WHEN TRUST MATTERS

FIXED OFFSHORE WIND STRUCTURE DESIGN

What Sesam and Bladed can do for fixed offshore wind turbine structure design and analysis.

April 2022

WHITE PAPER

Table of contents

1 EXECUTIVE SUMMARY

This document aims to guide the reader through the different analyses related to fixed offshore wind turbine support structures and how Sesam and Bladed support those.

Sesam can perform a multitude of different analyses applicable to the offshore wind turbine (OWT) support structure industry, based on many years of proven technology from the offshore oil and gas industry, and extended with new functionalities specific for the offshore wind industry in accordance with international standards such as IEC61400-3-1, the DNV Standards DNV-ST-0126 (Support Structures for Wind Turbines) and DNV-ST-0437 (Loads and Site Conditions for Wind Turbines) and the DNV Recommended Practices DNV-RP-C203 (Fatigue Design of Offshore Steel Structures) and DNV-RP-0585 (Seismic Design of Wind Power Plants).

In preliminary design, Sesam for fixed offshore wind turbine structurescan be used for modelling and the various types of analysis. The support structure can be modelled in the 3D modelling environment. Benefits during modelling include reference point modelling and parametric scripting, resulting in a powerful interface to perform a trade-off study of several conceptual designs in a quick and efficient manner. Some of the analyses that can be performed in the conceptual design phase are natural frequency analysis (eigenvalue analysis), ultimate limit state (ULS) and serviceability limit state (SLS) analysis including member and joint code checking, as well as fatigue limit state (FLS) analysis using damage equivalent loads or wave loads. Non-linear pile-soil analysis can be performed in these static analyses, whereas equivalent linearized pile-soil spring matrices to be used in dynamic analyses can be automatically obtained by the software.

In the detailed design phase, Sesam for fixed offshore wind turbine structures can be used to perform time domain analyses from a customized workbench, Sesam Wind Manager. Sesam Wind Manager can perform both fatigue analysis (FLS) as well as ultimate strength analysis (ULS) and earthquake analysis in the time domain. These analyses can be perfomed in two ways, either using a superelement approach or a fully integrated approach:

- In the superelement approach, the model and wave (and optionally, seismic) loads are converted into a superelement and load files by Sesam. The wind turbine is simulated together with the superelement and loads in an aeroelastic tool (e.g. Bladed), after which the wind turbine loads are extracted at the interface point between superelement and turbine. Wind turbine loads from any third-party wind turbine tool can be used in Sesam, and converters are available for Bladed, BHawC, VTS/Flex5 and HAWC2. These loads are then merged into the analysis in Sesam, followed by a time domain analysis to obtain the stress time histories in the structure. These stresses are then post-processed for FLS and/or ULS requirements. Superelement generation is available for Bladed, Siemens Gamesa's BHawC and MHI Vestas' VTS/Flex5.
- In the integrated approach, the model can be created in Sesam, after which it is converted to Bladed and combined with the (tower and) rotor-nacelle assembly. After the analyses are run in Bladed, all results of the complete structure are converted from Bladed into a Sesam result file. The stress time histories are then post-processed in Sesam for FLS and/or ULS requirements. In recent years, the integrated approach is being used more in-house by foundation designers too, running both Bladed and Sesam, for R&D purposes or early stage development. To support these activities, DNV offers wind turbine concept models and/or assistance in running the software.

Reports for FLS and ULS results are generated automatically. For FLS the report includes the combined fatigue damage over all included design load cases, giving a good insight into the critical locations in the structure and the critical design load cases, and can take into account changes of the SN curve over the lifetime of the structure. For ULS the report includes an overview over the worst load case and utilization factor for each member and joint in the structure, taking into account load factors. Detailed results per position in the model per design load case can be reported as well. The results can be visualized with colour coding on the 3D model using Sesam Insight as well.

After the analyses have been performed, redesign can be performed in an efficient manner, without the need to re-run the full analysis for intermediate checks. During redesign the code check results are based on changes in the section or material properties using the initial analysis results. This will give results that are sufficient during redesign. When all design changes have been performed, consistency between model and analysis results is established and updated code check results are obtained.

Sesam contains the latest versions of all main offshore standards, such as API, AISC, Danish Standard, Eurocode, ISO and Norsok for ultimate strength analysis. The fatigue analysis in the time domain is based on rainflow counting and can automatically compute the stress concentration factors (SCFs) for every hotspot in the structure based on the applied loads, both for tubular joints and for butt-welds. A library of SN curves is included and the splash zone limits can be entered so that corresponding SN curves are applied automatically if desired.

All time domain analyses can be run in parallel, either locally or in the Sesam Cloud, thereby significantly reducing the analysis time required. When using Sesam Cloud, multiple design iterations can be run within a single day. The reduced analysis time results not only in time and cost savings, but also allows for further structural optimization and thereby further cost reductions.

Conversion tools to/from other software are included. Wind turbine loads can be extracted from results from Bladed (although Bladed can output these directly in Sesam format if desired too), BHawC, VTS/Flex5 and HAWC2, and loads or displacements from any (other) wind turbine tool can be used when outputted as a simple text file. Conversions from Sesam to a Bladed foundation model, a Bladed superelement model, a BHawC (Siemens Gamesa) superelement model and a VTS/Flex5 (MHI Vestas) superelement model are possible. Structural models can be imported from SACS, Ansys, Staad, Nastran, SOLIDWORKS (Acis SAT or DXF format), etc.

Besides primary steel design, Sesam can be used for secondary steel design of typically boat landings and J-tubes, including (operational and accidental) boat impact analysis and vortex-induced vibration analysis of J-tubes. Other analysis during the life-cycle, such as transportation, lifting and corrosion protection, can be performed as well. All Sesam modules use the same model, thereby easing the process of running multiple analyses. Sesam can also be used for designing ofsfhore substations, installation (jack-up) and other vessels, as well as to perform fatigue and strength analysis of power cables.

2 INTRODUCTION

Sesam software has been used to analyse marine and offshore structures since 1969. The long experience and constant improvements of the software have made it one of the most used software applications in the offshore industry. Its strong history with offshore oil and gas, combined with hands-on experience with wind turbines, has made DNV a global leader in risk management of offshore wind projects. Sesam for fixed offshore wind turbine (OWT) structures and Bladed are based on this knowledge and use the proven technology from the offshore industry as well as functionalities developed based on requirements specific for the offshore wind industry.

Sesam for fixed OWT structures is a tailor-made solution for structural strength analysis of fixed offshore wind turbine structures, addressing the industry's need to account for the combined effect of wind turbine, hydrodynamic and seismic loads. The analysis functionality offered is in accordance with international standards such as IEC61400-3-1, the DNV Standards DNV-ST-0126 (Support Structures for Wind Turbines) and DNV-ST-0437 (Loads and Site Conditions for Wind Turbines) and the DNV Recommended Practices DNV-RP-C203 (Fatigue Design of Offshore Steel Structures) and DNV-RP-0585 (Seismic Design of Wind Power Plants).

Bladed is the industry-leading wind turbine design package and is well-integrated with Sesam. It is used not only by wind turbine manufacturers, but can also be used by foundation designers and other parties involved in the design of offshore wind turbine structures, and is therefore also referenced at multiple places in the document.

This document aims to guide the reader through the different analyses related to fixed offshore wind turbine support structures and how Sesam and Bladed support those. Floating wind is not part of this document, and the reader is instead referred to the separate whitepaper for that, availabe through www.dnv.com or available upon request via digital@dnv.com.

For more information, questions or to learn best practice in using Sesam and Bladed for offshore wind turbine support structure analysis, please contact us via digital@dnv.com.

DNV

3 PRELIMINARY DESIGN

The following is a selection of analyses involved in the preliminary design of a support structure that can be performed in Sesam.

3.1 Modelling

Modelling of the support structure is done in the 3D modelling environment of Sesam GeniE. Any kind of structure can be modelled, from monopile to tripod and jacket type structures, including secondary steel, such as boat landings and J-tubes, if desired.

Beams and joints can be modelled as concepts. For example, a leg can be modelled as one (segmented) member, instead of multiple members spanning from joint to joint. GeniE will take care of meshing this concept member into multiple mesh elements automatically, taking into account connections to other structure and changes in section and material. Joint modelling includes automatic creation of cans and stubs, as well as options to flush braces, add gaps and add local joint flexibilities.

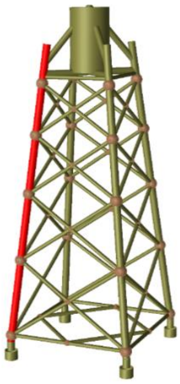


Figure 3.1: The leg is modelled as a beam concept.

GeniE offers the capability to model both beam and shell models, as well as a combination of these. This means that complex transition pieces and complex joints can be modelled as local shell models within a beam model. Tubular joints can easily be converted from beam to shell elements. This allows the user to obtain stress concentration factors for these complex local shell models. Besides modelling parts explicitly, it is possible to include external matrices representing parts of the model, such as the transition piece, suction buckets or joints.

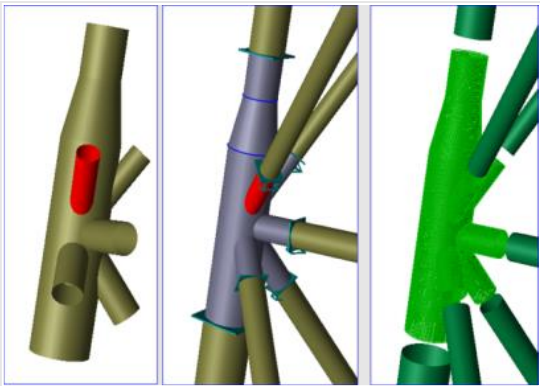


Figure 3.2: A joint modelled with shell elements within a beam model.

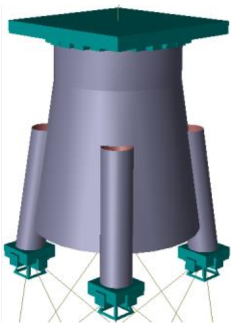


Figure 3.3: Shell transition piece connected to the jacket and tower beam structures.

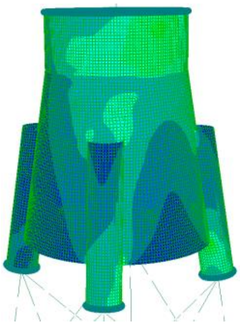


Figure 3.4: Results showing mesh with the stress distribution over the transition piece.

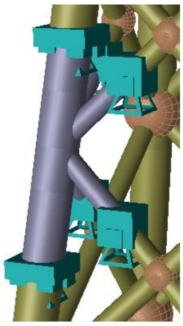


Figure 3.5: A joint designed with shells together with the jacket beam model.

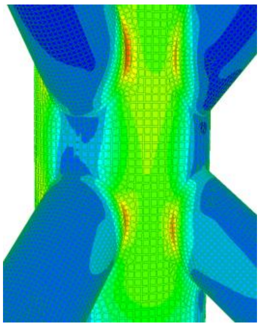


Figure 3.6: Results showing mesh with the stress distribution over the joint.

Environmental conditions, such as hydrodynamic effects (wave, current, flooding, marine growth) can be applied to the model, as well as non-linear pile-soil interaction. Automatic pile-soil linearization is included, as natural frequency analysis, dynamic analyses and superelement generation for the aeroelastic tool require a linear model.

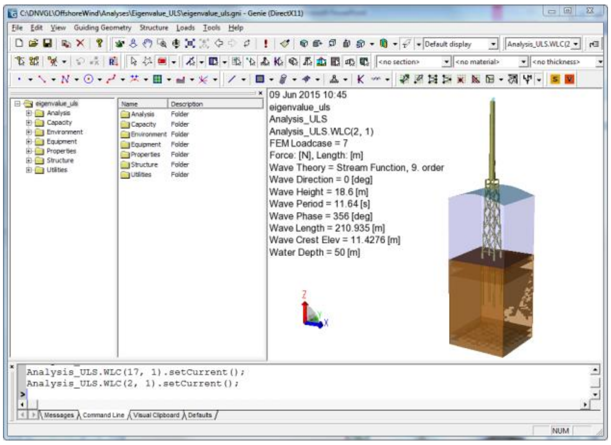


Figure 3.7: A jacket including wave and soil environment in GeniE.

Benefits during modelling include reference point modelling and parametric scripting, resulting in a powerful interface to perform a trade-off study of several conceptual designs in a quick and efficient manner. Code checking, including redesign options, are included in GeniE as well.

GeniE and other Sesam modules can be used within Sesam Manager, providing further possibilities of automating workflows.

3.2 Natural frequency analysis

Natural frequency analysis is easily run from within GeniE.

The effects of the tower and rotor-nacelle assembly as well as hydrodynamic added mass can be included in the analysis.

In the eigenvalue analysis options, the user can indicate the desired number of mode shapes to be calculated. If required, modal participation factors can be obtained too, to determine the importance of each mode.

Visualization options are included in both GeniE and Xtract, with animation of the mode shapes for easy identification of the modes being available in Xtract.

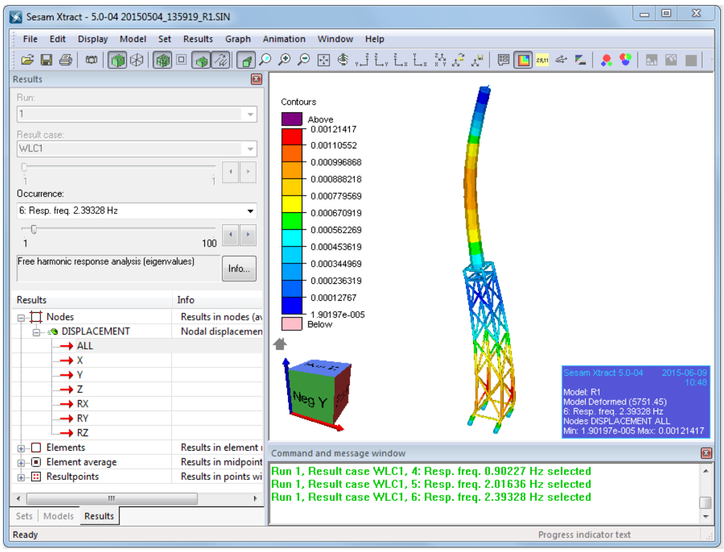


Figure 3.8: An example mode shape of a jacket structure visualized in Xtract.

3.3 FLS analysis using damage equivalent loads and waves

By applying cyclic loads onto the structure, it is possible to obtain the fatigue damage for damage equivalent loads (DELs). The DELs can be applied to the structure through point loads (e.g. representing the wind turbine loads). The number of occurrences per load cycle is indicated in the fatigue analysis.

Fatigue due to hydrodynamic loads corresponding to wave conditions can also be simulated. The combined fatigue damage from DEL and wave loads can then be calculated.

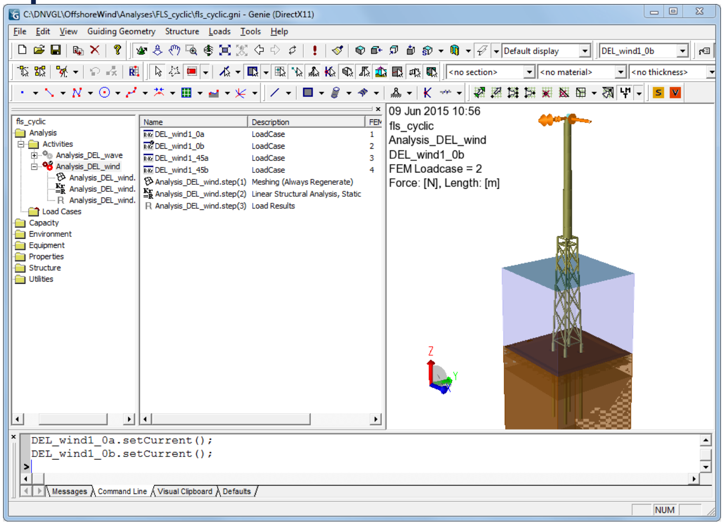


Figure 3.9: A damage equivalent load can be modelled in GeniE as a load case.

Stress concentration factors (SCFs) for joints and butt welds can be computed automatically (e.g. through Efthymiou equations) and/or assigned manually. A library of SN curves is included and subtypes (e.g. in air, free corrosion, or with corrosion protection) can be assigned automatically based on defined splash zone limits. SN curves can also be specified and/or assigned manually. Miner's sum is used to find the total damage over all load cycles included.

3.4 ULS and SLS analysis using extreme loads

Extreme checks are included in GeniE for the main offshore codes, including API, AISC, Danish Standard, Eurocode, ISO and Norsok.

Both member checks and tubular joint checks are available. Code check parameters can be assigned both globally and locally.

Redesign capabilities are included, calculating the effect of property changes on the code check results using the initial analysis results, without the need of re-running the complete analysis at each step of the redesign.

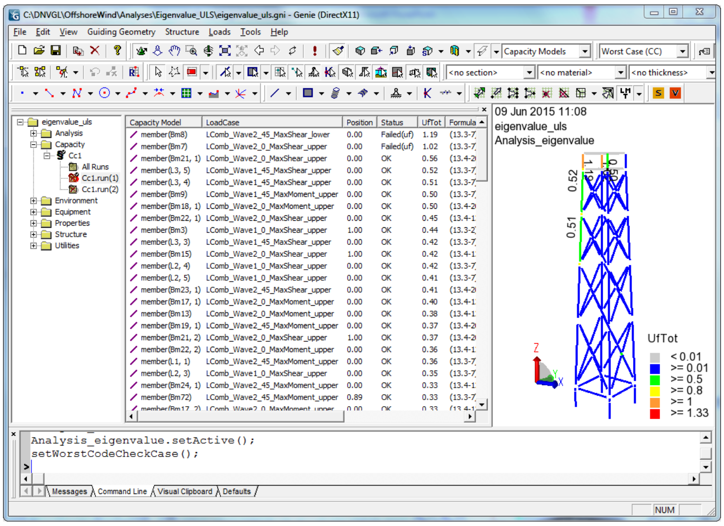


Figure 3.10: Member check results in the GeniE user interface tabulated and visualized in the 3D model.

Results (as well as model, loads, etc.) can be written to a report in tabulated form and including graphics.

4 DETAILED DESIGN

The following is a selection of steps involved in the detailed design of a support structure. Most of these analyses are performed using dynamic analysis in the time domain.

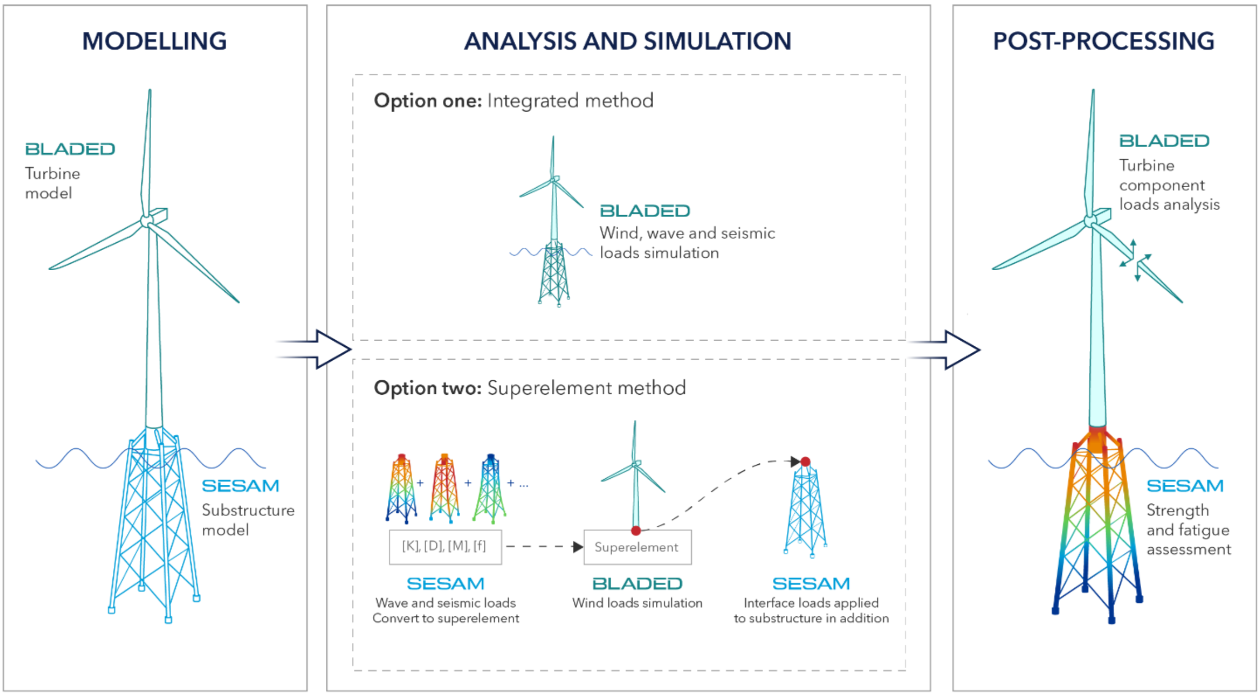


Figure 4.1: Industry workflows are supported – integrated and superelement.

The analyses can be performed in two ways:

- Integrated analysis: The modelling is done in Sesam. The model is then imported/ converted and linked to a wind turbine in a program such as Bladed, after which the resulting forces and moments are extracted for every beam in the structure. These results are then converted into Sesam format, where fatigue and ultimate strength analysis is performed.
- Superelement (or sequential) analysis: The modelling is done in Sesam. The model and the wave loads and (optionally) seismic effects are converted to a superelement and linked to a wind turbine in a program such as Bladed, BHawC, VTS/Flex5, HAWC2, etc. Structural analysis is then run in the wind turbine tool after which the forces and moments are extracted at an interface point. These loads are then applied to the model in Sesam, together with the wave loads and seismic effects, and the structural analysis is run. Fatigue and ultimate strength analysis is subsequently performed in Sesam.

4.1 Modelling

The same modelling capabilities are available for detailed design as for preliminary design, based on Sesam's GeniE modelling environment.

4.2 Wave load generation in time domain

Sea states can be modelled in time domain according to the standards. Hydrodynamic loads are computed by Wajac based on he waves, current, marine growth, flooding, and added mass, using Morison theory and optionally the MacCamy-Fuchs correction for large diameter monopiles. Wheeler stretching can be applied if desired.

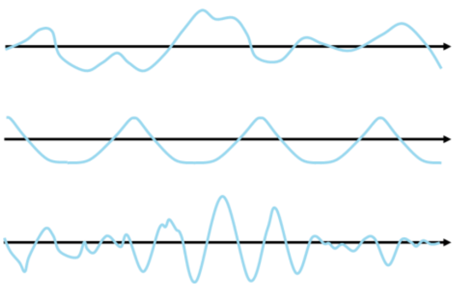


Figure 4.2: Irregular, regular and constrained waves can be used.

Multiple wave types are available, such as:

- irregular waves (Airy, random seeds) either using a single-peaked (PM, Jonswap) or double-peaked spectrum (Ochi-Hubble, Torsethaugen full/simplified)
- regular waves (Airy, Stokes, Stream, Cnoidal)
- constrained waves (irregular combined with NewWave and/or Stream function)

If desired, it is possible to import/export waves (surface elevation or wave components), e.g. to use measured waves or to use the same wave in another program.

4.3 Structural analysis in time domain

Structural analysis is performed using Sesam's finite element solver Sestra. Time domain analyses can be dynamic (either using direct time integration or modal superposition) or quasi-static.

4.4 FLS analysis in time domain

The fatigue limit state (FLS) analysis is performed using Sesam Wind Manager. The program is developed with the requirements of the offshore wind standards in mind. All design load cases (DLCs) can be set up according to their wind and sea state combinations, after which the total fatigue damage for each hotspot on each beam is summed and reported over the included DLCs, considering the relative occurrence of each DLC over the lifetime. Partial fatigue damage results per hotspot in the model per design load case can be reported as well. Changes in SN curves over the lifetime of the structure can be considered as well.

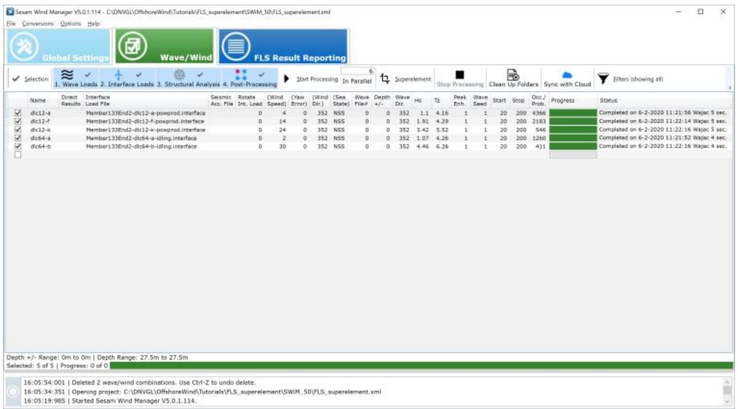


Figure 4.3: Sesam Wind Manager includes a grid to easily enter all design load cases for the analysis.

The fatigue analysis itself uses Framework and is based on rainflow counting of the stress time histories. It includes automatic computation of stress concentration factors (SCFs) for tubular joints, based on geometry or loadpath of the tubular joints in the model, as well as for butt welds.

SN curves can be assigned per hotspot around the cross-section and along each member. A library of SN curves is included and subtypes (e.g. in air, free corrosion, or with corrosion protection) can be assigned automatically based on defined splash zone limits. SN curves and SCFs can also be specified and/or assigned manually.

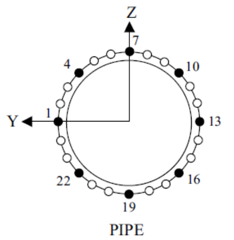


Figure 4.4: Hotspots around a tubular cross-section are used in the fatigue analysis.

Visualization and animation of the stress time histories for the model are available through Xtract. Visualization of the FLS results on the 3D model is available through Sesam Insight.

4.5 ULS analysis in time domain

The ultimate limit state (ULS) analysis is performed through Sesam Wind Manager, like the FLS analysis.

Both random sea states, regular sea states and constrained sea states can be set up from Sesam Wind Manager. Load factors for environmental loads, buoyancy and gravity can be included if desired.

The ultimate strength checks are performed through Framework, which allows for fast and efficient running of all code checks for members and tubular joints. Standards available include: API, AISC, Danish Standard, Eurocode, ISO and Norsok.

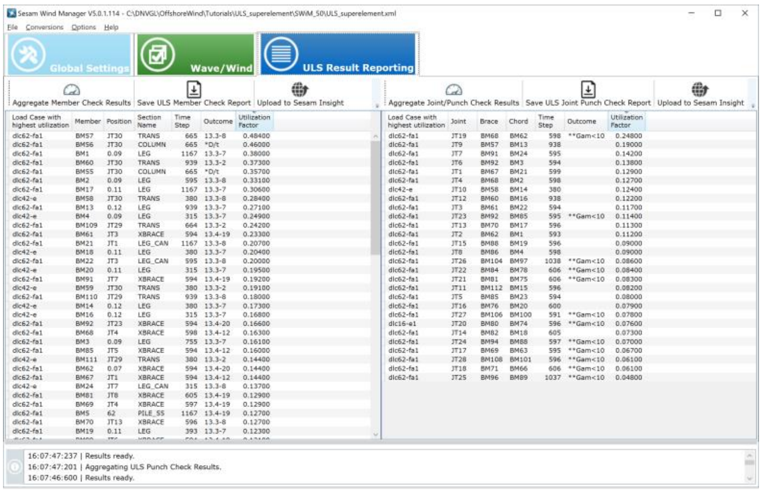


Figure 4.5: Code check results (worst utilization factors) reported in Sesam Wind Manager over all design load cases.

All time steps of all DLCs can be checked for ultimate strength checks, or a selection of time steps can be checked if desired.

ULS result reporting is included through Sesam Wind Manager. The report includes an overview over the worst load case and utilization factor for each member and joint in the structure, taking into account load factors, over all design load cases. Detailed results per position in the model per design load case can be reported as well. For each DLC, graphical and textual output can be generated. Visualization of the ULS results on the 3D model is available through Sesam Insight.

4.6 Earthquake analysis in time domain

Earthquake analysis can be performed in two ways in Sesam, one being in the frequency domain and one being in the time domain. For wind turbines, the time domain method allows for adding the wind turbine load time series and wave load time series.

The earthquake is applied to the structure through prescribed accelerations and/or displacements, based on a time history defined by the user. The applied seismic time histories can be defined individually per point in the model if desired, for example to take into account different seismic waves at different depths.

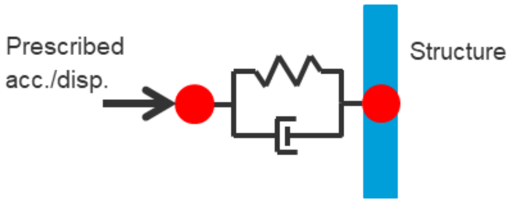


Figure 4.6: Seismic time histories can be included as prescribed accelerations and/or displacements. Optionally, a zero-length spring and/or damper can be included to represent the pile-soil spring stiffness and/or damping respectively.

Optionally, a zero-length spring and/or damper can be included to represent the equivalent linear pile-soil spring stiffness and/or damping matrix.

After the analysis, similar post-processing capabilities exist as for the ULS analysis in the time domain.

Seismic effects can also be used in a superelement analysis. Two options are available:

1. The superelement contains additional interface points, so that the seismic signal can be applied to the superelement in the aeroelastic tool. No seismic effects are included in the superelement loads file.
2. The seismic effects are included in the superelement loads file. No additional interface nodes are included in the superelement.

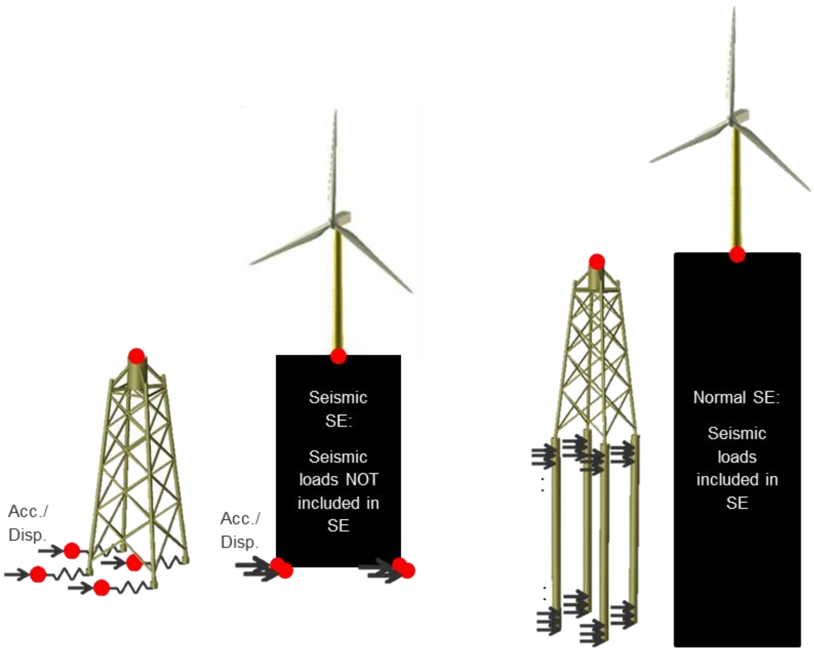


Figure 4.7: Various options for using seismic effects in a superelement analysis.

4.7 Post-processing of concrete structures in time domain

In addition to post-processing of steel structures, it is possible to perform design calculations of reinforced concrete shell structures using ShellDesign. ShellDesign is developed by Dr. techn. Olav Olsen and based on their extensive development and project experience. It uses Sesam results as input and supports ULS, ALS, SLS and FLS post-processing.

ShellDesign is based on plate and shell theory. It is applicable for both solid and shell FE models, for parts of structures where in-plane dimensions are much larger than out of plane. Supported design codes are NS3473, NS-EN1992-1-1 and DNV-OS-C502. It accounts for non-linear behaviour of reinforced concrete in sectional response. It also includes the option to calculate the non-linear structural response (using the Consistent Stiffness Method) and triaxial response (using the Modified Compression Field Theory).

Figure 4.8: Concrete GBS support structure.

4.8 Boat impact analysis

Boat impact, either operational or accidental (ALS), requires non-linear analysis capabilities. Sesam's Usfos is able to run these kinds of analyses based on a GeniE model. The boat impact can be defined on a boat landing or on any other beam in the structure. The plastic utilization of the model can be checked and push-over analysis can be performed to check the remaining capacity of the structure. See also [1] for more information.

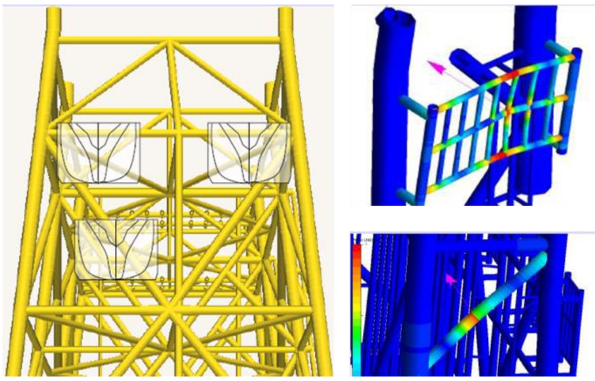


Figure 4.9: Usfos can perform non-linear analyses, such as boat impact analysis and push-over analysis.

4.9 Vortex induced vibrations of J-tubes

Using Framework's wind-induced fatigue functionality, it is possible to perform vortex induced vibration analysis of J-tubes. The wind fatigue module evaluates fatigue damage of frame structures subjected to wind loading. Buffeting loads due to wind gusts and the vortex shedding effects due to steady state wind can be considered.

4.10 Cathodic protection system design assessment

To identify areas of under and over-protection of the corrosion protection throughout the asset lifecycle, Sesam's FNCorrosion tool for managing the risk of corrosion can be used. The software sets itself apart in the market as an integrated part of a structural design system.

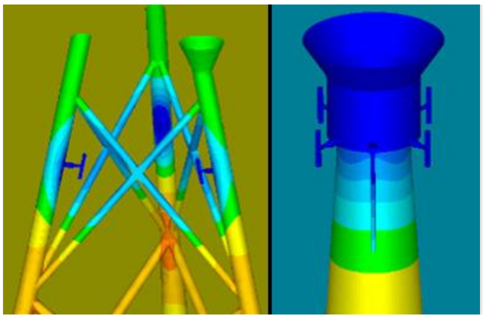


Figure 4.10: Surface corrosion potentials on a jacket and monopile.

4.11 Redesign

One of the strengths of Sesam is that redesign can be performed in an easy and efficient manner.

After running the analyses in Sesam Wind Manager, it is possible to make changes to the structural geometry, without the need to re-run the meshing and structural analysis, to see the updated (approximate) result of the FLS and ULS analysis. This is done by re-using the analysis results while using updated structural properties as part of the FLS or ULS post-processing. This allows for quick design iterations. Once all design changes have been made, the complete analysis can be re-run once to obtain the final results.

Redesign is available both when performing local runs as well as when performing cloud runs. In both cases, Sesam Wind Manager can pick up the structural analysis results from a previously run job to re-run the post-processing.

4.12 Cloud analysis and parallel computing

Sesam Wind Manager offers parallel computing, thereby significantly speeding up analyses of a large set of design load cases (DLCs).

In addition to local parallel computing, Sesam Wind Manager is integrated with [Sesam's Cloud Compute Services](). This enables users to analyse all DLCs in DNV's OneCompute cloud analysis platform, allowing for fast, simultaneous running of many DLCs. The status of the analyses can be monitored online and results are downloaded automatically once the runs are finished.

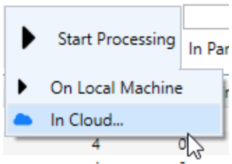


Figure 4.11: Sesam Wind Manager can run analyses in parallel locally or in the Sesam Cloud, speeding up analyses.

Significant time savings can be achieved by analysing the design load cases in parallel runs in the cloud, thereby enabling the performance of multiple design iterations in a single day[1]. The reduced analysis time not only results in time and cost savings, but also allows for further structural optimization and thereby further cost reductions.

[1] Based on measurements of FLS analysis for a selection of wind turbine jacket foundation models in detailed design projects run by Sesam customers. Analysing 2000 design load cases in parallel in the cloud took 45min-1hr (depending on jacket complexity, number of time steps and other settings). For a superelement analysis run one also needs to convert the Sesam model and wave loads to superelement format. The time required for this superelement file generation is not included in the run time here and would add some additional time to the Sesam part of the workflow.

5 TURBINE CONCEPT MODELS FOR INTEGRATED LOAD ANALYSIS

In recent years, foundation designers have increasingly been performing in-house analysis of the complete system of support structure and wind turbine. Reasons for this include foundation design initialization, sensitivity studies, additional structural design loops (outside of the turbine OEM) and learning process of turbine foundation design. This can allow the foundation designers to achieve a more optimized design from the early design phases in a project.

Such analyses are possible using Bladed and Sesam combined, and can use either the superelement or integrated approach (see chapter 4).

In order to perform this process, the foundation designer will need a wind turbine model. Foundation designers often either do not have the required data to model a turbine and/or controller, or they do not possess the competence to set up and run such wind turbine models and controllers. DNV can assist in such cases by offering wind turbine concept models. This empowers the non-turbine OEM stakeholders like foundation designers to perform integrated offshore wind turbine loads analyses.

Several wind turbine concept models are available upon request. This includes models representative of existing/coming commercial wind turbine models as well as turbine sizes beyond those available on the market today. The models are created by DNV's Turbine Engineering department (with a track record of 21 full turbine concept designs for turbine OEMs) and are based on public information and their many years of experience in turbine design. A closed loop and supervisory controller designed by DNV control experts can be included and tuned to the support structure frequency, enabling simulations for power production and other operating modes.

Figure 5.1: Combined turbine and support structure model in Bladed.

A wide range of turbine models are available off-the-shelf. At the time of writing, the following models are available:

- Ready off-the-shelf:
  - Concept 6MW 154
  - Concept 7MW 154
  - Concept 8MW 167
  - Concept 9.5MW 164
  - Concept 9.5MW 174
  - Concept 10MW 178
  - Concept 12MW 218
  - Concept 14MW 222
  - Concept 15MW 240
  - Concept 16MW 240
  - Concept 20MW 252
- Future plans:
  - Concept 20+MW

If desired, DNV can also run representative turbine models, acting as a wind turbine manufacturer and exchanging data with the foundation designer.

6 BEYOND IN-PLACE ANALYSIS

The analyses described above are all considering in-place analysis of the support structure. In addition to this, other analyses in the lifetime of the structure can be performed in Sesam. The benefit of this is that the same models can be used throughout the different analyses in Sesam.

6.1 Transportation analysis

Structural analysis of transportation of structures can be performed using GeniE and Sestra. Static loads due to ship accelerations (translational and/or rotational) can be considered and code checks can be performed on the structure.

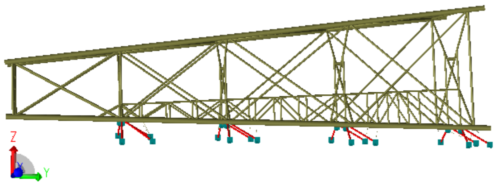


Figure 6.1: Transportation analysis of a large jacket with sea-fasteners.

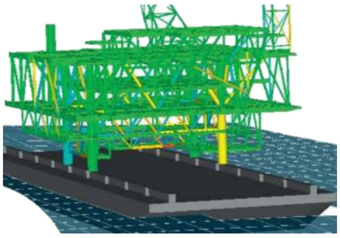


Figure 6.2: A large topside on a barge.

Sesam also offers tools (HydroD, Sima) to perform seakeeping and hydrodynamics analyses, hydrostatics and stability analyses and to simulate transport and other marine operations.

6.2 Lifting analysis

Structural analysis of lifting of structures can be performed using GeniE and Sestra. Static loads due to the lifting can be considered and code checks can be performed on the structure.

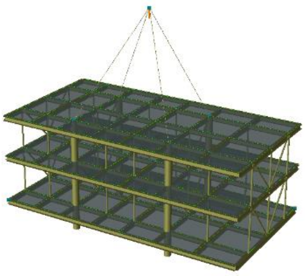


Figure 6.3: Lifting analysis of a topside.

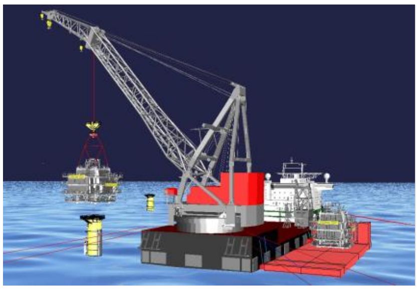


Figure 6.4: Lifting operations in Sima.

Sesam also offers tools (HydroD, Sima) to perform lifting operations, including through-surface effects, and other marine operations.

7 BEYOND OFFSHORE WIND TURBINE SUBSTRUCTURES

In addition to performing analysis on the wind turbine structures, Sesam can be used for other structures related to offshore wind.

7.1 Substations

Sesam can be used for the design and analysis of fixed and floating substations, such as jackets, monopiles or semi-submersibles.

Figure 7.1 Hohe See offshore wind farm substation. Source: DNV's S&LP customer story.

7.2 Installation (jack-up) and other vessels

Sesam can be used for the design and analysis of jack-up vessels used for offshore wind farm installation, as well as for the marine operations performed with such vessels.

Other vessels can be designed and analysed in Sesam as well, such as those used for crew transfer or O&M.

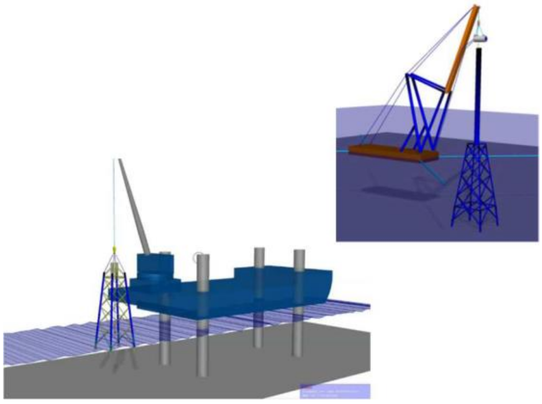


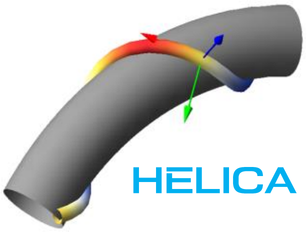
Figure 7.2 Lifting operations in Sima.

7.3 Power cables

Sesam's tools Helica and FatFree are applicable for the assessment of power cables, such as those inside a wind farm or connecting the wind farm to shore, both for fixed and floating wind farms.

Power cables are characterized by a multi-layered cross-section containing helix-shaped components. It makes the cable flexible in bending, but calculating mechanical properties and component stresses is not straightforward. Finite element analyses may be used. However, modelling of contact and friction between layers using finite element software is complex and time consuming. Consequently, simplifications and/or conservative assumptions are often made, causing either too conservative or unconservative results.

Figure 7.3 Power cable cross-section.



Helica software is a stress analysis tool that is tailor made for cross-section analysis of flexible pipes, umbilicals and power cables. Helica calculates mechanical properties, including stick and slip bending stiffness, load share between cross-section components, angle of twist as a function of tension, and allowable tension versus curvature. Most importantly, Helica calculates stresses in the cross-section due to combined loading, accounting for inter-layer contact due to tension and pressure, radial deformation, and stick/slip effects. A stress calculation in Helica is completed in just a fraction of a second, enabling efficient fatigue calculations, consistent with the response from the global analyses.

Helica performs short-term fatigue analyses based on histograms or time series imported directly from global analyses. Long-term fatigue damage is calculated by accumulating damage over all load conditions and directions. Helica also performs extreme load capacity checks in accordance with the ISO umbilical standard, and can calculate fatigue damage due to vortex-induced vibrations.

Helica can be used in combination with Sesam's FatFree to perform free span analyses of flexible pipes, umbilicals and power cables. FatFree is relevant for the seabed part of the cables before they are buried, or for the parts not being buried, for both bottom fixed and floating wind cable systems.

8 OVERVIEW OF SESAM AND BLADED SOFTWARE MODULES FOR FIXED OFFSHORE WIND STRUCTURES

This chapter gives a short introduction of the different Sesam programs and Bladed relevant for fixed offshore wind projects. For more information, also see the Sesam Feature Description available on https://sesam.dnv.com.

8.1 Sesam Wind Manager

Sesam Wind Manager is a cloud-enabled tool for time domain fatigue and ultimate strength analysis of fixed and floating offshore wind turbine support structures subjected to wave and wind turbine interface loads or displacements (and optionally seismic loading). Examples of for fixed offshore wind support structures are jackets, tripods and monopiles. After the analyses the results are combined over all design load cases and FLS/ULS results can be reported. It is also used to generate superelements and load files for wind turbine tools and for converting Sesam models to Bladed and results back. It is intended for use in the detailed design phase of offshore wind projects.

8.2 Sesam Manager

Sesam Manager is free for all Sesam users. It helps the user create and execute an analysis workflow based on previous best practices. It provides users with a single, shared environment for different Sesam modules and other applications – comprising all kinds of analyses needed. It supports any Sesam analysis, from simple to very complex, including the flexibility of Javascript. It is therefore very suitable to automate workflows in the various stages of an offshore wind project.

8.3 GeniE

GeniE is used for modelling of beams, plates and curved surfaces with stiffeners. Load modelling includes equipment with automatic load transfer, explicit loads, wave and wind loads and compartments. The model is transferred to Sestra for linear structural analysis, to Wajac for hydrodynamic analysis, to Splice for pile-soil analysis and to Usfos for non-linear structural analysis. GeniE includes predefined analysis set-ups (workflows) involving Wajac, Splice and Sestra. General basic results presentation can be carried out as well as code checking of members and tubular joints.

8.4 Wajac

Wajac calculates wind, wave and current loads on fixed and rigid frame structures. Typical examples of such structures are offshore jacket platforms and jack-up rigs. The loads are calculated according to Morison's equation (plus optionally MacCamy-Fuchs) deterministically, in the frequency domain or in the time domain simulation. Time domain simulation allows calculation of hydrodynamic loads due to irregular (random) waves, regular waves and constrained waves. Added mass due to marine growth, flooding, etc. can be included. Loads are transferred to structural analysis in Sestra.

8.5 Splice

Splice is a program for non-linear analysis of the structure-pile-soil interaction problem for structures supported by piles driven into the seabed. The non-linear soil stiffnesses are generated by Gensod (part of Splice) based on soil modelling performed in GeniE. Splice is also able to compute equivalent linear spring stiffness matrices to replace the non-linear pile-soil interaction, e.g. for application in a dynamic analyses.

8.6 Sestra

Sestra is the static and dynamic structural analysis program within the Sesam suite of programs. It is based on the displacement formulation of the finite element method. In addition to linear structural analysis, Sestra can analyse gap/contact problems as well as tension/compression-only members. Moreover, linear buckling, stress stiffening and inertia relief analyses may be performed.

8.7 Framework

Framework is a postprocessor for frame structures with the ability of performing fatigue analysis due to wave and/or wind, earthquake analysis and ultimate strength code checking.

8.8 Sesam Insight

Sesam Insight is an online tool that lets you easily share and collaborate on engineering models through any browser and across devices, such as computer, tablet and smartphone. There is no need for pre-installed desktop applications. It includes viewing of model and results data (such as FLS and ULS results) as well as revision and collaboration capabilities.

8.9 Xtract

Xtract is the model and results visualization program of Sesam. It offers general-purpose features for extracting, further processing, displaying, tabulating and animating results from static and dynamic structural analysis as well as results from various types of hydrodynamic analysis.

8.10 ShellDesign

ShellDesign is owned and developed by Dr.techn. Olav Olsen and sold and supported by DNV. ShellDesign is a design program and post-processor for reinforced concrete shell structures subjected to stresses in-plane and out-of-plane.

8.11 Usfos

The Usfos software module is a special purpose non-linear program for progressive collapse and accident analysis of jackets, topsides, floaters and other frame type structures. Accidental damage caused by explosion, fire, dropped objects, extreme environmental events and ship collision poses a major threat to the safety and operation of offshore structures.

8.12 FNCorrosion

Sesam's FNCorrosion tool lets you visually simulate, test and evaluate cathodic protection systems throughout the asset lifecycle. The software provides the ability to visualize the surface potentials and current density in 3D and run multiple 'what-if' scenarios showing the levels of protection around the submerged structure. It provides assurance that the selected cathodic protection system will protect fixed and floating structures and any subsea equipment.

8.13 Helica

Sesam's Helica software is a stress analysis tool that is tailor-made for cross-section analysis of flexible pipes, umbilicals and power cables. Helica facilitates extreme load capacity checks and highly efficient (short-term and long-term) fatigue analyses consistent with the global response determined through dynamic analysis.

8.14 FatFree

FatFree software is used for the design, assessment and reassessment of submarine pipeline spans in compliance with the DNV Recommended Practice DNVGL-RP-F105. The software enables the user to design fatigue lifetime for new free spanning pipelines due to Vortex Induced Vibrations and direct wave loading and re-assess fatigue lifetime of pipelines in operation.

8.15 Bladed

DNV's wind turbine design tool Bladed is renowned as the industry leading device modelling tool. For over 30 years, Bladed has been the industry standard aero-elastic wind turbine design tool, providing critical insight into turbine dynamics and optimization. It is closely integrated with Sesam for both integrated design and superelement analysis.

9 INTEGRATION OF SESAM WITH OTHER SOFTWARE

Besides the import and export formats that are available for model exchange in GeniE, Sesam Wind Manager has the capability to exchange data with some wind turbine-specific analysis tools.

9.1 Integration between Sesam and Bladed

Models can be taken from Sesam to Bladed by the provided converter in Sesam Wind Manager, either as a complete foundation model or as a superelement model and load files (see [2] for more information on both methods).

Automatic conversion into Sesam format is included from Bladed. Two options are available, either importing the results for the full structure in an integrated analysis (i.e. the complete load time history of each beam) for post-processing in Sesam, or importing the wind turbine loads time series at the interface (the latter can also be exported directly from Bladed in Sesam format, see [2] for more information on both methods).

9.2 Integration between Sesam and third-party software

Conversion of Sesam models and loads into Siemens Gamesa's BHawC (superelement) format (see [3] for more information) and MHI Vestas' VTS/Flex5 (superelement) format is available through Sesam Wind Manager. Automatic conversion of turbine interface loads from BHawC (see [3] for more information), VTS/Flex5 and HAWC2 into Sesam format is available in Sesam Wind Manager too.

It should be noted that wind turbine load or displacement time series from any third-party tool can easily be used in Sesam. The loads (or displacements) are read as a simple text file with seven columns, which include time and forces/moments (or translations/rotations) in 6 degrees of freedom.

For the foundation model, many import formats exist in Sesam for model import from other tools, such as SACS, Ansys, Staad, Nastran, SOLIDWORKS (ACIS SAT or DXF format), etc. Sesam also supports a variety of model export options. For example, it allows geometry to be taken into programs as SOLIDWORKS (ACIS SAT or DXF format) to create fabrication drawings or into Ansys for detailed bolted flange connection analyses.

10 REFERENCES

[1] J. Cunha, “Boat impact in fixed offshore structures,” DNV GL, Høvik, Norway, 2018.

[2] L. M. Alblas, “Verification report of Sesam’s Bladed interface (report no. 2016-0866, rev. 3),” DNV GL, Høvik, Norway, 2019.

[3] L. M. Alblas, “Verification report of Sesam’s BHawC interface (report no. 2016-0681, rev. 1),” DNV GL, Høvik, Norway, 2016.

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.