

SESAM EXAMPLE

Earthquake Response Spectrum Analysis in GeniE





Sesam Example

Jacket Earthquake Analysis Performed in GeniE

Date: April 2025

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

Table of c	ontents	3
1	Introduction	1
2	Open New GeniE Workspace and Import Model	1
3	Earthquake Conditions	2
4	Earthquake Response Spectrum Analysis	3
4.1	Eigenvalue Analysis	3
4.2	Static Analysis	5
4.3	Earthquake Response Spectrum Analysis	6
4.3.1	Input Earthquake Condition	6
4.3.2	Create and Run Earthquake Response Spectrum Analysis	6
4.3.3	Check Analysis Results	9
5	Beam and Joint Code Checks	10
5.1	Beam Code Checks	10
5.2	Joint Minimum Strength Code Checks	12



1 Introduction

In an earthquake response spectrum analysis, the foundation linearization needs to be performed first. After the equivalent springs are applied at pile heads, the response spectrum earthquake analysis can be performed. The analysis workflow is like below.



This example skips the foundation linearization and focuses on the earthquake response spectrum analysis. The jacket model is shown in the figure below (equivalent springs are at pile heads). The units used in the example is kN and m.

The earthquake response spectrum analysis is controlled by GeniE. There are three analyses included, the Eigenvalue analysis, the static load analysis, and the earthquake response spectrum analysis. Capacity managers are created after beam forces are generated. Member and joint code checks are performed in capacity managers.



2 Open New GeniE Workspace and Import Model

The license Earthquake is required to create and perform an earthquake response spectrum analysis.

To create all analyses included in GeniE from a new GeniE workspace, do the following:

• Open a new GeniE workspace, check the license for earthquake analysis, and create a new workspace. Units are kN



and m.

✓

-				Workspace name: C Overvirite Existing file	
	License	Use	-	GenE_EQK C Regenerate	
1	CurvedGeom	2	_	G Keep existing	
2	FrameCodeC	<u>v</u>	- 11	Locadon:	
3	Patecodeche	×	- 11	C:///	
5	SesamCore	C	_	Create directory for workmans 00	
6	Earthouake	2	-	1 Contraction & Low Manusplace 8 F 1 Store Monopole anectory	
	inforce use of Ger inforce use of Ger	nE. RCLite license nE. ShipMode licens nE. Starter license	e	P? Length m Porce KN Temperature delc Porce Connected copy on by default Use Dual Assembly P?	
I D	to not show this d	lalog again OK		OK Cancel App	ly
					_

3 Earthquake Conditions

API acceleration spectrum is used in this example. Based on the information of the structural site and the exposure level, the below earthquake acceleration spectrum will be used in the earthquake response spectrum analysis.



No.	ω	Sa,ELE(m/s^2)
1	628.3185	1.2123
2	125.6637	1.5507
3	62.8319	1.9736
4	41.8879	2.3965
5	31.4159	2.8194
6	25.1327	2.8194
7	19.6350	2.8194
8	12.5664	1.8044
9	8.3776	1.2029
10	6.2832	0.9022
11	4.1888	0.6015
12	3.1416	0.4511
13	2.8690	0.4120
14	2.5133	0.3609
15	2.0944	0.3007
16	1.7952	0.2578
17	1.5708	0.2256
18	1.2566	0.1444
19	1.0472	0.1002
20	0.8976	0.0736



The above acceleration spectrum and the damping ratio of 0.05 will be applied in the global X, Y, and Z directions.

4 Earthquake Response Spectrum Analysis

The earthquake response spectrum analysis includes three analyses, the Eigenvalue analysis, the static load analysis, and the earthquake response spectrum analysis. All three analyses are set up as below.

4.1 Eigenvalue Analysis

✓ Create a new Eigenvalue analysis and name it as Analysis1_Eigen. Wave Load Activity and Linear Structural Analysis are included.

Name	Analysis1_Eigen			
	ck concepts after meshing (<mark>?</mark> ? ases		
Availab	le activities eshing ull Girder Load Adjuster cal Analysis ave Load Activity ear Structural Analysis e Soil Analysis e Soil Analysis ension/Compression Analysis dep. Tank Coupling Analysis ad Results rthquake	C Static C Eigenvalue C Dynamic Use Equivalent Use Equivalent Wave Load Condition Condition 1 Vse Sestra 10	Static Loads - Fatigue Static Loads - ULS 9? •	83 83
Force Temper	M analysis units ??		OK Can	cel

Insert Load Combination

- ✓ Only masses are required in the Eigenvalue analysis. Then Load case LCBuoy can be excluded from the analysis.
- ✓ The structural mass is included automatically. Two load cases, LCEq, and LCLLoad need to be converted to mass and then included in the analysis. A load combination LComb_Mass is created to convert the equipment loads and user defined point and line loads to masses.
- Name:
 LComb_Mass

 Convert to loadcase independent mass
 ✓

 ✓ Point and line loads
 Global scale factor:
 1

 ✓ Placed equipment
 Footprint-Mass
 ✓

 Load Case
 Factor
 Phas...
 Scan ...

 Load Case
 Factor
 Phas...
 Scan ...

 Load Case
 Factor
 Phas...
 Scan ...

 Description

 false
 Reference to L....

 Ø Factor
 1
 0
 false
 Reference to L....

 ØK
 LOLload
 1
 OK
 Cancel

? ×

- \checkmark Set up the below options for each step.
 - ✓ Edit Mesh Activity: Choose *Always Regenerate Mesh* and *Exclude Piles* as Pile boundary condition.
 - ✓ Edit Wave Load Run: It is not necessary to calculate wave loads. Choose mass calculation options on *Added* mass and damping only tab.

DNV		
Mesh activity ? ×		
Meshing Rules Regenerate Mesh	Edit Wave Load Run Run: Analysis1_Eigen.step(2) V Wave load condition:	? ×
Export beams as members First level combinations as BSELL Smart load combinations Praileize load application P? Indude loads on mesh P? Write FPM file P?	○ Load calculation ● Added mass and damping only □ Data check only Deterministic seastates Added mass and damping Rules Buoyancy Mo	Automatic generation of input files 💡?
Override Global Superelement Data Top Superelement Type I Superelement Type I	Calculate added mass The control water	8. 8.
Mesh Priority Mesh Subset Distriction	 Include longitudinal mass of marine growth and internal water Include longitudinal hydrodynamic added mass 	85 85 85
Subset Keep mesh of previous subset	Calculate damping	85 85
Pile boundary condition Exclude Piles ©? Subset		

✓ Edit Linear Sestra Analysis: The number of modes is 70. Two runs need to be performed. The options and the purpose of each run are as below.

Linear Analysis

 Check the option *Mass Matrix* and click *OK* to close the dialog box. Run the analysis to generate the mass matrix file M1.SIF.

NOTE: If the model contains supernodes, GeniE created Sestra.inp file includes **DRED** card. User needs to manually generate the Sestra.inp file by unchecking Automatic generation of input files, manually delete **DRED** card to revise the Sestra.inp file, and then perform the analysis with the revised Sestra input file.

 Uncheck Mass Matrix, check Modal Mass Factors, and make sure Automatic generation of input file is selected, and click OK to close the dialog box. Run the analysis to create the result file _R1.SIN

Static Analysis Solver Multifront Lanczos P? Bigenvalue Analysis Number of Modes 70 Shift 0 Direct Integration P? Modal Mass Factors Mass Matrix P? inear Analysis P Modal Mass Factors Mass Matrix P? inear Analysis P Eigenvalue Analysis P? C Static Analysis Solver Multifront Lanczos P? C Static Analysis Solver Multifront Lanczos P C Static Analysis Solver Multifront Lanczos P C Static Analysis Solver Multifront Lanczos P Multifront Lanczos P Solver Solver Multifront Lanczos P P P Solver Multifront Lanczos P P Shift D P P S	naiysis type	Eigenvalues		
● Eigenvalue Analysis Number of Modes 70 ● Direct Integration Shift 0 ● Direct Integration P? ■ Modal Mass Factors ● Mass Matrix P? inear Analysis ■ Datacheck Only ■ Automatic generation of input files ■ Analysis type ■ </td <td>O Static Analysis</td> <td>Solver Multifro</td> <td>ont Lanczos</td> <td>✓ 8?</td>	O Static Analysis	Solver Multifro	ont Lanczos	✓ 8?
Opynamic Response Shift 0 O Direct Integration Modal Superposition Image: Modal Mass Factors Image: Mass Matrix Image: Mas	C Eigenvalue Analysis	Number of Modes	70	
● Direct Integration ●? ● Modal Superposition ●? ■ Modal Mass Factors ● Mass Matrix ● Integration ●? ■ Modal Mass Factors ● Mass Matrix ● Datacheck Only ■ Automatic generation of input files Analysis type ● ● Static Analysis ● ● Direct Integration ● ● Direct Integration ●	O Dynamic Response	Shift	0	_
Modal Superposition P? Inear Analysis Datacheck Only Analysis type C Static Analysis C Static Analysis C Dynamic Response (* Direct Integration	O Direct Integration			
	O Modal Superposition 8?	Modal Mass Facto	mass Mass M	latrix V?
Inear Analysis Datacheck Only Automatic generation of input files Analysis type Static Analysis Eigenvalue Solver Multifront Lanczos Dynamic Response O Dynamic Response O Direct Integration To the total of total of the total of total of the total of total				
Datacheck Only Automatic generation of input files Analysis type Static Analysis Eigenvalue Analysis Dynamic Response O Direct Integration Direct Integration	Linear Analysis			
Datacheck Only Automatic generation of input files Analysis type Static Analysis Eigenvalue Analysis Dynamic Response (© Direct Integration () () ()			-	
Analysis type Eigenvalues C Static Analysis Solver Multifront Lanczos ~ © Eigenvalue Analysis Number of Modes 70 © Dynamic Response Shift 0	Datacheck Only Automa	tic generation of input file	s	
C Static Analysis Solver Multifront Lanczos C Eigenvalue Analysis Number of Modes 70 Dynamic Response Shift 0 O	Analysis type	Eigenvalues		
Eigenvalue Analysis Number of Modes 70 Dynamic Response Shift 0 O	C Static Analysis	Solver	Multifront Lancz	os 🚽 💡
C Dynamic Response Shift 0	Eigenvalue Analysis	Number of M	lodes 70	
© Direct Integration	C Dynamic Response	Shift	0	
	C Direct Integration		1.	_
C Modal Superposition 8?	C Modal Superposition	P? Modal Ma	iss Factors	 Mass Matrix
	Linear Analysis			
inear Analysis				
inear Analysis	🗌 Datacheck Only 🛛 🗹 Automatic	generation of input files		
inear Analysis	Analysis type	Eigenvalues		
inear Analysis Datacheck Only Automatic generation of input files Analysis type Eigenvalues		Solver M	Iultifront Lanczos	
Inear Analysis Datacheck Only Automatic generation of input files Analysis type Static Analysis Solver Multifront Lanczos	Static Analysis			
Inear Analysis Datacheck Only Q Automatic generation of input files Analysis type Static Analysis Eigenvalue Analysis Eigenvalue Analysis Number of Modes 70	 Static Analysis Eigenvalue Analysis 	Number of Mod	es 70	

The generated mass matrix file **M1.SIF** and the result file **_R1.SIN** are in the analysis folder. They will be used later in the earthquake response spectrum analysis.

Modal Superposition 8?

NOTE: The mass matrix must be created before the Eigen values and mode shapes are calculated, i.e, the analysis creating mass matrix must be run first, and the analysis calculating Eigen values and mode shapes must be run secondly. To make M1.SIF generated faster, it is suggested that Sestra10.19-00 is used.

✓ Generate the mass participation report using *Save Report* feature. The number of modes included in the earthquake response spectrum analysis is based on the total mass participation ratios listed in columns AMM-X, AMM-Y, and AMM-Z.



-		-			-					_	
ResultCase	AngFreq [rad/s]	Period [s]	EMM-X [tonne]	EMM-Y [tonne]	EMM-Z [tonne]	DMM-X	DMM-Y	DMM-Z	AMM-X	AMM-Y	AMM-Z
Analysis1_Eigen.resultCase(1)	1.4381	4.3691	0.05202	12434.30000	0.38309	0.00000	0.82928	0.00003	0.00000	0.82928	0.00003
Analysis1_Eigen.resultCase(2)	1.7640	3.5620	12910.30000	0.12163	0.17539	0.86533	0.00001	0.00001	0.86534	0.82929	0.00004
Analysis1_Eigen.resultCase(3)	3.2890	1.9104	15.07190	22.77650	0.06264	0.00101	0.00152	0.00000	0.86635	0.83081	0.00005
Analysis1_Eigen.resultCase(4)	4.5271	1.3879	6.01972	2315.17000	7.79755	0.00040	0.15441	0.00061	0.86675	0.98521	0.00066
Analysis1_Eigen.resultCase(5)	4.6845	1.3413	1801.87000	4.22854	1.96354	0.12077	0.00028	0.00015	0.98752	0.98549	0.00082
Analysis1_Eigen.resultCase(65)	27.7277	0.2266	0.05787	2.27255	3.15986	0.00000	0.00015	0.00025	0.99680	0.99755	0.94184
Analysis1_Eigen.resultCase(66)	27.9545	0.2248	0.13787	6.10419	14.37090	0.00001	0.00041	0.00113	0.99681	0.99795	0.94297
Analysis1_Eigen.resultCase(67)	28.0462	0.2240	2.65173	2.10261	0.82429	0.00018	0.00014	0.00006	0.99699	0.99809	0.94304
Analysis1_Eigen.resultCase(68)	28.5633	0.2200	0.01018	0.20718	0.17171	0.00000	0.00001	0.00001	0.99699	0.99811	0.94305
Analysis1_Eigen.resultCase(69)	28.6271	0.2195	0.01798	0.38393	7.41770	0.00000	0.00003	0.00058	0.99699	0.99813	0.94364
Analysis1_Eigen.resultCase(70)	29.1030	0.2159	0.05270	0.00715	13.90960	0.00000	0.00000	0.00109	0.99700	0.99813	0.94473
	ResultCase Analysis1_Eigen.resultCase(1) Analysis1_Eigen.resultCase(2) Analysis1_Eigen.resultCase(3) Analysis1_Eigen.resultCase(4) Analysis1_Eigen.resultCase(5) Analysis1_Eigen.resultCase(65) Analysis1_Eigen.resultCase(66) Analysis1_Eigen.resultCase(67) Analysis1_Eigen.resultCase(68) Analysis1_Eigen.resultCase(69) Analysis1_Eigen.resultCase(70)	ResultCase AngFreq [rad/s] Analysis1_Eigen.resultCase[1) 1.4381 Analysis1_Eigen.resultCase[2) 1.7640 Analysis1_Eigen.resultCase[3) 3.2890 Analysis1_Eigen.resultCase[4) 4.5271 Analysis1_Eigen.resultCase[5] 4.6845 Analysis1_Eigen.resultCase[6] 27.7277 Analysis1_Eigen.resultCase[65] 27.7277 Analysis1_Eigen.resultCase[65] 28.0462 Analysis1_Eigen.resultCase[66] 28.5633 Analysis1_Eigen.resultCase[69] 28.6271 Analysis1_Eigen.resultCase[69] 28.6271 Analysis1_Eigen.resultCase[67] 28.021	ResultCase AngFreq [rad/s] Period [s] Analysis1_Eigen.resultCase(1) 1.4381 4.3691 Analysis1_Eigen.resultCase(2) 1.7640 3.5620 Analysis1_Eigen.resultCase(3) 3.2890 1.9104 Analysis1_Eigen.resultCase(4) 4.5271 1.3879 Analysis1_Eigen.resultCase(5) 4.6845 1.3413 Analysis1_Eigen.resultCase(5) 27.7277 0.2266 Analysis1_Eigen.resultCase(66) 27.9545 0.2248 Analysis1_Eigen.resultCase(67) 28.0462 0.2240 Analysis1_Eigen.resultCase(69) 28.5633 0.2200 Analysis1_Eigen.resultCase(69) 28.6271 0.2195 Analysis1_Eigen.resultCase(67) 29.1030 0.2195	ResultCase AngFreq [rad/s] Period [s] EMM-X [tonne] Analysis1_Eigen.resultCase(1) 1.4381 4.3691 0.05202 Analysis1_Eigen.resultCase(2) 1.7640 3.5620 12910.30000 Analysis1_Eigen.resultCase(2) 1.7640 3.5620 12910.30000 Analysis1_Eigen.resultCase(3) 3.2890 1.9104 15.07190 Analysis1_Eigen.resultCase(4) 4.5271 1.3879 6.01972 Analysis1_Eigen.resultCase(5) 4.6845 1.3413 1801.87000 Analysis1_Eigen.resultCase(65) 27.7277 0.2266 0.05787 Analysis1_Eigen.resultCase(66) 27.9545 0.2248 0.13787 Analysis1_Eigen.resultCase(66) 28.5633 0.2200 0.01018 Analysis1_Eigen.resultCase(68) 28.6271 0.2195 0.01798 Analysis1_Eigen.resultCase(69) 28.6271 0.2155 0.01798 Analysis1_Eigen.resultCase(67) 29.1030 0.2159 0.05270	ResultCase AngFreq [rad/s] Period [s] EMM-X [tonne] EMM-Y [tonne] Analysis1_Eigen.resultCase(1) 1.4381 4.3691 0.05202 12434.30000 Analysis1_Eigen.resultCase(2) 1.7640 3.5620 12910.30000 0.12163 Analysis1_Eigen.resultCase(3) 3.2890 1.9104 15.07190 22.77650 Analysis1_Eigen.resultCase(4) 4.5271 1.3879 6.01972 2315.17000 Analysis1_Eigen.resultCase(5) 4.6845 1.3413 1801.87000 4.22854 Analysis1_Eigen.resultCase(6) 27.7277 0.2266 0.05787 2.27255 Analysis1_Eigen.resultCase(66) 27.9545 0.2244 0.13787 6.10419 Analysis1_Eigen.resultCase(66) 28.5633 0.2200 0.01018 0.20718 Analysis1_Eigen.resultCase(69) 28.6271 0.2155 0.01798 0.38393 Analysis1_Eigen.resultCase(670) 29.1030 0.2159 0.00715 0.00715	ResultCase AngFreq [rad/s] Period [s] EMM-X [tonne] EMM-Y [tonne] EMM-7 [tonne	ResultCase AngFreq [rad/s] Period [s] EMM-X [tonne] EMM-Z [tonne] EMM-Z [tonne] DMM-X Analysis1_Eigen.resultCase[1) 1.4381 4.3691 0.05202 12434.30000 0.38309 0.00000 Analysis1_Eigen.resultCase[2) 1.7640 3.5620 12910.30000 0.12163 0.17539 0.86533 Analysis1_Eigen.resultCase[3) 3.2890 1.9104 15.07190 22.77650 0.06264 0.00101 Analysis1_Eigen.resultCase[4) 4.5271 1.3879 6.01972 2315.17000 7.79755 0.00404 Analysis1_Eigen.resultCase[65) 4.6845 1.3413 1801.87000 4.22854 1.96354 0.12077 Analysis1_Eigen.resultCase[65) 27.7277 0.2266 0.05787 2.27255 3.15966 0.00000 Analysis1_Eigen.resultCase[66) 27.9545 0.2248 0.13787 6.10419 14.37090 0.00001 Analysis1_Eigen.resultCase[66) 28.6462 0.2240 2.65173 2.10261 0.632429 0.0018 Analysis1_Eigen.resultCase[68) 28.6533	ResultCase AngFreq [rad/s] Period [s] EMM-X [tonne] EMM-X [tonne] EMM-X [tonne] EMM-X [tonne] DMM-X DMM-X Analysis1_Eigen.resultCase[1) 1.4381 4.3691 0.05202 12434.30000 0.38309 0.00000 0.82928 Analysis1_Eigen.resultCase[2) 1.7640 3.5620 12910.30000 0.12163 0.17539 0.86533 0.00001 Analysis1_Eigen.resultCase[3) 3.2890 1.9104 15.07190 22.77650 0.06264 0.00101 0.0152 Analysis1_Eigen.resultCase[4) 4.5271 1.3879 6.01972 2315.17000 7.79755 0.00040 0.15441 Analysis1_Eigen.resultCase[6] 4.6845 1.3413 1801.87000 4.22854 1.96354 0.12077 0.00028 Analysis1_Eigen.resultCase[65] 27.7277 0.2266 0.05787 2.27255 3.15986 0.000001 0.00015 Analysis1_Eigen.resultCase[66] 27.95745 0.2240 0.27617 2.10261 0.082429 0.00018 0.00014 Analysis1_Eigen.resultCase[68] 28	ResultCase AngFreq [rad/s] Period [s] EMM-X [tonne] EMM-X [tonne] EMM-Z [tonne] DMM-X DMM-X DMM-Y DMM-X Analysis1_Eigen.resultCase[1) 1.4381 4.3691 0.05202 12434.30000 0.38309 0.00000 0.82928 0.00003 Analysis1_Eigen.resultCase[2) 1.7640 3.5620 12910.30000 0.12163 0.17539 0.86533 0.00001 0.00001 Analysis1_Eigen.resultCase[3) 3.2890 1.9104 15.07190 22.77650 0.06264 0.00101 0.0152 0.00000 Analysis1_Eigen.resultCase[4) 4.5271 1.3879 6.01972 2315.17000 7.79755 0.00040 0.15441 0.00061 Analysis1_Eigen.resultCase[65) 4.6845 1.3413 1801.87000 4.22854 1.96354 0.12077 0.00028 0.00015 Analysis1_Eigen.resultCase[65) 27.7277 0.2266 0.05787 2.27255 3.15986 0.00001 0.00015 0.00025 Analysis1_Eigen.resultCase[66] 27.9545 0.2240 0.27173 2.1026	ResultCase AngFreq [rad/s] Period [s] EMM-X [tonne] EMM-Z [tonne] DMM-X DMM-Y DMM-X AMM-X Analysis1_Eigen.resultCase[1) 1.4381 4.3691 0.05202 12434.3000 0.38309 0.00000 0.82928 0.00001 0.86534 Analysis1_Eigen.resultCase[2) 1.7640 3.5620 12910.30000 0.12163 0.17539 0.86533 0.00001 0.086534 Analysis1_Eigen.resultCase[3) 3.2890 1.9104 15.07190 22.77650 0.06264 0.00101 0.00152 0.00000 0.86635 Analysis1_Eigen.resultCase[4) 4.5271 1.3879 6.01972 2315.17000 7.77755 0.00040 0.15441 0.00061 0.86635 Analysis1_Eigen.resultCase[5) 4.6845 1.3413 1801.87000 4.22854 1.96354 0.12077 0.00028 0.00015 0.98752 Analysis1_Eigen.resultCase[65) 27.7277 0.2266 0.05787 2.27255 3.15986 0.00001 0.00013 0.99870 Analysis1_Eigen.resultCase[66) 27.9545 <td>ResultCase AngFreq [rad/s] Period [s] EMM-X [tonne] EMM-Z [tonne] DMM-X DMM-Y DMM-X AMM-X AMM-X Analysis1_Eigen.resultCase[1) 1.4381 4.3691 0.05202 12434.30000 0.38309 0.00000 0.82928 0.00001 0.82928 Analysis1_Eigen.resultCase[2) 1.7640 3.5620 12910.30000 0.12163 0.17539 0.86533 0.00001 0.86534 0.82929 Analysis1_Eigen.resultCase[3) 3.2890 1.9104 15.07190 22.77650 0.06264 0.00101 0.00152 0.00001 0.86635 0.83081 Analysis1_Eigen.resultCase[4) 4.5271 1.3879 6.01972 2315.17000 7.77755 0.00040 0.15441 0.00061 0.86635 0.83081 Analysis1_Eigen.resultCase[65) 4.6845 1.3413 1801.87000 4.22854 1.96354 0.12077 0.00028 0.00015 0.98752 0.98689 Analysis1_Eigen.resultCase[65) 27.7277 0.2266 0.05787 2.27255 3.15986 0.00001 0.00</td>	ResultCase AngFreq [rad/s] Period [s] EMM-X [tonne] EMM-Z [tonne] DMM-X DMM-Y DMM-X AMM-X AMM-X Analysis1_Eigen.resultCase[1) 1.4381 4.3691 0.05202 12434.30000 0.38309 0.00000 0.82928 0.00001 0.82928 Analysis1_Eigen.resultCase[2) 1.7640 3.5620 12910.30000 0.12163 0.17539 0.86533 0.00001 0.86534 0.82929 Analysis1_Eigen.resultCase[3) 3.2890 1.9104 15.07190 22.77650 0.06264 0.00101 0.00152 0.00001 0.86635 0.83081 Analysis1_Eigen.resultCase[4) 4.5271 1.3879 6.01972 2315.17000 7.77755 0.00040 0.15441 0.00061 0.86635 0.83081 Analysis1_Eigen.resultCase[65) 4.6845 1.3413 1801.87000 4.22854 1.96354 0.12077 0.00028 0.00015 0.98752 0.98689 Analysis1_Eigen.resultCase[65) 27.7277 0.2266 0.05787 2.27255 3.15986 0.00001 0.00

In this example all 70 modes are used in the earthquake response spectrum analysis.

4.2 Static Analysis

A static analysis is performed to analyze the structure for static loads.

✓ Create a static analysis and name it as **Analysis2_Static**.



- ✓ Set up the below options for each step.
 - ✓ Edit Mesh Activity: Same as what options used in **Analysis1_Eigen**.
 - ✓ Edit Wave Load Run:

On **Buoyancy** tab, uncheck the option *Include weight of marine growth* to make Wajac calculate the buoyancy and the weight of marine growth separately in two wave load cases, click Apply.

terministic seastates Added mass and damping Rul Buoyancy forces with non-horizontal water-plane	es Buoya
Buoyancy forces with non-horizontal water-plane Q? • Assuming horizontal free surface Q? • Using actual free surface	
Q? • Assuming horizontal free surface Q? • Using actual free surface	
8? C Using actual free surface	
End Forces	
♀?	nal method)
♀ ○ Exclude end forces for all members (mar	ine method)
♀ ○ Include end forces for non-flooded mem	bers only
✓ Include Buoyancy due to steel area	
Include buoyancy of beams at mudline	

Choose Only and Weight as Buoyancy option on Deterministic seastate tab.



	Seastate	Period	Direction	Height	Phase	Wave mod.	Order	Current	Wind	Stretching	Step length [deg]	Num.steps	Buoyancy	Design load	Current b.fac.	Wave k.fac.	Water levels	Doppler Effect	1.LC nu
1	1					CalmSea				NoStretching	0 deg	1	Only	NoDesignLoads	1	1	0 m	Off	5
2	2					CalmSea				NoStretching	0 deg	1	Weight	NoDesignLoads	1	1	0 m	Off	6
3																			

✓ Run the static analysis to create the result file _R1.SIN, which is in the analysis folder.

4.3 Earthquake Response Spectrum Analysis

GeniE supports the earthquake response spectrum analysis with user input acceleration spectra, velocity spectra, or displacement spectra.

Before the earthquake analysis is created, damping ratios and earthquake spectra need to be input into the *Environment* folder. When the earthquake response spectrum analysis is created, damping ratios and spectra can be selected.

The earthquake response spectrum analysis can be performed to generate the base loads only, which can be used to linearize the foundation. It also can be performed to analyze the structure and generate an earthquake load case, which includes beam forces and moments. The earthquake load case need be combined with the static load cases. The combined load cases will be used in beam and joint code checks.

4.3.1 Input Earthquake Condition

✓ Input overall damping coefficients. Only one damping coefficient is used in this example.

NOTE: GeniE only supports the overall damping coefficients currently, i.e., one damping ratio is used for one direction.

Constant da	mping for Earthquake				
Name	Damping1	▼ 8?			
Coefficient	0.05				

✓ Input earthquake spectra. Only one spectrum is used for global X, Y, and Z directions in this example. Use the down arrow key↓ to add enough empty line, copy and paste (Ctrl + V) the spectrum shown in Section 2 into the table.

E Environment	
庄 🧰 Air	
Directions	
Earthquake Damping	
🛅 Earthquake Spectra	
E Location1	New Earthquake Spectrum
Condition 1	Color code all visible properties

Name Spectrum1						
Frequency [rad/s]	Acceleration [m/s^2]	-				
0.8976 rad/s	0.0736 m/s^2					
1.0472 rad/s	0.1002 m/s^2					
1.2566 rad/s	0.1444 m/s^2					
1.5708 rad/s	0.2256 m/s^2					
1.7952 rad/s	0.2578 m/s^2					
2.0944 rad/s	0.3007 m/s^2					
2.5133 rad/s	0.3609 m/s^2					
2.869 rad/s	0.412 m/s*2					
3.1416 rad/s	0.4511 m/s^2					
4.1888 rad/s	0.6015 m/s^2					
6.2832 rad/s	0.9022 m/s^2					
8.3776 rad/s	1.2029 m/s^2					
12.5664 rad/s	1.8044 m/s^2					
19.635 rad/s	2.8194 m/s^2					
25.1327 rad/s	2.8194 m/s^2					
31.4159 rad/s	2.8194 m/s^2					
41.8879 rad/s	2.3965 m/s^2					

4.3.2 Create and Run Earthquake Response Spectrum Analysis

Follow the below steps to create the earthquake response spectrum analysis and create load combinations.



✓ To create an earthquake analysis, uncheck the option Automatically import global loadcases first, and then select the option Earthquake.

reate carthqu	iake Activ	ity			?	\times
Name Anal	ysis3_EQK					
Lock conc	cepts after	meshing t global load	8 ? cases			
Available activ	vities					
Meshing Hull Girde Local Ana Wave Loz Linear Str Pile Soil A Tension/C Indep. Ta Load Res V Earthqual	r Load Adj Ilysis ad Activity ructural An nalysis Compressio ank Couplir ults ke	uster alysis m Analysis ng Analysis	Use Sestra10	83		
FEM anal	ysis units	83				
Length	m					
Force	kN					
	1.10					

- \checkmark The options for each analysis step are as below.
 - ✓ Edit Mesh Activity: Select the same options used in Analysis1_Eigen.
 - ✓ Edit Earthquake Analysis:
 - Input earthquake load: Select the defined damping ratio and the spectrum, and then input the scaling factors for X, Y, and Z directions.
 - Select number of mode shapes: Choose the Eigenvalue analysis performed previously and select the number of mode shapes included in the earthquake analysis. The modal combination method *CQC* is selected with the default directional combination method SRSS. Check type is *Force*.

ponse spe	ctrum analysis Earthq	uake Load Combi	inations	Earthquake Load Mode s	Earthquake Li hape selection	oad Combinations	
rthouake I	0ad Mode shape sele	iction		Eigenvalue Analysis	90 Analysis 1	_Eigen	• 87
orquaria a	I houe shape sele	cuon	1	Available ResultCases	Add ->	Selected ResultCases	
Slobal X Dir	ection			Name -		ResultCase	•
Damping	A Damping 1				<+ Kemove	Analysis1_Eigen.resultCase(11)	
	Company				Add All	Analysis 1_Eigen.resultCase(14)	
Spectrum	Spectrum1		-		Remove All	Analysis1_Eigen.resultCase(20)	
				-		Analysis 1_Eigen.resultCase(16)	
scaling	1					Analysis1_Eigen.resultCase(17)	
						Analysis 1_Eigen.resultCase(10)	
Global Y Dir	ection					Analysis1_Eigen.resultCase(30)	_
amping						Analysis1_Eigen.resultCase(23)	
Jamping	O Damping1		-			Analysis 1_Eigen.resultCase(27)	
Spectrum	Construct					Analysis 1_Eigen_resultCase(29)	
pecualit	o spectrum1		-			Analysis 1 Eigen.resultCase(26)	
Scaling	1					Analysis1_Eigen.resultCase(28)	
	•					Analysis1_Eigen.resultCase(24)	
	College -					Analysis1_Eigen.resultCase(25)	
sioual 2 Dil	ecuon					Analysis 1 Eigen resultCase(21)	
Damping	Damping1		-			Analysis1_Eigen.resultCase(2)	
						Analysis 1_Eigen.resultCase(4)	
pectrum	Spectrum1		-			Analysis 1_Eigen.resultCase(1)	
caling						Analysis 1 Eigen.resultCase(3)	-1
calling	0.5					1	· [
				Modal Combination Metho	COC		-
				Church Theory			_

Create load combinations: Two load combinations, LComb_C and LComb_T are created. Load combinations include all basic load cases and wave load cases in *Analysis2_Static*, and the load case from the current earthquake analysis. Use the down arrow key to insert the new combined load case.

Earthquake Run				?	
Automatic generation of in esponse spectrum analysis	nput files 🔗? Earthquake Load Combinations				
Create base load only)? 				
Name	Available LoadCases	Add ->	Selected LoadCases	~	< <u></u> ₩?
.Comb_T	Name	<- Remove Add All Remove All	LoadCase ELCGrav RefLCEq RefLCLoad RefLCBuoy Analysis2_Static.WLC(1, 1) Analysis2_Static.WLC(2, 1) Earthquake analysis load	Factor 1 1 1 1 1 1 1 2 Case	
	Method Tension	/	type and scaling factor	~	
	Institutional Factors 1			_	

NOTE: Result case *EQK* will be generated from the analysis. It is the combined modal response and contains the beam internal forces at two ends of FE beams. The value of *EQK* forces and moments are non-negative. Three combination criteria are supported in GeniE with the special option in **axial** force component: 1) *Maximum* (get the largest absolute value by adding the earthquake response to the static component using the sign of the static component), *Compression* (assume the earthquake response is compressive and add "-" earthquake response to the static component, and Tension (assume the earthquake response is tensile and add "+" earthquake response to the static component). Other components are calculated as the *Maximum* method, i.e., static component + or – EQK component to get the maximum [ABS] values using the sign of the static component.

NOTE: If the option *Create base load only* is checked, the earthquake analysis only generates base loads, and the structure is not analyzed for load combinations.

 \checkmark Run the analysis. The progress of the analysis is displaying as below.

DNV

```
EarthquakeRSA version: 1.1.15.0
Input args length is 5
args index 0 is [C:/Truerle____/GeniE_EQK\Analysis3_EQK/]
args index 1 is [earthquakeRSA.inp]
args index 2 is [Eigen_R1.SIN]
args index 3 is [EigenMass_M1.SIF]
args index 4 is [Static_R1.SIN]
 WorkspaceFolder: C:\Training_Project\GeniE_EQK\Analysis3_EQK
  InputFileName: earthquakeRSA.inp
 EigenvFileName: Eigen_R1.SIN
   MassfileName: EigenMass_M1.SIF
 StaticFileName: Static_R1.SIN
- Open Eigenvalue SIF Model start ...
- Open Eigenvalue SIF done
Open Eigenvalue SIF Model Execution Time: 3.306 s
- Open Static SIF Model start ...
- Open Static SIF done
Open Static SIF Model Execution Time: 0.486 s
--- Analysis: EQK combination and create new SIN file ---
Modal Combination start ...
[################### ] 100%
Modal Combination CQC done!
Modal Combination Execution Time: 39.448 s
Load Combination Execution Time: 0.063 s
Create NewResultFile start ...
Create NewResultFile done!
Create New SIF file Execution Time: 1.849 s
 --- Analysis: Base Load calculation ---
Mass matrix processing start
Mass matrix processing done!
Mass matrix processing Execution Time: 0.753 s
- Check Mode Shapes Orthogonality start
[#################### ] 100%
- Check Mode Shapes Orthogonality done
Calculate Base Loads start
77%
```

NOTE: GeniE created input file can be modified if necessary, and the analysis can be performed using the modified input file. The input file format follows the commands in the Framework JNL input file with minor modification or simplification. User may refer to GeniE reference document *Earthquake Response Spectrum Analysis and Code check* and Framework User's Manual for more description.

4.3.3 Check Analysis Results

After the analysis is completed, the analysis input file, the information file and the result listing file are created in the analysis folder.

Activity	Duration	Status	Generate Input
I - Analysis3_EQK - Analysis	150s	Success	
1.1 - Meshing (Always Rege	2s	Success	
1.1.1 - Delete loads	0s	Success	
1.1.2 - Generate loads	0s	Success	
1.1.3 - Delete mesh	0s	Success	
1.1.4 - Generate mesh	2s	Success	
Kr =R 1.2 - Earthquake Analysis,	148s	Success	Vac
R 1.3 - Load Results	0s	Success	Edit Earthquake Analysis earthquakeRSA.inp earthquakeRSA.mlg earthquakeRSA.lis



EarthquakeRSA.MLG file contains the analysis options, inputs, and the intermediate results. If warnings or errors are issued in the analysis, they are included in the file.

EarthquakeRSA.LIS file includes the modal mass report, the modal load factor report, and the modal effective inertia force report.

Check *EarthquakeRSA.LIS* file, we can find below result cases.

- ✓ The result case *EQK* is the *ResultCase* 7, which is included in the load combination *LComb_C* and *LComb_T*.
- ✓ Result cases for load combinations *LComb_C* and *LComb_T* are *ResultCase 8* and *ResultCase 9*. They will be used in beam and joint code checks.

ex	RESName
1	LCGrav
2	LCEq
3	LCLLoad
4	LCBuoy
5	WLC1
6	WLC2
7	EQK
8	LComb C
9	LComb_T
	1 2 3 4 5 6 7 8 9

Resultcase(TDRESREF) in New EQK Result File

Right click *Analysis3_EQK.resultCase(8)* and *Analysis3_EQK.resultCase(9)* to define their load case *Design Condition* as *Earthquake*.

ame	Description	FEM Loadcase	FEM LC Rule	
Analysis3_EQK.resultCase(1)	ResultCase			Design Condition
Kr Analysis3_EQK.resultCase(2)	ResultCase			
Analysis3_EQK.resultCase(3)	ResultCase			De la contra Contrante
Analysis3_EQK.resultCase(4)	ResultCase			Design Condition
Kr Analysis3_EQK.resultCase(5)	ResultCase			
Kr Analysis3_EQK.resultCase(6)	ResultCase			
Analysis3_EQK.resultCase(7)	ResultCase			
Analysis3_EQK.resultCase(8)	ResultCase	Set Current		
Analysis3_EQK.resultCase(9)	ResultCase	Set Current		
Analysis3_EQK.step(1)	Meshing (Always	Properties		
R Analysis3_EQK.step(2)	Earthquake Analy,			
{ Analysis3_EQK.step(3)	Load Results			

NOTE: The *Design Condition* of a load case or a result case will determine the code check parameters, such as increased allowable stress factors in API RP 2A or the hydrostatic pressure factors in ISO 19902. When the *Design Condition* is selected, the appropriate parameters will be used in code checks automatically.

5 Beam and Joint Code Checks

Two capacity managers are created. One is for beam code checks and the other one is for critical joint code checks.

5.1 Beam Code Checks

- ✓ Create the capacity manager Cc1_setCCJacket, the results from Analysis3_EQK will be transferred into the capacity model, and design code API WSD 2014 is selected for beam code checks.
- Capacity members are created for all beams included in set CCJacket. Beam design parameters, such as beam buckling lengths, buckling factors, and moment amplification factors defined in the concept model will be transferred to the capacity model automatically.

Nalysis Analysis3_EQK Value Check API WSD 2014 Value Check API WSD 2014 Value Check API WSD	I⊽ Subset: CCJadket _
nde Check API WSD 2014 V	
	Structure Criteria
xde Edition 🖉 🖓 ?	Split at incoming beam Solit at beam end
orr Add Rule No Addition 👻 😵	☑ Consider beam offset 👂

- Add a run into the capacity manager.
 - ✓ Only check *Member* to make the run only for beam code checks.

reate Cod	e Check Run		2
	Col catCCladet	 	ОК
apacity	CC1_SelCCJacket		

✓ LoadCase tab: select result case (8) and (9), one is for earthquake inertia axial force in compression and the other one is for the axial force in tension.

Create Code Check Run activ Cc1_setCCJadeet activ Cc1_setCladeet activ Cc1	? ○ Canc ○ DesignC		e Check Run Cc1_setCCJacket PLWSD 2014 Members Joints eneral Member	de Check Run Cc1_setCCJacket API WSD 2014 Members Joints General Member
acity Cc1_setCCJacket c Check: SetWesp 2016 wde: Members Joints Generate listing file adcases General Member Available LoadCases Included LoadCases Name DesignCondition Add -> Name DesignCondit	OK OK		Cc1_setCCJacket PTWSD 2014 Members Joints eneral Member	Cc1_setCCJacket API WSD 2014 Members Joints General Member
e Check:	Cano DesignC		API WSD 2014 Members Joints eneral Member	APT WSD 2014 Members Dioints General Member
ude: Members Joints Generate listing file adcases Generate listing file advalable LoadCases Included LoadCases Name DesignCondition Bit Analysis3_EQK:resultCase(1) Operating VersultCase(2) Coverating	DesignC		Members Joints eneral Member	Members Joints General Member
adcases General Member Available LoadCases Included LoadCases Name DesignCondition Add -> Namesis_EQK.resultCase(1) Operating < Very Analysis_EQK.resultCase(2) Operating <	DesignC		eneral Member	General Member
Available LoadCases Included LoadCases Name DesignCondition Bit Analysis3_EQK.resultCase(1) Operating Sin Analysis3_EQK.resultCase(2) Operating Sin Analysis3_EQK.resultCase(2) Operating Sin Analysis3_EQK.resultCase(3) Earth Compared to the Analys	DesignC		dCareer	
Name DesignCondition Add -> Name Design Mik Analysis3_EQK.resultCase(1) Operating <-Remove Mark Analysis3_EQK.resultCase(3) Earthcase(3) Mark Analysis3_EQK.resultCase(2) Operating <-Remove Mark Analysis3_EQK.resultCase(3) Earthcase(3)	DesignC		Jucases	oadCases
Analysis3_EQK.resultCase(1) Operating <- Remove KersultCase(2) Operating KersultCase(3) Earthc KersultCase(dtCaca(0) Earthquaka	Condition	Der	Design
kw Analysis3 EOK, resultCase(2) Operating	itCase(o) caruiquake	ing	3_EQK.resultCase(1) Op	s3_EQK.resultCase(1) Operat
	ultCase(9) Earthquake	ing	3_EQK.resultCase(2) Op	s3_EQK.resultCase(2) Operat
Analysis3_EQK.resultCase(3) Operating		ing	3_EQK.resultCase(3) Op	s3_EQK.resultCase(3) Operat
Analysis3_EQK.resultCase(4) Operating Add All		ing	3_EQK.resultCase(4) Op	s3_EQK.resultCase(4) Operat

✓ General tab: All default options are used. Option Cap-end force included is checked since the rational method is used to calculate the buoyancy in Analysis2_Static.

	le Check Run			?
Capacity	Cc1_setJac	ket		OK
Code Check:	API WSD 2014		*	Cancel
include:	Members Member	loints	Genera	ate listing file
API WSD :	2014			
Cap-end	forces included	?		
Use indiv	idual brace to can	end distance		
Azimuthal To	lerance Angle: Joi	nt Design (deg.) 5		
Joint Validi	ty Range			
(Not ch	ecked 8?			
C Use ge	ometric limits 🔗?			
C Use mo	dified geometry	83 Si		
Joint Minim	um Capacity	100000000000000000000000000000000000000		
Use 50	% effective streng	oth check 🔗?		
I Earthq	Jake strength check	sk so uppgo fostern [[1	
Prace upit	theck perceptage			
biace unit (neok percentage	05 7689		
Required C	hord Thickness	bord thickness		
	al steo value	0.001 m m	· · · · · · · · · · · · · · · · · · ·	
Increment		1		
Increment	and the second second			



✓ Member tab: The member design parameters, such as buckling lengths and buckling factors, are input into the concept model. *From Structure* option will transfer them into the capacity model. Input **0.85** for Moment amplification for all capacity members.

NOTE: Since the calculated earthquake inertia forces in beams are only at FE element ends, GeniE calculated moment amplification factors from any equation might be not appropriated. Therefore, a constant value, such as 0.85, can be input as *Moment amplification* and used in beam code checks. The values input in the capacity model do not affect the inputs in the concept model.

NOTE: GeniE will not check the maximum moment along the member length for earthquake load cases. The option *Max Bending Moment* and *Local Bending Moment* doesn't affect the member code check results.

cc1_set	CJacket		OK
Check: API WSD 2014		~	Cano
e: 🗹 Members 🗌] Joints Generate listing file		
cases General Memb	ber		
PIWSD 2014 ALSC 9	n		
Buckling length	From Structure		
Effective length factor	From Structure		-
Effective length factor		~	-
Moment amplification	0.85	~	1
about z-axis 🔽 y-z sym	metry		
Buckling length	From Structure		m 🔗 ?
Effective length factor	From Structure		
Moment amplification	0.85		
Axial compression and b	ending		
Bending moment option	Max Bending Moment 9?		
Stiffener spacing			
Member	From Structure	~	m 🔗 ?
Cone	From Structure	~	m 🔗 ?
Floriday	Frank Shulah wa		1

Perform beam code checks and list the code check results for Worse Case (CC).

Capacity Model	LoadCase	Position	Status	UfTot	Formula	SubCheck	GeomCheck
/ member(Bm253)	Analysis3_EQK.resultCase(8)	0.00	OK	0.48	uf6.13	API WSD 2014 member	Geom OK
/ member(Bm255)	Analysis3_EQK.resultCase(8)	0.00	OK	0.48	uf6.13	API WSD 2014 member	Geom OK
/ member(Bm287)	Analysis3_EQK.resultCase(8)	1.00	OK	0.48	uf6.13	API WSD 2014 member	Geom OK
/ member(Bm284)	Analysis3_EQK.resultCase(8)	0.00	OK	0.48	uf6.13	API WSD 2014 member	Geom OK
/ member(Bm271)	Analysis3_EQK.resultCase(8)	0.00	OK	0.48	uf6.13	API WSD 2014 member	Geom OK
/ member(Bm281)	Analysis3_EQK.resultCase(8)	1.00	OK	0.48	uf6.13	API WSD 2014 member	Geom OK
/ member(Bm283)	Analysis3_EQK.resultCase(8)	1.00	OK	0.48	uf6.13	API WSD 2014 member	Geom OK
/ member(Bm279)	Analysis3_EQK.resultCase(8)	0.00	OK	0.48	uf6.13	API WSD 2014 member	Geom OK

5.2 Joint Minimum Strength Code Checks

This capacity manager is created for critical joint code checks. 100% of brace axial strengths will be applied as brace member loads in joint code checks. Then the capacity model must include brace members and joints.

Create capacity manager Cc2_CritJT_withBraces. Analysis results from Analysis3_EQK and API WSD 2014 will be
used to perform joint code checks. Capacity members and capacity joints are created from set CritJT_withBraces.

		and a state		Create Joints ? X
Name	Cc2_CritJT_withBraces		Capacity Manager: Cc2_CritJT_withBraces	
Analysis	Analysis3_EQK •	89	Subset: CritJT_withBraces	Capacity Manager: Cc2_CritJT_withBraces
Code Check	API WSD 2014	87	Structure Criteria	Subset: CritJT_withBraces
Code Edition	v	87	☐ Split at incoming beam	
Corr Add Rule	No Addition 👻	82	Consider beam offset 9?	OK Cancel
			Vise "From Structure" Member Options	
	OK Ca	ancel	□ Update 💡	

- Add a run into the capacity manager.
 - ✓ Check options for *Members* and *Joints*.

Capacity	Cc2_CritJT_withBraces	OK
Code Check:	API WSD 2014	Cancel
Include:	Members 🔽 Joints	



✓ LoadCase tab: Include result case (8) and (9) in code checks.



✓ General tab: Choose the option Use geometric limits to include geometric validity range checks and select the option Earthquake strength check for critical joint minimum strength checks. 100% of the brace member axial strengths will be used as brace loads in joint code checks.

✓ Member tab: Use the default options.

de check.	
ndude: 🔽 Members 🔽 Joints	Generate listing file
Loadcases General Member Joint	
API WSD 2014	
Cap-end forces included 9?	
☑ Use individual brace to can end distance	e -
Azimuthal Tolerance Angle: Joint Design (d	leg.) 5
- Joint Validity Range	
C Not checked 8?	
Use geometric limits 9?	
C Use modified geometry 9?	
Joint Minimum Capacity	
🔲 Use 50% effective strength check	
Earthquake strength check	
Minimum cut-off value for brace usage fa	ctors 0
Brace unit check percentage 85 %	

city Cc2_Cri	tJT_withBraces	
Check: API WSD 201	4	Cance
de: 🔽 Members dcases General Mem	Joints Generate listing file	
API WSD 2014 AISC 9	th .	
about y-axis		
Buckling length	From Structure	<u>▼</u> m 🗞 5
Effective length factor	From Structure	-
Moment amplification	From Structure	-
about z-axis 🔽 y-z syr	nmetry	
Buckling length	From Structure	∽ m 🔗?
Effective length factor	From Structure	+
Moment amplification	From Structure	Ψ.
Axial compression and I	pending	
Bending moment option	Max Bending Moment	
Stiffener spacing		
Member	From Structure	▼ m 🔗?
Cone	From Structure	▼ m 8 ?
Flooding	From Structure	-
Conical Transitions	, prces and bending moments	

✓ Joint tab: Check the option Critical Joint

DNV

Capacity	Cc2_CritJT_withBrac	es				
Code Check:	PI WSD 2014					
include: 🔽	Members 🔽 Joints	Genera	ate listing file			
		_				
I and an and a	Joint Jatanta Joint	h				
Loadcases G	eneral Member Joint					
Loadcases G	eneral Member Joint)				
Loadcases G	eneral Member Joint					
Loadcases G API WSD 2 Brace	eneral Member Joint	Gap [m]	Through Brace	Critical Joint	Brace Utilization	Joint Grouting [m
Loadcases G API WSD 2 Brace All Brace	eneral Member Joint D14 Brace Type ES	Gap [m]	Through Brace	Critical Joint	Brace Utilization	Joint Grouting [m

NOTE: The joint minimum strength check is only performed when the option *Earthquake strength check* on General tab and the option *Critical Joint* on Joint tab are both selected in the capacity run.

Perform joint code checks, and critical joint minimum strength check results are listed as below.

Capacity Model	LoadCase	Position	Status	LIFTot	Formula	SubCheck	GeomCheck	Required Thickness
K		rostoon	- d d O	01100	C al l	Subericek	ocomencer	required microicos
joint(Jt6)	Analysis3_EQK.resultCase(8)	Bm57	Failed(uf)	2.38	utearthquake	API WSD 2014 joint	Geom OK	Not checked
joint(Jt1)	Analysis3_EQK.resultCase(8)	Bm55	Failed(uf)	2.38	ufearthquake	API WSD 2014 joint	Geom OK	Not checked
joint(Jt3)	Analysis3_EQK.resultCase(8)	Bm66	Failed(uf)	1.90	ufearthquake	API WSD 2014 joint	Geom OK	Not checked
joint(Jt2)	Analysis3_EQK.resultCase(8)	Bm72	Failed(uf)	1.90	ufearthquake	API WSD 2014 joint	Geom OK	Not checked
(joint(JT191)	Analysis3_EQK.resultCase(8)	Bm207	Failed(uf)	1.54	ufearthquake	API WSD 2014 joint	Geom OK	Not checked
joint(JT 197)	Analysis3_EQK.resultCase(8)	Bm200	Failed(uf)	1.54	ufearthquake	API WSD 2014 joint	Geom OK	Not checked
joint(JT 199)	Analysis3_EQK.resultCase(8)	Bm206	Failed(uf)	1.53	ufearthquake	API WSD 2014 joint	Geom OK	Not checked
joint(JT 196)	Analysis3_EQK.resultCase(8)	Bm202	Failed(uf)	1.53	ufearthquake	API WSD 2014 joint	Geom OK	Not checked
joint(JT 190)	Analysis3_EQK.resultCase(8)	Bm 186	Failed(geo)	1.39	ufearthquake	API WSD 2014 joint	theta	Not checked
joint(JT177)	Analysis3_EQK.resultCase(8)	Bm 184	Failed(geo)	1.38	ufearthquake	API WSD 2014 joint	theta	Not checked
K joint(Jt118)	Analysis3_EQK.resultCase(8)	Bm 130	Failed(geo)	1.37	ufearthquake	API WSD 2014 joint	theta	Not checked
joint(Jt113)	Analysis3_EQK.resultCase(8)	Bm 129	Failed(geo)	1.37	ufearthquake	API WSD 2014 joint	theta	Not checked
🖌 joint(Jt100)	Analysis3_EQK.resultCase(8)	Bm94	Failed(uf)	1.35	ufearthquake	API WSD 2014 joint	Geom OK	Not checked
joint(Jt96)	Analysis3_EQK.resultCase(8)	Bm93	Failed(uf)	1.35	ufearthquake	API WSD 2014 joint	Geom OK	Not checked
joint(JT176)	Analysis3_EQK.resultCase(8)	Bm 183	Failed(geo)	1.29	ufearthquake	API WSD 2014 joint	theta	Not checked
joint(JT175)	Analysis3_EQK.resultCase(8)	Bm 191	Failed(geo)	1.28	ufearthquake	API WSD 2014 joint	theta	Not checked
joint(Jt112)	Analysis3_EQK.resultCase(8)	Bm128	Failed(uf)	1.23	ufearthquake	API WSD 2014 joint	Geom OK	Not checked
joint(Jt111)	Analysis3_EQK.resultCase(8)	Bm 155	Failed(uf)	1.23	ufearthquake	API WSD 2014 joint	Geom OK	Not checked
joint(Jt94)	Analysis3_EQK.resultCase(8)	Bm 108	Failed(uf)	1.15	ufearthquake	API WSD 2014 joint	Geom OK	Not checked
K joint(Jt95)	Analysis3_EQK.resultCase(8)	Bm92	Failed(uf)	1.15	ufearthquake	API WSD 2014 joint	Geom OK	Not checked
/ member(Bm253)	Analysis3_EQK.resultCase(8)	0.00	OK	0.48	uf6.13	API WSD 2014 member	Geom OK	

Save the model as *Model_ResponseSpectrumAnalysis_Done.gnx*



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.