2        1.2        1.2        1.510E+04  1.480E+00
```

We see from the listing file that the worst usage factor is 0.748 (calculated fatigue life 26.7 years). Compared to the beam fatigue results this is an improvement from 6.07 (calculated fatigue life 3.3 years) – or a factor of 8.1.

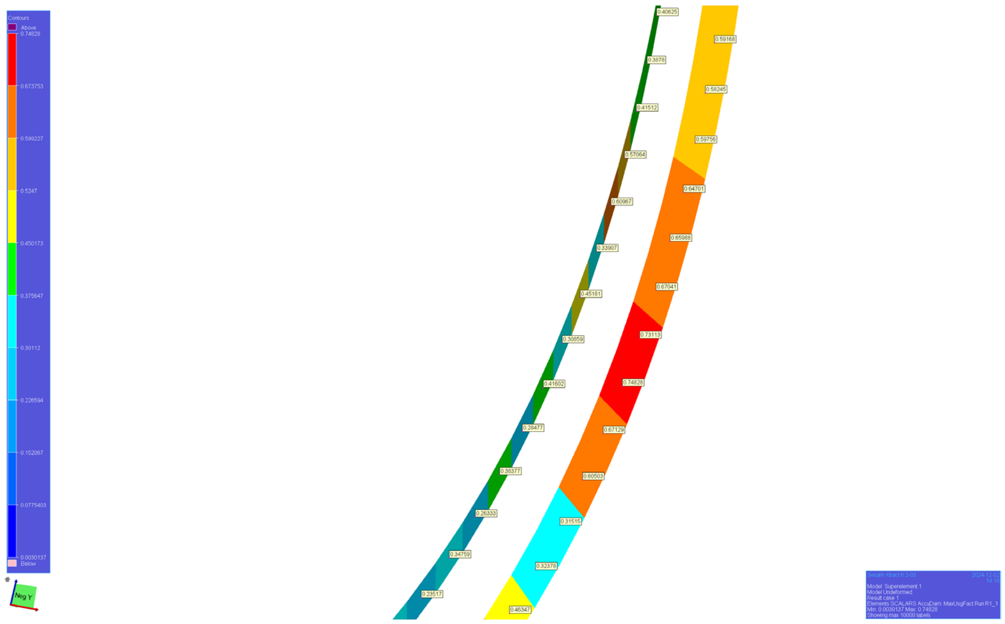
You can also investigate the results graphically in Xtract.



In Xtract's browser window, double click Superelement 1 and the result attribute Accumulated Damage. The maximum values may be added to the display from the pulldown menu Results and then Show Peaks.



The picture above shows fatigue damage (Uf) for the Outer_layer.



Zooming in to the maximum calculated fatigue life (Uf).

9 EXAMPLE DATA FILES

This example has two zip files for import into Sesam Manager.

The first zip file "Jacket_shell_fatigue_in.zip" (1.3 MB) includes files to

- Create the jacket model in GeniE
- A GeniE script file to convert tubular joint Jt3 to shell model (notice some manual steps must be done – they are described in the script file)
- Input files for beam and shell fatigue

The second zip file "Jacket_shell_fatigue_completed.zip" (392 MB) includes the above plus results from structural analysis and fatigue calculations.

Notice that the GeniE sessions in the zip files include both the usage of GeniE v8.10-01 and GeniE v8.11-02. You may need to modify in Sesam Manager set up to reflect what you have installed.

10 DISCLAIMER

This workshop has been made with the objective to explain the Sesam workflow for converting a tubular joint to shell models and how to conduct shell fatigue calculations based on a stochastic dynamic analysis.

The model and environmental data are simplified and the input data for the tubular joint conversion as well as the input data for beam and shell fatigue may not represent real project data. DNV takes no responsibility in the use, or the consequence of use, of the workshop material or input files.

The information and data provided by DNV in this workshop material is for general training purposes only. All information in the workshop material is provided in good faith, however DNV makes no representation on warranty of any kind, express or implied, regarding the accuracy, adequacy, validity, reliability, availability or completeness of any information in the workshop material. The same applies to the use of input parameters for calculating fatigue life based on DNV-RP-C203.

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.