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

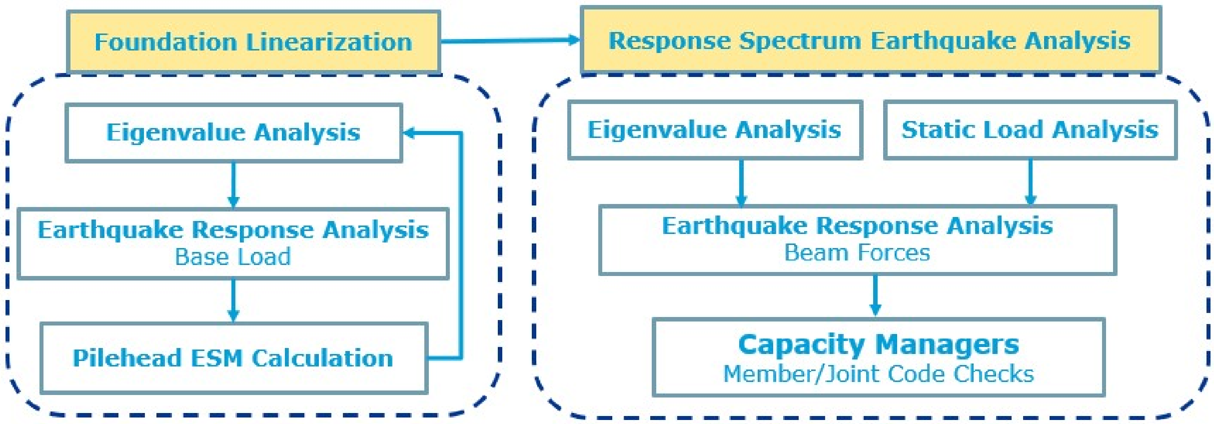
E-mail support: software.support@dnv.com

E-mail sales: digital@dnv.com

Table of contents

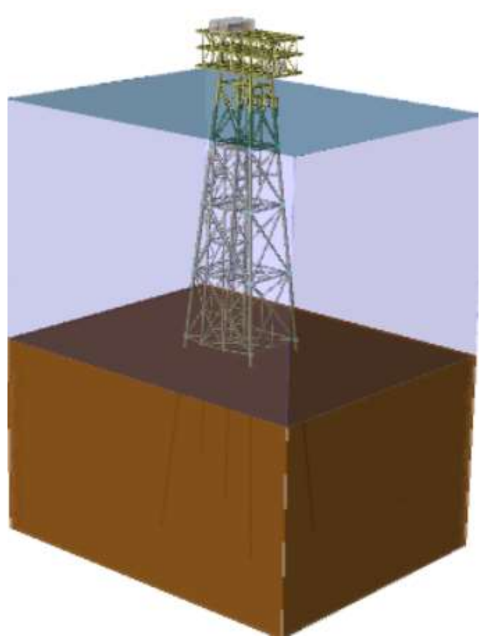
1 Introduction

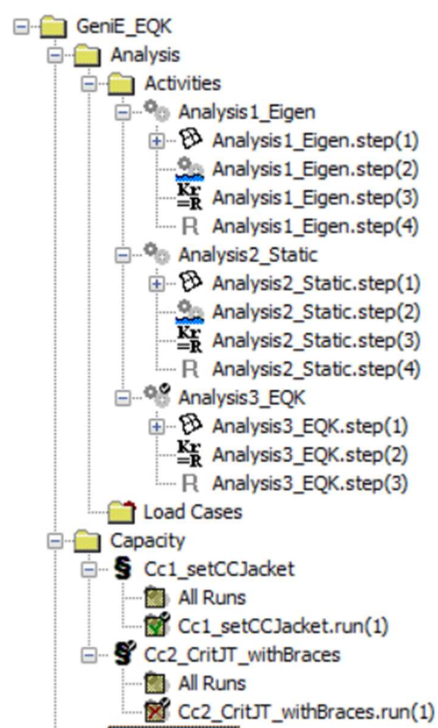
In an earthquake response spectrum analysis, the foundation linearization needs to be performed first. After the equivalent springs are applied at pile heads, the response spectrum earthquake analysis can be performed. The analysis workflow is like below.



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.



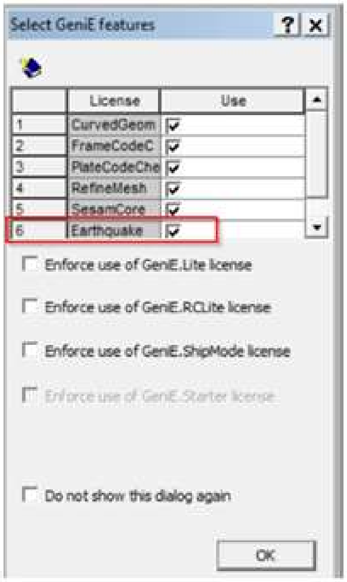


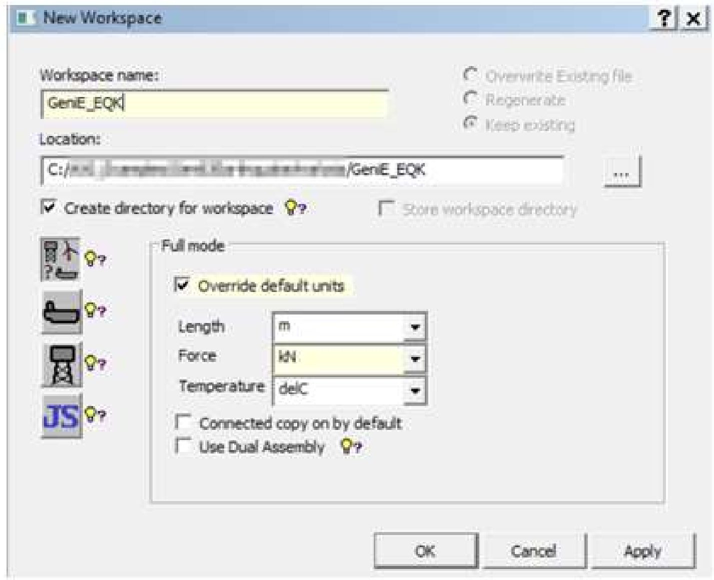
2 Open New GeniE Workspace and Import Model

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

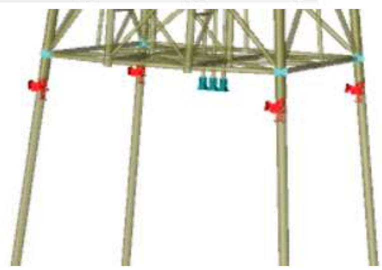
To create all analyses included in GeniE from a new GeniE workspace, do the following:

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



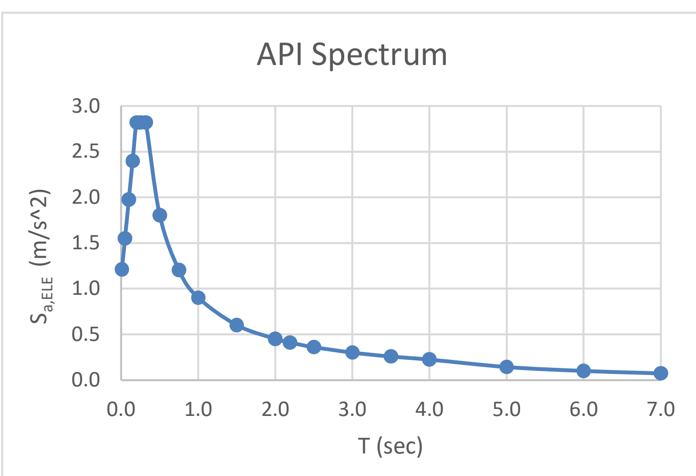


- ✓ Import the model ***Model_ResponseSpectrumAnalysis_Start.gnx***. The model contains equivalent springs at all pile heads and piles below the mudline. In all analyses only equivalent springs are used as the support condition.



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

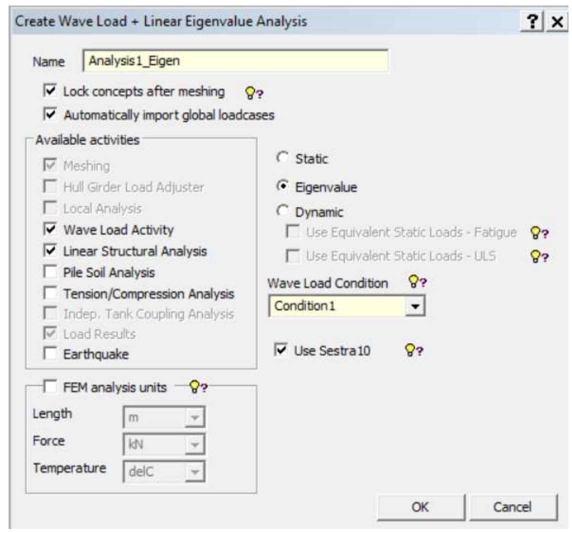
The above acceleration spectrum and the damping ratio of 0.05 will be applied in the global X, Y, and Z directions.

4 Earthquake Response Spectrum Analysis

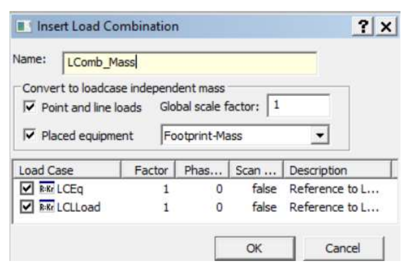
The earthquake response spectrum analysis includes three analyses, the Eigenvalue analysis, the static load analysis, and the earthquake response spectrum analysis. All three analyses are set up as below.

4.1 Eigenvalue Analysis

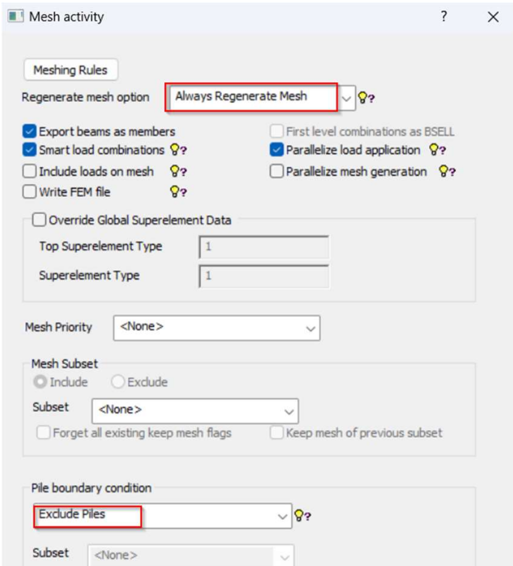
- ✓ Create a new Eigenvalue analysis and name it as **Analysis1_Eigen**. Wave Load Activity and Linear Structural Analysis are included.

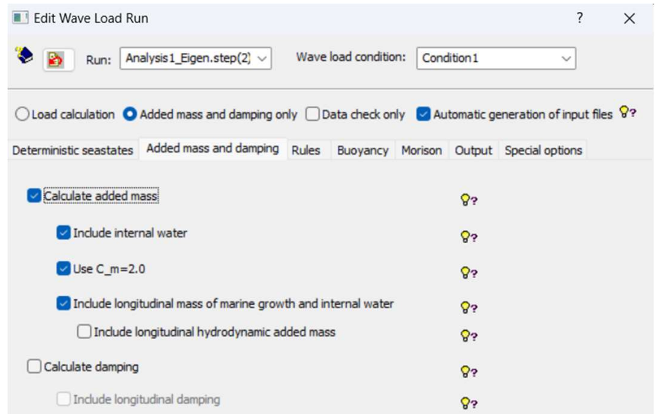


- ✓ Only masses are required in the Eigenvalue analysis. Then Load case LCBuoy can be excluded from the analysis.
- ✓ The structural mass is included automatically. Two load cases, LCEq, and LCLLoad need to be converted to mass and then included in the analysis. A load combination **LComb_Mass** is created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



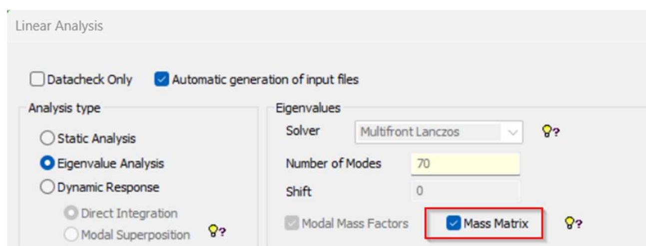
- ✓ Set up the below options for each step.
  - ✓ Edit Mesh Activity: Choose ***Always Regenerate Mesh*** and ***Exclude Piles*** as Pile boundary condition.
  - ✓ Edit Wave Load Run: It is not necessary to calculate wave loads. Choose mass calculation options on ***Added mass and damping only*** tab.



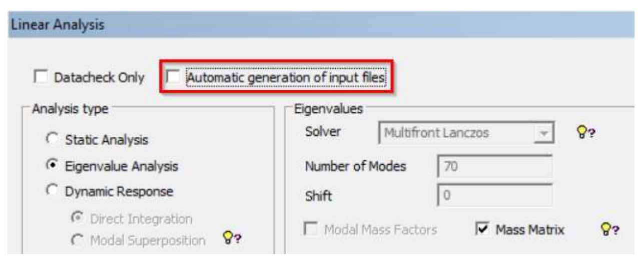


✓ Edit Linear Sestra Analysis: The number of modes is 70. Two runs need to be performed. The options and the purpose of each run are as below.

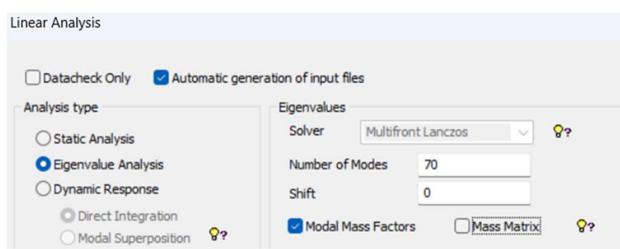
1) Check the option ***Mass Matrix*** and click ***OK*** to close the dialog box. Run the analysis to generate the mass matrix file **M1.SIF**.



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.



2) 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**



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.

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.

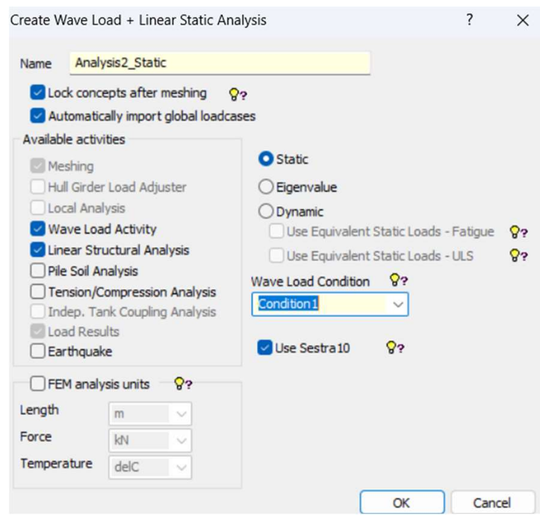
EigenMode	ResultCase	AngFreq [rad/s]	Period [s]	EMM-X [tonne]	EMM-Y [tonne]	EMM-Z [tonne]	DMM-X	DMM-Y	DMM-Z	AMM-X	AMM-Y	AMM-Z
1	Analysis1_Eigen.resultCase(1)	1.4381	4.3691	0.05202	12434.30000	0.38309	0.00000	0.82928	0.00003	0.00000	0.82928	0.00003
2	Analysis1_Eigen.resultCase(2)	1.7640	3.5620	12910.30000	0.12163	0.17539	0.86533	0.00001	0.00001	0.86534	0.82929	0.00004
3	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
4	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
5	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
...												
...												
65	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
66	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
67	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
68	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
69	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
70	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

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.

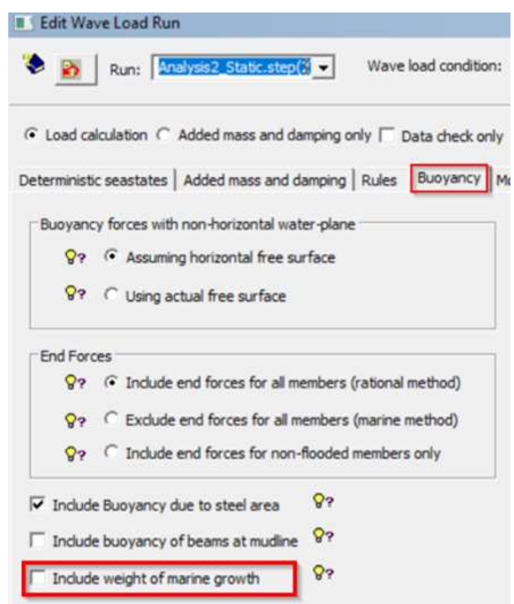
✓ Create a static analysis and name it as **Analysis2_Static**.



✓ 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.



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.

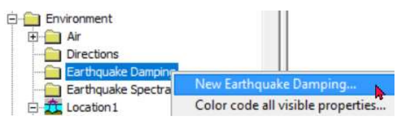
Before the earthquake analysis is created, damping ratios and earthquake spectra need to be input into the ***Environment*** folder. When the earthquake response spectrum analysis is created, damping ratios and spectra can be selected.

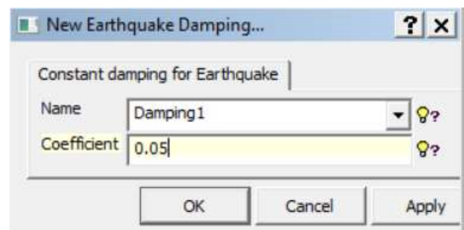
The earthquake response spectrum analysis can be performed to generate the base loads only, which can be used to linearize the foundation. It also can be performed to analyze the structure and generate an earthquake load case, which includes beam forces and moments. The earthquake load case need be combined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

4.3.1 Input Earthquake Condition

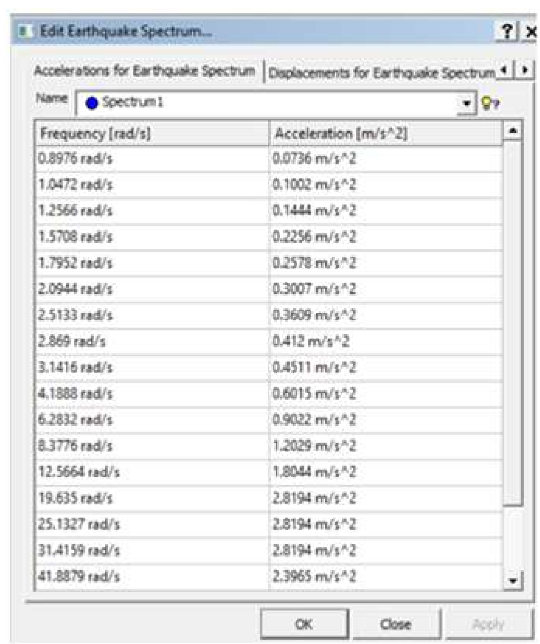
✓ Input overall damping coefficients. Only one damping coefficient is used in this example.

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





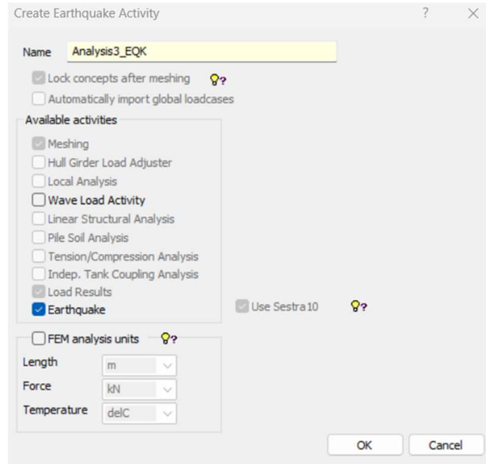
✓ Input earthquake spectra. Only one spectrum is used for global X, Y, and Z directions in this example. Use the down arrow key⬇ to add enough empty line, copy and paste (Ctrl + V) the spectrum shown in Section 2 into the table.



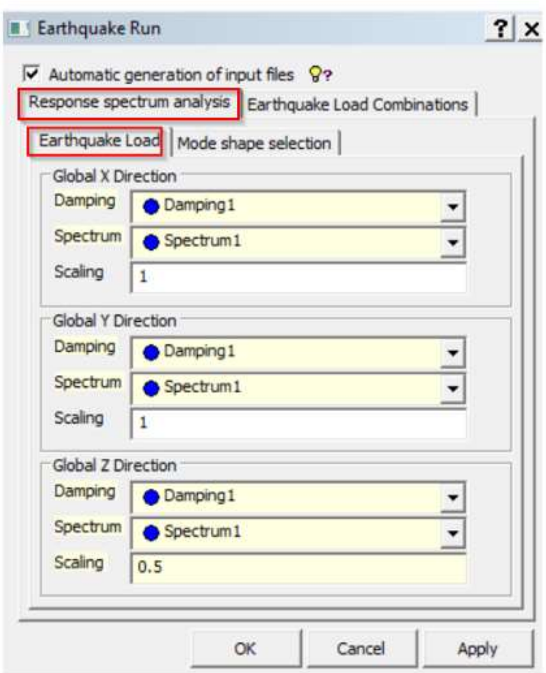
4.3.2 Create and Run Earthquake Response Spectrum Analysis

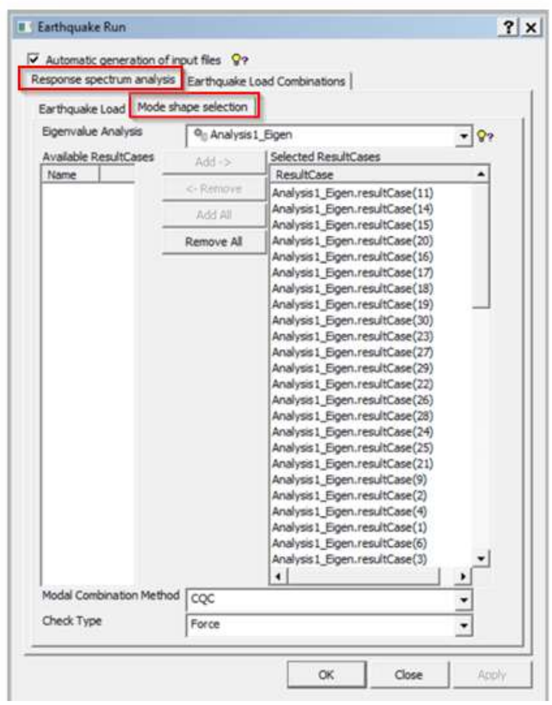
Follow the below steps to create the earthquake response spectrum analysis and create load combinations.

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

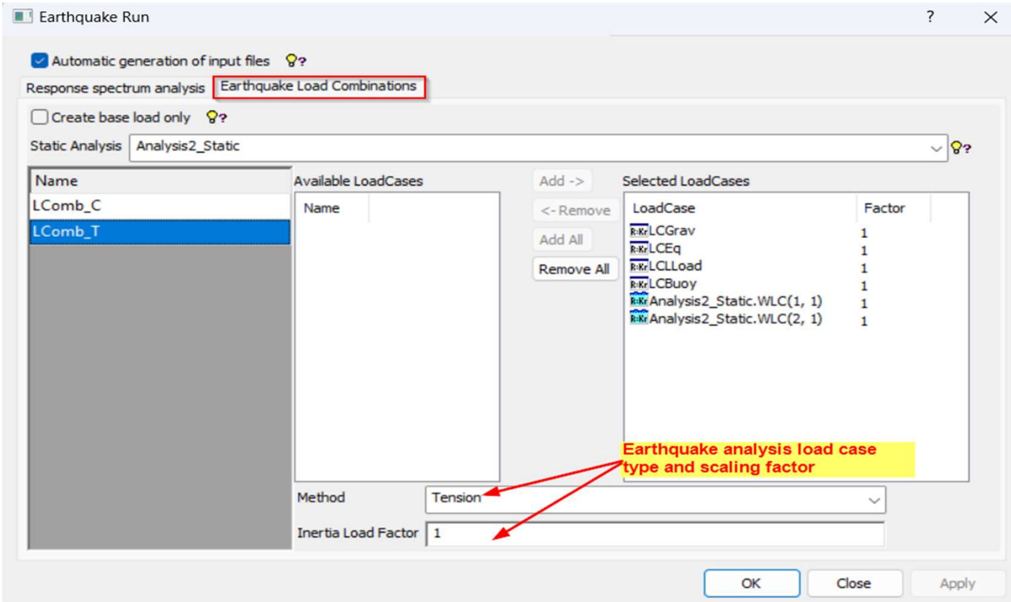


- ✓ The options for each analysis step are as below.
  - ✓ Edit Mesh Activity: Select the same options used in Analysis1_Eigen.
  - ✓ Edit Earthquake Analysis:
    - o Input earthquake load: Select the defined damping ratio and the spectrum, and then input the scaling factors for X, Y, and Z directions.
    - o 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***.





    - o 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.



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.

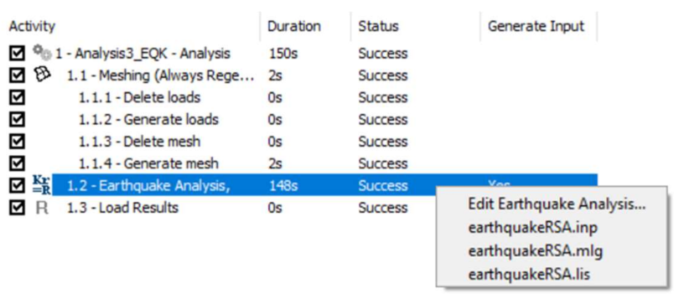
✓ Run the analysis. The progress of the analysis is displaying as below.

```
EarthquakeRSA version: 1.1.15.0
Input args length is 5
args index 0 is [C:/[redacted]/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.



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.

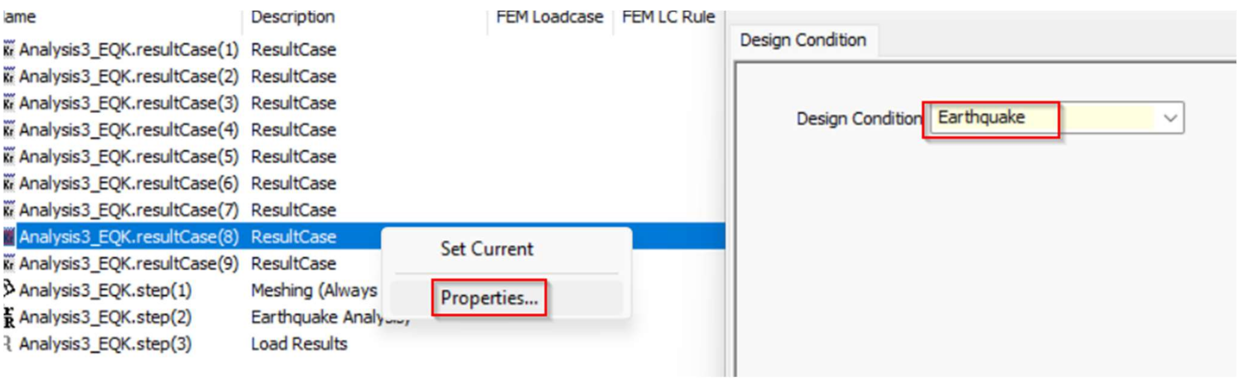
EarthquakeRSA.LIS file includes the modal mass report, the modal load factor report, and the modal effective inertia force report.

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.

```
Resultcase(TDRESREF) in New EQK Result File

        Index RESName
 ResultCase 1  LCGrav
 ResultCase 2  LCEq
 ResultCase 3  LCLLoad
 ResultCase 4  LCBuoy
 ResultCase 5  WLC1
 ResultCase 6  WLC2
 ResultCase 7  EQK
 ResultCase 8  LComb_C
 ResultCase 9  LComb_T
```

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



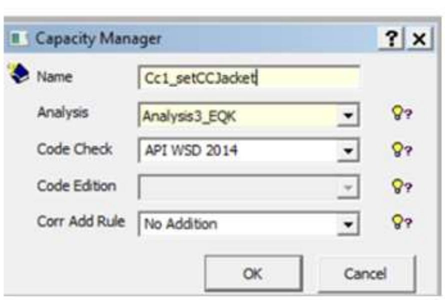
NOTE: The *Design Condition* of a load case or a result case will determine the code check parameters, such as increased allowable stress factors in API RP 2A or the hydrostatic pressure factors in ISO 19902. When the *Design Condition* is selected, the appropriate parameters will be used in code checks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

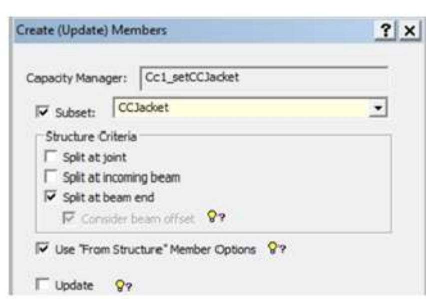
5 Beam and Joint Code Checks

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

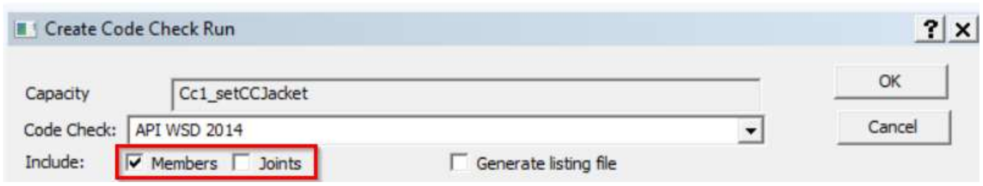
5.1 Beam Code Checks

- ✓ 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.

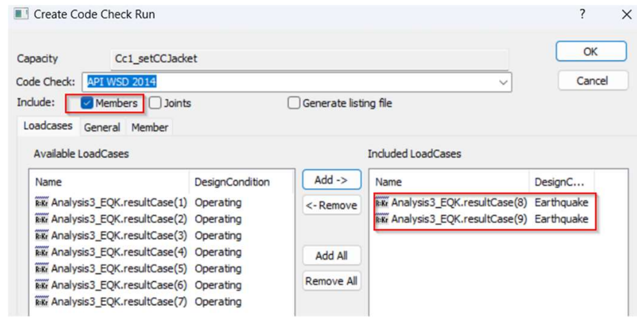




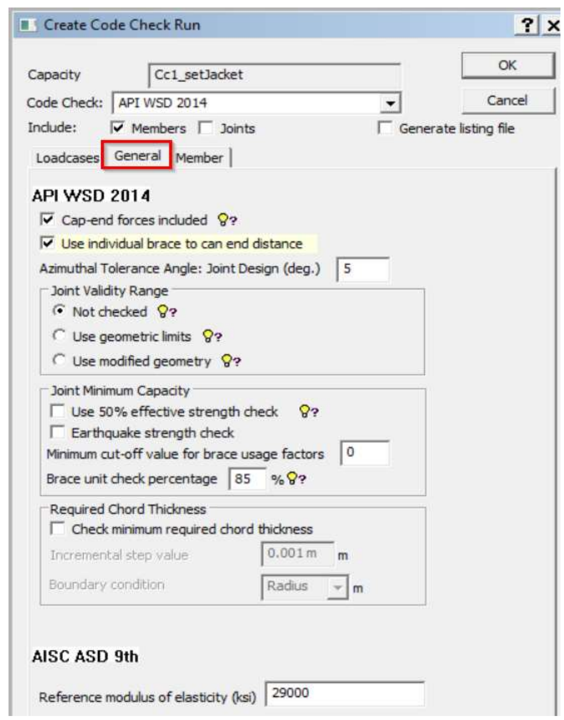
- Add a run into the capacity manager.
  - ✓ Only check *Member* to make the run only for beam code checks.

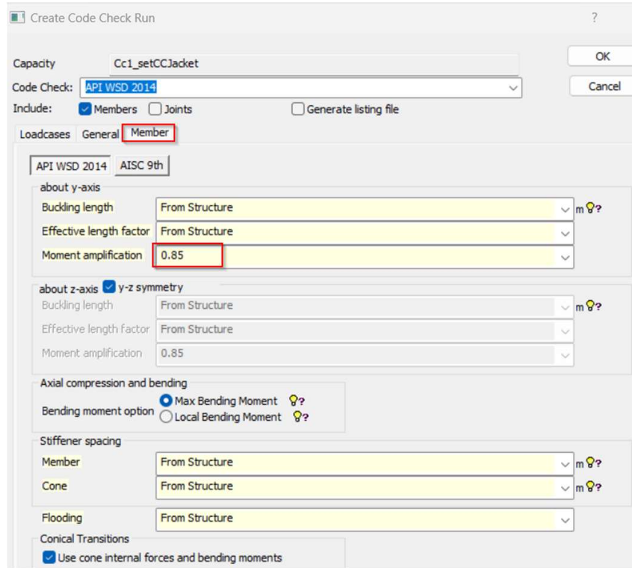


- ✓ LoadCase tab: select result case (8) and (9), one is for earthquake inertia axial force in compression and the other one is for the axial force in tension.



- ✓ 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***.





- ✓ 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.

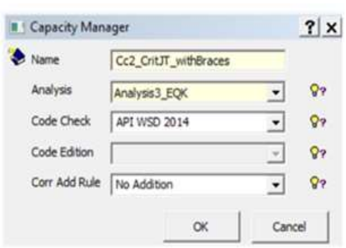
Perform beam code checks and list the code check results for Worse Case (CC).

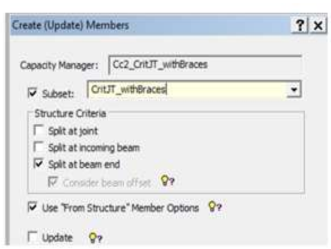
Capacity Model	LoadCase	Position	Status	UfTot	Formula	SubCheck	GeomCheck
member(Bm253)	Analysis3_EQK.resultCase(8)	0.00	OK	0.48	uf6.13	API WSD 2014 member	Geom OK
member(Bm255)	Analysis3_EQK.resultCase(8)	0.00	OK	0.48	uf6.13	API WSD 2014 member	Geom OK
member(Bm287)	Analysis3_EQK.resultCase(8)	1.00	OK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

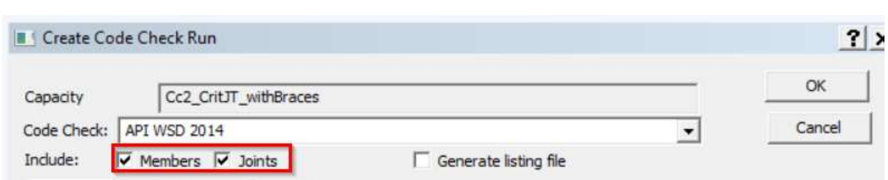
This capacity manager is created for critical joint code checks. 100% of brace axial strengths will be applied as brace member loads in joint code checks. Then the capacity model must include brace members and joints.

- 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.

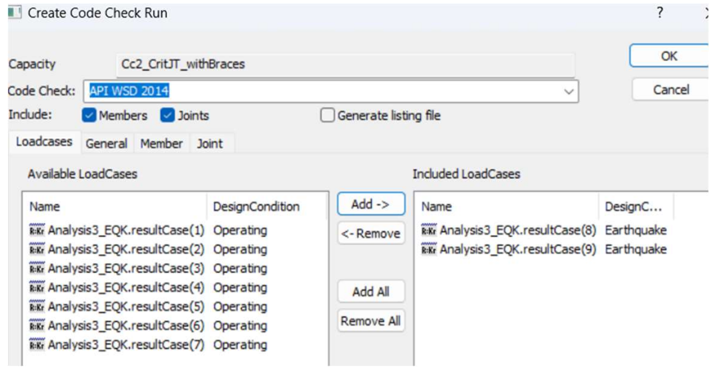




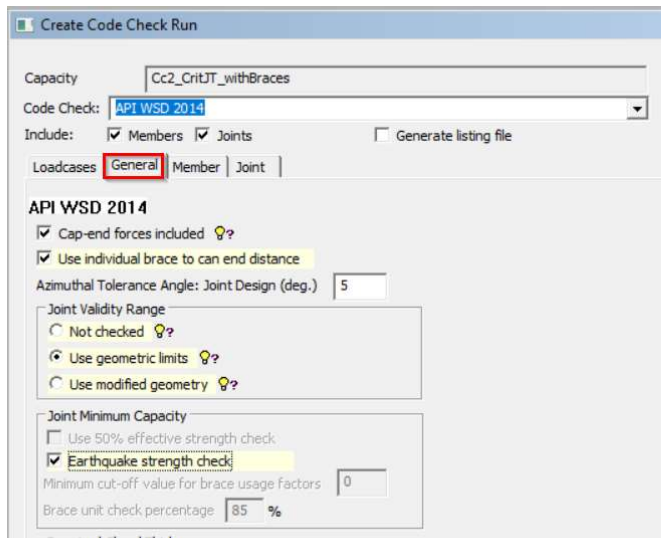
- Add a run into the capacity manager.
  - ✓ Check options for *Members* and *Joints*.



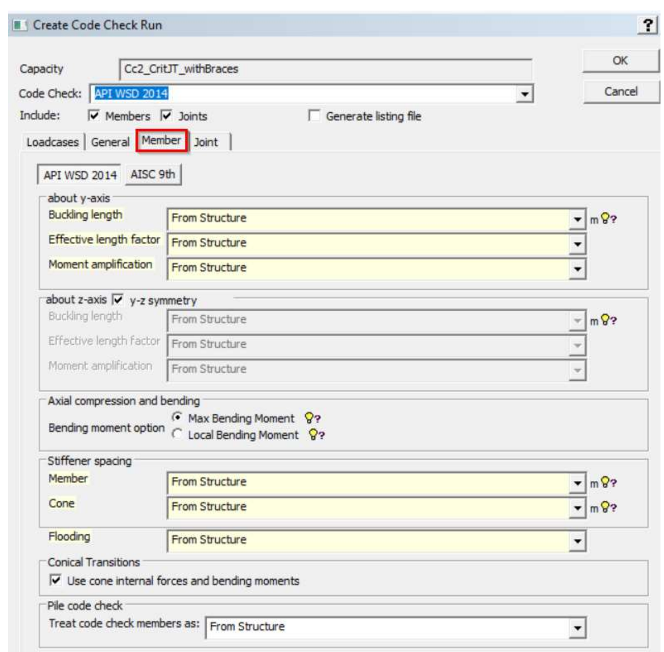
- ✓ LoadCase tab: Include result case (8) and (9) in code checks.



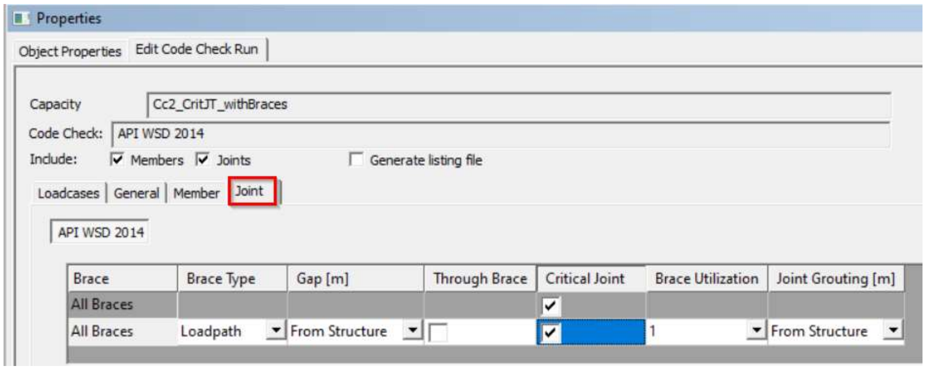
- ✓ General tab: Choose the option ***Use geometric limits*** to include geometric validity range checks and select the option ***Earthquake strength check*** for critical joint minimum strength checks. 100% of the brace member axial strengths will be used as brace loads in joint code checks.



- ✓ Member tab: Use the default options.



- ✓ Joint tab: Check the option *Critical Joint*



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	UfTot	Formula	SubCheck	GeomCheck	Required Thickness
joint(Jt6)	Analysis3_EQK.resultCase(8)	Bm57	Failed(uf)	2.38	ufearthquake	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(JT197)	Analysis3_EQK.resultCase(8)	Bm200	Failed(uf)	1.54	ufearthquake	API WSD 2014 joint	Geom OK	Not checked
joint(JT199)	Analysis3_EQK.resultCase(8)	Bm206	Failed(uf)	1.53	ufearthquake	API WSD 2014 joint	Geom OK	Not checked
joint(JT196)	Analysis3_EQK.resultCase(8)	Bm202	Failed(uf)	1.53	ufearthquake	API WSD 2014 joint	Geom OK	Not checked
joint(JT190)	Analysis3_EQK.resultCase(8)	Bm186	Failed(geo)	1.39	ufearthquake	API WSD 2014 joint	theta	Not checked
joint(JT177)	Analysis3_EQK.resultCase(8)	Bm184	Failed(geo)	1.38	ufearthquake	API WSD 2014 joint	theta	Not checked
joint(Jt118)	Analysis3_EQK.resultCase(8)	Bm130	Failed(geo)	1.37	ufearthquake	API WSD 2014 joint	theta	Not checked
joint(Jt113)	Analysis3_EQK.resultCase(8)	Bm129	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)	Bm183	Failed(geo)	1.29	ufearthquake	API WSD 2014 joint	theta	Not checked
joint(JT175)	Analysis3_EQK.resultCase(8)	Bm191	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)	Bm155	Failed(uf)	1.23	ufearthquake	API WSD 2014 joint	Geom OK	Not checked
joint(Jt94)	Analysis3_EQK.resultCase(8)	Bm108	Failed(uf)	1.15	ufearthquake	API WSD 2014 joint	Geom OK	Not checked
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.

DNV | digital@dnv.com | dnv.com/digital