WHEN TRUST MATTERS

SESAM EXAMPLE
Spectral Fatigue Analysis

Using Equivalent Static Load Analysis

Date: June 2025

Prepared by DNV – Digital Solutions

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

E-mail sales: software@dnv.com

Table of contents

1 INTRODUCTION

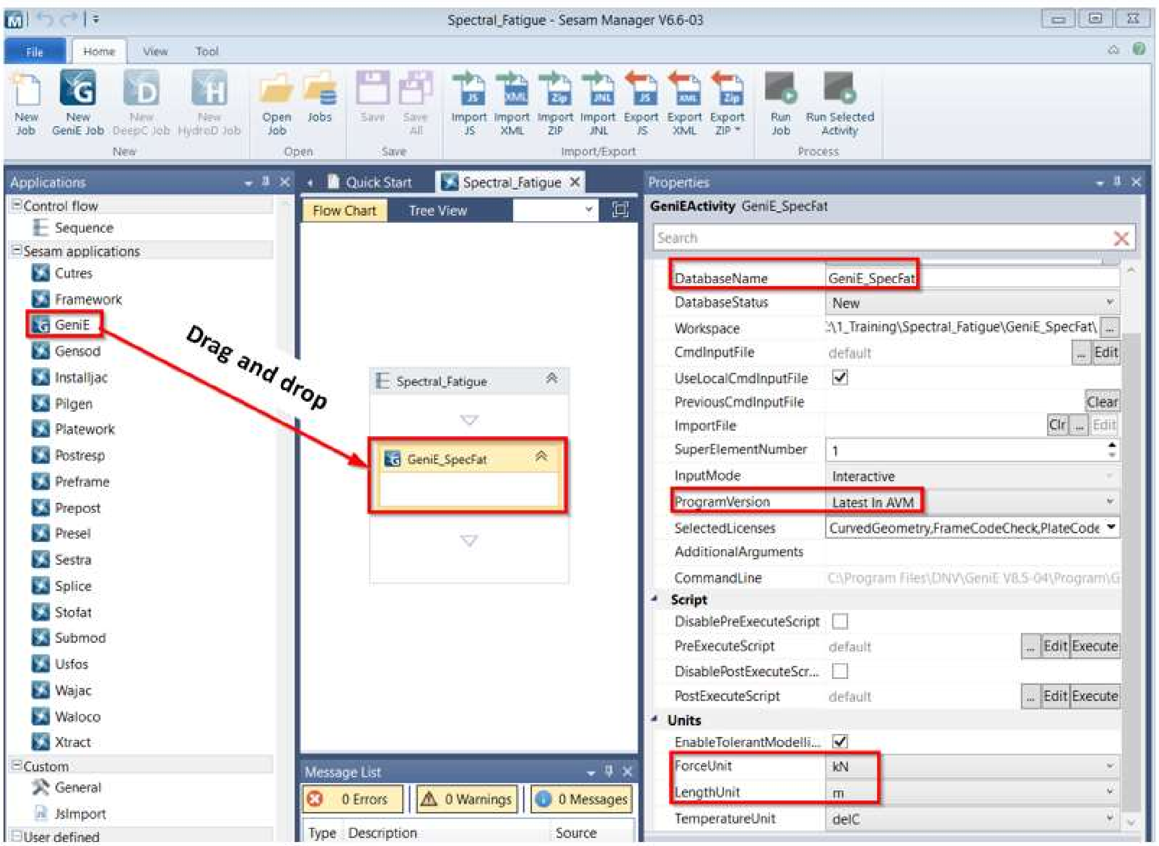
This example goes through steps to perform spectral fatigue analysis for an offshore fixed platform using Sesam software suite. The model file uses in this analysis is the model with linearized pilehead springs and the pile foundation.

2 MODEL MODIFICATION

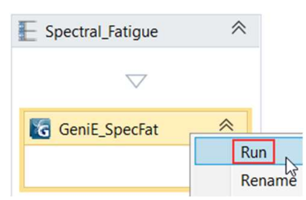
The model used to perform the Eigenvalue analysis will be modified to include a set of waves to perform a dynamic analysis for transfer function generations.

2.1 Importing Model from Eigenvalue Analysis

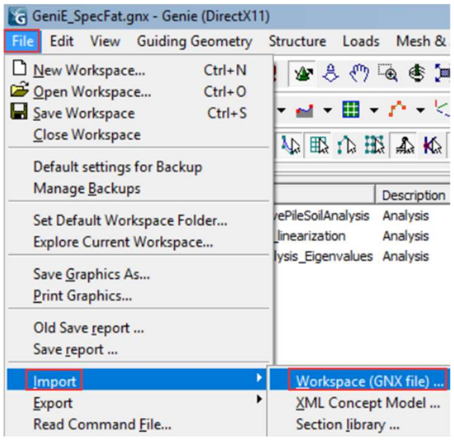
Open Sesam Manager, name it as **Spectral_Fatigue**. Then drag GeniE into the work area, and name is as **GeniE_SpecFat**. Change GeniE units to kN, m, and delC;

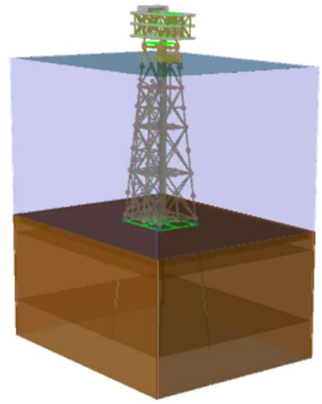


To launch GeniE, just RMB on **GeniE_ SpecFat** and click **Run**. A new GeniE workspace is now open.



To import the model file **Model_SpectralFatigue_Start.gnx** into GeniE, go to **File** > **Import** > **Workspace (GNX file)**. The structure will be displayed on the screen as below.





Note that the model file **Model_SpectralFatigue_Start.gnx** is the same model file as **Model_Eigenvalue_Analysis_Done.gnx**.

2.2 Creating Transfer Function Waves

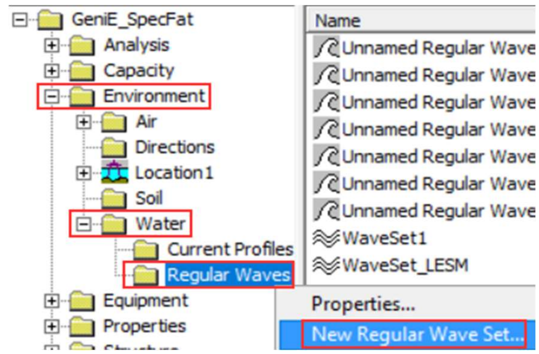
For the structure used in this example, the waves in three directions, 0 degree, 52 degrees, and 90 degrees, are as below and will be input into the model.

Seastate	Period	Height	Phase	Direction	Period	Height	Phase	Direction	Period	Height	Phase	Direction
1	1.800	0.2528	0.0	0.00	1.800	0.2528	0.0	52.00	1.800	0.2528	0.0	90.00
2	2.000	0.3122	0.0	0.00	2.000	0.3122	0.0	52.00	2.000	0.3122	0.0	90.00
3	2.250	0.3951	0.0	0.00	2.250	0.3951	0.0	52.00	2.250	0.3951	0.0	90.00
4	2.500	0.4877	0.0	0.00	2.500	0.4877	0.0	52.00	2.500	0.4877	0.0	90.00
5	2.600	0.5275	0.0	0.00	2.600	0.5275	0.0	52.00	2.600	0.5275	0.0	90.00
6	2.700	0.5689	0.0	0.00	2.700	0.5689	0.0	52.00	2.700	0.5689	0.0	90.00
7	2.850	0.6339	0.0	0.00	2.850	0.6339	0.0	52.00	2.850	0.6339	0.0	90.00
8	3.000	0.7023	0.0	0.00	3.000	0.7023	0.0	52.00	3.000	0.7023	0.0	90.00
9	3.070	0.7355	0.0	0.00	3.070	0.7355	0.0	52.00	3.070	0.7355	0.0	90.00
10	3.150	0.7743	0.0	0.00	3.150	0.7743	0.0	52.00	3.150	0.7743	0.0	90.00
11	3.220	0.8091	0.0	0.00	3.220	0.8091	0.0	52.00	3.220	0.8091	0.0	90.00
12	3.300	0.8498	0.0	0.00	3.300	0.8498	0.0	52.00	3.300	0.8498	0.0	90.00
13	3.500	0.9560	0.0	0.00	3.500	0.9560	0.0	52.00	3.500	0.9560	0.0	90.00
14	3.600	1.0114	0.0	0.00	3.600	1.0114	0.0	52.00	3.600	1.0114	0.0	90.00
15	3.700	1.0684	0.0	0.00	3.700	1.0684	0.0	52.00	3.700	1.0684	0.0	90.00
16	3.800	1.1269	0.0	0.00	3.800	1.1269	0.0	52.00	3.800	1.1269	0.0	90.00
17	3.920	1.1992	0.0	0.00	3.920	1.1992	0.0	52.00	3.920	1.1992	0.0	90.00
18	4.003	1.2505	0.0	0.00	4.003	1.2505	0.0	52.00	4.003	1.2505	0.0	90.00
19	4.080	1.2991	0.0	0.00	4.080	1.2991	0.0	52.00	4.080	1.2991	0.0	90.00
20	4.150	1.3440	0.0	0.00	4.150	1.3440	0.0	52.00	4.150	1.3440	0.0	90.00
21	4.200	1.3766	0.0	0.00	4.200	1.3766	0.0	52.00	4.200	1.3766	0.0	90.00
22	4.400	1.5108	0.0	0.00	4.400	1.5108	0.0	52.00	4.400	1.5108	0.0	90.00
23	4.500	1.5803	0.0	0.00	4.500	1.5803	0.0	52.00	4.500	1.5803	0.0	90.00
24	5.750	2.5802	0.0	0.00	5.750	2.5802	0.0	52.00	5.750	2.5802	0.0	90.00
25	6.440	3.2366	0.0	0.00	6.440	3.2366	0.0	52.00	6.440	3.2366	0.0	90.00
26	7.000	3.8239	0.0	0.00	7.000	3.8239	0.0	52.00	7.000	3.8239	0.0	90.00
27	7.750	4.6872	0.0	0.00	7.750	4.6872	0.0	52.00	7.750	4.6872	0.0	90.00
28	7.927	4.9038	0.0	0.00	7.927	4.9038	0.0	52.00	7.927	4.9038	0.0	90.00
29	8.006	5.0020	0.0	0.00	8.006	5.0020	0.0	52.00	8.006	5.0020	0.0	90.00
30	9.000	6.3211	0.0	0.00	9.000	6.3211	0.0	52.00	9.000	6.3211	0.0	90.00
31	10.500	8.6038	0.0	0.00	10.500	8.6038	0.0	52.00	10.500	8.6038	0.0	90.00
32	12.000	11.2376	0.0	0.00	12.000	11.2376	0.0	52.00	12.000	11.2376	0.0	90.00

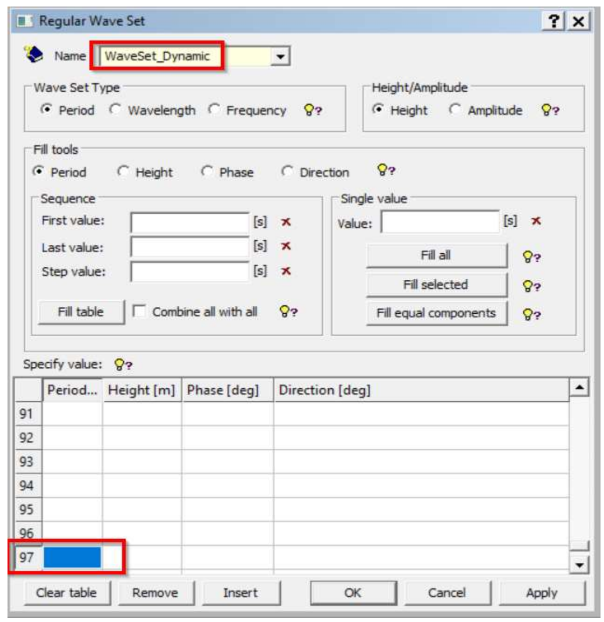
Note: Transfer function waves are selected by users based on the structural dynamic characters.

ISO 19902 2020 Section A.16.7.2.2 provides the guidelines for the selection of transfer function wave frequencies.

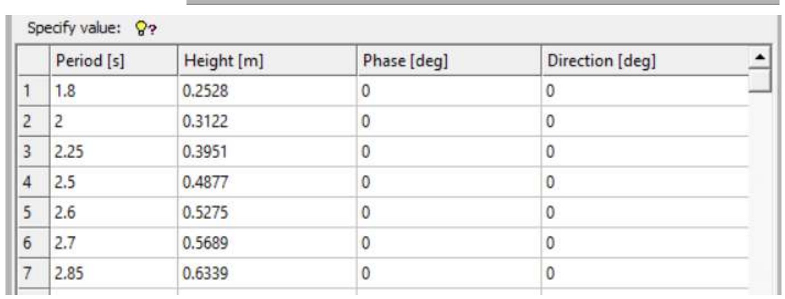
- To create new waves, go to **Environment** > **Water** > **Regular Waves**. Right-click **Regular Waves** folder and select **New Regular Wave Set**.



- In the dialog box, name this new wave set as **WaveSet_Dynamic** and add at least 96 (total number of seastates) lines by using ↓ downward key on the keyboard.



- Copy wave Period, Height, Phase angel, and Direction for all waves from the Excel spreadsheet and then paste them into this wave set.



Specify value:

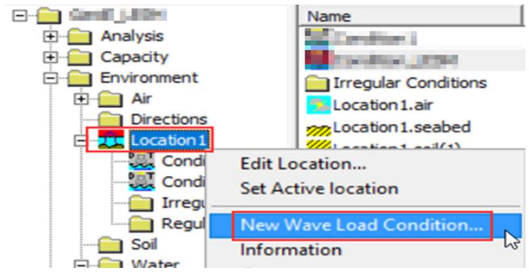
	Period [s]	Height [m]	Phase [deg]	Direction [deg]
1	1.8	0.2528	0	0
2	2	0.3122	0	0
3	2.25	0.3951	0	0
4	2.5	0.4877	0	0
5	2.6	0.5275	0	0
6	2.7	0.5689	0	0
7	2.85	0.6339	0	0

Once completed, click **OK** and the wave set will be created.

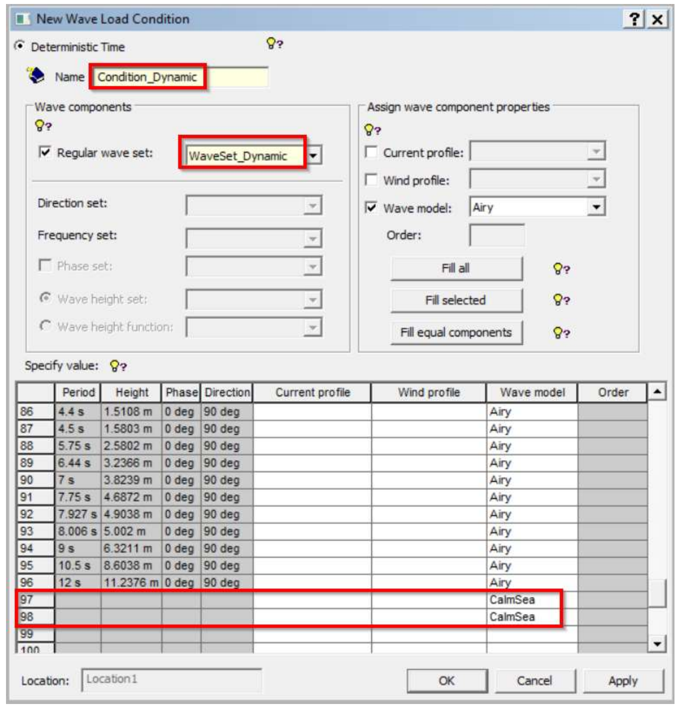
2.3 Defining Wave Load Condition

A new wave condition will be created to include the wave set **WaveSet_Dynamic** in order to include waves in the analysis.

- Create wave load condition under **Location1**, right-click on **Location1** and select **New Wave Load Condition**.



- Define the wave condition name as **Condition_Dynamic**, select wave set WaveSet_**Dynamic**, use **Airy** as wave model for all waves, and then include two additional seastates with **CalmSea** wave model.



- Click **OK** and **Condition_ Dynamic** is now created. This wave load condition will be included in the dynamic analysis.

2.4 Creating a Set for Fatigue Damage Calculations

A set containing a joint and all beams connected to the joint is created and will be included in the Framework fatigue damage calculations.

- Select **JT176** and all connected beams, see right. Create a set and name it as **Set_Fat**. This set will be included in the fatigue analysis in Framework.

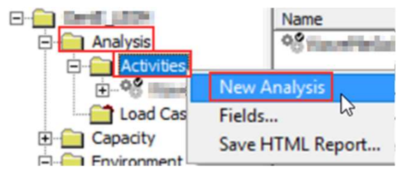
3 DYNAMIC ANALYSIS – GENIE

A new analysis containing a linear dynamic analysis and a non-linear Splice analysis is created. The linear dynamic analysis is performed to generate the structural dynamic responses with equivalent static loads, and then the Splice analysis is executed to analyze the structure with the pile-soil foundation.

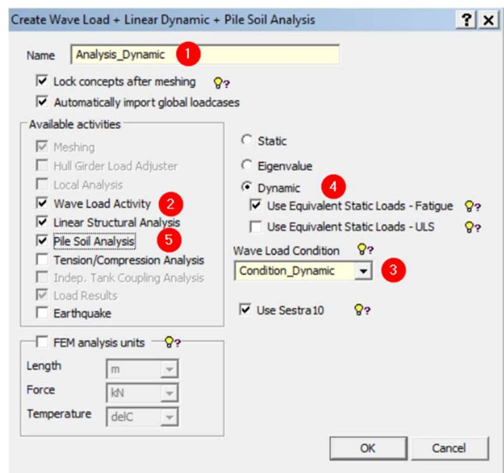
Please note that there are two sets of support conditions included in the model. The pilehead springs will be included in the linear dynamic analysis and the pile-soil foundation will be used in the subsequent non-linear Splice analysis.

3.1 Creating Analysis

- To create the analysis, go to **Analysis** > **Activities,** right-click and select **New Analysis**.



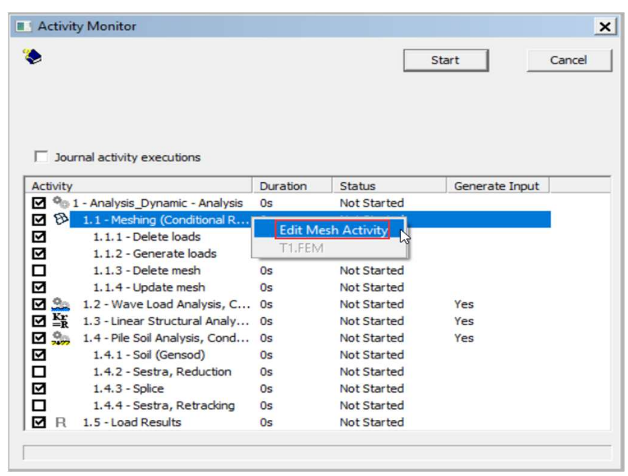
- Generate the analysis following the sequence: 1) Name the analysis to **Analysis_Dynamic**, 2) keep **Linear Structural Analysis** checked, and check **Wave Load Activity**, 3) include **Condition_Dynamic** for **Wave Load Condition**, 4) select **Dynamic** and check **Use Equivalent Static Loads – Fatigue**, 5) select **Pile Soil Analysis**.
- Click **OK** to confirm selections.



Now the new dynamic analysis is created. Exclude wind load cases, **LCEqWind0**, **LCEqWind52**, and **LCEqWind90** from the analysis.

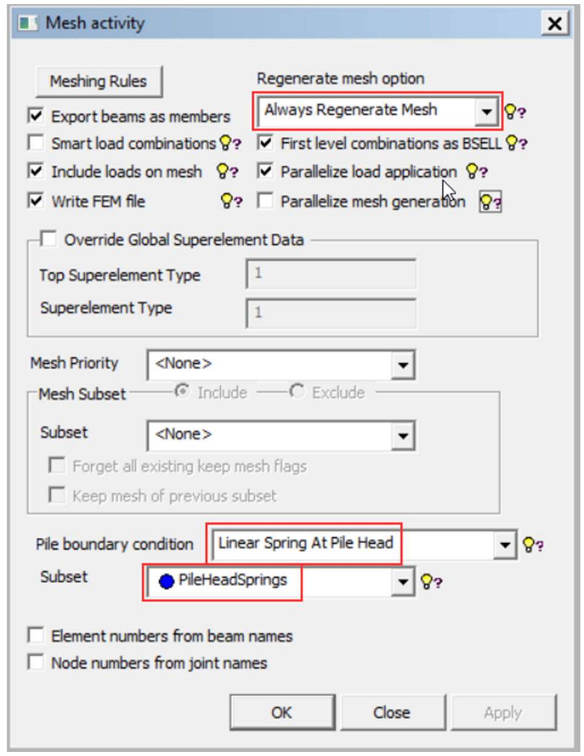
3.2 Editing Meshing

Use **Alt+D** to open the **Activity Monitor**, right-click on **1.1 – Meshing** and select **Edit Mesh Activity**.



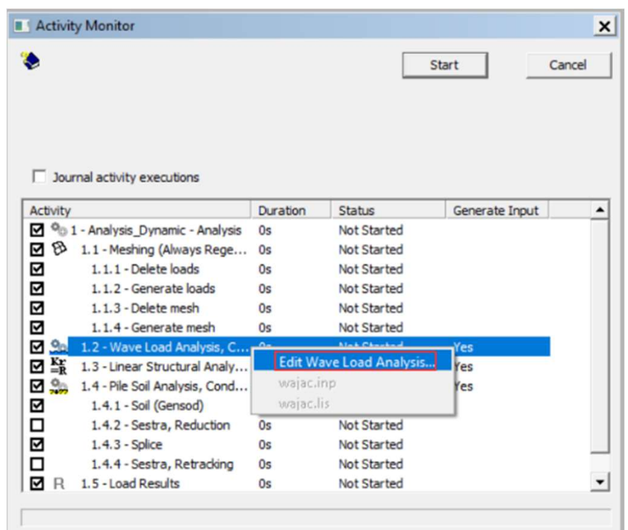
- Change **Regenerate mesh option** to **Always Regenerate Mesh**. Uncheck **Smart Load Combination** and check the option **First level combinations as BSELL**. This will make the load combination passed to add to the generated equivalent static loads (ESL).
- Select **Linear Springs At Pile Head** as the **Pile boundary condition** to provide the support condition for the linear dynamic analysis. Enter the set **PileHeadSprings** as the Subset.

  Note that all piles contained in the model will be automatically included in the Splice analysis.
- Click **OK** to save the selections.



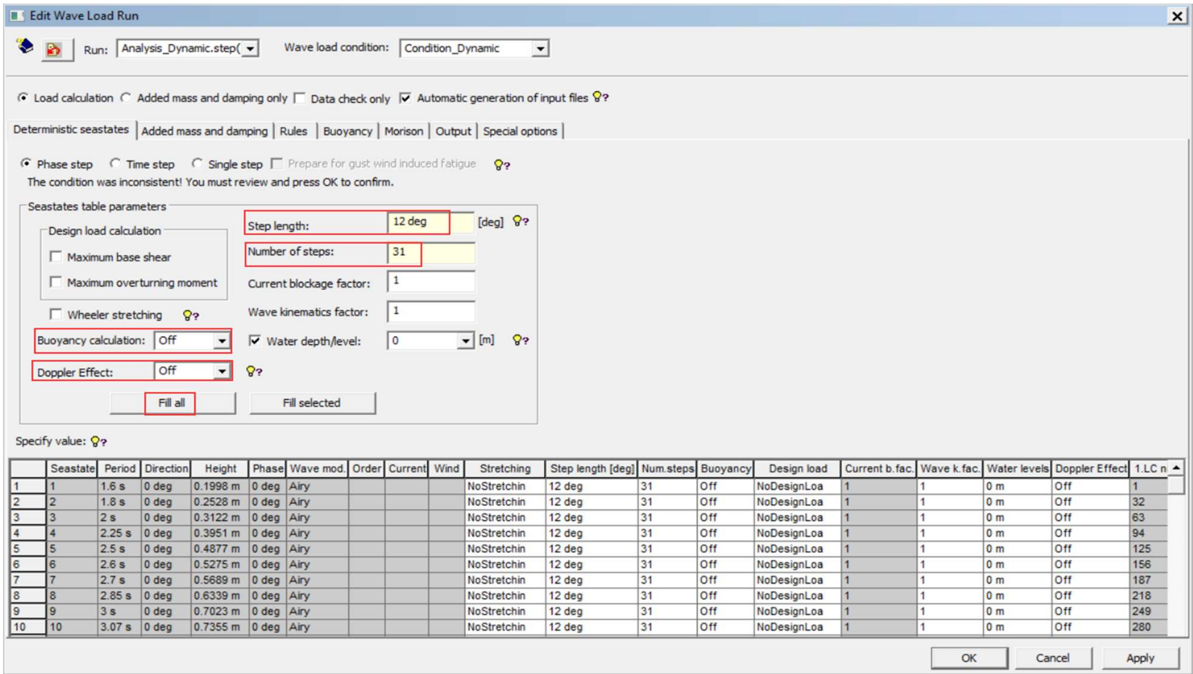
3.3 Editing Wave Load Analysis

- Righ-click on **1.2 – Wave Load Analysis** and select **Edit Wave Load Analysis.**

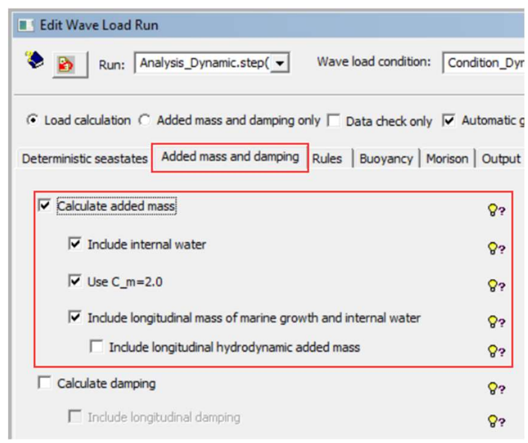


- Edit the wave load activity in the newly created analysis. Set the wave step length to 12 degrees, the number of steps to 31. Set buoyancy calculation **Off** and doppler effect **Off**, then click **Fill all**. Click **Apply** to save the input but keep the dialog box open.

  NOTE: In a dynamic analysis wave steps should include a full cycle 0 – 360 degrees wave propagation, such as if wave step length equal to 12 degrees, then 31 (instead 30) wave load steps should be used for wave load generation. Also, the number of wave steps must be less than 36.

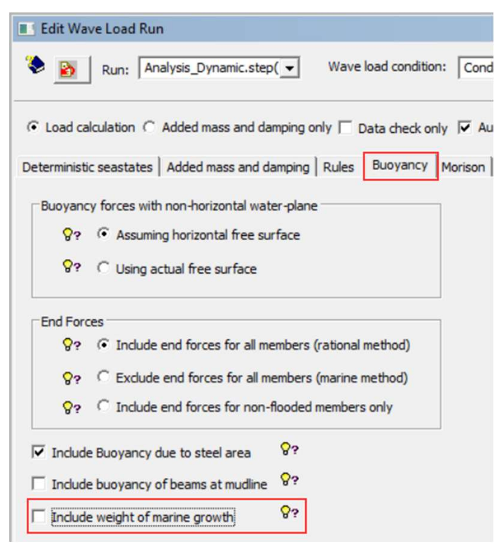


- On the **Added mass and damping** tab, select **Calculate added mass** and choose options **Include internal water**, **Use Cm=2.0**, and **Include longitudinal mass of marine growth and internal water**. Click **Apply**.



- On the **Buoyancy** tab, uncheck **Include weight of marine growth** option.

  NOTE: This option excludes the marine growth weight and buoyancy in the wave load calculations for seastates 1 to 99, which will be used in subsequent direct time domain dynamic analysis. Click **Apply** to save the inputs.



- Go back to **Deterministic Seastates** tab. Set two **Calmsea** load cases with the first calmsea line to account for the buoyancy of structure and buoyancy of marine growth, the second calmsea load case to account for the weight of marine growth.

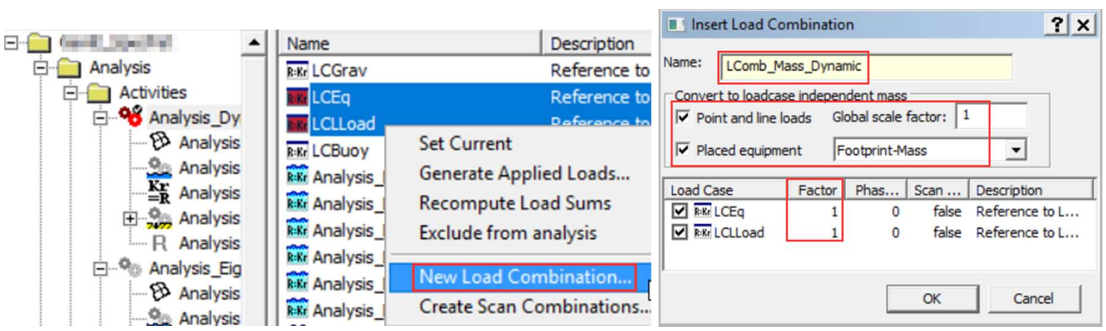
95	95	10.5 s	90 deg	8.6038 m	0 deg	Airy			NoStretching	12 deg	31	Off	NoDesignLoads
96	96	12 s	90 deg	11.2376 m	0 deg	Airy			NoStretching	12 deg	31	Off	NoDesignLoads
97	97					CalmSea			NoStretching	0 deg	1	Only	NoDesignLoads
98	98					CalmSea			NoStretching	0 deg	1	Weight	NoDesignLoads

- Click **OK** to apply all inputs and close the dialog box.

3.4 Generating Load Combinations

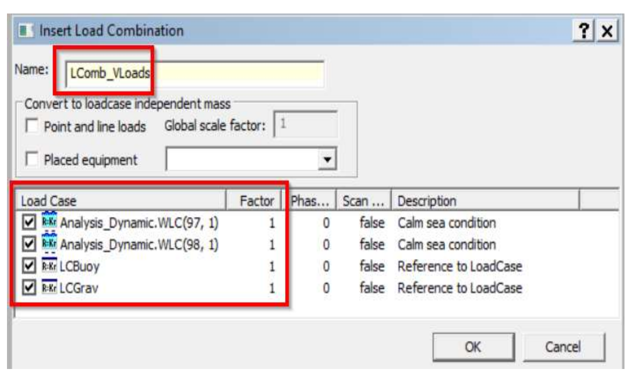
The mass of equipment and live load should be included in the dynamic analysis. Therefore, a load combination for these two load cases needs to be created and loads included in the load combination will be converted to mass in the dynamic analysis.

- Go to **Analysis** > **Activities** > **Analysis_Dynamic**, select load cases, LCEq and LCLLoad, which are included in the load combination, right-click and select **New Load Combination**.
- Name the load combination with name **LComb_Mass_Dynamic**. Check options **Point and line loads** and **Placed equipment** with **Footprint-Mass**. Give the load factor for each load cases as shown below. Loads included in this load combination will be converted to mass in the analysis and the weight of these masses will be included in the load case, LCGrav, in which the structural self-weight is included.

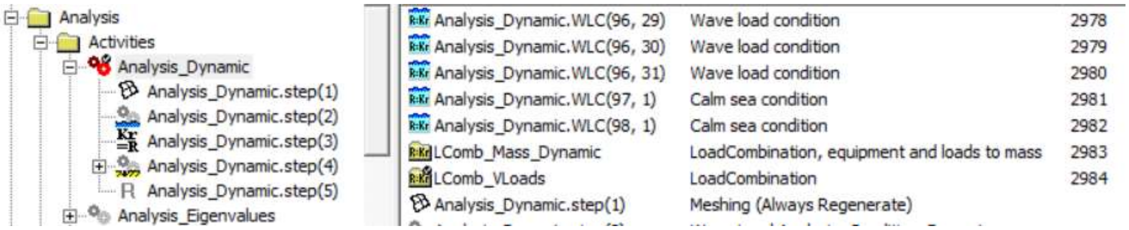


Since a Splice analysis will be performed after the dynamic analysis, a load combination containing all vertical loads, such as the structural self-weight, equipment loads, deck live loads, marine growth weight, and all buoyancies, need to be created. The calculated dynamic effect plus the loads in this load combination will be included in the static Splice analysis.

- To create a load combination for ESL analysis, select load case **LCGrav**, **LCBuoy**, **Analysis_Dynamic.WLC(97,1)** and **Analysis_Dynamic.WLC(98,1)**, right-click and choose **New Load Combination**. Name it as **LCOMB_VLoads.**
- The load case factors are shown in the image.



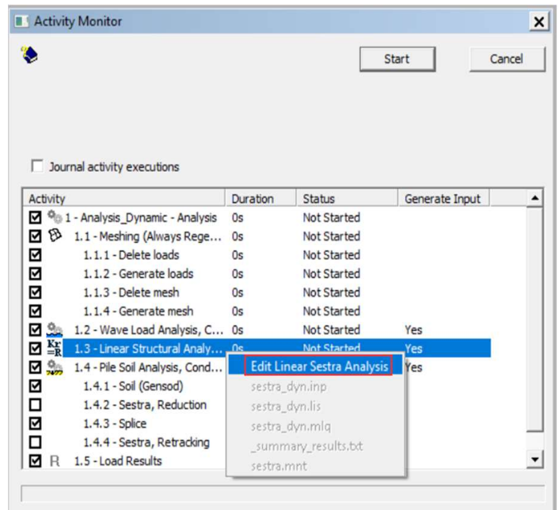
Now two load combinations are included in the analysis.



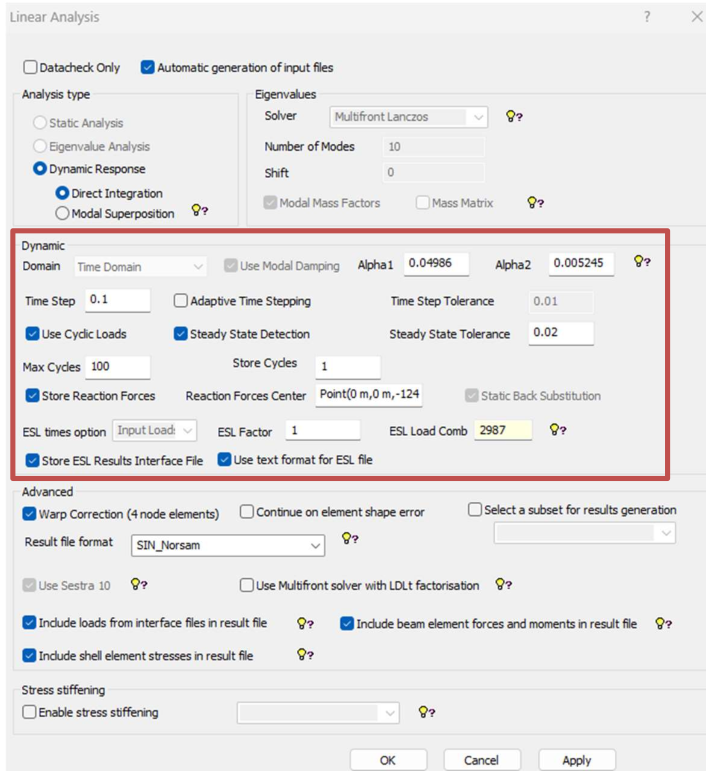
3.5 Editing Linear Sestra Analysis

The dynamic analysis parameters, such as the analysis method, Rayleigh damping ratios, the time integration step size, the option to store reaction forces, the options to include the combined vertical loads, and the options of storing ESL result interface file, need to be defined.

- Right-click on **1.3 – Linear Structural Analysis** and select **Edit Linear Sestra Analysis**.



- Enter the following parameters:
  i. Raileigh damping ratios **Alpha1** = 0.04986; **Alphat2** = 0.005245;
  ii. Set the **Time step** to 0.02;
  iii. Check **Use Cyclic Loads** and **Steady State Detection**;
  iv. Enter **Max Cycles** to 100 and **Store Cycles** to 1;
  v. Check **Store Reaction Forces**;
  vi. Set the **Reaction Forces Centre** to point (0m, 0m, -124m);
  vii. Input **ESL Factor** to 1. Use FEM Loadcase number of **LCOMB_VLoads** and set it as **ESL Load Comb.**
  viii. Check **Store ESL Result Interface File** and **Use text format for ESL file**.



Click **OK** to save all inputs.

3.6 Editing Splice Analysis

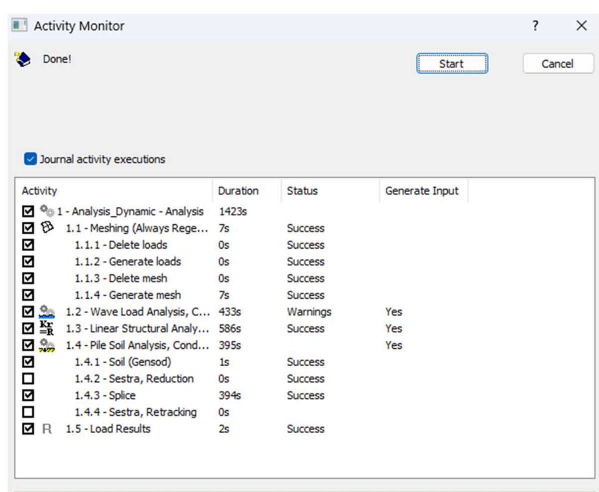
The Splice analysis will include the structural dynamic effects and the vertical loads. The results from this analysis will be used to perform a spectral fatigue analysis for the jacket structure and piles below the mudline.

- All default options will be used in the Splice analysis.

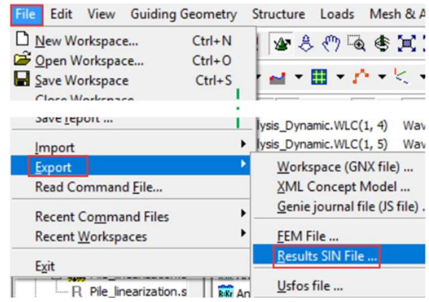
3.7 Executing Analysis

With all the parameters set up, the analysis can now be run.

- Click **Alt+D** on the keyboard to open the **Activity Monitor** and click **Start** to run the analysis.
- The run will take some time to finish.



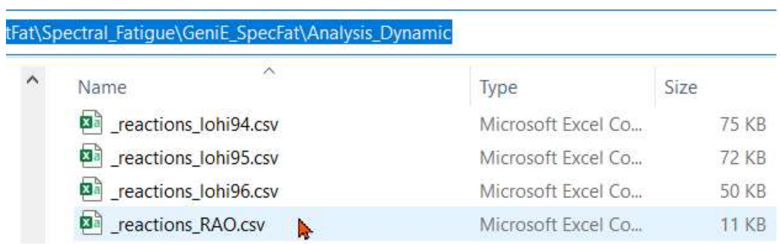
- To export the result file into **_repository** folder, go to **File** > **Export** > **Result SIN File**, and name the result file as **Spectral_R1.SIN**.



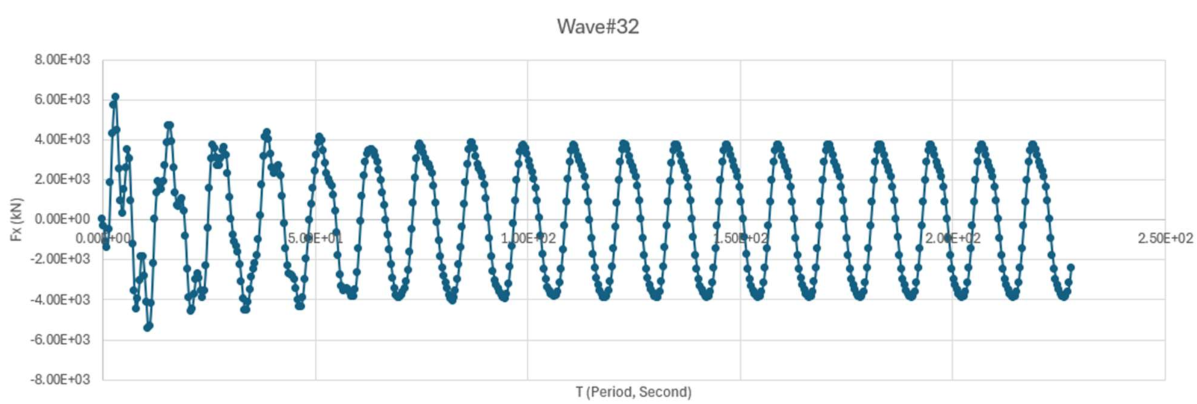
Save the model and export it to **_repository** folder as **Model_SpectralFatigue_Done.gnx**.

3.8 Structural Dynamic Responses

Structural response files, reaction files for each wave (**_reactions_history##.csv**), and a transfer function file (**_reactions_RAO.csv)** are created automatically when the analysis is completed. All files are located at the analysis folder … **Spectral_Fatigue\GeniE_SpecFat\Analysis_Dynamic**.



Below is the structural response for wave Period = 12.0 seconds, Height = 11.2376 m, Direction = 0 degree taken from **__reactions_history32.csv** file. There are 20 cycles analysed, and the steady state is reached at the last cycle. Responses from the last cycle are saved and will be used in the Splice analysis.



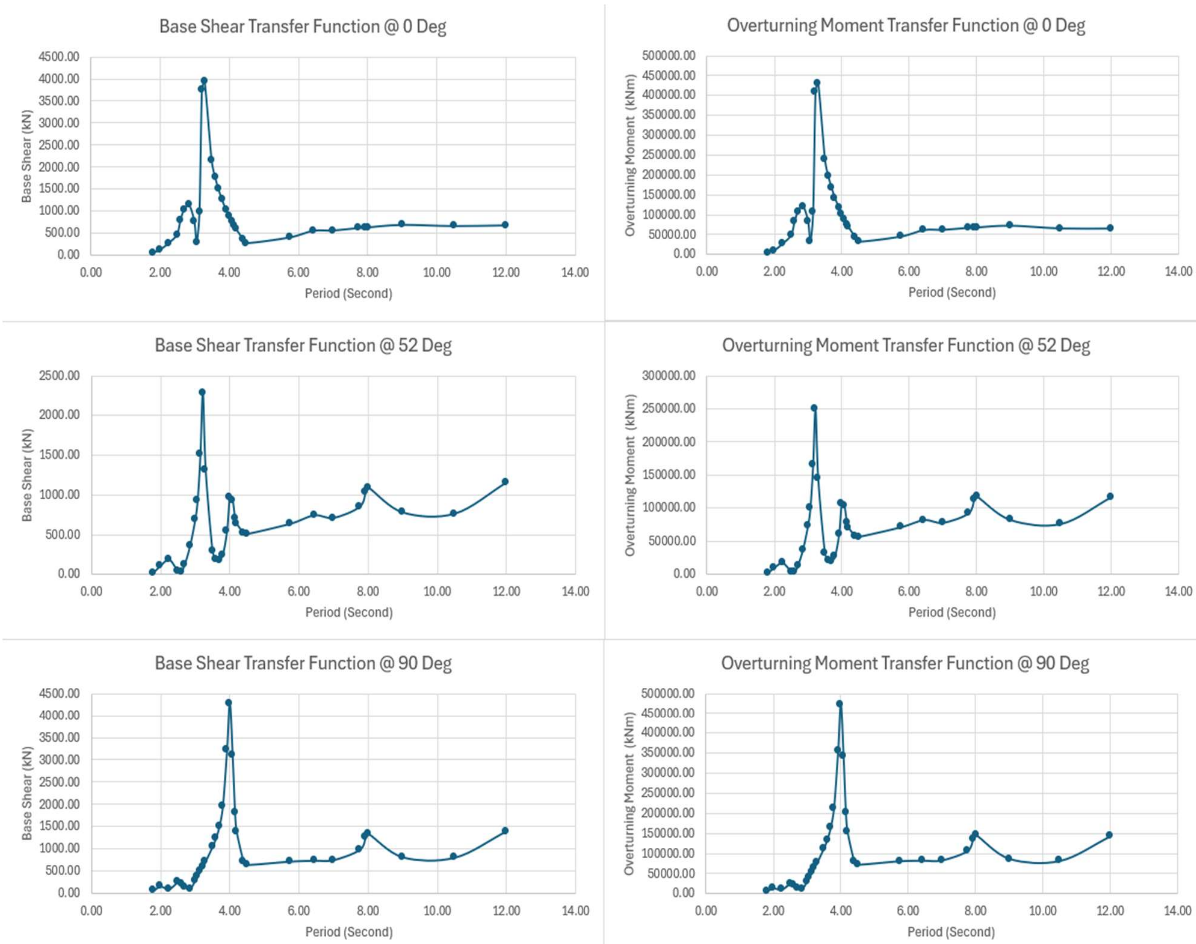
The **__reactions_RAO.csv** file can be opened to inspect the base shear and overturning moment transfer functions.

- Create the base shear and overturning moment transfer functions using the data contained in **__reactions_RAO.csv** file.

i. For base shears, compute the resultant base shears from the reported base forces in global X-direction and Y-direction.

ii. For overturning moments, compute the resultant overturning moments from the reported overturning moments about the global X-axis and the moments about the global Y-axis.

- Generate base shear and overturning moment transfer function plots in Excel spreadsheet.



4 SPECTRAL FATIGUE ANALYSIS – FRAMEWORK

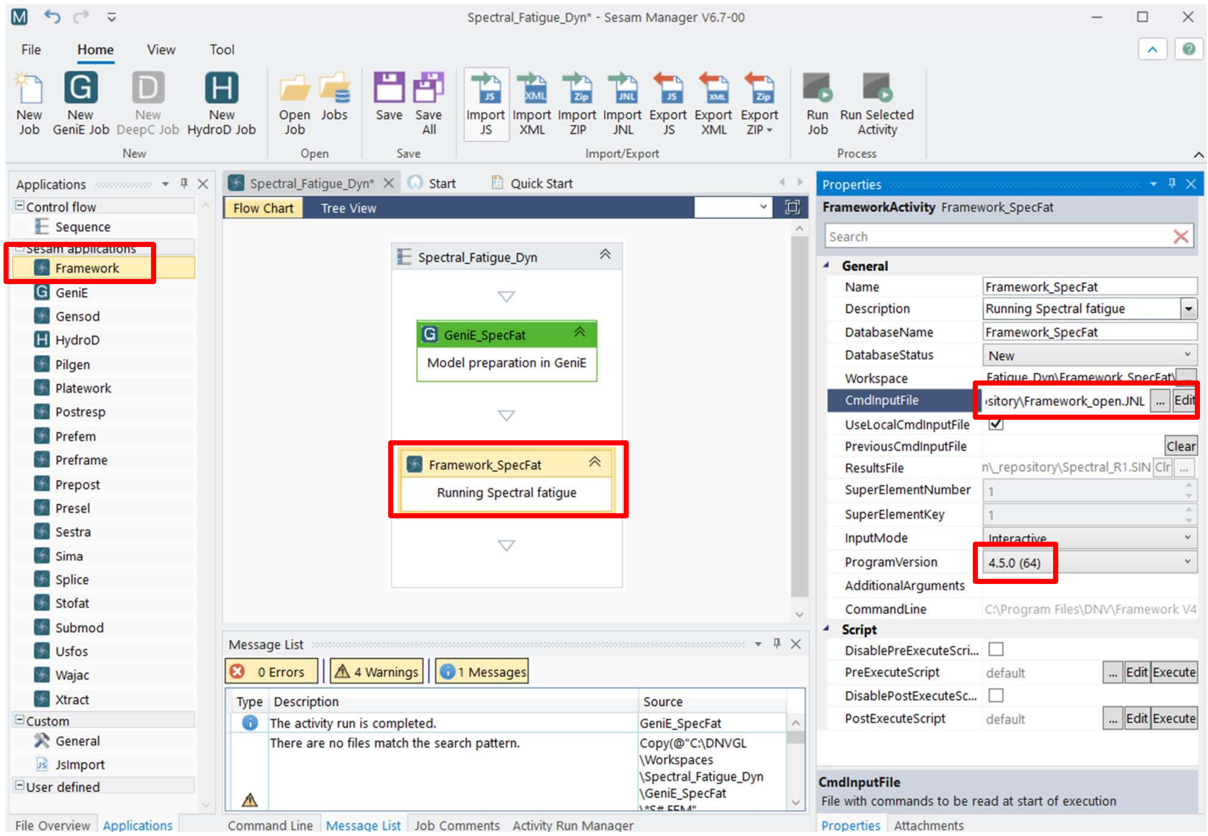
With the results in place, the spectral fatigue analysis can be run.

First, create an empty text file in **_repository** folder named **Framework_open.jnl**. This will be used to start Framework without any commands and without opening the results file yet, so that some default settings can be specified first (which are applied to the result file upon opening it).

Drag Framework into the work area, and name it as **Framework_SpecFat**. In the Framework activity properties browse to select the **CmdInputFile** and select the newly created **Framework_open.jnl** file in **_repository** as the input file.

After specifying the command input file, click Framework activity to run Framework.

Framework_open.jnl is also provided in the input file folder in this example.

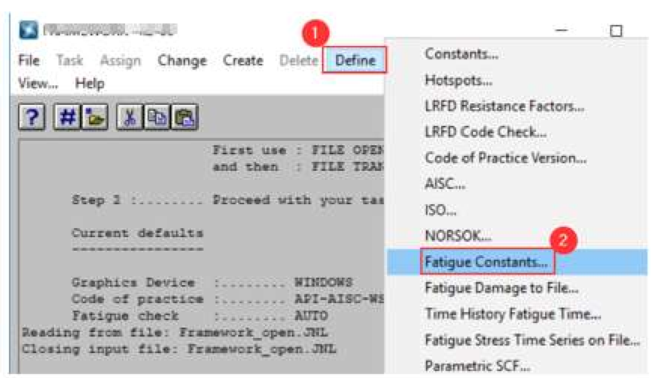


4.1 General Input

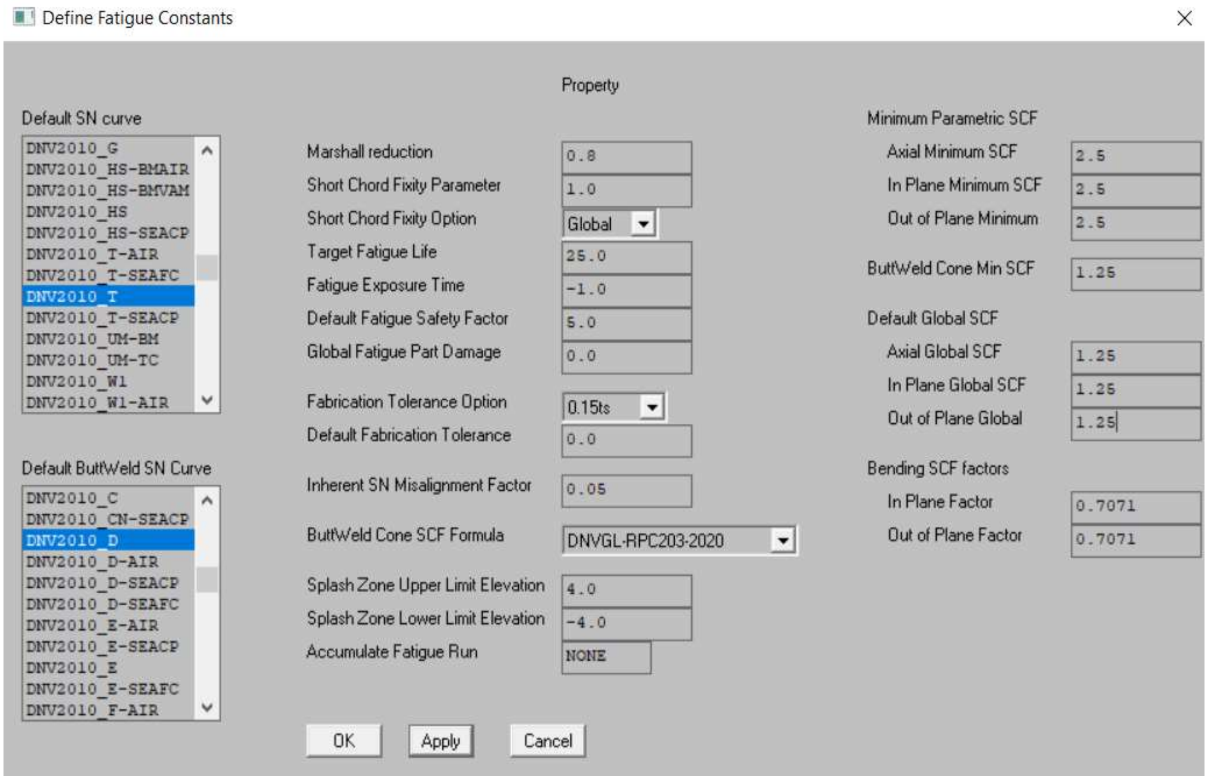
Before import the result file, global fatigue parameters need to be defined in a new Framework workspace.

4.1.1 Fatigue Constants and Global Settings

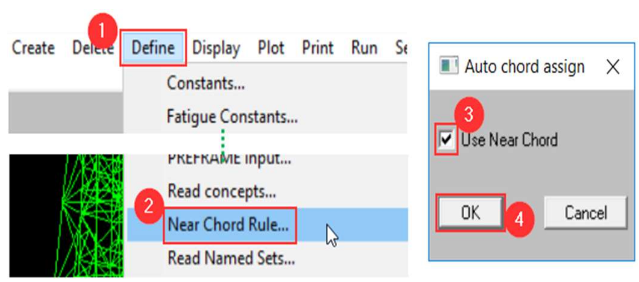
- Go to **Define** > **Fatigue Constants** to open the dialog for inputting the global fatigue check options.



- Enter the selections as shown below. Click **OK** to confirm inputs.

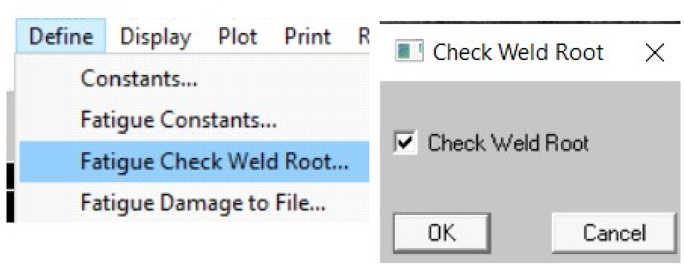


- Go to **Define** > **Near Chord Rule** and enable the checkbox **Use Near Chord**.



NOTE: This will ensure that the chord member and the aligned chord member of a brace do not change when the structure and its load pattern are subject to a rotation. This is useful when multiple fatigue analyses are performed on the same model in different states, e.g. when the model is standing and when the same model is placed horizontally for transportation in another fatigue analysis. This ensures that the hotspots are the same in both analyses, so that results can be summed correctly.

- Go to **Define** > **Fatigue Check Weld Root** and enable the option. Then fatigue damages will be checked at weld root positions for tubular joint welds and butt-welds.



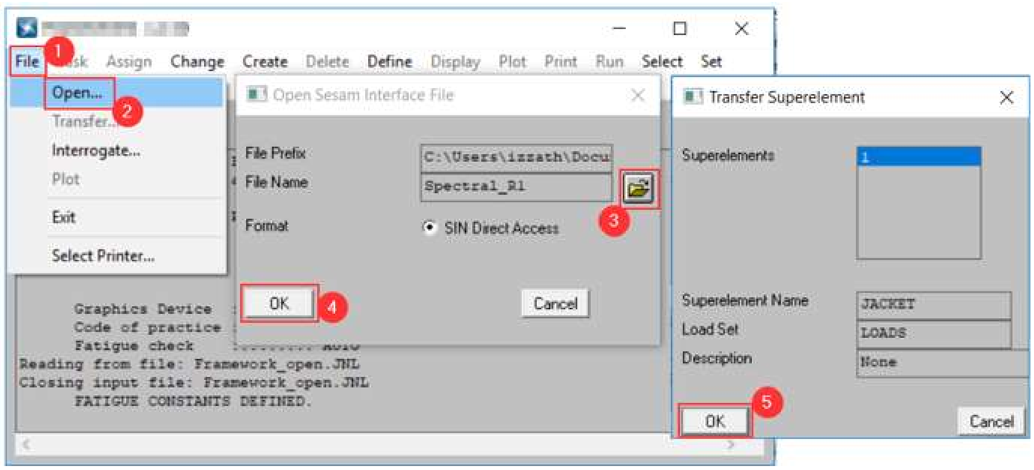
Note: The commands **DEFINE NEAR-CHOR-RULE** and **DEFINE FATIGUE-CHECK-WELD-ROOT** must be issued before the result file is read in.

4.1.2 Importing Result File

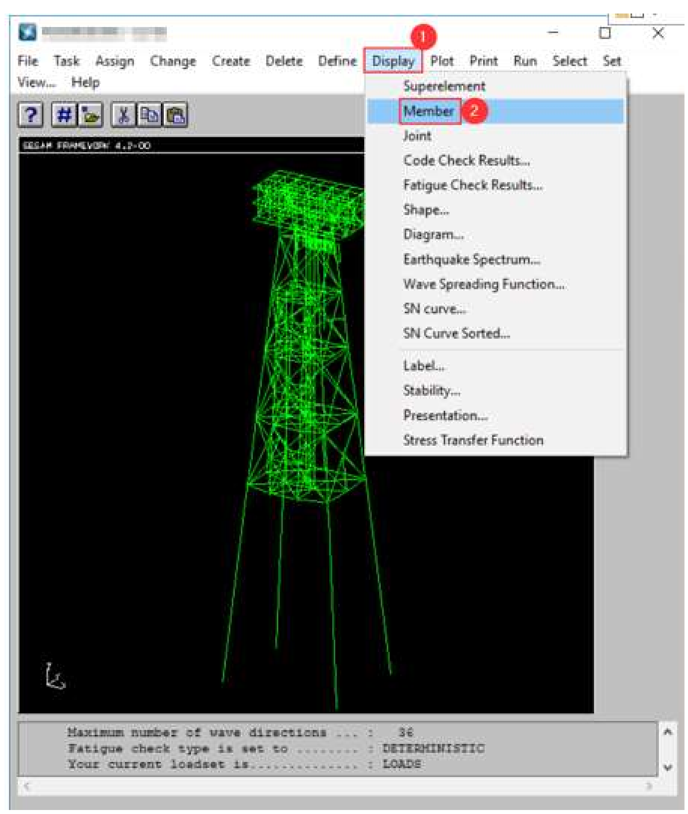
Now the result file can be read into Framework.

- Go to **File** > **Open**. Click the browse folder button and locate the **Spectral_R1.SIN** file in _repository folder, press **OK**.

  Transfer Superelement dialog box is opened. Leave all defaults as is and press **OK** to start reading in the results file.

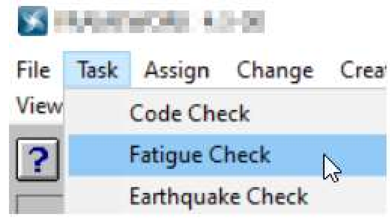


- After the file is read in, click **Display** > **Member** to get the structure displayed.

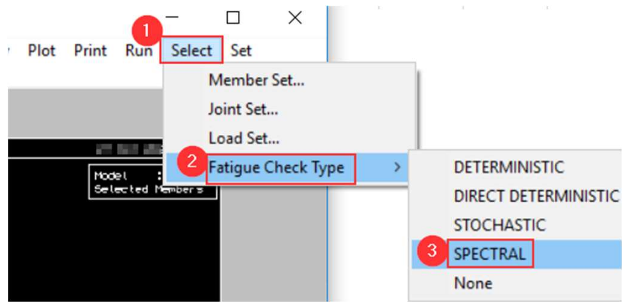


4.1.3 Specifying Task and Analysis

- Define the task by going to **Task** > **Fatigue Check**. This will limit the displayed menu items in Framework to those relevant for a fatigue analysis.

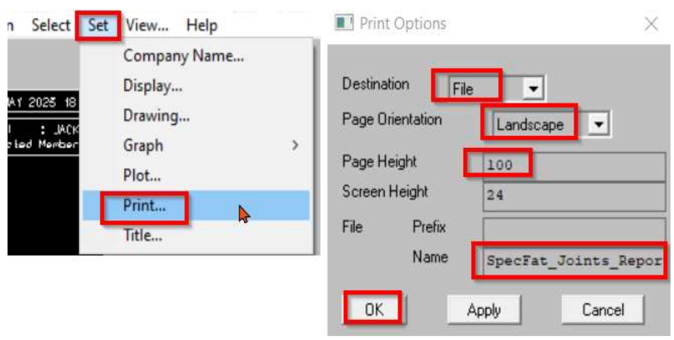


- Go to **Select** > **Fatigue Check Type** > **Spectral** to define that a spectral fatigue analysis will be run.



4.1.4 Setting Up Print Output

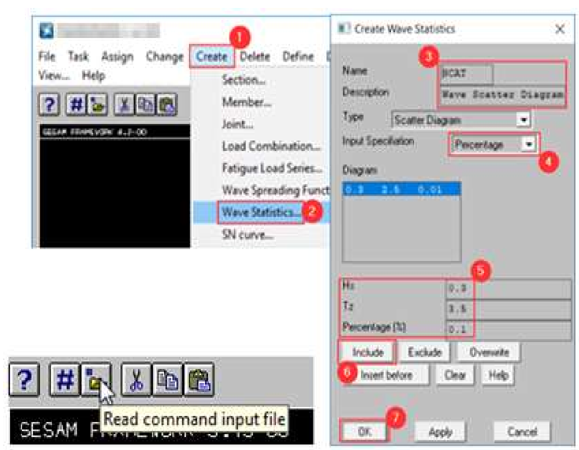
- Go to **Set** > **Print** to set **Print Options** as shown in the image, define the file name as **Framework_SpecFat_Joints_Report**



4.2 Fatigue Loads – Wave Scatter Diagrams

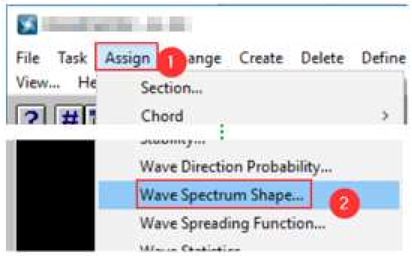
Next, the environmental data will be specified and assigned to each wave direction.

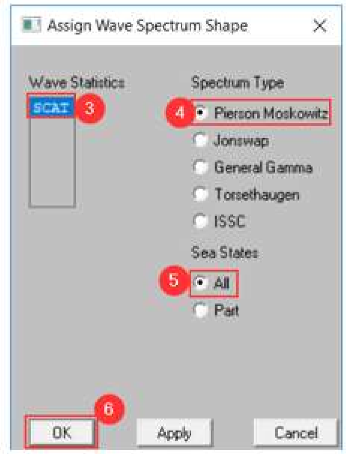
- Go to **Create** > **Wave Statistics**, input the wave scatter diagram. This scatter diagram will be applied in all three wave directions. Input Hs, Tz, & Percentage (%), and select **Include**. Repeat to complete the entire scatter diagram, then click **OK**.
- Alternatively, the data can be read in via a command file. Click the **Read Command Input File** button in the toolbar. Select the **wave_scatter.jnl** file from the input files.



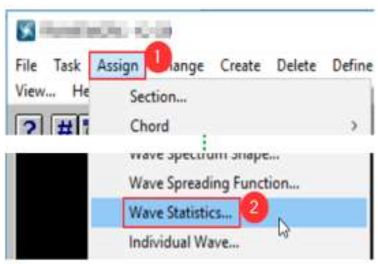
NOTE: Make sure that there are no spaces in the path to the input file, otherwise Framework will fail to read it in.

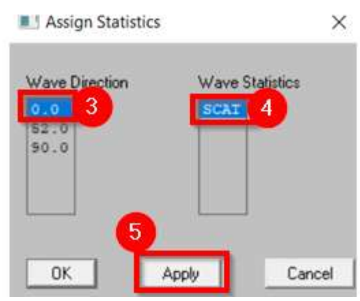
- Assign the wave spectrum shape to each wave scatter diagram via **Assign** > **Wave Spectrum Shape** as **Pierson-Moskowitz** spectrum.



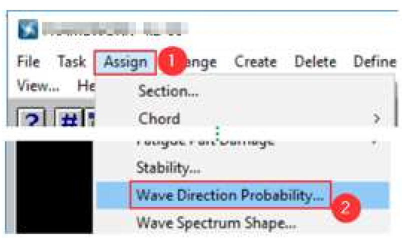


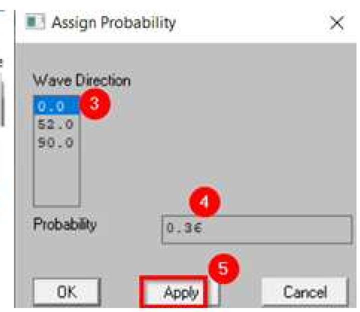
- Assign the scatter diagram to each wave direction. Go to **Assign** > **Wave Statistic**, select 0.0-degree wave direction and click **SCAT** wave statistic, press Apply. Repeat this for waves in 52 and 90 degree directions.
- Click **Cancel** to close the dialog box.





- Assign the wave direction probabilities via **Assign** > **Wave Direction Probability**. Assign a probability of 0.36 to direction 0 degree, a probability of 0.40 to direction 52 degrees, and a probability of 0.24 to direction 90 degrees.
- Click **Cancel** to close the dialog box.



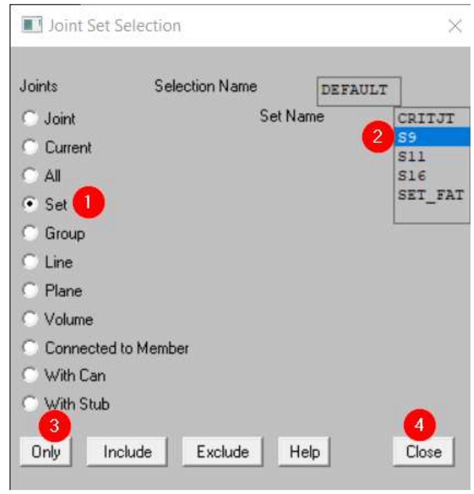


4.3 Joint and Member Selection

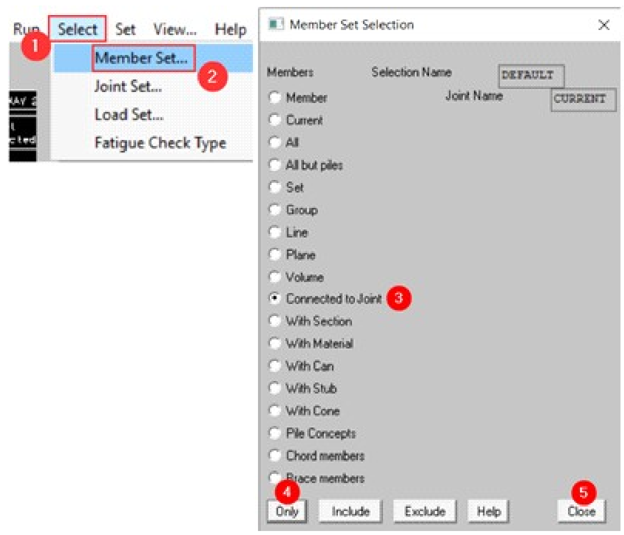
The joints and members included in the fatigue analysis can be selected before the analysis is performed. If the set name of joints and members are longer than 8 characters, Framework will rename them. The naming map is reported in Framework.MLG file.

```
* SET        9 has name longer than      8 characters.
* Name is CritJT_withBraces
* Name S9        will be used
```

- Go to **Select** > **Joint Set** > **Set**, select the set **S9**, then click **Only** and **Close**.



- Go to **Select** > **Member Set**, select **Connected to Joint** option, click **Only** and **Close**.

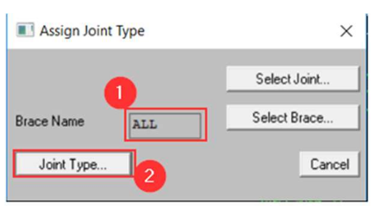


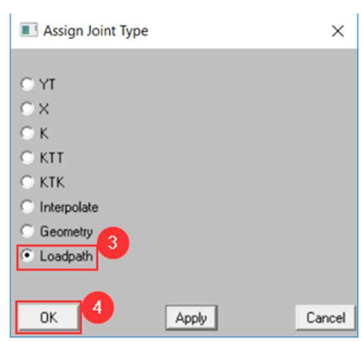
4.4 SCF Related Inputs

In a fatigue analysis, user needs to define SCF calculation rules for tubular connections and for butt-welds. If needed local SCFs can be defined for some selection tubular connections and the selected butt-weld positions.

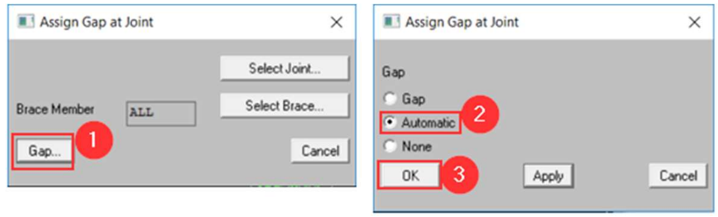
The following commands are under **Assign** menu. We only define SCFs for joints in this section.

- Go to **Assign** > **Joint Type**, click **Joint Type** and select **Loadpath**. This will be assigned to the selected joint set S9 and all braces connected to the joints. The defined joint classification rule will be used in SCF calculations.

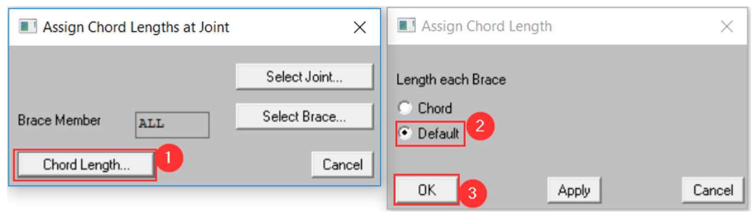




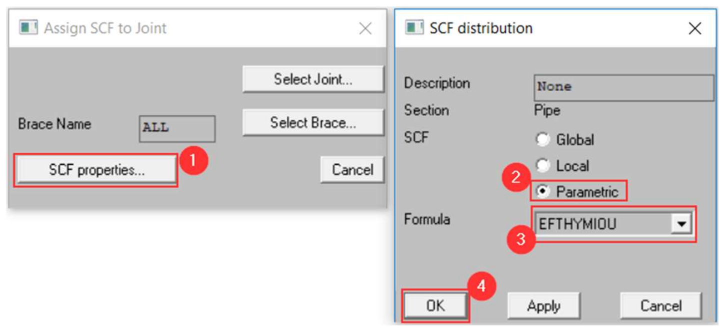
- Go to **Assign** > **Joint Gap** and make sure that gaps are automatically computed based on the modelled gaps for all selected joints.



- Go to **Assign** > **Joint Chord Length** and make sure that the calculated chord length from the model is used as the chord length for SCF calculations.



- Go to **Assign** > **SCF** > **Joint**, assign parametric SCFs according to Efthymiou to all joints.

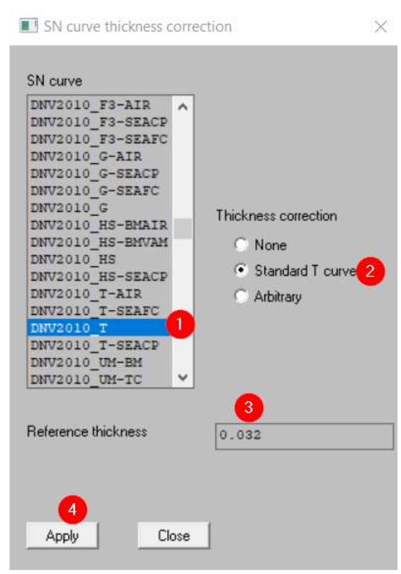


4.5 S-N Curve Related Inputs

In Framework, user can enter S-N curve related inputs using **Assign** commands.

The global S-N curve is defined in **Fatigue Constants**. User is able to define different ones for some selected joints. User can also specify the S-N curve thickness correction to for the selected S-N curves.

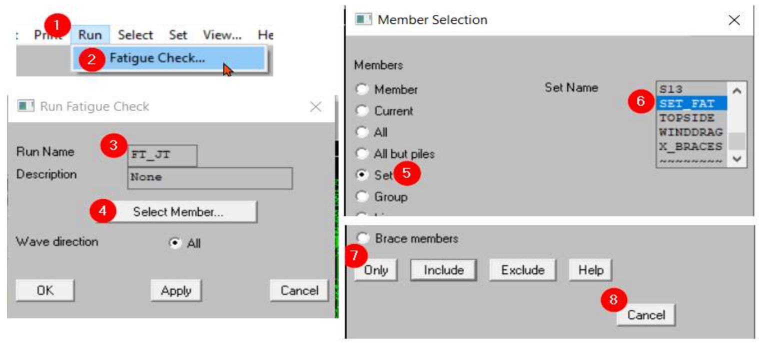
- Go to **Assign** > **Thickness Correction**, select the curve **DNV2010_T** and make sure a **Standard T-curve** is used for the thickness correction with the reference thickness set to 32 mm (i.e. 0.032 m). Press **Apply** and **Close**.



4.6 Fatigue Analysis for Tubular Connections

The spectral fatigue damage analysis for tubular connections will now be run. In this example the tubular connections and members included in set **Set_Fat** are included in the analysis.

- Go to **Run** > **Fatigue Check,** name the run as **FT_JT**, select members included in the set **SET_FAT**, press **OK** to run the analysis.



- After the analysis completed, go to **Print** > **Fatigue Check Results**, select the analysis results to be printed into the listing file for tubular connections included in set **SET_FAT**.

The fatigue listing file is in **Framework_SpecFat** activity folder. The results are reported as below.

```
                                     SPECTRAL          fatigue check results
                                     Run:       Superelement:      Loadset:
                                     FT_JT      JACKET             LOADS
                                     Priority....: Selected Joints
                                     Usage factor: Above    0.00                                                     SUB PAGE:     2

Joint      Brace           Outcome   Damage    Life     WeldSide        Hot  SCFrule    SCFax     SCFipb  SCFopb  SNcurve
           Chord                               Alpha    Symmet      DiaBra   ThiBra      Gap     ThiFac    QR       Cycles
                                               Theta    Jtype       DiaCho   ThiCho    LenCho    FixCho  SCFaxC     SCFaxS
--------------------------------------------------------------------------------------------------------------------------------
JT176      BM167                     6.93E-01  1.80E+02 CHORD-SID         7 EFTHYMIOU   6.053     2.500   3.274 DNV2010_T
           BM21                                180.158  CROWN-SAD  8.00E-01   0.025    0.00E+00  1.216   1.000       6.44E+08
                                                83.385  YT         1.90E+00   0.070    6.04E+01  1.000   6.053          4.810
                                     2.85E-01  4.39E+02 BRACE-SID         1 EFTHYMIOU   6.608     2.500   3.982 DNV2010_T
                                               180.158  CROWN-SAD  8.00E-01   0.025    0.00E+00  1.000   1.000       7.19E+08
                                                83.385  YT         1.90E+00   0.070    6.04E+01  1.000   4.257          6.608
                                     3.16E-01  3.96E+02 CHORD-ROO         7 EFTHYMIOU   3.632     2.500   2.500 DNV2010_E-AIR
                                               180.158  CROWN-SAD  8.00E-01   0.025    0.00E+00  1.229   1.000       6.88E+08
                                                83.385  YT         1.90E+00   0.070    6.04E+01  1.000   3.632          2.886
                                     3.93E-01  3.18E+02 BRACE-ROO        10 EFTHYMIOU   3.260     2.500   2.500 DNV2010_F3-AIR
                                               180.158  CROWN-SAD  8.00E-01   0.025    0.00E+00  1.000   1.000       8.06E+08
                                                83.385  YT         1.90E+00   0.070    6.04E+01  1.000   2.554          3.965

           BM182                     5.64E-02  2.22E+03 BRACE-ROO         1 EFTHYMIOU   2.500     2.500   3.054 DNV2010_F3-AIR
           BM21                                175.855  CROWN-SAD  1.00E+00   0.030    2.14E-01  1.047   1.000       6.28E+08
                                                84.838  KTT/LPD    1.90E+00   0.070    6.04E+01  1.000   2.500          2.500
                                     5.44E-02  2.30E+03 CHORD-ROO        19 EFTHYMIOU   2.500     2.500   2.941 DNV2010_F-AIR
                                               175.855  CROWN-SAD  1.00E+00   0.030    2.14E-01  1.294   1.000       6.73E+08
                                                84.838  KTT/LPD    1.90E+00   0.070    6.04E+01  1.000   2.500          2.500
                                     6.66E-02  1.88E+03 BRACE-SID         1 EFTHYMIOU   3.882     2.500   5.065 DNV2010_T
                                               175.855  CROWN-SAD  1.00E+00   0.030    2.14E-01  1.000   1.000       6.25E+08
                                                84.838  KTT/LPD    1.90E+00   0.070    6.04E+01  1.000   3.882          3.882
                                     1.41E-01  8.86E+02 CHORD-SID         1 EFTHYMIOU   4.057     2.500   4.877 DNV2010_T
                                               175.855  CROWN-SAD  1.00E+00   0.030    2.14E-01  1.216   1.000       6.29E+08
                                                84.838  KTT/LPD    1.90E+00   0.070    6.04E+01  1.000   4.057          4.057
```

NOTE: By default S-N curves DNV2010_E-AIR or DNV2010_F-AIR is assigned to the chord root position based on the brace thickness, and S-N curve DNV2010_F3-AIR is assigned to the brace root position.

There are some calculation data, such as tubular connection SCFs, the fatigue check data, and fatigue check positions, can also be printed. For example, the below command can be used to print out the details of S-N curves used in each connection and butt-weld position.

```
PRINT MEMBER FATIGUE-CHECK-POSITIONS ( )
```

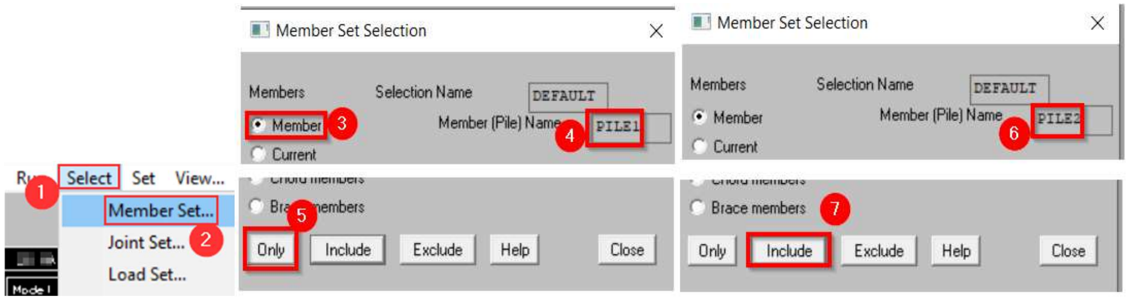
```
                                   Member fatigue check positions


NOMENCLATURE:

Member          Name of member
Joint/Po        Joint name or position within the member
SecTy           Section type
PositionName    Name to indentify position
CoorX           X coordinate of position
CoorY           Y coordinate of position
CoorZ           Z coordinate of position
LocSpl          Location w.r.t. splash zone. AIR-in air; SPL-inside splash zone; SEA-below splash zone.
SNcurve         Name of SN curve assigned

Note:
Splash zone UPPER limit is      4.000
Splash zone LOWER limit is     -4.000


Member   Joint/Po SecTy PositionName              CoorX    CoorY    CoorZ  LocSpl SNcurve
-------------------------------------------------------------------------------------------------
BM21     JT176    PIPE  CHORD-SIDE-0.0000          12.99     9.89   -19.00  SEA   DNV2010_T
         JT176    PIPE  BRACE-SIDE-0.0000          12.99     9.89   -19.00  SEA   DNV2010_T
         JT176    PIPE  CHORD-ROOT-0.0000          12.99     9.89   -19.00  SEA   DNV2010_F-AIR
         JT176    PIPE  BRACE-ROOT-0.0000          12.99     9.89   -19.00  SEA   DNV2010_F3-AIR
         0.12     PIPE  ROOTSectLEGGRUP-0.1217     12.66     9.63   -15.35  SEA   DNV2010_F3-AIR
         0.12     PIPE  Section-LEGGRUP-0.1217     12.66     9.63   -15.35  SEA   DNV2010_D
         0.12     PIPE  Section-LEGINTR-0.1217     12.66     9.63   -15.35  SEA   DNV2010_D
         0.12     PIPE  ROOTSectLEGINTR-0.1218     12.66     9.63   -15.35  SEA   DNV2010_F3-AIR
         0.90     PIPE  ROOTSectLEGINTR-0.9013     10.54     7.93     8.04  AIR   DNV2010_F3-AIR
         0.90     PIPE  Section-LEGINTR-0.9014     10.54     7.93     8.04  AIR   DNV2010_D
         0.90     PIPE  Section-LEGGRUP-0.9014     10.54     7.93     8.04  AIR   DNV2010_D
         0.90     PIPE  ROOTSectLEGGRUP-0.9014     10.54     7.93     8.04  AIR   DNV2010_F3-AIR
         JT197    PIPE  BRACE-ROOT-0.9999          10.27     7.72    11.00  AIR   DNV2010_F3-AIR
         JT197    PIPE  CHORD-ROOT-0.9999          10.27     7.72    11.00  AIR   DNV2010_F-AIR
         JT197    PIPE  BRACE-SIDE-1.0000          10.27     7.72    11.00  AIR   DNV2010_T
         JT197    PIPE  CHORD-SIDE-1.0000          10.27     7.72    11.00  AIR   DNV2010_T
```

4.7 Fatigue Analysis for Member Butt-Welds

The fatigue damage analysis can be performed for member butt-welds. In this analysis the butt-welds on piles are checked.

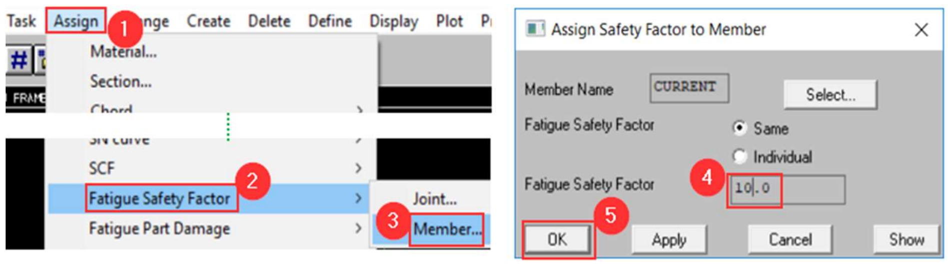
4.7.1 Pile Selection

- Go to **Select** > **Member Set**, select **Member** and input **Pile1** for **Member (Pile) Name**, click Only to include the selection.
- Input the second pile **Pile2** and click **Include** to add it to the selection.
- Repeat the selection to include Pile3 and Pile4.



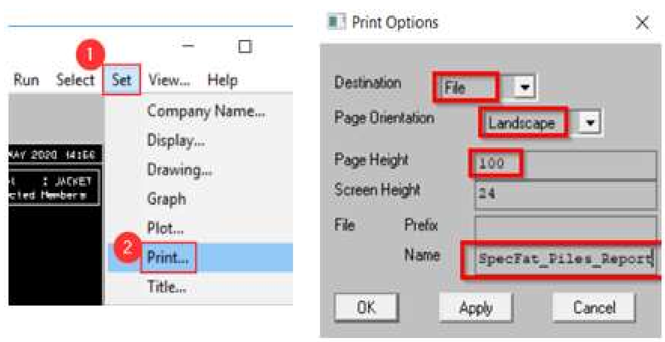
4.7.2 Pile Safety Factor

- To include a different safety factor for pile fatigue checks, to **Assign** > **Fatigue Safety Factor** > **Member**, assign a fatigue safety factor of 10 to all piles.



4.7.3 Set Up Print Output

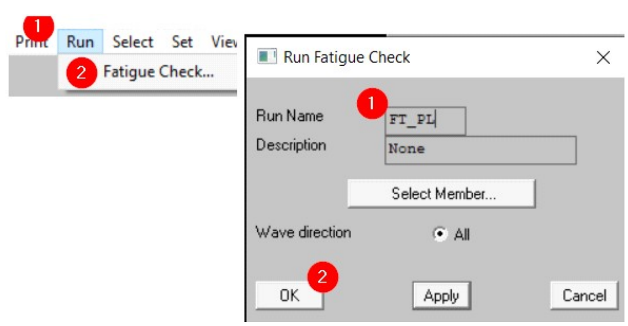
- Go to **Set** > **Print**, set **Print Options** as below, the listing file name is defined as **Framework_SpectFat_Piles_Report**.



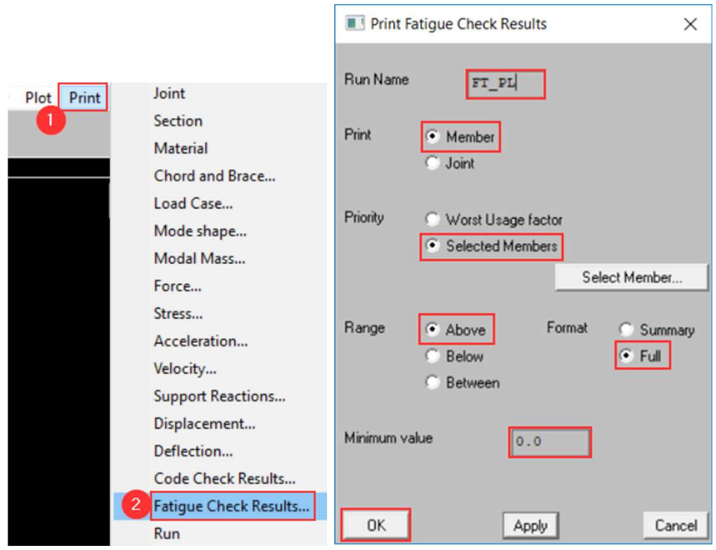
4.7.4 Executing Fatigue Analysis

The fatigue analysis for butt-welds on piles can now be run.

- Go to **Run** > **Fatigue Check**, name the run as **FT_PL**, press **OK** to run the analysis.



- After the analysis completed, go to **Print** > **Fatigue Check Results**, select the analysis results to be printed into the file.



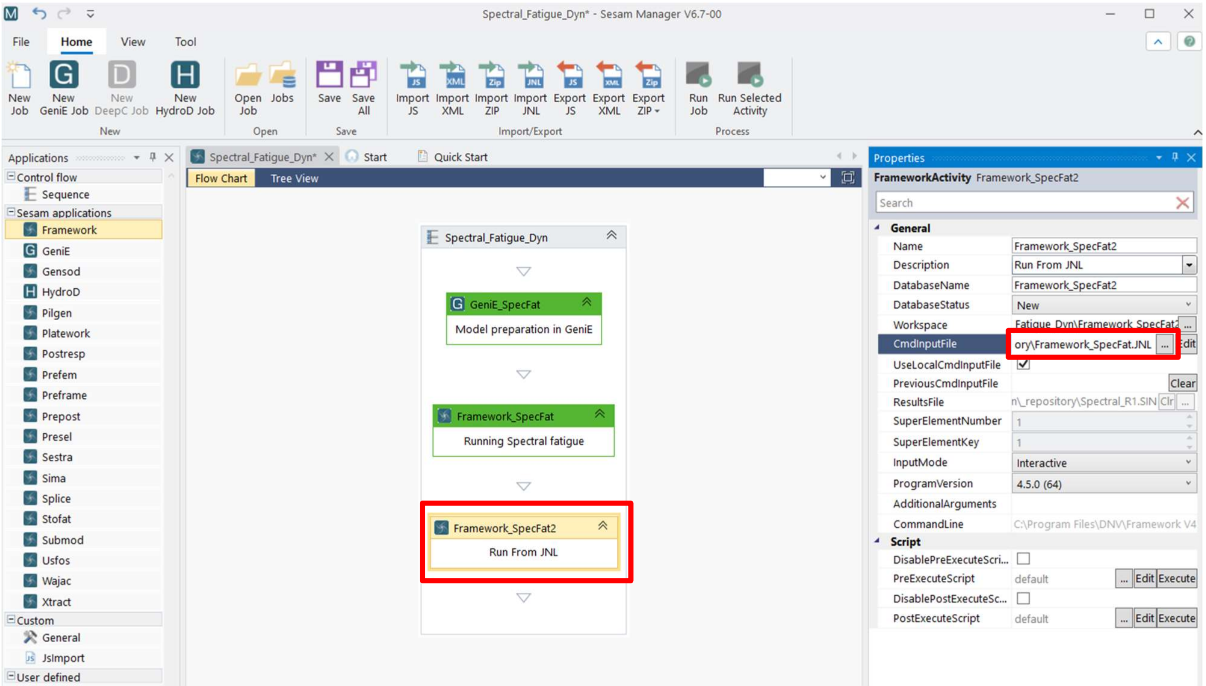
The fatigue analysis results for pile butt-welds are listed in the file located in **Framework_SpecFat** activity folder.

```
                                    SPECTRAL         fatigue check results
                                    Run:      Superelement:     Loadset:
                                    FT_PL     JACKET            LOADS
                                    Priority....: Selected Members
                                    Usage factor: Above    0.00                                          SUB PAGE:     2

Member       Type     Joint/Po Outcome   Damage     Life   WeldSide         Hot SCFrule    SCFax    SCFipb SCFopb SNcurve
             SctNam                                Alpha   Symmet        DiaBra  ThiBra      Gap    ThiFac   QR      Cycles
                                                   Theta   Jtype         DiaCho  ThiCho   LenCho    FixCho SCFaxC      SCFaxS
----------------------------------------------------------------------------------------------------------------------------------
PILE1        PIPE     763                7.12E-01 3.51E+02 BOTH-SIDE          4 GLOBAL     1.250    1.250  1.250 DNV2010_T
             PILESCT                                0.000  UNIFORM     1.70E+00   0.080 0.00E+00    1.257  1.000     1.54E+09
                                                    0.000              0.00E+00   0.000 0.00E+00    1.000  1.250        1.250
                      0.20               3.05E-02 8.20E+03 ROOTSectP         16 BUTT2020   1.250    1.250  1.250 DNV2010_F3-AIR
                                                    0.000  0.15TS      1.70E+00   0.080 4.00E-03    1.337  1.000     1.56E+09
                                                    0.000              0.00E+00   0.000 8.00E-02    1.000  1.250        1.250
                      0.20               2.18E-03 1.15E+05 PILESCT           16 BUTT2020   1.250    1.250  1.250 DNV2010_D
                                                    0.000  0.15TS      1.70E+00   0.080 0.00E+00    1.262  1.000     1.56E+09
                                                    0.000              0.00E+00   0.000 8.00E-02    1.000  1.250        1.250
             PIPE     0.20               5.06E-03 4.94E+04 PILEIN            16 BUTT2020   1.250    1.250  1.250 DNV2010_D
             PILEIN                                 0.000  0.15TS      1.70E+00   0.060 0.00E+00    1.191  1.000     1.56E+09
                                                    0.000              0.00E+00   0.000 8.00E-02    1.000  1.250        1.250
                      0.20               6.40E-02 3.90E+03 ROOTSectP         16 BUTT2020   1.250    1.250  1.250 DNV2010_F3-AIR
                                                    0.000  0.15TS      1.70E+00   0.060 4.00E-03    1.245  1.000     1.56E+09
                                                    0.000              0.00E+00   0.000 8.00E-02    1.000  1.250        1.250
                      0.98               4.88E-10 5.13E+11 ROOTSectP         19 BUTT2020   1.412    1.412  1.412 DNV2010_F3-AIR
                                                    0.000  0.15TS      1.70E+00   0.060 1.40E-02    1.245  1.000     1.56E+09
                                                    0.000              0.00E+00   0.000 8.00E-02    1.000  1.412        1.412
                      0.98               1.00E-10 2.50E+12 PILEIN            16 BUTT2020   1.250    1.250  1.250 DNV2010_D
                                                    0.000  0.15TS      1.70E+00   0.060 -9.00E-03   1.191  1.000     1.56E+09
                                                    0.000              0.00E+00   0.000 8.00E-02    1.000  1.250        1.250
             PIPE     0.98               1.00E-10 2.50E+12 PILESCT           19 BUTT2020   1.250    1.250  1.250 DNV2010_D
             PILESCT                                0.000  0.15TS      1.70E+00   0.080 -9.00E-03   1.262  1.000     1.56E+09
                                                    0.000              0.00E+00   0.000 8.00E-02    1.000  1.250        1.250
                      0.98               1.37E-10 1.82E+12 ROOTSectP         19 BUTT2020   1.412    1.412  1.412 DNV2010_F3-AIR
                                                    0.000  0.15TS      1.70E+00   0.080 1.40E-02    1.337  1.000     1.56E+09
                                                    0.000              0.00E+00   0.000 8.00E-02    1.000  1.412        1.412
                      724                1.00E-10 2.50E+12 BOTH-SIDE         16 GLOBAL     1.250    1.250  1.250 DNV2010_T
                                                    0.000  UNIFORM     1.70E+00   0.080 0.00E+00    1.257  1.000     1.56E+09
                                                    0.000              0.00E+00   0.000 0.00E+00    1.000  1.250        1.250
```

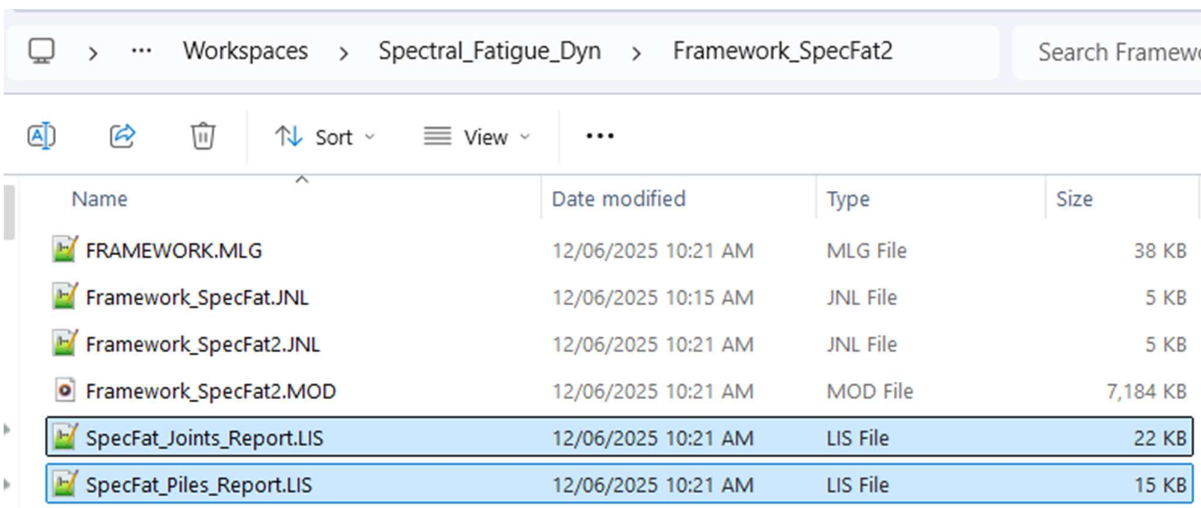
NOTE: By default S-N curve DNV2010_F3-AIR is assigned to the butt-weld root position.

4.8 Framework Analysis Using Manually Created Framework.jnl file

After the above analysis is finished, a journal file, **Framework_SpecFat.JNL** file is created in the Framework folder. Copy the file and paste it into **_repository** folder. Drag a new Framework activity into the work area, and name it as **Framework_SpecFat_2**. Choose the file **Framework_SpecFat.JNL** from **_repository** folder as **CmdInputFile**.



RMB click Framework activity and run it. The same analysis is performed, and the same listing files will be generated in the analysis folder.



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.