DNV

WHEN TRUST MATTERS

SESAM EXAMPLE
Static Spectral Fatigue Analysis

Static Spectral Fatigue Analysis

Date: July 2025

Prepared by DNV – Digital Solutions

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

E-mail sales: software@dnv.com

Table of contents

1 INTRODUCTION

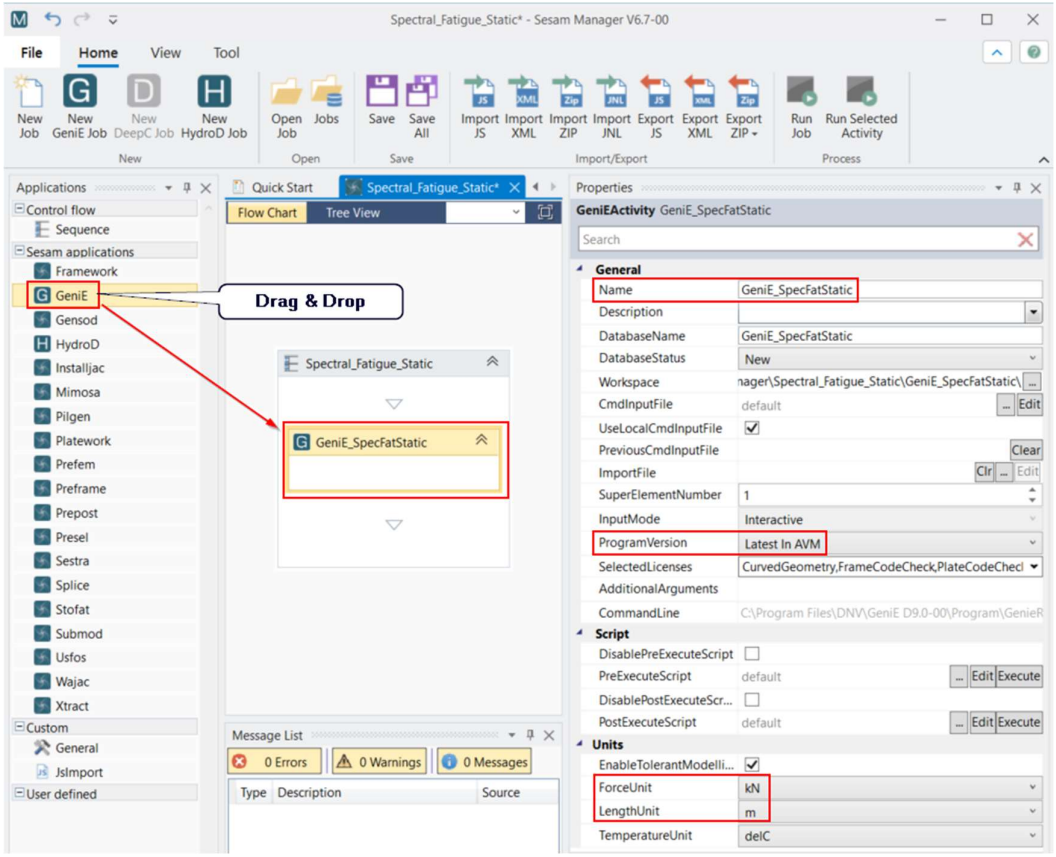
This example goes through steps to perform spectral fatigue analysis based on static analysis for an offshore fixed platform using Sesam software suite. The model file used in this analysis is the model from Inplace analysis with pile foundation.

2 MODEL MODIFICATION

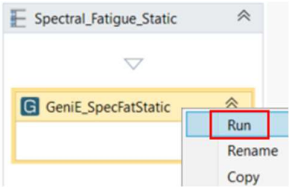
The model used to perform the Inplace analysis will be modified to include the deterministic wave load approach in Wajac, pile soil analysis in Sestra and Splice with the spectral fatigue approach in Framework.

2.1 Importing Model from Inplace Analysis

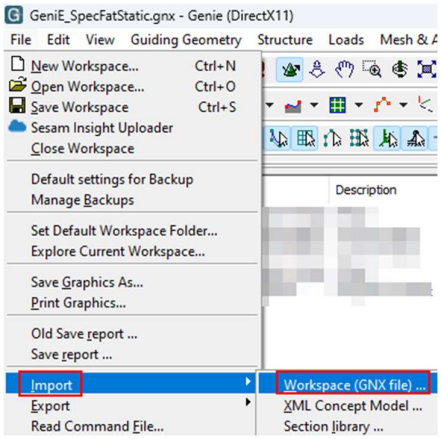
Open Sesam Manager, name it as **Spectral_Fatigue_Static**. Then drag GeniE into the work area, and name is as **GeniE_SpecFatStatic**. Change GeniE units to kN, m, and delC;

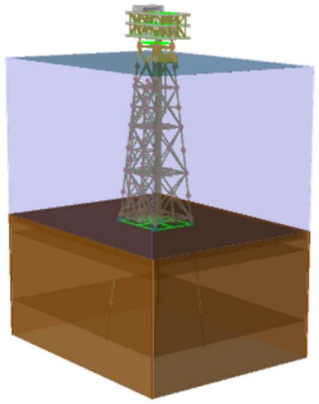


To launch GeniE, just RMB on **GeniE_ SpecFatStatic** and click **Run**. A new GeniE workspace is now open.



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.



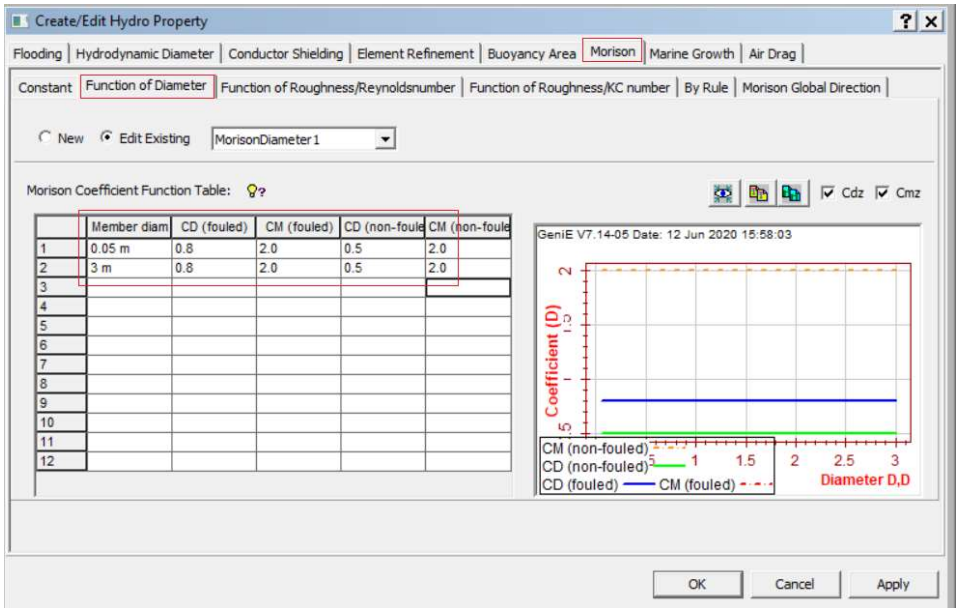


2.2 Revise the Model

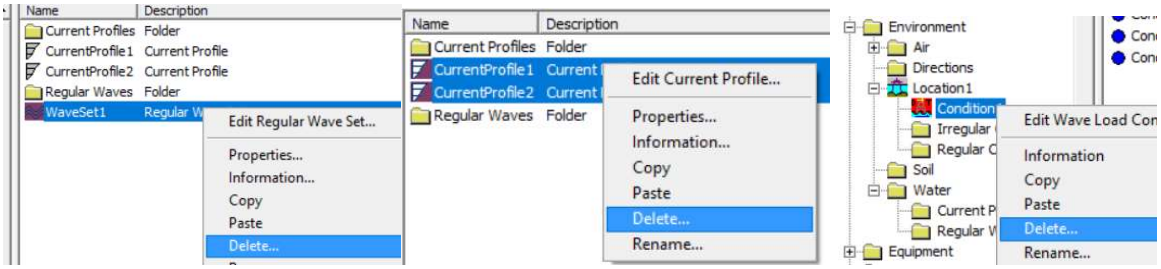
The model file **Model_SpectralFatigue_Start.gnx** is used for the inplace analysis. We need to revise it to make it suitable for the static spectral fatigue analysis.

2.2.1 Revise Hydro Properties and Delete Waves/Currents

The existing Morison coefficients were input for the inplace analysis. They need to be changed as below for a fatigue analysis.

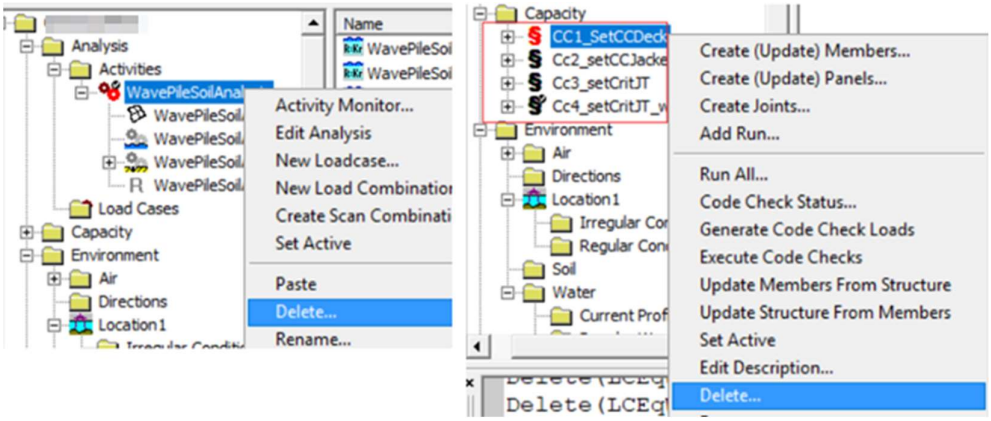


Delete the existing waves, currents, and the condition under the Location.



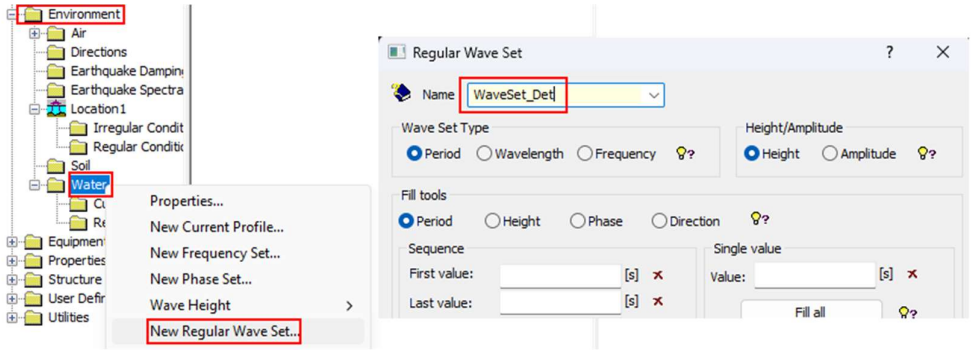
2.2.2 Delete Analysis and Capacity Models

Delete the existing analysis **WavePileSoilAnalysis** and all capacity models (if any).



2.3 Waves and Wave Condition

Input determanistic wave. To create the waves, go to **Environment** > **Water**, RMB click, select **New Regular Wave Set**. Name the wave set as **WaveSet_Det**.



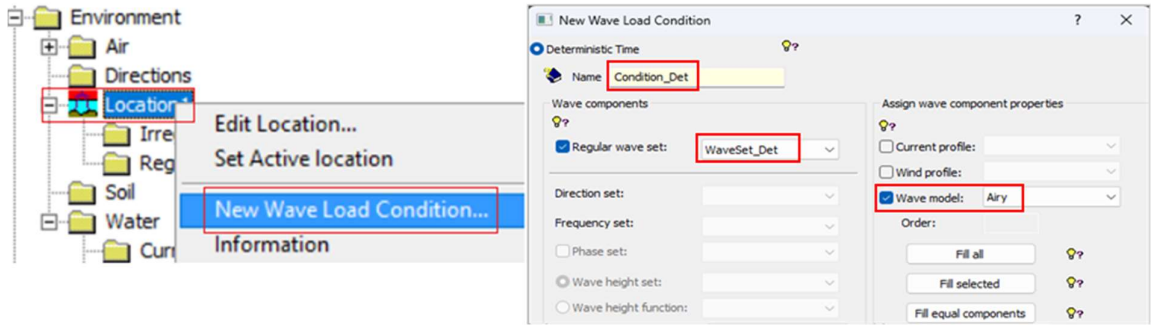
Add at least 15 lines by using ⮟ key on the keyboard. Enter five waves for each direction (0 deg, 52 deg, and 90 deg).

Specify value:

	Period [s]	Height [m]	Phase [deg]	Direction [deg]
1	5.66	2.5	0	0
2	8.004	5	0	0
3	9.803	7.5	0	0
4	11.32	10	0	0
5	12.4	12	0	0
6	5.66	2.5	0	52
7	8.004	5	0	52
8	9.803	7.5	0	52
9	11.32	10	0	52
10	12.4	12	0	52
11	5.66	2.5	0	90
12	8.004	5	0	90
13	9.803	7.5	0	90
14	11.32	10	0	90
15	12.4	12	0	90

Clear table | Remove | Insert | OK | Cancel | Apply

Create **Condition_Det** under Location1, set **WaveSet_Det** as the **Regular wave set**, and choose wave model as **Airy**.

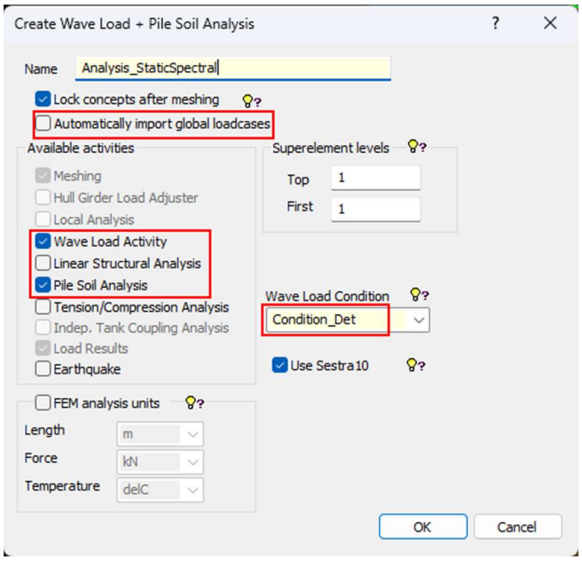


3 STATIC ANALYSIS – GENIE

A static analysis is created and performed in GeniE. The analysis includes the Wajac wave load calculations and the Splice/Sestra static analysis.

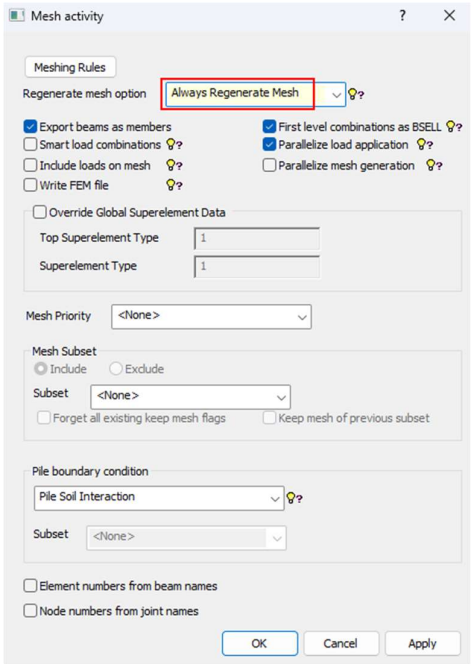
3.1 Creating Analysis

A static pile-soil analysis with wave load condition of **Condition_Det** is created. Uncheck the option **Automatically import global loadcases** to exclude all user defined load cases.



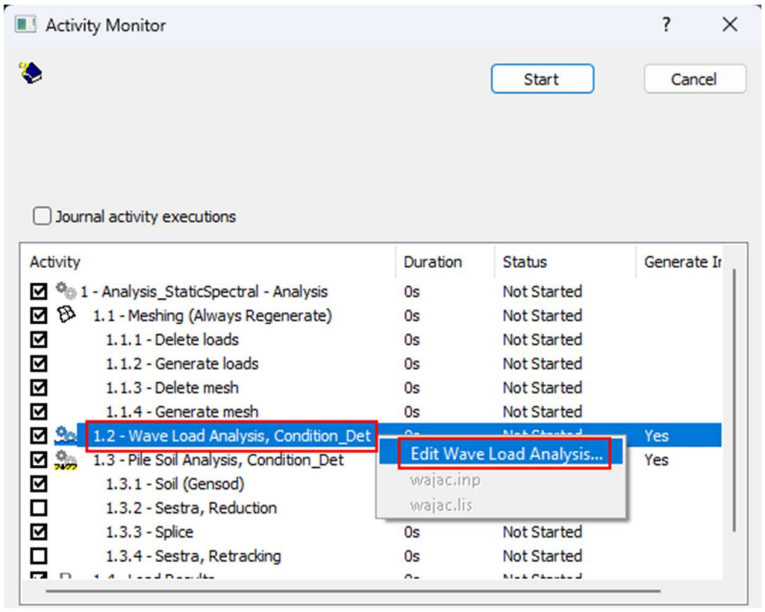
3.2 Meshing Activity

The following meshing options are selected.



3.3 Editing Wave Loads

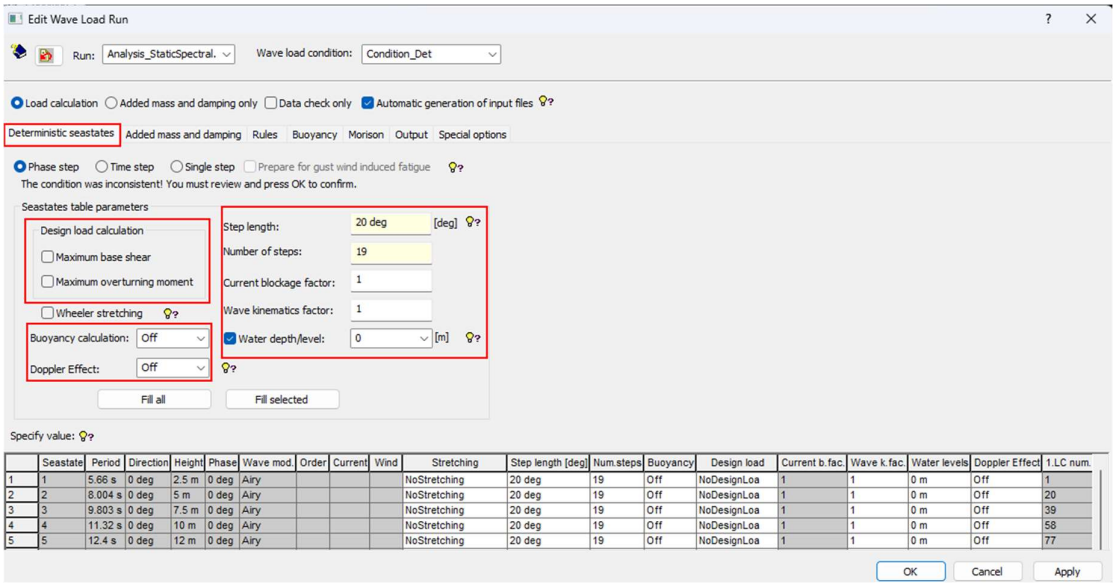
RMB click **1.2 – Wave Load** Analysis, select **Edit Wave Load Analysis**.



- **Deterministic seastates** tab

  When a wave is passing through the structure, the wave step size is selected as 20 degrees and the total number of steps is 19. The **Buoyancy calculation** and **Doppler Effect** are turned off. The design load criteria is **NoDesignLoads**. Wajac will calculate the wave load for each wave step.

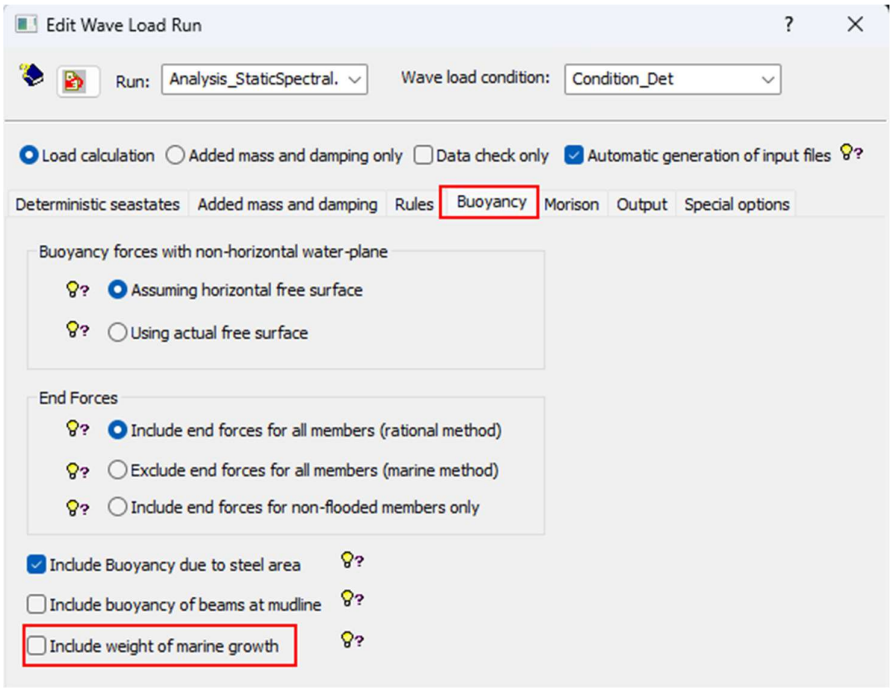
  Click **Fill all** to implement the selected options, and Click **Apply** to save the inputs and keep the dialog open.



Note: For a static analysis the number of wave steps can be 18 as well.

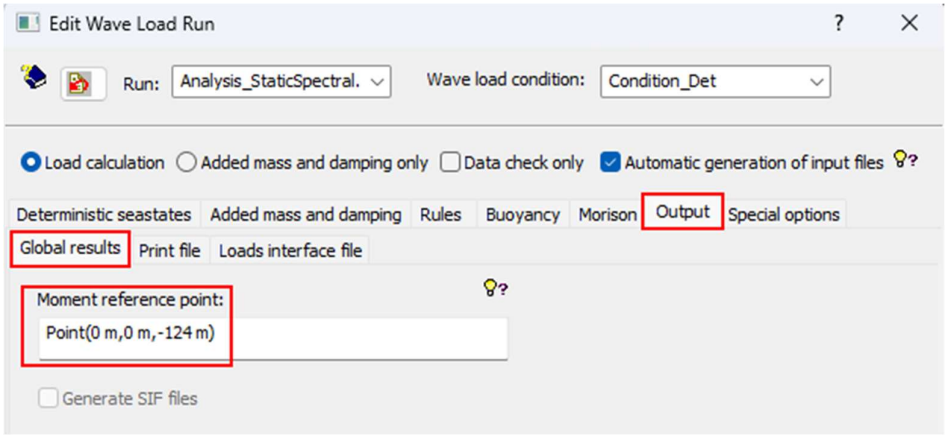
- **Buoyancy** tab

  The option **Include weight of marine growth** is unchecked. Click **Apply** to save the change and keep the dialog open.

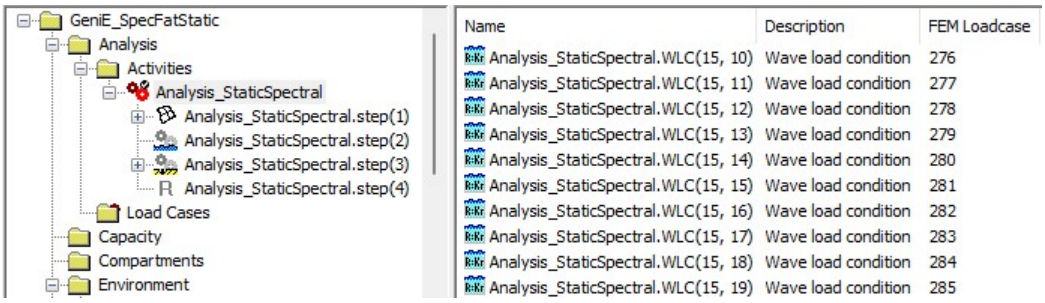


- **Output** tab

  The center point at the mudline is selected as the **Moment reference point**. Click **Apply** to save the input.



Finally, click **OK** to close the dialog box. Total 285 wave load cases are created and displayed in the analysis folder.



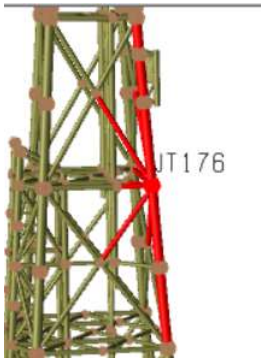
3.4 Editing Splice Soil Analysis

The default options are used in the Splice Soil Analysis. Splice soil analysis analyzes the structure for all wave load cases and generate beam forces and moments.

3.5 Creating a Set for Fatigue Damage Calculation

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

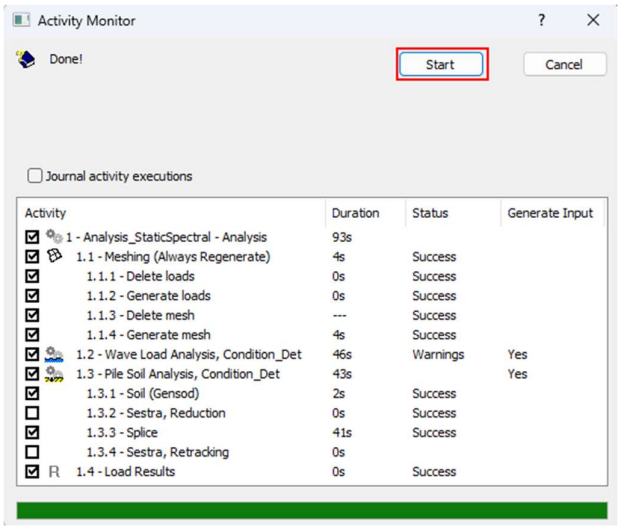
- Select **JT176** and all connected beams, see right. Create a set and name it as **Set_Fat**. This set will be included in the fatigue analysis in Framework.



3.6 Executing Analysis

Perform the analysis by clicking **Start** on **Activity Monitor** dialog box.

The analysis results are stored in the result file **_R1.SIN**.



Save the model and export it to **_repository** folder as **Model_SpectralFatigue_Done.gnx** Export the results SIN file to **_repository** folder with the name of **StaticSpectral_R1.SIN**.

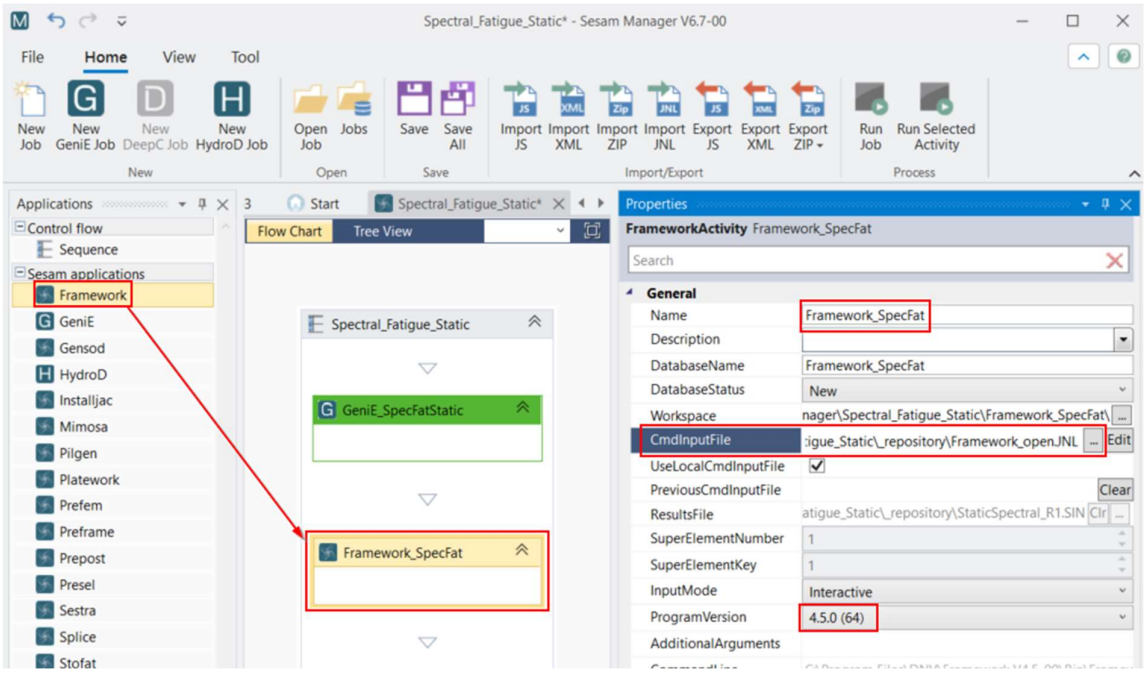
4 SPECTRAL FATIGUE ANALYSIS – FRAMEWORK

With the results in place, the static 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, right click Framework activity to run Framework.

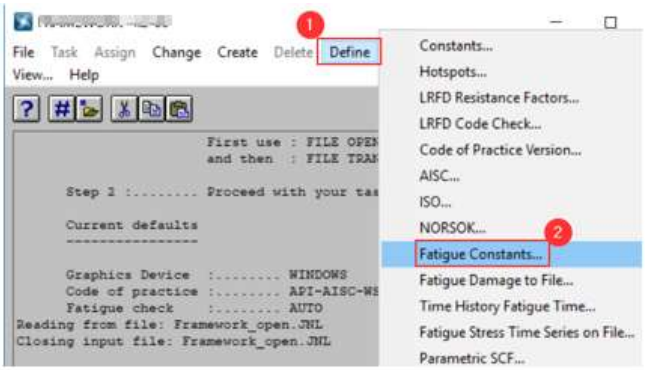


4.1 General Input

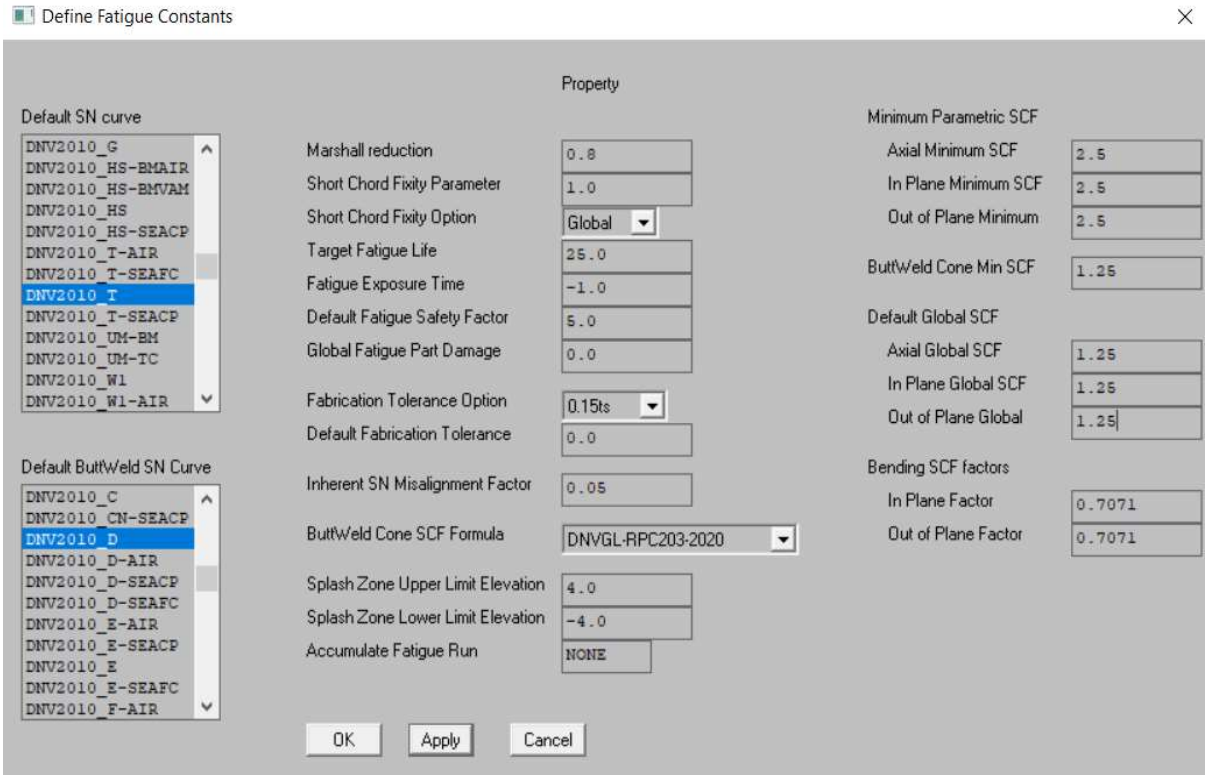
Before import the result file, global fatigue parameters need to be defined in a new Framework workspace.

4.1.1 Fatigue Constants and Global Settings

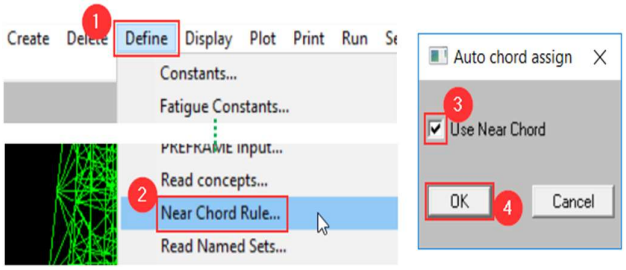
- Go to **Define** > **Fatigue Constants** to open the dialog for inputting the global fatigue check options.



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

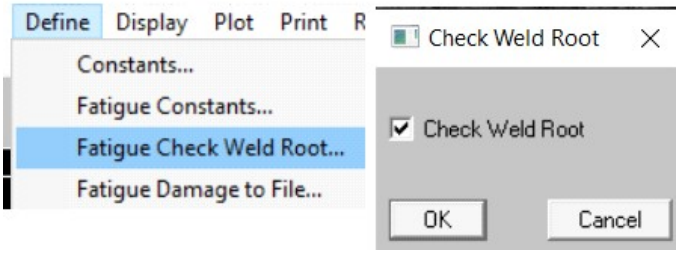


- Go to **Define** > **Near Chord Rule** and enable the checkbox **Use Near Chord**.



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.



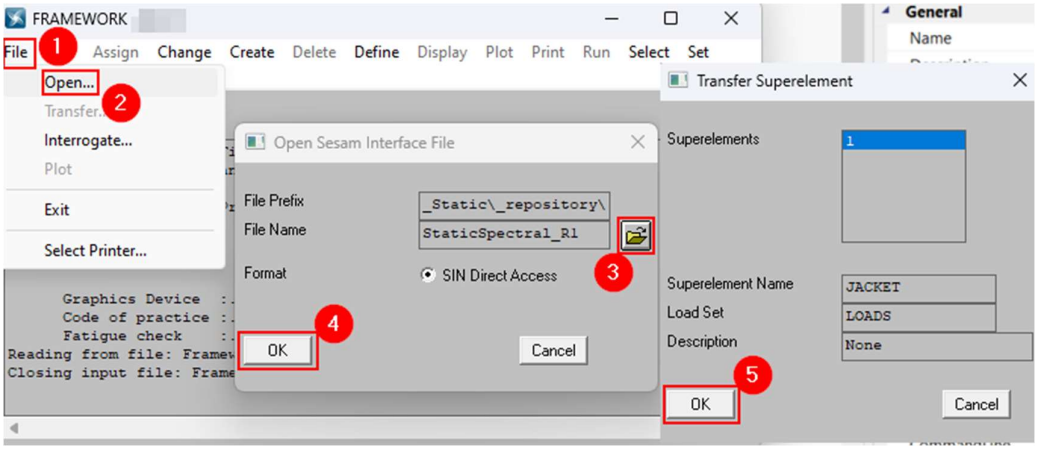
Note: The commands **DEFINE NEAR-CHOR-RULE** and **DEFINE FATIGUE-CHECK-WELD-ROOT** must be issued before the result file is read in.

4.1.2 Importing Result File

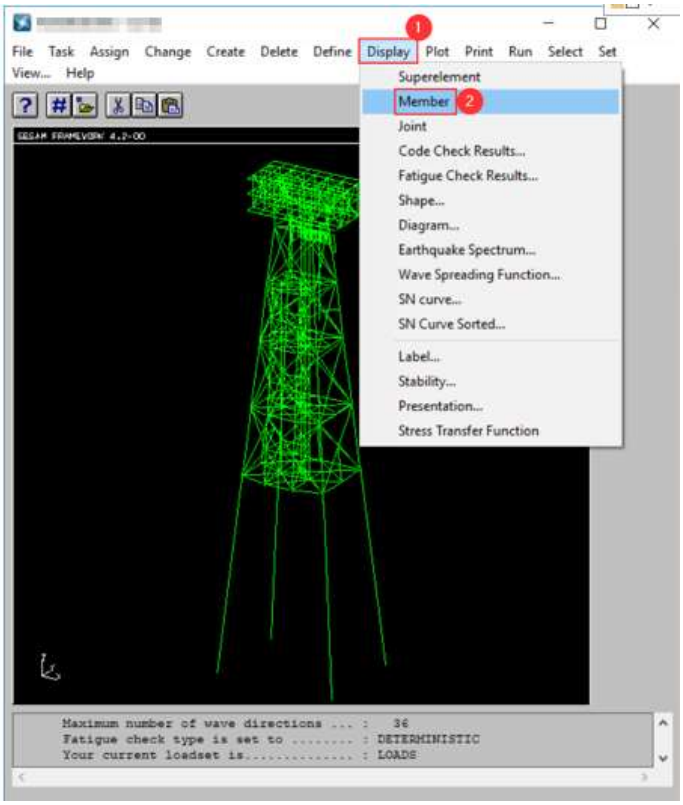
Now the result file can be read into Framework.

- Go to **File** > **Open**. Click the browse folder button and locate the **StaticSpectral_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.

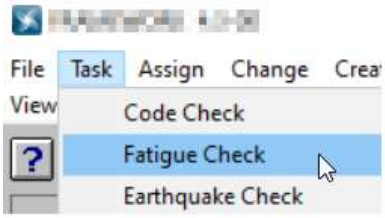


- After the file is read in, click **Display** > **Member** to get the structure displayed.

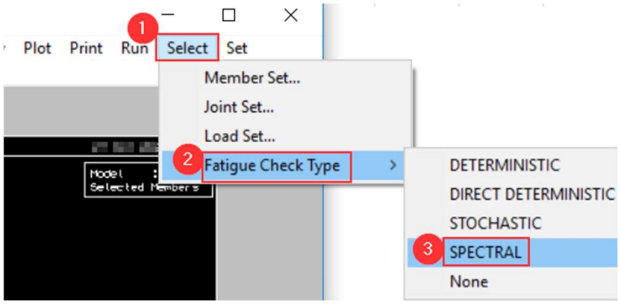


4.1.3 Specifying Task and Analysis

- 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.

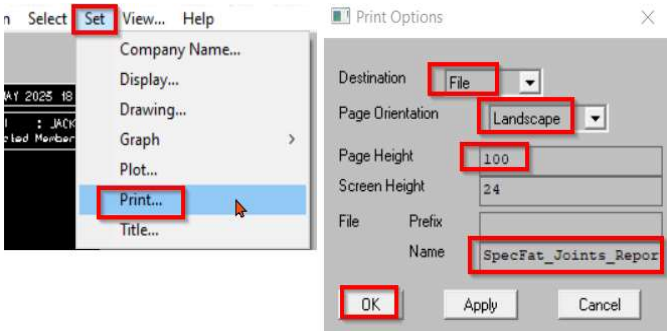


- Go to **Select** > **Fatigue Check Type** > **Spectral** to define that a spectral fatigue analysis will be run.



4.1.4 Setting Up Print Output

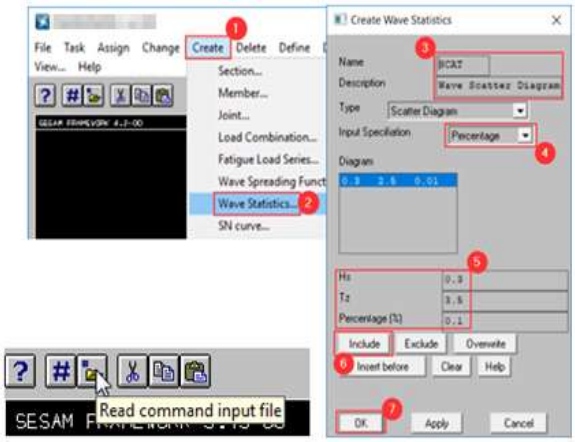
- Go to **Set** > **Print** to set **Print Options** as shown in the image, define the file name as **Framework_SpecFat_Joints_Report**



4.2 Fatigue Loads – Wave Scatter Diagrams

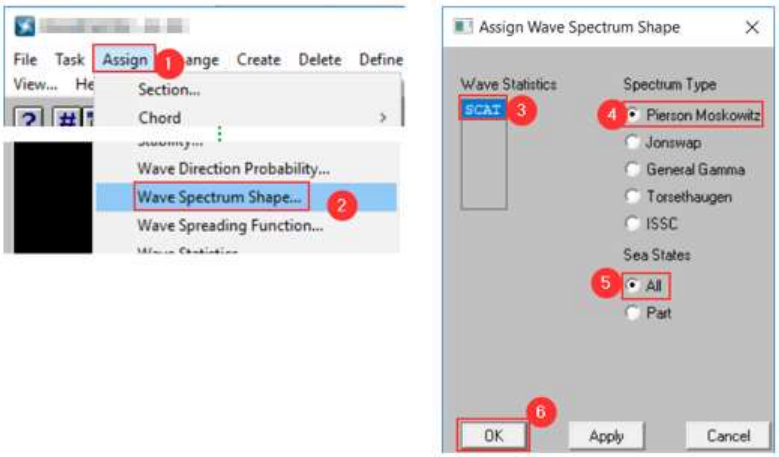
Next, the environmental data will be specified and assigned to each wave direction.

- 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.

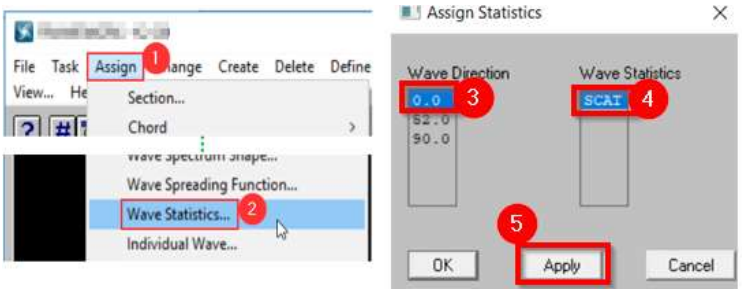


NOTE: Make sure that there are no spaces in the path to the input file, otherwise Framework will fail to read it in.

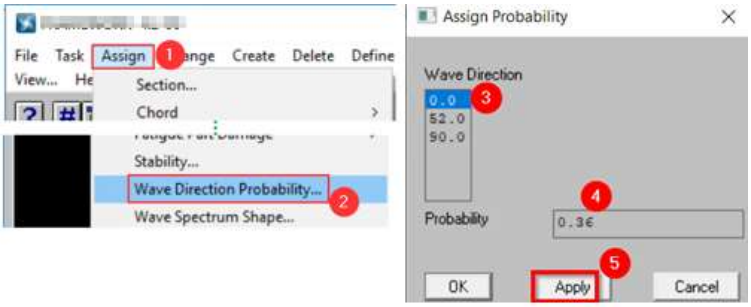
- Assign the wave spectrum shape to each wave scatter diagram via **Assign** > **Wave Spectrum Shape** as **Pierson-Moskowitz** spectrum.



- 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.

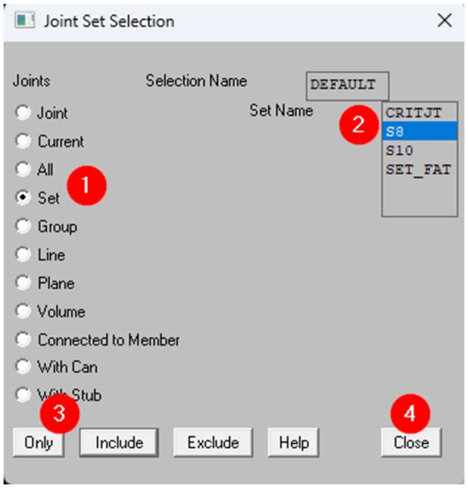


4.3 Joint and Member Selection

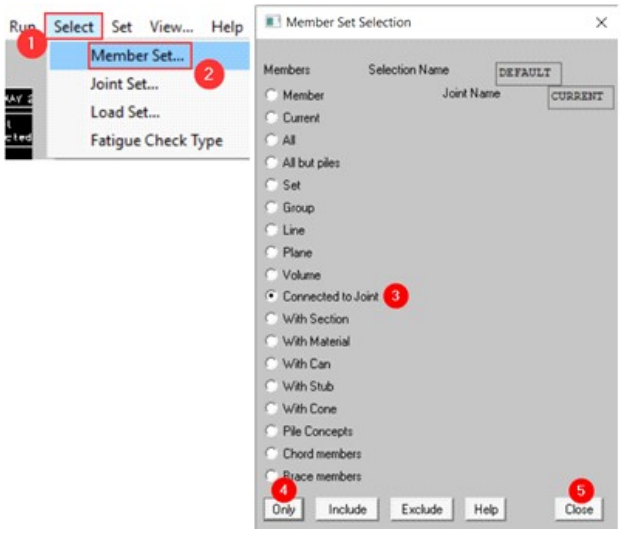
The joints and members included in the fatigue analysis can be selected before the analysis is performed. If the set name of joints and members are longer than 8 characters, Framework will rename them. The naming map is reported in Framework.MLG file.

```
* SET       8 has name longer than       8 characters.
* Name is CritJT_withBraces
* Name S8       will be used
```

- Go to **Select** > **Joint Set** > **Set**, select the set **S8**, then click **Only** and **Close**.



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

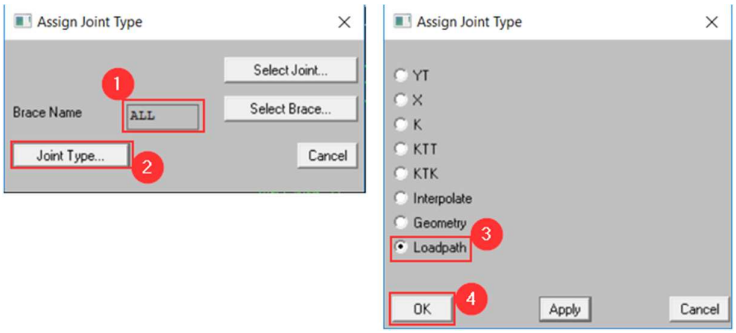


4.4 SCF Related Inputs

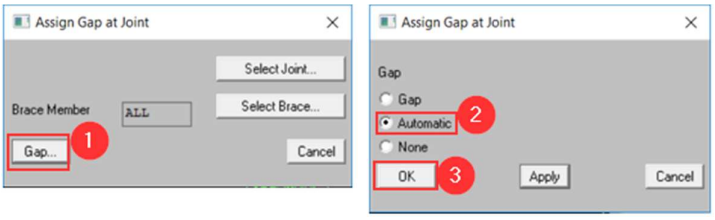
In a fatigue analysis, user needs to define SCF calculation rules for tubular connections and for butt-welds. If needed local SCFs can be defined for some selection tubular connections and the selected butt-weld positions.

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

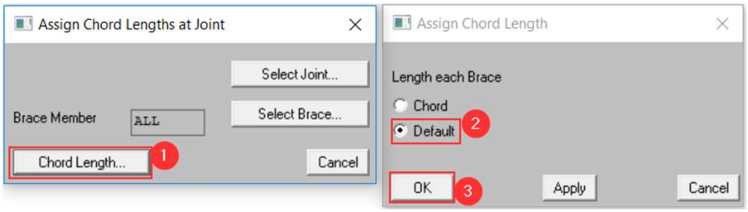
- Go to **Assign** > **Joint Type**, click **Joint Type** and select **Loadpath**. This will be assigned to the selected joint set S9 and all braces connected to the joints. The defined joint classification rule will be used in SCF calculations.



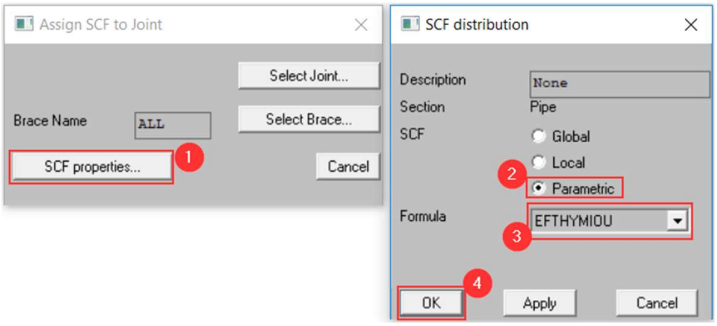
- Go to **Assign** > **Joint Gap** and make sure that gaps are automatically computed based on the modelled gaps for all selected joints.



- 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.

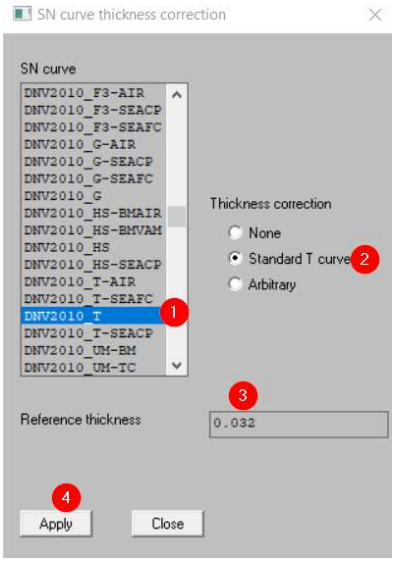


4.5 S-N Curve Related Inputs

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.

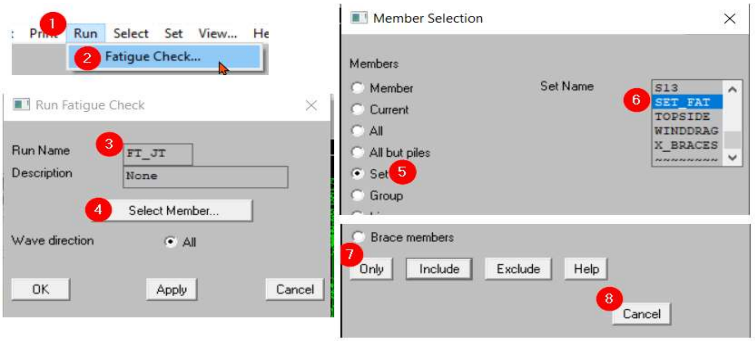
- Go to **Assign** > **Thickness Correction**, select the curve **DNV2010_T** and make sure a **Standard T-curve** is used for the thickness correction with the reference thickness set to 32 mm (i.e. 0.032 m). Press **Apply** and **Close**.



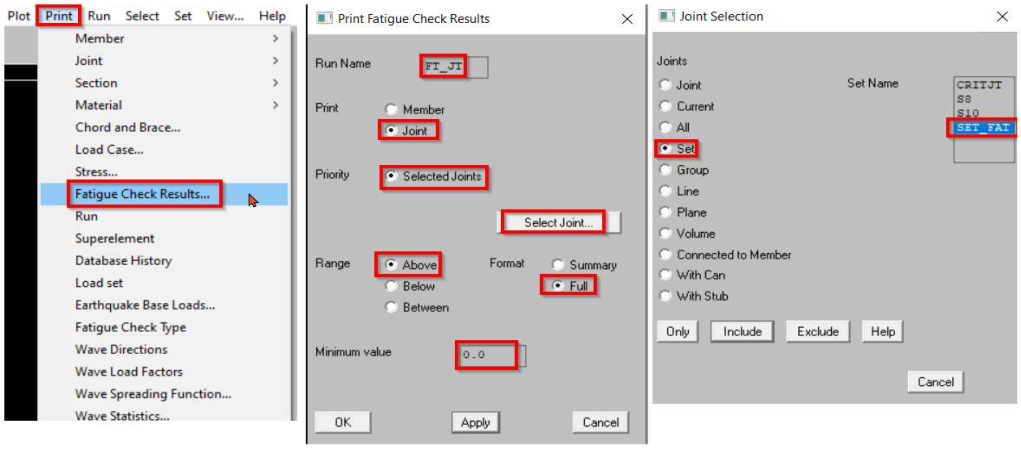
4.6 Fatigue Analysis for Tubular Connections

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 located 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                   3.48E-01  3.60E+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      5.94E+08
                                            83.385  YT         1.90E+00   0.070    6.04E+01 1.000  6.053        4.810
                                 1.22E-01  1.03E+03 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      6.00E+08
                                            83.385  YT         1.90E+00   0.070    6.04E+01 1.000  4.257        6.608
                                 1.91E-01  6.54E+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      6.11E+08
                                            83.385  YT         1.90E+00   0.070    6.04E+01 1.000  3.632        2.886
                                 2.40E-01  5.21E+02 BRACE-ROO         7 EFTHYMIOU   2.554    2.500  2.500 DNV2010_F3-AIR
                                           180.158  CROWN-SAD  8.00E-01   0.025    0.00E+00 1.000  1.000      6.22E+08
                                            83.385  YT         1.90E+00   0.070    6.04E+01 1.000  2.554        3.965

         BM182                   1.23E-02  1.01E+04 BRACE-ROO         4 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      5.56E+08
                                            84.838  KTT/LPD    1.90E+00   0.070    6.04E+01 1.000  2.500        2.500
                                 1.30E-02  9.60E+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      5.86E+08
                                            84.838  KTT/LPD    1.90E+00   0.070    6.04E+01 1.000  2.500        2.500
                                 9.16E-03  1.36E+04 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      5.96E+08
                                            84.838  KTT/LPD    1.90E+00   0.070    6.04E+01 1.000  3.882        3.882
                                 2.40E-02  5.20E+03 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      5.93E+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

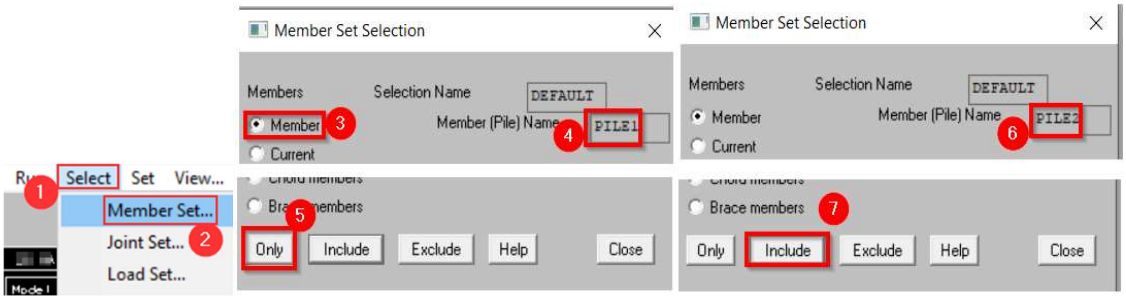
Member	Joint/Po	SecTy	PositionName	CoorX	CoorY	CoorZ	LocSpl	SNcurve
BM21	JT176	PIPE	CHORD-SIDE-0.0000	12.99	9.89	-19.00	SEA	DNV2010_T
	JT176	PIPE	BRACE-SIDE-0.0000	12.99	9.89	-19.00	SEA	DNV2010_T
	JT176	PIPE	CHORD-ROOT-0.0000	12.99	9.89	-19.00	SEA	DNV2010_F-AIR
	JT176	PIPE	BRACE-ROOT-0.0000	12.99	9.89	-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.

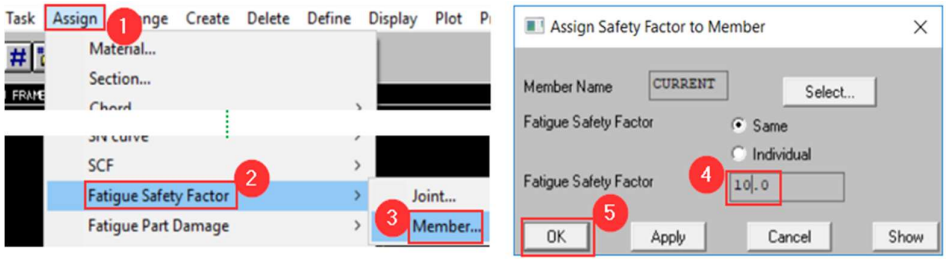
4.7.1 Pile Selection

- 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.



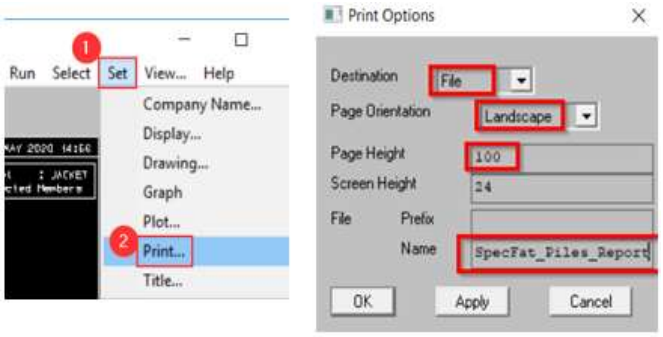
4.7.2 Pile Safety Factor

- 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

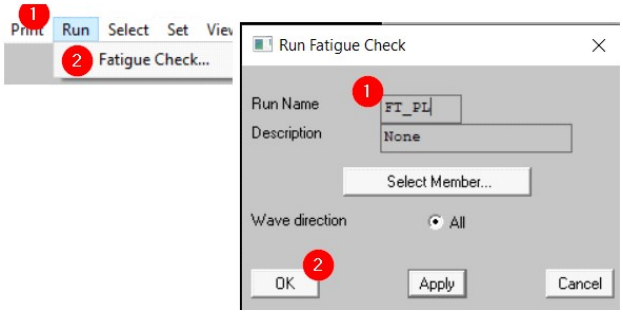
- Go to **Set** > **Print**, set **Print Options** as below, the listing file name is defined as **Framework_SpecFat_Piles_Report**.



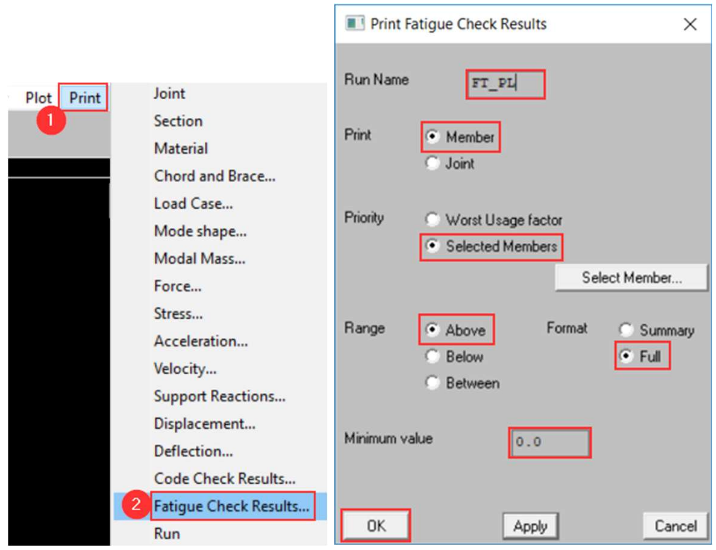
4.7.4 Executing Fatigue Analysis

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.

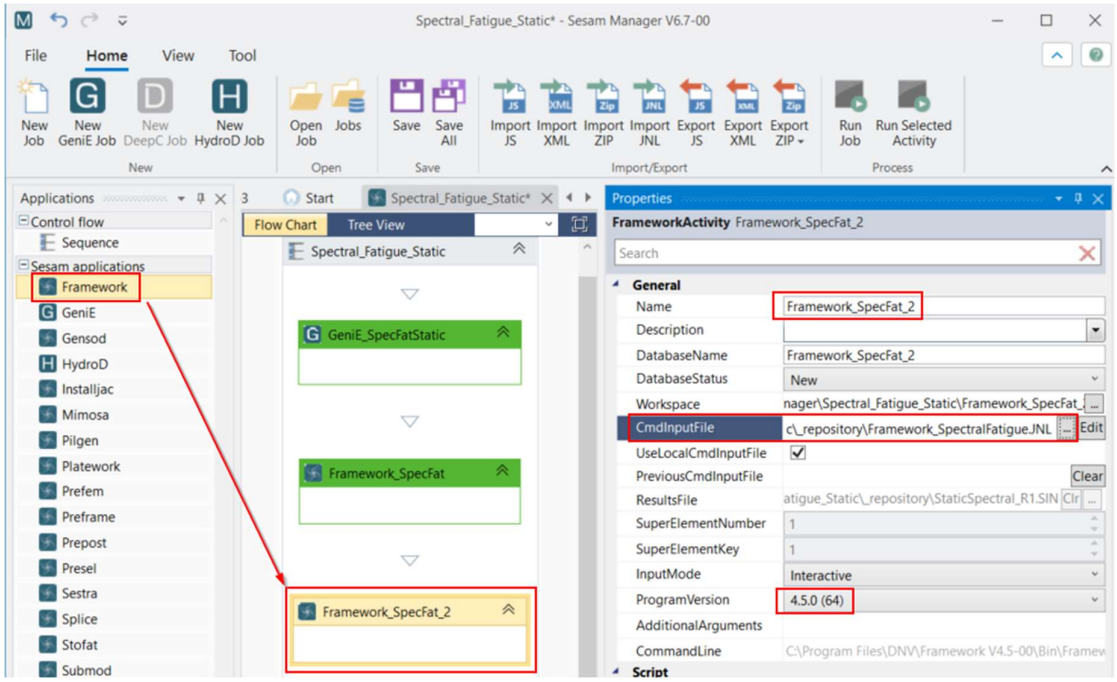
```
                         SPECTRAL          fatigue check results
                         Run:      Superelement:      Loadset:
                         FT_PL     JACKET             LOADS
                         Priority....: Selected Members
                         Usage factor: Above    0.00                                                  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                3.25E-02  7.69E+03 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     1.12E+09
                                                    0.000           0.00E+00   0.000 0.00E+00  1.000  1.250        1.250
                     0.20               1.29E-03  1.94E+05 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     1.13E+09
                                                    0.000           0.00E+00   0.000 8.00E-02  1.000  1.250        1.250
                     0.20               9.01E-05  2.78E+06 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     1.13E+09
                                                    0.000           0.00E+00   0.000 8.00E-02  1.000  1.250        1.250
            PIPE     0.20               2.09E-04  1.20E+06 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     1.13E+09
                                                    0.000           0.00E+00   0.000 8.00E-02  1.000  1.250        1.250
                     0.20               2.78E-03  8.99E+04 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     1.13E+09
                                                    0.000           0.00E+00   0.000 8.00E-02  1.000  1.250        1.250
                     0.98               1.00E-10  2.50E+12 ROOTSectP     16 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     1.13E+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        19 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     1.13E+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       10 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     1.13E+09
                                                    0.000           0.00E+00   0.000 8.00E-02  1.000  1.250        1.250
                     0.98               1.00E-10  2.50E+12 ROOTSectP     16 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     1.13E+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     13 GLOBAL    1.250    1.250  1.250 DNV2010_T
                                                    0.000 UNIFORM   1.70E+00   0.080 0.00E+00  1.257  1.000     1.13E+09
                                                    0.000           0.00E+00   0.000 0.00E+00  1.000  1.250        1.250
```

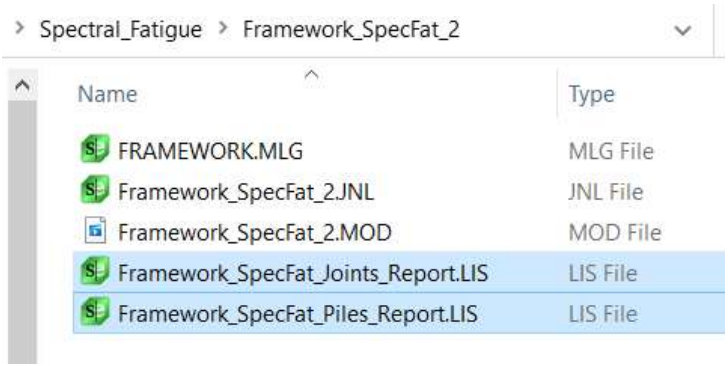
NOTE: By default S-N curve DNV2010_F3-AIR is assigned to the butt-weld root position.

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.