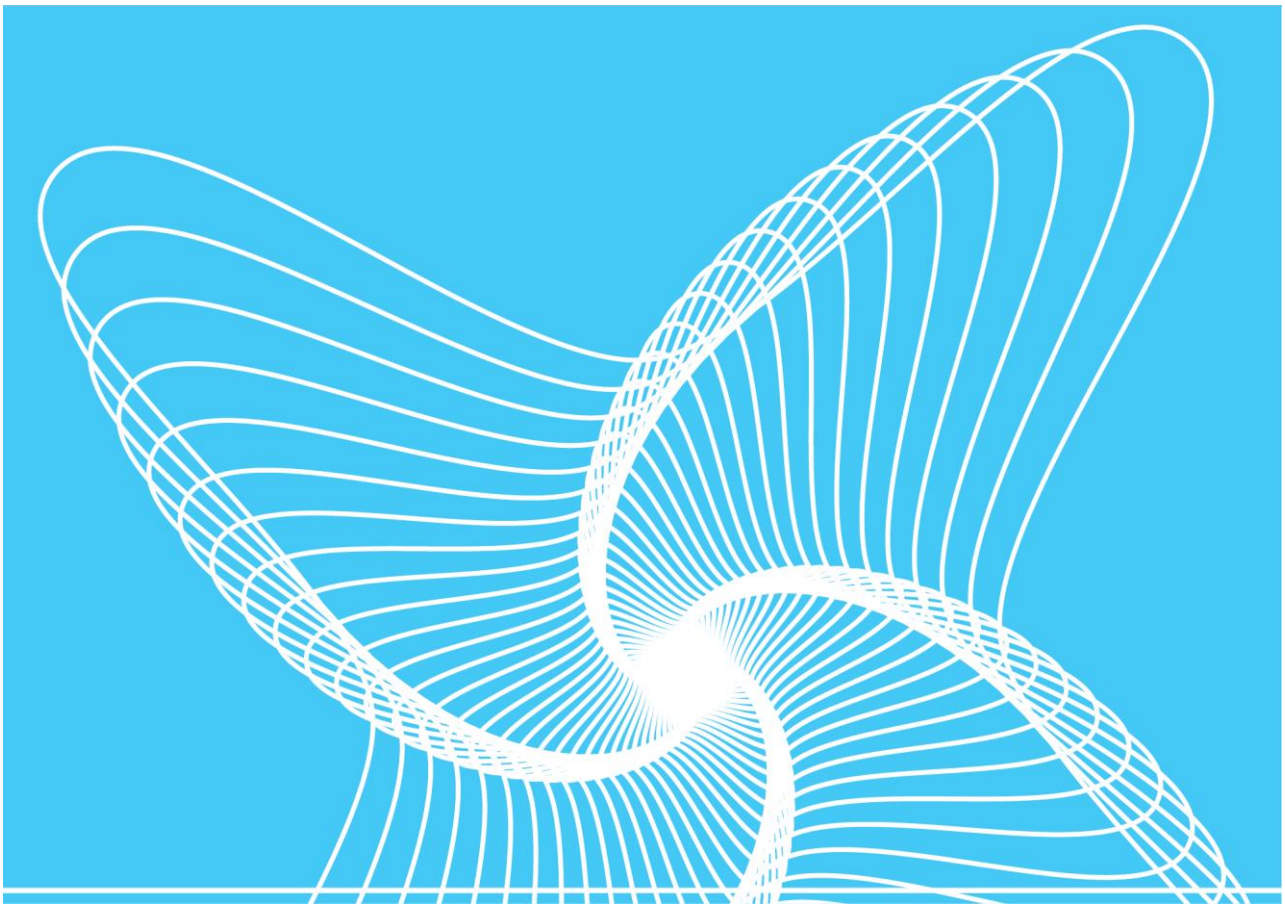
DNV·GL

SESAM USER MANUAL

Fatigue Manager

Fatigue analysis of fixed offshore structures

Valid from program version 3.2



SAFER, SMARTER, GREENER

Sesam User Manual

Fatigue Manager

Date: March 1st 2015

Valid from Fatigue Manager version 3.2

Prepared by DNV GL - Software

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

E-mail sales: software@dnvgl.com

© DNV GL AS. All rights reserved

This publication or parts thereof may not be reproduced or transmitted in any form or by any means, including copying or recording, without the prior written consent of DNV GL AS.

Table of contents

1 INTRODUCTION 2
1.1 Fatigue Manager 2
1.2 How to Read this User Manual 2
1.3 Terminology and Notation 2
1.4 Release Notes and Status List 2
1.5 Limitations 3

2 FEATURES OF FATIGUE MANAGER 4
2.1 Prerequisites for running Fatigue Manager 4
2.2 Features of Fatigue Manager 4
2.3 Analysis flow 4

3 USER'S GUIDE TO FATIGUE MANAGER 6
3.1 Graphical user interface 6
3.2 Global parameters 8
3.2.1 Structure and loads data 8
3.2.2 Timing and Environment Data 9
3.3 Wind loads 11
3.3.1 File import and usage 11
3.3.2 File format 11
3.4 Additional input files 12
3.4.1 Optional Wave Data 12
3.4.2 Sestra Input Files 13
3.4.3 Fatigue Data 14
3.5 Defining and executing the analyses (time simulations) 15
3.5.1 The data grid 15
3.5.2 The tool buttons 16
3.6 Processing in the Cloud 18
3.6.1 Registering for Sesam Cloud 18
3.6.2 The Cloud Interface inside Fatigue Manager 18
3.7 Result Reporting 20
3.7.1 Computation of damage 21

4 IMPORT WIND LOADS 22
4.1 HAWC Converter 22
4.2 Bladed Converter 24

5 DATA FLOW DESCRIPTION 26

6 RAINFLOW COUNTING METHOD 27

1 INTRODUCTION

1.1 Fatigue Manager

Fatigue Manager is a tool for doing time history fatigue analysis with wave and wind loads. The software package can do fatigue analysis on all types of offshore structures, such as jacket, tripods and monopiles.

In addition to the main analysis management tool, the load merge tool and the report tool are described in this document. Fatigue Manager will use the SESAM programs Wajac, Sestra, Splice, Gensod and Framework. Information about these programs is found in their respective User Manuals.

For information about how to set up a pile/soil analysis and create input data for Wajac, see the GeniE User manual.

1.2 How to Read this User Manual

Chapter 2 gives an overview of the functionality of the Fatigue Manager.

Chapter 3 gives a more detailed description of how to use the Fatigue Manager.

Tutorials describing the use of the application on example cases are provided as separate documents.

1.3 Terminology and Notation

Workspace:	A workspace defines the settings and contents for a specific fatigue analysis.
Workspace folder:	This is the root folder for the fatigue analysis. It contains the workspace settings and all input files.
Analysis:	An analysis of the structure for one selected sea state and one selected wind load time series. A workspace will typically contain a large number of Analyses.
Analysis folder:	Every analysis will be carried out in a separate folder. These folders will be sub-folders to the Workspace folder.
Wave load analysis:	Computation of wave loads on the support structure. The analysis is done using the program Wajac.

1.4 Release Notes and Status List

Additional information exists for Fatigue Manager next to this user manual (as for all other Sesam programs). This may be:

- Reasons for update (new version)
- New features
- Errors found and corrected (bug fixes)
- Frequently asked questions (FAQs)
- Etc.

To look up information, go to the DNV GL Software Customer Portal, found on http://dnvs.force.com/ and log in to this service. Here you can find the software, documentation, FAQs and the Sesam Status List.

1.5 Limitations

- Fatigue life will only be calculated for beams. The structural model may however include beams, shells and solids.
- The structural model must have super element number 1 (<prefix>_T1.FEM).
- All models and input must be in SI units.
- All wind files must at least have the same length as specified for the simulation length.

2 FEATURES OF FATIGUE MANAGER

2.1 Prerequisites for running Fatigue Manager

The model of the structure must be available before starting the analysis. If the user wants to include pile/soil analysis as part of the simulation, it is required that soil data and Splice input is also available. All this is most easily established by performing a pile/soil analysis in GeniE.

It is also possible to use e.g. GeniE to set up additional input data to the wave load analysis. Examples of such data are hydrodynamic coefficients and marine growth data.

For more details see section 3.1.

2.2 Features of Fatigue Manager

Fatigue Manager will perform a fatigue analysis in the time domain. The analysis will typically consist of a set of different environmental conditions. Each condition consists of a combination of a sea state and a time history for the load imposed from wind.

Each time simulation consists of the following steps:

- Wave load computation by Wajac.
- Merging of wave loads and wind loads. (this step is optional)
- Structural analysis using Sestra.
- (Optionally) Pile/soil analysis using Splice.
- Fatigue analysis using Framework

Alternatively, Fatigue Manager can also run the fatigue analysis directly on specified result files.

After execution of all simulations with the environmental conditions the fatigue data will be read by the Report tool which will add the fatigue damage from all time simulations to provide the total damage.

2.3 Analysis flow

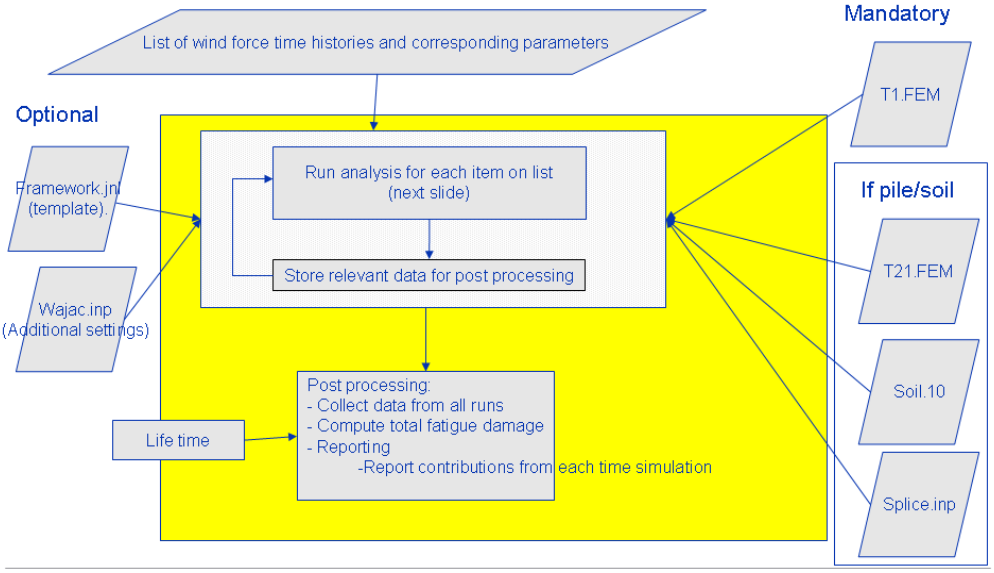


Figure 2.1: The overall workflow.

Figure 2.1 shows the overall workflow. All the input files except for the wind force time histories are most easily obtained by performing a wave load analysis in GeniE (including pile/soil if that is to be included in the wind analysis).

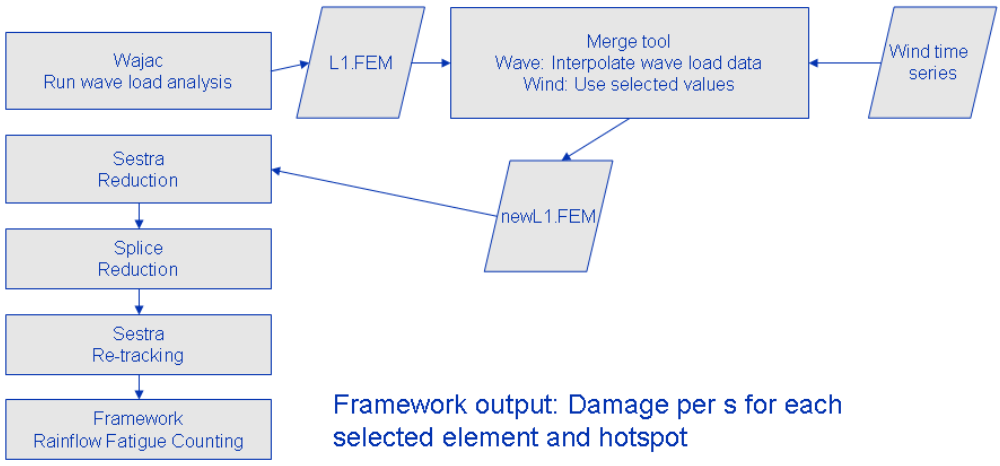


Figure 2.2: The workflow for each time-simulation.

Figure 2.2 shows the workflow for each time simulation. If pile/soil is not included the sequence Sestra – Splice – Sestra is replaced by only Sestra.

It is possible to skip the wind load and the merge step, in which case the application can be used to perform fatigue analysis in time domain for the wave loads (Wajac) only.

3 USER'S GUIDE TO FATIGUE MANAGER

This chapter provides a description of the different features of Fatigue Manager.

- Section 3.1 describes the graphical user interface
- Section 3.2 describes the global parameters
- Section 3.3 describes the wind load input file.
- Section 3.4 describes the other input files.
- Section 3.5 describes how to set up the analysis.
- Section 3.6 describes the report tool.

3.1 Graphical user interface

The Graphical user interface (GUI) is as shown in Figure 3.1 and Figure 3.2. It consists of three main tabs, the *Global Settings* tab and the *Wave/Wind* tab and the *Result Reporting* tab.

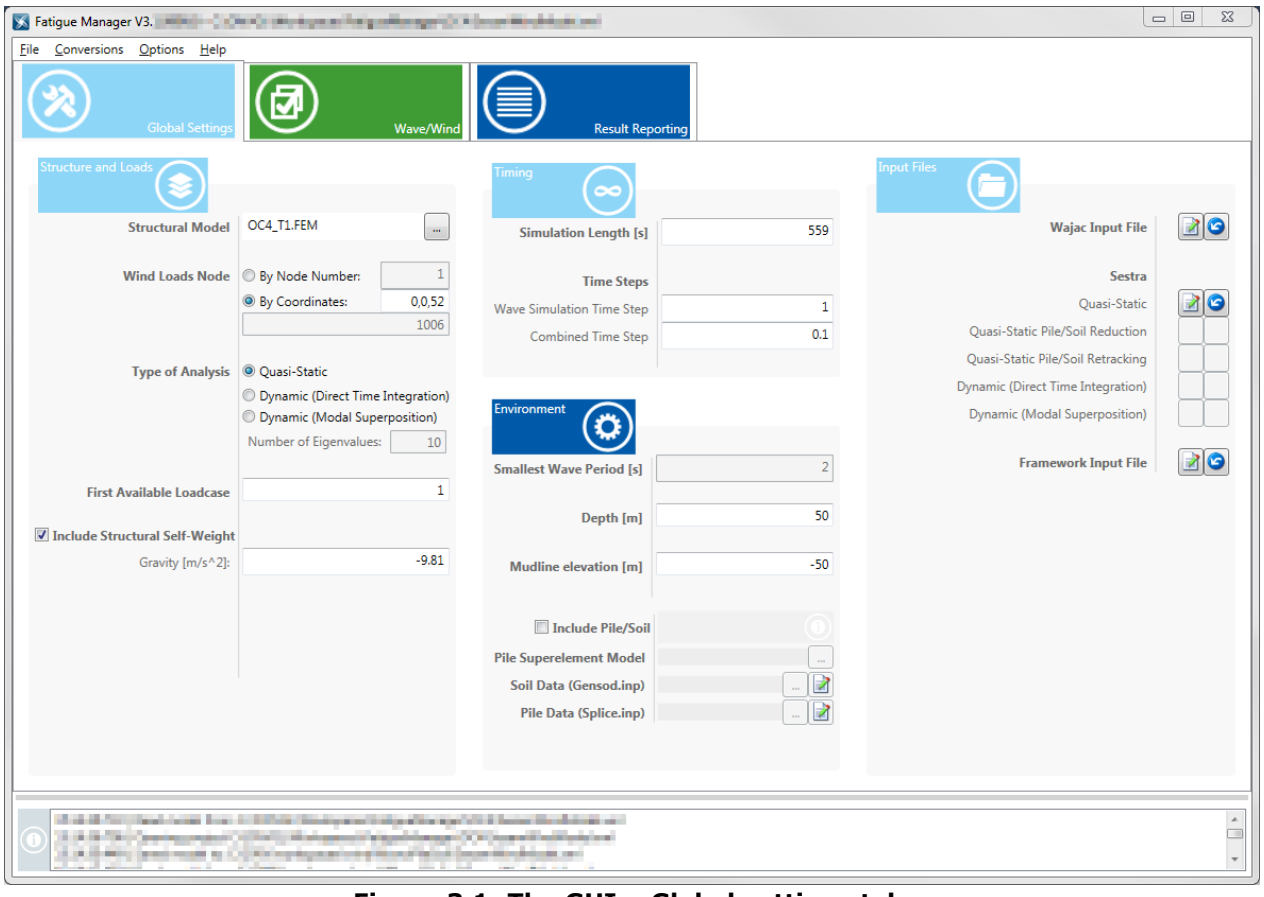


Figure 3.1: The GUI – Global settings tab.

The *Global Settings* tab is used to define parameters (see section 3.2) and the input files (see section 3.4) that are used for all different time simulations.

In this and the other tabs you have to actively move the cursor to another field in order to apply a value you have given.

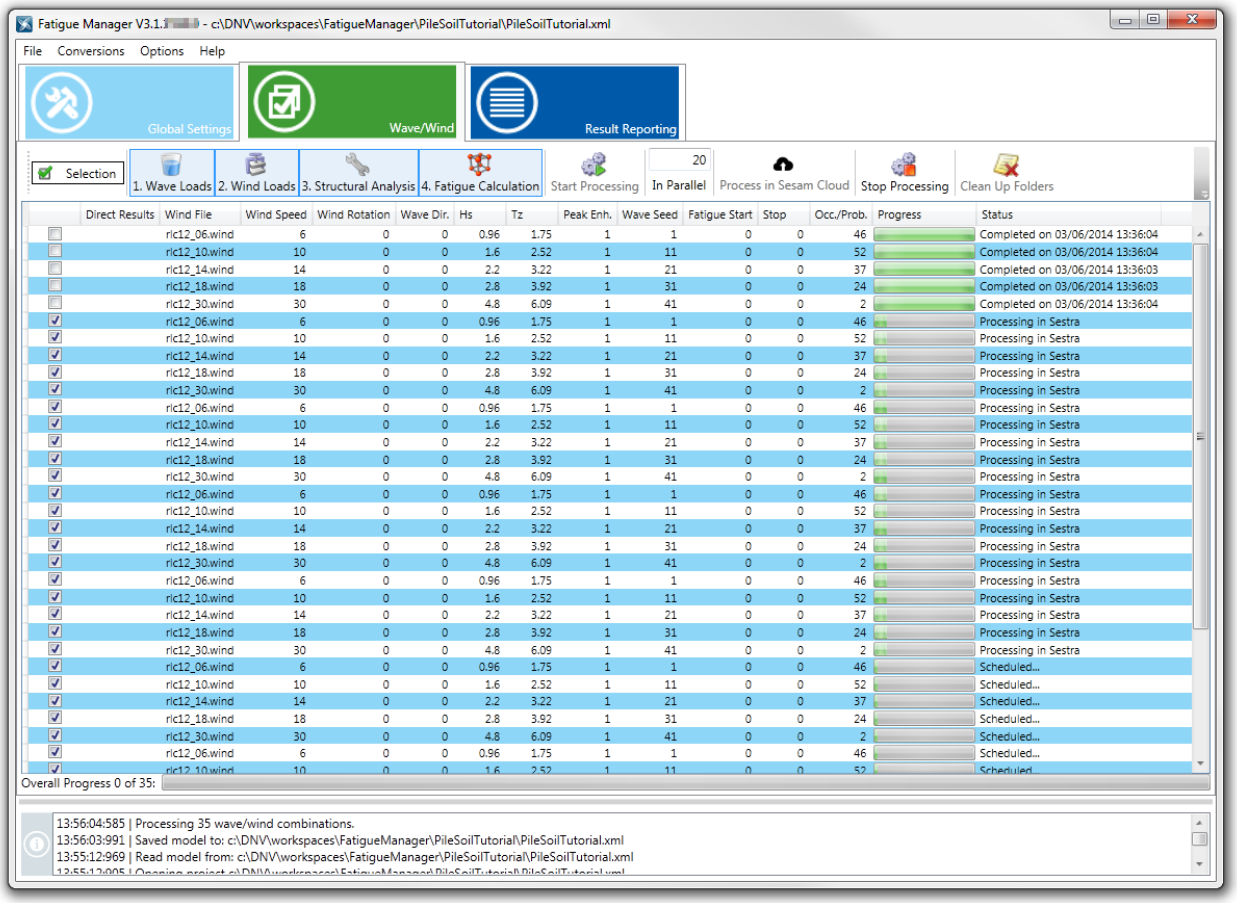


Figure 3.2: The GUI – Wave/Wind Combinations Tab.

The *Wave/Wind* tab is used to define and execute the different time simulations. For each such analysis a wind load file must be given (see section 3.3). In addition other parameters related to the wind or wave loads must be specified (see section 3.5).

For each simulation there is a status bar showing the progress of the analysis. There is also a progress bar at the bottom of the tab showing the overall progress of the analysis.

If results for a wave/wind combinations are available from a previous analysis, the progress bar will be at 100% and the status field shows when the combination had been completed

3.2 Global parameters

This section describes the global parameters that are set for all different time simulations.

3.2.1 Structure and loads data

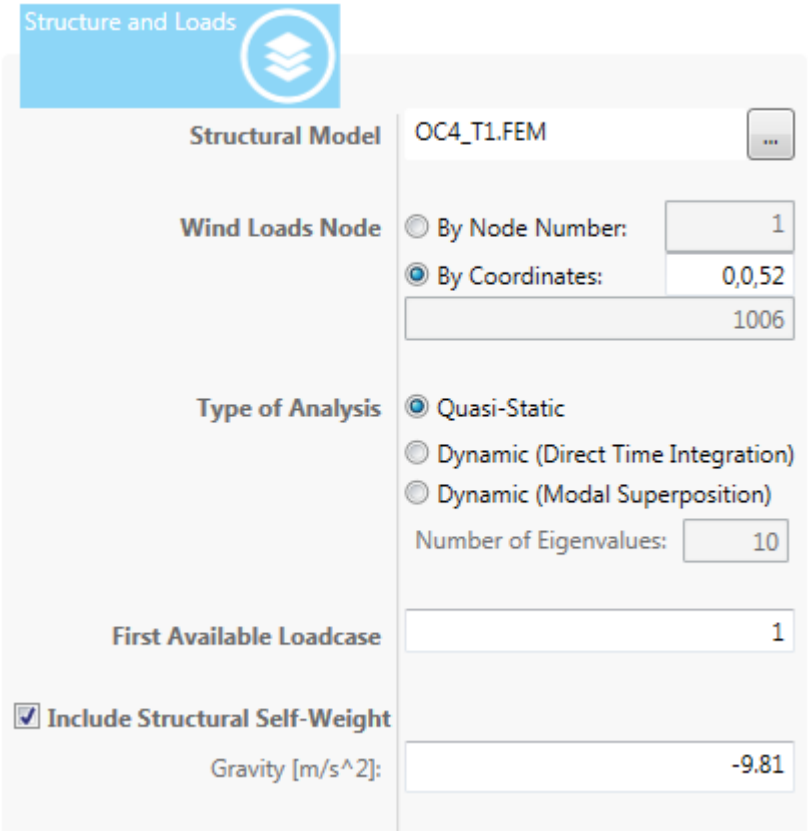


Figure 3.3: The Structure and Loads data field.

Structural Model: Click on the browse button to locate the structural model to be used in the analysis. A copy of this file will be created in the workspace folder for the fatigue analysis project.

In case you want to perform an analysis without piles and soil in Fatigue Manager, you will only need the _T1.FEM file. Note that the _T1.FEM file for a model including piles and soil is not the same as for an analysis without piles and soil included.

Wind-Loads Node: You can specify the load where the wind loads are to be applied either by giving the external node number (the number you see in GeniE when you label node numbers) or the node coordinates. The coordinates do not need to coincide exactly with the coordinates of a node in the model. The application will find the nearest node and apply the loads at that node. For both selections the corresponding internal node number is shown.

Type of Analysis: Three types of analyses can be selected. The difference between the different modes is the type of analysis being run in Sestra.

- **Quasi-Static:** Read the Sestra manual for more information.
- **Dynamic (Direct Time Integration):** In the case of dynamic analysis with direct time integration, remember to adjust the start time for the fatigue analysis. The results from the structural analysis will, in the start, contain a transient period, that should not be taken into account for the fatigue

analysis. Set the Fatigue Start time in for the analyses accordingly in the Wave/Wind combination part of the user interface.

- **Dynamic (Modal Superposition):** Read the Sestra manual for more information. If default number of mode shapes for dynamic analysis is not set (i.e. since it is zero by default in Fatigue Manager), then 10 mode shapes will be computed by Sestra.

First Available Loadcase: Wajac will create one load case per time step. These will be numbered sequentially starting with the load number defined here. It is recommended not to have any other loads defined on the structural model used for Fatigue Manager, in which case the default value of 1 should be used. If there are such loads you should specify a number which is higher than the load cases created by GeniE.

Include Structural Self-Weight: Enables or disables the adding of the self-weight of the structure to every time-step of the analyses. **Gravity** will be set to -9.80665 as default for new workspaces. This value is applied in global Z-direction, which is pointing upwards. Therefore this value has to be negative.

3.2.2 Timing and Environment Data

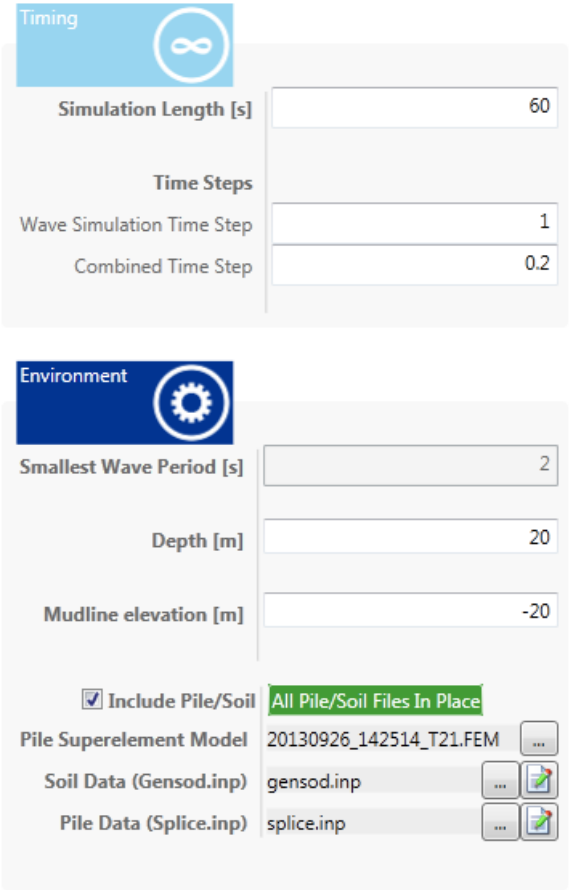


Figure 3.4: The Timing and Environment data fields.

Simulation Length: The duration of the time domain simulation (in seconds). This value will define the simulations length for every wave/wind combination.

Time Steps:

- **Wave Simulation:** The time step used in the Wajac analysis.
- **Combined:** The time step used in the structural and fatigue analysis. This time step must be a multiple of the time step of the wind load file. The time step can be lower than the Wave Simulation (Wajac) time step, in which case the Wajac load will be interpolated (see section 3.3).

Smallest Wave Period: The shortest period included in the Wajac realization of the specified sea state. Will always be 2x the Wave Simulation time step.

Depth from still water to mudline: The water depth. This is always a positive number.

Mudline elevation: The z-coordinate of the mudline in the structural model coordinate system.

Include Pile/Soil: Optionally the model can include piles and soil. In that case some extra input files are required.

All Pile/Soil Files in Place: This area will be green, when all necessary files for pile/soil are in place.

Pile Superelement Model (mandatory for pile/soil analysis): The pile/soil analysis must be done as a super element analysis. The top level super element must have number 21. This file (T21.FEM) is created by GeniE if a pile/soil analysis is carried out.

Soil data (mandatory for pile/soil analysis): This file has the name *gensod.inp* and is created by GeniE when doing a pile/soil analysis. It will serve as the input for Gensod. A template Gensod input file must be available, since Fatigue Manager will not create a complete Gensod input file. The user can select the file by using the browse button and optionally edit the selected file by selecting the "Edit" button and change and save the file in the text editor that will show up.

Splice Input File (mandatory for pile/soil analysis): This file has the name *splice.inp* and is created by GeniE when doing a pile/soil analysis. It will serve as the input for Splice. A template Splice input file must be available, since Fatigue Manager will not create a complete Splice input file. The user can select the file by using the browse button and edit the selected file by selecting the "Edit" button and change and save the files in the text editor that will show up. Fatigue Manager will modify this file automatically in order to set the correct amount of load cases.

3.3 Wind loads

In this section it is explained how to use wind load files in Fatigue Manager.

3.3.1 File import and usage

A wind load input file must be specified for each analysis, if no direct result input files is selected). The same wind load file may be used in many time analyses.

The file is imported into an analysis by selecting the analysis (i.e. the Wave/Wind combination) and selecting *Select Windfile* from the RMB menu. After selection the file is copied to the workspace folder.

3.3.2 File format

The loads on the support structure due to the wind must be available as a time series. The format of the file is as follows (with one line per time step and a space as separator):

Time Fx Fy Fz Mx My Mz

- **Time** is the time in seconds.
- **Fx** is the force in the wind direction (the direction the wind is blowing towards).
- **Fy** is the horizontal force normal to the wind.
- **Fz** is the vertical force (the z-axis is pointing upwards).
- **Mx**, **My**, **Mz** are the moments around axes parallel to x, y and z-axis as defined above

The wind load will be applied as a nodal load on the structure at the selected node. The moments are applied as given on the wind file, thus these moments must be given relative to axes going through this node.

3.3.2.1 Example of .wind file

```
2.00000E-02 9.69330E+06 -2.03934E+05 0.00000E+00 -2.51951E+06 3.24129E+06 -2.66348E+06
4.00000E-02 9.92221E+06 -2.08451E+05 0.00000E+00 -2.83301E+06 3.23756E+06 -3.02408E+06
6.00000E-02 1.02243E+07 -2.17529E+05 0.00000E+00 -3.04241E+06 3.20739E+06 -3.25410E+06
8.00000E-02 1.05831E+07 -2.30668E+05 0.00000E+00 -3.14535E+06 3.16025E+06 -3.34926E+06
1.00000E-01 1.09855E+07 -2.46911E+05 0.00000E+00 -3.15063E+06 3.10640E+06 -3.32258E+06
1.20000E-01 1.14218E+07 -2.65001E+05 0.00000E+00 -3.07429E+06 3.05651E+06 -3.20019E+06
1.40000E-01 1.18833E+07 -2.83581E+05 0.00000E+00 -2.93789E+06 3.02178E+06 -3.01395E+06
1.60000E-01 1.23631E+07 -3.01475E+05 0.00000E+00 -2.76080E+06 3.01204E+06 -2.79444E+06
1.80000E-01 1.28561E+07 -3.17957E+05 0.00000E+00 -2.55444E+06 3.03492E+06 -2.56427E+06
```

3.4 Additional input files

All the input files described below, will be copied to the Workspace folder when selected from the User Interface. To select the file use the browse button as shown in Figure 3.5.

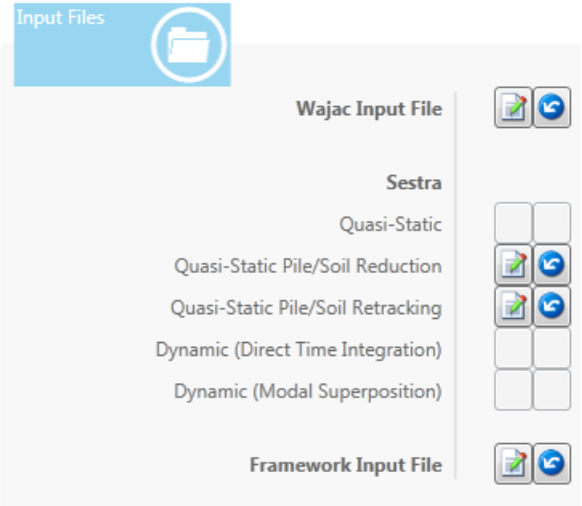


Figure 3.5: Input files.

3.4.1 Optional Wave Data

If the edit field is greyed out the default Wajac.inp will be used. If the user wants to edit the default Wajac.inp, this is possible by pressing the "Edit" button.

"Reset to Default" will overwrite the Wajac.inp in the workspace folder.

3.4.1.1 Variables

The following variables in the Wajac file are automatically replaced and added to the corresponding wave/wind combination folder by values from the input.

- `#.....TITLE#` is replaced with a descriptive title text.
- `#.....PREFIX#` is replaced with the prefix of the model file.
- `#.....OUTPUTPREFIX#` is replaced with the prefix of the output load file.
- `#....LOADNUMBER#` is replaced with the number entered for the first available loadcase.
- `#.MUDLINE#` is replaced with the value for mudline elevation.
- `#..ISPECT#` is replaced with the value 2.0.
- `#..SIMLEN#` is replaced with the value entered for Simulation Length.
- `#....SEED#` is replaced with the value entered for the seed for the corresponding wave/wind combination.
- `#....TCUT#` is replaced with the value for the smallest wave period.
- `#TIMESTEP#` is replaced with the value for Wave Simulation Time Step.

- #.....TIM# is replaced with the sum of Simulation Length and Wave Simulation Time Step.
- #....DELT# is replaced with the value for Wave Simulation Time Step.
- #....BETA# is replaced with the value for the Wave Dir.
- #......HS# is replaced with the entered value for Hs.
- #......TZ# is replaced with the entered value for Tz.
- #..SIGMAA# is replaced with the value 0.07.
- #..SIGMAB# is replaced with the value 0.09.
- #...GAMMA# is replaced with the value entered for Peak. Enh.
- #...DEPTH# is replaced with the value for Depth.
- #...NSTEP# is replaced by the sum of Simulation Length and Wave Simulation Time Step divided by the Wave Simulation Time Step.
- #BET# is replaced with the value for the Wave Dir. (used for fields that can only take a five digit number for this value.
- #...MOMTZ# is replaced with the value for mudline elevation.

3.4.2 Sestra Input Files

There are multiple Sestra input files available. Based on the selection for the type of analysis and if pile/soil is included, different Sestra input files will be used. The unused files are greyed out.

In the respective file some key words between two hash tags "#" will be replaced by variables from the user interface. For example #.......PREFIX# will be replaced with the model file prefix of the model used for the analysis.

All other commands can be changed or new can be added.

3.4.2.1 Sestra Quasi-Static

None of the #-variables are recommended to change by the user. All other commands can be changed or new can be added.

3.4.2.2 Sestra Dynamic

The #.......H# variable will be replaced by the combined wave/wind time step divided by two. If a different H –value is desired this has to be entered here. The number cannot be larger than ten digits and has to fit into the space that the #.......H#-variable occupies.

3.4.2.3 Sestra Quasi-Static Pile/Soil Reduction and Retracking.

None of the 3-variables used inside theses file are recommended to change.

3.4.2.4 Sestra Modal superposition

The `#EIG#`-variable will be replaced by the value entered in the user interface. It is not recommended to change the number of eigenvalues used by replacing the `#EIG#`-variable, but rather to enter the desired value in the user interface.

3.4.3 Fatigue Data

The input commands to Framework that are common for all time simulations must be specified. This file will typically contain commands for which members and hotspots we want to analyse, stress concentration factors etc.

The commands can be changed by editing the Framework input file from the User Interface by pressing the "Edit" button.

"Reset to Default" will overwrite the "Framework_const.jnl" in the workspace folder.

The `#FATIGUESTART#` variable will be replaced with the value set for Fatigue Start in for the corresponding wave/wind combination. The `#FATIGUESTOP#` variable will be replaced with the value given in the Stop field. If that value is set to zero, this value will be replaced with the complete simulation length.

For dynamic analysis with direct time integration it is recommended to start the rainflow counting analysis after a certain amount of seconds.

3.5 Defining and executing the analyses (time simulations)

This section explains how to set up the different time simulations.

3.5.1 The data grid

The time simulations are defined on the *Wave/Wind* tab, see Figure 3.2. This data grid supports Copy/Paste from Excel so it is possible to set up all the data in an Excel spread sheet.

The content of the different columns is as follows:

Selection: The selection column identifies which of the time simulations are to be included in the analysis. Only checked time histories are included in the analysis and in the reporting.

If you have done an analysis for many time simulations, but want to add some more, the procedure is:

1. Unselect All.
2. Check the new analyses.
3. Perform the analysis by using the Start Processing button. This will only execute the new time simulations.
4. Before running the report tool use the Select All button to make sure that all runs are included in the computation of the damage.

Direct Results: Either this field can be filled or the Wind loads field. If a _R1.SIN file is selected in this field the analysis will just perform a Framework fatigue calculation. All other steps are not applied to this wave/wind combination. If the time series in the selected direct input file is shorter than the simulation length given in the Global Settings, then the Fatigue Stop time has to be set explicitly for this combination.

Wind loads: The .wind file that is used in the load merge step. The file format is explained above.

Wind speed: This parameter is not used directly in any computation (it is implicitly included in the wind load time series). The value is only used as an identification parameter in the report tool.

Wind direction: The angle between the x-axis of the model and the x-axis in the wind load data. A positive angle is defined conform a positive z-axis upwards and using the right hand rule.

Wave direction: The direction between the model x-axis and the direction the waves are propagating towards. The wave and wind directions can be different input parameters to Wajac. A positive angle is defined conform a positive z-axis upwards and using the right hand rule.

Hs (significant wave height), Tz (mean zero up-crossing wave period), Peak enhancement: The parameters defining the Jonswap spectrum. Notice that the period is the Mean Zero Up-crossing period. Input parameters to Wajac.

Wave seed: Input parameter to the wave generation in Wajac. Changing this parameter will give rise to a different realization of the same wave spectrum.

Fatigue Start: The time in seconds when the fatigue counting shall start. This can be used to not take into account the first 100sec of a dynamic analysis for example. This value will replace the #FATIGUESTART# variable in the Framework input file.

(Fatigue) Stop: If set to 0 then the value of the simulation length of the global settings tab is taken. If a value above 0 is chosen, this this value will replace the #FATIGUESTOP# variable in the Framework input file. The fatigue counting will stop at the time set here.

Occurrences/Probability: The number identifying which part of total life time each time simulation represents, see section 3.6. If you are changing these numbers **it is not necessary to redo the analysis (time simulations)**. You only need to rerun the report tool.

Progress bar and Status column: Shows the progress and status of the simulations. If an analysis fails the status column will also contain some information about the error.

3.5.2 The tool buttons

The tool buttons on top of the grid are used to control the analysis. These buttons correspond to the steps in the workflow as illustrated in Figure 3.6. Step 1, 3 and 4 are mandatory before a report can be made, but it is possible to e.g. first do all Wajac runs, then all the rest of the simulations (so to do all steps 1 first, then all steps 2 etc.). Only the steps that are checked are executed when you press the Process button.

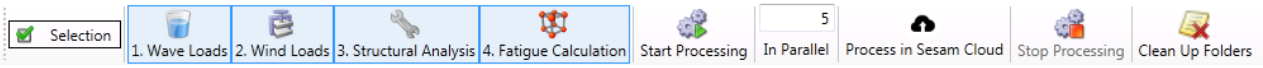


Figure 3.6: The Processing Toolbar.

Selection button: The sub-selections of the Selection button are:

- Select All
- Select All with Results (Will select rows where a Framework-results file exist.)
- Select All without Results
- Select The Next 10 Unselected Without Results
- Deselect All

Analysis steps:

1. **Wave Loads**: If checked, Wajac will be executed.
2. **Wind Load:** If checked, wind loads will be added with the wave loads.
3. **Structural Analysis**: Execute Sestra (possibly including Splice) if checked.
4. **Fatigue Calculation**: If checked, Framework will be executed.

Start Processing: Executes all those simulations which have been checked in the first column.

In Parallel: Here you specify how many parallel processes you want to run. You should be careful not to overrun on memory. What this means in practice depends on the size of the model, the number of time steps and the number of members/joints you want to analyse in Framework. For the case in the tutorial you can run 4 parallel runs on a 32-bit OS with 4 threads and 3 GB memory.

Process in Sesam Cloud: Will open a new window. From that window the selected wave/wind combinations can be started in the cloud. (See own chapter.)

Stop Processing: Stops all on-going analyses.

Clean Up Folders: Will – for all selected wave/wind – combinations delete all files from their folders, except for the files that result from the Fatigue Calculation.

3.6 Processing in the Cloud

For some projects, it can be very time-consuming analysing large number of wave/wind combinations. By selecting Process in Sesam Cloud, all analysis jobs can be processed in Sesam Cloud and results are downloaded to Fatigue Manager after each wave/wind combination is completed. When all jobs are completed, the results can be used the same way as if the analysis was done locally on the client.

3.6.1 Registering for Sesam Cloud

To be able to process wave/wind combinations, you have to be a registered user of Sesam Cloud and have been given the correct privileges.

For registration for Sesam Cloud go to this web-site and follow the instructions: http://sesam.cloudapp.net/.

After registration contact DNV GL – Software Support for setting the right privileges for the account.

3.6.2 The Cloud Interface inside Fatigue Manager

Pressing the "Process in the Cloud" button will open a new window.

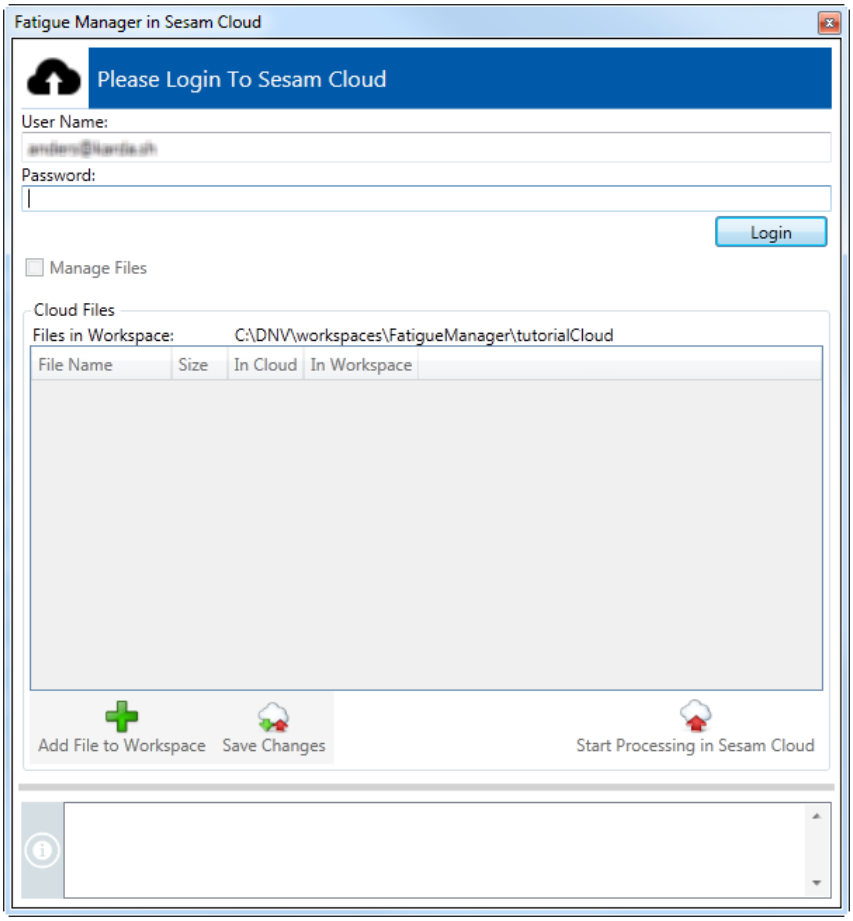


Figure 3.7: The Sesam Cloud Window.

User Name: Enter your Sesam Cloud user-name.

Password: Enter the password for the account.

Login: Pressing this button will send the user name and password to the cloud for login and then scan the cloud storage for existing files. Login usually takes some seconds, while analysis of file status in the cloud can take much longer time. It is recommended to delete unused workspaces in the cloud to speed up the login-process.

Manage Files: Fatigue Manager will upload all necessary files to the cloud automatically, but it is also possible to manage the files in the cloud manually by selecting Manage Files. With Manage Files, you can add or remove files in the cloud as well as adding new files (by selecting the *Add File to Workspace*.) This can be useful when a file is updated on the client and needs to be replaced in the cloud.

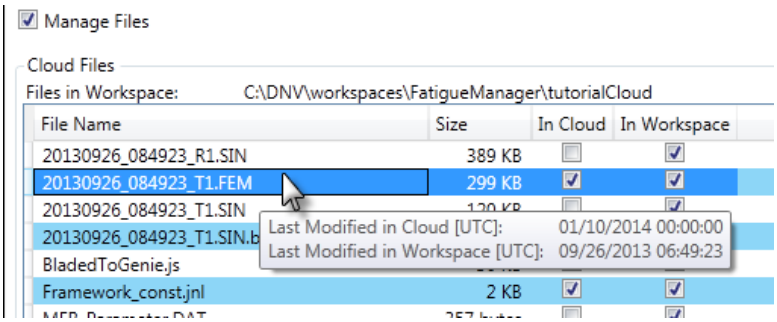


Figure 3.8: Manage files in Sesam Cloud.

Start Processing in Sesam Cloud: This will start uploading all necessary files to Sesam Cloud and start processing all selected wave/wind combination. Fatigue Manager will then return to the Wave/Wind combination-tab. The selected jobs will get status *Scheduled* and Fatigue Manager will track the cloud status for all jobs every 30 seconds. When a job is finished successfully, the Framework-result file will be copied from Sesam Cloud to the client.

3.7 Result Reporting

The result reporting tab will show the accumulated damage results for all wave/wind combinations that are checked on the *Wave/Wind* tab. Lines that are not checked on the *Wave/Wind* tab will not be taken into account in the results.

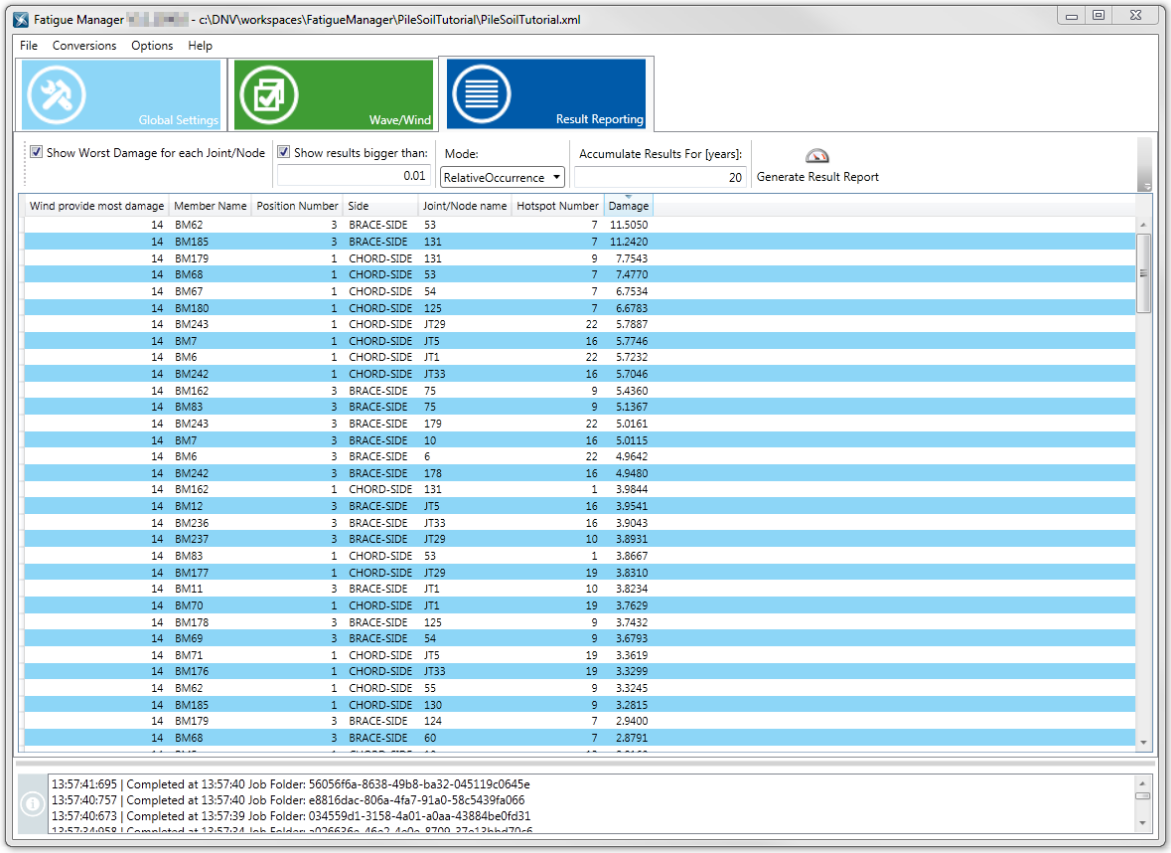


Figure 3.9: The Result Report.

If the button **Show worst damage for each Joint/Node** is selected, only the result for the worst hotspot will be printed.

If the button **Show results bigger than:** is selected, you can enter a minimum damage value in the following field. Only damages larger than this value will be reported.

Use the **Mode** selector to choose between 3 different fatigue analysis reports, depending on what kind of value has been entered in the Occurrences/Probability field on the *Wave/Wind* tab:

- Absolute Occurrence:
    - This number represents the actual number of occurrences of this environment in the fatigue life period.
    - If the fatigue life is to be doubled all these values must also be doubled.
    - With this option the parameter for Fatigue life (Accumulate results for (years)) is not used.
- Probability:

- Gives the probability of each environment, i.e. the fraction of the total fatigue life where this environment is to be applied (cf. Probabilities in a scatter diagram).
- The probabilities for the time simulations should add up to 1.0.

- Relative occurrence:
  - Gives the number of occurrences of the environments relative to each other (cf. Occurrences in a scatter diagram).
  - Is recomputed to probabilities by dividing each number of occurrences with the total number of occurrences in all environments (whether they are selected or not).

Use **Accumulate results for (years)** to specify the target fatigue life.

The report is created by pressing the **Generate Result Report** button.

The report can be written to a text file by selecting **Save Report** from the **File** menu.

A right-click and select *Show Detailed Damage List* on a line in the report will bring up a detailed report, listing the partial damages due to the different time simulations (see Figure 3.10).

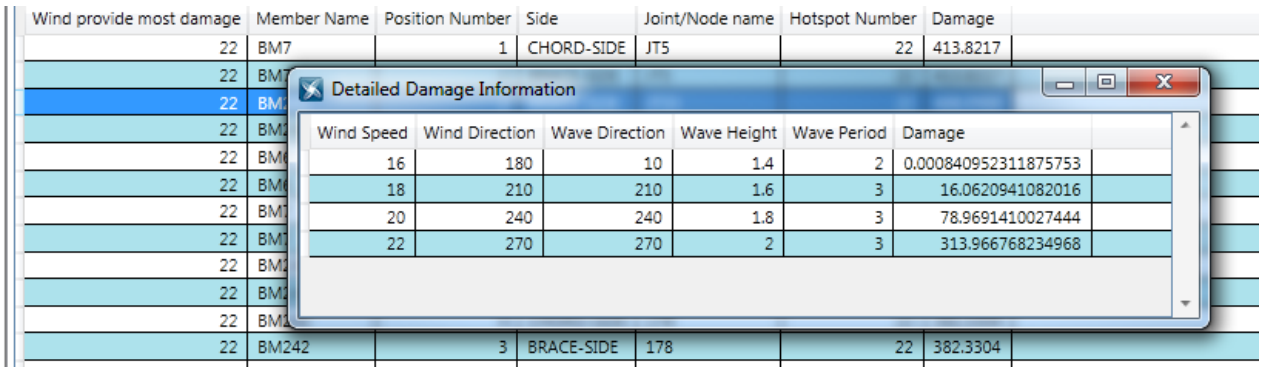


Figure 3.10: Partial damages from each simulation.

3.7.1 Computation of damage

For each of the time simulations Framework will output the *damage per second*. The total damage is computed by summing up the partial damages for each simulation. Thus, the total damage reported in the main window in figure 3.8 will always be identical to the sum of the partial damages shown in the detailed damage information window in the same figure.

The partial damage for each simulation, d_i, is computed as follows:

- If probabilities are specified: $d_i = D_i * f * p_i$
- If absolute occurrences are specified: $d_i = D_i * N_i * T_i$
- If relative occurrences are specified, then these are recomputed to probabilities: $d_i = D_i * f * N_i / \sum_j N_j$

Here D_i is the damage per second for simulation i, f is the fatigue life (in seconds), N_i is the number of occurrences of simulation i and T_i is the duration of simulation i.

4 IMPORT WIND LOADS

Use the top menu **Conversions** to import wind loads from other applications.

4.1 HAWC Converter

To convert HAWC result files into Fatigue Manager format, open the source folder for the HAWC dat-files.

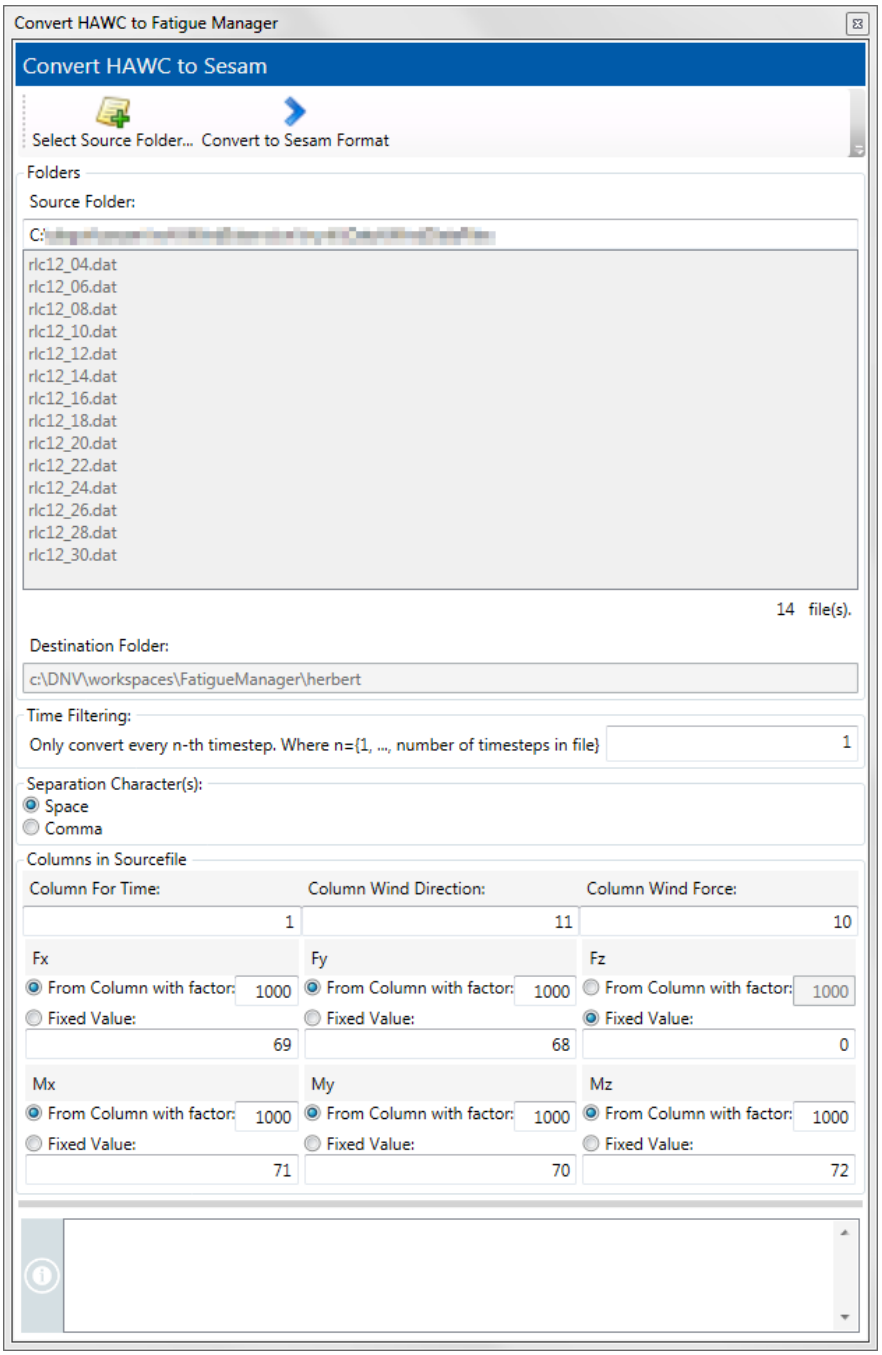


Figure 4.1: The HAWC Converter window.

The source folder is scanned for dat-files automatically.

In the options field, it is possible to specify the column number for the key values. These values are also defined in the HAWC sel-file.

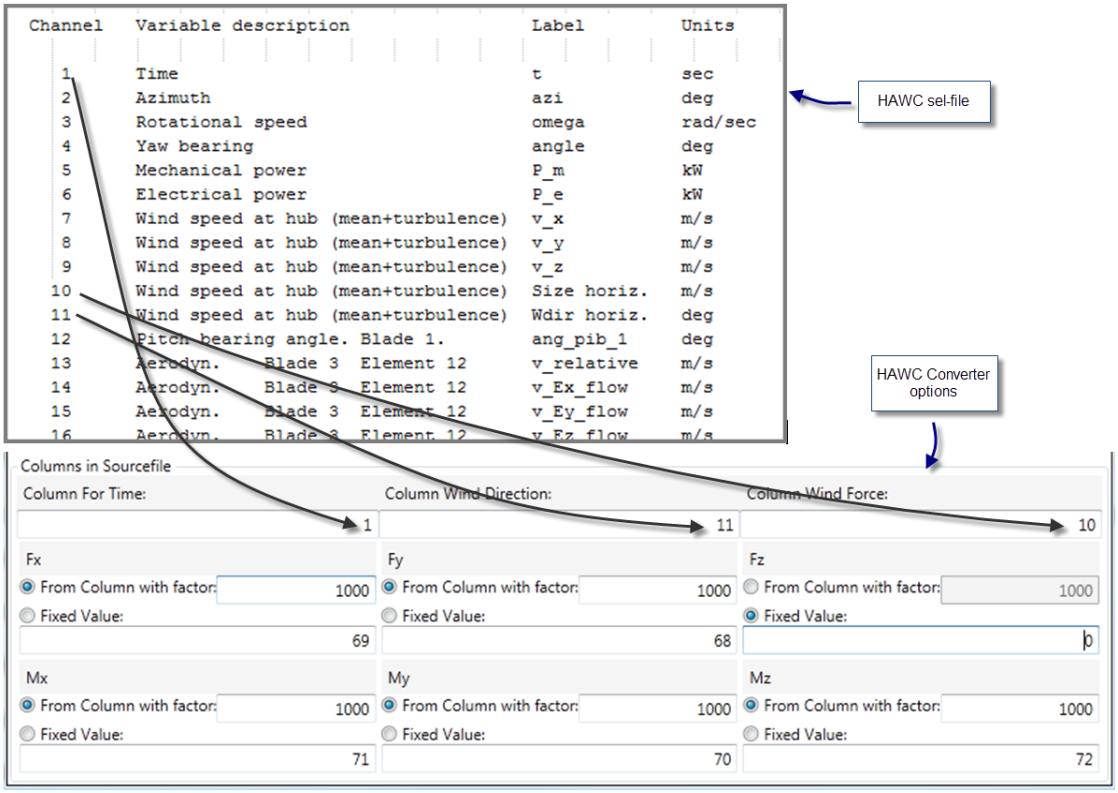


Figure 4.2: Values from the HAWC sel-file and the converter option fields.

Separation Characters: Choose between comma or space separated.

Column for time: Specify which column time is specified in the dat-file.

Column Wind Direction: Column for wind direction.

Column Wind Force: Column for wind speed.

Fx, Fy, Fz, Mx, My, Mz: Here you can insert a multiplication-factor if needed, or define a constant value instead of converting the value from the HAWC dat-file.

Pressing the **Convert to Sesam Format** will convert all dat-files in the source folder and save the result files in the destination folder.

4.2 Bladed Converter

The Bladed Converter converts Bladed Load files either to Fatigue Manager wind-files or SESAM _R1.SIN files. To convert files, you first need to select a Bladed project file. After the project file is selected, the converter will scan all directories below the selected project files for Bladed Load files and list the files in a table.

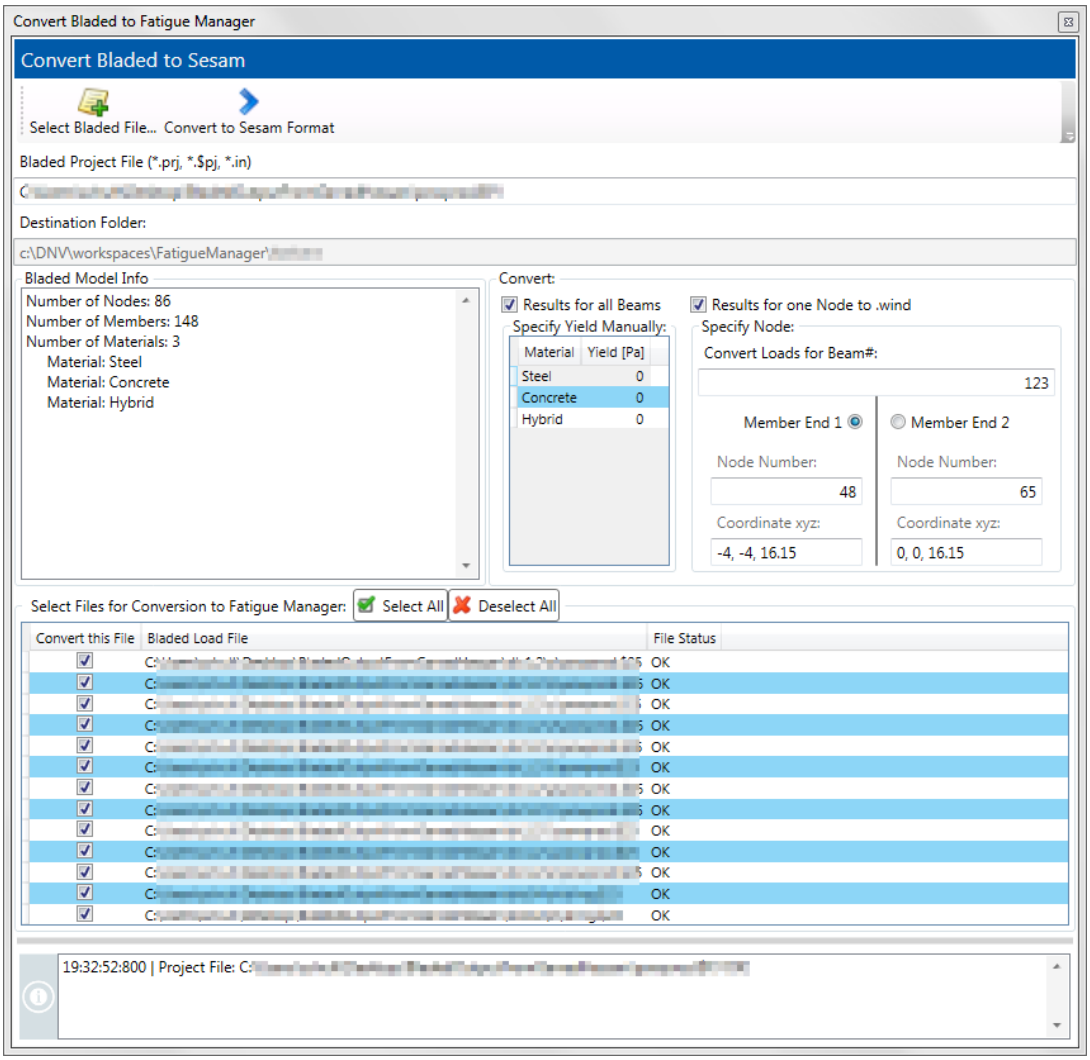


Figure 4.3: The Bladed Converter window.

At least one of the two following selections has to be selected in order for the **Convert to Sesam Format** button will be enabled.

Convert Results for all Beams: When this option as selected, all results on the beam ends of the Bladed model will be converted to results in SESAM's format and stored in a _R1.SIN for each Bladed results file. Since the Bladed project file does not contain yield information for the materials, the user can enter yield manually. This is not necessary for FLS analysis, but needed for code-check in GeniE.

Convert Results for one Node to .wind#: Enter a valid node number between 1 and the total number of members.

Member End 1/Member End 2: Select which end of the beam to convert loads from.

The converter will display the **Node Number** and the **Coordinate xyz** based on the member number selected in the **Member #** field.

While converting, the Bladed Converter will also make a BladedToGenie.js-file. This file can be used to create the Bladed Model in GeniE.

5 DATA FLOW DESCRIPTION

The following images describe the data flow for a simulation with or without wind turbine loads respectively.

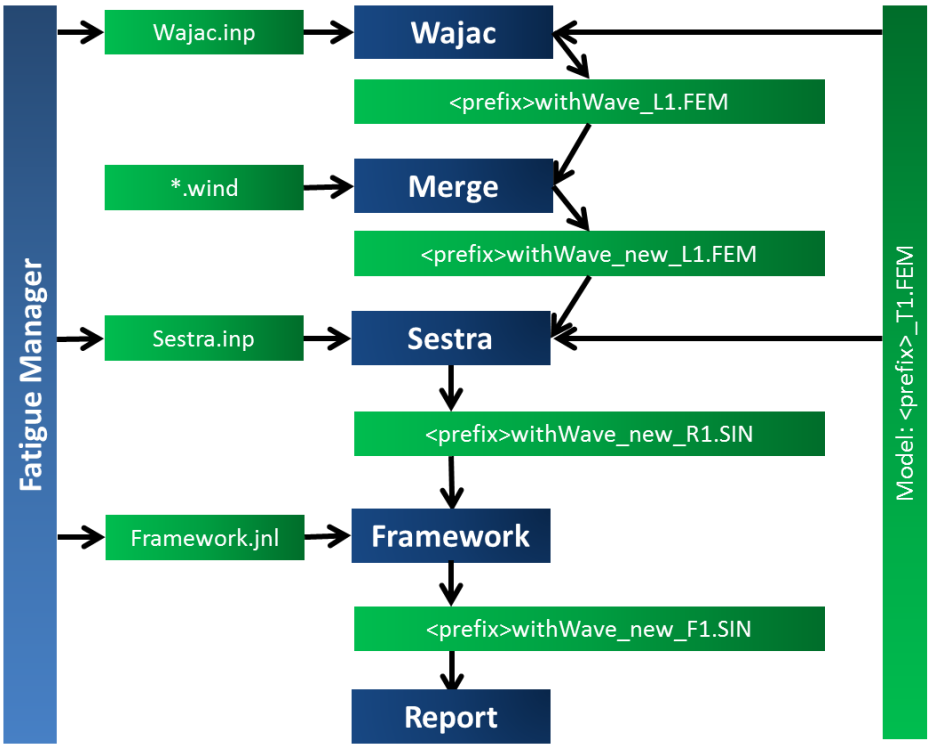


Figure 5.1: The data flow for a combined wave/wind simulation.

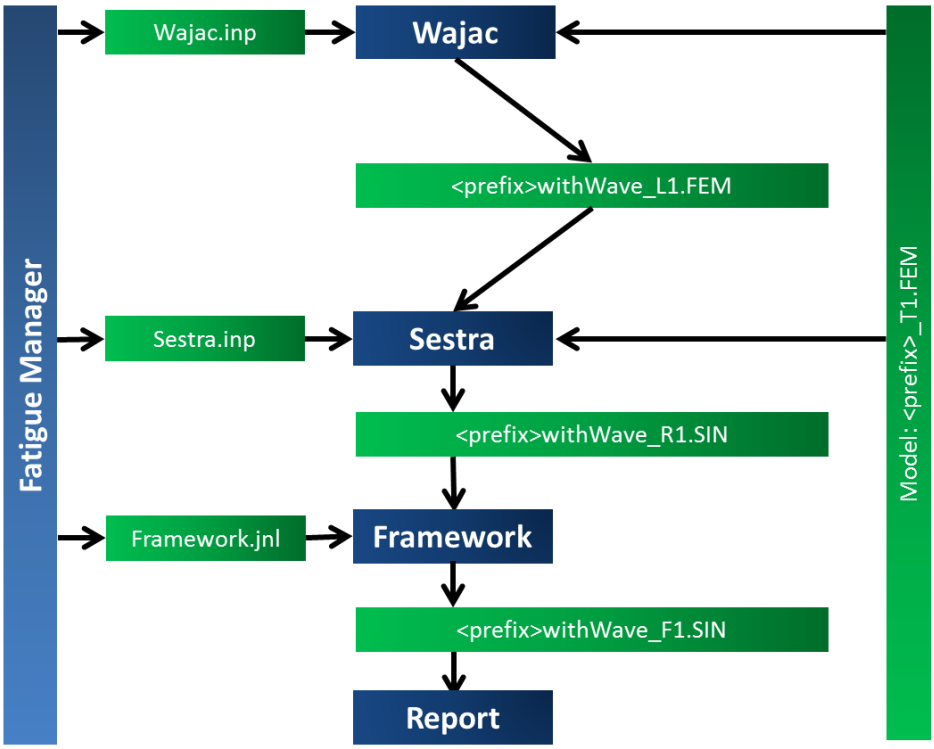


Figure 5.2: The data flow for a wave only simulation.

6 RAINFLOW COUNTING METHOD

Time series of the member forces are obtained by performing a quasi-static or dynamic time domain analysis in Sestra for the time history loading. If both wave and wind loads are present, the loads must be combined before running Sestra. Time series of the wave loading may be established by performing time series wave load analyses in Wajac. Time series of wind loading are not possible to generate in Sesam and must be provided from other sources, e.g. load files established from in-field wind measurement data. These are then read and merged into the Sesam analysis using the Wind step in the analysis. A fatigue analysis can then be performed in Framework, identifying the stress ranges at member positions in the selected parts of the model and applying the rainflow counting method to find the partial damage of each run. The damage is then summed by Fatigue Manager after all runs have been analysed in Framework.

By default Framework calculates fatigue damages per unit time and no fatigue safety factors are applied. It is therefore important to complete the Framework input file.

For more information on the time history fatigue analysis, rainflow counting method and on how to specify the fatigue input data, please refer to the Framework user manual.

About DNV GL

Driven by our purpose of safeguarding life, property and the environment, DNV GL enables organizations to advance the safety and sustainability of their business. We provide classification and technical assurance along with software and independent expert advisory services to the maritime, oil and gas, and energy industries. We also provide certification services to customers across a wide range of industries. Operating in more than 100 countries, our 16,000 professionals are dedicated to helping our customers make the world safer, smarter and greener.

Software

DNV GL is the world-leading provider of software for a safer, smarter and greener future in the energy, process and maritime industries. Our solutions support a variety of business critical activities including design and engineering, risk assessment, asset integrity and optimization, QHSE, and ship management. Our worldwide presence facilitates a strong customer focus and efficient sharing of industry best practice and standards.