

# SESAM EXAMPLE Shell fatigue of tubular joints

WORKSHOP Shell fatigue of tubular joints



Date: 6 December 2024

Prepared by DNV – Digital Solutions

E-mail support: <a href="mailto:software.support@dnv.com">software.support@dnv.com</a>

E-mail sales: software@dnv.com

© DNV AS. All rights reserved

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



# Table of Contents

1	INTRODUCTION	1
2 2.1 2.2 2.3	METHODOLOGY AND DISCUSSIONS Background Fatigue analysis by use of calculated SCF's Fatigue analysis by use of shell theory	2 2 3 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 7.1 7.2	STEP 3: CONVERT JT3 TO SHELL MODEL, EIGENVALUE AND DYNAMIC ANALYSIS Eigenvalue analysis Stochastic dynamic analysis	13 25 28
8	STEP 4: SHELL FATIGUE ANALYSIS	31
9	EXAMPLE DATA FILES	
10	DISCLAIMER	



# **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 <u>software.support@dnv.com</u> 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  $0.1 \times \sqrt{(r \times t)}$  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 2<sup>nd</sup> order thick shell is recommended, 1<sup>st</sup> order thick shell theory is generally accepted for use in industry and followed in this study. 1<sup>st</sup> 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
  - o 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.

M	ה (ק	-													
File	m	e View	Tool												
	G	D	H			e l	JS	XML	Zip		JS	XML	Zip		
New Job	New GeniE Job	New DeepC Job	New HydroD Job	Open Jobs Job	Save	Save All	Import JS	Import XML	Import ZIP	Import JNL	Export JS	Export XML	Export ZIP 🕶	Run Job	Run Selected Activity
		New		Open	Sa	ve			Impo	ort/Expc	ort				Process

- 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:

I 5 0 0		WS_Shell Fatigue* - Sesam Manager V6.7-00			- 0 ×
File Home View Tool C D D H Home View Tool New New New New New New New New New New	Save Save Save Save Save Save				
File Overview + 9 ×	WS Shel Fatiguet X 🔄 Test3 🕠 Start 👘 Quick Start		+ Properties		- 0 ×
22 07	Flow Chart Tree View		StofatActivity Shel fatigue	analysis	
😑 🐜 WS Shell Fatique			Search		×
WS Shell Fatigueumi		E WS_Shell_Faligue A	4 Count		
repository			Nome	Shell fatique analysis	
			Description	-Perform shell fatigue analysis of the relevant parts on the critical joint	
		🔁 Beam model and dynamic analysis 🔗	DatabaseName	Shell_fatigue_analysis	
		-import a GNX file with model, frequency wave loads,	DatabaseStatus	Old	~
		run eigenvalue and dynamic analyses.	Workspace	C\DW\Workspaces\WS_Shell_Fatigue\Shell_fatigue_analysis\	-
		(Tubular joint fatigue in.gnd)	CmdinputFile		_ Edit
		- Beam, model, eigenvalue, R1SN and	UseLocalCmdinputFile		1000
		Beam_model_dynamic_R13IN to repository	PreviousUmdinputrite	Communications and requestion and a particular and a state and a state	Clear
			SuperflementNumber	1	
			SuperElementKey		
		8	InputMode	Interactive	
		team Jaligue, analysis	ProgramWersion	4.1.2 (32)	~
		-Perform fatigue analysis of a selected set of members and joints.	AdditionalArguments		
		-SCF's are based on Efflymicu equations	CommandLine	CVProgram Files (x86)/DNV/Stofat V4.1-02/Jin/Stofat.exe /NAME="Shell_fatigue_analys	is" /INTER=PICK /STAT=
		-Import Beam (atigue_in.INL -Export Beam fatigue results.LIS	Script		
			DisablePreExecuteScript		C Goode To 1
			Preseoutescript	default	Edit Execute
		$\bigtriangledown$	PostExecuteScript	default	Edit Execute
	Connect (Inc	Comparison (and property and the first of the first	- X X		
	Dama codel and dynamic analysis.ProgramWester to * * West 25 (Version's) Basa_stippe, analysis.ProgramWestion * and version(Version's) Sama_stippe.Common. Sama_stippe.Common Sam	e Camero, 12,81,81,77,547); ex.Camero, 12,81,81,77,547); ex.Camero, 12,81,81,77,547); 14.1,27,1227;			
File Overview Applications	Command Line Message List Job Comments Activity Run Manager		Properties Attachments		

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:

Properties	- th X
GeniEActivity Beam_model_	_and_dynamic_analysis
Search	×
<ul> <li>General</li> </ul>	
Name	Beam_model_and_dynamic_analysis
Description	-Import a GNX file with model, frequency wave loads, run eigenvalue and dynamic analyses.
DatabaseName	Beam_model_and_dynamic_analysis
DatabaseStatus	New
Workspace	C:\DNV\Workspaces\WS_Shell_Fatigue\Beam_model_and_dynamic_analysis\
CmdInputFile	default Edit
UseLocalCmdInputFile	
PreviousCmdInputFile	$\label{eq:spaces} WS\_Shell\_Fatigue\Beam\_model\_and\_dynamic\_analysis\Shell\_model\_and\_dynamic\_analysis\_in.js \ \fbox{Clear}$
ImportFile	Clr Edit
SuperElementNumber	1
InputMode	Interactive
ProgramVersion	8.10.1 (64)

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):

Μ	<b>5</b> ♂ ⊽														
File	Home	View	Tool												
촵	G	D	H				XML	Zip		JS	XML	Zip			
New Job	New GeniE Job D	New DeepC Job	New HydroD Job	Open Jobs Job	Save Sav A	re Impo I JS	rt Import XML	Import ZIP	Import JNL	Export JS	Export XML	Export ZIP <del>•</del>	Run Job	Run Selected Activity	
	I	New		Open	Save			Imp	ort/Expo	ort				Process	



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.

Beam_model_and_dynamic_analys	is.gnx -	Genie (DirectX11)		
File Edit View Guiding Geometry	Stru	cture Loads Mesh & Ana	lysis Results Help	
🗅 🗃 🖬 📭 🗙 🕅 🖛 🎖	! 1	<b>*</b> & (*) 🗟 🕏 🗶	\$\$\$ <b>\$\$\$</b> \$ <b>\$</b> \$ <b>\$</b> \$	• 🖗 🕅 •
] • • ∧ • N • ⊙ • ≁ • ≯	* • •	<b>4 • Ⅲ • /^ •</b> < • ≯	< • Q: •   ∕ •   ■ • @	• <b>*</b> •   ;
▶ 44 🔳 - 🗮 📈 - 🗳			à 🕄 🛣 🗳 🖫 🐔 🕼 🕫	R
				×
Beam_model_and_dynamic_analy	sis	Name	Description	FEM Loadcas
🖻 🧰 Analysis		R:Kr DECK	Reference to LoadCase	4
Activities		R:Kr Eigenvalue.resultCase(1)	ResultCase	
🖻 💊 Eigenvalue			ResultCase	
Bigenvalue.step(	Acti	vity Monitor	ResultCase	
	Edit	Analysis	ResultCase	
Eigenvalue.step	Nev	Loadcase	ResultCase	
R Eigenvalue.step(	Nev	Load Combination	ResultCase	
	Crea	ate Scan Combinations	PerultCase	
Load Cases	View Guiding Geometry       Structure       Loads       Mesh & Analysis       Results       Help         Image: Structure       Image: Structur			
Capacity	Set	Active	ResultCase	
Compartments	Past	- <b>T</b>	ResultCase	
🗄 🧰 Environment	Dale	**	ResultCase	
🗄 💼 Equipment	Dele		Meshing (Always Regenerate)	
主 🧰 Properties	Ren	ame	Wave Load Analysis, Condition2	
🖻 💼 Structure	Field	ds	Linear Structural Analysis, Eigenvalue	
	Save	e HTML Report	Load Results	
Details	_	Kit Elgenvalue.vvLC(1, 1)	Wave load condition	5

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).



Activity Monitor			<u>?</u>
Done!		[	Start Cancel
Journal activity executions			
Activity	Duration	Status	Generate Input
1 - Eigenvalue - Analysis	13s	Warnings	
🛛 🖗 1.1 - Meshing (Always Rege	8s	Success	
1.1.1 - Delete loads	1s	Success	
1.1.2 - Generate loads	1s	Success	
1 1 3 - Delete mech	0s	Success	
1.1.5 - Delete mesh			
1.1.4 - Generate mesh	6s	Success	
1.1.4 - Generate mesh     1.2 - Wave Load Analysis, C	6s 2s	Success Warnings	No
1.1.3 - Delete mesh     1.1.4 - Generate mesh     1.2 - Wave Load Analysis, C     Kr     1.3 - Linear Structural Analy	6s 2s 3s	Success Warnings Success	No Yes
1.1.3 - Delete mesh     1.1.4 - Generate mesh     1.2 - Wave Load Analysis, C <b>Kr</b> 1.3 - Linear Structural Analy     R     1.4 - Load Results	6s 2s 3s 0s	Success Warnings Success Success	No Yes

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.						
С	T1	T2	Т3	Τ4	T5	Т6	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)



#### **STEP 2: BEAM FATIGUE ANALYSIS** 6

1. Click on the box Click on the box

 $\approx$ S Beam\_fatigue\_analysis -Perform fatigue analysis of a selected set of members and joints. -SCF's are based on Efthymiou equations -Import Beam fatigue in.JNL -Export Beam\_fatigue\_results.LIS

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

Properties		- ¶ ×
FrameworkActivity Beam_fa	atigue_analysis	
Search		×
<ul> <li>General</li> </ul>		
Name	Beam_fatigue_analysis	
Description	-Perform fatigue analysis of a selected set of members and joints.	•
DatabaseName	Beam_fatigue_analysis	
DatabaseStatus	New	~
Workspace	C:\DNV\Workspaces\WS_Shell_Fatigue\Beam_fatigue_analysis\	
CmdInputFile	C:\DNV\Workspaces\WS_Shell_Fatigue\_repository\Beam_fatigue_in.JNL	Edit
UseLocalCmdInputFile	$\checkmark$	
PreviousCmdInputFile	$\label{eq:c:DNV} Work spaces \ \ Shell_Fatigue \ Beam_fatigue\_analysis \ \ Beam_fatigue\_in. \\ JNL \ \ \ \ \ \ \ \ \ \ \ \ \ \ \ \ \ \ \$	Clear
ResultsFile		Clr
SuperElementNumber	1	 ▼
SuperElementKey	1	 ▼
InputMode	Interactive	~
ProgramVersion	4.4.0 (64)	~

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 LOT	JT3	**Fail**	6.07E+00
BM563	PIPE	JT5	**Fail**	4.66E+00
	G_1_DB3			
BM2339	PIPE G_1_X16	JT1	**Fail**	3.70E+00
BM647	PIPE	JT2	**Fail**	3.64E+00
	LQI			



# 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.

<ul> <li>General</li> </ul>		
Name	Shell_model_and_dynamic_analysis	
Description	-The js-input file will iImport same GNX from repository and convert critical joint to shell	•
DatabaseName	Shell_model_and_dynamic_analysis	
DatabaseStatus	New	~
Workspace	C:\DNV\Workspaces\WS_Shell_Fatigue\Shell_model_and_dynamic_analysis\	
CmdInputFile	default	Edit
UseLocalCmdInputFile		
PreviousCmdInputFile		Clear
ImportFile		Clr Edit
SuperElementNumber	1	*
InputMode	Interactive	$\sim$
ProgramVersion	8.10.1 (64)	~
SelectedLicenses	CurvedGeometry,FrameCodeCheck,PlateCodeCheck,RefineMesh,Earthquake	•

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 v(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).

? ×

R.-

• 87 A

,₀,<mark>B</mark>

- 8ªC

m x 🖻

m × F

• 8?



- 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:

Convert joint(s) to s	hells	? ×	Mesh Density	×
eneral Mesh Refiner	nent Zones   Mesh Transition Zones		Md_200mm 👻	
Joint	🗴 Jt3	<b>R</b> .•	Element Length 200mm m	
General options	1.0		Growth rate 1.05	
Delete converted bea	ams Make Converted Beams Nonstructural	▼ 8?	Enforce density on concept edges 🖓?	
Add rigid link	,		Element Length	
Disconnect the no	on-structural beams from the chord surface	<b>9</b> ?		
Mesh density	Md_200mm	<b>→</b> 83		
Mesh option	PSQ			
Joint options V Delete braces insi V Split beams at sm	ide chord 💡? art positions 💡?		OK Cancel Apply	
Split braces at			Mesh Option for Face	
Distance from jo	bint Distance 3.9	m 🔗 ?	PSQ 💌	
O Nearest segment	IT From chord surface ♀?		Force quad elements      Ŷ?     ✓ Override face edges mesh strategy      Ŷ?     ♥?	
Split chords at				
Oistance from jo	int Distance 7.2	m 8?	Verride mesh strategy 8?	
C Nearest segmen	T From furthest brace 8?		Patch surf quad mesher     Patch surf triangle mesher	
			C Advancing front quad mesher	
Set containing shells	Jt3_Shells	▼ 8?	C Advancing front triangle mesher C Sesam quad mesher Number of layers	
Create Refinemen	nt Zones 8?	_	C Template quad mesher	
Create Transition	Zones 8?		C Radial quad mesher 0 1 2 3 4	
	-			
	OK Cancel	Apply		
			OK Cancel	Ap



(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.

Brace	Layers	Total	Zone	Number of	Total Zone	4
Jt3.GetBrace(Br	2	-40m	m	76	-0.04093271495	
Jt3.GetBrace(Br	2	-40m	m	64	-0.04055220503	
Jt3.GetBrace(Br	2	-40m	m	99	-0.1455943155	
Jt3.GetBrace(Br	2	-40m	m	76	-0.0634344529	
Jt3.GetBrace(Br	2	-40m	m	106	-0.04093271495	
Jt3.GetBrace(Br	2	-40m	nm	64	-0.04055220503	
Set containing m First Brace	esh refinem	ent zone	Jt3Refiner ace	mentZones Through	▼ 8? Brace	¥
Set containing m First Brace	esh refinem	ent zone . Second Br	Jt3Refiner ace	mentZones Through	▼ §? Brace	*

(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.

Convert joint(s) to shells			? ×	Mesh Density	? ×
General Mesh Refinement Zones Me	sh Transition Zones	1			
Chord Offset Distance	-0.3360943216 m		m 😵 ?	Md_80mm	
Brace Offset Distance	-0.3810000122 m		m 🔗 ?	Element Length 80mm	m
Mesh density in mesh transition zones	Md_80mm	-	· 82	Growth rate 1.05	
Set containing mesh transition zone	Jt3TransitionZones		-   <mark>0</mark> ?	Enforce density on concept edges	83
	,	_	-	Element Length	
				v	
					OK Cancel Apply
					Cancer Apply
	ОК	Cancel	Apply		



(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 modalsuperposition method. Select all of them and modify to

me : *			
sition :   *			
Boundary Condition     C Boundary Stiffness Matrix     C Boundary Stil	ffness Per Ler	gth	
Boundary conditions			
Fixed Free Prescribed Dependent Super pring	Spring stiffness		
x	0 kN/m	[kN/m]	
у	0 kN/m	[kN/m]	
z	0 kN/m	[kN/m]	
I Let rx change ry and rz Fixed Free Prescribed Dependent Super Spring	Spring stiffness	85	
rxj	0 kN*m	[kN*m]	
ry	0 kN*m	[k/N*m]	
rz	0 kN*m	[kl\*m]	

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  $0.1 \times \sqrt{(r \times t)}$ . 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.

Create Mesh Priority Set	? ×
Name: Mpri1	
⊡ ♠ Priority 1	Add Mesh Priority
Jt3RefinementZones	Move Up
	Move Down
	Add Selection
	Add Selectio

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

Conceral EEM ontions	Other proferences
Use second order elements w?	
Superelement type 1	I Automatic load combination FEM numbering
lodel topology (Performed before mes	hing)
✓ Always simplify topology ♀?	I Adjust number of elements ♥?
Split periodic geometry if needed	
Repair split topology if possible	Eliminate internal eccentricities      ??
lement profesences	Ignore fillet on fillet sections §?
Drafer regular mach (m v n)	✓ Prefer linear dependencies
Prefer regular mesn (m x n)	Naming preferences
<ul> <li>Allow triangular elements</li> </ul>	✓ Use long Loadcase names
Prefer point mass as node mase	✓ Use long Set names
	Use long Property names
Use first order thick shell elements	V?
Use eccentric hinges V?	C Advancing Front Quad Mesher
<ul> <li>Longitudinal mass nonstruct element</li> </ul>	Its 9? C Advancing Front Triangle Mesher
Adjust edge density using ratio	
Ratio 0.1	C Template Quad Mecher
cantling idealizations	C Patch Surf Tri Machar
Thickness Gross	C Patch Surf Quad Masher
Ignore beam eccentricities 8?	Edge mesher
Use co-centric beams 9?	(• Uniform distribution
Loads based on gross dimensions	C Linear distribution
	Idealizations
	Remove internal vertices §?
	✓ Remove internal edges
	☐ Ignore structure not part of subset when idealizing ♀?
	□ Use uniformized face parameterization \$?



(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.



-	
Regenerate mesh option Alwa	ays Regenerate Mesh
Export beams as members	First level combinations as BSELL
Smart load combinations 8?	Parallelize load application §?
Include loads on mesh 9?	Parallelize mesh generation §?
Write FEM file	
Override Global Supereleme	nt Data
Top Superelement Type	1
Superelement Tune	1
and a second sec	
Mesh Subset Indude C Exclude	
Mesh Subset Include C Exclude Subset Dyn_FTG Forget all existing keep mes	sh flags Keep mesh of previous subset
Mesh Subset Indude C Exclude Subset Dyn_FTG Forget all existing keep me	sh flags Keep mesh of previous subset
Mesh Subset Include Exclude Subset Forget all existing keep mer Pile boundary condition	sh flags T Keep mesh of previous subset
Nesh Subset C Indude C Exclude Subset Dyn_FTG Forget all existing keep mer Pile boundary condition Pile Soil Interaction	sh Flags TKeep mesh of previous subset
Mesh Subset Indude Exclude Subset Dyn_FTG Forget all existing keep met Pile boundary condition Pile Soil Interaction Subset <a href="https://www.subset">subset</a>	sh flags T Keep mesh of previous subset

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.

File Falls Minus Cuiding Connectory Churchards Londo Mark & Analysis Desults Hele	Mesh activity	? ×
	Meshing Rules         Regenerate mesh option         Always Regenerate Mesh         P         Export beams as members         First level combinations as BSELL	
Shell_model_and_dynamic_analysis Name Description FEM Loadcase FEM LC Rule	Smart load combinations ♀?     I     Parallelize load application ♀?	
Analysis Analysis Activities G-Segenvalue.step(1) G-Segenvalue.step(2) Eigenvalue.step(3) R Eigenvalue.step(4) B-Stochastic_dynamic Load Cases Capacity Capacity	Indude loads on mesh       9?       Parallelize mesh generation       9?         Write FEM file       9?         Override Global Superelement Data         Top Superelement Type       1         Superelement Type       1         Mesh Priority       Image: Comparison of the second sec	•
	Subset  Dyn_FTG Forget all existing keep mesh flags Keep mesh of previous subset	

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

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

Activity Monitor				? X
😓 Done!		St	art	Cancel
Journal activity executions	Duration	Status	Generate Input	
I - Eigenvalue - Analysis	359s	Warnings	, centrate input ,	
☑ Ø 1.1 - Meshing (Always Rege	155s	Success		
1.1.1 - Delete loads	0s	Success		
1.1.2 - Generate loads	2s	Success		
<ul> <li>✓ 1.1.2 - Generate loads</li> <li>✓ 1.1.3 - Delete mesh</li> </ul>	2s Os	Success Success		- 1
<ul> <li>✓ 1.1.2 - Generate loads</li> <li>✓ 1.1.3 - Delete mesh</li> <li>✓ 1.1.4 - Generate mesh</li> </ul>	2s Os 153s	Success Success Success		
<ul> <li>✓ 1.1.2 - Generate loads</li> <li>✓ 1.1.3 - Delete mesh</li> <li>✓ 1.1.4 - Generate mesh</li> <li>✓ 1.2 - Wave Load Analysis, C</li> </ul>	2s Os 153s 7s	Success Success Success Warnings	No	
Image: Non-State State         Image: Non-State State         Image: Non-State         Imag	2s 0s 153s 7s 197s	Success Success Success Warnings Warnings	No Yes	
✓         1.1.2 - Generate loads           ✓         1.1.3 - Delete mesh           ✓         1.1.4 - Generate mesh           ✓         1.2 - Wave Load Analysis, C           ✓ <b>**</b> 1.3 - Linear Structural Analy           ✓         R           1.4 - Load Results	2s 0s 153s 7s 197s 0s	Success Success Success Warnings Warnings Success	No Yes	
Image: 1.1.2 - Generate loads         Image: 1.1.3 - Delete mesh         Image: 1.1.4 - Generate mesh         Image: 1.2 - Wave Load Analysis, C         Image: Image: 1.3 - Linear Structural Analy         Image: Image: R         Image: I	2s Os 153s 7s 197s Os	Success Success Success Warnings Warnings Success	No Yes	

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;
                                                  Period
                 Eigenvalue;
                                 Frequency;
            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).



08.52 Model R1 Vasalsse dynamics Length, Im) Force, Iv/(I)(Native) Model Devormed (210.507) Result case 1 1. Resp. trag.0.384411 Hz Nodes DISPLACE/MENT ALL Min. O Max: 0.00844396 Limited to current set

(5) Export the result SIN file to repository. You could use the name Shell\_model\_eigenvalue\_R1.SIN.



# 7.2 Stochastic dynamic analysis

 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.



egenerate mean option	Always Regener	ate Mesh
Export beams as member	ers	First level combinations as BSEL
Smart load combinations	8?	▼ Parallelize load application ♀?
Include loads on mesh	83	Parallelize mesh generation
Write FEM file	83	
Override Global Super	element Data	
Top Superelement Type	1	
Superelement Type	1	
Mesh Priority	e	•
Mesh Priority Mpri Mesh Subset G Indude C Exclud Subset Dyn_FTG Forget all existing ke	e ep mesh flags	Keep mesh of previous subset
Mesh Priority Mpril Mesh Subset Include Exclud Subset Dyn_FTG Forget all existing ker Pile boundary condition —	e ap mesh flags	Keep mesh of previous subset
Mesh Priority Mpril Mesh Subset Exclud Subset Dyn_FTG Forget all existing kee Pile boundary condition Pile Soil Interaction	L e ep mesh flags	Keep mesh of previous subset
Mesh Priority Mpri: Mesh Subset Exclud Subset Dyn_FTG Forget all existing kee Pile boundary condition Pile Soil Interaction Subset Chinne's	e ep mesh flags	Keep mesh of previous subset

(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.



near Analysis				?)
Datacheck Only Automatic gen	eration of input files			
Analysis type	Figenvalues			
C Static Analysis	Solver Multifro	nt Lanczos 👻	83	
C Figenvalue Analysis	Number of Modes	10		
Dynamic Response	Shift	0		
C Direct Integration <ul> <li>Modal Superposition</li> </ul>	Modal Mass Facto	rs 🔽 Mass Matrix	<b>8</b> ?	
Dynamic Domain Frequency Domain 💌 🔽	Use Modal Damping	Overall Modal Damp	ing 0.02	
Time Step 0.1 Adaptive	e Time Stepping	Time Step Tolerance	0.01	
	State Colution 92		129 00	
Iv Steady:	state solution with	- FFT Number	120 ¥?	
Max Cycles 100	Store Cycles			
Store Reaction Forces Reaction	Forces Center Point(0 m,	0 m,-93 n 🔽 St	atic Back Substitution	
	,			
ESL times option   Input Loads V ESL	Factor 1	ESL Load Comb   0	85	
Store ESL Results Interface File	Use text format for ESL file			
Advanced				
₩ Warp Correction (4 node elements)	Continue on element s	hape error 🔽 S	elect a subset for results gene	ration
Result file format SIN Norsam	▲ 8 <sup>3</sup>		Outer_layer	-
1				
✓ Use Sestra 10 💡	Use Multifront solver v	with LDLt factorisation	83	
			and an and the latent of the file	0-
Include pags from interface files in re-	suit file 🤡 ?   Ind	ude beam element ford	es and moments in result file	83
✓ Include shell element stresses in result	t file 🛛 🖓 ?			
Stress stiffening				
Enable stress stiffening		✓ 8?		
		OK Cance	Apply	

(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.

		1	Start	Cancel
Journal activity executions Activity	Duration	Status	Generate Inp	ut
• 90 1 - Stochastic_dynamic - Anal	778s	Warnings		
🗹 🖗 1.1 - Meshing (Always Rege	150s	Success		
1.1.1 - Delete loads	0s	Success		
	15	Success		
1.1.2 - Generate loads				
<ul> <li>1.1.2 - Generate loads</li> <li>1.1.3 - Delete mesh</li> </ul>	Os	Success		
1.1.2 - Generate loads     1.1.3 - Delete mesh     1.1.4 - Generate mesh	0s 149s	Success		
	0s 149s 30s	Success Success Warnings	No	
1.1.2 - Generate loads     1.1.3 - Delete mesh     1.1.4 - Generate mesh     2.1.4 - Generate mesh     2.2. Wave Load Analysis, S     1.3 - Linear Structural Analy	0s 149s 30s 598s	Success Success Warnings Warnings	No Yes	



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.

Shell\_fatigue\_analysis -Perform shell fatigue analysis of the relevant parts on the critical joint - Import Shell\_fatigue\_in.JNL - Export Shell\_fatigue\_results.LIS and export to vtf for viewing in Xtract

(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".

Properties		- ¶ ×
StofatActivity Shell_fatigue	_analysis	
Search		×
🖌 General		
Name	Shell_fatigue_analysis	
Description	-Perform shell fatigue analysis of the relevant parts on the critical joint	•
Database, 'ame	Shell_fatigue_analysis	
DatabaseStatus	New	~
Workspace	C:\DNV\Workspaces\WS_Shell_Fatigue\Shell_fatigue_analysis\	
CmdInputFile	C:\DNV\Workspaces\WS_Shell_Fatigue\_repository\Shell_fatigue_in.JNL	Edit
UseLocalCmdInputFile		
PreviousCmdInputFile	$\label{eq:c:DNV} Work spaces \ \ Shell_Fatigue \ \ Shell_fatigue \ \ analysis \ \ Shell_fatigue \ \ in \ \ JNL$	Clear
ResultsFile		Clr
SuperElementNumber	1	*
SuperElementKey	1	 ▼
InputMode	Interactive	~
ProgramVersion	4.1.2 (32)	Ŷ

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).

<ul> <li>Image: Shell_model_eigenvalue_R1.SIN</li> <li>Image: Shell fatigue analysis</li> <li>Image: Input</li> <li>Image: Output</li> </ul>	Exclude Explore Open
— ►□ Stofat.vtf	Open with Notepad
- •□ Jt3 shell fatique.LIS - •□ STOFAT.MLG	View with Xtract <b>T</b> Delete



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

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	
-------------	--

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.



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.