SESAM EXAMPLE

Jacket In-Place, Fatigue, Earthquake and Transportation Analyses in Sesam

Sesam Example

Jacket In-Place, Fatigue, Earthquake and Transportation Analyses in Sesam

Date: October 2022

Prepared by: Digital Solutions at DNV

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

E-mail sales: digital@dnv.com

Table of contents

1 Introduction

This document explains how to do various jacket analyses in Sesam, including in-place (ULS), free vibration, fatigue by alternative methods (FLS), earthquake and transportation analyses. Emphasis is put on steps demanding special attention, for example proceeding from static to dynamic analysis and converting a tubular joint from beam to shell model for shell fatigue analysis. The jacket model is shown in **Figure 1-1**.

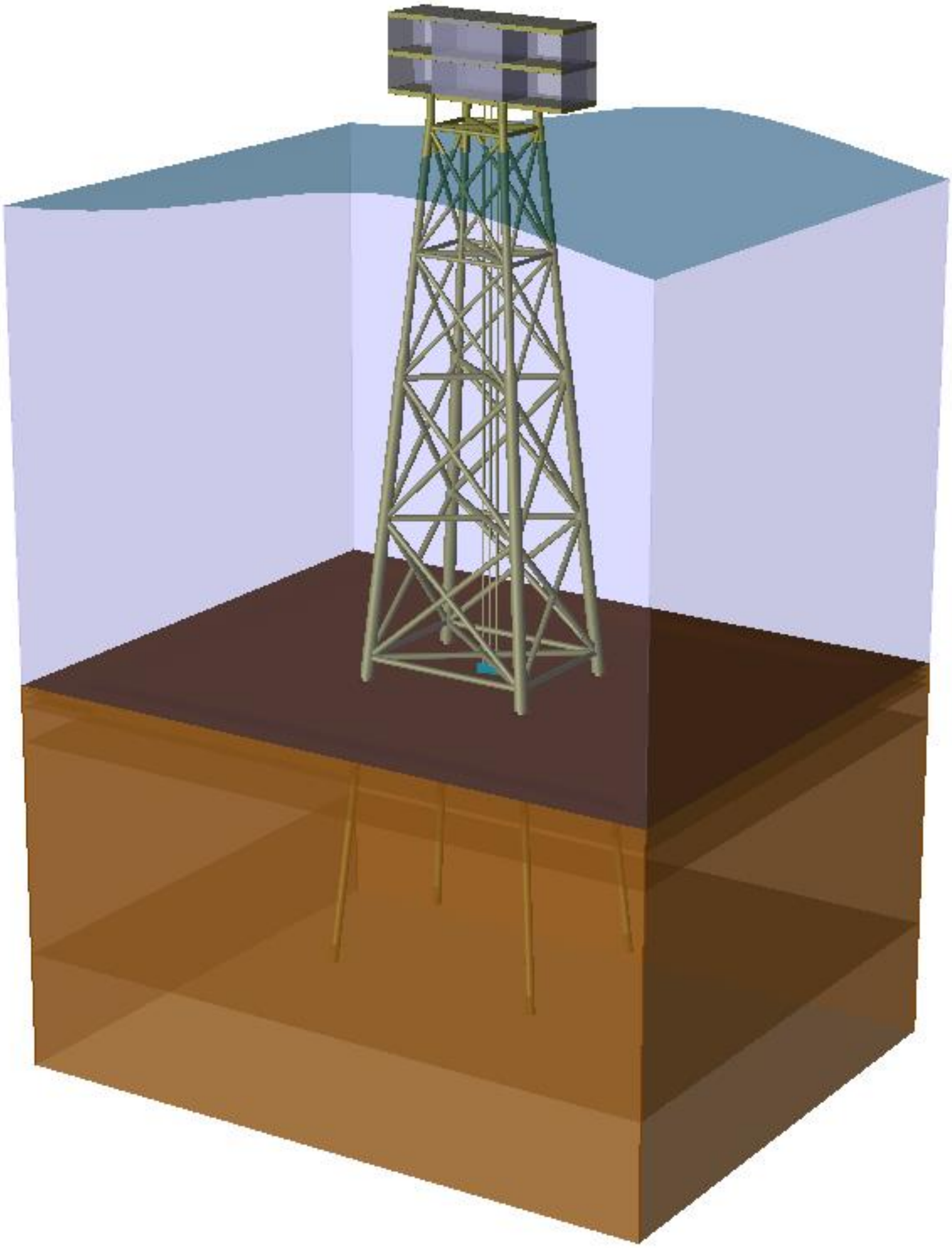


Figure 1-1 Four-legged jacket

This guide should, however, not be taken as a complete description of all necessary steps for the analyses included. Moreover, the model and other input is simplified and therefore in part somewhat unrealistic. This in turn causes the analysis results, the FLS results in particular, to be dubious.

The analysis workflow is controlled by Sesam Manager. The workflow is shown in so-called *Tree View* in **Figure 1-2**. Following activities for ULS, pile-soil-linearisation and free vibration analyses, there are sequences (sub-workflows) for:

- Deterministic fatigue
- Spectral fatigue, static analysis
- Spectral fatigue, dynamic analysis
- Spectral fatigue, equivalent static loads (ESL) analysis
- Stochastic fatigue of pure beam model
- Stochastic fatigue of a tubular joint converted from beam model to shell model
- Earthquake analysis
- Transportation analysis

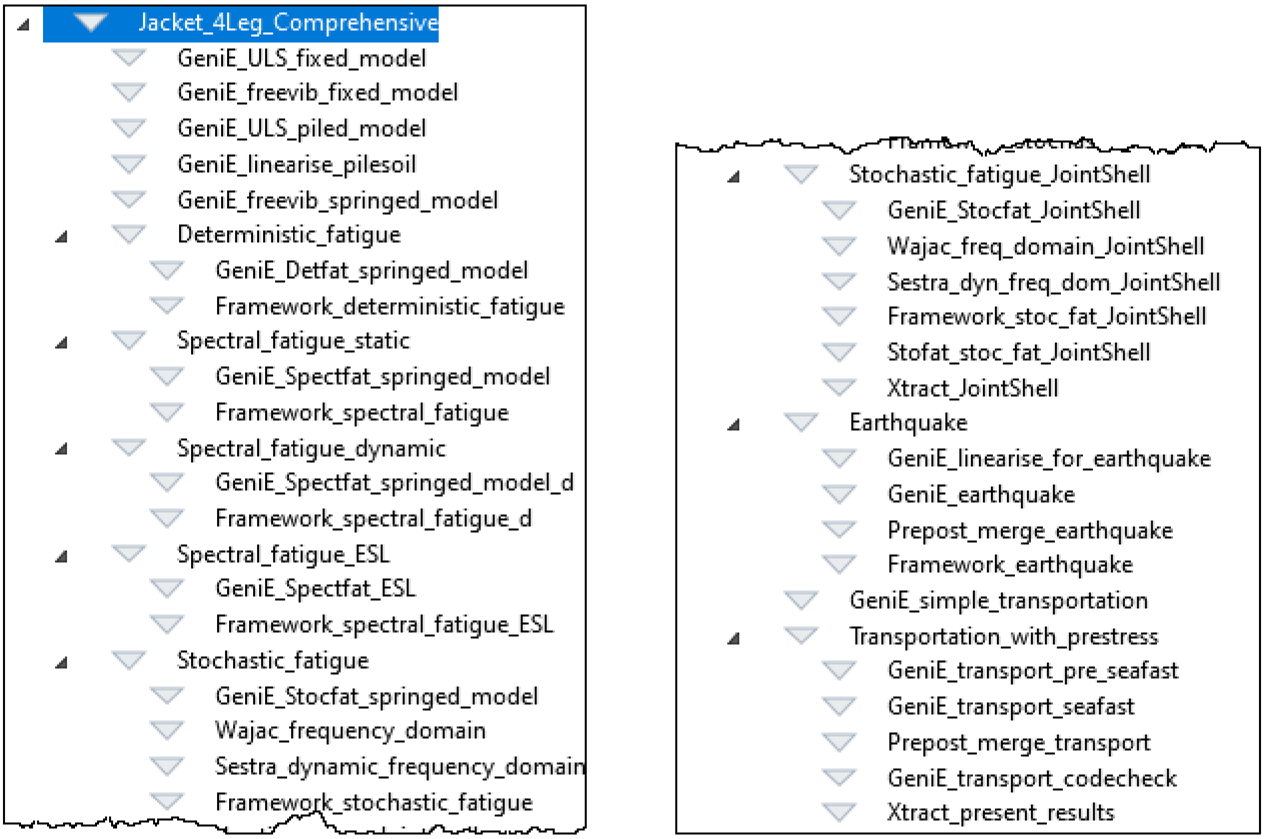


Figure 1-2 Sesam Manager workflow

The example is based on units kN and m. Acceleration of gravity is set to 9.80665.

This example is based on use of GeniE 8.4, Wajac 7.8, Sestra 10.16, Splice 8.0, Framework 4.3, Stofat 4.1-02, Xtract 6.0, Prepost 8.4 or later versions of these programs.

2 Static Analysis of Fixed Model

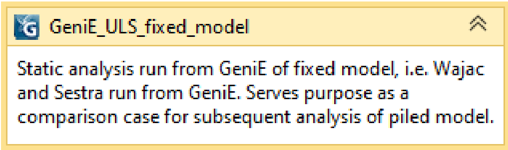


Figure 2-1 Activity for linear static analysis of fixed model in *Flow Chart* view

A linear static analysis including code checking is performed by the activity **GeniE_ULS_fixed_model**. The jacket is fixed at the leg bottom and subjected to wave and wind loads from north (270°), east (90°), south (180°) and west (0°). 0° is in positive X direction. A wave plus wind load analysis in Wajac and a linear static analysis in Sestra are run. Wajac and Sestra are run under the control of GeniE, the name of the GeniE analysis activity is WaveAnalysis. Alternatively, GeniE may be used merely as a modelling tool with Wajac and Sestra run under the control of Sesam Manager.

The purpose of this analysis is to serve as a basis of comparison for a subsequent non-linear structure-pile-soil interaction analysis. After starting this activity and completion of the execution, GeniE is left open and you may view the code check results and other results before exiting. Code check result *UfTot* for the worst case (*Worst Case (CC)*) is shown in Figure 2-2 below.

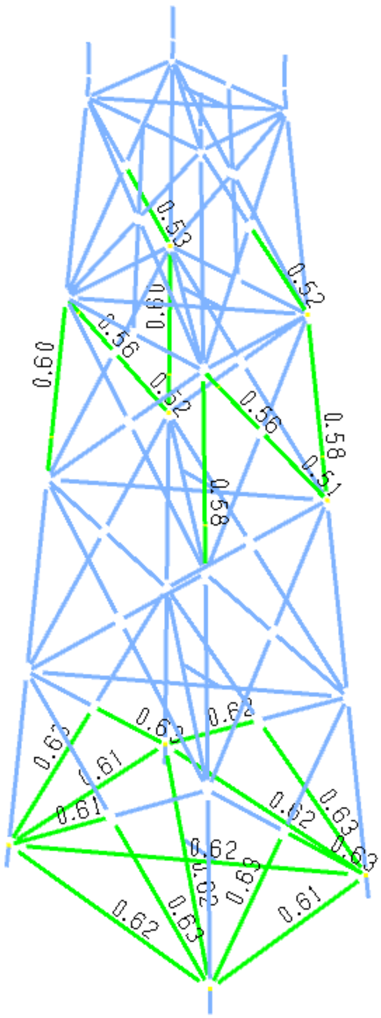


Figure 2-2 Code check result *UfTot* for *Worst Case (CC)* for fixed model

3 Free Vibration Analysis of Fixed Model

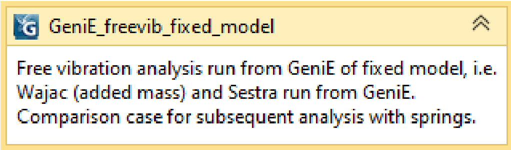


Figure 3-1 Activity for free vibration analysis of fixed model in *Flow Chart* view

A free vibration (eigenvalue) analysis is performed by the activity **GeniE_freevib_fixed_model**. The GeniE workspace of the previous activity, GeniE_ULS_fixed_model, is imported and modified. The mesh density for the topside is reduced and computation of added mass in Wajac and linear free vibration analysis in Sestra are set up. Wajac and Sestra are run under the control of GeniE, the name of the GeniE analysis activity is FreevibAnalysis. The purpose of the analysis is to serve as a basis of comparison for a subsequent free vibration analysis of a model with spring supports. The first five eigenperiods found are 2.281, 1.857, 1.171, 0.650 and 0.583 (reported in the Sestra.lis file).

4 Non-Linear Static Analysis of Piled Model

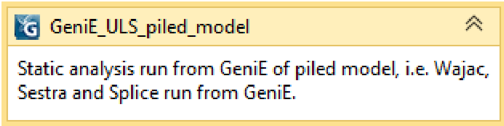


Figure 4-1 Activity for non-linear static analysis of piled model in *Flow Chart* view

A non-linear structure-pile-soil (in-place) analysis with the same loads as in section **2** is performed by the activity **GeniE_ULS_piled_model**. A model with piles and soil is created, a wave load analysis in Wajac is run, and a non-linear structure-pile-soil interaction analysis in Splice and Sestra is run, all under the control of GeniE, the analysis activity is named StructurePileSoilAnalysis:

- Gensod computes soil curves/stiffnesses,
- Splice controls the non-linear structure-pile-soil interaction analysis:
  - Sestra reduces the jacket, i.e., eliminates all nodes not connected to the piles,
  - Splice solves the pile-soil interaction non-linearly,
  - Sestra retracks the jacket, i.e., computes displacements in nodes throughout the model.

This is an ultimate limit state (ULS) analysis. You may view the code check results and other results before exiting GeniE. Code check result *UfTot* for the worst case (*Worst Case (CC)*) is shown in Figure 4-2 below.

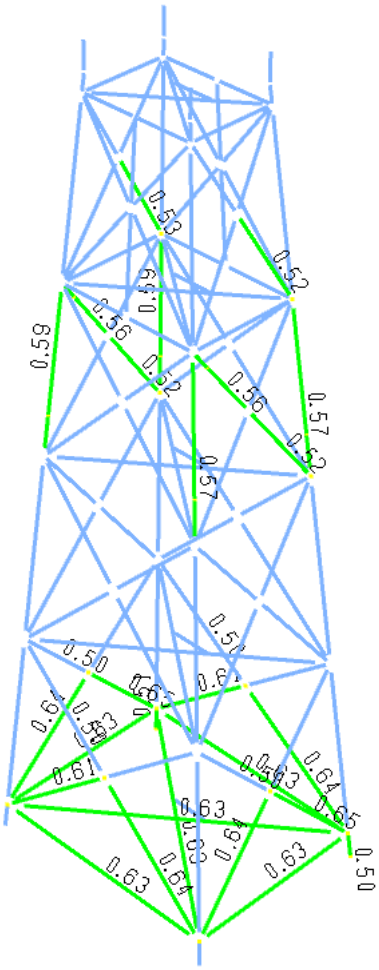


Figure 4-2 Code check result *UfTot* for *Worst Case (CC)* for piled model

5 Change the Model When Proceeding from Static ULS Analysis

Before proceeding with the fatigue limit state (FLS) analyses some changes to the model as used for the static ULS analysis must be made. The changes required are discussed below.

The non-linear pile-soil foundation should be replaced by linear spring-to-ground elements. Section **6** explains how to find appropriate linear spring stiffnesses. The motivation for this replacement is:

- Both the spectral and stochastic fatigue analyses, see section **8**, are spectral methods that are based on linearity.
- In a fatigue limit state (FLS) analysis the loads are generally small or moderate compared with the ULS loads. This means that the pile-soil foundation is utilized primarily in the linear range. For fatigue analysis of the jacket, the pile-soil foundation may therefore be replaced with linear spring-to-ground elements.
- Nevertheless, the equivalent static loads analysis, see section **12**, accounts for the non-linearity of the pile-soil foundation as follows: Run a dynamic analysis with spring-to-ground elements, compute equivalent static loads as the sum of the damping and mass related forces, and run a static analysis with pile-soil foundation, adding the equivalent static loads.

Dynamic structural analysis normally necessitates some additional changes to the model:

- In a dynamic analysis the model must be dynamically sound meaning that the stiffness must correspond with the mass on a detailed level, i.e., for each single dof contributing to the dynamic analysis, as well as overall. In a static analysis, soft parts, e.g., due to simplified modelling or low-quality mesh, may be accepted since unrealistically high static displacements in local areas may simply be neglected. In a dynamic analysis, however, incorrect representation of mass and stiffness (too high mass/stiffness quotient) for a detail may adversely affect the overall results. As a consequence, parts of the model may have to be modified, this could be both refinement and simplification of which the latter is normally preferred.
- Explicit loads (point-, line- and surface loads) representing equipment and other dead weights in a static analysis must be replaced by mass in a dynamic analysis.
    - In GeniE, proper modelling of equipment and other dead loads is done by use of either the ‘equipment feature’ or the ‘weight list feature’ rather than explicit loads. For a static analysis the equipment and weight list items will be converted into loads. For a dynamic analysis, however, the equipment and weight list items must be converted into mass. In the property dialog for the load case(s) containing equipment or weight list items select *Placed equipment* under *Convert to loadcase independent mass*. Use either the *Footprint-Mass*, *Beams-And-Mass* or *Vertical-Beams-And-Mass* representation. The *Eccentric-Mass* option is less suitable for structural dynamic analysis and should be avoided.
    - If the model has been imported from other software in which masses have been modelled as loads, or such modelling technique has been used in GeniE, then loads representing equipment and other dead loads must be converted to mass. This may be done by checking *Point and line loads* under *Convert to loadcase independent mass* in the property dialog for the load case.
- There should be no static loads defined in GeniE. Such static loads are irrelevant for fatigue analysis in Framework and may also cause failure in Framework if included. So, either delete any such loads in GeniE or create a new analysis activity without importing such loads.
    - Note that load cases including equipments, weight list items, and explicit point and line loads will be stripped of their loads when *Convert to loadcase independent mass* is selected. Such loads will, therefore, not cause any problem for a fatigue analysis. The *Convert to loadcase independent mass* feature does not convert explicit surface loads though.

- An exception to excluding static loads is spectral fatigue based on equivalent static loads (ESL) structural analysis, see section **12**.

- Damping must be specified. Proportional damping is used as explained in section **11**.
- For a dynamic analysis, a reduction technique may be required if the model is big. Of the two reduction techniques in Sestra, the Master-Slave and the Component Mode Synthesis, the former is more appropriate for fatigue analysis. If the model is split into superelements then a reduction technique will necessarily be used.
  - Assuming the Master-Slave technique is used, and whether the model is a single or multi-superelement model, the 1st level superelements must be modified by introducing master nodes (supernodes) distributed over the model. This is required to properly represent the dynamic energy of all parts of the model. Master (super) nodes should be defined with the following in mind:
    - Define only the three translations as super and let the three rotations be free as the contribution of the rotations to the dynamic energy is normally modest.
    - In case of a multi-superelement model remember that all six degrees of freedom of the supernodes defined for coupling superelements should be super to fully couple them (only the translations as super would result in hinge coupling).
    - Distribute supernodes all over the model.
    - Have more supernodes where the dynamic energy is expected to be high due to large displacements and/or due to high mass.
    - Select nodes with high stiffness to be super. (The Master-Slave technique involves lumping of mass to the master (super) nodes so low stiffness would result in too large displacements of these nodes.)

6 Determine Linear Spring Stiffness for Fatigue

A procedure for determining a linear spring stiffness idealisation of the non-linear pile-soil is described in section 14 ‘Linear Equivalent Stiffness Matrices’ of the document ‘Splice Engineering Documentation’. Find the document from *File | Help* in Sesam Manager. Type ‘splice’ in the *Sesam user documentation* search field and open the document named *spl-ed.pdf*. This procedure is implemented as a feature in Splice. In GeniE this feature is available by editing the pile-soil analysis as shown in the Figure 6-1 below.

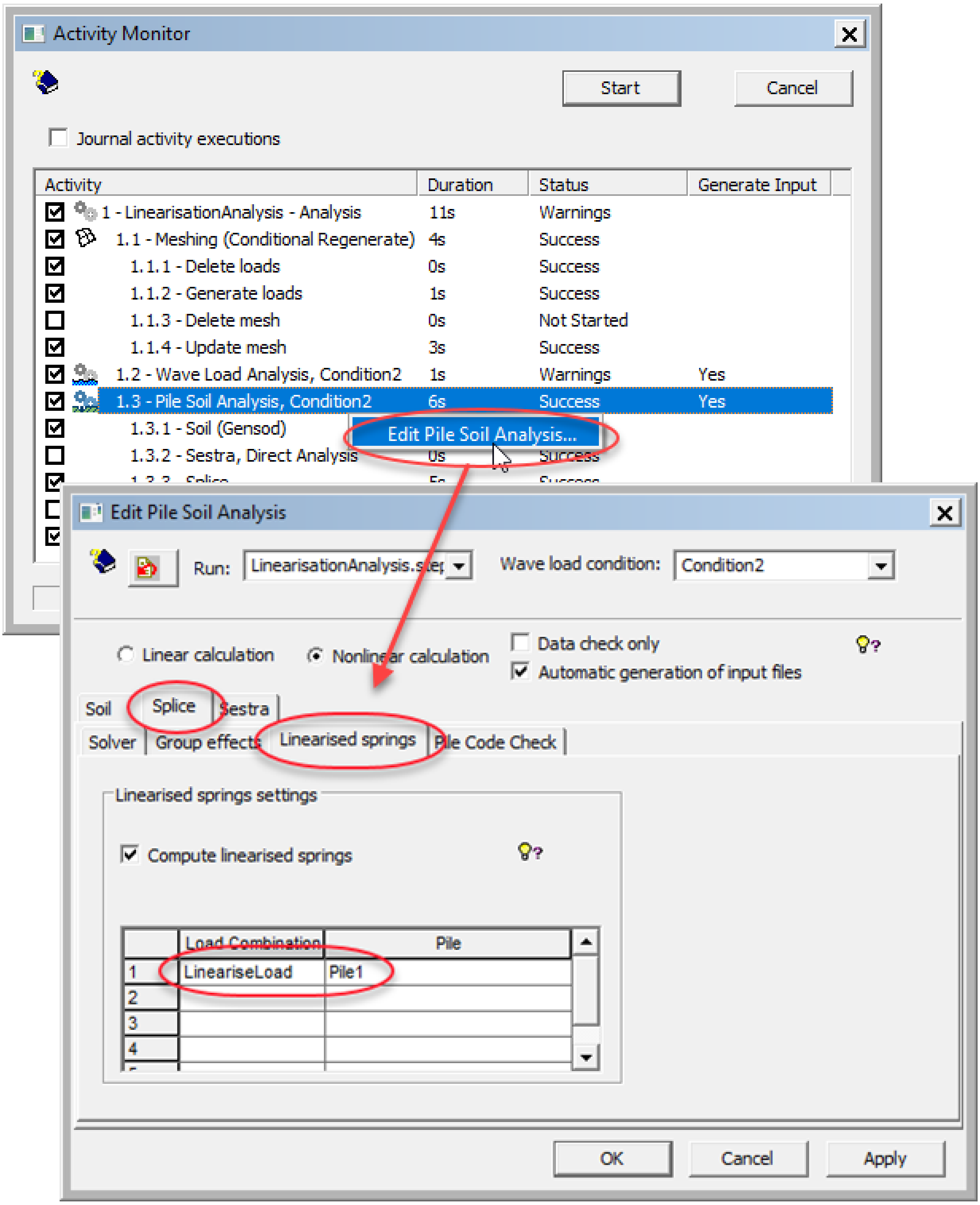


Figure 6-1 Edit the pile-soil analysis activity in GeniE to linearise pile-soil

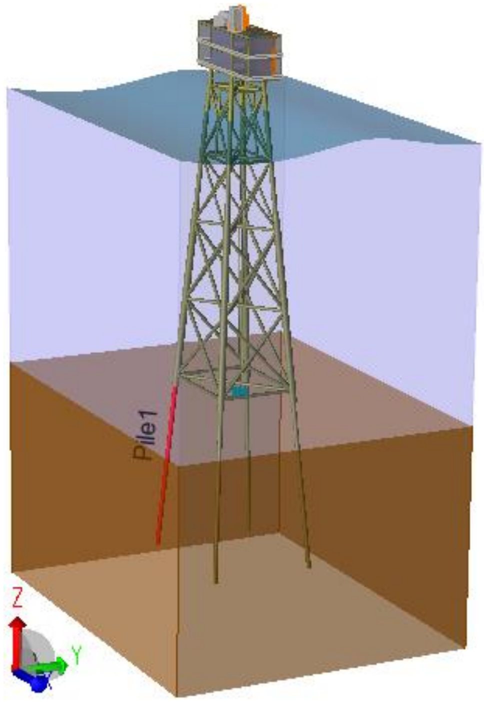


Figure 6-2 Edit the pile-soil analysis activity in GeniE to linearise pile-soil

A load and a pile must be selected for the linearisation process. In this case there are four piles, all equal. Pile1, see the Figure 6-2 above, is chosen.

The selected load (combination) must include a typical wave. This wave may be the one contributing the most to fatigue damage, or, alternatively, the one for which 50% of the damage occurs for smaller waves and 50% for bigger waves. One can imagine some advanced procedure for selecting this wave but the easiest may be to simply make a qualified guess and then verify this guess based on fatigue results from Framework. Note that the Framework command *DEFINE FATIGUE-DUMP* dumps detailed results like damage per sea state, damage per direction, damage per hotspot, etc.

In this tutorial a wave of 6 m and 8 sec. with direction from north (270°) is chosen.

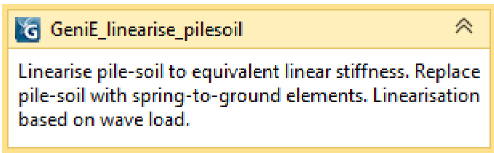


Figure 6-3 Activity for determining linear springs replacing non-linear pile-soil in *Flow Chart* view

The pile-soil linearisation run is performed by the activity **GeniE_linearise_pilesoil**. The GeniE workspace of the previous activity, GeniE_ULS_piled_model, is imported and modified. The input creates a new wave load plus a pile-soil analysis activity named LinearisationAnalysis, and runs Wajac, Sestra and Splice. The Splice run produces two files, SPLICE_EQSTIFF.LIS and SPLICE_EQSTIFF.FEM, both found in the analysis folder. These files contain the linearised stiffnesses of the piles for the selected load. The former file is for viewing and the latter file is for importing into GeniE. Even though the linearization process focuses on a single pile, linearised stiffnesses are printed for all piles.

The SPLICE_EQSTIFF.FEM file is imported into GeniE (interactively by *File* | *Import* | *Import linearised pilehead springs*) and the piles are deleted. The model is now ready for further analyses.

After importing the linearised springs, but *before* deleting the piles, the GeniE workspace is exported as the file ModelWithSpringToGroundAndPilesSoil.gnx. The reason for this is that this model is used in the equivalent static loads (ESL) analysis, see section **12**.

7 Free Vibration Analysis of Model with Springs

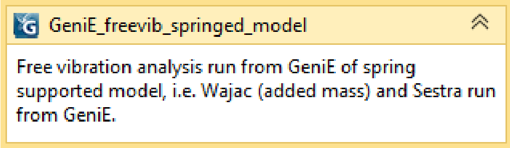


Figure 7-1 Activity for free vibration analysis of springed model in *Flow Chart* view

A free vibration analysis with the linearised stiffnesses (spring-to-ground elements) replacing the pile-soil foundation is performed by the activity **GeniE_freevib_springed_model** to find the resonance frequencies of the jacket. The GeniE workspace of a previous activity, GeniE_linearise_pilesoil, is imported and modified. The mesh density for the topside is reduced and computation of added mass in Wajac and linear free vibration analysis in Sestra are set up. Wajac and Sestra are run under the control of GeniE, the analysis activity is named FreevibAnalysis. The first five eigenperiods found are 2.963, 2.390, 1.282, 0.905 and 0.839. These are as expected higher than those of the fixed model since the spring-to-ground fixation is softer. These eigenperiods should be taken into account when specifying wave periods in the later forced response analysis in frequency domain. The eigenperiods will also be used to find proper proportional damping coefficients as explained in section **11**.

Mode shapes should be studied in Xtract (using animation if necessary) to reveal possibly irrelevant or incorrect modes. You can start Xtract from within GeniE by *Tools* | *Analysis* | *Advanced Results (Xtract)*.

You can also start Xtract as shown in Figure 7-2 below from the *File Overview* tab that is found next to the *Applications* tab in the lower left corner of Sesam Manager. Click the *Show All Files* button and find the result file *R1.SIN under FreevibAnalysis under the activity folder GeniE_freevib_springed_model. Right-click the results file and select *View with Xtract*.

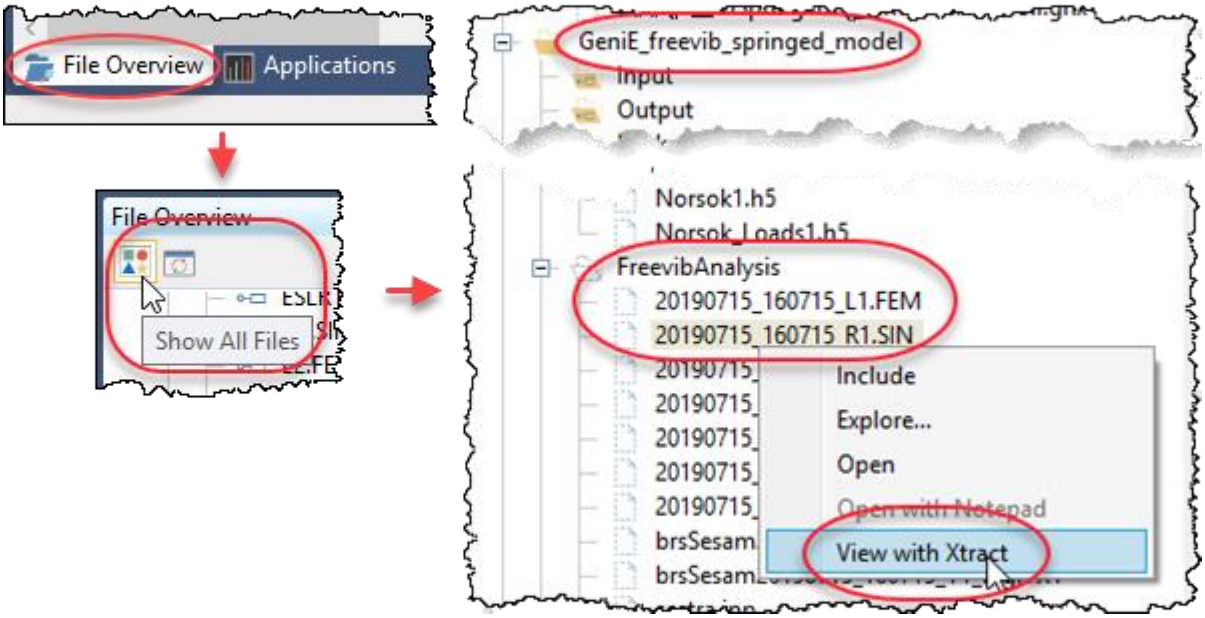


Figure 7-2 Start Xtract from Sesam Manager to view mode shapes

The first five modes are shown in Figure 7-3 below. Modes 1 and 4 are the two first bending modes in Y, modes 2 and 5 are the two first bending modes in X. Mode 3 is a torsional mode.

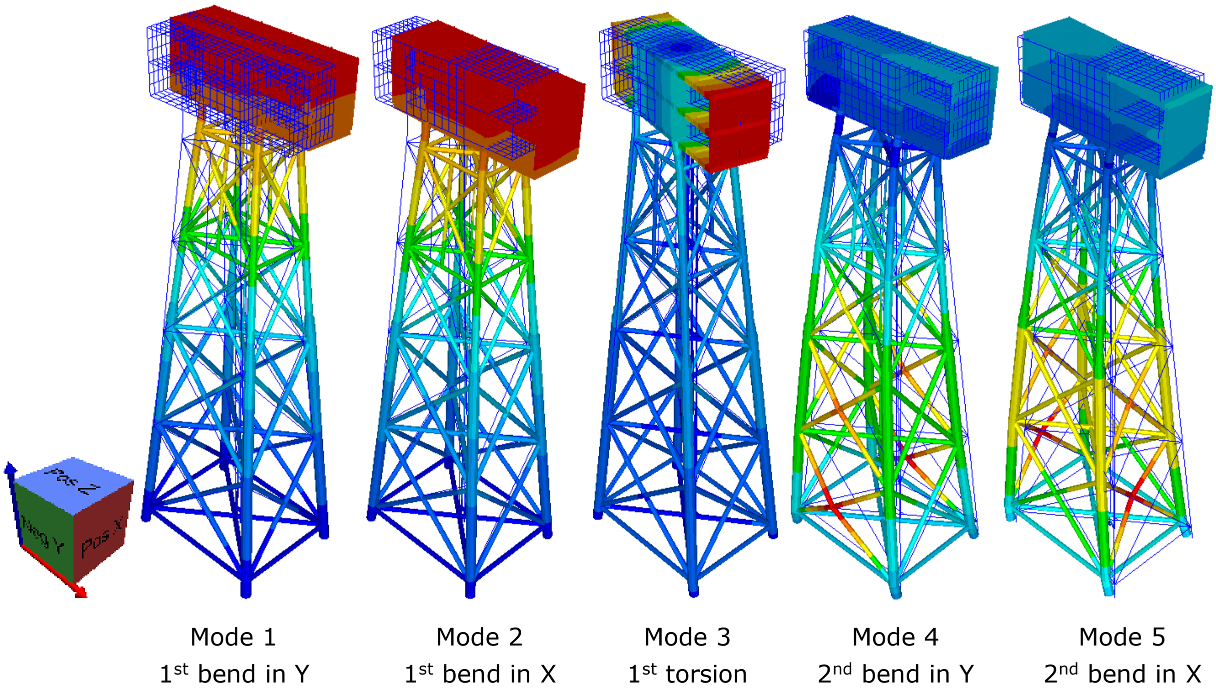


Figure 7-3 The five first mode shapes as displayed in Xtract

8 Fatigue Analyses in Sesam

There are four main methods for fatigue analysis available in Sesam as presented in the Table 8-1 below with their pros and cons. Except for the time domain method, all fatigue methods are demonstrated in this tutorial.

Table 8-1 Pros and cons for the four methods for fatigue analysis in Sesam

Fatigue method	Pros and cons for fatigue analysis method
Deterministic	The deterministic fatigue analysis method is for dynamically insensitive structures in shallow to medium water depths where non-linearities in the wave force such as drag and variable submergence are of importance. The energy content of the sea-states is not directly represented in the method, so judgment and experience are required in selecting the discrete waves to include in the analysis.
Spectral	The spectral fatigue analysis method is for dynamically sensitive and insensitive structures in shallow to medium water depths where non-linearities in the wave force such as drag and variable submergence are of importance. The structural dynamic analysis, if required, may be computer intensive. The method properly represents the energy content of the sea-states.
	As an alternative to being based on a dynamic analysis of a model with linear spring-to-ground elements, a spectral fatigue analysis may be based on a so-called equivalent static loads (ESL) analysis. This involves a dynamin analysis followed by a static analysis in which dynamic effects are added as static loads. This is primarily intended for fatigue analysis of piles.
Stochastic	The stochastic fatigue analysis method is for dynamically sensitive and insensitive structures in deep water where the non-linearities in the wave force are less important. The structural dynamic analysis, if required, may be computer intensive. The method properly represents the energy content of the sea-states.
Time domain	The time history fatigue analysis method is for studies allowing inclusion of wave force non-linearities as well as structural dynamics. The method is computer intensive involving a large number of time domain simulations in order to capture the energy of the sea-states properly.

An illustration of the processes in Sesam for the four fatigue analysis methods is shown in Figure 8-1 below.

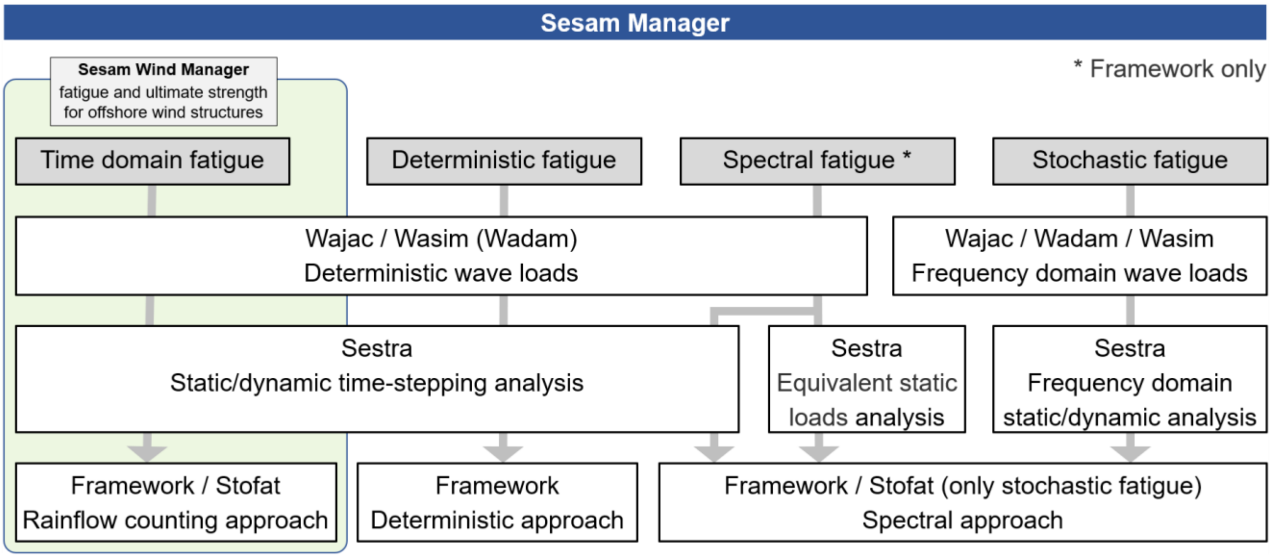


Figure 8-1 The processes in Sesam for the four fatigue analysis methods

9 Deterministic Fatigue Analysis

The sequence (sub-workflow) for deterministic fatigue is shown below in *Flow Chart* view.

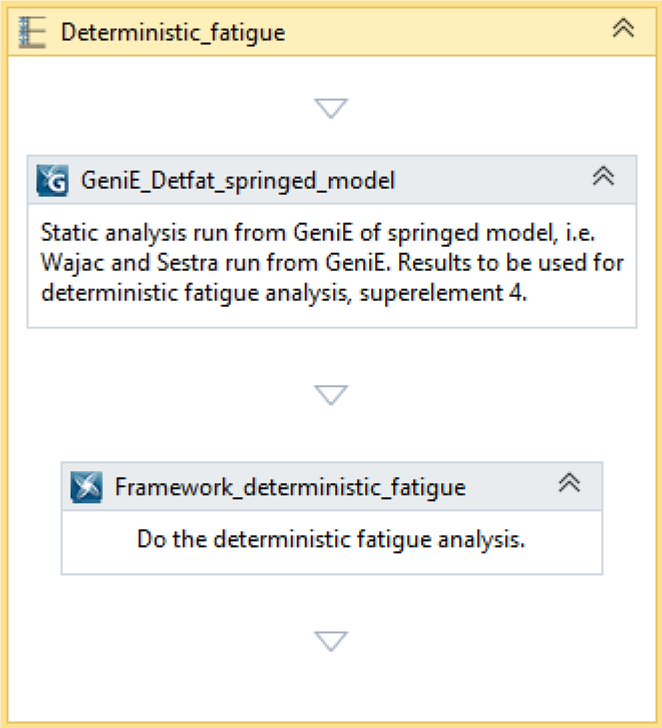


Figure 9-1 The sequence for deterministic fatigue

The deterministic fatigue analysis method is highlighted with red frames and yellow text highlighting in Figure 9-2 below.

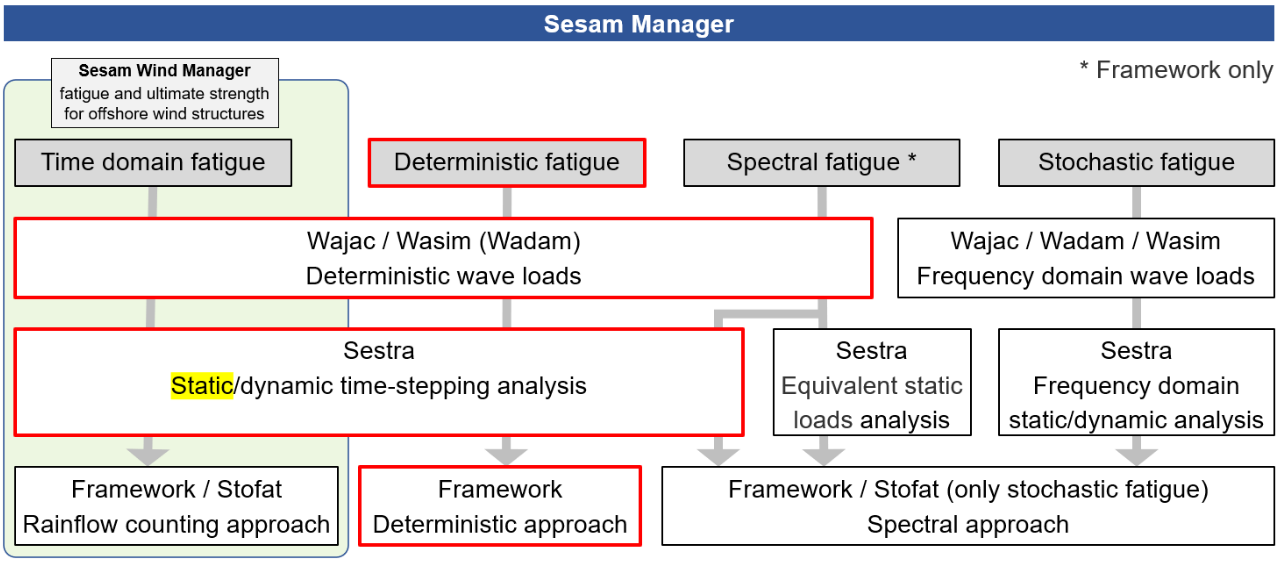


Figure 9-2 Highlighting the processes in Sesam for deterministic fatigue analysis

A deterministic fatigue analysis is not based on linearity and therefore allows taking into account non-linearities such as:

- Load from current (normally not taken into account)
- Drag term of the Morison equation
- Variable submergence during wave cycle

- Pile-soil foundation (normally not included)

As discussed in section **5**, the non-linear pile-soil foundation is, however, often deemed less important since most of the fatigue occurs for small to medium sized waves for which the foundation behaves mostly linearly. The pile-soil foundation is therefore replaced by linear spring-to-ground elements. See section **6** for how to determine the linear spring-to-ground stiffnesses.

A disadvantage of the deterministic fatigue analysis is that it involves a static structural analysis. Dynamic effects, if important, therefore have to be approximately accounted for by use of dynamic amplification factors (DAFs). See sections **13.1** and **13.2** about computing DAFs. Moreover, compared with the stochastic method the deterministic fatigue analysis requires a certain element of judgement guided by experience, especially when selecting the discrete waves on which to base the fatigue assessment.

The deterministic fatigue analysis is contained in the sequence **Deterministic_fatigue**.

A static structural analysis with deterministic wave loads is performed by the activity named **GeniE_Detfat_springed_model**. The GeniE workspace of a previous activity, GeniE_linearise_pilesoil, is imported and modified. A deterministic wave load run in Wajac and a static analysis in Sestra are set up and run under the control of GeniE, the analysis activity is named Detfat. To enable running Framework a *PostExecuteScript* for the GeniE activity in Sesam Manager moves the R4.SIN file (produced by Sestra under the control of GeniE) to the repository. After running the activity ensure that the file R4.SIN is found in the repository.

To see the Sesam Manager *PostExecuteScript*, select the GeniE activity and find it in the right pane as shown in Figure 9-3 below.

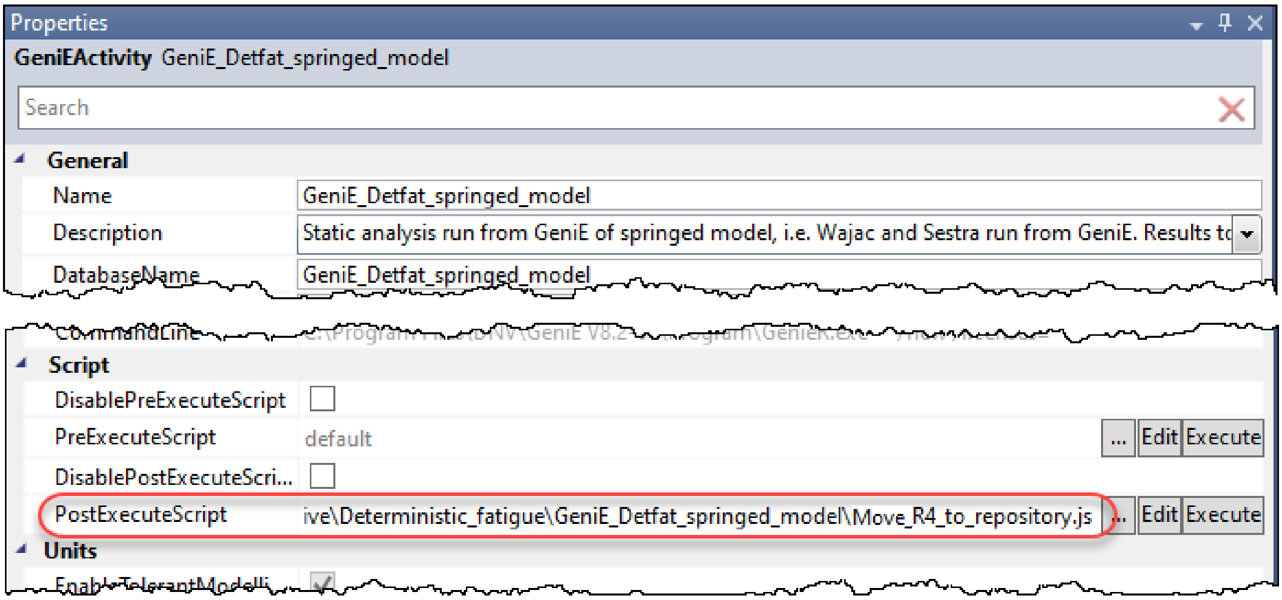


Figure 9-3 View/edit/execute a *PostExecuteScript*

The *PostExecuteScript* for GeniE contains the following command:

```
Move(CurrentActivity.Workspace + @"Detfat\*R4.SIN", @"_repository\R4.SIN");
```

The script moves the R4.SIN file from the analysis folder Detfat (the GeniE analysis activity name) to the repository and at the same time strips it of the time stamp. Alternatively, the R4.SIN file could have been exported from GeniE (*File | Export | Results SIN File*) in which case the default *PostExecuteScript* for GeniE would be suitable.

The activity **Framework_deterministic_fatigue** runs the deterministic fatigue analysis. Only the jacket (i.e., not the deck structure) is included in the run.

Figure 9-4 below is a screen dump of the fatigue results (usage factors).

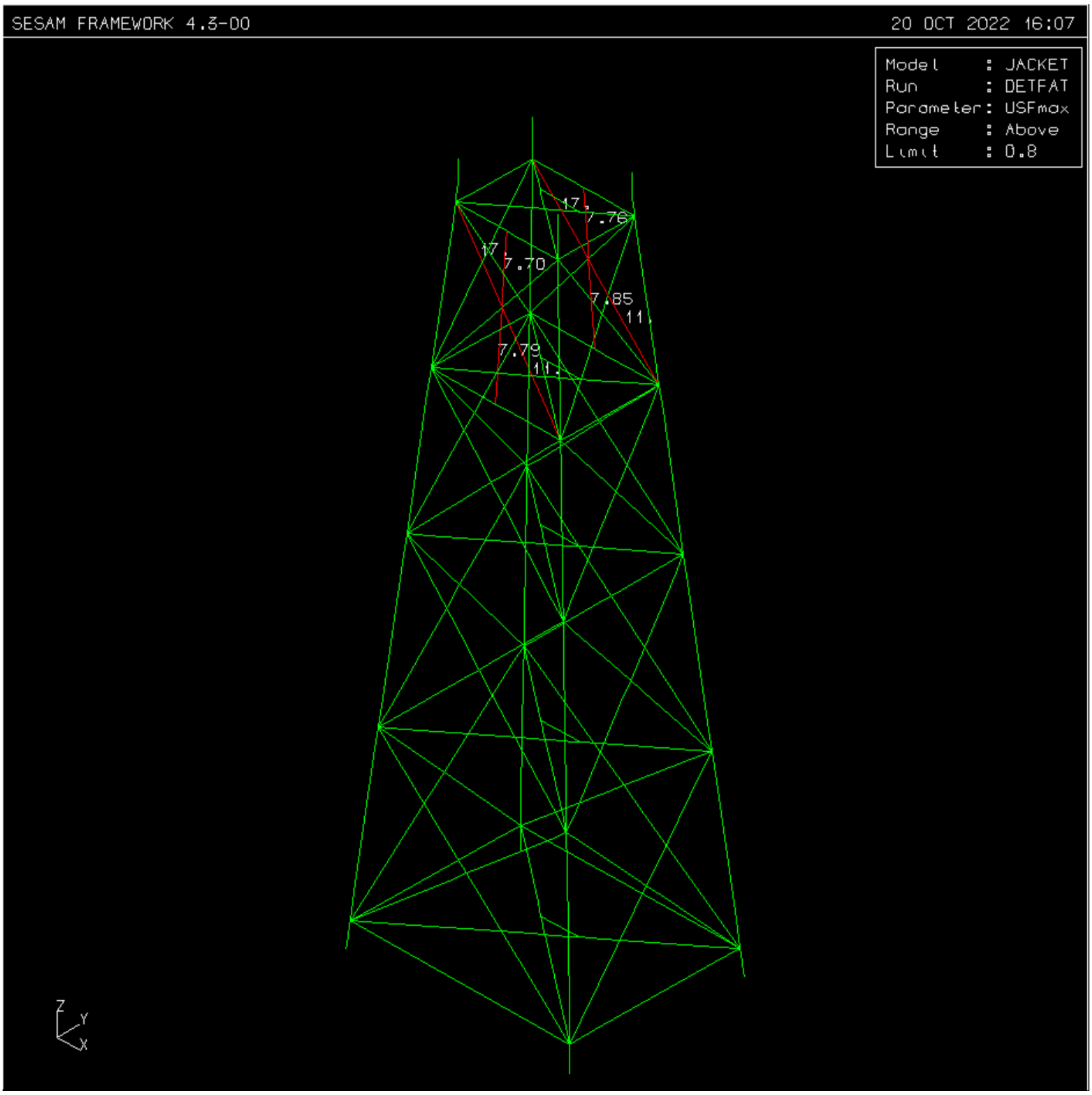


Figure 9-4 Fatigue check result *Max Usage Factor* from deterministic fatigue analysis

10 Spectral Fatigue Analysis – Static Analysis

The sequence for spectral fatigue analysis based on static structural analysis is shown below in *Flow Chart* view.

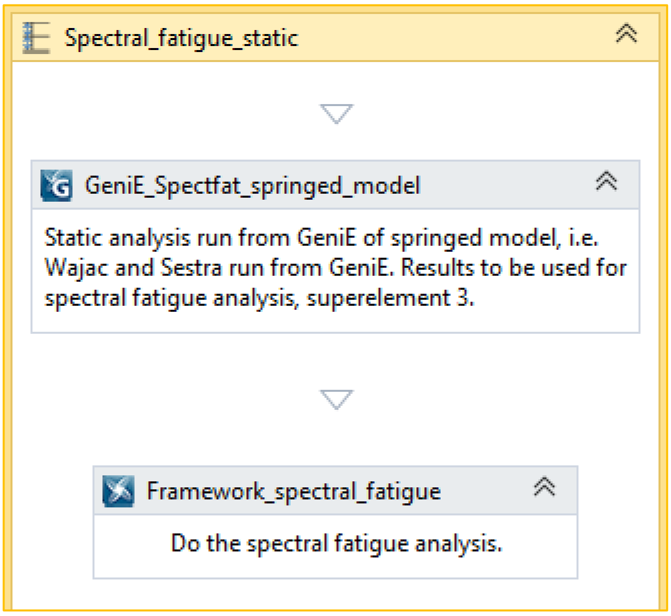


Figure 10-1 The sequence for spectral fatigue analysis based on static analysis

The spectral fatigue analysis method based on static structural analysis is highlighted with red frames and yellow text highlighting in Figure 10-2 below.

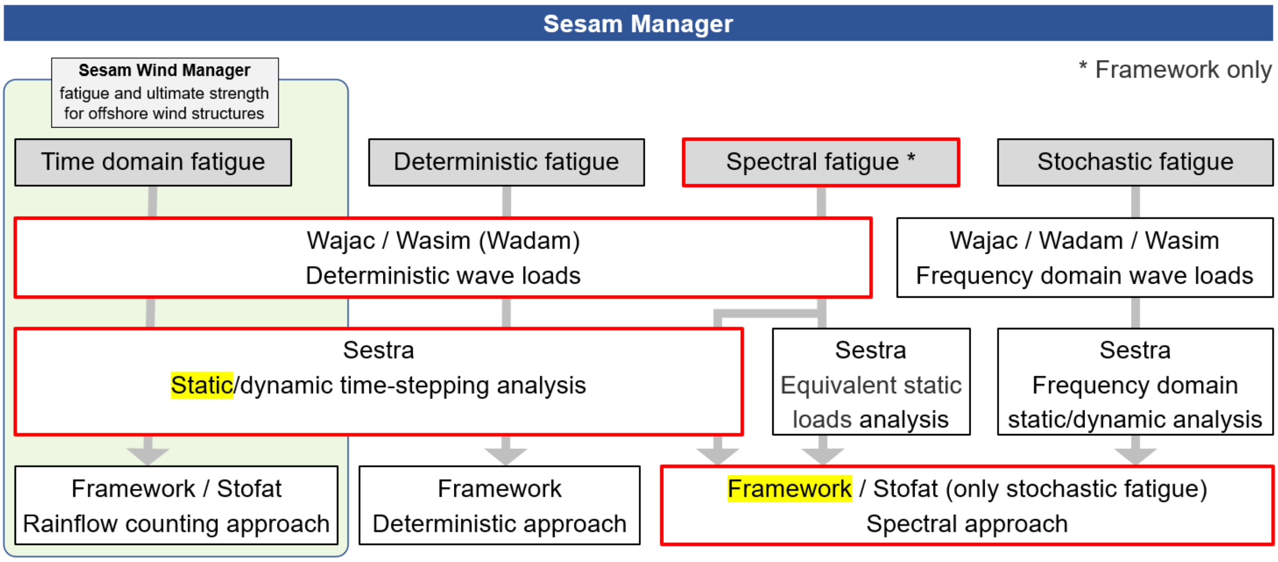


Figure 10-2 Highlighting the processes in Sesam for spectral fatigue analysis based on static analysis

A spectral fatigue analysis combines the deterministic wave load approach in Wajac and Sestra with the spectral fatigue approach in Framework. I.e., the advantages of capturing non-linearities in the wave loading as offered by the deterministic approach, see section **9**, are combined with the advantages of better representation of the energy content of the sea states of the stochastic fatigue analysis, see section **13**.

This section deals with spectral fatigue analysis based on *static* structural analysis. For spectral fatigue analysis based on *dynamic* structural analysis see section **11**. And for spectral fatigue analysis based on *equivalent static loads (ESL)* analysis see section **12**.

The spectral fatigue analysis based on static analysis is contained in the sequence **Spectral_fatigue_static**.

A static structural analysis with deterministic wave loads is performed by the activity named **GeniE_Spectfat_springed_model**. The GeniE workspace of a previous activity, GeniE_linearise_pilesoil, is imported and modified. A deterministic wave load run in Wajac and a static analysis in Sestra are set up and run under the control of GeniE, the analysis activity is named Spectfat. To enable running Framework a *PostExecuteScript* for the GeniE activity moves the R3.SIN file (produced by Sestra under the control of GeniE) to the repository. After running the activity ensure that the file R3.SIN is found in the repository.

The activity **Framework_spectral_fatigue** runs the spectral fatigue analysis. Only the jacket (i.e., not the deck structure) is included in the run.

Figure 10-3 below is a screen dump of the fatigue results (usage factors).

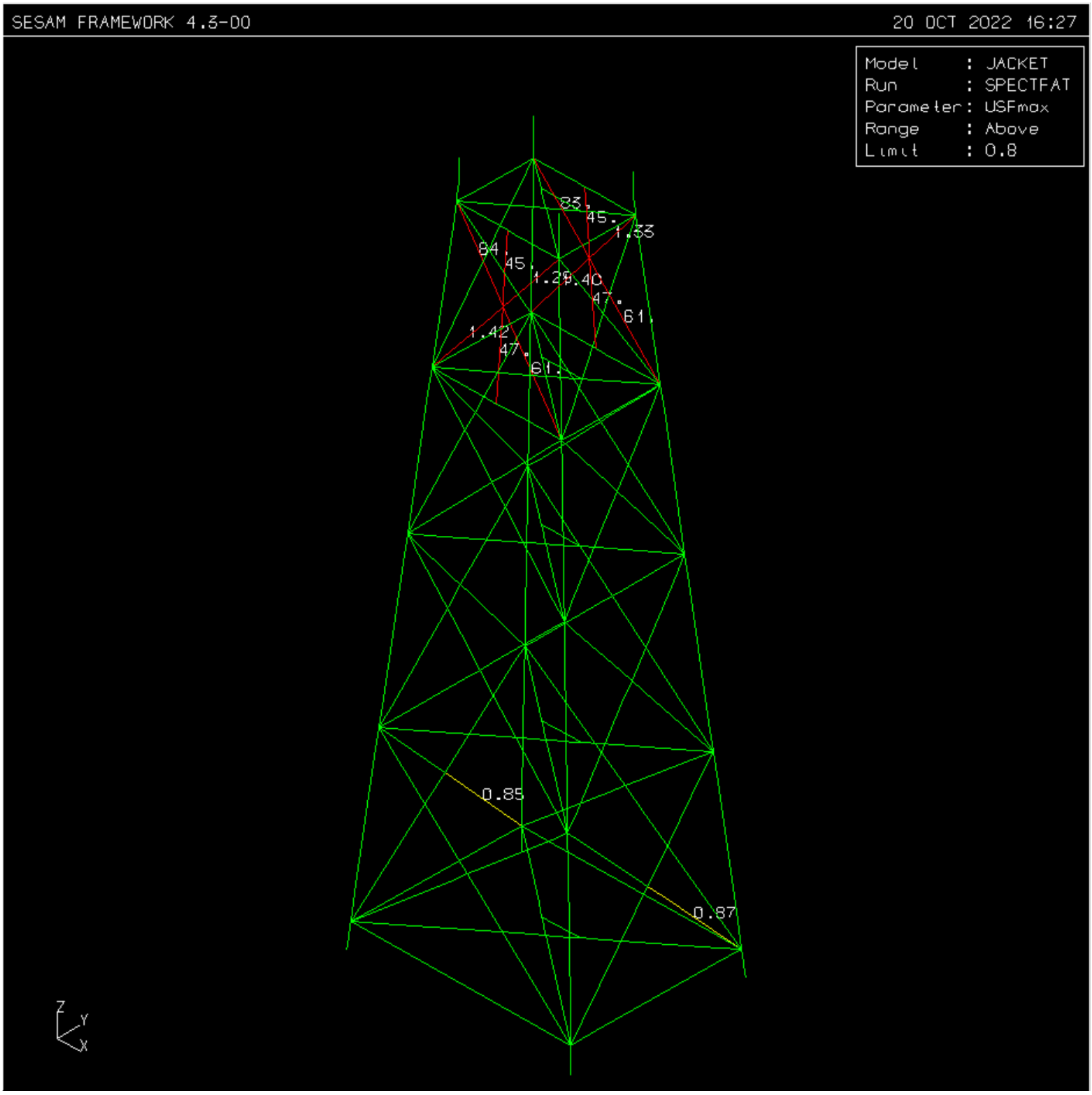


Figure 10-3 Fatigue check result *Max Usage Factor* from spectral fatigue – static analysis

11 Spectral Fatigue Analysis – Dynamic Analysis

The sequence for spectral fatigue analysis based on dynamic structural analysis is shown below in *Flow Chart* view.

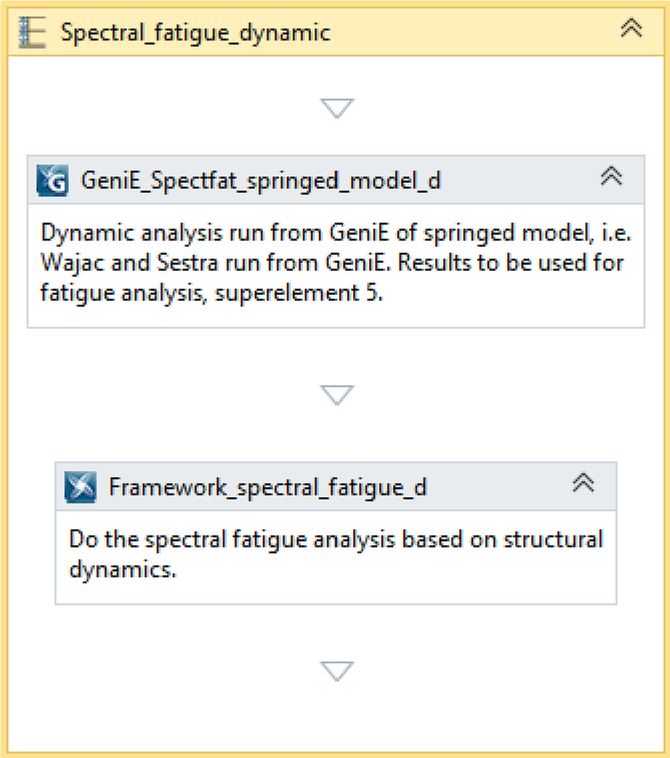


Figure 11-1 The sequence for spectral fatigue analysis based on dynamic analysis

The spectral fatigue analysis method based on dynamic structural analysis is highlighted with red frames and yellow text highlighting in Figure 11-2 below.

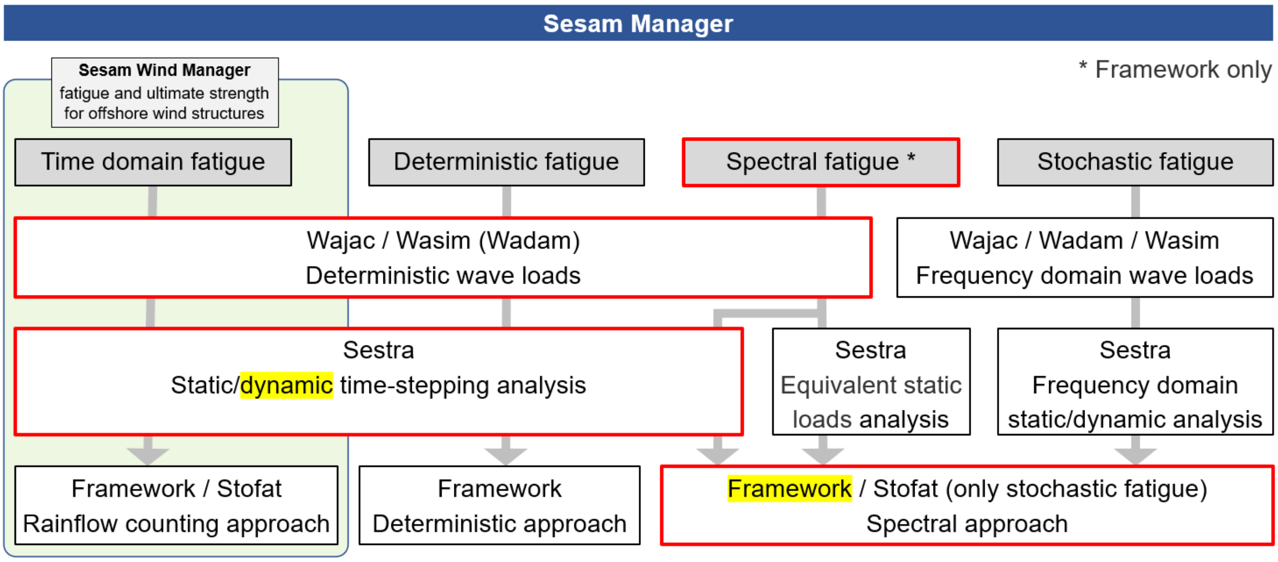


Figure 11-2 Highlighting the processes in Sesam for spectral fatigue analysis based on dynamic analysis

The spectral fatigue analysis based on dynamic analysis is contained in the sequence **Spectral_fatigue_dynamic**.

A dynamic structural analysis with deterministic wave load analysis is performed by the activity named **GeniE_Spectfat_springed_model_d**. The GeniE workspace of a previous activity, GeniE_linearise_pilesoil, is imported and modified. A deterministic wave load run in Wajac and a dynamic analysis in Sestra are set up and run under the control of GeniE, the analysis activity is named Spectfat_d.

At the onset of each wave the structural response will be transient. Each wave is time stepped 0.1 seconds and repeating the wave cycle until steady state of the structural response is detected. The tolerance (see the Sestra user manual about this) used to decide when steady state has been reached is set to 0.01.

Results for *only the last wave cycle*, the first steady state cycle, is stored in the results file and subsequently used in the fatigue analysis. The number of wave cycles required to reach steady state depends on the wave. The number of cycles is about 35 for the smallest waves and about 16 for the largest waves. The other waves are in between these number of cycles. A maximum of 50 cycles is set to ensure that the run does not continue indefinitely.

Proportional damping (Rayleigh damping) that sets the damping matrix to be a factored sum of the mass and stiffness matrices has been selected with proportional factors α_1 (factor for mass matrix) and α_2 (factor for stiffness matrix) determined by:

$$\propto_1 = \frac{2\omega_j\omega_k\left(\lambda_j\omega_k - \lambda_k\omega_j\right)}{\omega_k^2 - \omega_j^2}$$

$$\propto_2 = \frac{2\left(\lambda_k\omega_k - \lambda_j\omega_j\right)}{\omega_k^2 - \omega_j^2}$$

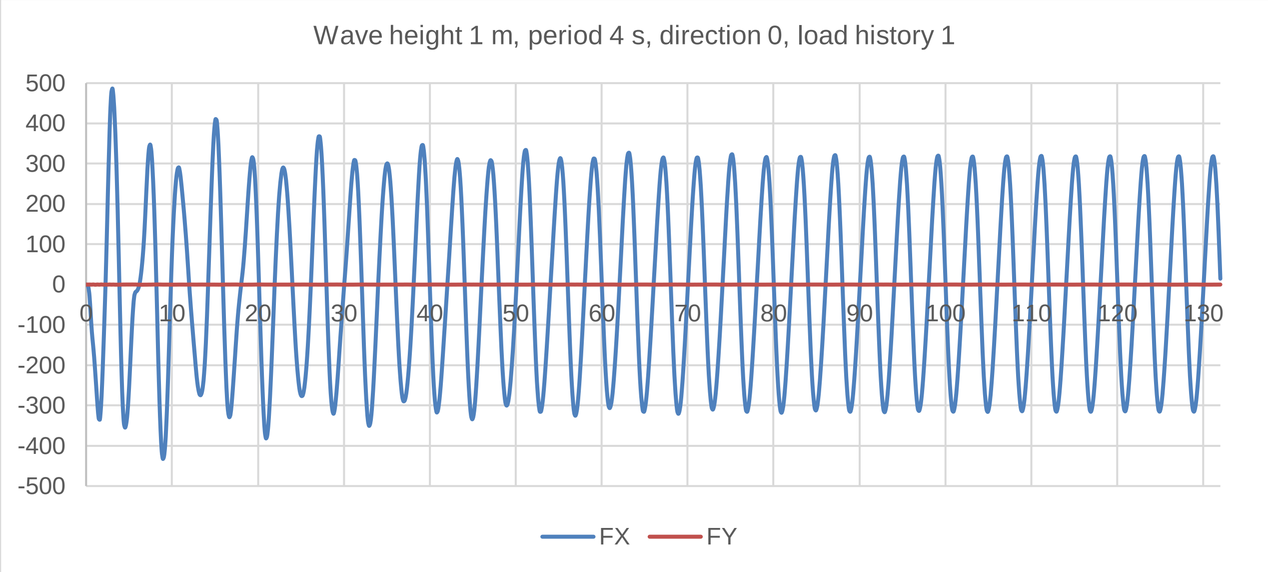
Here ω_j and ω_k are two selected angular eigenfrequencies, normally those of the first and second bending modes. λ_j and λ_k are the modal damping ratios belonging to the two selected modes.

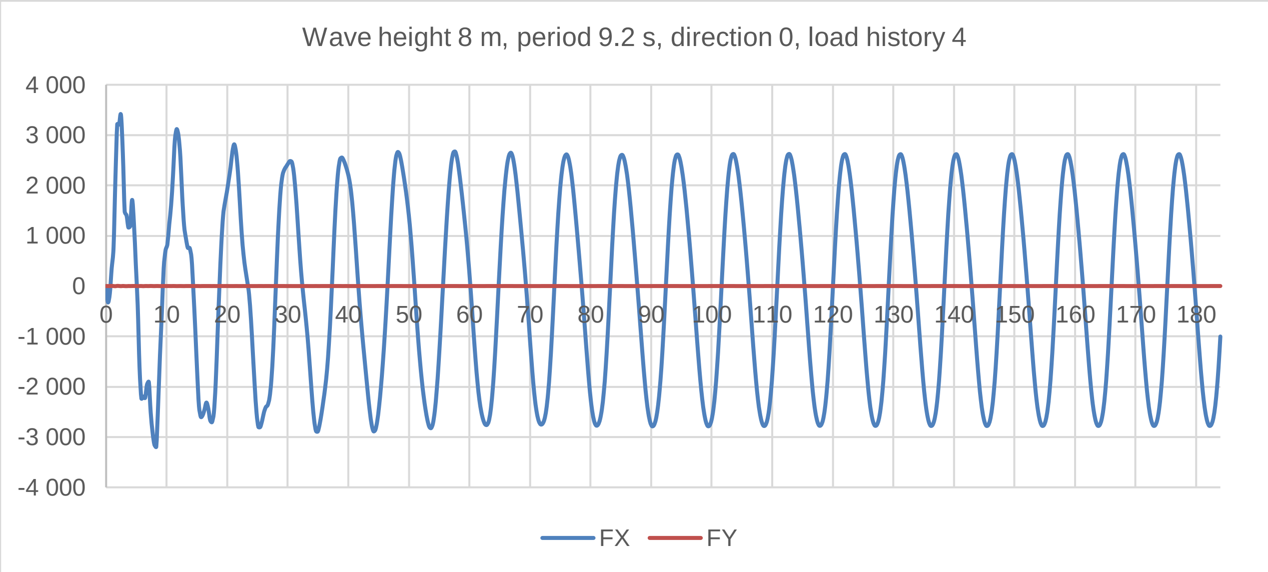
From the free vibration analysis, see section **7**, we find the eigenperiods for the first and second bending in X direction to be 2.390 and 0.839, respectively, and 2.963 and 0.905 for the first and second bending in Y direction. These values, converted to angular frequencies ($\omega = {}^{2\pi}/_{T}$), plugged into the equations above together with modal damping ratios set to 0.02, give two sets for α_1 and α_2, one set for X and one set for Y direction. Averaging these sets gives $\alpha_1 = 0.071$ and $\alpha_2 = 0.0042$. (A spreadsheet for computing these values is attached to this Sesam Manager job.)

Ensure that the results file R5.SIN is found in the repository after the execution.

In addition to the results file with results for the last wave cycle, the time domain dynamic analysis in Sestra produces a file containing the sum of reaction forces for each time step through *all cycles* of each wave, i.e., one file for each load history (number of deterministic waves times number of wave directions), in total 56 load histories. The names of the files are _reactions_lohi*i*.csv prefixed by a time stamp and where *i* is the load history number. The files are found under the GeniE analysis activity folder Spectfat_d.

Graphs of the reaction sums for the two horizontal directions are presented in Figure 11-3 through Figure 11-5 below for three waves (smallest, middle and largest) in directions 0, 45 and 90. The graphs confirm that steady state has been reached for the load histories.





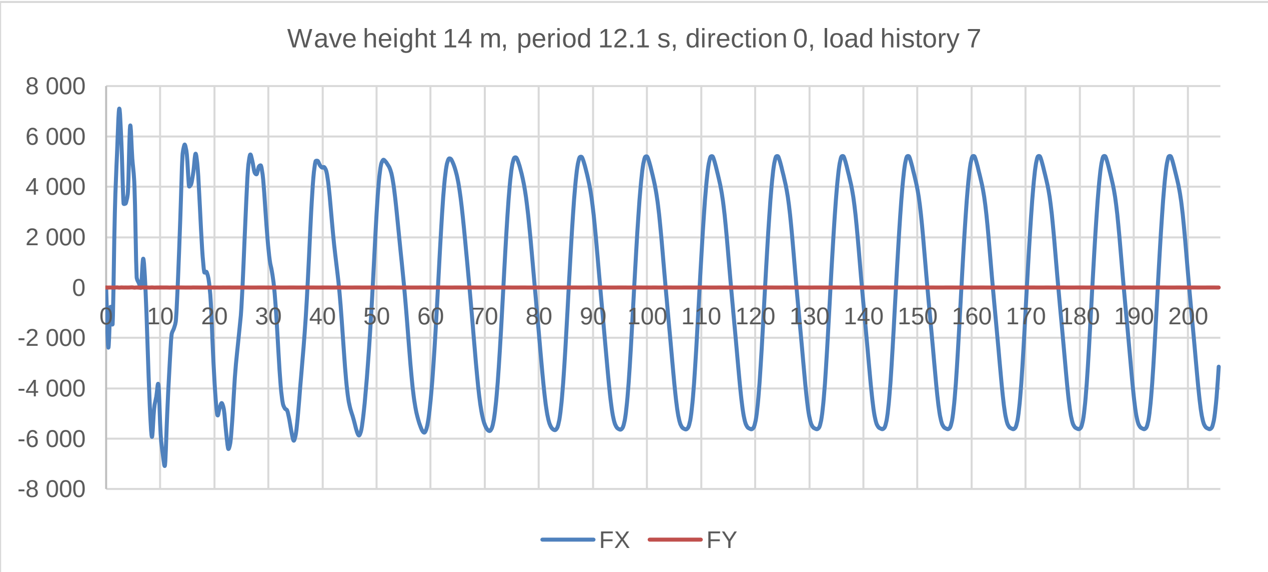


Figure 11-3 Graphs of reaction sums in X and Y for load histories 1, 4 and 7

Figure 11-4 Graphs of reaction sums in X and Y for load histories 8, 11 and 14

Figure 11-5 Graphs of reaction sums in X and Y for load histories 15, 18 and 21

The activity **Framework_spectral_fatigue_d** runs the spectral fatigue analysis. Only the jacket (i.e., not the deck structure) is included in the run.

Figure 11-6 below is a screen dump of the fatigue results (usage factors).

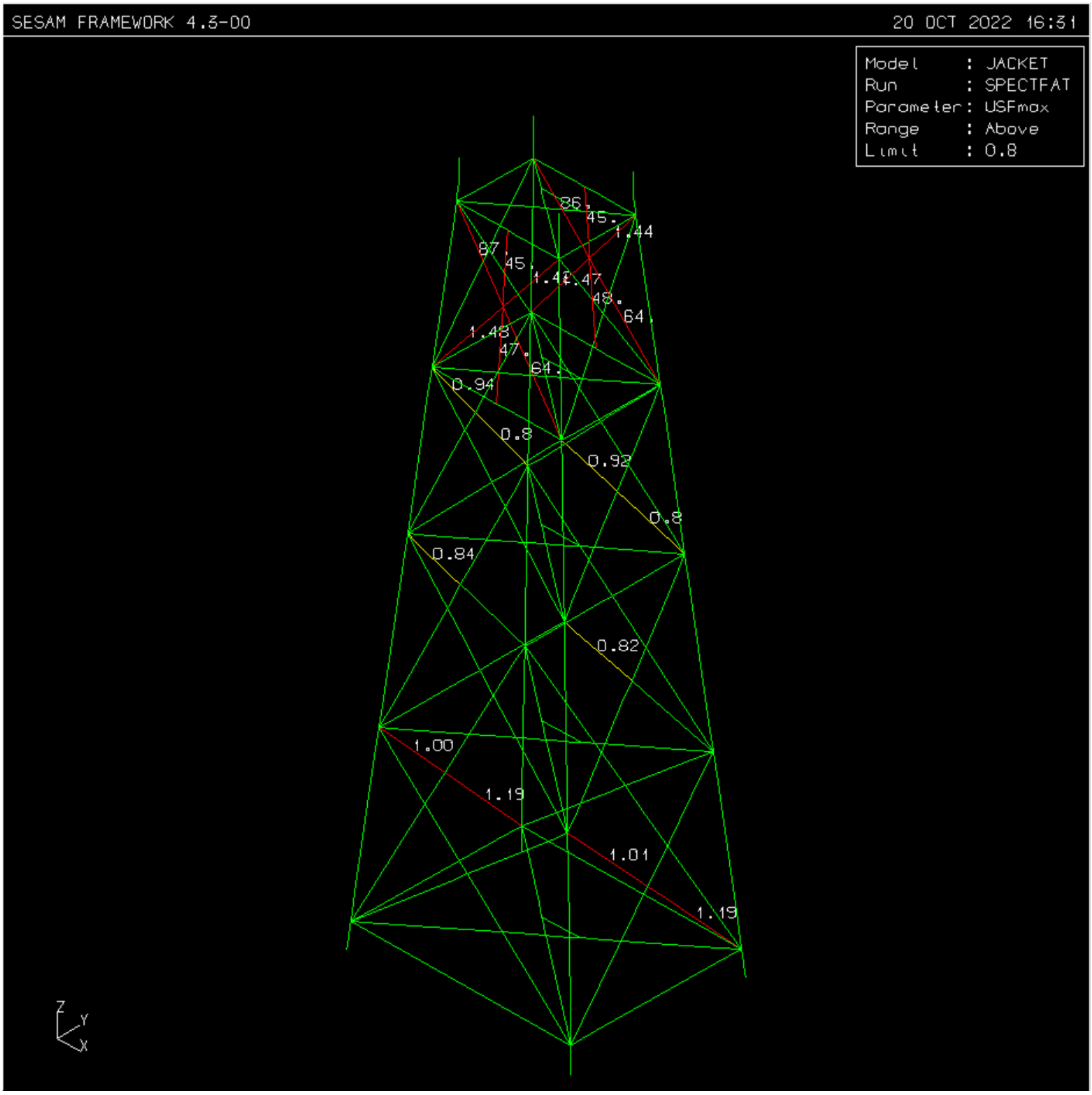


Figure 11-6 Fatigue check result *Max Usage Factor* from spectral fatigue – dynamic analysis

12 Spectral Fatigue Analysis – Equivalent Static Loads (ESL) Analysis

The sequence for spectral fatigue analysis based on ESL structural analysis is shown below in *Flow Chart* view.

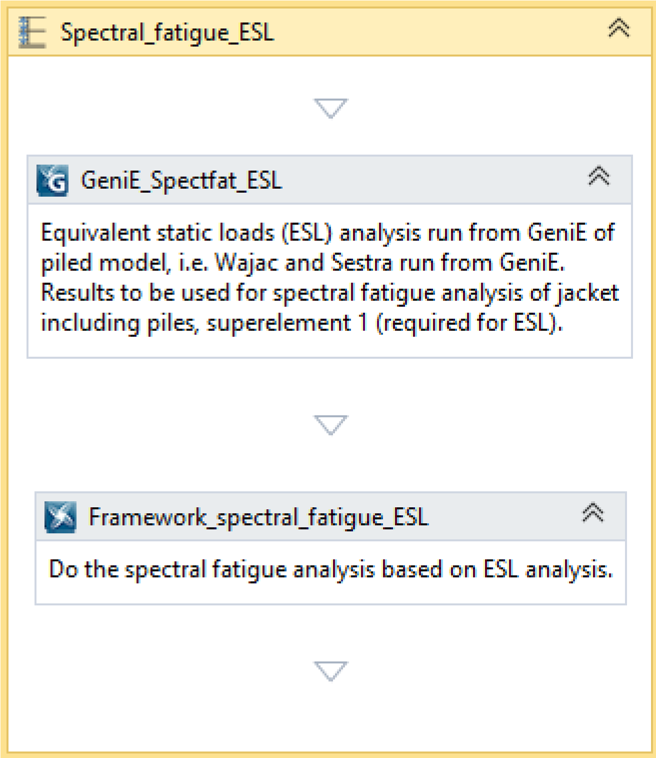


Figure 12-1 The sequence for spectral fatigue analysis based on ESL analysis

The spectral fatigue analysis method based on ESL structural analysis is highlighted with red frames and yellow text highlighting in Figure 12-2 below.

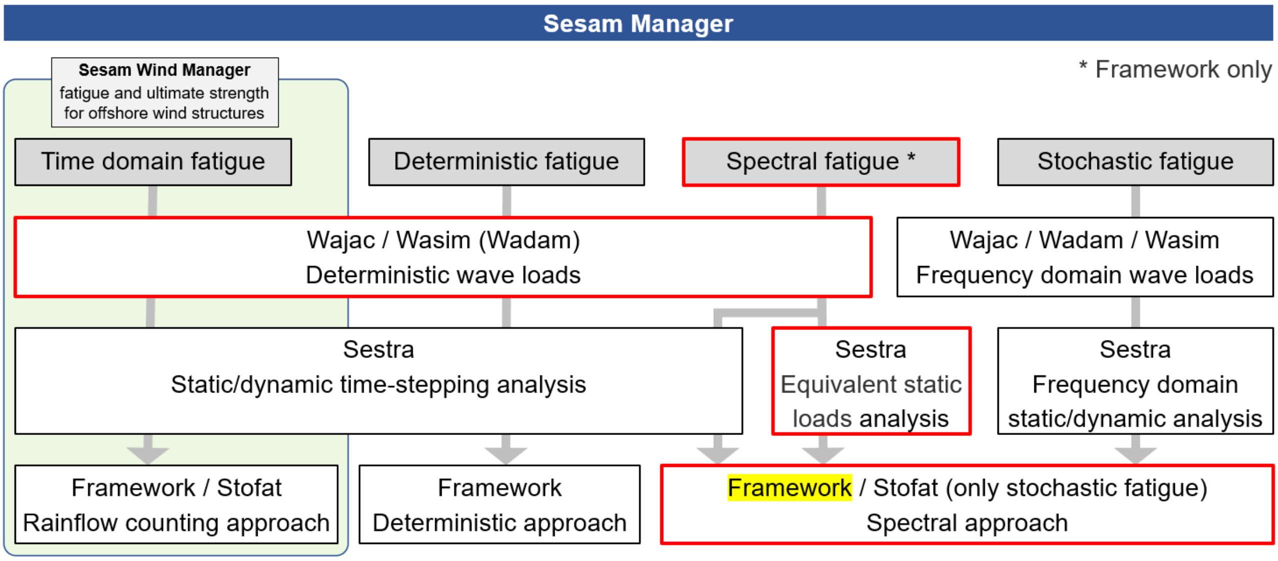


Figure 12-2 Highlighting the processes in Sesam for spectral fatigue analysis based on ESL analysis

The spectral fatigue analysis based on equivalent static loads analysis is contained in the sequence **Spectral_fatigue_ESL**.

An equivalent static loads analysis with deterministic wave loads is performed by the activity named **GeniE_Spectfat_ESL**. The GeniE workspace ModelWithSpringToGroundAndPilesSoil.gnx (exported from the GeniE activity GeniE_linearise_pilesoil) is imported and modified. A deterministic wave load run in Wajac and an equivalent static loads analysis in Sestra are set up and run under the control of GeniE, the analysis activity is named Spectfat_ESL.

Note that an equivalent static loads analysis involves two Sestra runs:

1. A time domain dynamic analysis similar to the one of the spectral fatigue analysis method based on dynamic structural analysis, see section **11**, is performed. In this analysis equivalent static loads for the last wave cycle (and first steady state wave cycle) are stored to file SestraESL.srs. The equivalent static loads are essentially the external dynamic loads subtracted by internal dynamic forces (inertia and damping). See the Sestra user manual about this.

2. A static analysis is performed with import of the equivalent static loads from file SestraESL.srs. This analysis may be a structure-pile-soil interaction analysis, i.e., running Splice that in turn runs Sestra.

To enable the above, the model must contain *both* pile-soil foundation *and* spring-to-ground elements. The spring-to-ground elements are used in the first analysis and the pile-soil foundation in the second.

To enable running Framework a *PostExecuteScript* for the GeniE activity moves the R1.SIN file to the repository while adding the prefix ESL. The *PostExecuteScript* achieves this by the two commands:

```
Move(CurrentActivity.Workspace + @"Spectfat_ESL\*R1.SIN", @"_repository\R1.SIN");
Move(@"_repository\R1.SIN", @"_repository\ESLR1.SIN");
```

After running the activity ensure that the file ESLR1.SIN is found in the repository. Note that due to a restriction in Splice/GeniE the superelement number must be 1.

As for the time domain dynamic structural analysis described in section **11**, files are produced containing the sums of reaction forces for each time step through *all cycles* of all 56 load histories. The names of the files are _reactions_lohi*i*.csv prefixed by a time stamp and where *i* is the load history number. The files are found under the GeniE analysis activity folder Spectfat_ESL.

Graphs of the reaction sums for the two horizontal directions are presented below for three waves (smallest, middle and largest) in directions 0, 45 and 90. The graphs confirm that steady state has been reached for the load histories.

The activity **Framework_spectral_fatigue_ESL** runs the spectral fatigue analysis. The jacket with piles is used in the run.

Figure 12-3 below is a screen dump of the fatigue results (usage factors).

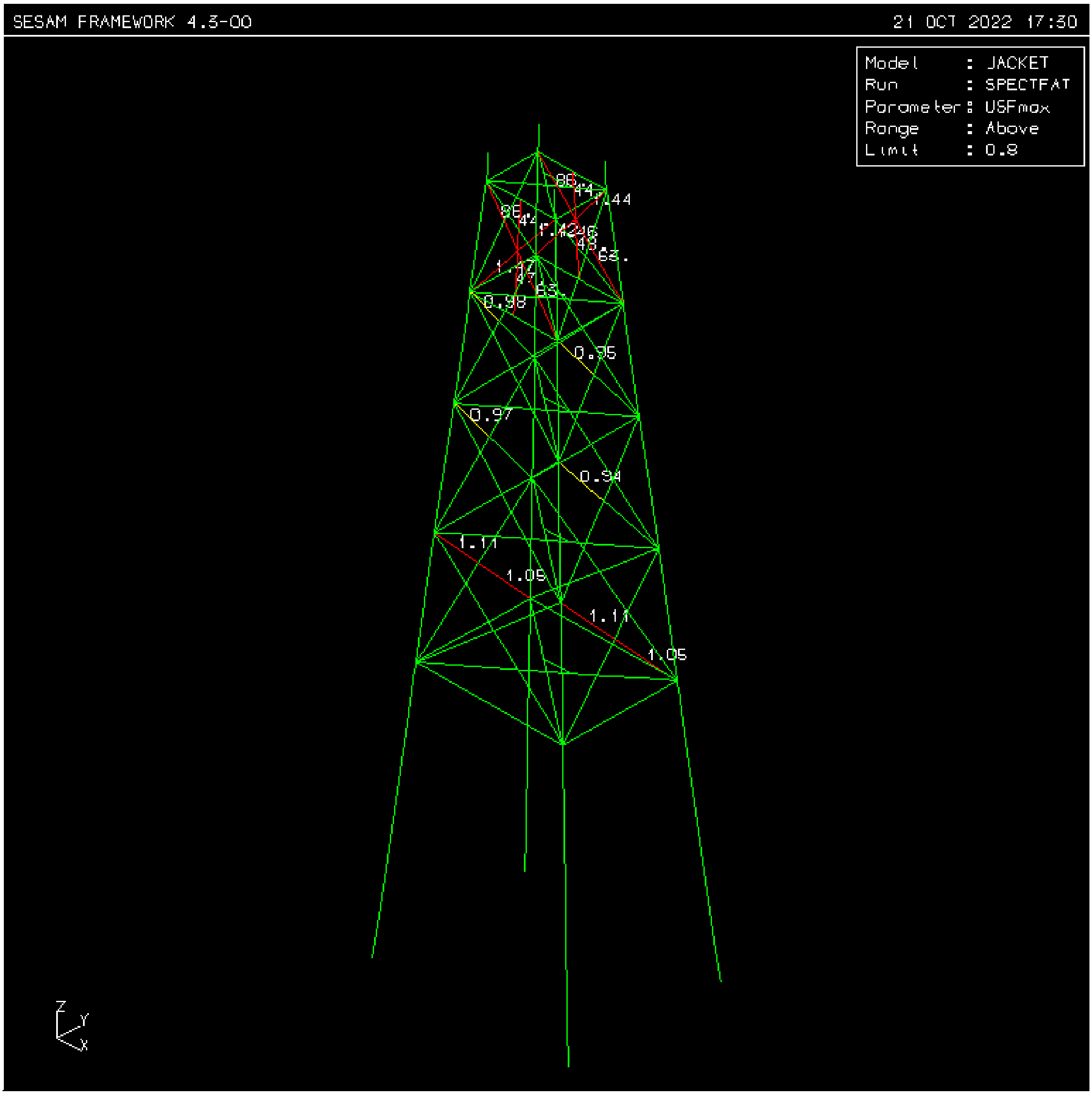


Figure 12-3 Fatigue check result *Max Usage Factor* from spectral fatigue – ESL analysis

13 Stochastic Fatigue Analysis by Dynamic Forced Response Analysis – Find DAFs

The sequence for stochastic fatigue analysis for beam model of joints is shown below in *Flow Chart* view.

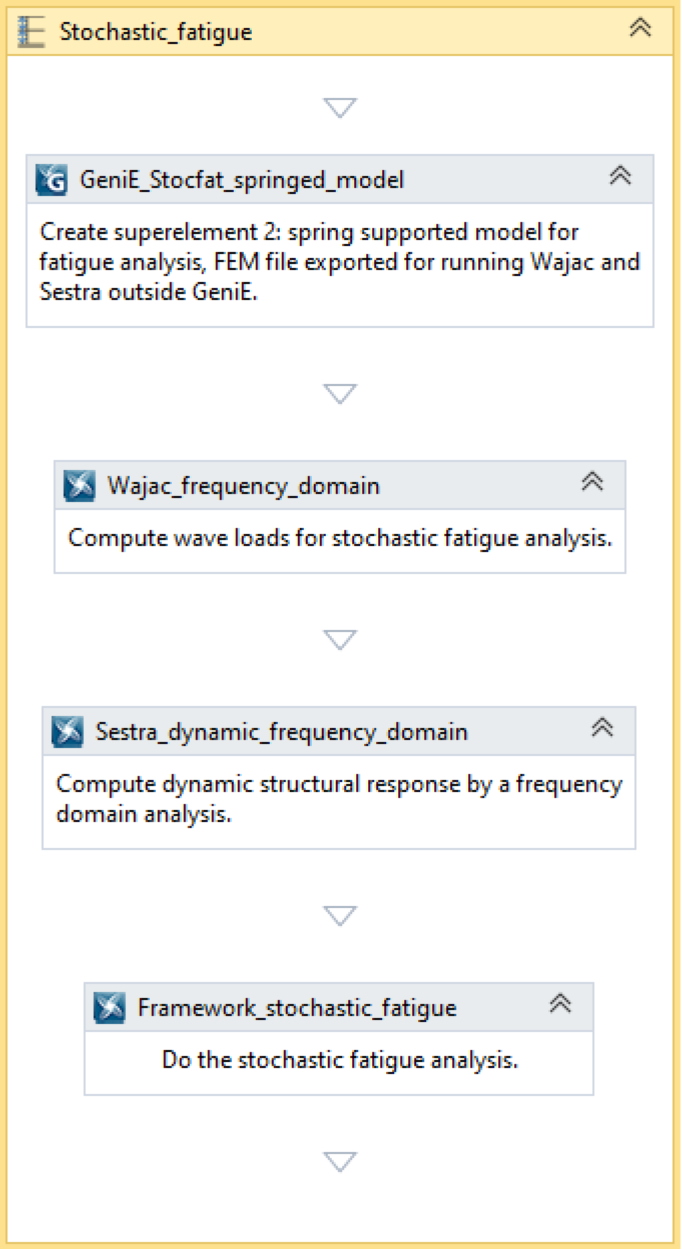


Figure 13-1 The sequence for stochastic fatigue analysis of beam model of joints

The stochastic fatigue analysis method for beam model of joints is highlighted with red frames and yellow text highlighting in Figure 13-2 below.

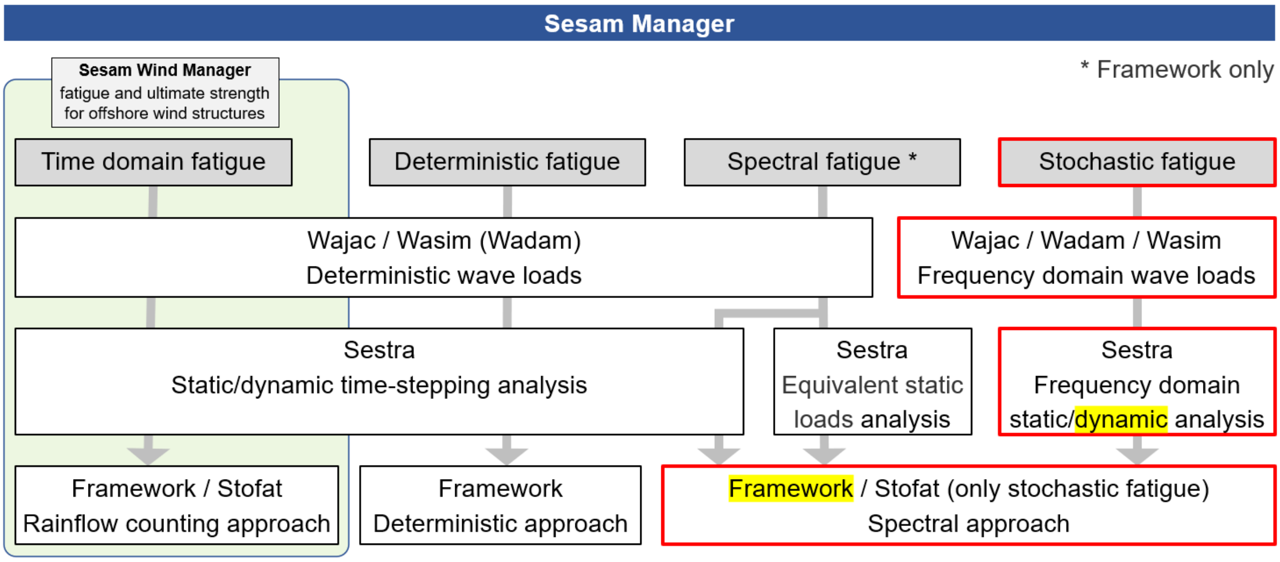


Figure 13-2 Highlighting the processes in Sesam for stochastic fatigue analysis of beam model of joints

The stochastic fatigue analysis is contained in the sequence **Stochastic_fatigue**.

The activity **GeniE_Stocfat_springed_model** imports and modifies the workspace of the activity GeniE_linearise_pilesoil. The model is exported (T2.FEM, i.e., superelement 2) since both Wajac and Sestra will be run from Sesam Manager rather than from GeniE. (GeniE cannot set up and run a frequency domain analysis.) The analysis activity creating the model is named Analysis1. After running the GeniE activity ensure that the file T2.FEM is found in the repository.

Run Wajac for computing wave transfer functions and added mass. The execution is found in the activity named **Wajac_frequency_domain**. The wave periods given as input should be selected based on:

- Eigenperiods of the structure
- Amplification effects (caused by wave crests hitting the legs simultaneously so that the wave forces act in the same direction, the peaks in the transfer function graph further below) and cancellation effects (caused by crest and trough hitting the legs simultaneously so that the wave forces act in the opposite direction, the dips in the graph)
- Where the wave energy is high
- Periods more or less evenly distributed in the range with high wave energy

Ensure that the files L2.FEM and S2.FEM are produced by the Wajac run and copied (by Sesam Manager) to the repository.

In addition to the loads file, a frequency domain dynamic analysis in Wajac produces a file containing the transfer functions for all wave directions. The name of the file is TRF.out.

Now run the Sestra forced response dynamic analysis using the direct frequency response method. This is activity **Sestra_dynamic_frequency_domain**. In consistency with the previous dynamic analyses, see sections **11** and **12**, proportional damping (Rayleigh damping) has been selected with proportional factors 0.071 and 0.0042 for the mass and the stiffness matrices, respectively. Ensure that the results file R2.SIN is found in the repository after the Sestra execution.

In addition to the results file, a frequency domain dynamic analysis in Sestra produces a file containing the transfer functions for reaction forces all wave directions. The name of the file is _reactions_RAO.csv.

13.1 Global Dynamic Amplification Factors (DAFs)

We may now compute global dynamic amplification factors by comparing the dynamic response found in the file _reactions_RAO.csv with the static response found in the file TRF.out. For this purpose, we can use the base shear.

Figure 13-3 below shows extracts from the TRF.out file (right) and _reactions_RAO.csv file (left). The relevant column in the csv file depends on the wave direction. For wave direction 0 the relevant column is DF_X/H (base shear magnitude in X direction) while for direction 90 the relevant column is DF_Y/H (base shear magnitude in Y direction). For wave direction 45 the two columns must be vectorially combined.

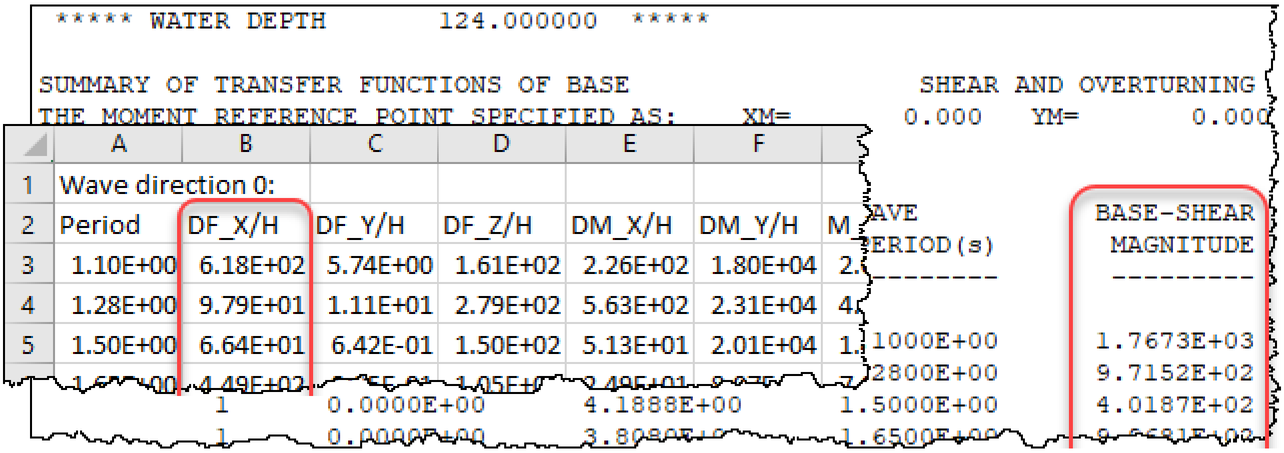


Figure 13-3 Transfer functions – the dynamic reactions (left) and sum of loads (right)

Comparison of transfer functions from Wajac and Sestra gives dynamic amplification factors (DAF) at base level. The graph below presents static and dynamic base shears for wave directions 0 and 90. The horizontal axis is the wave period in seconds. At the red dotted peak, the DAF is 19. This is base shear in direction 90 (Y) for period 2.96 that corresponds to the first bending mode in Y.

The graph of the transfer functions may be used to revise the choice of wave frequencies.

The DAFs extracted from the graph in Figure 13-4 below are valid at base level, i.e., they can be used to factorise static results for the nodes and beams close to the seabed. To get DAFs at various levels and not only at the bottom one must do a quasi-static (static frequency domain) analysis in Sestra and compare this with the dynamic analysis. This is briefly explained in section **13.2**.

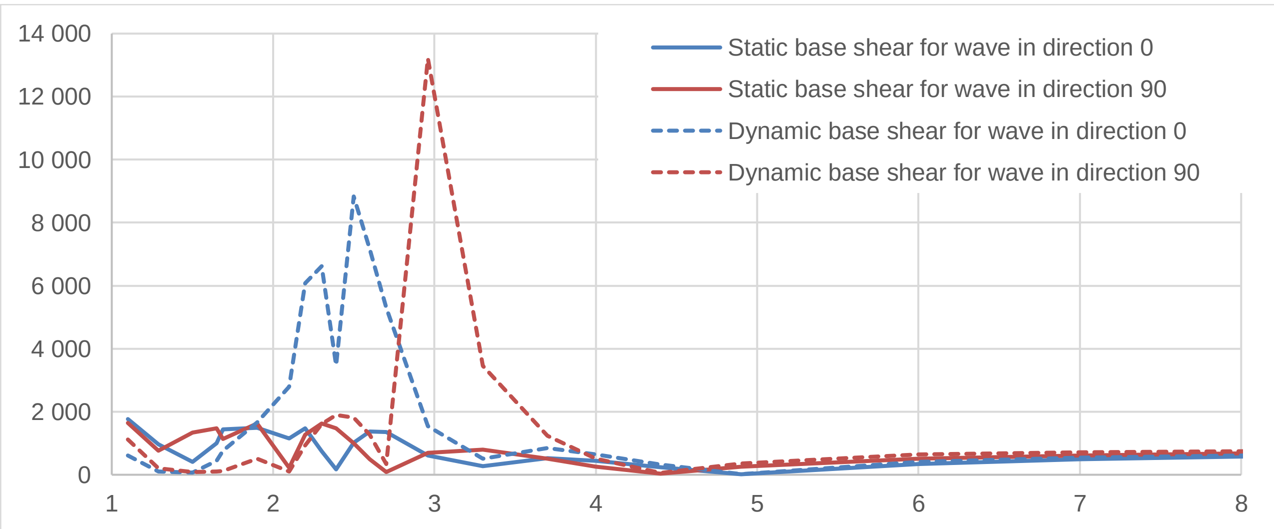


Figure 13-4 Transfer functions – comparing dynamic and static base shear responses

13.2 Dynamic Amplification Factors (DAFs) at Various Elevations

As explained in section **13.1**, the DAF at the base of the jacket is found by comparing the sum of loads computed by Wajac with the dynamic reactions computed by Sestra. To compute the DAFs at higher elevations of the jacket the results of a dynamic analysis in Sestra must be compared with the results of a static analysis in Sestra. A static analysis is required since Wajac cannot present the sum of wave loads from top and down to a certain elevation. For a chosen elevation (horizontal section through the jacket) the forces and moments must be summed for all intersected members. The quotient between such a sum for the dynamic analysis results and the static analysis results represents the DAF at that elevation. The elevation should be just above or just below a horizontal bracing.

Obviously, the above procedure for computing DAFs for a large model requires a certain amount of work.

Having found the DAFs at various elevations you may perform a deterministic fatigue analysis (which is based on a static analysis in Sestra) and account for dynamic amplifications by multiplying the structural response at the various elevations with the corresponding DAFs. This may for instance be done by multiplying the stress concentration factors (SCFs) in the fatigue analysis with the DAFs.

13.3 Stochastic Fatigue Analysis for Beam Joint

The stochastic fatigue analysis in Framework may now be performed based on the dynamic frequency domain results from Sestra. The activity **Framework_stochastic_fatigue** runs Framework. Only the jacket (i.e., not the deck structure) is included in the fatigue analysis. The scatter diagram shown in **Figure 13-5** is together with a cosine-squared wave spreading function, Pierson- Moskowitz wave spectrum, and Efthymiou stress concentration factors input to Framework.

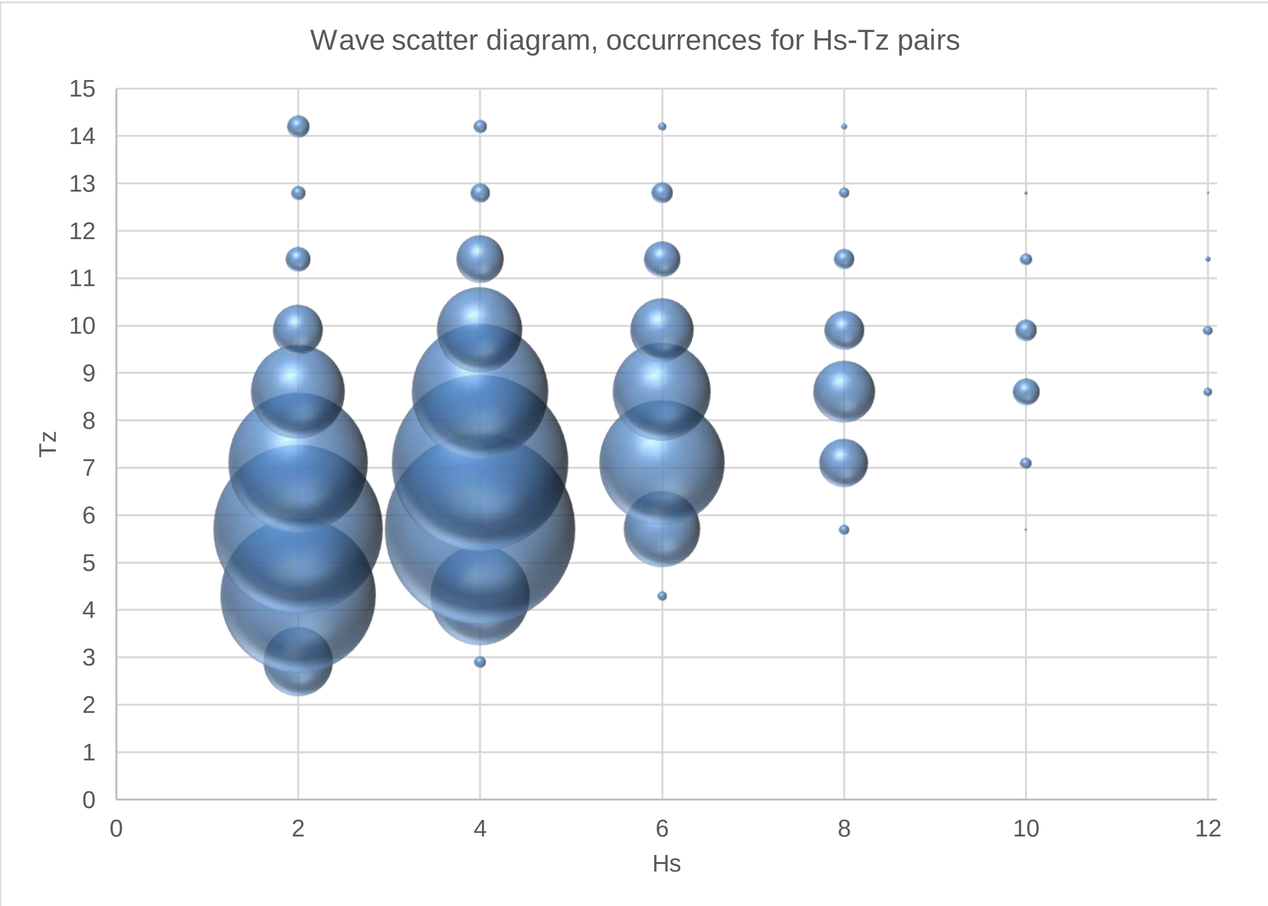


Figure 13-5 Scatter diagram

Figure 13-6 below is a screen dump of the fatigue results. The values are usage factors with a target fatigue life of 20 years which means that a usage factor of 1 gives a fatigue life of 20 years while a usage factor of 2 gives a fatigue life of 20/2=10 years.

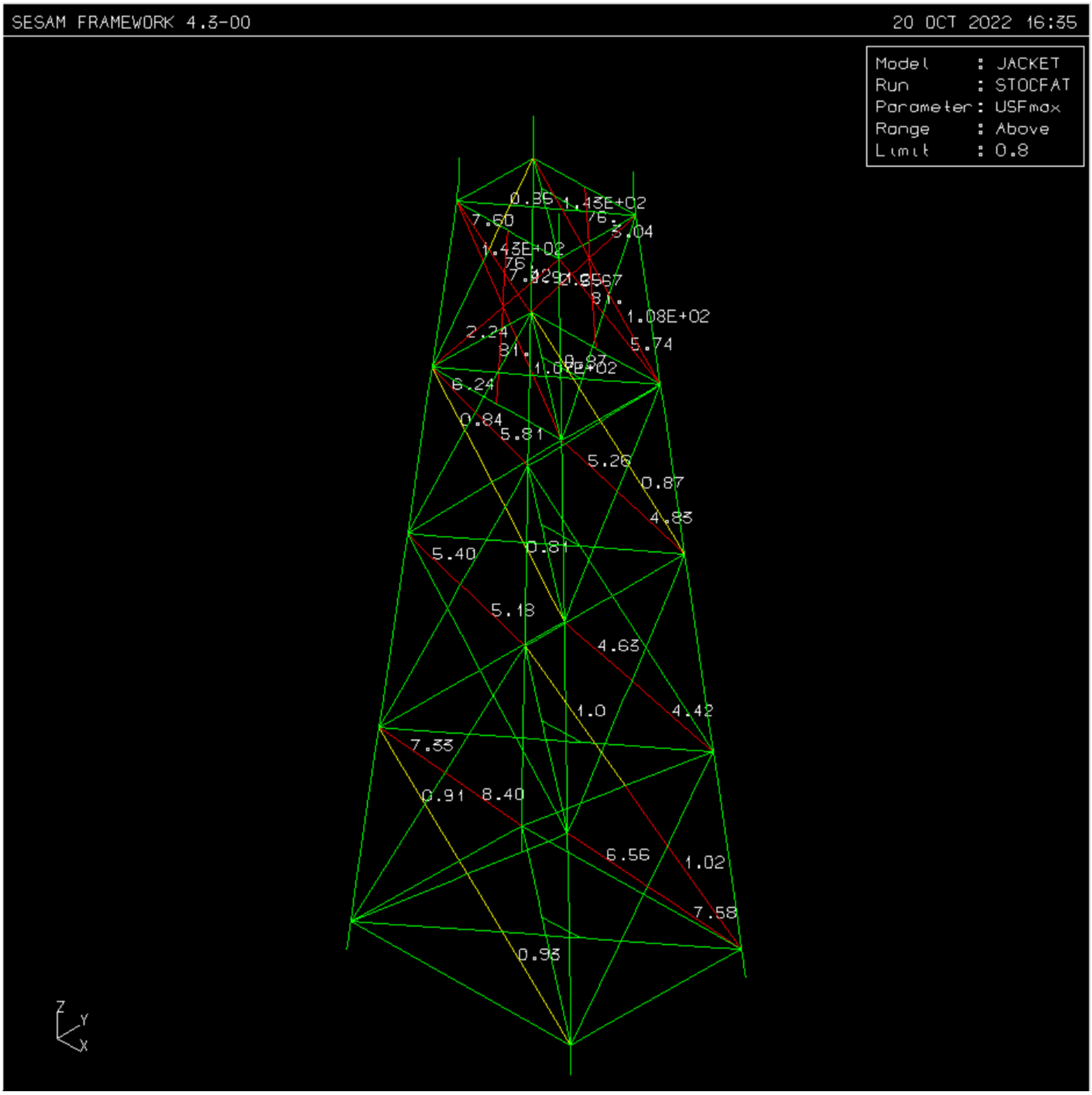


Figure 13-6 Fatigue check result *Max Usage Factor* from stochastic fatigue

14 Stochastic Fatigue Analysis of Shell Model of Joints

The sequence for stochastic fatigue analysis for shell model of joints is shown below in *Flow Chart* view.

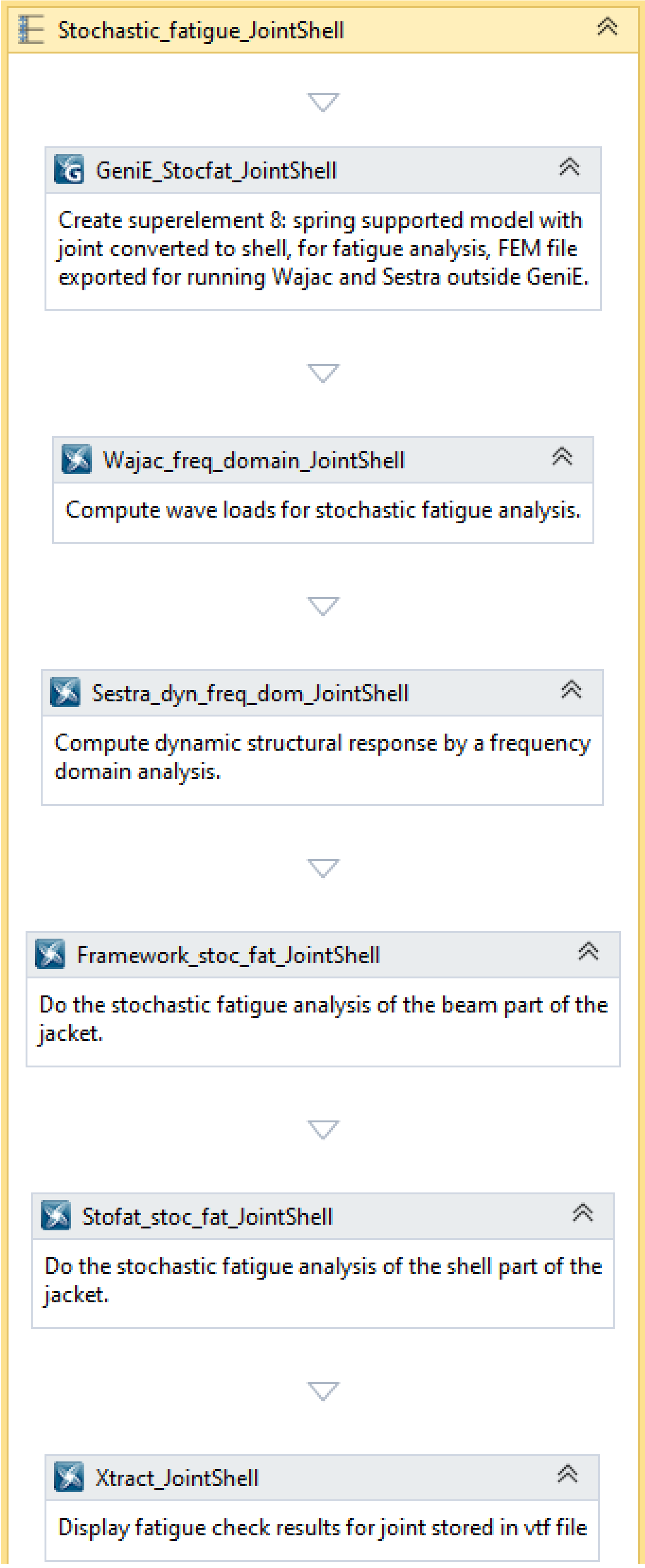


Figure 14-1 The sequence for stochastic fatigue analysis of shell model of joints

The stochastic fatigue analysis method for shell model of joints is highlighted with red frames and yellow text highlighting in Figure 14-2 below.

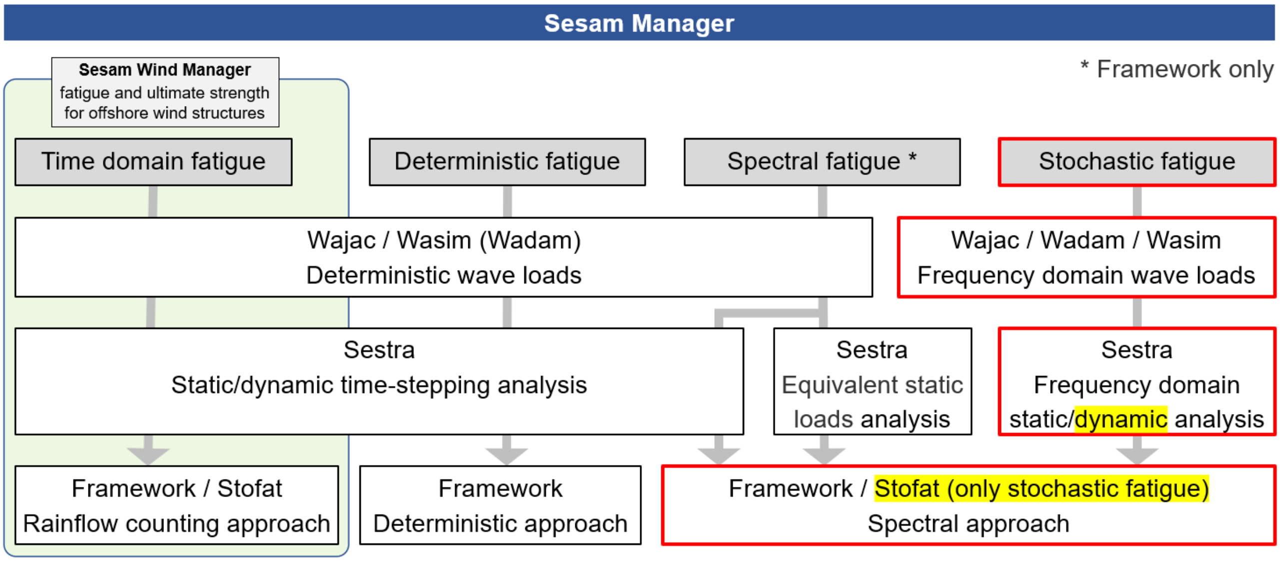


Figure 14-2 Highlighting the processes in Sesam for stochastic fatigue analysis of shell model of joints

Two of the tubular joints of the jacket are converted into shell models with FE meshes consisting of 4-node quadrilateral elements (and some 3-node triangular elements). One of the joints, Jt3, a multiplanar joint including a cone, is positioned at the middle of one of the legs. The other joint, Jt41, is a T-joint between two braces.

A stochastic fatigue analysis method is used for the weld zones. The method is in principle the same as the one of section **13** only that Stofat is used for shell/plate elements rather than Framework for beam elements.

The stochastic fatigue analysis is contained in the sequence **Stochastic_fatigue_JointShell**.

The activity **GeniE_Stocfat_JointShell** imports and modifies the workspace of the activity GeniE_linearise_pilesoil. The model is modified by converting two tubular joints from beam models to shell models. There is an automatic feature in GeniE for performing this conversion. In interactive mode, right-click a joint and select *Convert joint(s) to shells*.

In the *Convert joint(s) to shells* dialog, see Figure 14-3, there are several options among which are:

- How to handle the converted beams, in this case *Make Converted Beams Nonstructural* is selected so that they contribute with wave and buoyancy loads but not with stiffness.
- *Add rigid link* causes so-called support rigid links to be added. This involves that the ring of nodes at each end of the shell are linearly coupled to the beam node thereby properly transferring forces and moments.
- Mesh density and mesh option for the shells. The mesh option involves that *Advancing Front Quad Mesher* is used.
- Where to split chords and braces thereby deciding lengths of the chords and braces to be converted from beam model to shell model. In this case, the lengths of the braces are set to approximately three times their diameters measured from the chord-brace intersections. A study has shown that the stresses in the chord-brace intersection zones depend on the lengths converted to shell with convergence when the lengths approach three times the diameters.
- The tabs *Mesh Refinement Zones* and *Mesh Transition Zones*, see Figure 14-4, offer options for determining widths of and mesh densities for these two zones. The refinement zones are next to the chord-brace intersections while the transition zones connect the refinement zones with the rest of the shell part of the joint.

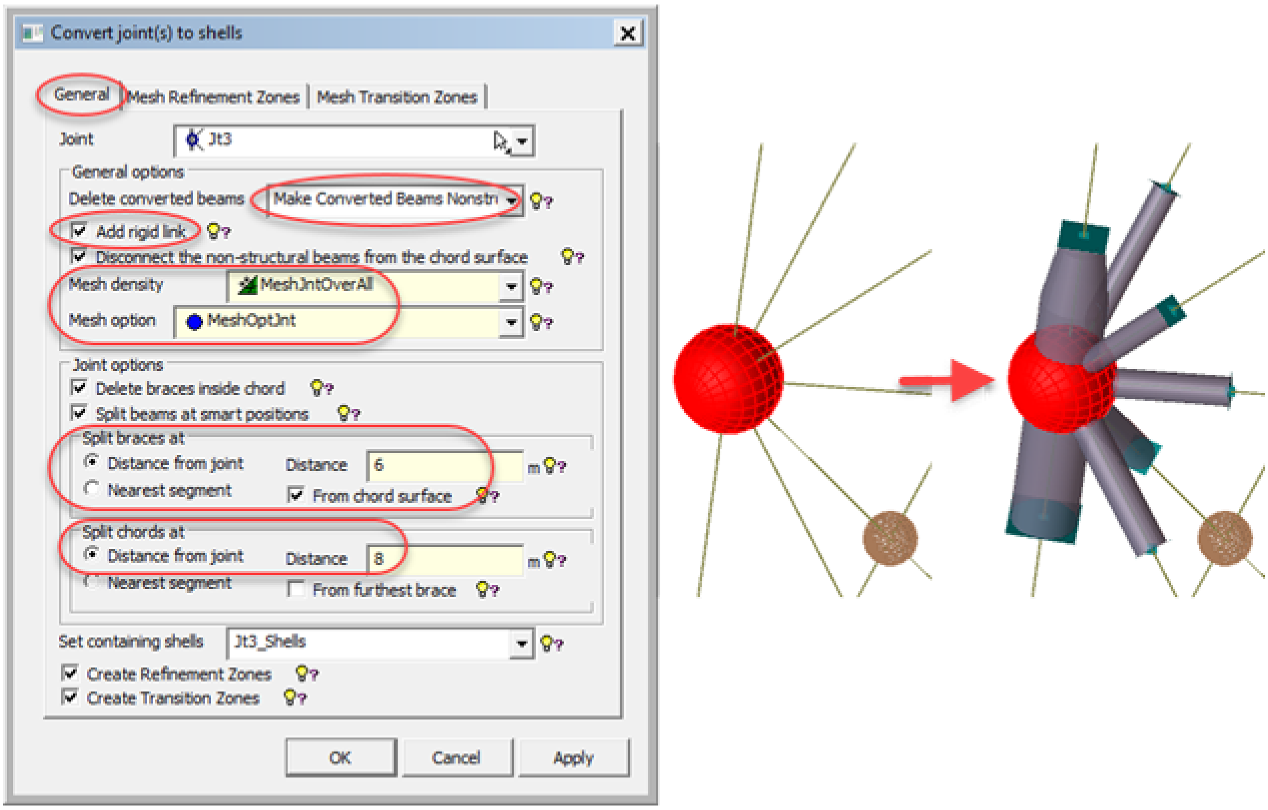


Figure 14-3 Dialog for automatic conversion of beam model of joint to shell model

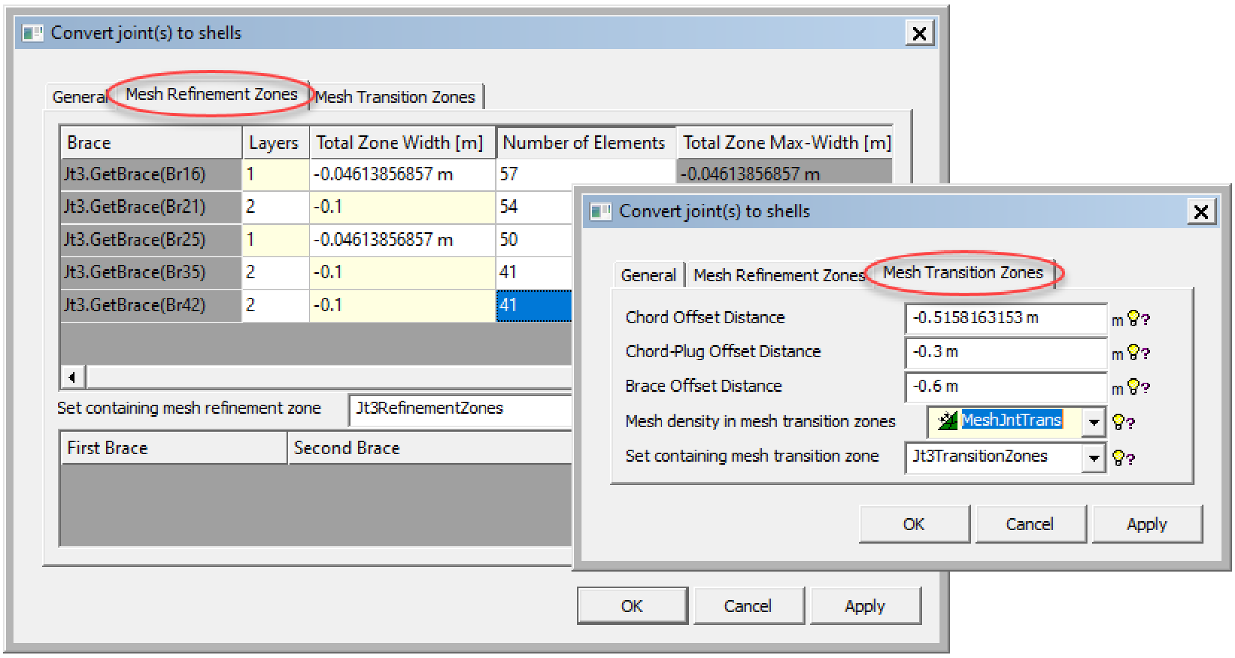


Figure 14-4 Dialog for automatic conversion of beam model of joint to shell model – additional tabs

Figure 14-5 below shows a part of the FE mesh for the joint Jt3. The mesh density is approximately 0.06 m which is in the proper range for shell fatigue analysis given that the thickness of the chord is 0.09 m and the thicknesses for the braces are 0.03 m. In the case of joint Jt41 the chord and brace thicknesses are both 0.03 m so a mesh density of approximately 0.03 m is used.

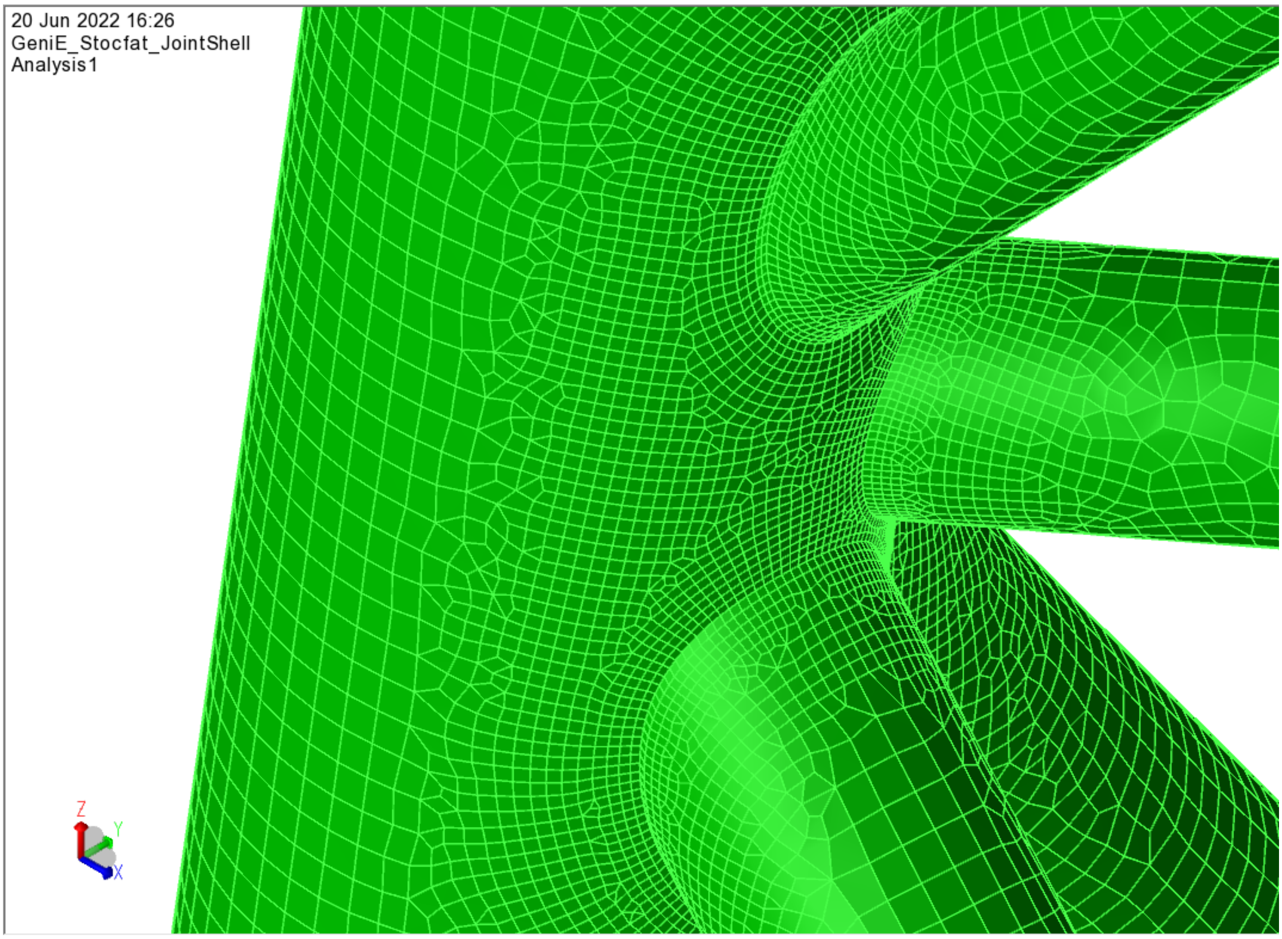


Figure 14-5 FE mesh for joint Jt3

The model is exported (T8.FEM, i.e., superelement 8) since both Wajac and Sestra will be run from Sesam Manager rather than from GeniE. The analysis activity creating the model is named Analysis1. After running the GeniE activity ensure that the file T8.FEM is found in the repository.

Run Wajac for computing wave transfer functions and added mass. The execution is found in the activity named **Wajac_freq_domain_JointShell**. The wave periods are the same as in section **13**.

Ensure that the files L8.FEM and S8.FEM are produced by the Wajac run and copied (by Sesam Manager) to the repository.

Now run the Sestra forced response dynamic analysis using the direct frequency response method. This is activity **Sestra_dyn_freq_dom_JointShell**. This analysis is the same as in section **13** but as the model now is much bigger the analysis time is 10-20 minutes or more depending on the computer. Ensure that the results file R8.SIN is found in the repository after the Sestra execution.

The same stochastic fatigue analysis in Framework as in section **13** is performed for the part of the jacket not converted to a shell model. This is merely for verification purposes; the results should be similar to the results in section **13.3**.The activity **Framework_stoc_fat_JointShell** runs Framework. Figure 14-6 below is a screen dump of the fatigue results. There are gaps where the joints are converted to shell models.

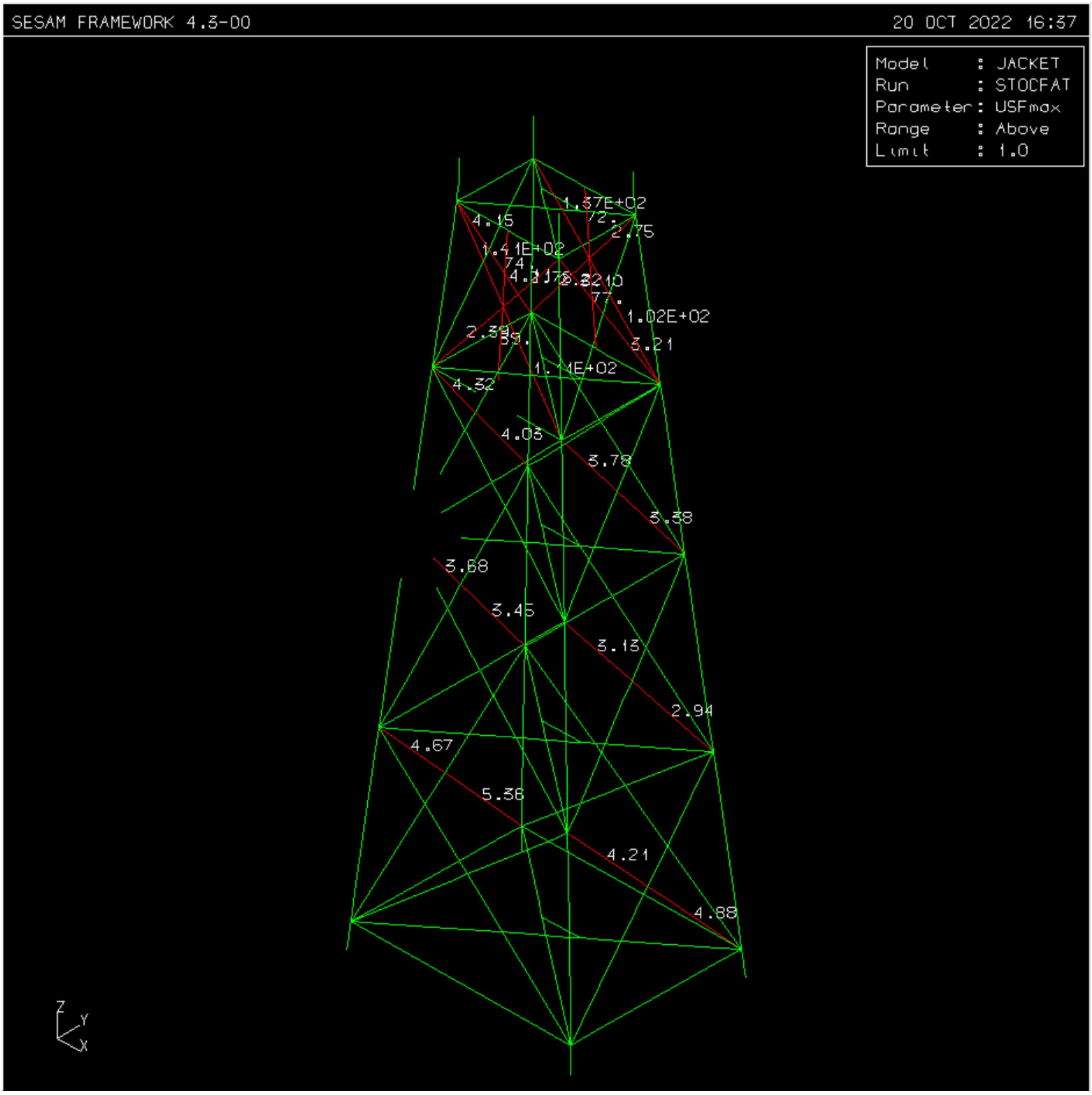


Figure 14-6 Fatigue check result *Max Usage Factor* from stochastic fatigue with two joints converted to shell

The stochastic fatigue analysis in Stofat of the joint shell models is performed by the activity **Stofat_stoc_fat_JointShell**.

Two types of fatigue analysis are performed in Stofat:

- Element check
- Hotspot check

14.1 Element Check

Only the finely meshed weld zones (termed *Mesh Refinement Zones* in the GeniE conversion to shell process) for the two converted joints are included in the fatigue analysis. The joints and their fatigue results are:

- Jt41 – A few elements at the saddle position of the T-joint show usage factors just above 3, see results presented by Xtract – activity **Xtract_JointShell** – in Figure 14-7.
- Jt3 – A few elements at the crown position of one of the braces show usage factors less than 1, see Figure 14-8.

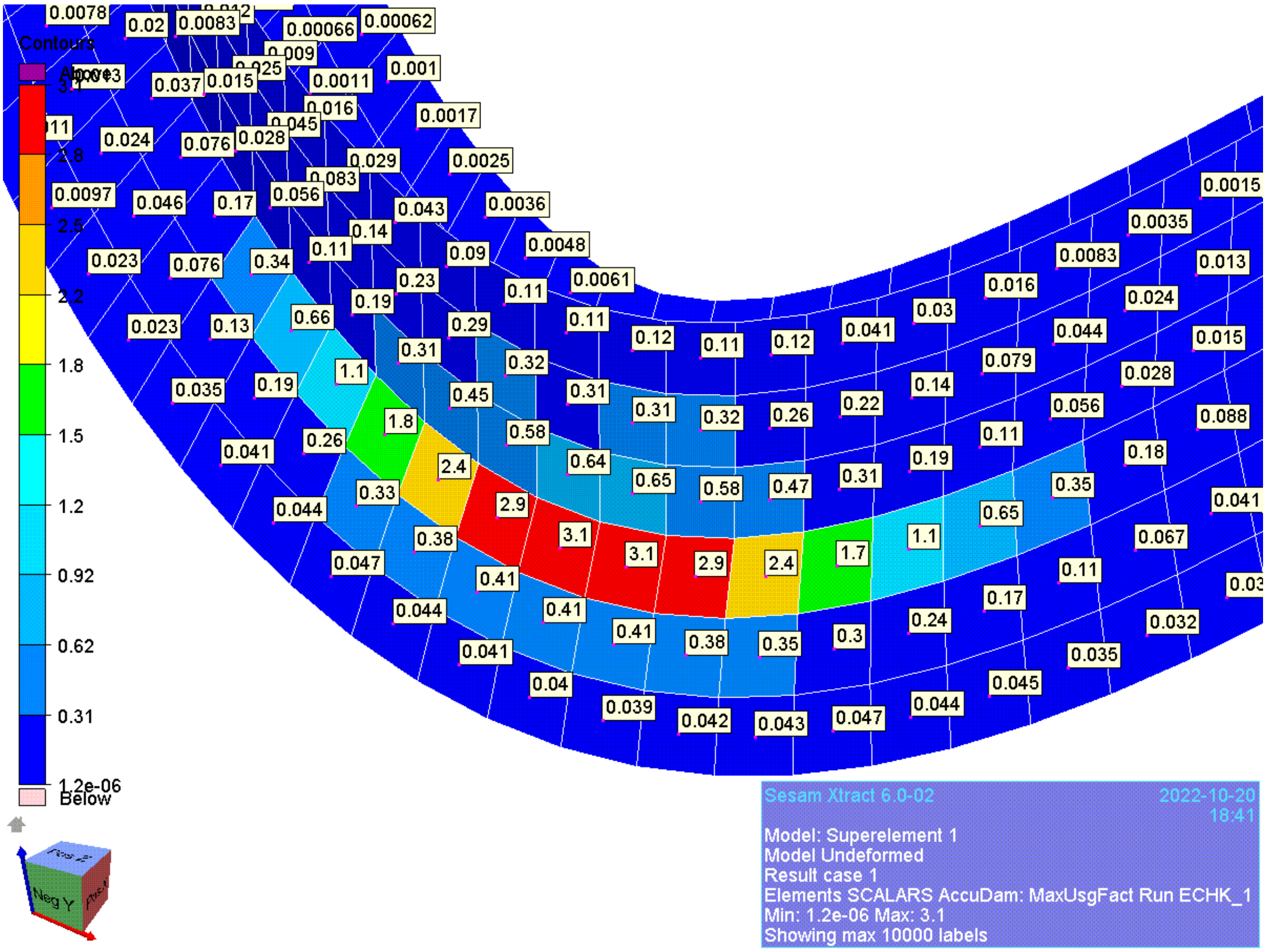


Figure 14-7 Element fatigue check results for joint Jt41 converted to shell

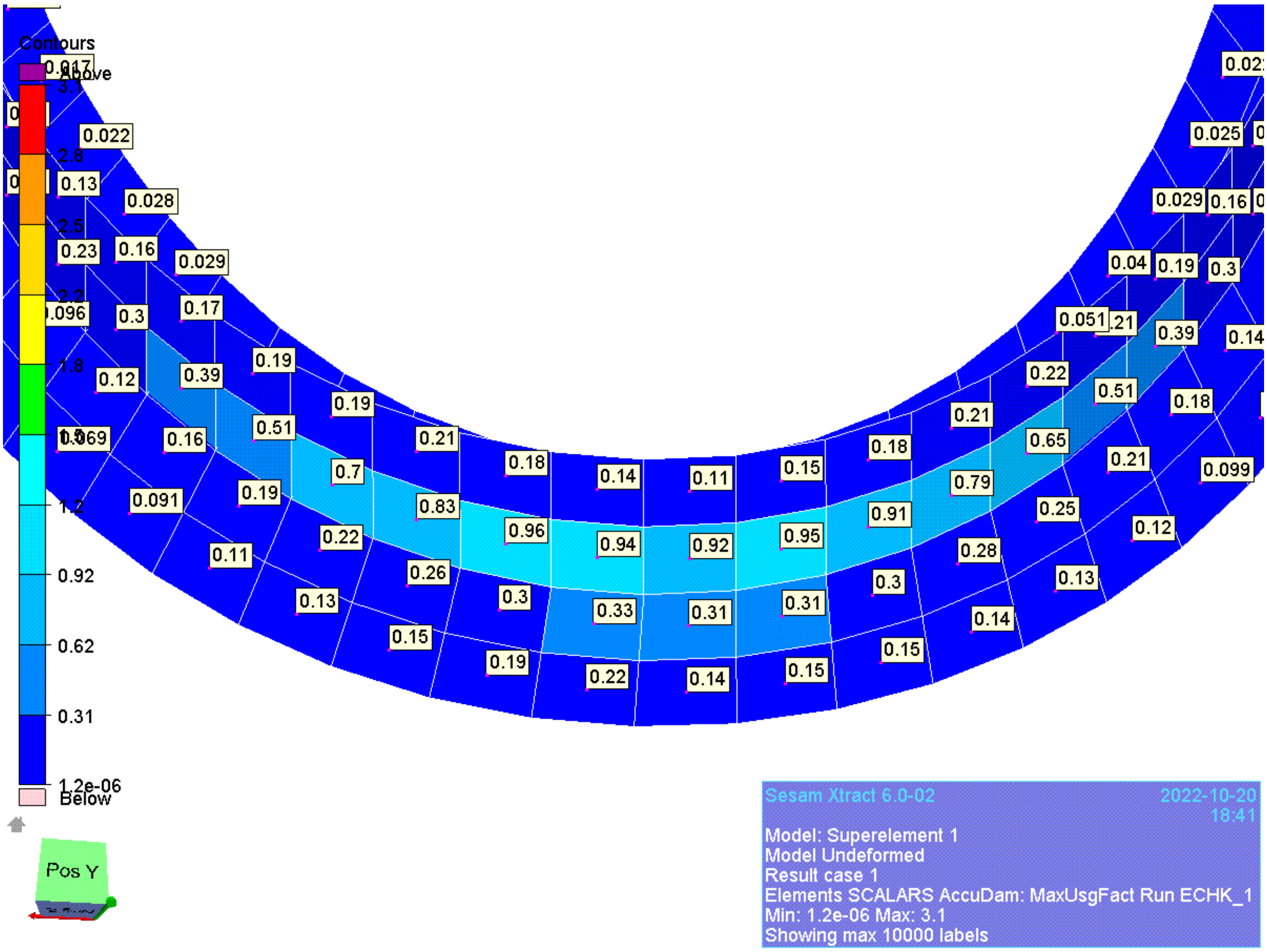


Figure 14-8 Element fatigue check results for joint Jt3 converted to shell

14.2 Hotspot Check

The element at the saddle position of the T-joint Jt41 where the highest usage factor of 3.1 is found is element number 6365 (the element number can be determined in e.g. Xtract). A hotspot in this element is checked to determine the fatigue damage more accurately. A hotspot check involves extrapolating stresses from two points to the hotspot (the stress in the hotspot itself is not used).

Xtract can be used to determine the coordinates of the extrapolation points, see Figure 14-9.

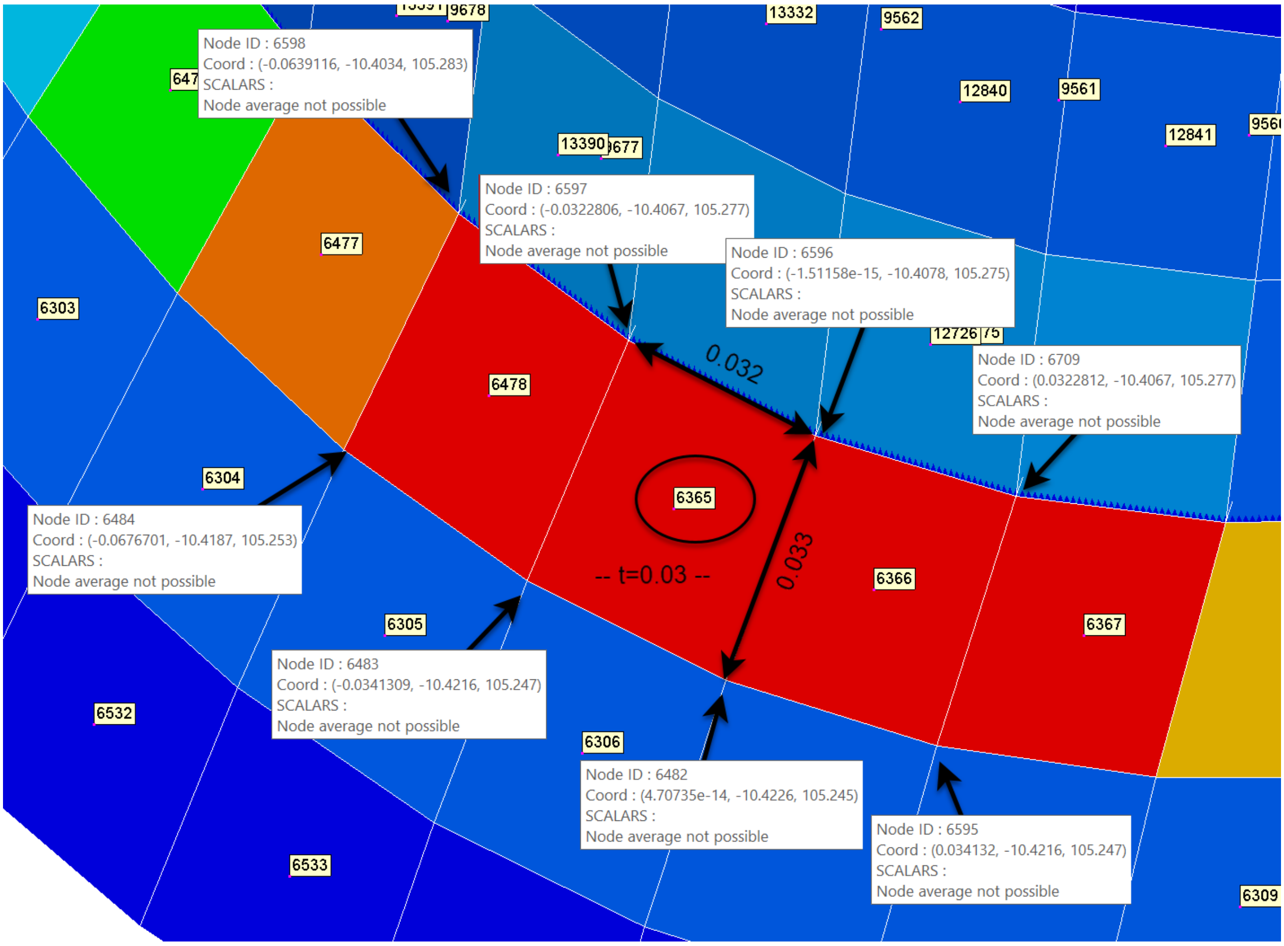


Figure 14-9 Determine coordinates of extrapolation points with the help of Xtract

Three extrapolations to the same point are defined, see Figure 14-10.

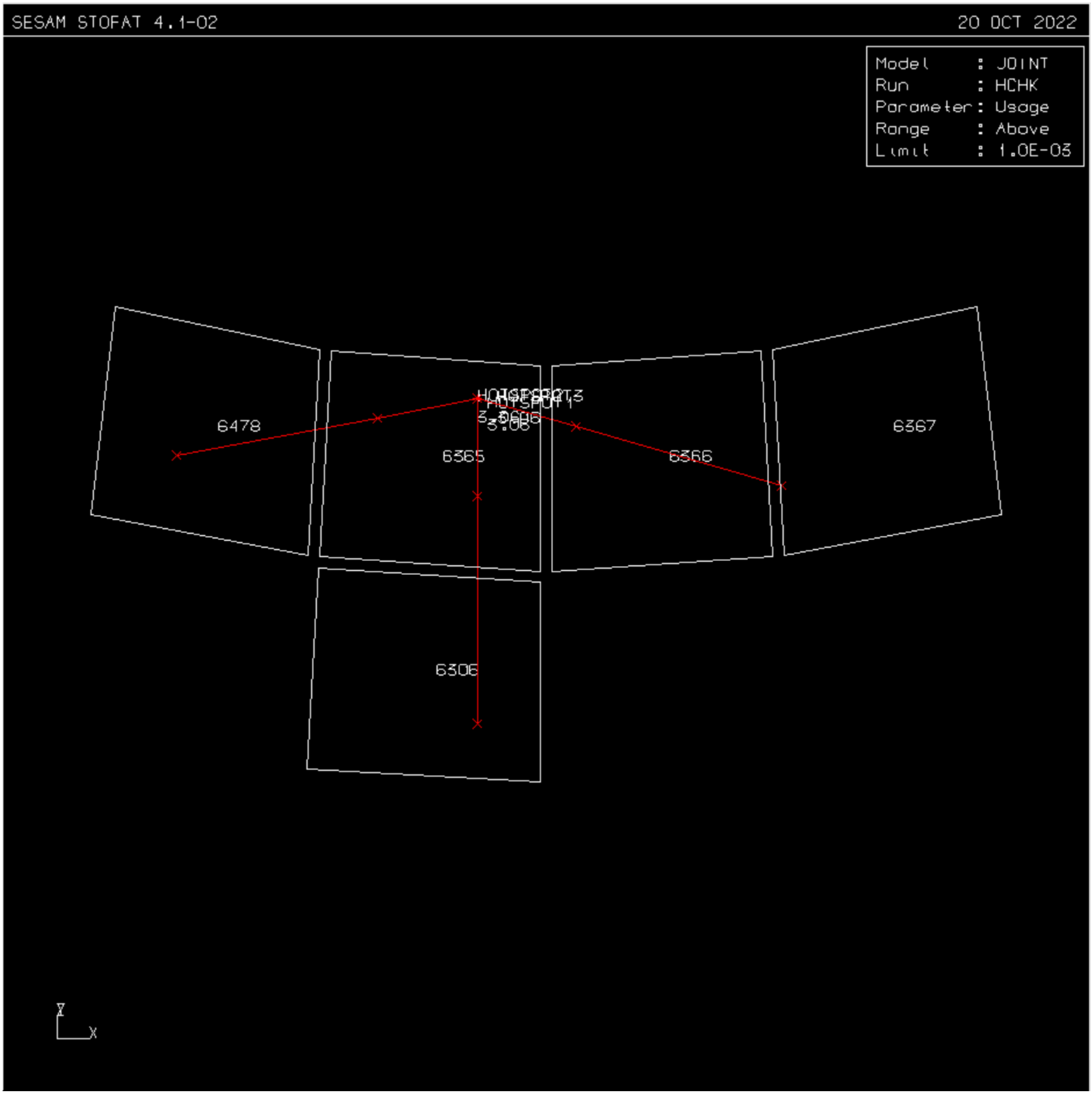


Figure 14-10 Three extrapolations to the same hotspot

Zooming in on element 6365, see Figure 14-11, shows that all three extrapolations give a usage factor of 3.06 which confirms the result of 3.1 from the element check.

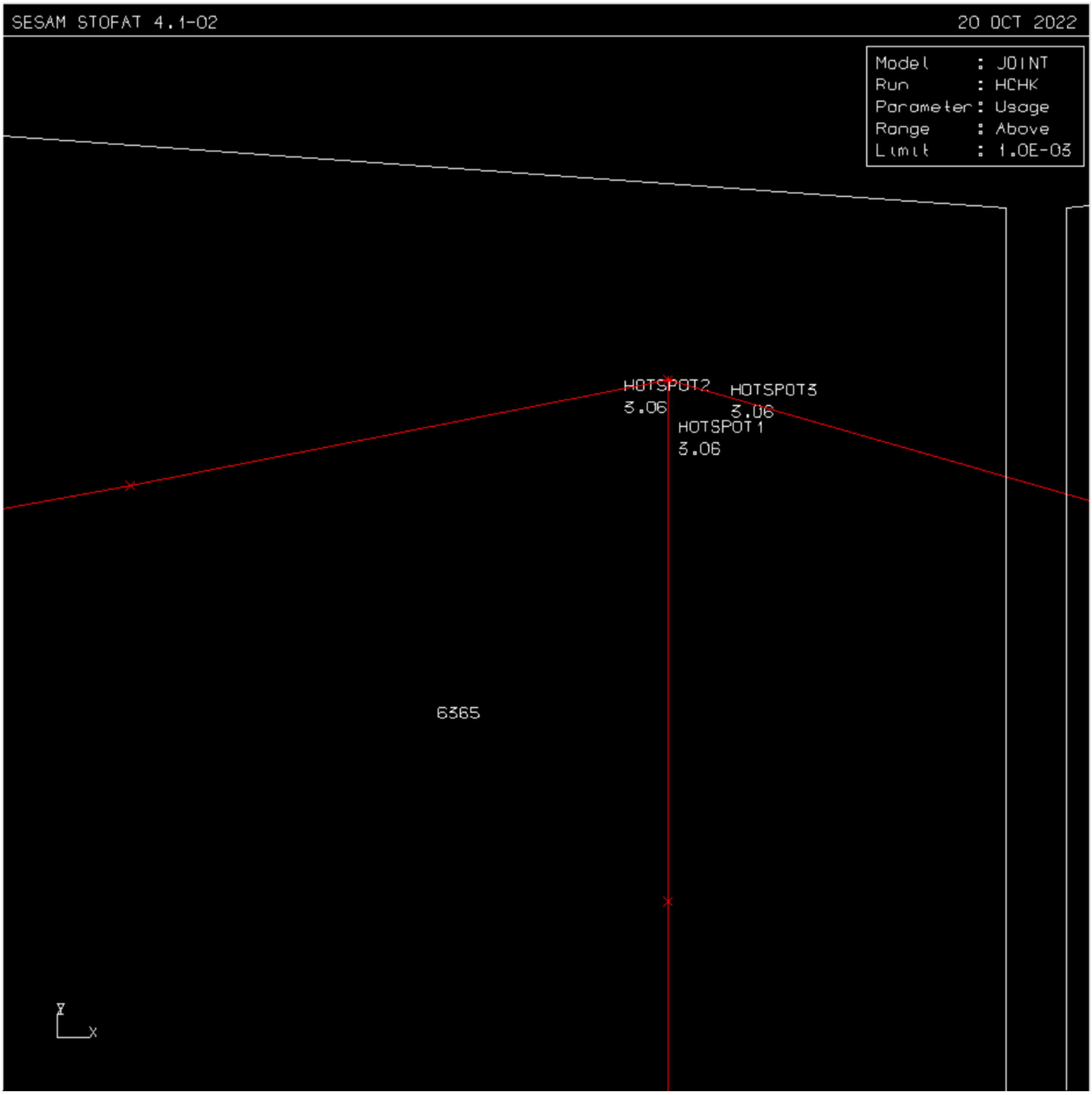


Figure 14-11 Hotspot fatigue check result from three extrapolations

15 Earthquake Analysis

The sequence for earthquake analysis is shown below in *Flow Chart* view.

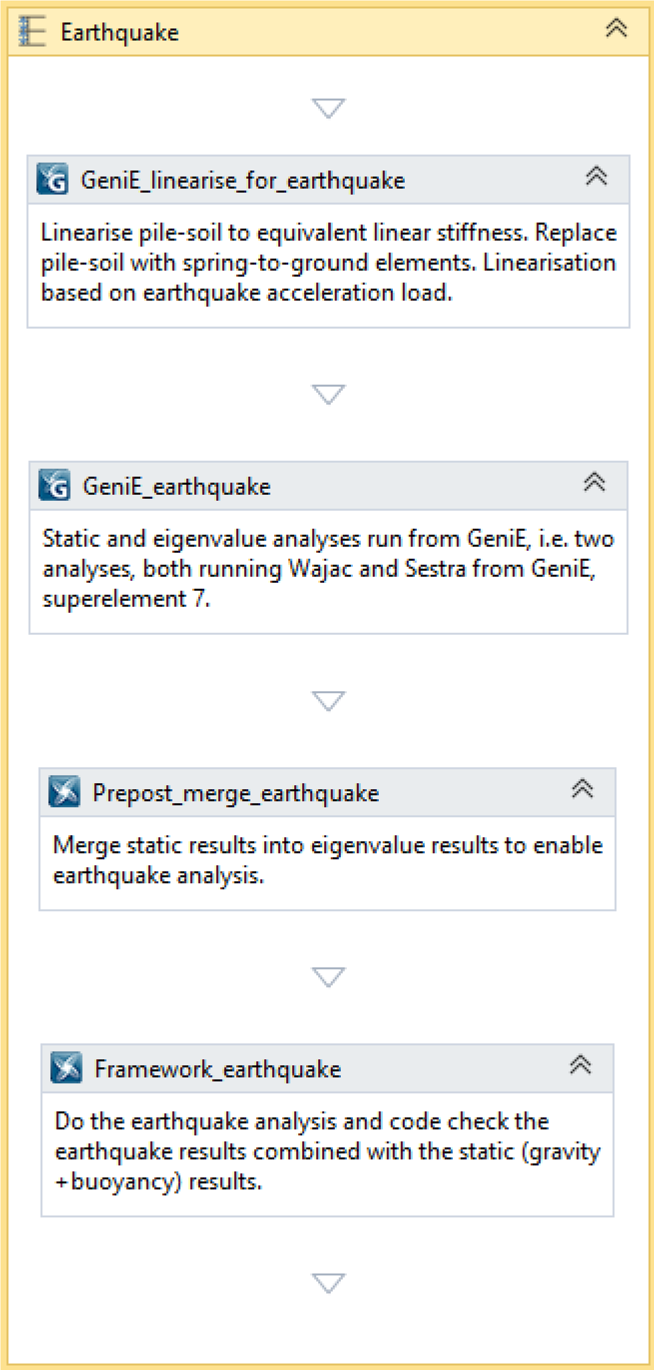


Figure 15-1 The sequence for earthquake analysis

An earthquake analysis is based on a merged results file from two analyses in Sestra:

- Static analysis of gravity and buoyancy loads
- Eigenvalue (free vibration) analysis

It is required that the two models are identical (same node numbering and coordinates and same element numbering).

The static analysis should preferably be merged into the eigenvalue analysis and not the other way around. (If the eigenvalue analysis is merged into the static analysis then the resulting file will contain mass representation from the static analysis. The print of modal masses in Framework will in such case present incorrect modal mass as fraction of total mass. Otherwise, the earthquake results will be correct.)

The non-linear pile-soil foundation must be replaced by linear spring-to-ground elements. The determination of the spring-to-ground elements used in the fatigue analyses, see section **6**, is based on a typical wave. More appropriate for an earthquake analysis is to base the linearisation on a typical horizontal acceleration. The activity **GeniE_linearise_for_earthquake** performs this linearisation based on a selected horizontal acceleration in the Y direction. The acceleration selected is 2.25 m/s^2.

Both the static and eigenvalue analyses are set up and run from the activity **GeniE_earthquake**. It imports and modifies the workspace of the activity GeniE_linearise_for_earthquake.

The static analysis activity is named Static and includes Wajac for computing buoyancy and Sestra for computing the static response to gravity (including equipments) and buoyancy.

The eigenvalue analysis activity is named Eigenvalue and includes Wajac for computing added mass and Sestra for the eigenvalue analysis. The equipment is represented as mass. This eigenvalue analysis must include computation of modal mass factors (modal earthquake excitation forces). An option for this is available through editing the eigenvalue analysis in the *Activity Monitor* of GeniE. The Multifront Lanczos eigenvalue solver is used to compute 20 modes.

A *PostExecuteScript* of Sesam Manager moves the two results files to the repository with the names StaticR7.SIN and EigenvalueR7.SIN. The *PostExecuteScript* looks like this:

```
Move(CurrentActivity.Workspace + @"Static\*R7.SIN", @"_repository\R7.SIN");
Move(@"_repository\R7.SIN", @"_repository\StaticR7.SIN");
Move(CurrentActivity.Workspace + @"Eigenvalue\*R7.SIN", @"_repository\R7.SIN");
Move(@"_repository\R7.SIN", @"_repository\EigenvalueR7.SIN");
```

The merging of the two results files in Prepost is done by activity **Prepost_merge_earthquake** setting *AnalysisType* to *MergeResultFiles*. The *InputFilePrefix* is set to Eigenvalue and the *MergeInputFilePrefix* is set to Static so as to merge the static results into the eigenvalue results. This run produces the file MERGED_EigenvalueR7.SIN. Ensure that this file exists. It should be bigger than EigenvalueR7.SIN since the static results have been merged into it.

Then run the earthquake analysis in Framework. This is activity **Framework_earthquake**. The earthquake spectrum for X and Y directions is shown in Figure 15-2 below. The same spectrum with a factor of 0.5 is used for the Z direction.

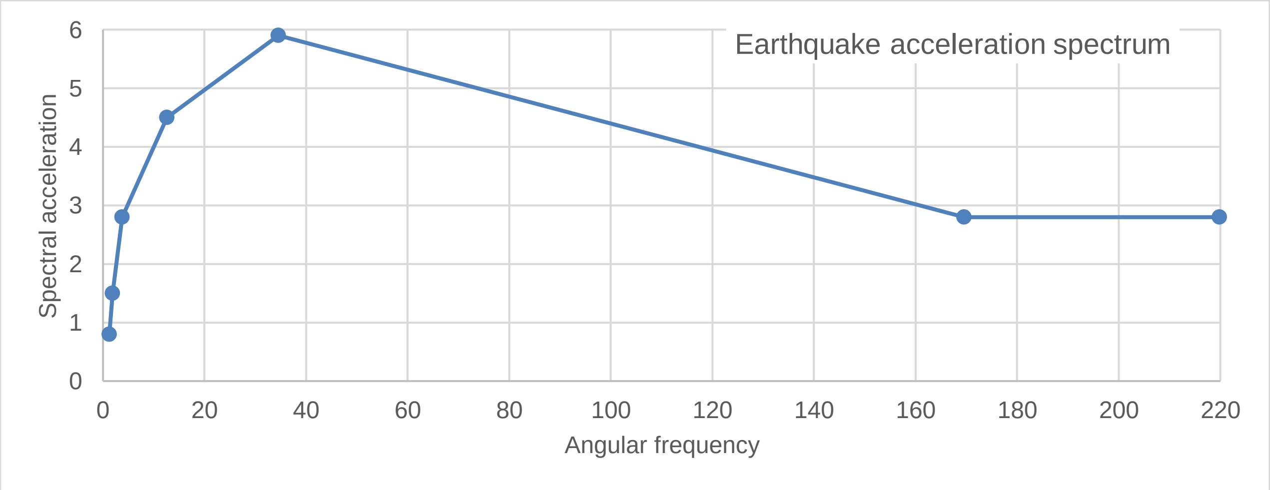


Figure 15-2 The earthquake acceleration spectrum

A screen dump of code check results (usage factors) is shown in Figure 15-3 below.

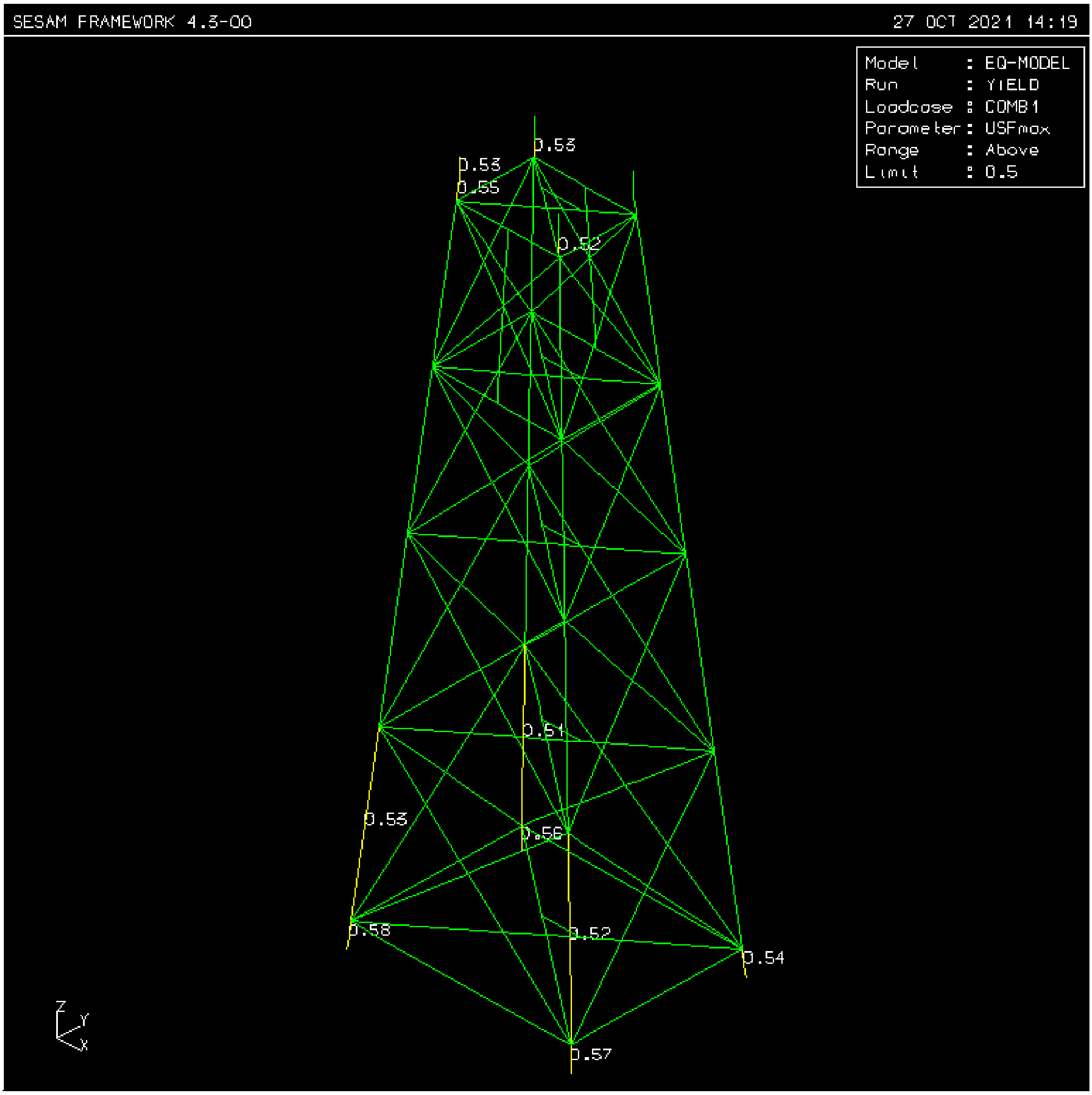


Figure 15-3 Code check result *USFmax* from code checking earthquake results

To verify the total mass participation (effective modal mass) in the earthquake analysis the individual modal mass participation factors and their fractions as well as accumulated fractions of the total mass are printed in Framework to a file named EarthquakeMass.LIS. The table in Figure 15-4 below is taken from this file. The mass participation fractions accumulated over 20 modes are 0.9922, 0.9912 and 0.7986, i.e., 99%, 99% and 80% for the X-, Y- and Z-directions, respectively.

The accumulated mass fractions may be taken into Excel and presented, see Figure 15-5. This graph shows that rather few modes contribute with nearly all mass participation. This is typical for jacket structures. The contribution to the Z-direction is from higher modes since the jacket is much stiffer in vertical than horizontal direction. To achieve a high percentage contribution for the vertical direction a high number of modes must be included.

```
                          Effective Modal Mass (EMM)
                          (modal load factors squared)


NOMENCLATURE:

Mode      Loadcase name of the mode shape
Period    Period of mode, T
EMM(X)    X-direction effective modal mass
EMM(Y)    Y-direction effective modal mass
EMM(Z)    Z-direction effective modal mass
DMM(X)    Fraction to total X mass
DMM(Y)    Fraction to total Y mass
DMM(Z)    Fraction to total Z mass
AMM(X)    X-direction acumulated mass fraction
AMM(Y)    Y-direction accumulated mass fraction
AMM(Z)    Z-direction accumulated mass fraction
```

Mode	Period	EMM(X)	EMM(Y)	EMM(Z)	DMM(X)	DMM(Y)	DMM(Z)	AMM(X)	AMM(Y)	AMM(Z)
1	2.953	1.46E-01	1.72E+04	6.31E-01	0.0000	0.7042	0.0000	0.0000	0.7042	0.0000
2	2.379	1.74E+04	1.53E-01	8.89E-02	0.7273	0.0000	0.0000	0.7273	0.7042	0.0000
3	1.294	4.44E-01	3.04E+01	3.00E-03	0.0000	0.0012	0.0000	0.7274	0.7055	0.0000
4	0.923	2.68E-05	6.51E+03	1.44E+00	0.0000	0.2671	0.0001	0.7274	0.9726	0.0001
5	0.858	6.10E+03	5.53E-04	1.02E+00	0.2556	0.0000	0.0001	0.9830	0.9726	0.0002
6	0.710	1.09E-01	4.56E+00	5.71E-03	0.0000	0.0002	0.0000	0.9830	0.9727	0.0002
7	0.551	2.21E-03	7.88E-01	6.87E+02	0.0000	0.0000	0.0394	0.9830	0.9728	0.0396
8	0.545	3.43E-04	2.84E-01	5.21E+01	0.0000	0.0000	0.0030	0.9830	0.9728	0.0426
9	0.531	7.52E-06	6.37E+01	7.68E+00	0.0000	0.0026	0.0004	0.9830	0.9754	0.0431
10	0.456	8.74E+00	4.08E+00	1.14E+04	0.0004	0.0002	0.6569	0.9834	0.9756	0.6999
11	0.445	9.58E+01	4.63E-01	7.07E+02	0.0040	0.0000	0.0406	0.9874	0.9756	0.7405
12	0.439	3.34E-02	7.99E-01	6.00E+00	0.0000	0.0000	0.0003	0.9874	0.9756	0.7409
13	0.431	2.29E-03	3.30E+02	2.74E+01	0.0000	0.0135	0.0016	0.9874	0.9891	0.7424
14	0.421	2.57E-04	4.63E+01	1.14E+00	0.0000	0.0019	0.0001	0.9874	0.9910	0.7425
15	0.413	6.49E-03	2.44E-01	1.17E+02	0.0000	0.0000	0.0067	0.9874	0.9910	0.7492
16	0.387	1.30E-04	2.41E-02	8.89E+00	0.0000	0.0000	0.0005	0.9874	0.9910	0.7497
17	0.373	1.11E-03	8.09E-02	1.74E-01	0.0000	0.0000	0.0000	0.9874	0.9910	0.7497
18	0.361	6.38E-03	2.50E+00	1.06E-04	0.0000	0.0001	0.0000	0.9874	0.9911	0.7497
19	0.358	1.13E+02	2.89E-03	1.22E+01	0.0048	0.0000	0.0007	0.9921	0.9911	0.7504
20	0.353	1.13E-02	5.38E-02	9.11E-02	0.0000	0.0000	0.0000	0.9921	0.9912	0.7504

Direction	X	Y	Z
Sum of EMM	2.3675E+04	2.4172E+04	1.3066E+04
Total Mass	2.3863E+04	2.4387E+04	1.7412E+04

Figure 15-4 Extract from EarthquakeMass.LIS printed by Framework

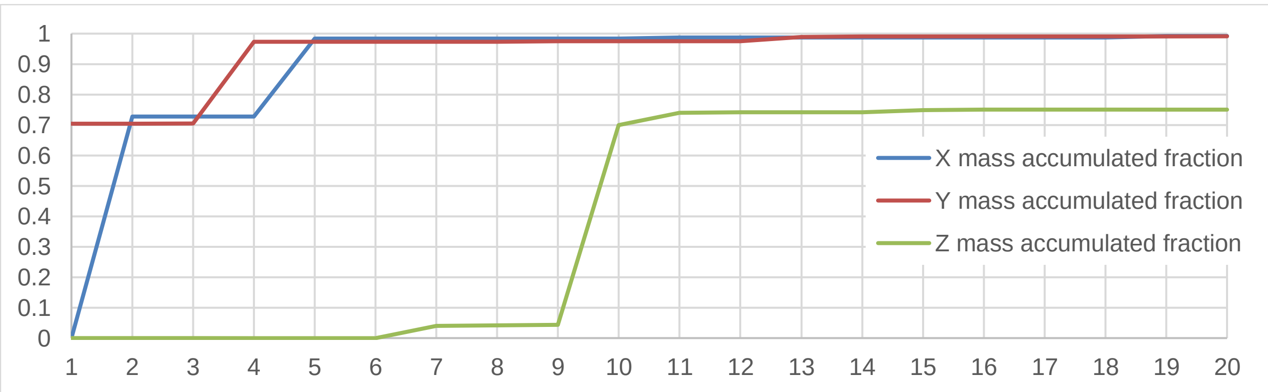


Figure 15-5 Accumulated mass fractions

The file EarthquakeBaseLoads.LIS, also printed by Framework, contains the earthquake base loads. The graph in Figure 15-6 below presents the absolute values of these base loads. As seen, high values correspond to jumps in the graph of mass accumulated fraction above.

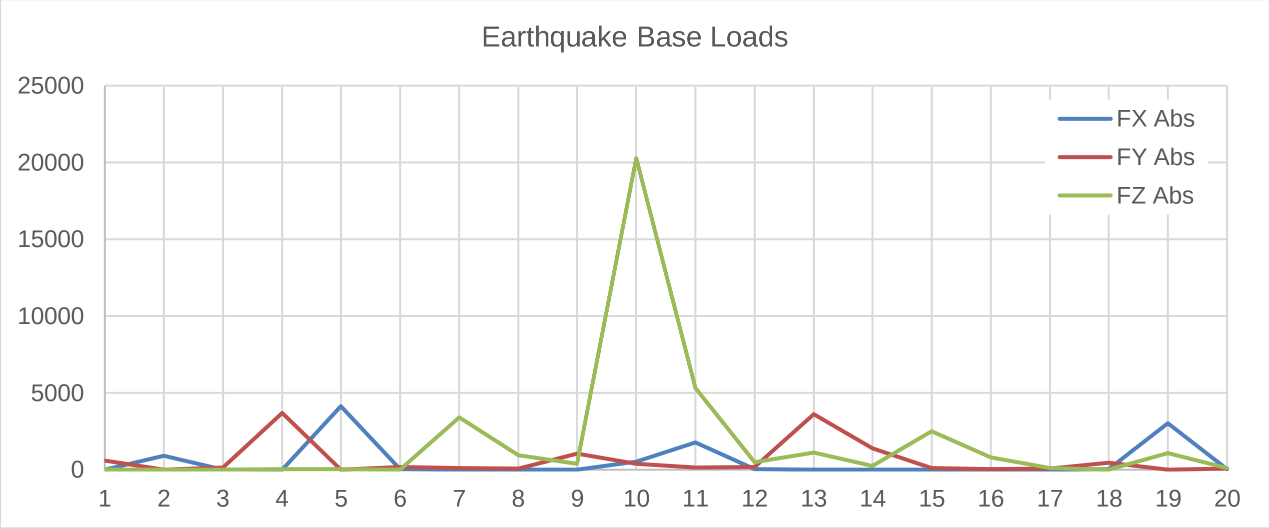


Figure 15-6 Earthquake base loads

16 Transportation Analysis

The model used for transportation analysis should be only the jacket without conductors and topside.

In a transportation analysis the jacket is resting on its side on a barge. The model must therefore be rotated. A way to find the rotation angle is as follows. Create two auxiliary beams as shown in Figure 16-1 below. The angle between them will be the rotation angle for the jacket to make it rest on its side. (In GeniE angles between intersecting beams may be labelled.) Deleted the two auxiliary beams after the rotation.

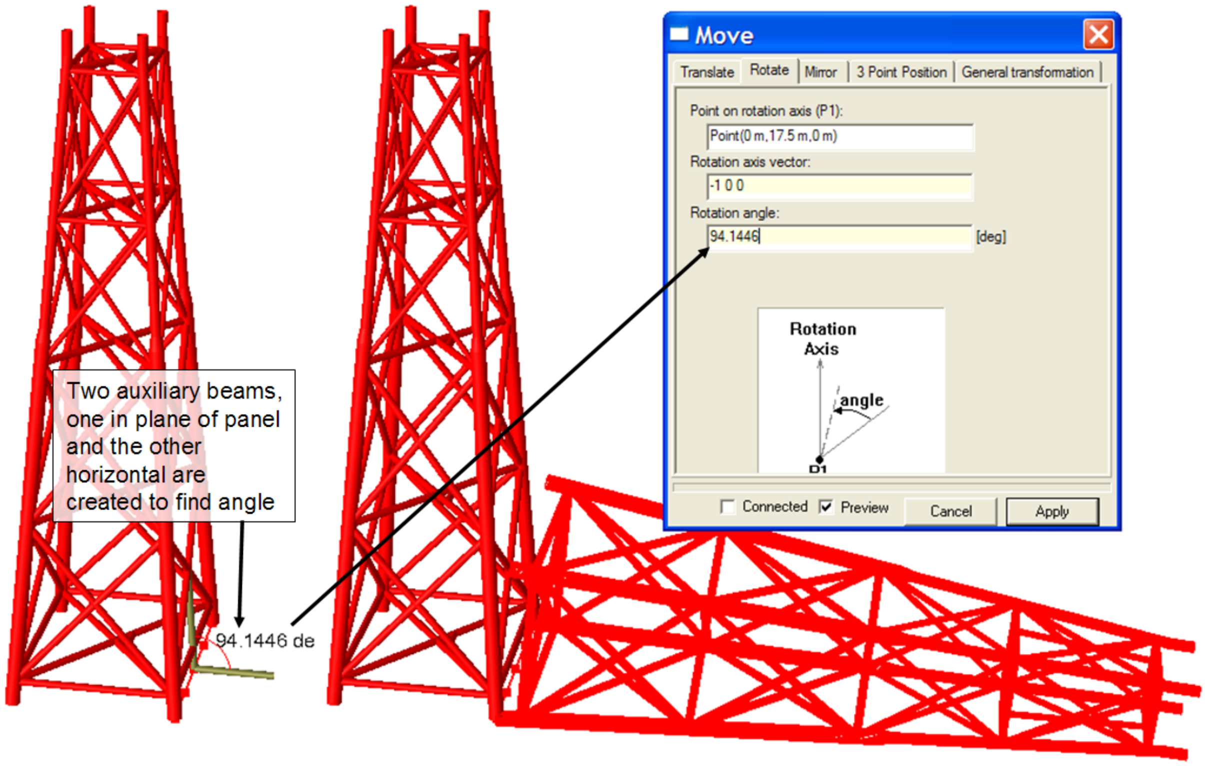


Figure 16-1 Determine the rotation angle and rotate the jacket so that it rests on its side

The jacket is resting on its side supported by compression-only stools. The sea-fastening is a set of inclined beams supporting the jacket sideways and is welded to the jacket after the jacket's deflection due to gravity. I.e., the sea-fastening is not loaded by the gravity of the jacket. The rolling of the barge then subjects the jacket to rotational accelerations which in turn exerts loads on the sea-fastening.

16.1 Simplified Transportation Analysis

Just for comparison purposes a simplified transportation analysis is performed by the activity **GeniE_simple_transportation**. In this analysis the effect of adding the sea-fastening after the jacket's deflection due to gravity is neglected. I.e., the sea-fastening will be loaded by the deflection due to gravity. The stools are modelled as compression-only trusses. Sestra is run under the control of GeniE.

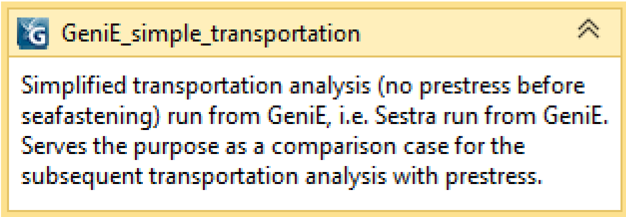


Figure 16-2 Activity for simplified transportation analysis in *Flow Chart* view

An example of code check results presentation is shown in Figure 16-3 below.

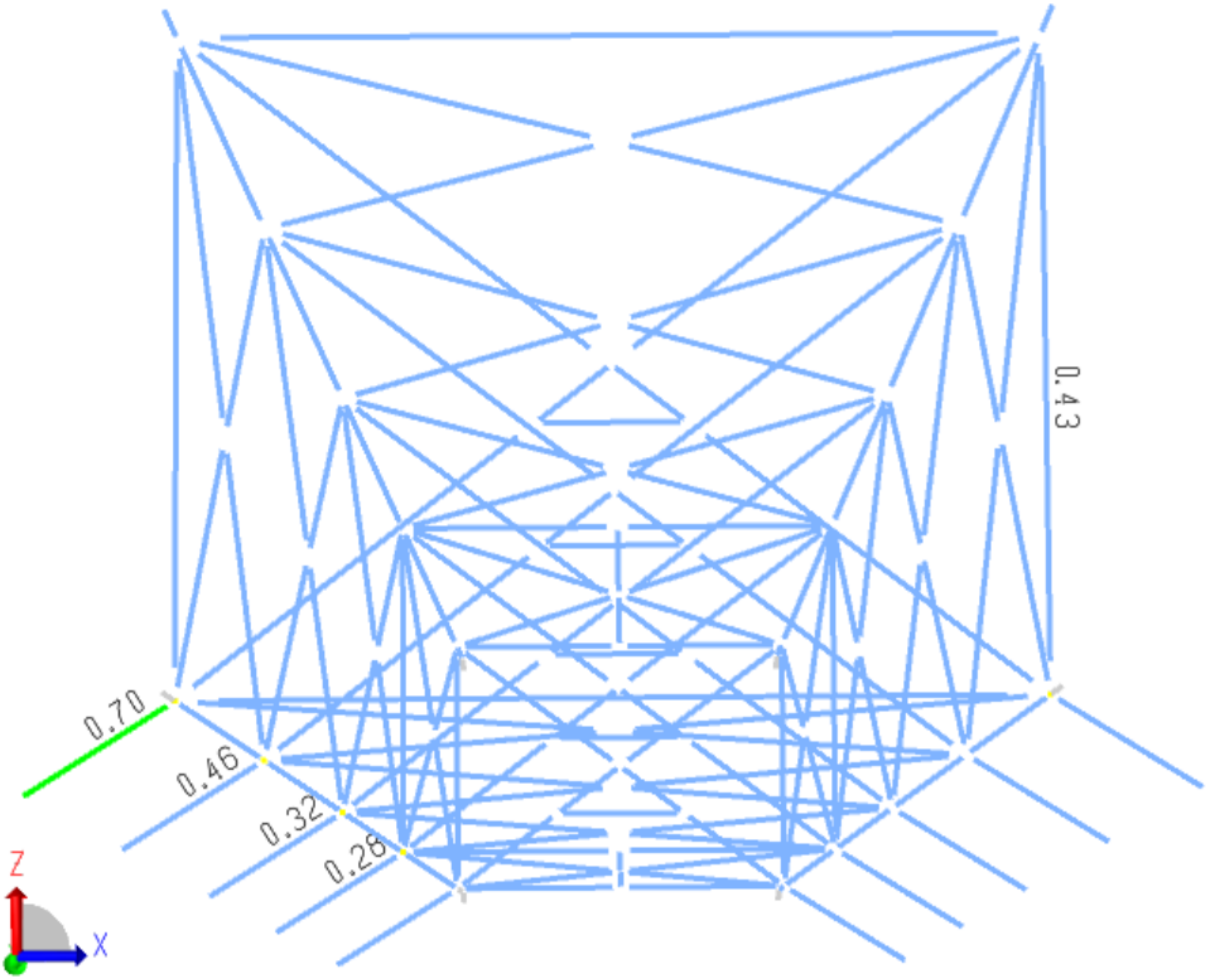


Figure 16-3 Code check result *UfTot* for *Worst Case (CC)* for simplified transportation analysis

16.2 Proper Transportation Analysis

A proper transportation analysis is then performed by the sequence **Transportation_with_prestress**.

The sequence for transportation analysis is shown below in *Flow Chart* view.

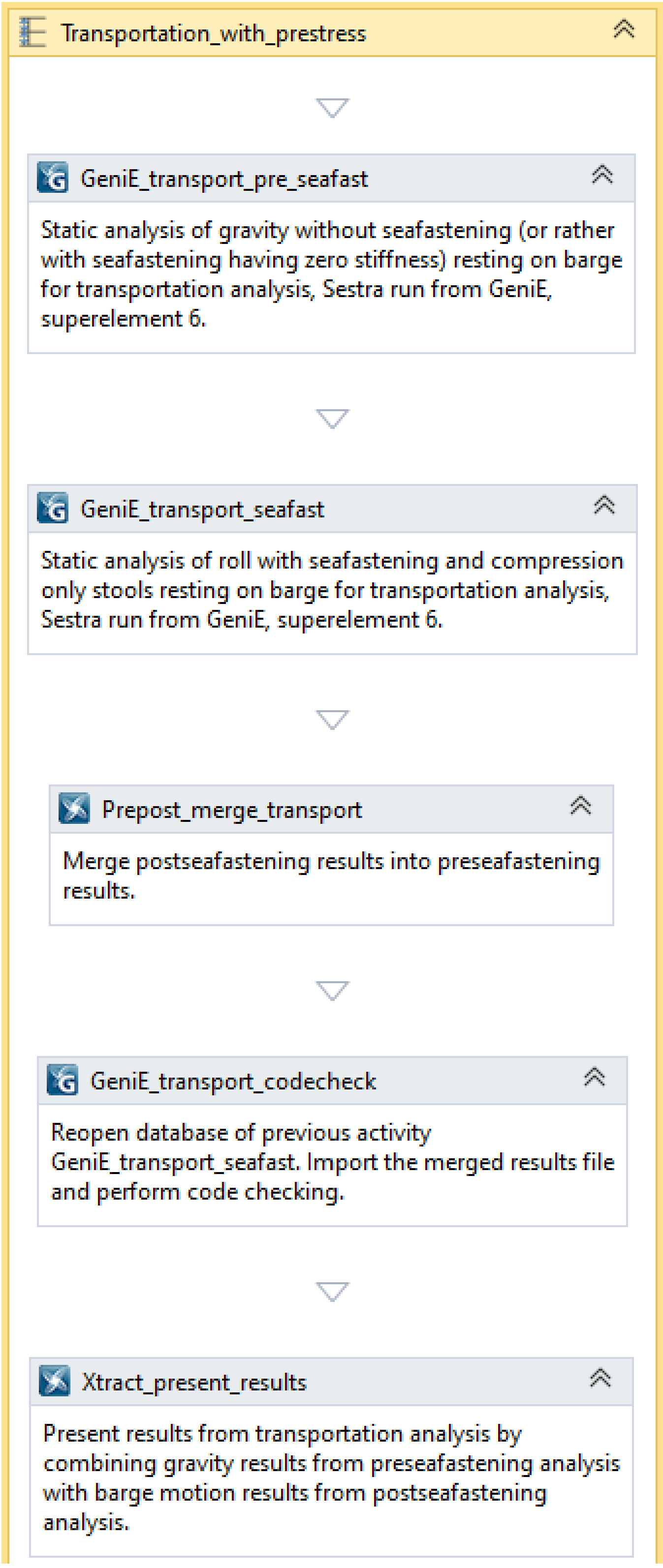


Figure 16-4 The sequence for transportation analysis

The loading of the sea-fastening only after the jacket's deflection due to gravity is achieved by the following procedure:

- First a static analysis with gravity load only is performed and with a very soft material (but not zero to avoid singular analysis) assigned to the sea-fastening. The stools are modelled as compression-only trusses. This is the activity **GeniE_transport_pre_seafast** running Sestra in the background to produce the file PreSeafastR6.SIN.
- Then a new static analysis with rotational load is performed, this time with normal material properties for the sea-fastening. This is the activity **GeniE_transport_seafast** running Sestra in the background to produce the file SeafastR6.SIN.
- These two result files are then merged using Prepost. This is activity **Prepost_merge_transport** with *AnalysisType* set to *MergeResultFiles*. The *InputFilePrefix* is set to PreSeafast and the *MergeInputFilePrefix* is set to Seafast so as to merge the latter into the former. This run produces the file MERGED_PreSeafastR6.SIN. Ensure that this file exists.

The activity **GeniE_transport_codecheck** reopens the workspace of the activity GeniE_transport_seafast. In Sesam Manager this is achieved by setting *Workspace* and *DatabaseName* equal to those of the activity GeniE_transport_seafast and *DatabaseStatus* to *Old*. The input to the activity imports the merged results file and performs a code checking of a combination of the gravity and roll result cases. An example of code check results presentation is shown in Figure 16-5 below.

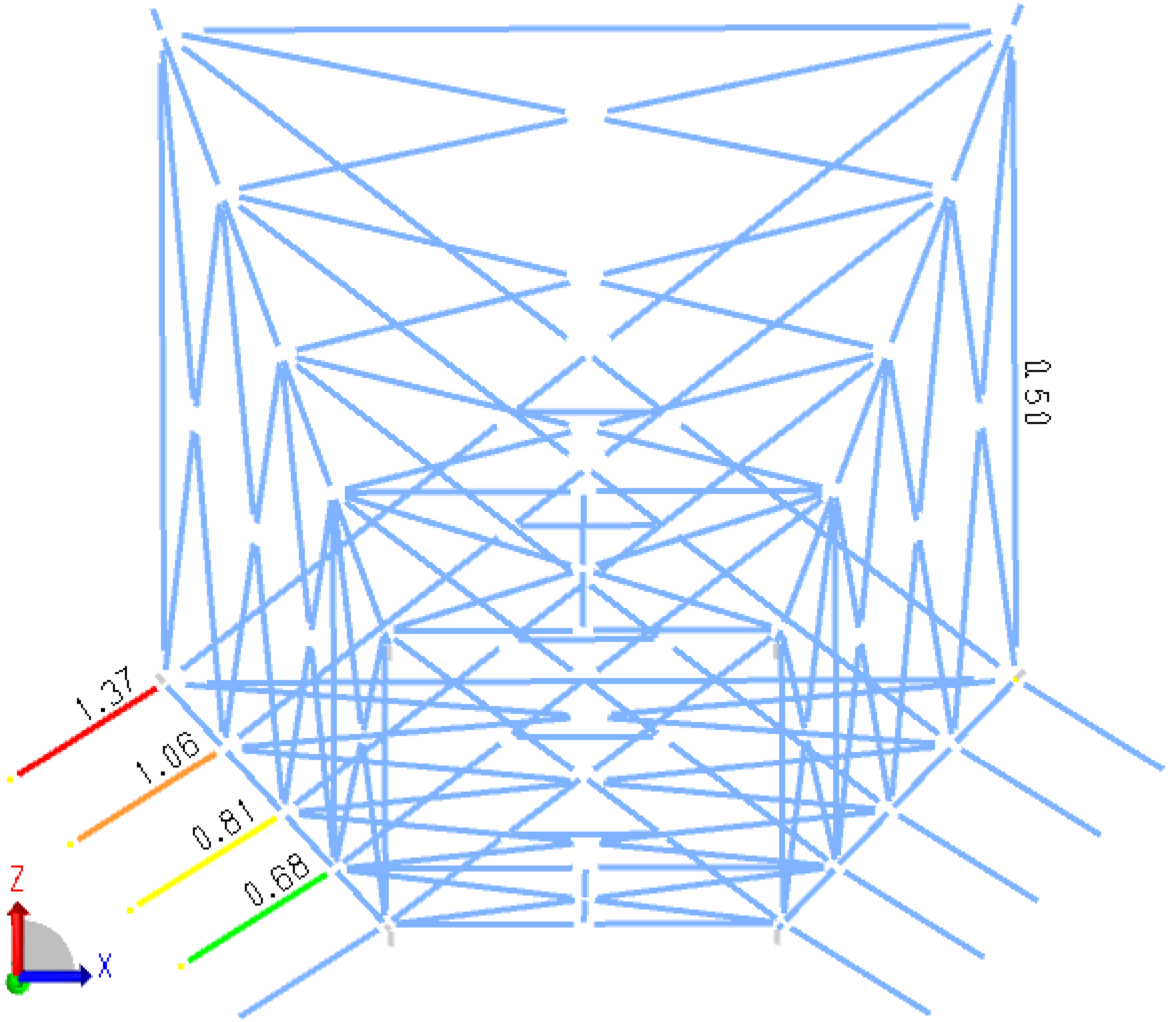


Figure 16-5 Code check result *UfTot* for *Worst Case (CC)* for transportation analysis

It should be noted that combining the gravity and roll result cases is strictly incorrect since the two merged analyses both are non-linear analyses due to the compression-only stools. Experience shows, however, that this procedure produces acceptable results.

In Xtract the gravity result case is combined with the rotational result cases. This is activity **Xtract_present_results**. Figure 16-6 shows deformed model overlaying the undeformed. The axial forces in the beams (*Elements G-FORCE NXX*) are colour coded. As seen the roll combined with gravity causes tensile axial forces in the sea fastening and the jacket lifts from the stool on the tension side.

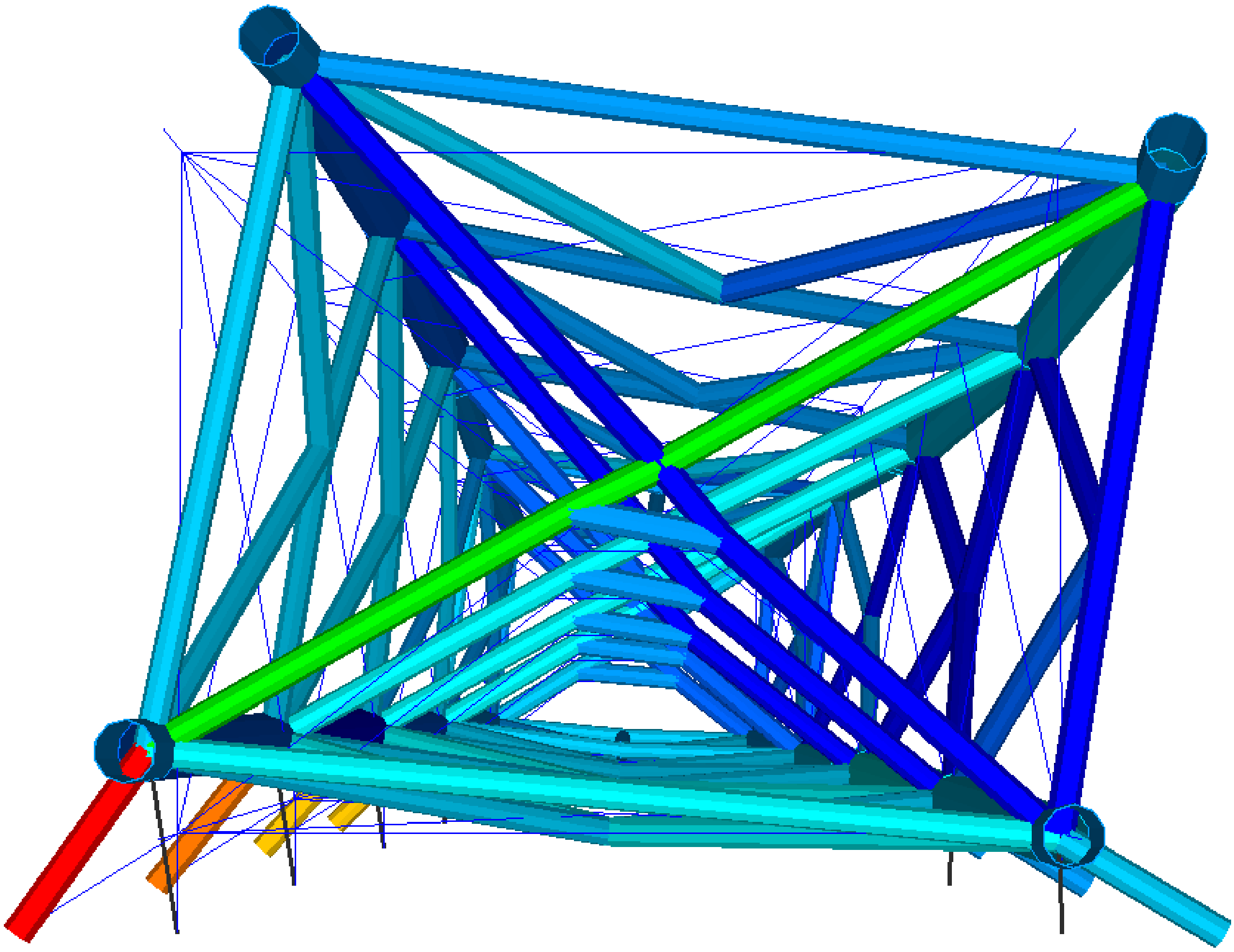


Figure 16-6 Deformed model for transportation analysis

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