

SESAM EXAMPLE

Eigenvalue Analysis for Jacket Structure





Sesam Example

Eigenvalue Analysis for Jacket Structure

Date: May 2024

Prepared by: Digital Solutions at DNV

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

E-mail sales: digital@dnv.com

© DNV AS. All rights reserved

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



Table of contents

Contents

1	Introduction	.1
2	Converting the load into point mass	.2
3	Create new Wave Load Conditions	.3
4	Create new analysis	.3
5	Running the analysis	.4
6	View the results	.6
About DN	V	.7
Digital Sol	utions	.7



1 Introduction

This document is about Eigenvalue Analysis for a jacket structure using GeniE and Sestra. It is recommended to use the latest version of GeniE and Sestra for this example. You may check and download the latest version of the programs at https://sesam.dnv.com/. The example is run in GeniE. Download Model_Eigenvalue.zip file. Unzip the file and there will be two GNX files (Model_Eigenvalue_Completed.gnx and Model_Eigenvalue_start.gnx). Launch GeniE and import Model_Eigenvalue_start.gnx to start. In GeniE, go to File > Import Workspace (GNX) and search for Model_Eigenvalue_start.gnx.

The model is shown in **Figure 1-1**. This is a four-legged jacket structure with diagonal X braces. The jacket is supported by four stiffness springs. There are two decks at the topside with plates.

This workspace was created in SI units (N, m and C). The following programs and versions are recommended to run this example: GeniE 8.9-04, Wajac 7.11-01, Sestra 10.17-02 and Xtract 6.1-01



Figure 1-1 Jacket model for this example



2 Converting the load into point mass

Under Analysis folder, click on Load Cases folder. Select LCEq (equipment load) by left click on the load case. Right click on it again and select Properties:



This will open Load Case properties for LCEq. Under Structural Analysis Load and Mass management, check Placed equipment and select Vertical-Beams-And-Mass from the drop-down menu. Change the FEM load case number to 100:

Load Case Properties: LCEc	1			>
eneral Equipment Loads	Rotation Field Wind Field Desi	gn Condition		
Environment			. 1	
Acceleration field: Vector	3d(0 m/s^2, 0 m/s^2, -9.80665)	m/s^2)		
Structural Analysis Load and	Mass management			
Delete Explicit Loads	Generate Applied Loads			
Convert to loadcase indep	endent mass	1		
Point and line loads				
Placed equipment	/ertical-Beams-And-Mass			
Include structure self-we	eight in structural analysis 🛛 🔽	⁷ Include structure n	nass with rotation field	
Sum over Equipments				
Mass [kg]:	1e+06			
COG [m]:	(3.925, 0.2625, 155.475)			
Applied load [N]:	Fx=0, Fy=0, Fz=0			
Conceptual load [N]:	Fx=0, Fy=0, Fz=-9.80665e+	06		
Explicit conceptual load [N]:	No loads			
Total applied load [N]:	Fx=0, Fy=0, Fz=0			
External matrices load:	No			
E	loot		 Display in Input Units 	
IF FEM Loadcase number:	100		C Display in Database Units	

Click OK or Apply and then Cancel.

Note : A beam cross section and a material must be assigned to the equipment. Right-click on the equipment, select Properties and go to the Section and Material. A suitable beam cross section for the auxiliary beam stubs is a medium to weak one, possibly with reduced shear stiffness. They have a length of 1 m only to limit deformations in the dynamic analysis. A suitable material is one with zero density so as not to add mass to the model. The equipment in this example has been assigned to Deq1 section and DMat1 material.

|SESAM EXAMPLE | Eigenvalue Analysis of Jacket Structure |



3 Create new Wave Load Conditions

Under Environment folder, right-click on Location1 and select New Wave Load Conditions. Select CalmSea conditions under wave model:

terministic Time	83					
Name Condition 1						
ave components	Assign	wave component proper	ties			
?	<mark>8</mark> ?					
Regular wave set:	→ Curr	ent profile:	Ŧ			
	E Wind	d profile:	*]		
irection set:	🛫 🗆 🗆 Wav	ve model:	v]		
requency set:	- Ord	ler:				
Phase set:	*	Fill all	82			
Wave beight set:		Fill colocted	00			
* Maria balabb frankland		1 m selected				
wave neight rundlichen:	F	ill equal components	83			
		in Equal components				
cify value: 💡		in equal components				
cify value: 0? Period Height Phase Direction	Current profile	Wind p	ofile	Wave m	odel	Order
cify value: Period Height Phase Direction	Current profile	Wind p	ofile	Wave m CalmSea CalmSea	odel	Order
afy value: 9? Period Height Phase Direction	Current profile	Wind p	ofile	Wave m CalmSea CalmSea	odel	Order
dfy value: Period Height Phase Direction	Current profile	Wind p	ofile	Wave m CalmSea CalmSea	odel	Order
dfy value:	Current profile	Wind p	ofile	Wave m CalmSea CalmSea	odel	Order
dfy value: 9/2 Period Height Phase Direction	Current profile	Wind p	ofile	Wave m CalmSea CalmSea	odel	Order
cify value:	Current profile	Wind p	ofile	Wave m CalmSea CalmSea	odel	Order

Click OK.

4 Create new analysis.

Right click on the activities folder and select New Analysis. Check Linear Structural Analysis and Wave Load Activity, select Condition1, set the analysis to be Eigenvalue and use Sestra 10 for the solver. Click OK:

eate Wave Load + Linear Eigenvalu	e Analysis	<u>? ×</u>
Name Analysis1		
 ✓ Lock concepts after meshing ✓ Automatically import global loade 	<mark>0</mark> ? Tases	
Available activities	C Static	
F Hull Girder Load Adjuster	Eigenvalue	
Local Analysis	CDynamic	
Wave Load Activity	Use Equivalent Static Loads -	Fatigue 💡?
	Use Equivalent Static Loads -	ULS <mark>9</mark> ?
	Wave Load Condition	
Indep, Tank Coupling Analysis	Condition 1	
I♥ Load Results	✓ Use Sestra10 Ø?	
FEM analysis units		
Length m 🛒		
Force N -		
Temperature delC 🚽		
		Crawl

New Analysis1 is now created. Exclude all load cases except LCEq from the analysis.



5 Running the analysis

Go to Activity Monitor (Alt + D).

Right click on 1.2 Wave Load Analysis. Under Added mass and damping, set as below:

${\bf \widehat{\bullet}}$ Load calculation \subset Added mass and damping only \square Data check only	✓ Automatic generation of input files ♥?
Deterministic seastates Added mass and damping Rules Buoyancy Mo	orison Output Special options
Calculate added mass	Ø?
V Indude internal water	85
✓ Use C_m=2.0	<mark>8</mark> 5
$\overrightarrow{\mathbf{V}}$ Include longitudinal mass of marine growth and internal water	<mark>8</mark> 3
Include longitudinal hydrodynamic added mass	<u>8</u> ;
Calculate damping	8 3
Include longitudinal damping	83

Under Buoyancy tab, set as below:

eterministic	seastates Added mass and da	amping Rules	Buoyancy	Morison	Output	Special options
Buoyancy	/ forces with non-horizontal wate	er-plane				
83	• Assuming horizontal free sur	face				
8?	C Using actual free surface					
End Force	• Include end forces for all me	mbers (rational i	method)			
83	C Exclude end forces for all me	embers (marine r	method)			
83	C Include end forces for non-f	looded members	only			
Include	Buoyancy due to steel area	<mark>8</mark> ?				
Indude	buoyancy of beams at mudline	83				

Click Apply.

Go to Deterministic seastates tab, and select Buoyancy On (from drop down menu) for the first seastate and select Weight for the second seastate:

Maximum overturning moment Maximum overturning moment Moseler stretching	urrent blockage Vave kinematics Water depth, ? Fill selecte	e factor: 1 s factor: 1 s/level: 0 ed		▼ [m] \$?									
ecify value: @?	Wave mod C	Order Current	Wind	Stratching	Step length Idea	Numetane	Buoyancy	Design load	Current h fac	Waya k fac	Water levels	Dopplar Effect	110 000
1	CalmSea	Sider Current	WING	NoStretching	0 deg	1	On	NoDesign oads	1	1	0 m	Off	101
	CalmCaa	_		No Otrotohing	0 deg	4	Wataki	NoDesignLouds	-	4	0	0#	107



Click OK.

Right click on 1.3 Linear Structural Analysis and change Number of Modes to 20:

iear Analysis	?
Datacheck Only ✓ Automatic gene Analysis type ✓ Static Analysis ✓ Eigenvalue Analysis ✓ Dynamic Response	ration of input files Eigenvalues Solver Multifront Lanczos
Dynamic Domain Time Domain V	Use Modal Damping Alpha1 0.01 Alpha2 0.001 9?
✓ Use Cyclic Loads ✓ Steady S	tate Detection Steady State Tolerance 0.01
Max Cycles 100	Forces Center Point(0 m, 0 m, 0 m)

Keep other settings as it is and click OK.

Click Start to run the analysis. Check the analysis status:

Don	ne!			Start	Cancel
Jour	rnal activity executions	Duration	Status	Generate Input	
9 %) 1 B 1 1	 Analysis 1 - Analysis 1.1 - Meshing (Conditional R 1.1.1 - Delete loads 1.1.2 - Generate loads 1.1.3 - Delete mesh 	4s 0s 0s 0s	Success Success		
	1.1.4 - Update mesh 1.2 - Wave Load Analysis, C 1.3 - Linear Structural Analy 1.4 - Load Results	0s 2s 2s 0s	Success Warnings Success Success	Yes Yes	



6 View the results.

The results may be examined in GeniE. Change the view to Results – With Mesh and check the displacement for each result case.

For more details such as viewing of mode shapes animation, click on Results > Advance Results (Xtract). This will open the result file in Xtract. In Xtract, go to Animation > Mode Shape Animation. Use default settings for number of frame and click OK. Click **v** button to play the animation.



The print of all Eigenvalues is listed in Sestra.lis file:

```
Print of eigenvalues.
Eigenvalues have unit sec^-2; frequency = sqrt(eigenvalue) / (2 * pi); period = 1 / frequency.
         Number;
                       Eigenvalue;
                                           Frequency;
                                                                   Period
                1; 6.784234e+00; 4.145439e-01; 2.412290e+00
                2; 8.752629e+00; 4.708574e-01; 2.123785e+00
               3; 1.394419e+01; 5.943152e-01; 1.682609e+00
4; 2.488864e+01; 7.940004e-01; 1.259445e+00
5; 2.697936e+01; 8.266771e-01; 1.209662e+00
                6; 4.109720e+01; 1.020296e+00; 9.801076e-01
                    8.935131e+01; 1.504425e+00; 6.647057e-01
                7;
                    9.075874e+01; 1.516227e+00; 6.595316e-01
1.039518e+02; 1.622692e+00; 6.162599e-01
                8;
                9;
               10; 1.039647e+02; 1.622793e+00; 6.162215e-01
               11; 1.039691e+02; 1.622827e+00; 6.162084e-01
              12; 1.039752e+02; 1.622875e+00; 6.162084e-01
13; 1.040685e+02; 1.623603e+00; 6.159142e-01
14; 1.041469e+02; 1.624215e+00; 6.156822e-01
               15; 1.042072e+02; 1.624684e+00; 6.155042e-01
               16; 1.042345e+02; 1.624897e+00; 6.154235e-01
                    1.254252e+02; 1.782430e+00; 5.610319e-01
1.267107e+02; 1.791541e+00; 5.581786e-01
               17;
                                                          5.610319e-01
               18;
               19; 1.637114e+02; 2.036384e+00; 4.910666e-01
               20; 1.689654e+02; 2.068803e+00; 4.833714e-01
```



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.