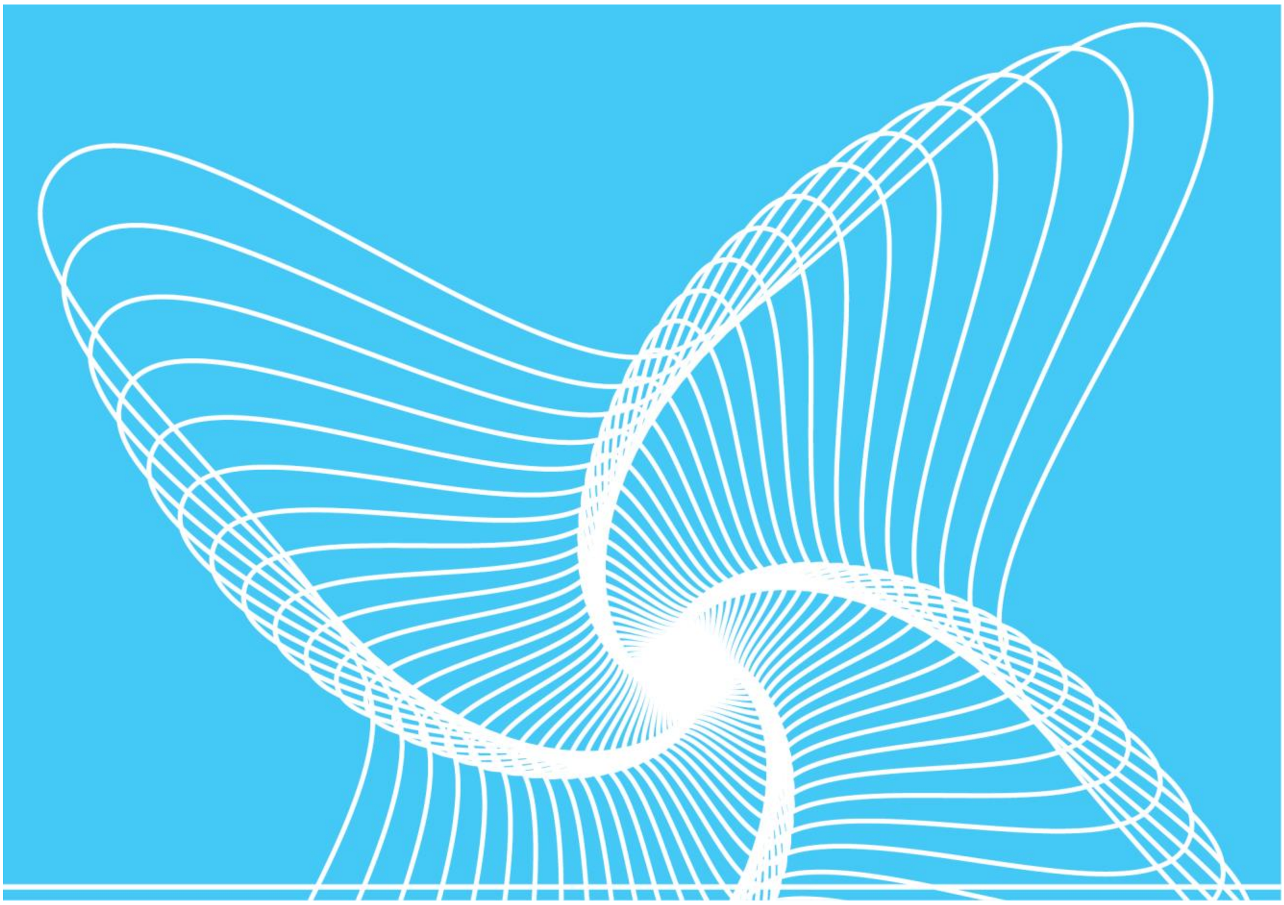
DNV·GL

FEATURE DESCRIPTION

Sesam

Software suite for hydrodynamic and structural analysis of ships and offshore structures



SAFER, SMARTER, GREENER

Last revised: March 12, 2019

Prepared by DNV GL – Digital Solutions

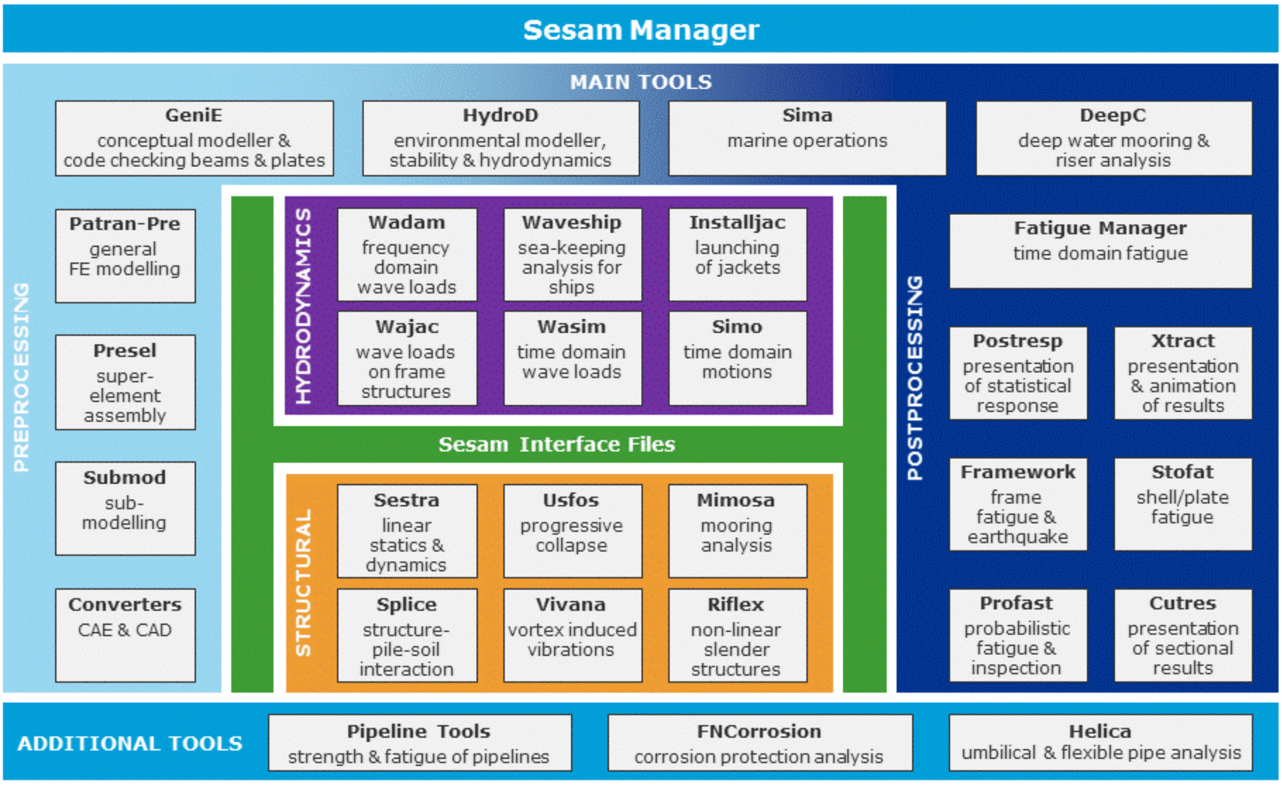
TABLE OF CONTENTS

Introduction to Sesam

Sesam is a software suite for hydrodynamic and structural analysis of ships and offshore structures. It is based on the displacement formulation of the finite element method. An overview of Sesam is shown below. The four groups of programs: **preprocessors**, **hydrodynamic analysis programs**, **structural analysis programs** and **postprocessors**, are bound together by a set of **Sesam Interface Files**, the green "H" in the figure. All major inter-program communication goes via this well-defined set of files.

Sesam Manager at top of the figure is the master control program of Sesam. Analysis workflows including any of the Sesam programs and of any complexity may be set up and run.

The main tools **GeniE**, **HydroD**, **Sima** and **DeepC** are through their features for modelling and controlling execution of analysis programs entry points to Sesam packages for specific industries. Typically, programs in the hydrodynamics and structural groups are run from GeniE, DeepC and HydroD.



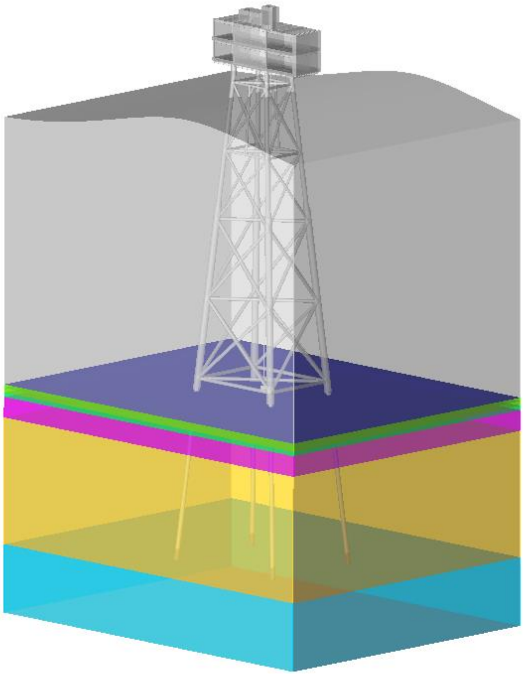
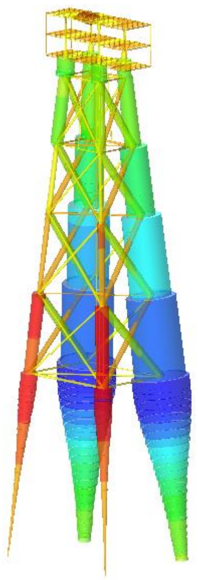
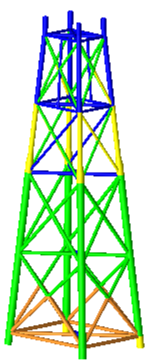
Sesam overview

This introduction to Sesam is organised in sections:

- Sesam packages – About the industry specific packages of Sesam
- Sesam Manager – About the master control program of Sesam
- Application Version Manager (AVM) – About the version control manager of Sesam
- Sesam Interface File – About the files binding Sesam together
- Import and export features of Sesam – About import from/export to CAE/CAD
- Hardware and operating systems – Sesam computer recommendations

Sesam packages

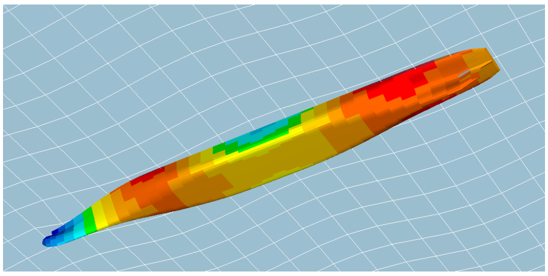
For specific industry applications, Sesam packages are available as described below. More details for the programs included in the packages are found in separate sections of this document.

<table>
<tr><th>PACKAGE</th><th>DESCRIPTION</th></tr>
<tr><td>Sesam for fixed structures

Jacket concept model with soil layer colouring
 
Axial force results Code check results</td>
<td>GeniE is the entry point to the packages for designing and analysing fixed offshore structures such as jackets, topsides, jack-ups, and offshore wind turbine (OWT) support structures.

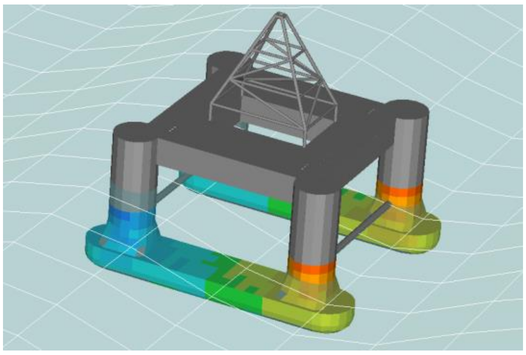
Through combining analysis programs with GeniE, the following application packages are offered.

General structure design
GeniE – modelling, analysis control, and code checking (limited model size)
Sestra – static structural analysis
Topside design
GeniE – modelling, analysis control and code checking
Sestra – static structural analysis
Jacket design
GeniE – modelling, analysis control and code checking
Wajac – computation of wave loads on frame structures
Splice – pile-soil analysis
Sestra – static/dynamic structural analysis
Framework – fatigue analysis of frame structures
Fixed OWT design
GeniE – modelling and code checking
Wajac – computation of wave loads on frame structures
Splice – pile-soil analysis
Sestra – static/dynamic structural analysis
Framework – fatigue analysis of frame structures
Fatigue Manager – analysis control for time series fatigue analysis under combined wind and wave loads
Usfos, the non-linear progressive collapse analysis program, is an optional add-on to the above.</td></tr>
</table>

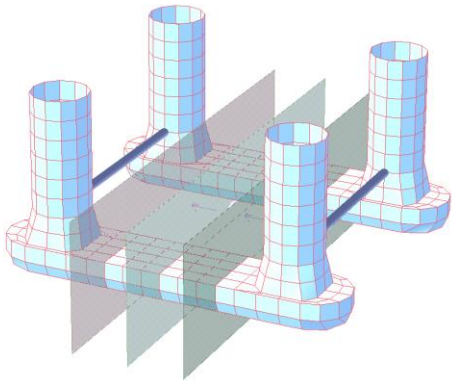
Sesam for floating structures



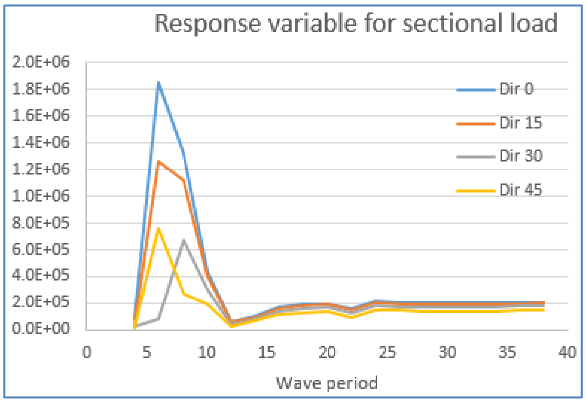
Ship hull with pressures viewed from underneath



Semisubmersible motion animation and hydrodynamic pressures



Semisubmersible panel model with load sections



HydroD and GeniE are the entry points to the packages for floating structures. By combining analysis programs with HydroD and GeniE, the following application packages are offered.

Stability analysis

- HydroD – modelling and stability analysis

Stability analysis extended

- GeniE Panel – floater modelling
- HydroD – modelling and stability analysis

Linear hydrodynamics

- HydroD – modelling and analysis control
- Wadam – freq. domain hydrodynamics
- Wasim – time domain hydrodynamics
- Postresp – statistical postprocessing

Linear hydrodynamics with forward speed

- HydroD – modelling and analysis control
- Wasim – hydrodynamics including forward speed
- Postresp – statistical postprocessing

Advanced hydrodynamics (add-on to above)

- Wadam – 2nd order hydrodynamics, wave/current interaction and multi-body
- Wasim – non-linear hydrodynamics including forward speed

Structural design

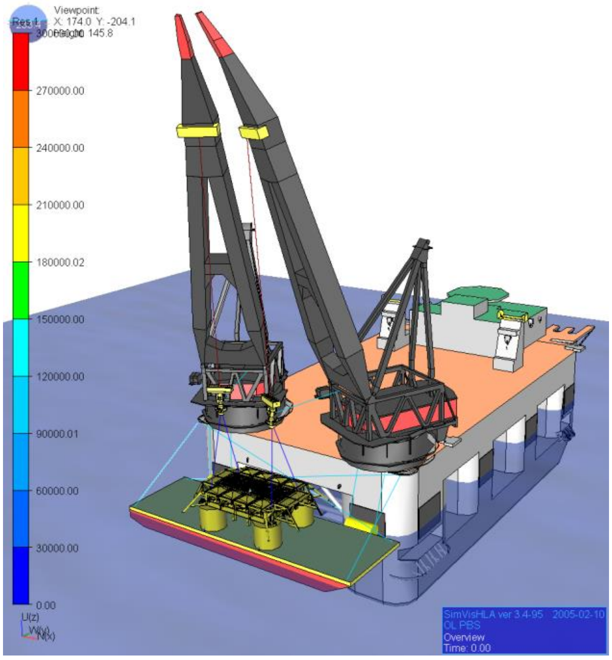
- GeniE – modelling, analysis control and plate code checking
- Sestra – static/dynamic structural analysis
- Xtract – FE results post-processing

Structural design extended

- GeniE – modelling, analysis control, and beam and plate code checking
- Presel – superelement modelling
- Submod – sub-modelling
- Wadam – wave load transfer extension
- Wasim – wave load transfer extension
- Presel – superelement modelling
- Sestra – static/dynamic structural analysis
- Stofat – fatigue of stiffened plates/shells
- Cutres – sectional results presentation
- Xtract – FE results post-processing

Xtract extension for animation is an optional add-on to the hydrodynamics and structural packages.

Sesam for marine systems



Sima is the entry point to packages for analysing and visualising marine systems in 3D.

Through combining analysis programs with Sima the following application packages are offered.

Marine operations

- Sima – modelling, analysis control and results presentation
- Simo – simulation of motions

Marine dynamics

- Sima – modelling, analysis control and results presentation
- Simo – simulations of motions
- Riflex – analysis of moorings

Marine dynamics extended

- Sima – modelling, analysis control and results presentation
- Simo – simulation of motions
- Riflex – analysis of moorings
- Vivana – vortex induced vibration analysis

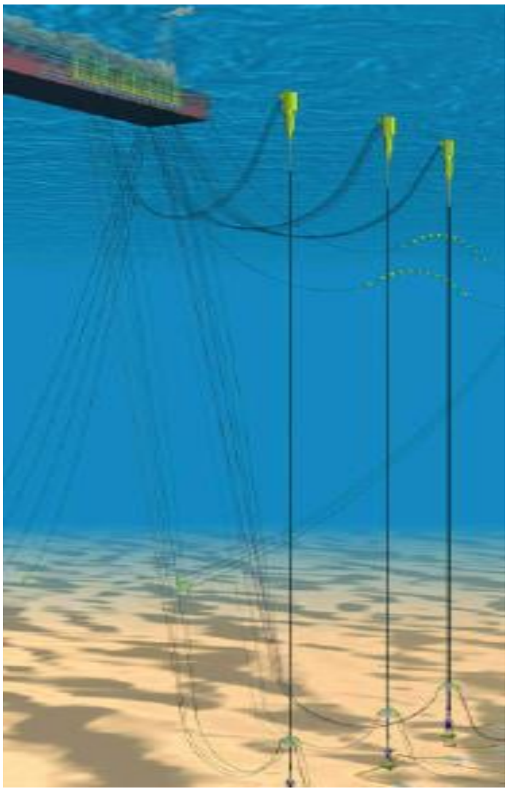
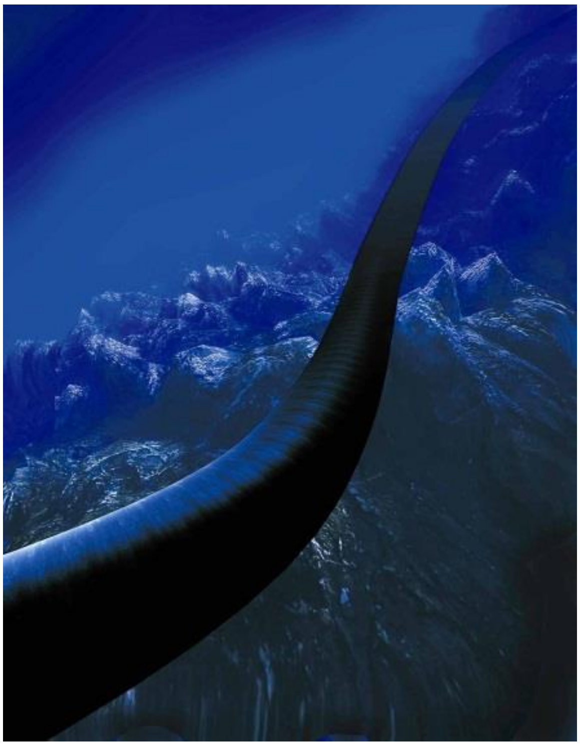
Floating OWT design

- Sima – modelling, analysis control and results presentation
- Simo – simulation of motions
- Riflex – analysis of moorings

Hydrodynamic coefficients

- HydroD – modelling and analysis control
- Wadam – frequency domain motion analysis to output mass/damping coefficients and RAOs

Vivana is an optional add-on to all packages except Hydrodynamic coefficients for which GeniE Panel is an optional add-on.

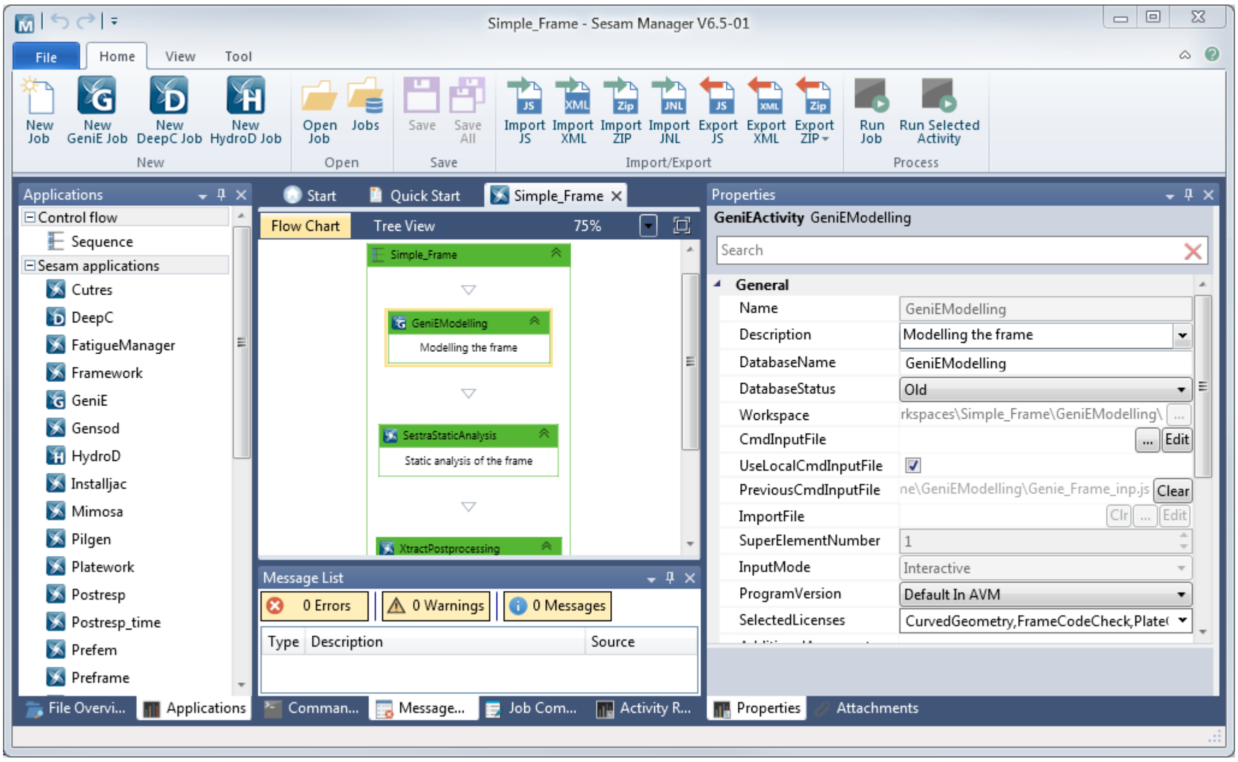
Sesam for moorings and risers 	DeepC is the entry point to this package for mooring and riser design. Through combining analysis programs with DeepC, the following application packages are offered. **Mooring and riser design** • DeepC – modelling, analysis control and results presentation • Simo – simulation of motions • Riflex – analysis of mooring lines **Hydrodynamic coefficients** * • HydroD – modelling, analysis control and results presentation • Wadam – frequency domain motion analysis to output mass/damping coefficients and RAOs * Offered as an optional add-on to the Mooring and riser design package together with GeniE Panel. Helica, Mimosa, Sima and Vivana are optional add-on programs to the Mooring and riser design package.
Sesam for pipelines 	Sesam for pipelines is a set of independent programs. • PET (Pipeline Engineering Tool) for early phase pipeline assessment covering different aspects of pipeline design. • StableLines for pipeline on-bottom stability based on DNV Recommended Practice DNV-RP-F109. • FatFree for analysis of free spanning pipelines according to the DNV Recommended Practice, RP-F105. • OS-F101 Code Compliance is related to the re-issue of the DNV Offshore Standard for Submarine Pipeline Systems. • RP-F101 Spreadsheet for assessment of corroded pipelines in compliance with DNV-RP-F101 • SimBuck for simplified global buckling analysis of submarine pipelines

Sesam Manager

Last revised: January 8, 2019. Describing version 6.6-02.

Sesam Manager manages Sesam analyses of any kind, from the simplest to the most comprehensive.

An analysis job is Sesam programs (applications) organised as activities in a workflow. The workflow may be of any length and complexity. Any other program/application may also be added to the workflow, e.g. your own program or an MS Office application.



Sesam Manager

Sesam Manager takes care of the data flow between the Sesam programs. The default file operation is transparent and can be modified to meet special requirements. Any document and file, e.g. analysis specifications and reports, may be attached to the job.

Taking advantage of the JavaScript® scripting language of Sesam Manager a job may be exported, edited and imported to establish a new revised job. A built-in ZIP import/export functionality allows jobs to be transferred between users whether they are in progress or completed.

In short, the purpose of Sesam Manager is to:

- Be a common starting point for all Sesam programs
- Ease the execution of Sesam programs and establish parts of the input
- Organise execution of the Sesam programs in the proper sequence for the task at hand
- Manage the files involved in an analysis project
- Establish workflow templates for analysis tasks of any complexity
- Provide easy archiving of an analysis job with its input and results files plus attachments

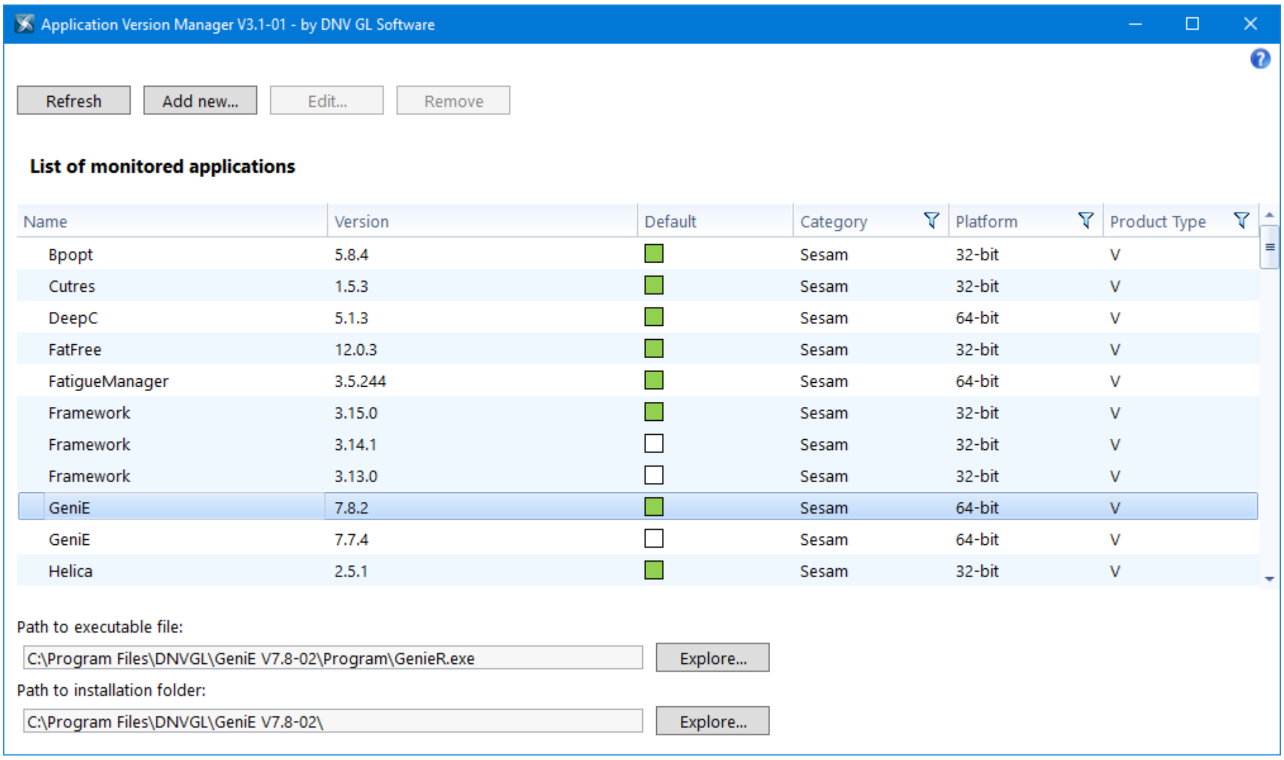
Applications Version Manager (AVM)

Last revised: April 9, 2018. Describing version 3.1.

Application Version Manager (AVM) provides an overview of Sesam program versions installed and offers control of which versions to be used. It also allows manually adding programs not installed as part of Sesam. Such programs are then easily available in e.g. Sesam Manager.

AVM is embedded in GeniE, HydroD and DeepC. This means that changing for example which Framework version is default takes effect next time GeniE is started and Framework is started from GeniE.

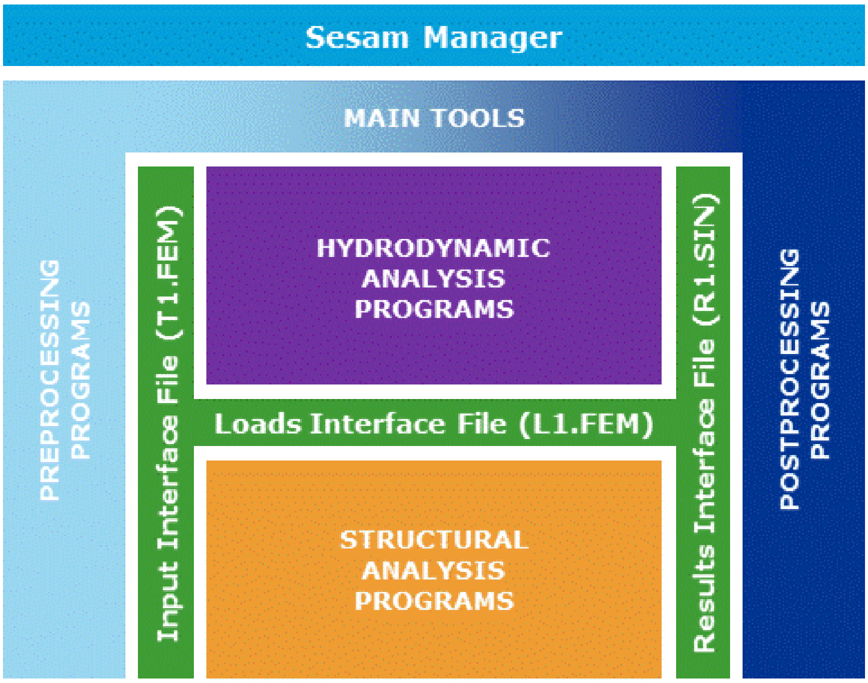
Start AVM as a separate application or from the Tool menu in Sesam Manager.



Applications Version Manager (AVM)

Sesam Interface Files

The Sesam Interface Files are comprised of a set of files for which the most commonly used names are T1.FEM, L1.FEM and R1.SIN. These are shown in the simplified Sesam overview figure below.



Sesam overview with focus on Sesam Interface Files

The Sesam Interface Files are comprised of the following:

- Input Interface Files – e.g. T1.FEM, T3.FEM, T21.FEM, etc.
  The model created by the preprocessors is contained in these files. The number in the file name can be any number from 1 to 9999 and is used to distinguish separate models (e.g. panel mesh for hydrodynamic analysis and FE mesh for structural analysis, different versions of the same model, different superelements, etc.). The short names T-file and FEM file are often used for these files. The contents of the file are:
  - FE/panel model with nodes, elements, material, boundary conditions and loads
  - 2nd or higher level superelements when using the multilevel superelement technique
- Loads Interface Files – e.g. L1.FEM, L3.FEM, etc.
  Hydrodynamic loads computed by environmental programs are stored in these files. They pertain to corresponding Input Interface Files: L1.FEM belongs to T1.FEM, L3.FEM belongs to T3.FEM, etc. The contents of the file are:
  - Hydrodynamic beam line and surface pressure loads, deterministic or transfer functions
  - Inertia and gravity loads
  - Point loads from anchor or TLP elements
- Matrix Interface Files – e.g. M21.SIF (or M21.SIU or M21.SIN)
  These files are for exchange of matrix data like stiffness, mass, damping and loads. The most common usage is exchange of data between Sestra and Splice. The contents of the file are:

    - Stiffness, mass and damping matrices
    - Load vectors
    - Nodal displacements
- Structural Results Interface Files – e.g. R21.SIN (or R21.SIF or R21.SIU)
  Structural (FE) analysis results are stored in these files ready for further processing by a postprocessor. The short names R-file and SIN file are often used for this file. The contents of the file are:
    - FE model (= Input Interface File)
    - Nodal displacements
    - Beam forces
    - Element stresses
- Hydrodynamic Results Interface Files – typically named G1.SIF (or G1.SIU or G1.SIN)
  Hydrodynamic rigid body motion results are stored in these files. The short name G-file is often used for this file. The contents of the file are:
    - Transfer functions for rigid body motion of floating structure
    - Hydrodynamic coefficients
    - Sea surface elevation and off-body kinematics
    - Transfer functions for base shear and overturning moments for fixed frame structure
    - Transfer functions for sectional loads
    - Transfer functions for forces and stresses in selected elements

Tools for conversion between Sesam and other formats, i.e. CAE and CAD programs, is covered in section Import and export features of Sesam below.

There are also auxiliary tools for manipulating the Sesam Interface Files:

- Loads Interface Files may be manipulated in various ways by the auxiliary program Waloco:
    - Merge two and more files from different Wajac/Wadam/Wasim runs
    - Renumber the load cases
    - Conversion between formatted and unformatted (FEM extension for both)
- Results Interface Files may be manipulated in various ways by the auxiliary program Prepost:
    - Merge two and more files from different Sestra runs
    - Copy data from one file to another
    - Conversion between formatted (SIF), unformatted (SIU) and database format (SIN)
    - Result combinations may be created (alternatively to creating combinations in GeniE)
    - Extraction of transfer functions for selected elements and results and storage on Hydrodynamic Results Interface Files (G-file)

Import and export features of Sesam

The table below presents the possibilities for import and export of model data between Sesam and other CAE and CAD data formats.

FORMAT	WHICH DATA	IMPORTED BY	EXPORTED BY
SACS INP file	Structure, weight, loads, load combinations, wave, wind loads on members, pile, soil and code checking data for member buckling lengths and moment amplification factors	GeniE Prepost except weight, pile and soil data	
StruCad3D S3D file	Structure and loads	GeniE and Prepost	
Spatial Technology ACIS SAT file	Structure only (surfaces)	GeniE	GeniE
CadCentre PDMS SDN (SDNF) file	Structure only, member system lines and eccentricities	GeniE	GeniE
Intergraph PDS SDN (SDNF) file	Structure only, member system lines and eccentricities	GeniE and Prepost	GeniE and Prepost
Rhinoceros (Rhino) GRC file	Guiding NURBS curves using a plug-in functionality in Rhinoceros, plug-in provided by DNV GL – Software	GeniE	GeniE
AutoCAD DXF file	Guiding points and NURBS curves, other curve definitions imported by script functionality	GeniE	
Ansys CDB + S0* files	Structure and loads	Prepost	Prepost
Nastran BDF + OP2 files	Structure, loads and results	GeniE and Sesam Converters	Sesam Converters
STAAD.Pro STD file	Structure, loads and load combinations	StaadToSesam	

Hardware and operating systems

Sesam is supported on Windows 7 and 10 (64-bit).

Minimum hardware recommendation

This recommendation is for tasks normally limited to jacket and topside design analyses including wave and pile-soil analysis. It is also suitable for modelling shell/plate superelements. Moreover, hydrostatic and smaller hydrodynamic analysis in frequency domain can be done.

- Graphics card: Open GL compatible. May be integrated with a processor (e.g. Intel HD).

- Memory: 4 GB
- Processor: Dual core
- 64-bit version of Windows operating system
- Storage: 200 GB
- Display: 17” supporting 1280x1024, alternatively laptop 15” supporting 1280x1024

Preferred hardware recommendation

This recommendation is for all types of Sesam analysis.

- Graphics card: Separate Open GL compatible graphics card (NVIDIA or ATI) with 512 MB graphics memory. If OpenGL is not supported, then use DX9 as provided in the Sesam installation.
- Memory: 16 GB
- Processor: Quad core
- 64-bit version of Windows operating system
- Storage: 500 GB
- Display: 24” supporting 1900x1200 (or-1080), alternatively laptop 17” supporting 1900x1200 (or-1080)

Graphics driver

By “graphics driver” below is meant the system level software provided by your Graphics Card supplier (most likely Intel, NVIDIA or ATI) to interface between Windows and the GPU. This is supplied with your operating system or graphics card.

By “GeniE graphics driver” below is meant the software used by GeniE to interface with the graphics driver defined above.

Use of DX9

DirectX 9.0 (DX9) is the preferred GeniE graphics driver and it is the default on installation.

DirectX 9.0c Runtime version 9.27.1734 distributed on June 2010 or a later version of DirectX 9 must be installed on your system. The Sesam installer will install DirectX 9.0c.

Windows 7 comes with DirectX 9 pre-installed. However, GeniE uses extra components so DirectX 9 must be explicitly installed using the Sesam installer or an installer from the Microsoft website.

The GeniE DX9 driver is supported on any DirectX 9.0 compliant graphics hardware (Microsoft Shader Model 3) with the latest vendor-supplied drivers.

DirectX 9.0c was first released in August 2004 so older systems will not support the DX9 driver.

Use of OpenGL

GeniE supports two different OpenGL drivers. The standard GeniE OpenGL driver is a legacy driver that attempts to support all OpenGL 1.1 hardware.

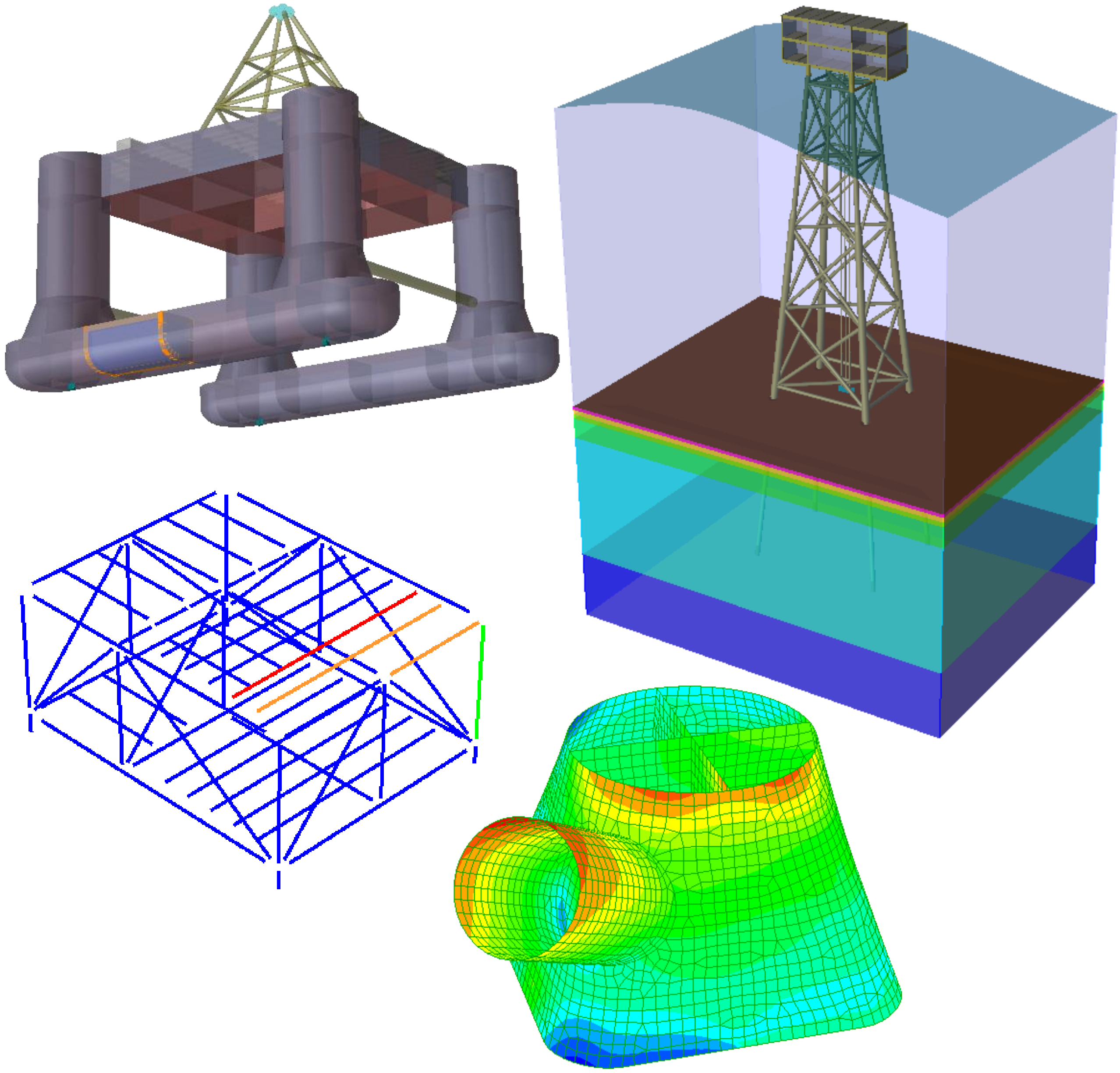
The GeniE OpenGL2 driver is a shader-based driver that is offered as an alternative should a user encounter problems with other drivers. It attempts to support all OpenGL 2.0+ hardware.

GeniE

CONCEPT MODELLING OF BEAM, PLATE AND SHELL STRUCTURES, ANALYSIS WORKFLOWS AND CODE CHECKING

Last revised: January 8, 2019. Describing version 7.11 (64-bit).

GeniE is a tool for concept (high level geometry) modelling of beams and stiffened plates and shells (curved surfaces). Load modelling includes equipment (with automatic load transfer), explicit loads (point, line and surface) and wind loads. The model is transferred to Sestra for structural analysis, to Wajac and Wadam for hydrodynamic analysis, to Splice for pile-soil analysis and to Installjac for launching and upending analysis. GeniE includes predefined analysis set-ups (workflows) involving Sestra, Wajac and Splice. General basic results presentation can be carried out as well as code checking of members and tubular joints.



FEATURES OF GENIE

The features of GeniE are organised in sections:

- Beam, plate and surface modelling
- Finite elements and features for meshing
- Modelling for structural analysis in Sestra
- Modelling for wave and wind analysis in Wajac
- Modelling for wave and motion analysis in HydroD/Wadam
- Modelling for pile-soil analysis in Splice
- Explicit (point, line, surface) load modelling
- Post-processing and reporting
- Member and tubular joint code checking – requires extension CCBM
- Supported standards for member and tubular joint checking
- Plate code checking – requires extension CCPL
- Import and export data in GeniE

GeniE has several extensions, i.e. features screened off for users of the basic version of the program. Access to an extension is subject to agreement and a valid license file. These extensions are:

CGEO – curved geometry modelling, includes partial meshing and all mesh editing except features covered by the REFM extension

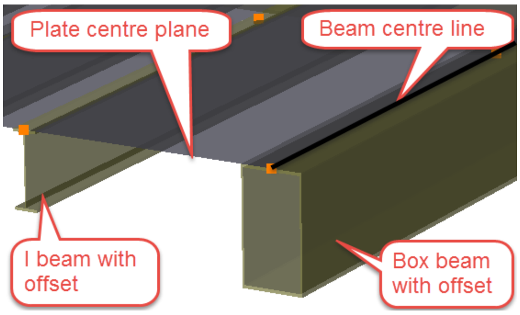
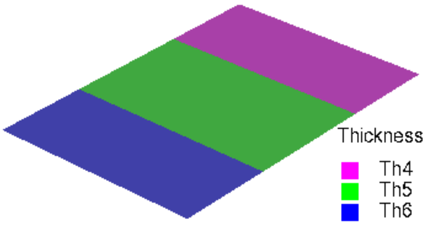
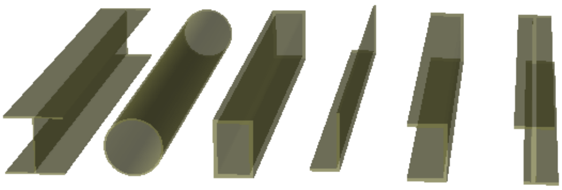
REFM – refined meshing, includes refine mesh for grid, edge and box, detail box for refined meshing and convert beam to plate

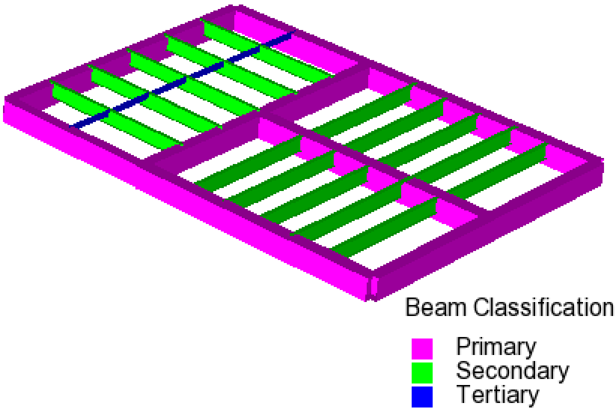
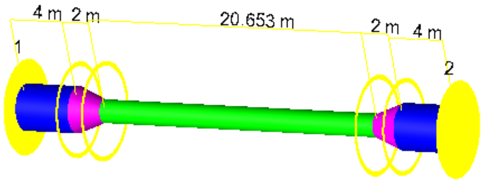
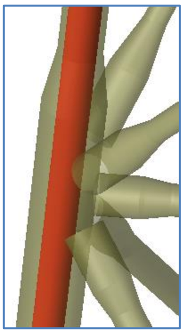
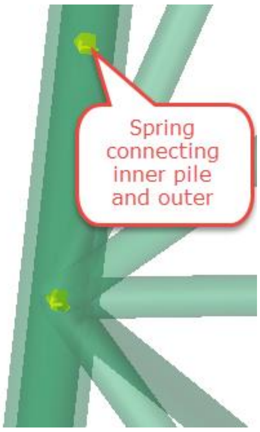
CCBM – code checking beams

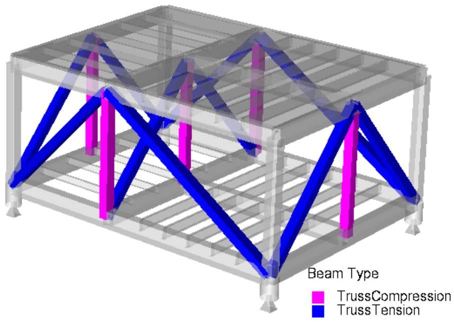
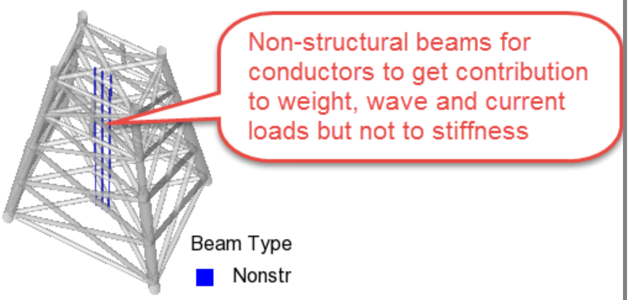
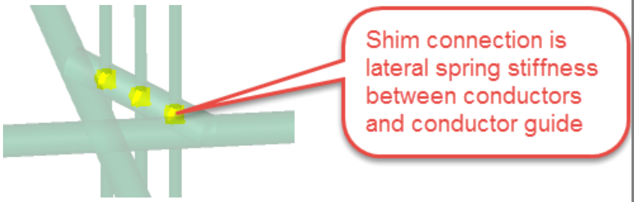
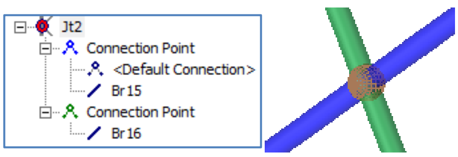
CCPL – code checking stiffened plates

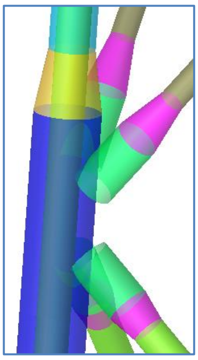
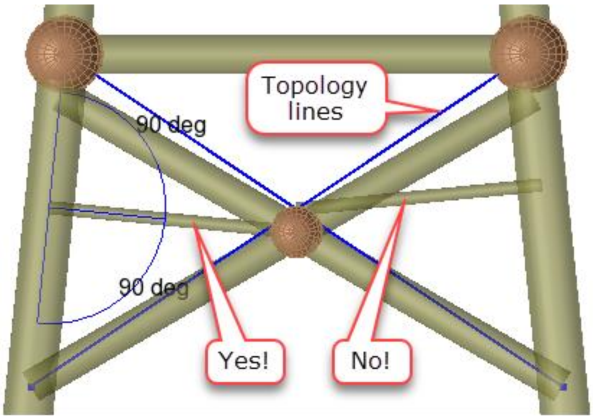
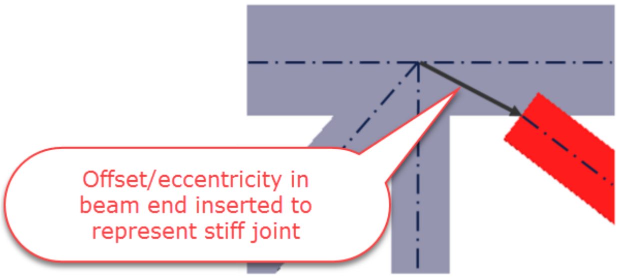
There is also a special version that is the same as GeniE including extensions CGEO, CCBM and CCPL only limited to 10,000 finite elements and 500 beam concepts. Wave loads (Wajac) and pile-soil analysis (Splice) is not included.

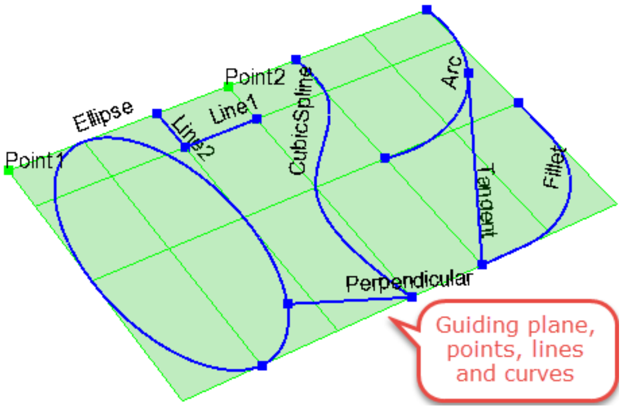
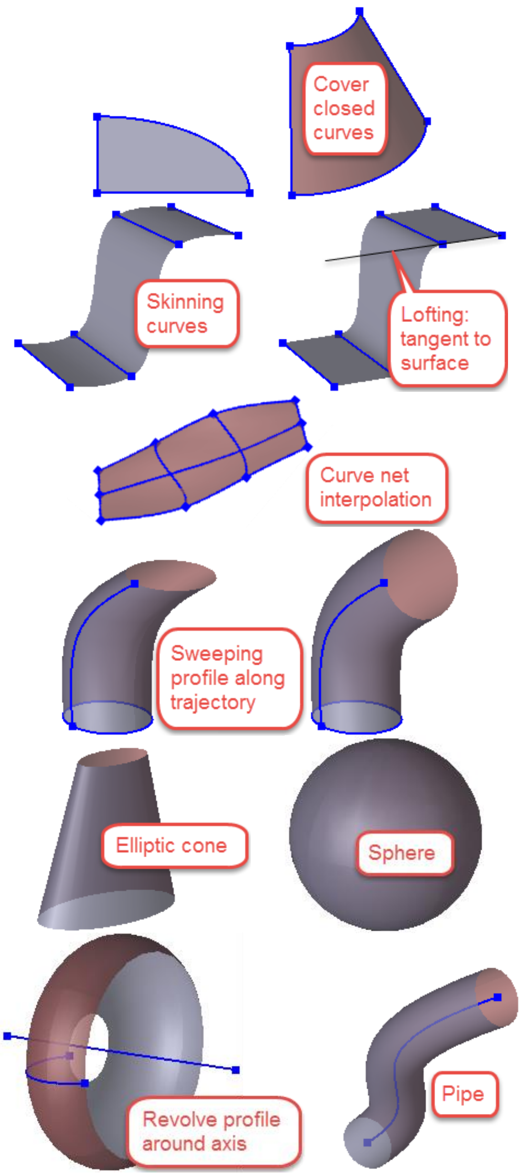
Beam, plate and surface modelling

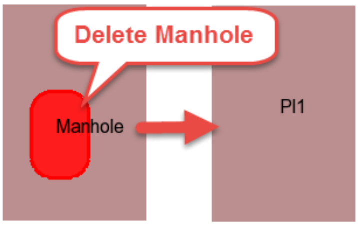
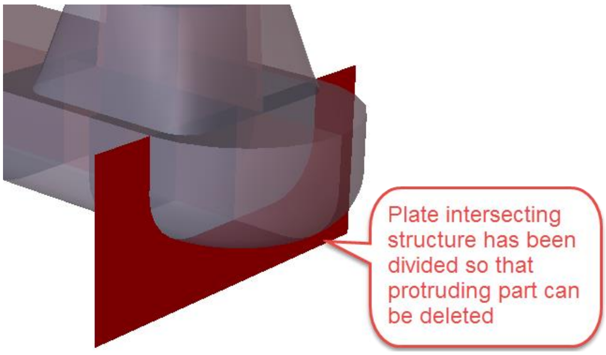
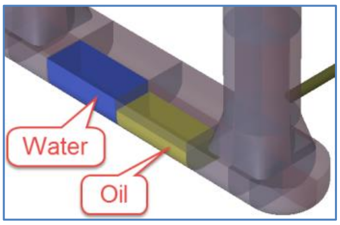
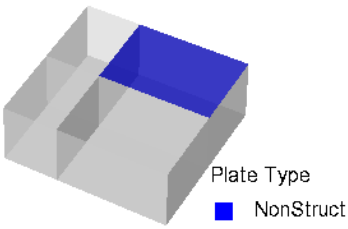
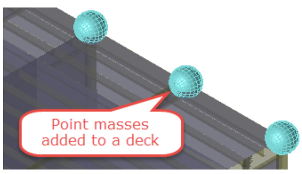
FEATURE	DESCRIPTION
Unit support **m? mm? inch?**	Units may be mixed throughout the modelling. The data logging (scripting) ensures that re-creating the model gives the same result. Unit information is stored on the Sesam Input Interface File (FEM file).
Flat plates and beams  	By default, there is connectivity between beams and plates that geometrically connect at their centre lines/planes. The user may, however, disconnect structural components. Beams connected to plates may be flushed (given offsets/eccentricities) so as to become plate stiffeners. Beams and plates may be created in GeniE or imported from other CAE systems. A flat plate may be changed to a membrane (no bending stiffness).
Thickness  	The thickness is applied to a flat plate (membrane or shell) or a surface (shell).
Beam cross sections (profiles) 	The user may define profiles for pipe, I/H (symmetrical and unsymmetrical), channel, angle, massive bar, box, tubular cone and of type general. Derived properties (area, moment of inertia, section modulus) based on geometry may be modified. In addition, GeniE includes section libraries from AISC, EURONORM, Norwegian Standard and BS in addition to typical ship libraries.

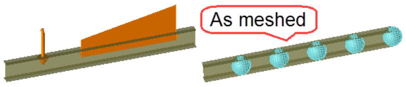
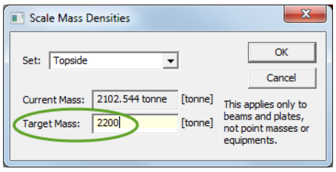
Beam classification 	Beams may optionally be assigned beam classifications primary, secondary, tertiary and auxiliary. This eases keeping the most important structural components in focus. In later versions, this classification will be used to auto setup code checking parameters.
Segmented beams 	A segmented beam is a beam split into multiple parts. Segmented beams are typically used for modelling beams with variable section and/or material. The automatic tubular joint modelling of cans, stubs and cone also involves segmented modelling.
Overlapping beams 	Overlapping beams are typically used to define pile in leg and other cases of double beams. By default, overlapping geometry is prevented so a particular command is used to create overlapping beams.
Grouted members 	Easy definition of outer leg and inner pile to define overlapping beams including the connectivity (fixed, free, stiffness) along the member length. The stiffness of grout between pile and leg may thus be included in a linear analysis. The mass of grout must be added to the overlapping member. Overlapping beams with connectivity may be modelled either in a single operation (grouted beam modelling) or by first modelling a normal beam and thereafter adding overlapping beam, inner beam and mesh properties.

Truss, tension only, compression only elements 	A truss is a straight beam that has stiffness in axial direction only, i.e. no bending stiffness. The truss element can be active in both directions or it can be used to represent a tension-only or compression-only element. Note that use of tension-only and compression-only elements involve non-linear analysis (tension/compression analysis).
Non-structural beams 	A non-structural beam does not include stiffness. It is used to contribute to a model with weight, wave and current loads. Typical uses are conductors, risers and secondary structures not contributing with significant stiffness. Non-structural beams must be connected to the structural model at both ends to avoid nodes with zero stiffness. Alternatively, they can be connected to support rigid links (master-slave), point-point connections or be fixed.
Shim connections 	Between conductors and conductor guides there are typically shim connections. These are lateral connections using linear spring elements. Assigning shim property to conductor beams automatically generates the shim connections. This is available for non-linear as well as for linear analysis.
Point-point connections 	The default behaviour of GeniE is full connection between structural components that overlap geometrically. However, it is possible to specify the connection between overlapping beams (such as pile and outer leg) to be fully connected, disconnected or linear spring connection.
Disconnected structural components 	It is possible to disconnect structural components. Typical examples are disconnected beams in an X joints and stiffener beams disconnected from the plate. It is also possible to disconnect plate edges (e.g. a crack). Such disconnection is available from the script language.

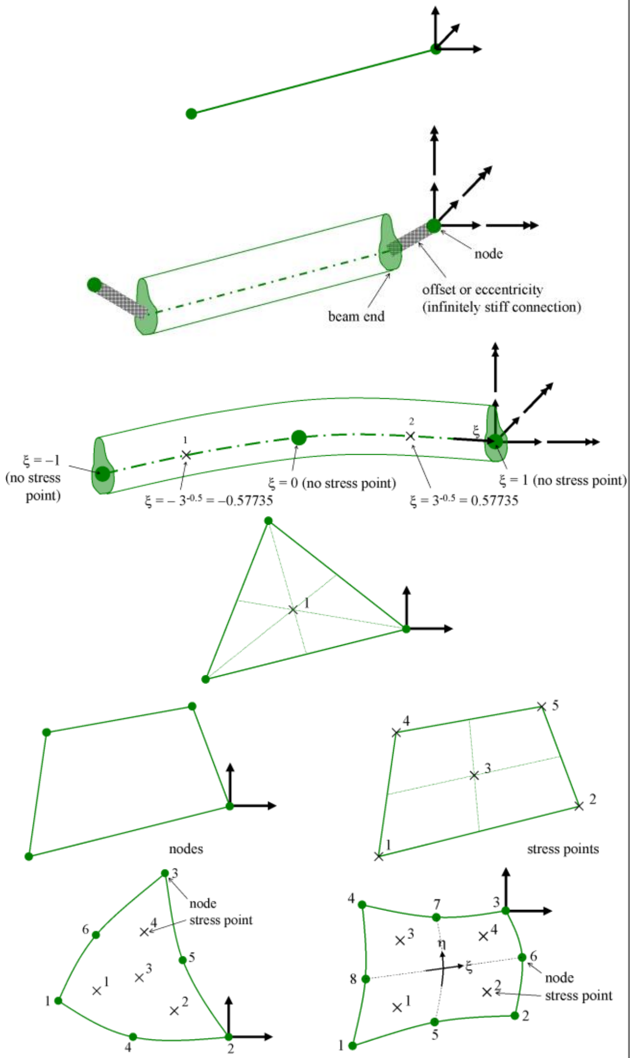
Tubular joints 	Tubular joints may automatically be refined to include gaps (brace offsets ensuring proper plane-wise gaps), cans, stubs and cones according to API and Norsok rules for joint design. The rules may be modified. The braces may be flushed to the chord wall to more accurately represent mass and buoyancy. The braces may be coupled to the chord by spring stiffness connections according to the Buitrago formulae (geometric or load path). It is also possible to use hinges to represent the joint stiffness.
X joints and offsets/eccentricities 	Offsets for an X brace typically cause the X joint to be offset from the topology lines. A brace added to the X joint should connect to the structural joint and not to the intersection of the topology lines. In the example to the left the diagonal braces have large offsets in their upper ends. The braces from the X joint to the legs should remain connected to the structural joint – and also remain perpendicular to the leg if relevant – when the brace offsets are introduced or changed.
General offsets/eccentricities 	The offset/eccentricity feature used for flushing plate stiffeners and ensuring proper gaps in tubular joints is quite general. In addition to the automatic flush and gap features, any offset may be manually specified in the two ends. An example of use is a structural joint with internal stiffeners making it so stiff that the flexible beam should not extend all the way to the FE node in the middle of the joint.

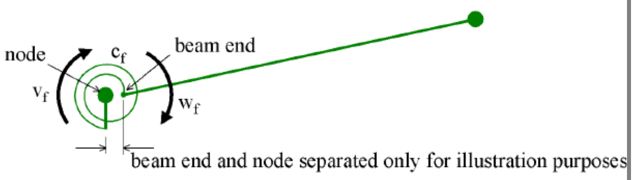
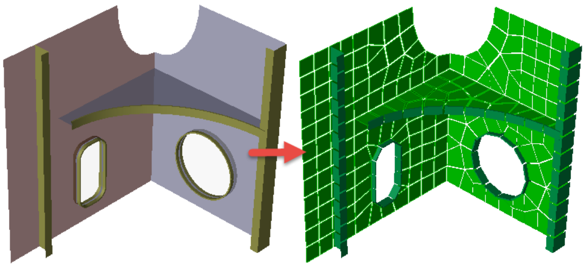
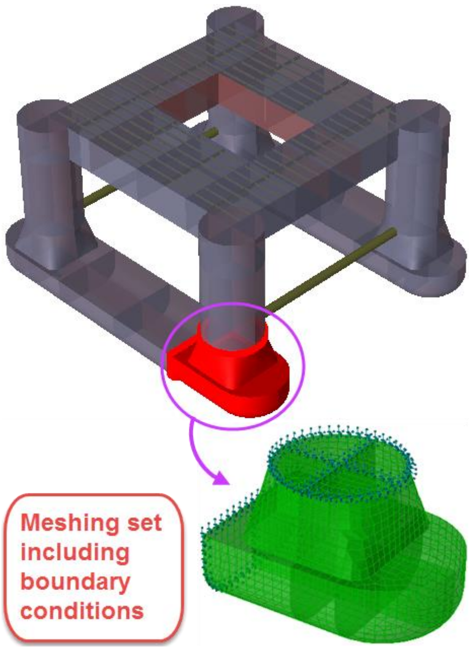
Guiding geometry 	An extensive library of guiding geometry tools helps creating beams, plates and curved surfaces: • Point, point on curve/surface, point closest to selection • Lines, tangents, perpendiculars, polylines • Circular/elliptic arcs, circles, ellipses, fillets • Cubic splines, polycurves, B-splines and polynomials • Quadrilateral, trapezoidal and triangular planes, and much more.
Curved surfaces and beams 	Curved surfaces and beams are created by referring to points and lines. These may be predefined guiding points and lines or data entered when creating the surfaces/beams. Guiding points and lines may be imported from CAD systems. Curved surfaces may be created using several techniques; • Cover set of closed curves, • Skinning and lofting set of curves, • Curve-net interpolation, • Sweeping profile along a trajectory/vector, • Circular/elliptic cylinder/cone, • Sphere, • Revolve profile around axis, • Pipe, • Various shells from net of points, • Automated features for ship hull stiffeners and • Punching holes. Curved beams may be automatically or user defined flushed to a surface (beam stiffeners on shell). Curved surface and beam modelling requires the GeniE extension CGEO.

Hole 	Holes may be defined as objects, i.e. a hole overlapping a plate involves that there is a hole in the plate. This differs from deleting an area in a plate in that deleting the hole patches up the plate. Holes are defined by punching a plate by a closed guide curve or a profile.
General about topology modelling 	When inserting, moving or deleting beams and surfaces the connectivity between structural objects is kept updated. This means that if two objects geometrically coincide they will be connected. Hence there is no need to manually define or delete connections. Beams and surfaces may be split at intersections thereby enabling trimming (deleting protruding parts) to flush parts.
Compartment modelling 	Compartments may automatically be created for volumes enclosed by surfaces. The compartments may be filled with liquid and solid matter in GeniE or from Nauticus Hull. The compartment contents will contribute with loads (e.g. weight) in a structural analysis. Compartment contents may also be used by Sesam HydroD in hydro-static and hydrodynamic analyses.
Non-structural plates 	A non-structural plate has no mass or stiffness. Its sole purpose is to close a volume in case there are openings in the enclosing surfaces. A typical example is an open compartment in a bulk ship.
Point masses 	Point masses may be inserted at given positions along a beam or a plate edge. The point mass may be of type uniform (mass is specified) or generic (properties for all 6 degrees of freedom are specified).

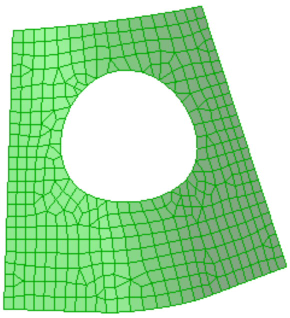
Loads converted to mass  	Various features for converting explicit loads and load combinations including load factors to mass for dynamic analysis.
Mass scaling  	A named set (a group of structural components) may be defined to have a designated mass. GeniE will scale the material density for the set to hit the target mass. That way the centre of gravity is maintained.

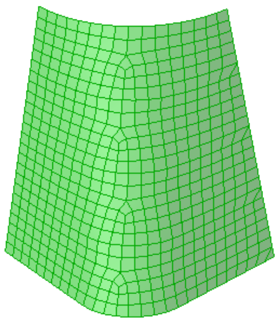
Finite elements and features for meshing

FEATURE	DESCRIPTION
Finite element types  	GeniE can create the following finite element types: • Truss, including tension-only and compression-only • Two node beam • Three node beam (GeniE term: second order) • Three node triangular flat plate • Four node quadrilateral flat plate • Six node triangular curved shell (GeniE term: second order) • Eight node quadrilateral curved shell (GeniE term: second order)

Hinges 	Hinges may be inserted at beam ends to fully or partly release their connection to the node of the joint. Each of the six degrees of freedom may be fixed, released or connected with spring stiffness to the node. Command: Properties > Hinges > New Hinge
Mesh always reflects geometry 	Basic geometry (beams, plates and shells) determines the FE mesh. Where geometry intersects, there will be mesh points (nodes) and lines. There is no need for ensuring mesh connectivity.
Mesh part of structure 	A set containing a part of the model may be meshed as a separate FE model. This could be to create a superelement (for superelement analysis) or to create a sub-model (for sub-modelling analysis). The boundary conditions at the cut planes (super for superelement analysis and prescribed displacements for sub-modelling) are contained in the set. Any number of such part FE models may be created. The example to the left shows a part of a model meshed as a separate FE model with boundary conditions. Command: Right-click meshing activity | Edit Mesh Activity

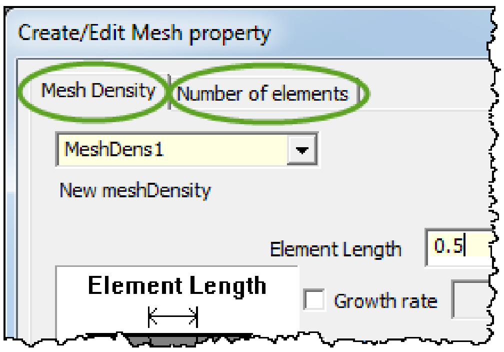
<table>
<tr>
<td>
Meshing algorithms





Advancing front mesher   Quad mesher
</td>
<td>
GeniE supports two different meshing algorithms. The first option is a quad mesh algorithm (the Sesam quad mesher) that will give the best mesh in the middle of a surface – it is thus intended for regular structures like topsides and rectangular parts of a floating structure. The other option is an advancing front mesher (also known as paver meshing) that gives the best mesh along edges – this is best for details and irregular structural parts such as a joint, a hole or the fore or aft part of a vessel. It is possible to use both options in a model.

Command: Edit | Rules | Meshing
</td>
</tr>
<tr>
<td>
Mesh density

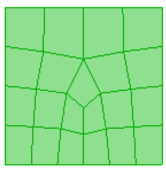


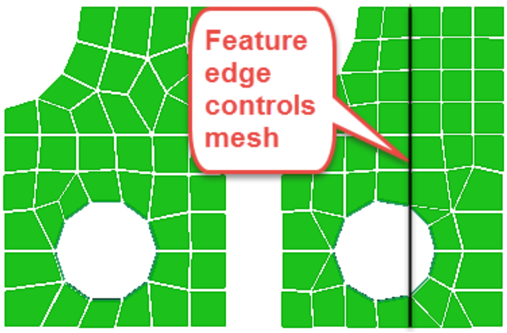
</td>
<td>
The density or size of elements is determined by mesh properties:

- Mesh density specifying length of element edge
- Number of elements along a line or plate edge

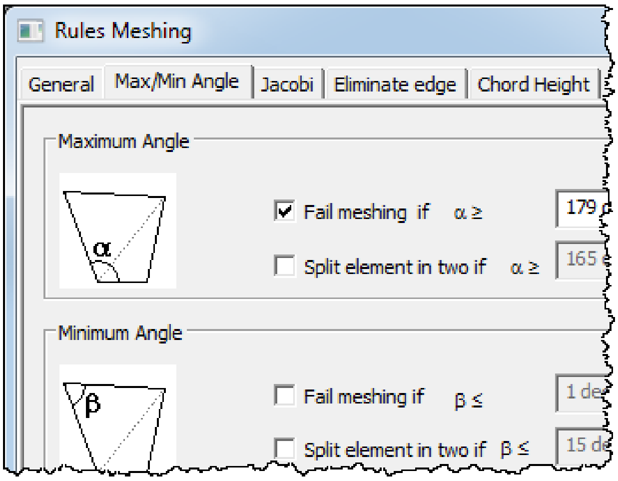
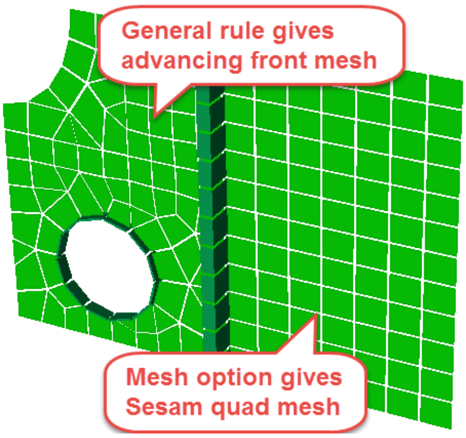
Any number of such properties are defined and assigned to various parts of the model. One of the mesh properties may also be set as default, i.e. valid where no particular property has been assigned.

Command: Edit | Properties | Mesh Property
</td>
</tr>
<tr>
<td>
Mesh transition


</td>
<td>
With different mesh densities for various parts there will be a mesh transition zone from fine to coarse mesh. By default, the extent of the transition zone will be as short as possible. The user may extend the transition zone by specifying a growth rate for a mesh density.
</td>
</tr>
</table>

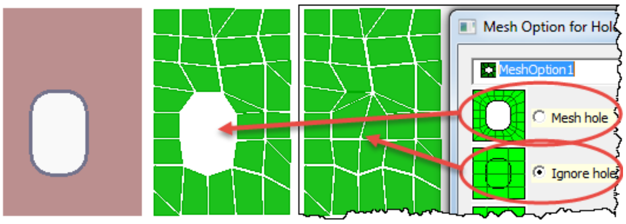
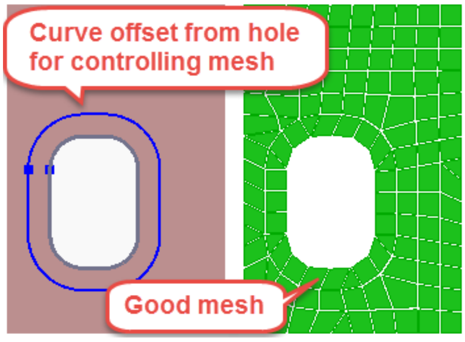
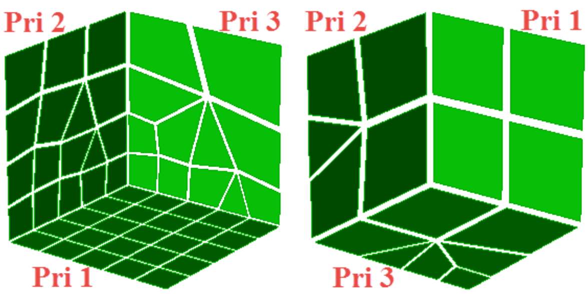
<table>
<tr>
<td>Feature edges for mesh control
</td>
<td>So-called feature edges may be inserted to control the mesh. Where there is a feature edge crossing a plate, there will be a mesh line in same way as for a beam stiffener.

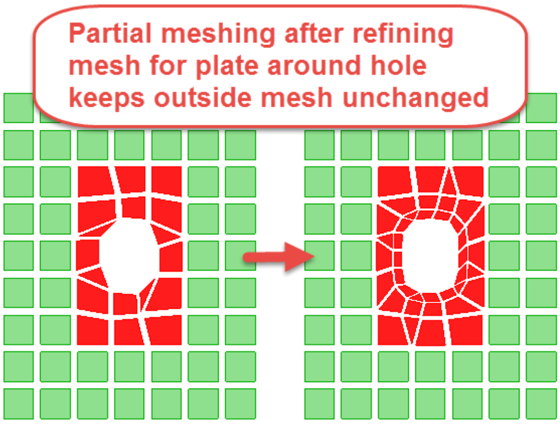
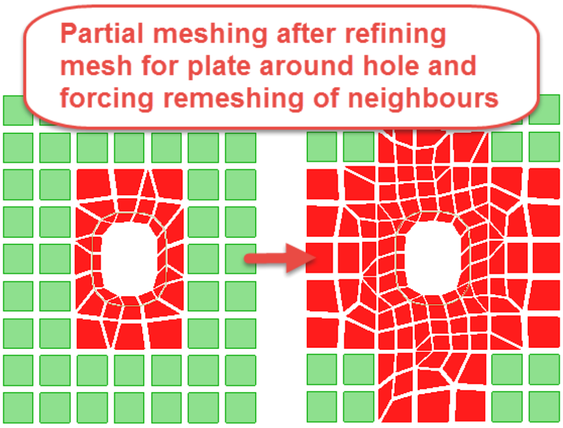
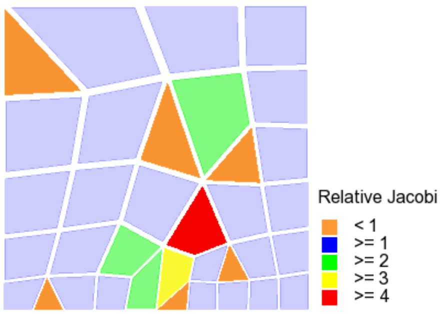
The example to the left shows how introducing a feature edge alters the mesh: there is a mesh line along the feature edge.

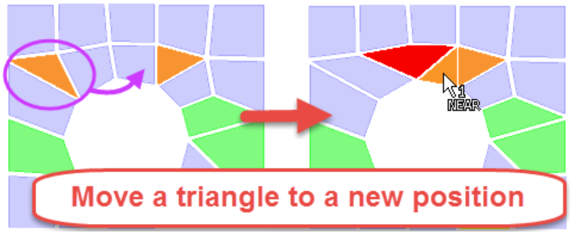
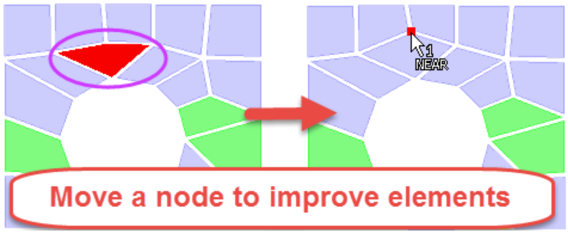
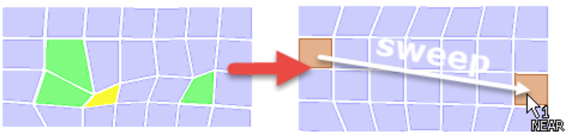
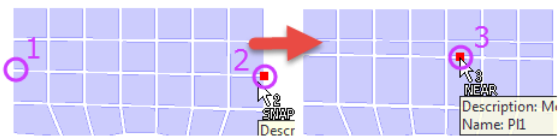
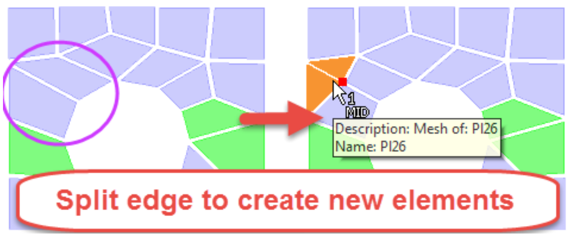
Command: Structure | Features and Holes | Feature Edge</td>
</tr>
<tr>
<td>Meshing rules
</td>
<td>The mesh creation is subject to several user defined rules:
Various idealisations of topology to improve mesh Preferences re. regular (mxn) mesh, triangular elements and more Max/min angle of element corner Max relative and minimum Jacobian determinant Elimination of short edges to avoid degenerated mesh Etc. Command: Edit | Rules | Meshing</td>
</tr>
<tr>
<td>Overrule general rules and settings
</td>
<td>Meshing rules and meshing algorithm (Sesam quad or advancing front) may be specified for individual parts thereby overruling general (global) rules and settings.

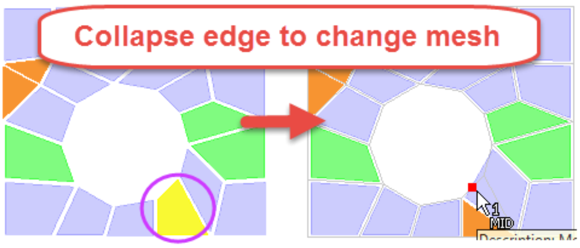
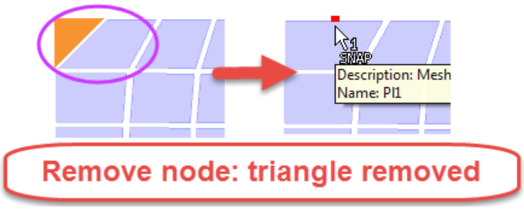
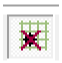
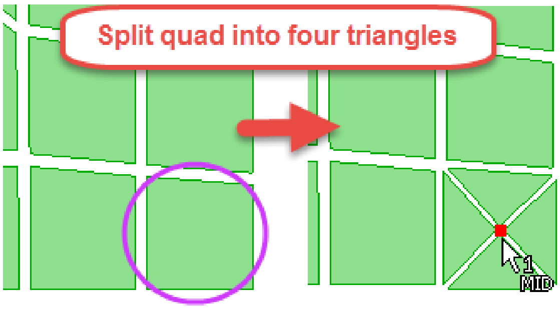
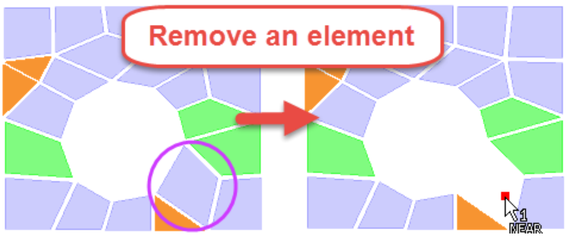
For the example to the left advancing front mesh is generally used to get a proper mesh for holes and cavities. But the Sesam quad mesher is assigned to the right part as this is a better choice for rectangular parts.

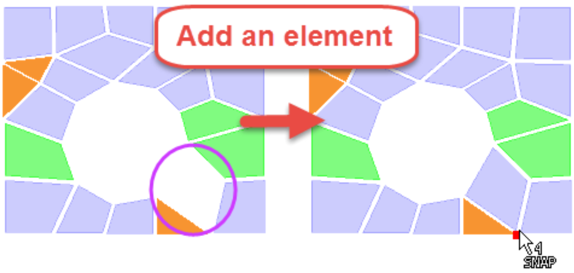
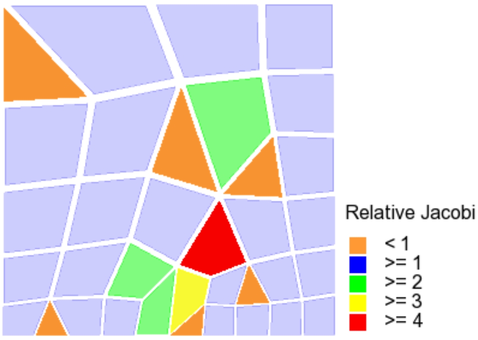
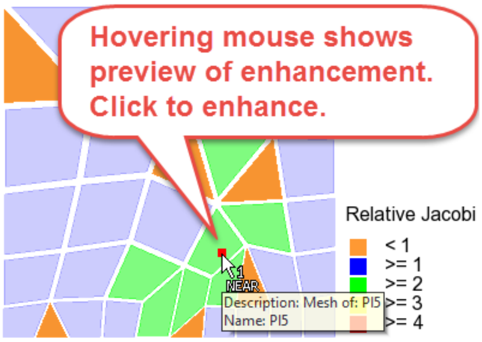
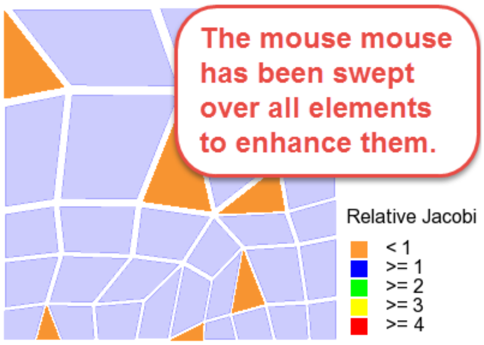
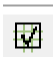
Command: Edit | Properties | Mesh Option</td>
</tr>
</table>

Option for ignoring holes 	Holes may optionally be ignored in the mesh creation. Command: Edit | Properties | Mesh Option
Control mesh around holes/cavities 	There is a feature for easy offsetting guide curves around holes, cavities and similar details by constant values. This may be used to improve the mesh in such critical areas. Command: Right-click model curve at hole edge | Offset
Prioritized meshing 	To achieve an optimal mesh for important parts the meshing can be prioritized. I.e. important parts are meshed first (priority 1) and less important parts are meshed later. Any number of priority levels may be specified, each priority level is assigned plates/shells. The priority levels may easily be reordered. The example to the left shows three perpendicular planes with different mesh densities. The plate with the highest priority gets a regular mxn mesh. Command: Utilities | Mesh Priorities | New Mesh Priority

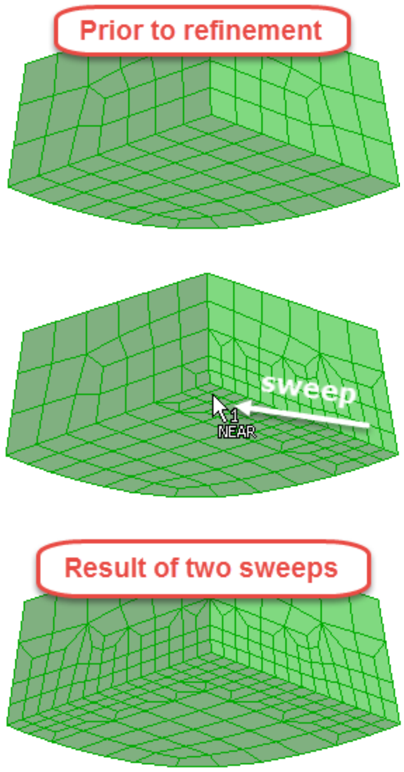
Partial meshing 	Partial meshing allows the user to alter or refine the mesh for a part of the structure while using the surrounding mesh as a constraint. I.e. the surrounding mesh including its node and element numbering is kept unchanged while remeshing the part. The advantages of partial meshing are speed-up (only modified parts are remeshed), preserve mesh outside part and better user control. Partial meshing is set by default. Command: Right-click analysis activity | Edit Analysis | Use Partial Meshing
Force remeshing 	In partial meshing concepts (beams and plates/shells) are identified as in need of remeshing or not depending on the model change done. You may, however, force remeshing of parts marked as *not* in need of remeshing. Also, you may force remeshing of the neighbours of the parts marked as in need of remeshing. The example to the left shows the result of forcing the neighbours of the middle plate to be remeshed. Command: Right-click concept | Remeshing | Force remesh (including neighbours)
Colour coding relative Jacobi 	The relative Jacobian determinant (measure on element quality where 1 is best for a quadrilateral element and higher value is inferior) may be colour coded. The example to the left shows a mesh with an inferior (red) element. Command: Right-click mesh | ColorCode | Mesh | Relative Jacobi

Mesh editing: Manipulate triangle  	The manipulate triangle feature has two options: • Moving a triangular element to a new position where it possibly may be merged with another triangular element into a quadrilateral element • Splitting a quadrilateral element into two triangular elements Button:
Mesh editing: Move node  	Drag any node to a new position and see while dragging the colour coding of maximum relative Jacobian determinant changes. Release node at the optimal position. The example to the left shows how the inferior (red) element is improved by moving a node. Button:
Mesh editing: Align nodes in element grid  	An irregular mxn element grid may be aligned by dragging the mouse diagonally over the area. Button:
Mesh editing: Align nodes along line and move by constant vector  	Aligned nodes may be shifted sideways by a given value. First select the aligned nodes by two clicks and then move sideways by a third. Button:
Mesh editing: Split edge  	Split an element edge by creating a new node there and new elements. Button:

Mesh editing: Collapse an edge 	The edge of an element may be collapsed to a point. In the example to the left the yellow element edge is collapsed. A quadrilateral element may also be collapsed to an edge by collapsing its diagonal. Button: 
Mesh editing: Remove node 	Only in certain cases a node may be removed. These are cases when elements may be merged as a result of the removal. To the left is an example where a triangular element is merged with a quadrilateral element. Button: 
Mesh editing: Imprint node 	A new node may be inserted in two cases: • Inside a triangular element thereby splitting it into three new triangular elements • Inside a quadrilateral element thereby splitting it into four new triangular elements Button: 
Mesh editing: Remove element 	Click an element to remove it leaving a hole in the mesh. Button: 

Mesh editing: Add plate element 	Add an element by clicking its corners. Button: 
Mesh editing: Enhance quality   	The mesh resulting from the automatic generation may be enhanced by merely clicking single elements or sweeping mouse over several elements. The example to the left shows how clicking the inferior element of the example above enhances it. The example to the left shows the result of sweeping the mouse over all elements. Button: 

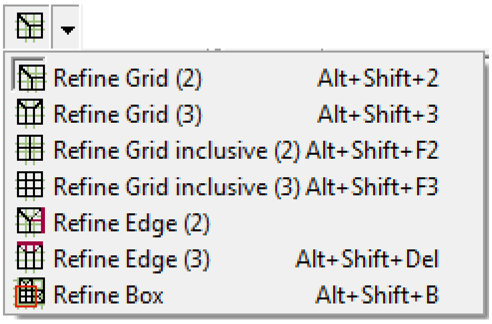
<table>
<tr>
<td>
Mesh editing: Refine grid/edge/box



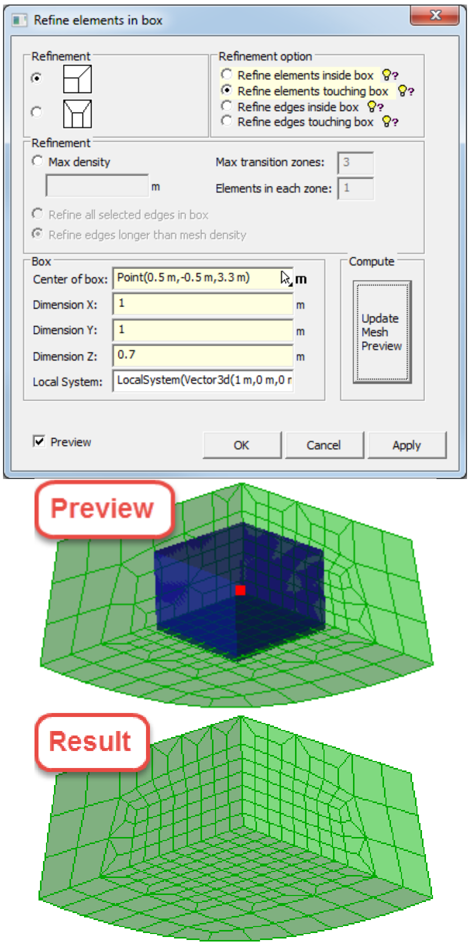
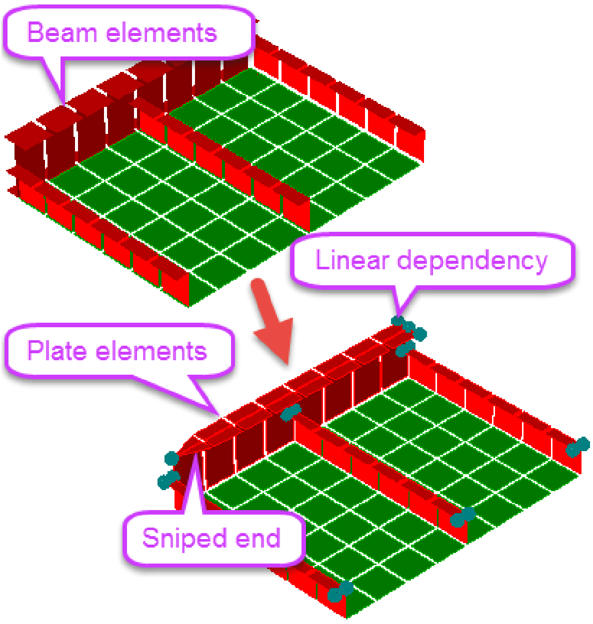
</td>
<td>
Grids (mxn) of quadrilateral elements may be refined by sweeping operations. There are alternative refinements; refine each element to 2 or 3 elements, include all selected elements in the refinement (transition zone outside selection) and refine a grid, an edge or all inside a box defined by coordinates.

To the left is an example of refining grids for three intersecting planes. The 'Refine Grid inclusive (2)' option is used. Two sweeps refine the grid in the intersection zones as shown.

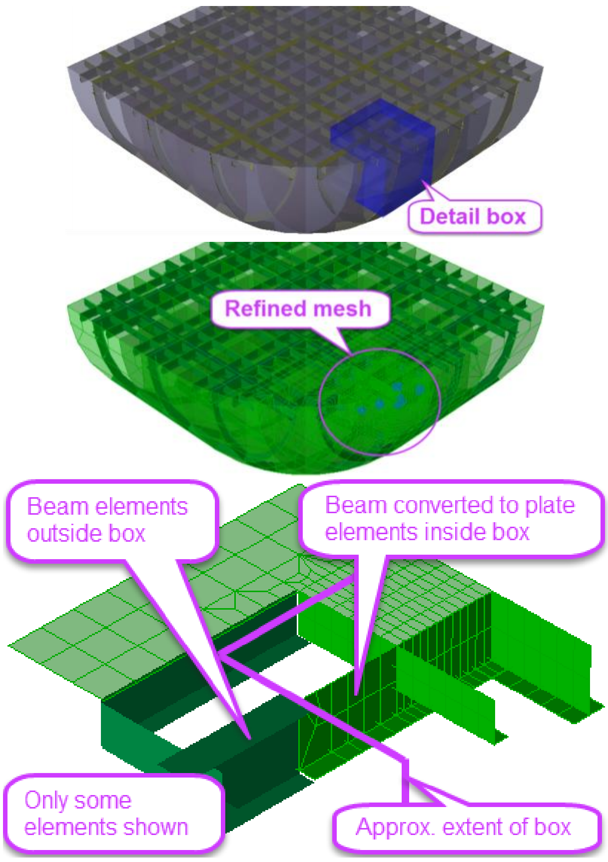
An example of Refine box is found below.

Buttons: 

</td>
</tr>
</table>

Mesh editing: Refine box  	Here is an example of using the refine box feature. All elements inside the box are refined according to the specifications in the dialog. The centre of the box is inserted by clicking in the model and manual adjustment of coordinates if required. In the preview a blue box shows the extent of the refinement.
Converting beams to plates/shells  	Beam elements with sections bar, I or L may be converted into plate elements. At the beam ends the nodes of the plate elements are linearly coupled to the node at the plate centre-plane to ensure deformation according to beam theory. The ends of the beams may optionally be sniped (tapered). This conversion from beam elements to plate elements will *not* alter the concept model. I.e. the beams will still be beam concepts. Note that the GeniE Snack Pack also provides features for converting beams to plates. These are in the form of JS-scripts that convert beam concepts into plate concepts. Command: Right-click beam elements | Convert to Plate Elements

Detail box for refined meshing



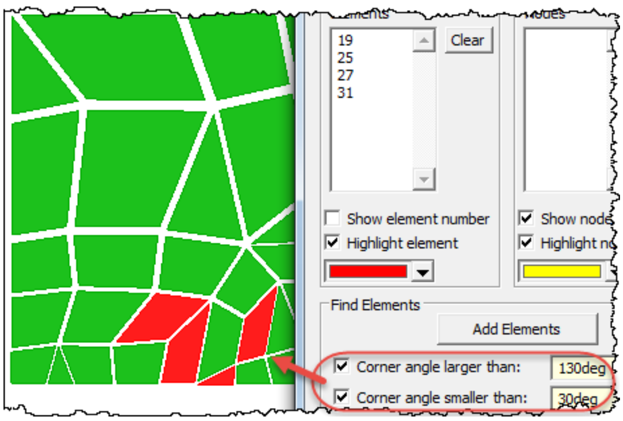
A box may be defined enclosing a part of a model with the purpose of creating a refined mesh inside the box. The box is defined by its centre and extent in X, Y and Z (optionally in a local coordinate system). The box definition includes a mesh density, how to handle holes and whether beams shall be converted into plate/shell elements.

The example to the left shows how a part of the rounded box model has a refined mesh.

Some of the elements are shown in the bottom figure. Beams inside the detail box are converted to plate elements, beams outside not.

Command: Right-click Structure > Details folder | New Detail Box

Identify inferior elements

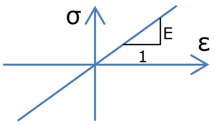
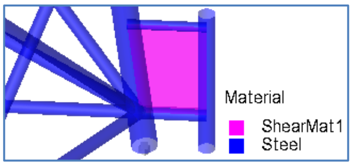


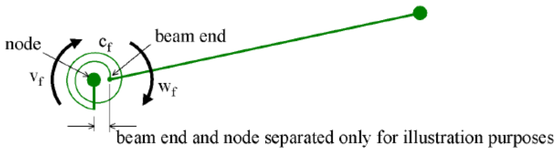
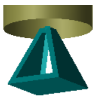
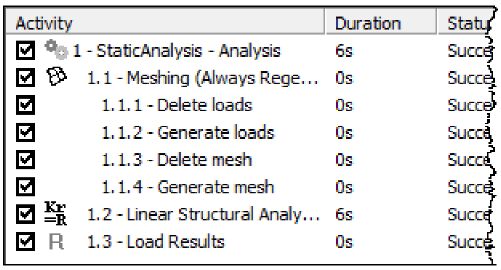
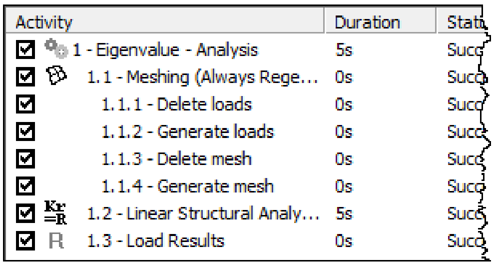
Inferior elements may be identified based on several criteria:

- Max/min angle of element corner
- Max relative and minimum Jacobian determinant
- Maximum aspect ratio
- Warping (twisting)
- Etc.

Command: Tools | Analysis | Locate FE

Modelling for structural analysis in Sestra

<table>
<tr><th>FEATURE</th><th>DESCRIPTION</th></tr>
<tr><td>Materials
</td><td>GeniE supports linear isotropic material, isotropic shear material and orthotropic material (material axes are aligned with the global axis system).

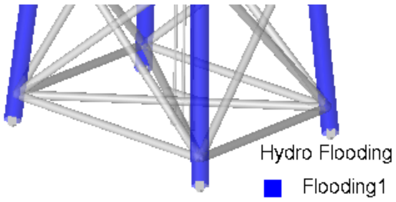
Plates with shear material are typically used for connecting pile sleeve and leg.</td></tr>
<tr><td>Hinges
</td><td>Hinges may be inserted at beam ends to fully or partly release their connection to the node of the joint. Each of the six degrees of freedom may be fixed, released or connected with spring stiffness to the node.</td></tr>
<tr><td>Boundary conditions
</td><td>Boundary conditions (or supports) may be of type fixed, free, spring-to-ground or spring matrix. Furthermore, it is also possible to specify prescribed displacements for each load case.</td></tr>
<tr><td>Corrosion addition
</td><td>Corrosion additions are applied to a plate, surface or beam to reduce the thickness to be used in the structural analysis. The corrosion addition is specific to an analysis which means that it is possible to run several analyses with alternative corrosion additions.</td></tr>
<tr><td>Analysis types

<table>
<tr><th>Activity</th><th>Duration</th><th>Statu</th></tr>
<tr><td>1 - StaticAnalysis - Analysis</td><td>6s</td><td>Succe</td></tr>
<tr><td>1.1 - Meshing (Always Rege...</td><td>0s</td><td>Succe</td></tr>
<tr><td>1.1.1 - Delete loads</td><td>0s</td><td>Succe</td></tr>
<tr><td>1.1.2 - Generate loads</td><td>0s</td><td>Succe</td></tr>
<tr><td>1.1.3 - Delete mesh</td><td>0s</td><td>Succe</td></tr>
<tr><td>1.1.4 - Generate mesh</td><td>0s</td><td>Succe</td></tr>
<tr><td>1.2 - Linear Structural Analy...</td><td>6s</td><td>Succe</td></tr>
<tr><td>1.3 - Load Results</td><td>0s</td><td>Succe</td></tr>
</table>

<table>
<tr><th>Activity</th><th>Duration</th><th>Statu</th></tr>
<tr><td>1 - Eigenvalue - Analysis</td><td>5s</td><td>Succ</td></tr>
<tr><td>1.1 - Meshing (Always Rege...</td><td>0s</td><td>Succ</td></tr>
<tr><td>1.1.1 - Delete loads</td><td>0s</td><td>Succ</td></tr>
<tr><td>1.1.2 - Generate loads</td><td>0s</td><td>Succ</td></tr>
<tr><td>1.1.3 - Delete mesh</td><td>0s</td><td>Succ</td></tr>
<tr><td>1.1.4 - Generate mesh</td><td>0s</td><td>Succ</td></tr>
<tr><td>1.2 - Linear Structural Analy...</td><td>5s</td><td>Succ</td></tr>
<tr><td>1.3 - Load Results</td><td>0s</td><td>Succ</td></tr>
</table></td><td>There are built-in analyses running analysis programs in the background:

Linear static structural analysis (in Sestra)
Eigenvalue analysis (in Sestra)
Time/frequency domain dynamic analysis, direct or Modal Superposition (in Sestra)
Non-linear tension-compression analysis (in Sestra)
Wave load calculation (in Wajac)
Wave load plus integrated structure-pile-soil analysis (Wajac, Sestra, Splice)

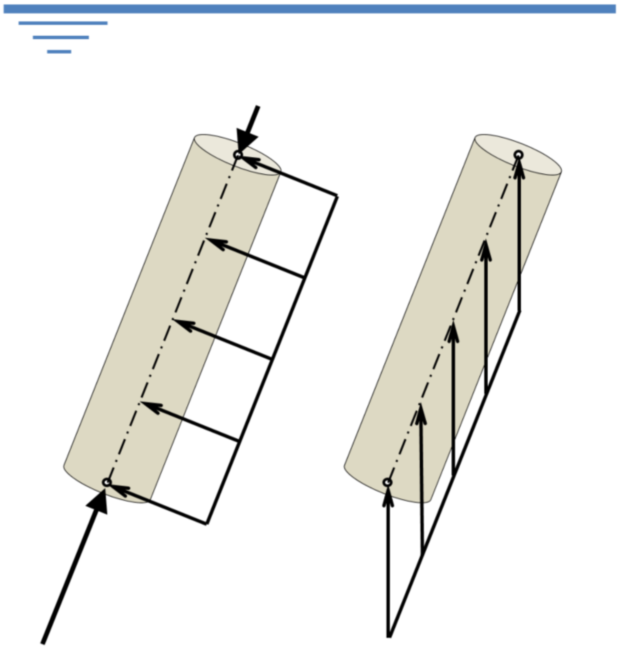
Sesam Manager is used to run non-linear analysis (in Usfos) where the model is created in GeniE.</td></tr>
</table>

Modelling for wave and wind analysis in Wajac

<table>
<tr><th>FEATURE</th><th>DESCRIPTION</th></tr>
<tr><td>Flooding



</td><td>Flooding is used to specify which members are filled with water. Members are by default non-flooded. Flooding involves that only the steel volume contributes to buoyancy.</td></tr>
<tr><td>Buoyancy



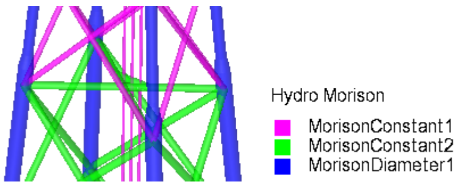
Rational method Marine method
</td><td>Buoyancy may be combined with the wave forces or singled out as a separate case. The buoyancy accounts for the steel sectional area plus entrapped air (unless flooded). The buoyancy may be switched off for selected members.

A buoyancy area may optionally be defined to override the area computed by Wajac for tubular or non-tubular sections for use in the buoyancy calculation.

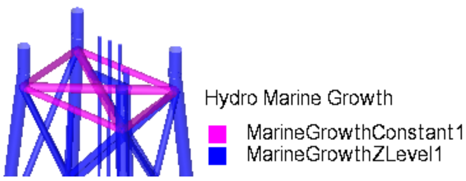
The buoyancy is calculated up to the wave crest/trough in deterministic analysis.

The buoyancy may be computed as:

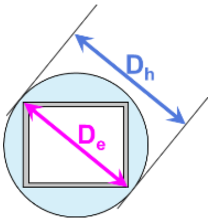
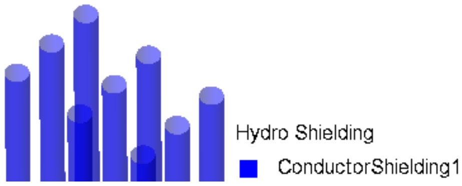
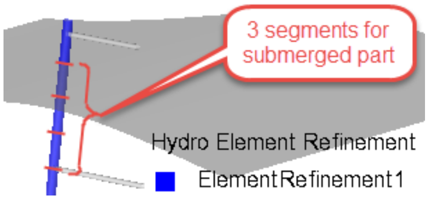
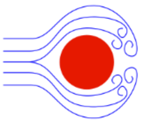
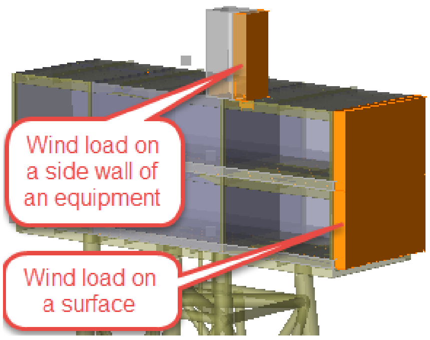
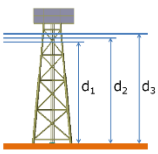
- Line load perpendicular to the member plus concentrated forces in the ends (rational method)
- Vertical line load (marine method)
</td></tr>
<tr><td>Morison coefficients for wave loads

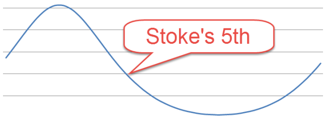
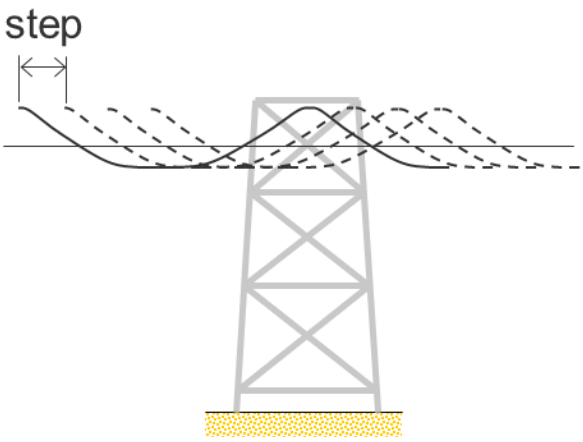
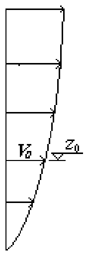
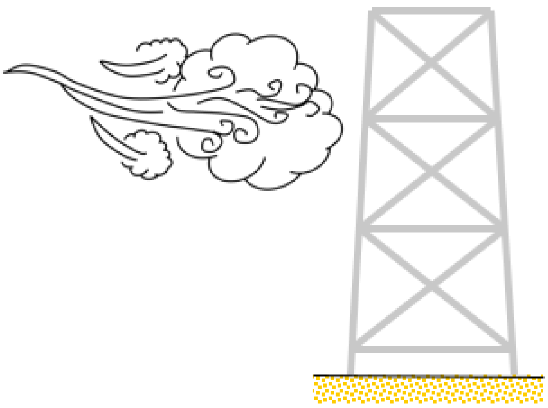


</td><td>The Morison coefficients (C_m and C_d) may be defined in several ways: constant value, function of diameter, function of Roughness/Reynolds number, function of Roughness/KC number, by rule (API RP 2A-WSD 21st edition) and directionally dependent.</td></tr>
<tr><td>Marine growth

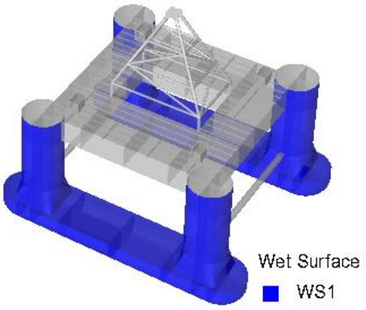


</td><td>By adding marine growth, the hydrodynamic diameter is increased. In addition, the mass and added mass of the marine growth is included in the analysis. The weight of the marine growth may optionally be included in the hydrostatic buoyancy calculation.</td></tr>
</table>

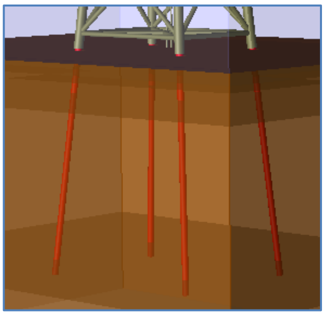
Hydrodynamic diameter 	The hydrodynamic diameter is used to manually override the diameter computed by Wajac for tubular or non-tubular sections for use in wave load calculation. An application may be to substitute the equivalent diameter, the diagonal, for a box section with a more suitable value.
Conductor shielding 	Reduce drag and inertia coefficients for conductor arrays due to shielding effects (conductor shielding factor) according to API (API RP 2A-WSD 21st edition).
Element refinement 	A member is by default divided into two segments for calculation of the wave loads. Only the submerged part is considered. Using the element refinement property, a member may be divided into up to 20 segments for more precise wave load analysis.
Air drag 	The Morison coefficient C_d for wind load calculation may be defined as a constant or as a function of Reynolds number. Wind shielding is achieved by setting $C_d = 0$.
Wind load area 	Wind load area is an area defined by the user for computation of wind loads. A wind load area can be a surface, a dummy wall connected to members, or a side wall of an equipment.
Water depth 	The water depth is used to define the location of the sea surface. It is possible to include multiple sea surface elevations in one analysis.

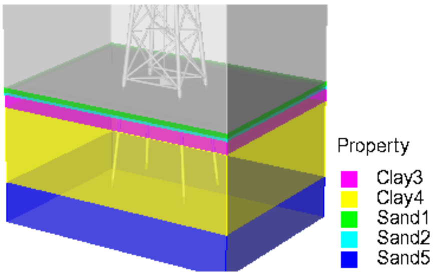
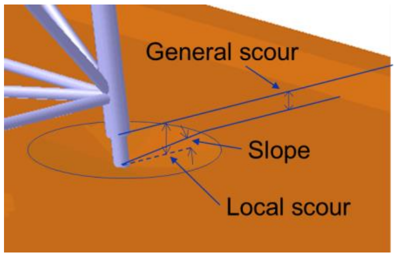
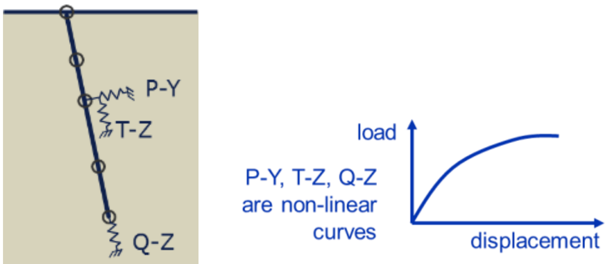
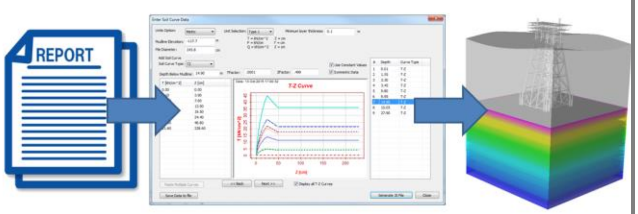
Wave theory 	The selection of wave theory (Airy, Stokes 5th order, Cnoidal and Stream Function) to be used in Wajac is done in GeniE. For more information see Wajac.
Current	The current may be defined to act in the same direction as the wave or in any specified direction.
Wave load analysis 	Deterministic wave load analysis (stepping a wave) may be performed by using GeniE's built-in analysis activity. All model and execution data are generated in GeniE. Wajac computes and stores for subsequent structural analysis the wave plus current loads for all wave steps or for only steps of maximum/minimum base shear and/or overturning moment. Short term sea state simulation (time domain) and frequency domain wave load analyses may be executed by using model data given in GeniE plus manually edited additional Wajac input data. Such Wajac analyses may also be part of a workflow established in Sesam Manager.
Wind profile 	Several wind profiles may be defined. There are two user defined profiles (normal and general) and two profiles according to API, the 1st edition (termed Extreme) and 21st edition (termed Extreme API21).
Wind load analysis 	A wind load analysis may be performed separately or in combination with a wave load analysis. As for a wave load analysis, the loads are stored for a subsequent structural analysis. All model and execution data are generated in GeniE. Wind loads are a combination of member (beam) loads and area loads. The member wind load calculation is done in Wajac while wind loads on areas, surfaces and equipments are calculated in GeniE.

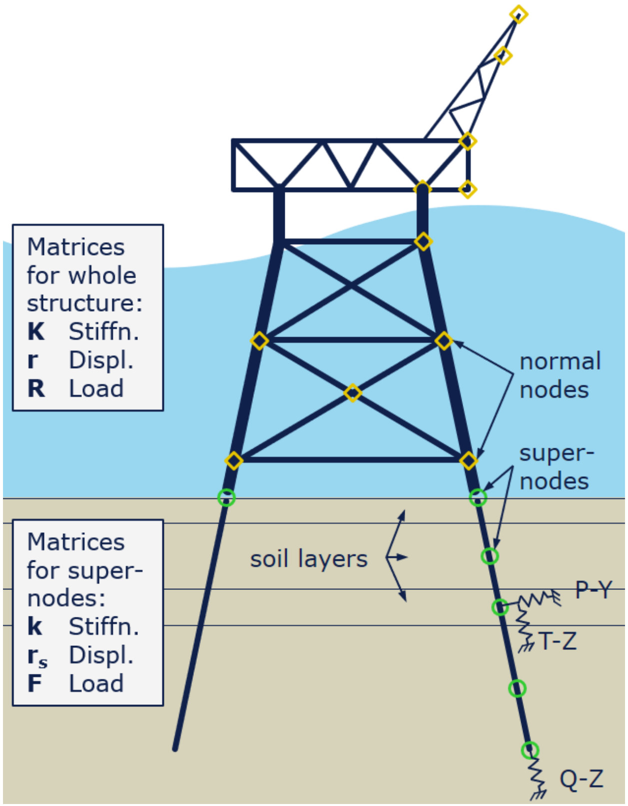
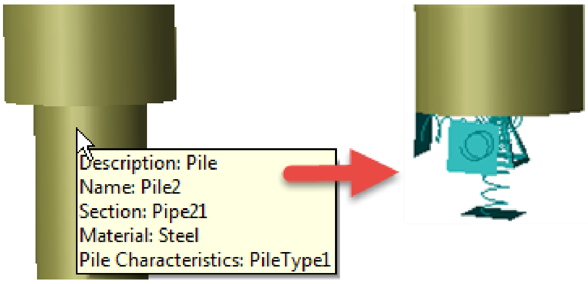
Modelling for wave and motion analysis in HydroD/Wadam

FEATURE	DESCRIPTION
Wet surface 	Wet surfaces are used to identify which side of which surfaces are exposed to water. This information is used by HydroD and Wadam for calculation of motion and hydrodynamic pressures.

Modelling for pile-soil analysis in Splice

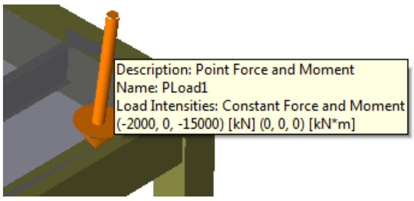
FEATURE	DESCRIPTION
Piles 	A pile is modelled as a straight beam. In most cases, it is represented as a segmented beam as there are changes in the thickness along the pile length. There is no need to manually specify the intersections with the soil layer as this is cared for by GeniE.
Pile characteristics	The pile characteristics is a separate type of property allowing section stiffness values to be overruled (e.g. overruling un-grouted pipe values due to grouting). Moreover, special pile tip boundary conditions may be specified. It is also possible to add the density of fluid/soil inside the pile.

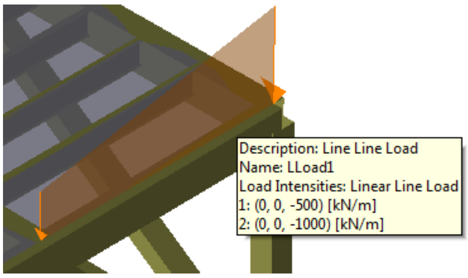
Soil type sand and clay 	The sand property is defined by specifying angle of internal friction, mass density and other geotechnical parameters. The clay property is defined by specifying undrained shear strengths, mass density and other geotechnical parameters.
Scour 	Scour is specified as consisting of two components: • General scour (i.e. for the whole sea bottom around the structure) • Local scour around the piles (depth and slope)
Soil data	Soil data is a property with additional soil characteristics like the initial value of soil shear modulus, soil Poisson’s ratio and details for skin friction and tip resistance.
Soil curves 	Soil curves are properties that control the generation of the: • P-Y (lateral stiffness) • T-Z (skin friction stiffness) • Q-Z (tip stiffness) They can be generated based on a set of pre-defined curves or by manual input of data.
Soil utility tool 	Easy conversion of soil data from a design premise report into analysis data format

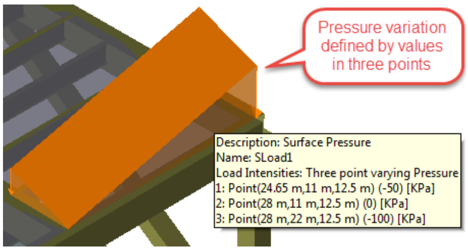
Non-linear pile-soil analysis 	All data required for a structure-pile-soil interaction analysis is generated in GeniE and the analysis is executed automatically from GeniE by running Splice in the background. Splice in turn runs Sestra in the background (API as a service). The analysis procedure is as follows: 1. Sestra reduces the stiffness and loads (including wave, current and wind loads) of the jacket (or other structure with piled foundation) by elimination of all nodes except the structure-pile connection nodes at the sea bed. 2. By a non-linear pile-soil interaction analysis Splice computes the displacements along the piles. 3. Based on the displacements of the structure-pile connection nodes (the pile heads) the displacements and forces of the jacket are computed by a back-substitution (retracking) process.
Linearised pilehead spring 	The feature of Splice for computing and storing a linearised spring stiffness matrix at the pileheads may be controlled from GeniE. Moreover, the linearised pilehead stiffness matrix computed may be imported into GeniE rather than being manually entered as a spring support matrix.

Explicit (point, line, surface) load modelling

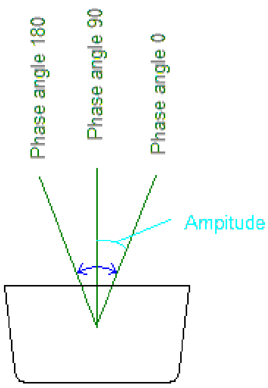
<table>
<tr><th>FEATURE</th><th>DESCRIPTION</th></tr>
<tr><td>Point loads


</td><td>A force or a moment applied at a given position. Must be connected to a beam or a plate edge.</td></tr>
<tr><td>Line loads

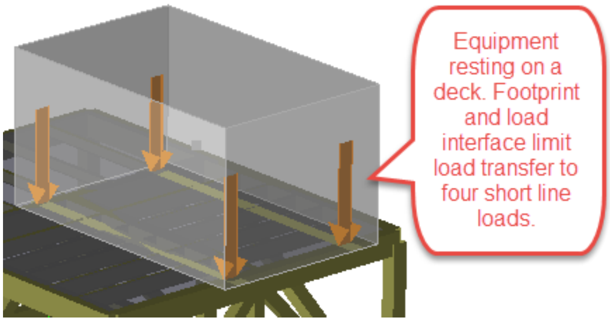
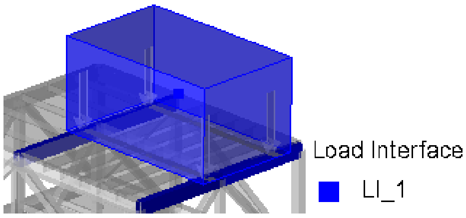
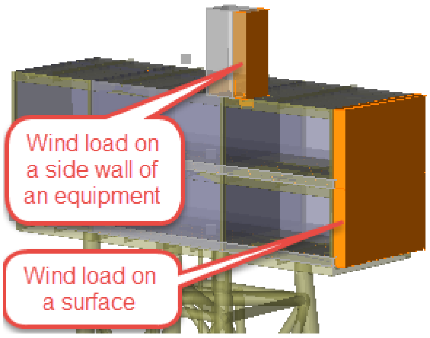

</td><td>A line load (constant or linearly varying) applied along the whole or part of a beam. May be applied to segmented beams and individual parts of overlapping beams.</td></tr>
<tr><td>Surface loads


</td><td>A surface load may be specified as X, Y and Z component pressure, normal pressure and traction. The variation of these may be constant, linearly varying and any variation as given by a JavaScript function.

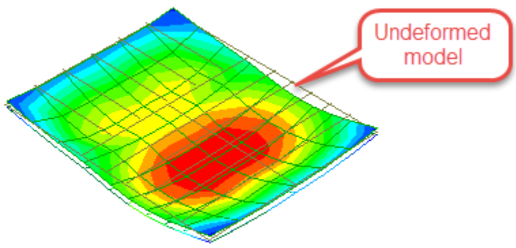
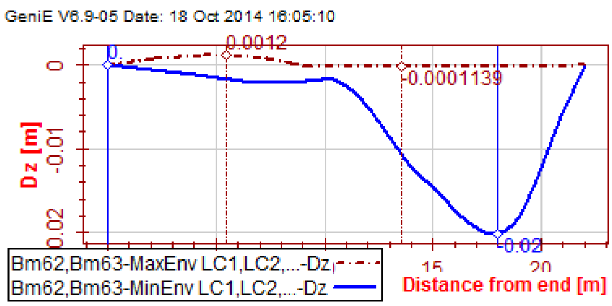
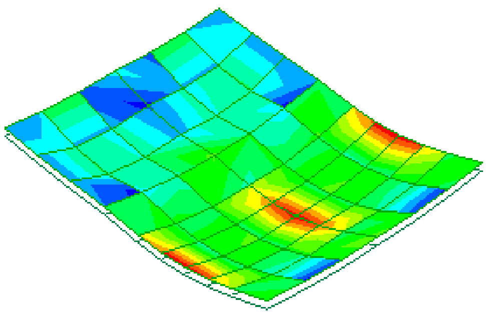
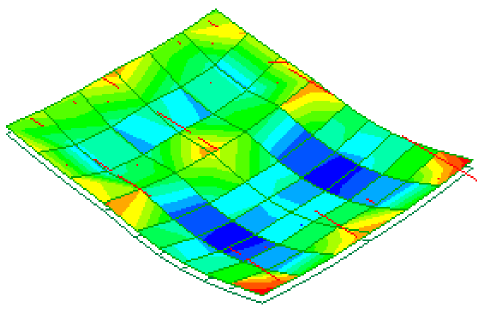
The load may be applied to any part of any surface.</td></tr>
<tr><td>Temperature loads</td><td>Temperature intensities may be applied along a beam – constant or linearly varying.</td></tr>
<tr><td>Acceleration loads

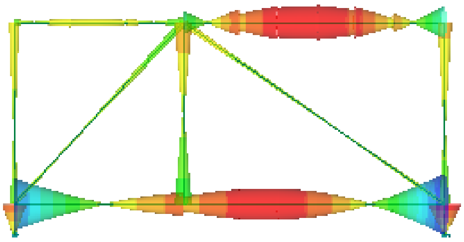
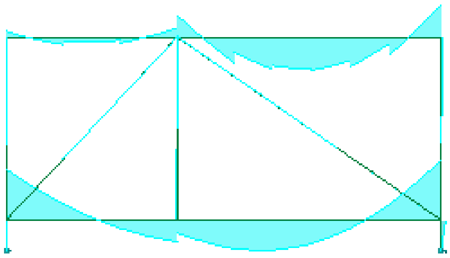
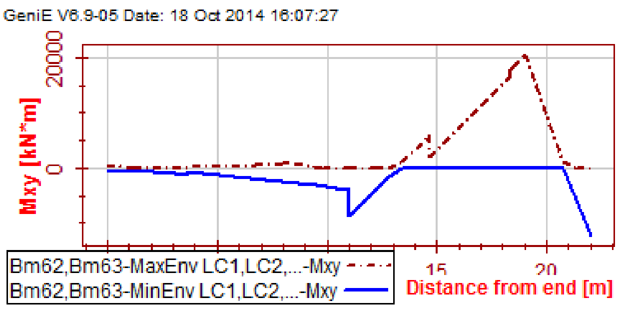

</td><td>Accelerations are used to generate inertia forces based on structural mass and user defined masses. Constant acceleration fields in X, Y or Z directions may be defined.

In addition, rotational acceleration fields may be defined. The angular velocity and acceleration of the rotational field may be given directly or in terms of a harmonic (wave induced) motion giving angular motion and period. The latter method is used for analysis of structures resting on a barge or ship with known motion.</td></tr>
</table>

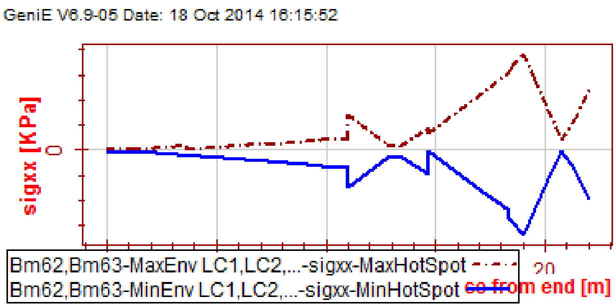
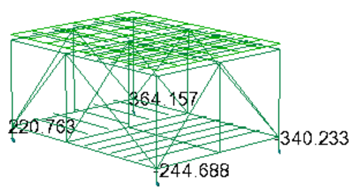
Equipment 	Equipments are used to create: • Point or line loads on beams for use in static analysis and • Masses for use in eigenvalue and dynamic analysis. The equipment includes user control of size and location, footprint, mass, method for calculation of forces and how to handle mass in dynamic analysis. Equipments may be resting on top or hanging below a deck, it may be resting on an inclined deck and be hanging on a vertical wall.
Load interface 	The load interface is used to limit equipment loading to certain beams, e.g. girders only.
Wind loads 	Wind loads are combinations of member loads and area loads. The area loads are generated in GeniE and combined with member loads from Wajac. The area loads are computed based on user defined areas, surfaces or equipment walls that are exposed to wind.
Load combinations **LComb1 = LCGrav*1.2 + LCBuoy*1.0 + LCWave*1.6 + LCWind*1.6**	Load combinations may be nested, i.e. a load combination may include another load combination. Load combinations can include loads from hydrodynamic and wind load analysis and they can be designated as an operation or storm condition for use by the API member code check.

Post-processing and reporting

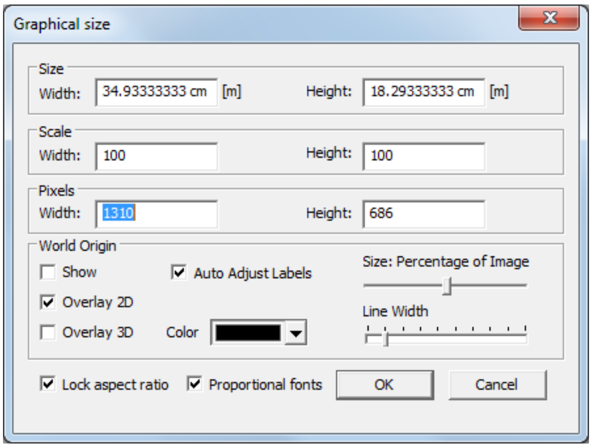
FEATURE	DESCRIPTION
Displacements 	Displacements for beams and plates are shown as contour plots in a 3D view. The deformed model may be viewed together with the un-deformed model. For a beam a cubic interpolation between the beam ends may be performed by including end rotations – this gives a realistic deformation pattern with a single beam element along a member.
Beam deflections 	Deflections along a member may be computed and presented in a 3D view and a 2D graph. It is also possible to do a check against the AISC provision of maximum deflection 1/180, 1/240 and 1/360 of the span. Envelopes (maxima and minima) over result cases may be presented in the 2D graph.
Plate and shell stresses 	GeniE presents element stresses (G-stress) for plates and shells as contour plots in 3D view. These are stresses extrapolated and interpolated from the result points within the individual elements. There is no averaging between adjoining elements. Stress components (sigxx, sigyy, tauxy, …), von Mises stress and principal stresses (P1, P2, P3) may be presented. More post-processing capabilities for stresses are available in Xtract that can be started from the GeniE user interface.
Principal plate and shell stress vectors 	Principal stresses P1, P2 and P3 may be shown as vectors on top of a contour plot of any displacement or stress component.

<table>
<tr>
<td>Beam forces and moments



</td>
<td>Beam forces and moments may be presented as a coloured wireframe in 3D view. Optionally, and for better identification of high values, the lines of the wireframe may be displayed as cylinders with diameter in proportion to the absolute value of the force/moment.

Beam forces and moments may also be shown as diagrams in a 3D view.

Finally, beam forces and moments may be presented in a 2D graph together with other beam results. The 2D graph allows envelopes (maxima and minima) over result cases to be presented.</td>
</tr>
<tr>
<td>Beam stresses

</td>
<td>Beam stresses are presented for selected members in a 2D graph. Envelopes (maxima and minima) over result cases may be presented.</td>
</tr>
<tr>
<td>Reaction forces

</td>
<td>Reaction forces in the support points may be shown as numeric values in a 3D view.</td>
</tr>
</table>

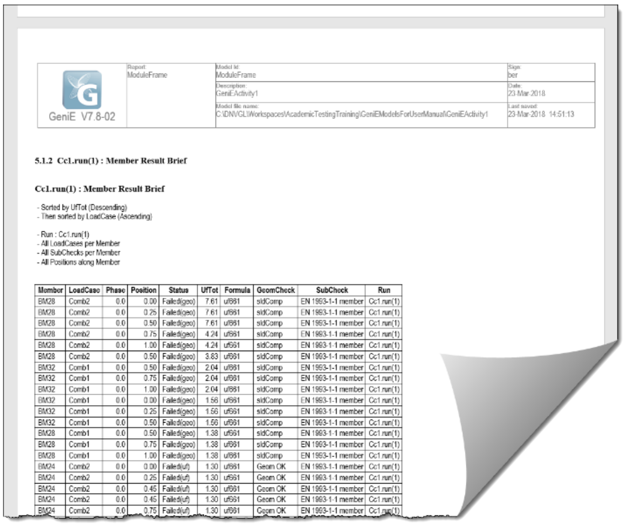
Save graphics

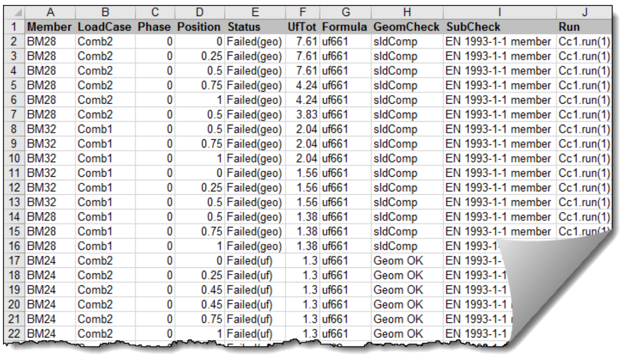


Graphics of the 3D view may be saved to alternative formats: gif, png, jpg, ps, bmp, tga, tif, pdf, hsf, hmf, obj and ply. The resolution of the graphics file is controlled by specifying number of pixels.

The 2D graph may be copied as a bitmap to the clipboard for pasting into a document.

Tabular report





	A	B	C	D	E	F	G	H	I	J
1	Member	LoadCase	Phase	Position	Status	UfTot	Formula	GeomCheck	SubCheck	Run
2	BM28	Comb2	0	0	Failed(geo)	7.61	uf661	sldComp	EN 1993-1-1 member	Cc1.run(1)
3	BM28	Comb2	0	0.25	Failed(geo)	7.61	uf661	sldComp	EN 1993-1-1 member	Cc1.run(1)
4	BM28	Comb2	0	0.5	Failed(geo)	7.61	uf661	sldComp	EN 1993-1-1 member	Cc1.run(1)
5	BM28	Comb2	0	0.75	Failed(geo)	4.24	uf661	sldComp	EN 1993-1-1 member	Cc1.run(1)
6	BM28	Comb2	0	1	Failed(geo)	4.24	uf661	sldComp	EN 1993-1-1 member	Cc1.run(1)
7	BM28	Comb2	0	0.5	Failed(geo)	3.83	uf661	sldComp	EN 1993-1-1 member	Cc1.run(1)
8	BM32	Comb1	0	0.5	Failed(geo)	2.04	uf661	sldComp	EN 1993-1-1 member	Cc1.run(1)
9	BM32	Comb1	0	0.75	Failed(geo)	2.04	uf661	sldComp	EN 1993-1-1 member	Cc1.run(1)
10	BM32	Comb1	0	1	Failed(geo)	2.04	uf661	sldComp	EN 1993-1-1 member	Cc1.run(1)
11	BM32	Comb1	0	0	Failed(geo)	1.56	uf661	sldComp	EN 1993-1-1 member	Cc1.run(1)
12	BM32	Comb1	0	0.25	Failed(geo)	1.56	uf661	sldComp	EN 1993-1-1 member	Cc1.run(1)
13	BM32	Comb1	0	0.5	Failed(geo)	1.56	uf661	sldComp	EN 1993-1-1 member	Cc1.run(1)
14	BM28	Comb1	0	0.5	Failed(geo)	1.38	uf661	sldComp	EN 1993-1-1 member	Cc1.run(1)
15	BM28	Comb1	0	0.75	Failed(geo)	1.38	uf661	sldComp	EN 1993-1-1 member	Cc1.run(1)
16	BM28	Comb1	0	1	Failed(geo)	1.38	uf661	sldComp	EN 1993-1-	
17	BM24	Comb2	0	0	Failed(uf)	1.3	uf661	Geom OK	EN 1993-1-	
18	BM24	Comb2	0	0.25	Failed(uf)	1.3	uf661	Geom OK	EN 1993-1-1	
19	BM24	Comb2	0	0.45	Failed(uf)	1.3	uf661	Geom OK	EN 1993-1-1	
20	BM24	Comb2	0	0.45	Failed(uf)	1.3	uf661	Geom OK	EN 1993-1-1	
21	BM24	Comb2	0	0.75	Failed(uf)	1.3	uf661	Geom OK	EN 1993-1-1	
22	BM24	Comb2	0	1	Failed(uf)	1.3	uf661	Geom OK	EN 1993-1-1	

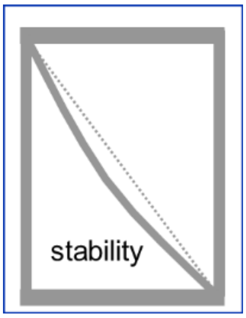
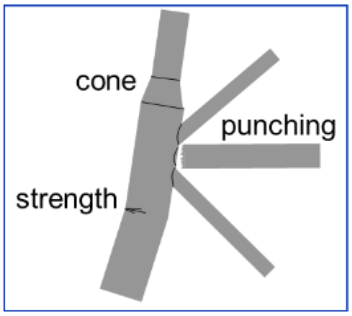
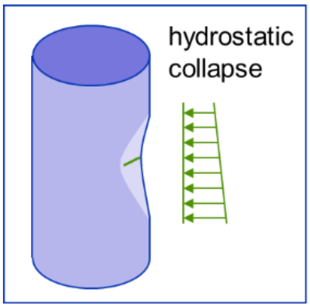
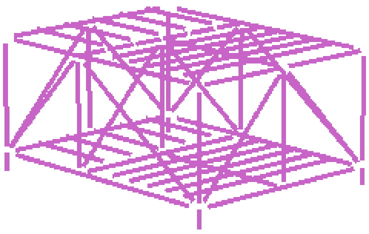
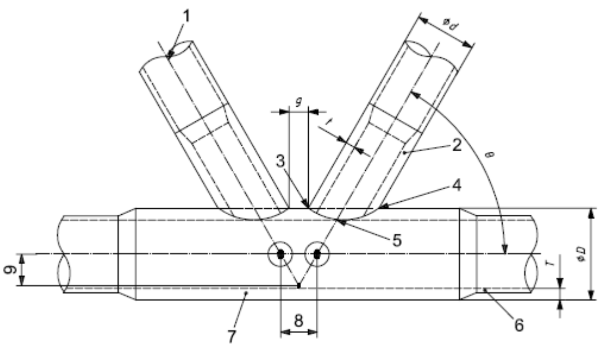
A tabular report can be saved to alternative formats: txt, html, MS Word (xml) and MS Excel (xml and csv).

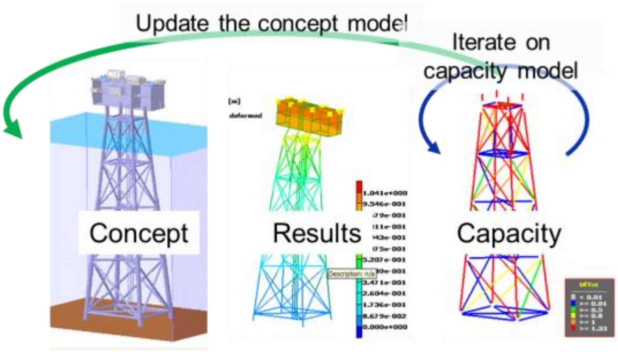
The user has full control of the content of the report. The report content is stored in the database so that it is easy to reproduce the report after model changes.

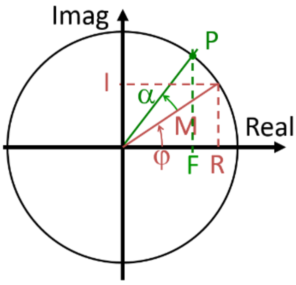
The report may include:

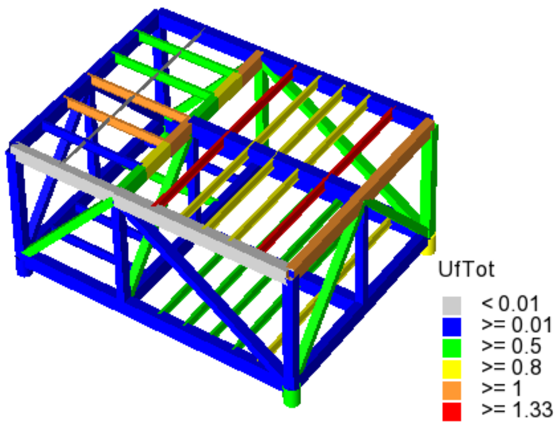
- Model data:
  - Beam and plate data
  - Properties
  - Masses
  - Loads
- FE analysis results
- Code checking results
- Graphics
- Your own text added

Member and tubular joint code checking – requires extension CCBM

FEATURE	DESCRIPTION
Code check analysis   	The following code checks are performed: • Member check • Hydrostatic collapse • Punching shear • Conical transition Code checks are performed for section profiles: • Pipe • Symmetrical/un-symmetrical I/H • Channel • Box • Massive bar • Angle • General • Torsion warping included for non-tubulars
Code checking model 	The capacity model is generated in GeniE and normally includes code checking parameters such as member buckling length and buckling factors, chord, can, stub and cone.
Code checking parameters 	Each standard has its own set of code checking parameter values. These values are the default values in GeniE but the user can manually override them if desired. The implementation of each code check standard is described in technical notes that are part of the GeniE installation.

<table>
<tr>
<td>Member redesign

</td>
<td>The redesign tool allows the user to modify parameters like section size, material quality and buckling length and immediately see the effect on the code check result. Multiple members may be evaluated at the same time. Such redesign is based on the assumption of no redistribution of forces caused by the redesign.

Final code check results based on redistribution of forces are generated by transferring redesign changes back to the model, re-running the structural analysis, generating code check results and updating the report(s). This is available from a single action in GeniE.</td>
</tr>
<tr>
<td>Tubular joint – chord thickness requirement</td>
<td>For API WSD 2005 GeniE will report the chord thickness required to pass the code check (i.e. utilisation factor less than 1.0).</td>
</tr>
<tr>
<td>Complex results

</td>
<td>Complex results from a frequency domain analysis may also be code checked. A combination of a static (non-complex) and a complex result case is code checked at given phase step intervals:

$F(\propto) = Static + R \cdot \cos(\propto) - I \cdot \sin(\propto)$</td>
</tr>
<tr>
<td>Result presentation


<table>
<tr><th>Capacity Model</th><th>LoadCase</th><th>Position</th><th>Status</th><th>UfTot</th><th>Formula</th></tr>
<tr><td>member(BM28)</td><td>Comb2</td><td>0.00</td><td>Failed(geo)</td><td>7.61</td><td>uf661</td></tr>
<tr><td>member(BM32)</td><td>Comb1</td><td>0.50</td><td>Failed(geo)</td><td>2.04</td><td>uf661</td></tr>
<tr><td>member(BM24)</td><td>Comb2</td><td>0.00</td><td>Failed(uf)</td><td>1.30</td><td>uf661</td></tr>
<tr><td>member(BM60)</td><td>Comb2</td><td>0.18</td><td>Failed(uf)</td><td>1.19</td><td>uf62</td></tr>
<tr><td>member(BM56)</td><td>Comb2</td><td>0.73</td><td>Failed(uf)</td><td>1.18</td><td>uf62</td></tr>
</table></td>
<td>The code check results may be presented in a 3D view or in tables on screen.

The code check results may also be saved to a report as shown above.</td>
</tr>
</table>

Supported standards for member and tubular joint checking

FEATURE	DESCRIPTION
American offshore standards	API-WSD 2002 – Offshore structures • Tubular: American Petroleum Institute RP 2A-WSD (21st edition December 2000, Errata and Supplement 1, December 2002) • Non-tubular: American National Standard; Specification for Structural Steel Buildings, AISC 360-xx (Steel Construction Manual 13th, 14th and 15th editions) API-WSD 2005 – Offshore structures • Tubular: American Petroleum Institute RP 2A-WSD (21st edition December 2000, Errata and Supplement 2, October 2005) • Non-tubular: American National Standard; Specification for Structural Steel Buildings, AISC 360-xx (Steel Construction Manual 13th, 14th and 15th editions) API-WSD 2014 – Offshore structures • Tubular: American Petroleum Institute RP 2A-WSD (22nd edition November 2014) • Non-tubular: American Institute of Steel Construction, Allowable Stress Design and Plastic Design, AISC 9th (June 1, 1989) API-LRFD 2003 – Offshore structures • Tubular: American Petroleum Institute LRFD (1st Edition/July 1, 1993/ Reaffirmed, May 16, 2003) • Non-tubular: American National Standard; Specification for Structural Steel Buildings, AISC 360-xx (Steel Construction Manual 13th, 14th and 15th editions)

NORSOK offshore standards	NORSOK 2004 and 2013 – Offshore structures • Tubular: NORSOK STANDARD N-004, Rev. 2, October 2004, and Rev. 3, February 2013. Design of steel structures • Non-tubular: EUROCODE 3, EN 1993 Part 1-1: General rules and rules for buildings. It is also possible to select the preferences according to the Norwegian and Danish National Annexes
ISO offshore standards	ISO 19902 2007 – Offshore structures • Tubular: INTERNATIONAL STANDARD ISO 19902, Petroleum and natural gas industries — Fixed steel offshore structures (First edition 1 December 2007) • Non-tubular: EUROCODE 3, EN 1993 Part 1-1: General rules and rules for buildings. It is also possible to select the preferences according to the Norwegian and Danish National Annexes
American onshore standards	AISC 360-05, 360-10 and 360-16 – Onshore structures: • Tubular and non-tubular: American National Standard; Specification for Structural Steel Buildings", versions from March 9 2005, June 22 2010 and July 7 2016. These versions are supported by AISC Steel Construction Manual 13th, 14th and 15th editions. The check covers design/utilisation of members according to the provisions for Load and Resistance Factor Design (LRFD) or to the provisions for Allowable Strength Design (ASD). AISC 335-89 – Onshore structures • Tubular and non-tubular: American National Standard; Specification for Structural Steel Buildings", June 1 1989. This version is supported by AISC Steel Construction Manual 9th edition. The check covers design/utilisation of members according to the provisions for Allowable Stress Design and Plastic Design (ASD).
EUROCODE onshore standard	EUROCODE 3 – Onshore structures • Tubular and non-tubular: EUROCODE 3, EN 1993 Part 1-1: General rules and rules for buildings. It is also possible to select the preferences according to the Norwegian and Danish National Annexes

Danish onshore and offshore standard DS DANSK STANDARD DANISH STANDARDS	DANISH STANDARD 412 / 449 – Onshore and offshore structures • Tubular profiles only in both DS 412 and DS 449

Plate code checking – requires extension CCPL

FEATURE	DESCRIPTION
Code check analysis	For CSR Bulk both panel yielding and buckling are assessed. For CSR panel buckling according to PULS is assessed.
Code checking model	The capacity model is generated in GeniE. There are three different ways to create the panel capacity models.
Supported standards	CSR Bulk: Common Structural Rules for Bulk Carriers, IACS, January 2006 CSR Tank – July 2008: Common Structural Rules for Double Hull Oil Tankers with Length 150 Metres and Above, IACS, July 2008
Redesign	The user may add local details to the code checking models such as stiffener locations. For refined design of panels according to the PULS standard it is possible to export the data to a separate tool (requires an installation of Nauticus Hull). By this tool, it is possible to modify all parameters like load, thickness, material, section type, locations and boundary conditions.
Result presentation	The code check results may be presented in a 3D view or in tables.
Fatigue for floating structures	Fatigue screening and calculations

Import and export data in GeniE

FEATURE	DESCRIPTION
Section library	GeniE includes section libraries for the AISC, Euronorm & Norwegian and the British Standards. There are also several hull specific profiles for angle, bulb, flatbar and tbar. Users may create their own libraries for sharing and re-use.
Material library	GeniE comes with a material library consisting of about 70 material types. Users may also create their own libraries.
GeniE XML file	The XML (eXtended Markup Language) is the most complete storage format for GeniE models. It contains all definitions to recreate structure, loads, analysis set-ups and member code checking. It is the most secure source of data export and import. It is also used when upgrading model data to a newer release of GeniE.
GeniE JS file	The js file based on JavaScript and is a log of all user commands except graphic manipulations. It can be edited and used as input. Utilizing the power of the JavaScript the js file can be used for parametric modelling.
GeniE condensed JS file	GeniE may export a condensed js file (also referred to as "clean JS file"). This is a file for recreating the model as is without the steps of its creation. The exported js file is intended for use in jacket and topside modelling as it does not cover curved surfaces or punched plates/surfaces.
Sesam Input Interface File also termed FEM file and T file	This file includes the finite element data. It is used by the other Sesam analysis tools. The FEM file can also be imported into GeniE for modification. This file is used for converting old Sesam models to GeniE format. Note that a curved surface is represented as facetted elements in a FEM file and will be imported as such.
Wajac.INP Wajac analysis control (input) file	GeniE will export a Wajac.inp file for use by the wave and wind load analysis in Wajac. GeniE may also import data from a Wajac.inp file.
Gensod.INP Gensod input file	GeniE will export a Gensod.inp file for use by the pile-soil analysis. GeniE may also import data from a Gensod.inp file.

For import and export towards external software and formats see subsection 'Import and Export Features of Sesam' in section 'Introduction to Sesam'.

HydroD

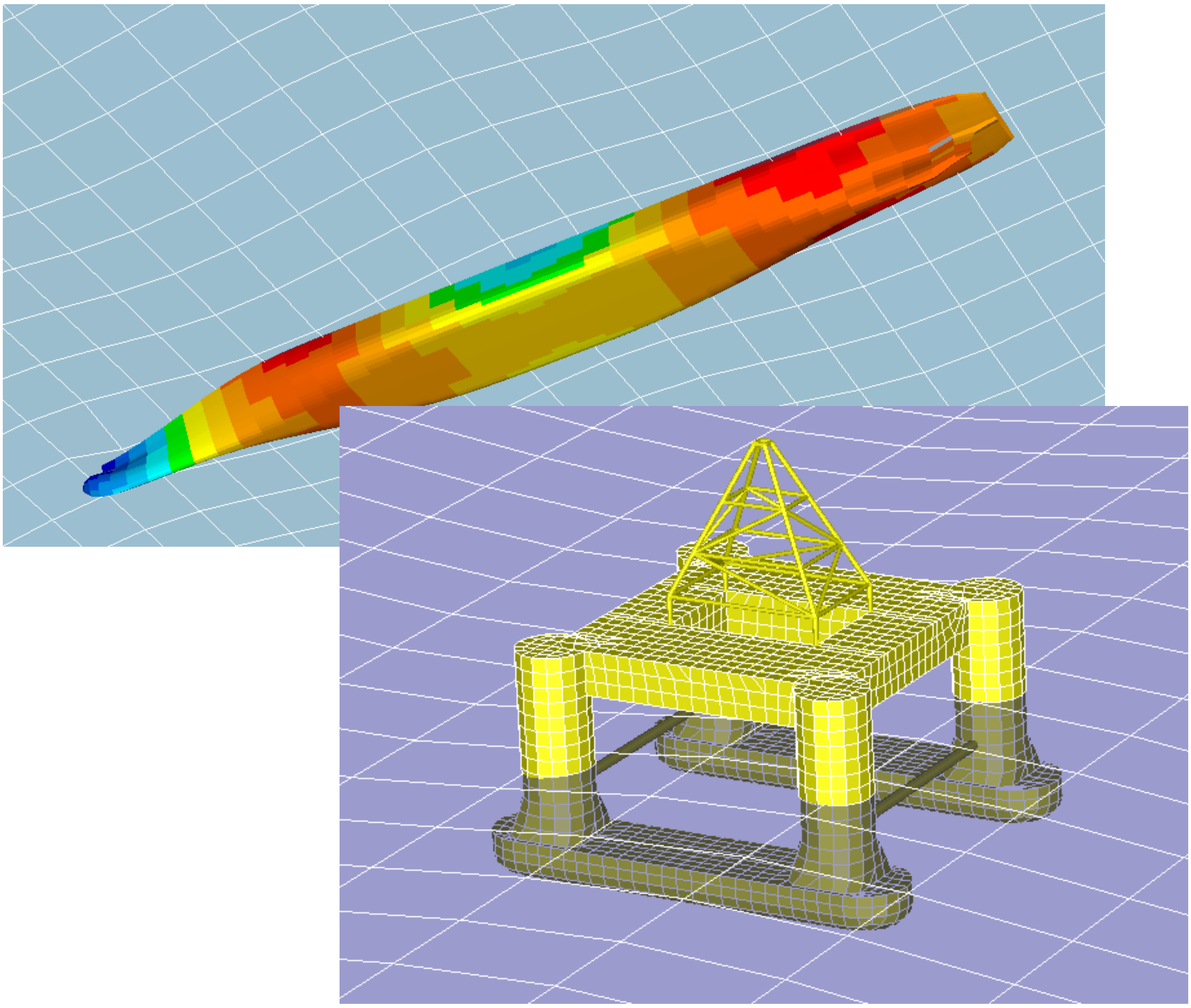
HYDRODYNAMIC AND HYDROSTATIC ANALYSIS

Last revised: January 9, 2019. Describing versions 4.10-01 (32-bit) and 5.1-07 (64-bit).

HydroD is a tool for:

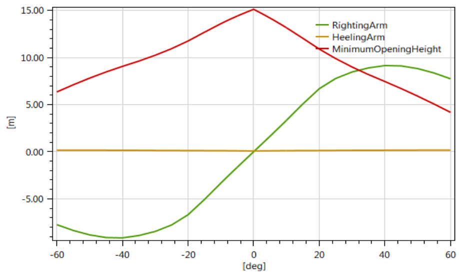
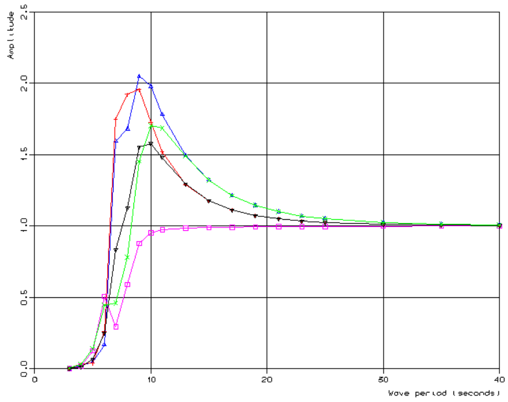
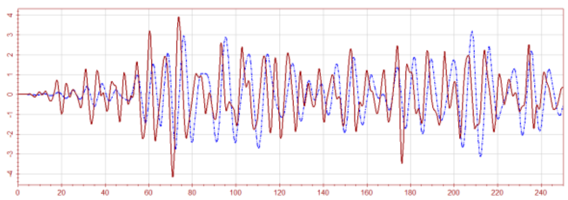
- Hydrostatic and stability analysis (version 5.1-07)
- Hydrodynamic (wave load and motion response) analysis (version 4.10-01)

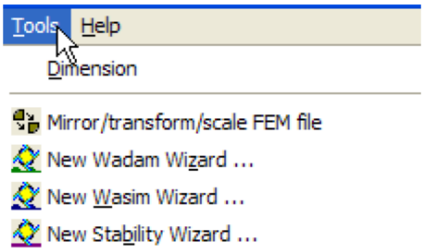
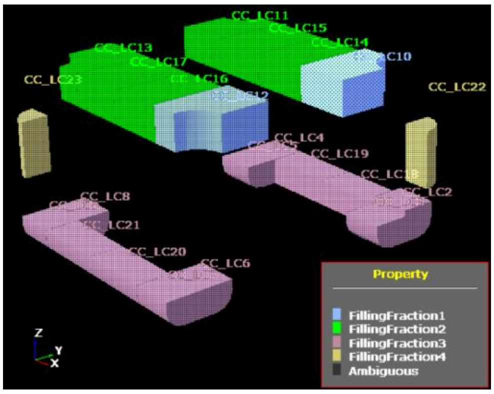
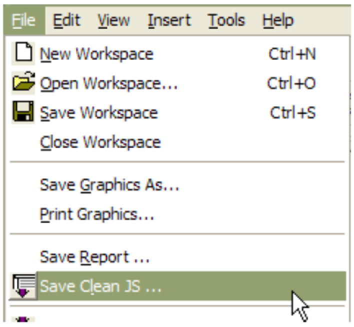
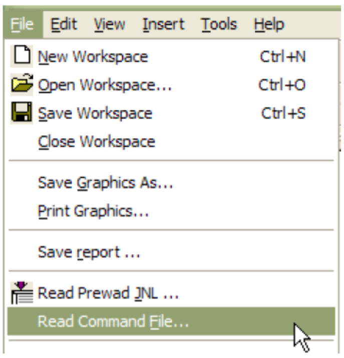
As concerns hydrodynamics, the environment is modelled in HydroD while the hydrodynamic analysis is performed by running Wadam (linear frequency domain theory) or Wasim (non-linear time domain theory) in the background. The panel model is normally created in GeniE but will in certain cases be created in HydroD. HydroD provides analysis workflows for execution of Wadam and Wasim. Multiple floating equilibrium positions can be computed depending on mass and compartment filling.



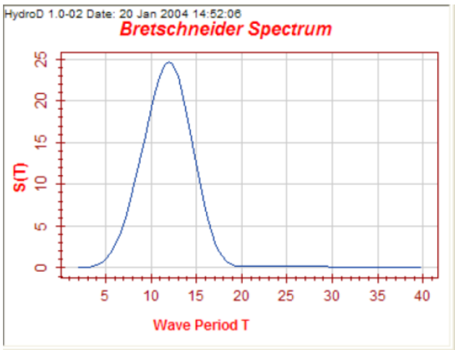
FEATURES OF HYDROD

General features

FEATURE	DESCRIPTION
Analysis types - stability 	HydroD provides the following stability analysis types: • Hydrostatic balancing • Stability analysis • Maximum KG analysis • Strength analysis • Watertight and weathertight surfaces
Analysis types - hydrodynamics  	HydroD provides the following hydrodynamic analysis types: • Frequency domain analysis of stationary floating or fixed rigid bodies (Wadam) • Deterministic analysis of floating or fixed rigid bodies (Wadam) • Time domain analysis of floating or fixed rigid bodies (Wasim) • Time domain analysis of ships with forward speed, with conversion to frequency domain (Wasim)
Unit support m? mm? inch?	The user may mix units throughout the modelling. The data logging (scripting) ensures that re-creating the model gives the same result.

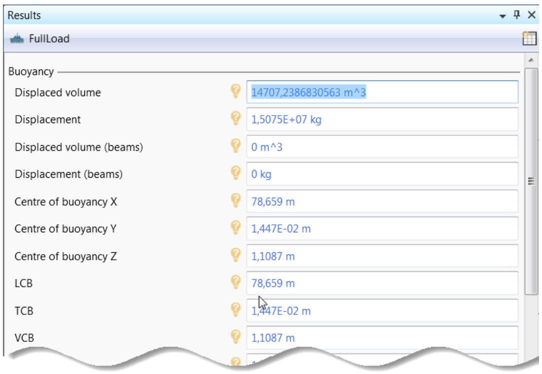
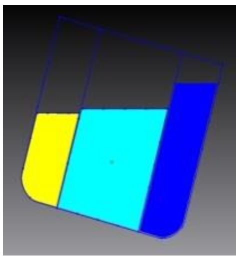
<table>
<tr>
<td>Wizards

</td>
<td>Wizards are available for guiding the user through all necessary steps to set up the following analyses:
Stability
Linear hydrodynamic analysis in Wadam
Non-linear hydrodynamic analysis in Wasim</td>
</tr>
<tr>
<td>Colour coding properties

</td>
<td>All properties may be displayed in separate colours for verification purposes.</td>
</tr>
<tr>
<td>Save Clean JS

</td>
<td>Save a complete and cleaned-up JS script file containing all the objects in your workspace in a defined sequence.</td>
</tr>
<tr>
<td>Read Command File (JS file)

</td>
<td>A JS script file may be edited and used as input.

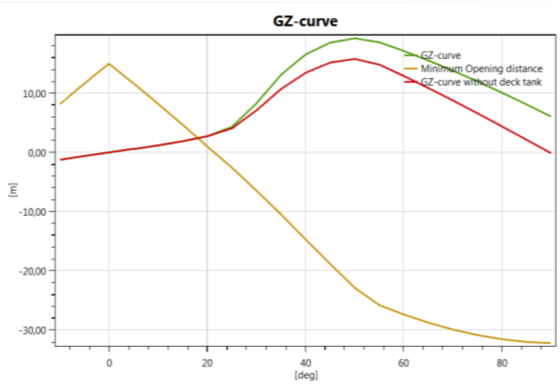
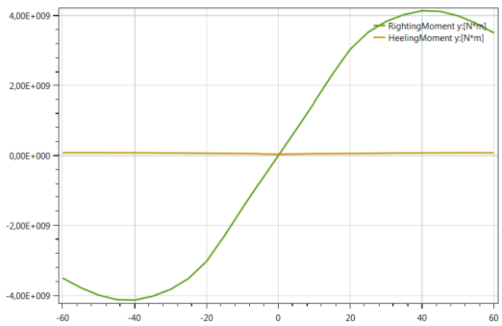
HydroD 5.1 and later can read a JS file also from HydroD 4.10.</td>
</tr>
</table>

Graph control 	Many of the dialogs utilize a graph control allowing manipulation of the graph.

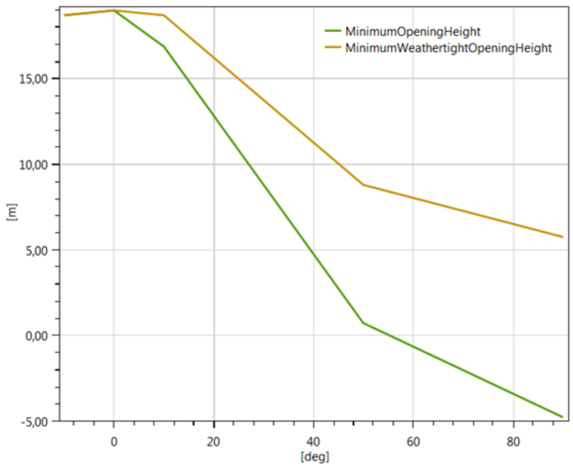
Features for hydrostatic analysis

Hydrostatic and stability computations may be run for both intact and damage conditions. HydroD will compute the draught and heel/trim angles to ensure equilibrium. Compartments may be flooded or balanced by HydroD. A wind heeling moment, user defined or calculated by HydroD, may be included.

FEATURE	**DESCRIPTION**
Hydrostatic data 	Hydrostatic data are computed: • Displaced volume • Mass with and without compartment fluid • Centre of gravity and centre of buoyancy • Centre of flotation • Metacentre • Trim moment • Compartment information
Compartments 	• Compartments are employed in hydrostatic and stability computations • The free surface inside compartments is always horizontal • The free surface in damaged compartments is always at the free surface level.
Hydrostatic balancing of compartments	HydroD may calculate the filling ratio of compartments necessary to obtain equilibrium of gravity and buoyancy forces.

GZ curve 	HydroD computes the GZ curve for the structure (with and without the influence of deck compartments for offshore structures). • The GZ curve is displayed • The shortest distance between a flooding opening and the sea surface is displayed • Zero crossings of the GZ-curve are reported • Zero crossings of the lowest flooding opening are reported • Change in trim and waterline is reported at each heel angle • Integrals of the GZ-curve are reported
Righting and heeling moment 	HydroD computes the righting and heeling moment. • The righting moment curve is displayed • Righting moment zero crossings are reported • Integral of righting moment is reported • The wind heeling moment is computed according to ship or offshore rules, or it may be user defined • The heeling force model may be defined from a combination of several element models • The wind area is computed at every heel angle • The heeling moment curve is displayed • Integral of heeling moment is computed

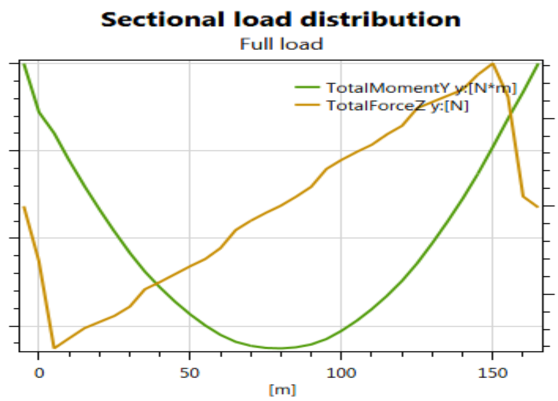
Openings



Flooding openings in the hull may be defined at selected locations.

- Openings can be Unprotected, Weathertight or Watertight
- Heeling angle of intersection with the waterline is printed
- An unprotected opening may be connected to a compartment, making this flooded when the opening becomes submerged

Strength analysis



HydroD computes the sectional loads in the still water condition on the specified side of the defined cross sections.

- Cross sectional loads, forces and moments, may be both displayed, e.g. longitudinal bending moment, and printed
- Mass and buoyancy distribution may also be displayed

Hydrostatic rule checks

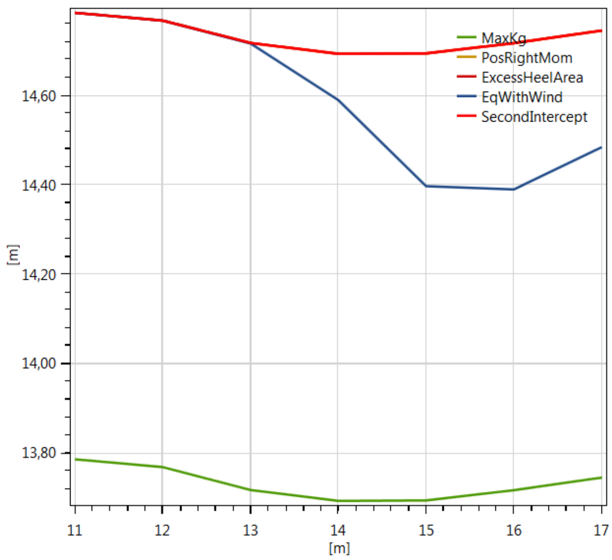
StabilityRuleCheck1

	Stability Analysis	Load case	Result
1	StabilityAnalysis1	LoadingCondition1 / A0 deg	PASS
2	StabilityAnalysis1	LoadingCondition1 / DamageCase1 / A0 deg	FAIL
3	StabilityAnalysis1	LoadingCondition1 / DamageCase2 / A0 deg	FAIL
4	StabilityAnalysis1	LoadingCondition1 / A45 deg	PASS
5	StabilityAnalysis1	LoadingCondition1 / DamageCase1 / A45 deg	PASS
6	StabilityAnalysis1	LoadingCondition1 / DamageCase2 / A45 deg	PASS
7	StabilityAnalysis1	LoadingCondition1 / A90 deg	PASS
8	StabilityAnalysis1	LoadingCondition1 / DamageCase1 / A90 deg	PASS
9	StabilityAnalysis1	LoadingCondition1 / DamageCase2 / A90 deg	PASS
10	StabilityAnalysis1	LoadingCondition1 / A135 deg	PASS
11	StabilityAnalysis1	LoadingCondition1 / DamageCase1 / A135 deg	PASS
12	StabilityAnalysis1	LoadingCondition1 / DamageCase2 / A135 deg	PASS
13	StabilityAnalysis1	LoadingCondition1 / A180 deg	PASS
14	StabilityAnalysis1	LoadingCondition1 / DamageCase1 / A180 deg	FAIL
15	StabilityAnalysis1	LoadingCondition1 / DamageCase2 / A180 deg	FAIL
16	StabilityAnalysis1	LoadingCondition1 / A225 deg	PASS
17	StabilityAnalysis1	LoadingCondition1 / DamageCase1 / A225 deg	PASS
18	StabilityAnalysis1	LoadingCondition1 / DamageCase2 / A225 deg	PASS
19	StabilityAnalysis1	LoadingCondition1 / A270 deg	PASS
20	StabilityAnalysis1	LoadingCondition1 / DamageCase1 / A270 deg	PASS
21	StabilityAnalysis1	LoadingCondition1 / DamageCase2 / A270 deg	PASS
22	StabilityAnalysis1	LoadingCondition1 / A315 deg	PASS
23	StabilityAnalysis1	LoadingCondition1 / DamageCase1 / A315 deg	PASS
24	StabilityAnalysis1	LoadingCondition1 / DamageCase2 / A315 deg	PASS

HydroD may check the analysis results against rules defined by internationally recognised codes:

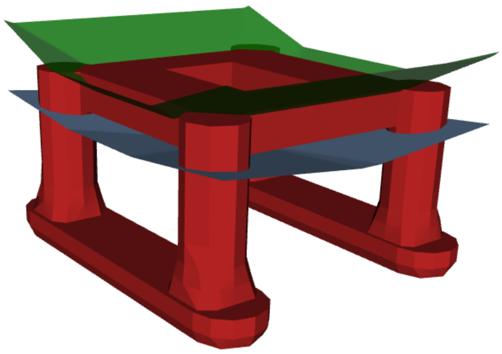
- IMO general
- MARPOL (tanker) intact
- MARPOL (tanker) damage
- IGC (liquefied gas in bulk) damage
- IBC (chemicals in bulk) damage
- NMD (mobile offshore units) intact
- NMD (mobile offshore units) damage
- DNV GL intact
- DNV GL damage
- IMO MODU (mobile offshore units) intact
- IMO MODU (mobile offshore units) damage
- ABS MODU (mobile offshore units) intact
- ABS MODU (mobile offshore units) damage
- User defined

Maximum KG analysis



HydroD can perform an AVCG/MaximumKG analysis based on the following rules

- IMO general
- MARPOL (tanker) intact
- MARPOL (tanker) damage
- IGC (liquefied gas in bulk) damage
- IBC (chemicals in bulk) damage
- NMD (mobile offshore units) intact
- NMD (mobile offshore units) damage
- DNV GL intact
- DNV GL damage
- IMO MODU (mobile offshore units) intact
- IMO MODU (mobile offshore units) damage
- ABS MODU (mobile offshore units) intact
- ABS MODU (mobile offshore units) damage

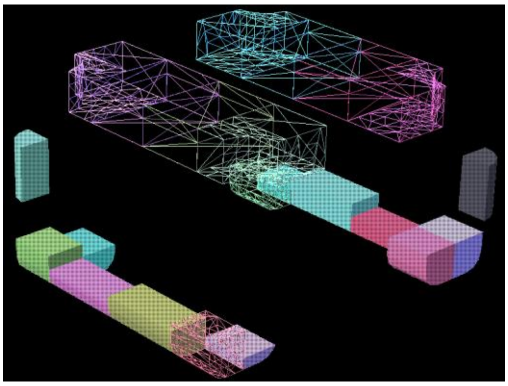
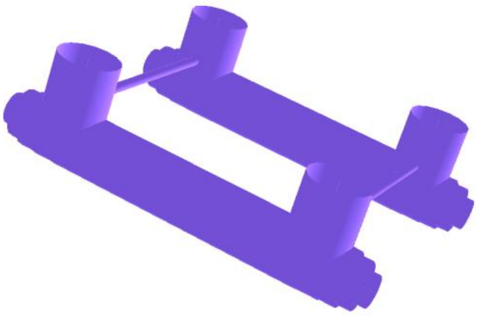
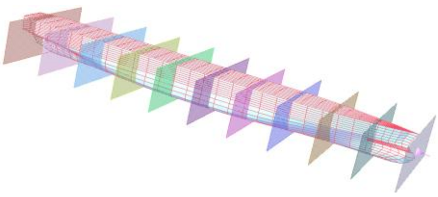
Limit surfaces 	Watertight and weathertight integrity surfaces can be computed and displayed.
Beach lines 	Beach lines, intersection of a horizontal plane and a limit surface, can be computed and displayed.

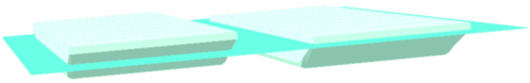
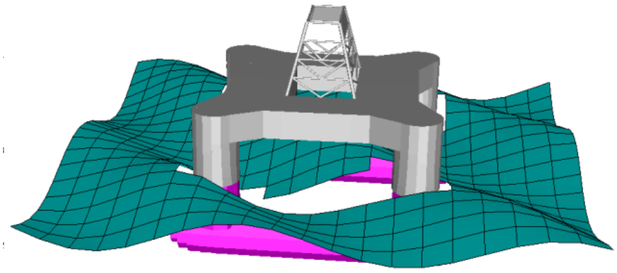
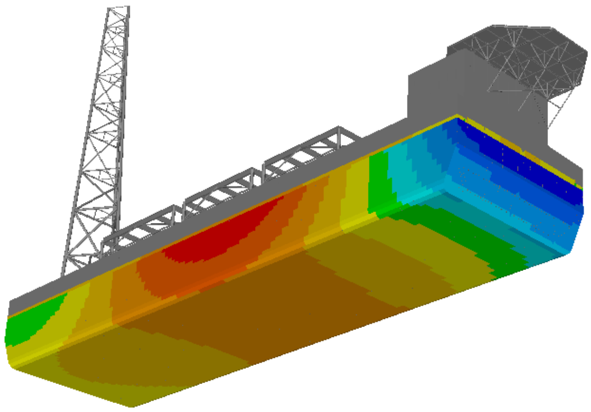
Features for hydrodynamic analysis (Wadam and Wasim)

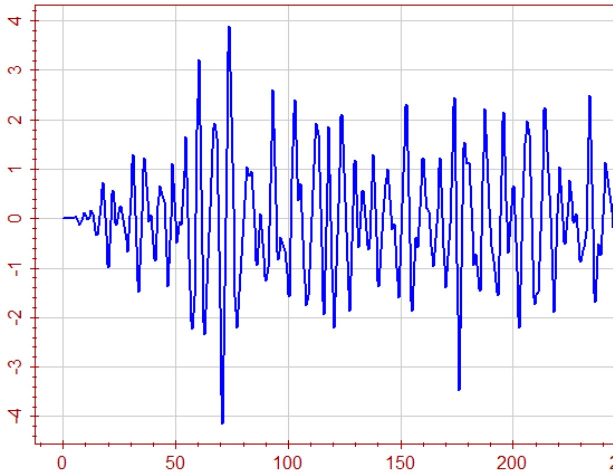
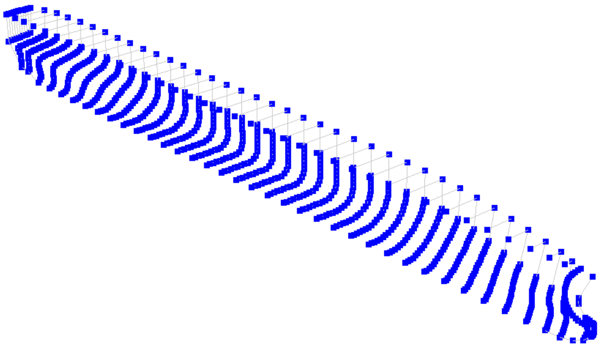
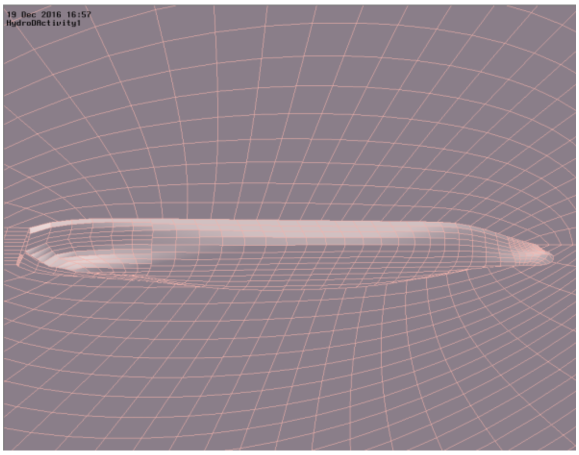
The hydrodynamic analysis is performed by the programs Wadam and Wasim.

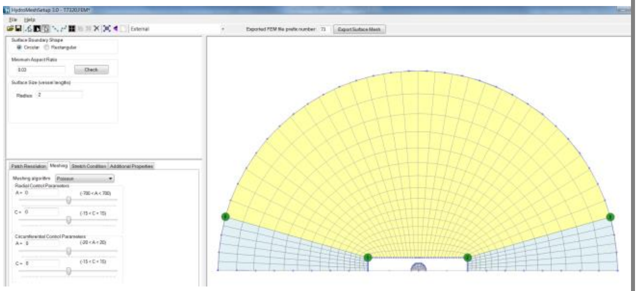
Wadam uses the Morison equation and first and second order 3D potential theory for the wave load calculations. The incident wave is an Airy wave and the analysis is performed in the frequency domain.

Wasim uses the Morison equation and solves the 3D diffraction/radiation problem by a Rankine panel method. The incident wave is an Airy, Stream Function or Stokes 5th wave and the analysis is performed in the time domain.

FEATURE	DESCRIPTION
Compartments 	Define fluids and filling fractions for the compartments. Define how the compartments are to be included in the hydrodynamic analysis • Full hydrodynamic solver for the internal fluid • Quasi-static method Specify compartment content or ballast compartments to achieve required equilibrium
Morison model 	A Morison model can be included in addition to a panel model to handle slender members • Structural parts not included in the panel model (e.g. braces) where all loads are computed from Morison's equation • Structural parts also included in the panel model (e.g. legs and pontoons) to get the combined effect of radiation/diffraction and viscous drag
Load cross sections 	Define cut planes for computation of section loads • Individual planes • Sequence of planes • Normal to x-, y- or z-axis

Multi-body model 	Create a multi-body model. The model can include different bodies or multiple occurrences of the same body, possibly with different loading conditions.
Global response  	A global response analysis may include: • First order wave excitation forces and moments • Second order wave excitation forces and moments (used to model springing effects, low frequency forces etc.) • Hydrodynamic added mass and damping • First and second order rigid body motions • Sectional forces and moments • Steady drift forces and moments • Wave drift damping • Sectional load components (mass, added mass, damping and excitation forces) • Panel pressures • Fluid particle kinematics (for gap calculations and free surface animation) • Second order free surface elevations • Calculation of selected global responses of a multi-body system
Transfer of structural loads 	Transfer of structural loads to a finite element (FE) model may include: • Inertia loads • Line loads on beam elements from Morison model • Point loads from pressure areas, anchor elements etc. from Morison model • Pressure loads on plate/shell/solid elements • Internal tank pressure in compartments

Environment 	Define the environment that is to be used in a hydrodynamic analysis. • Frequency domain condition for computation of transfer functions • Regular wave set for time domain simulation • Irregular time domain condition (a realization of a given sea state)
Section model 	A set of curves describing the hull. Can be imported from a "pln-file" (Sesam internal format) or DXF file with curves given as polylines. HydroD can make a panel model from a section model.
Automatic creation of mesh from section model 	Create mesh on hull for Wadam (panel model) and/or create mesh on hull and free surface for Wasim. Free surface mesh can also be used for Wadam wave/current interaction or wave elevation animation.

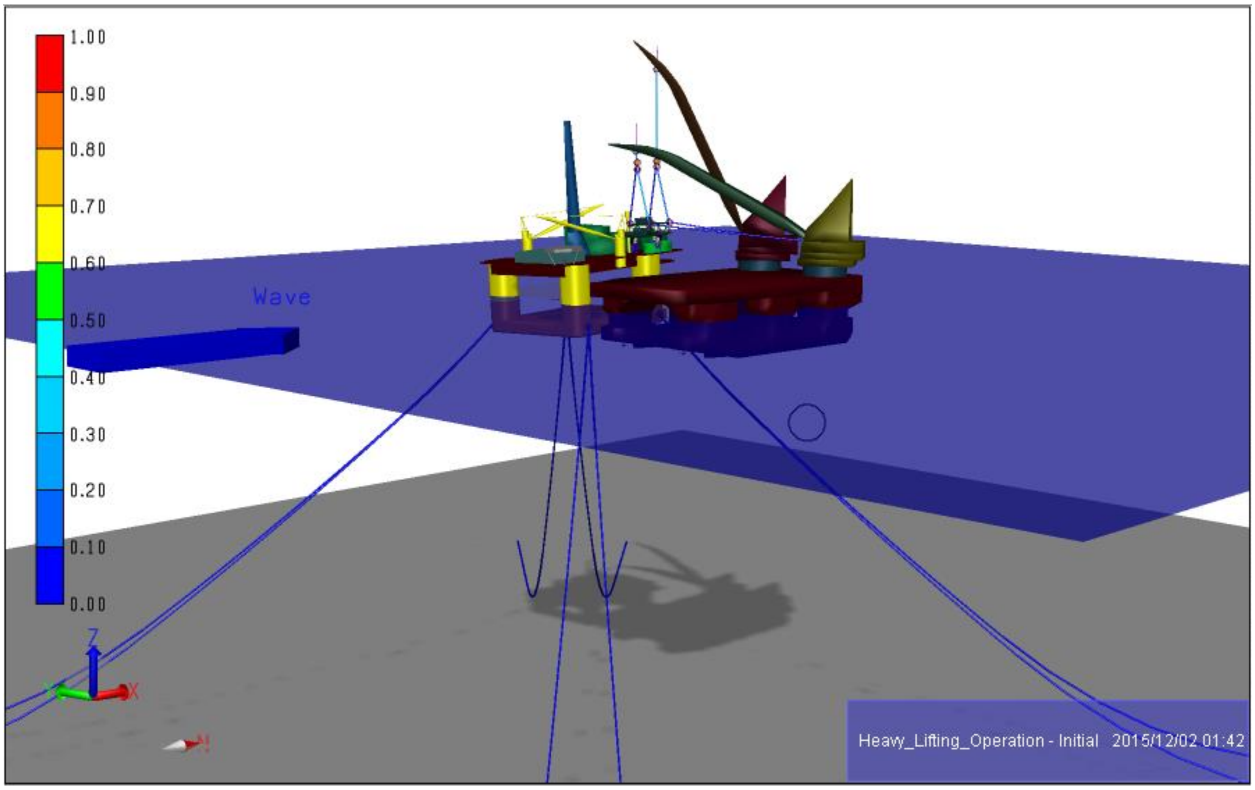
Interactive creation of free surface meshes 	A separate tool, HydroMesh version 3.1-01, is started from HydroD to create free surface meshes for: • Damping lids for single or multi-body problems • Internal lids for irregular frequency removal • Free surface mesh for wave elevation animation • Free surface mesh for second-order analysis or wave/current interaction and wave drift damping analysis • Free surface mesh for Wasim analysis if the automatically created mesh is not good enough

Sima

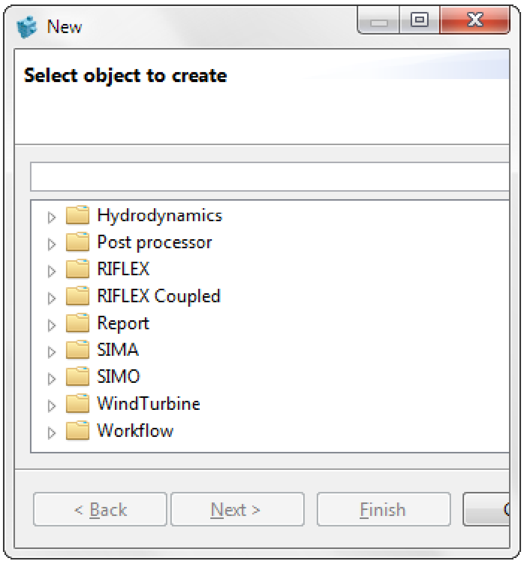
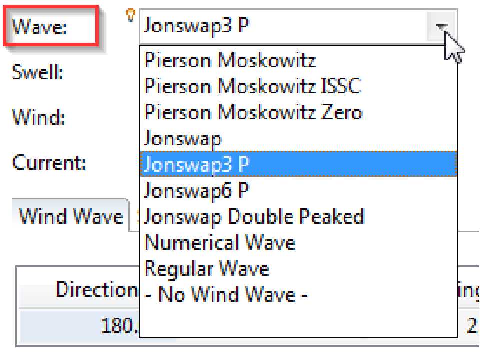
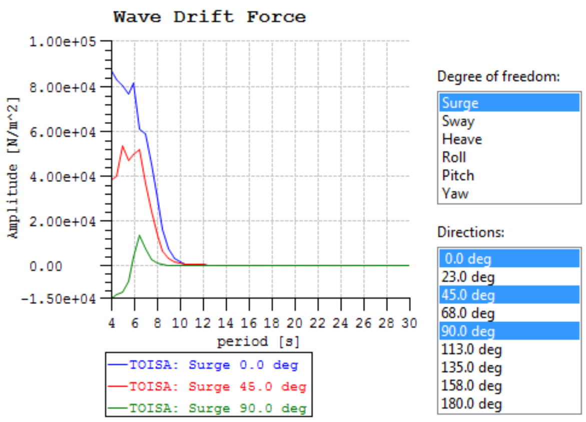
SIMULATION OF MARINE OPERATIONS

Last revised: March 12, 2019. Describing version 3.6-01 (64-bit).

Sima is a complete tool for simulation of marine operations from modelling to results presentation. Programs for dynamic analysis are run in the background under control of Sima. Both 3D and 2D graphics make understanding the results fast and intuitive.

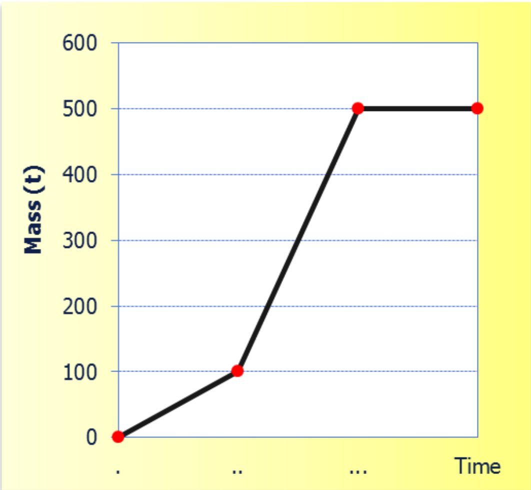


FEATURES OF SIMA

<table>
<tr><th>FEATURE</th><th>DESCRIPTION</th></tr>
<tr><td>GUI for Simo, Riflex and Vivana

</td><td>Sima supports various types of analysis. Sima is used as the general GUI and platform for various calculation programs such as Simo, Riflex and Vivana.</td></tr>
<tr><td>Wave, swell, current and wind

</td><td>Wave, swell, current and wind can be specified with different spectrums and profiles. They can have either collinear or different incoming directions.</td></tr>
<tr><td>Hydrodynamic coefficients

</td><td>Hydrodynamic coefficients can be read into Sima directly, i.e. added mass and damping coefficients, first and second order wave forces, etc.

Retardation function can also be calculated.</td></tr>
</table>

<table>
<tr>
<td>
Options and applications

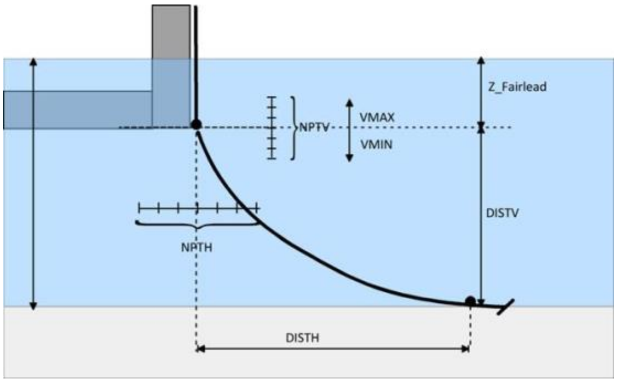


</td>
<td>
Options:

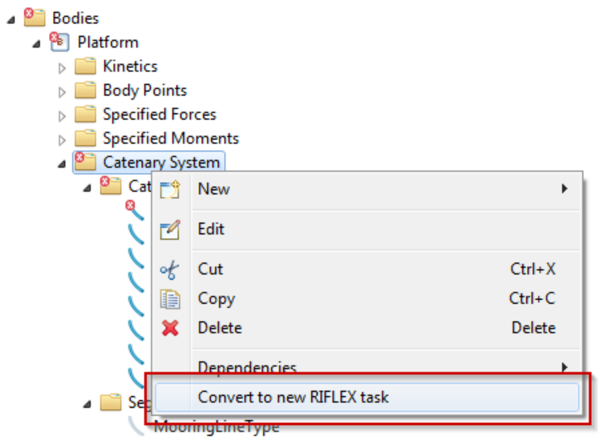
- Specified mass change rate at given position
- Filling of tank with simple geometry, specify flow rate, effect of slack tanks

Applications:

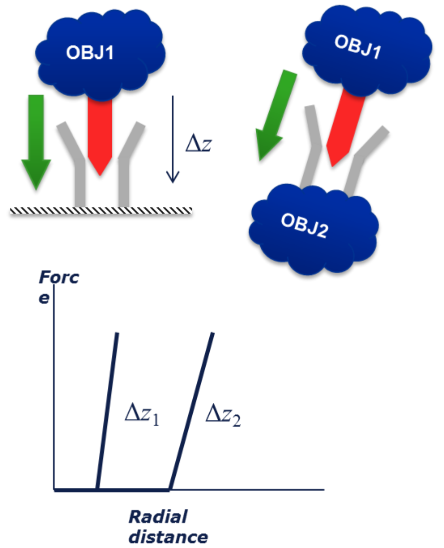
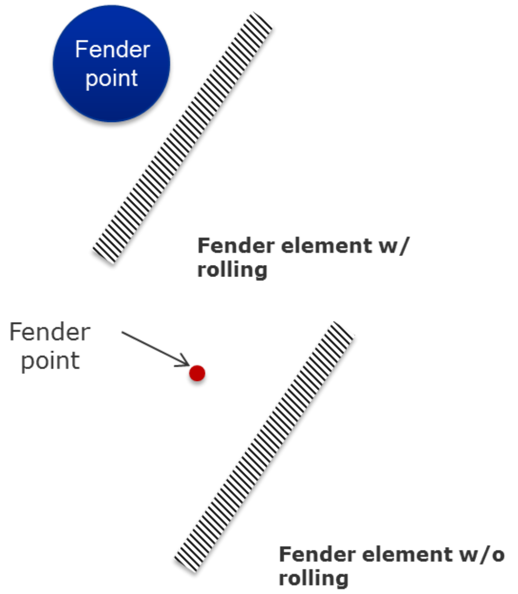
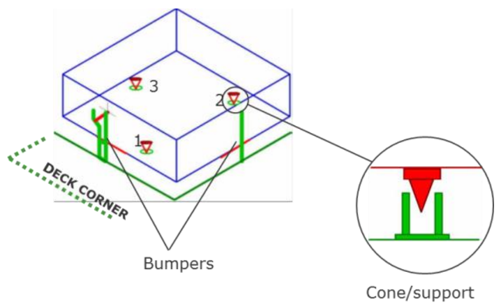
- Simulate a ballasting sequence and its influence on the static and dynamic response of a vessel
- Model a sudden or gradual water filling of a subsea module after a specified time
- The specified increase or decrease in mass both affect the gravity force and the mass matrix
</td>
</tr>
<tr>
<td>
Catenary method



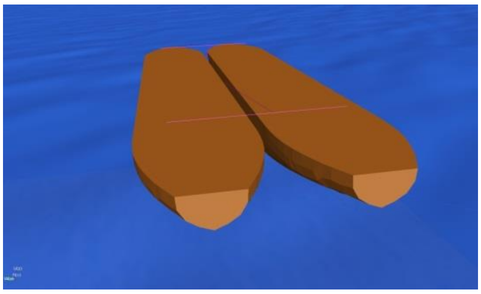
</td>
<td>
Catenary method (in Simo) can be used to simulate mooring system with quasi-static method. Three different approaches can be used with different types of input parameters.
</td>
</tr>
<tr>
<td>
Simo catenary to Riflex slender system



</td>
<td>
An advanced function in Sima can help users convert a Simo catenary system to a Riflex slender system. In Riflex a FE analysis of the slender structure (e.g. mooring lines and risers) can be carried out. Users can also compare the results from the catenary method and the dynamic FE method.
</td>
</tr>
</table>

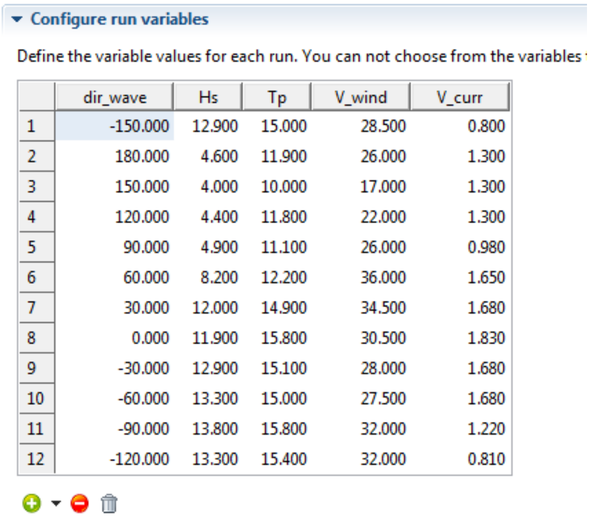
Docking cone 	A docking cone model can be used both as a global positioning element and a coupling element between bodies.
Fender model 	The fender model can be used both as a positioning element and as a coupling element. It expresses the contact force between a fender (point or cylinder) and a plane. The following characterizes the fender model: • Zero contact force for distances larger than a specified value • Compression force is normal to the plane calculated from a specified deformation – force relation • In-plane friction is proportional to the normal force (static and dynamic friction may be different) – shear stiffness and deformation of the fender is included • The plane can have any position and orientation
Bumper element 	The bumper element model is used to model contact force between a body and a globally fixed cursor or bumper, or contact forces between bodies. The bumper element is particularly useful in the analysis of offshore installation operations where deflectors/bumper bars are used to guide a module to its correct position and to protect existing equipment from impact damages.

Hydrodynamic coupling coefficients



For multibody problems, such as side-by-side mooring analysis, hydrodynamic coupling coefficients can be included in Sima.

Multiple analyses



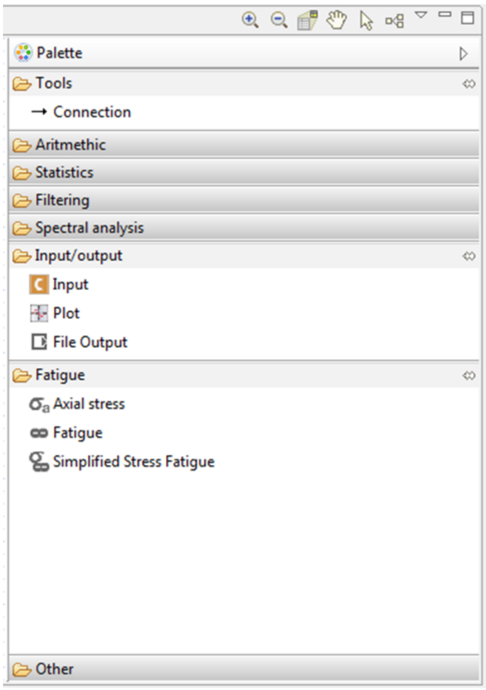
Configure run variables

Define the variable values for each run. You can not choose from the variables

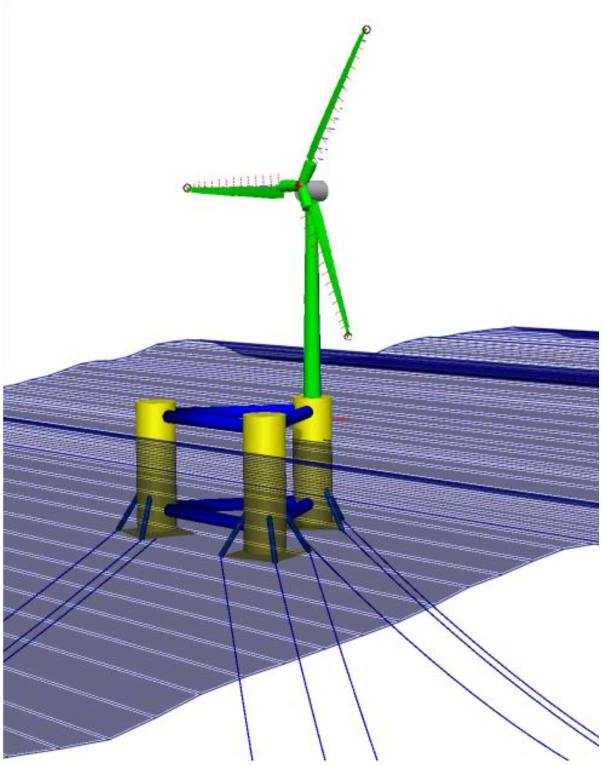
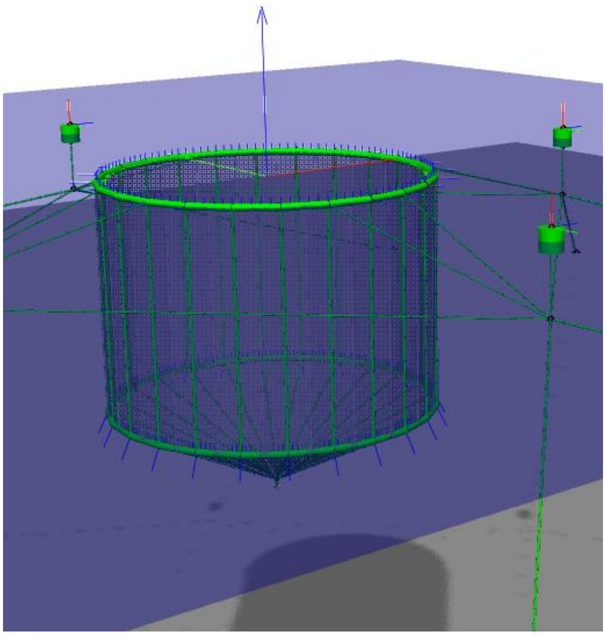
	dir_wave	Hs	Tp	V_wind	V_curr
1	-150.000	12.900	15.000	28.500	0.800
2	180.000	4.600	11.900	26.000	1.300
3	150.000	4.000	10.000	17.000	1.300
4	120.000	4.400	11.800	22.000	1.300
5	90.000	4.900	11.100	26.000	0.980
6	60.000	8.200	12.200	36.000	1.650
7	30.000	12.000	14.900	34.500	1.680
8	0.000	11.900	15.800	30.500	1.830
9	-30.000	12.900	15.100	28.000	1.680
10	-60.000	13.300	15.000	27.500	1.680
11	-90.000	13.800	15.800	32.000	1.220
12	-120.000	13.300	15.400	32.000	0.810

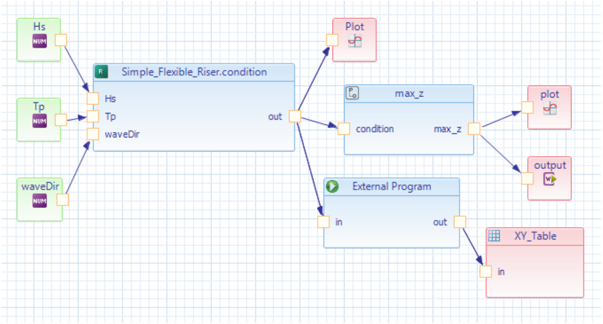
In addition to a single condition a condition set and condition space can be specified to define multiple analyses and run them in parallel.

Built-in postprocessor



The built-in post-processor includes different filters and input/output control. Customized plots can also be defined.

Floating wind turbine 	A floating wind turbine can also be simulated in Sima. The blades, turbine and control system can either be specified in Sima or be given by link to an external controller. The hydrodynamics of the floating substructure can be read from general hydrodynamic analysis programs. The mooring system is defined in Sima. The whole system can be analysed in time domain by a fully coupled method.
Aquaculture 	Sima also provides modelling possibilities for fish nets and fish cages. The floating ring, bottom ring etc. have formulations for partial submergence of horizontal slender structures. The net is modelled as a slender structure, where the user inputs the flow reduction factor manually. Fish net models can be combined with other Simo and Riflex elements, e.g. mooring system, connection cables and vessels.

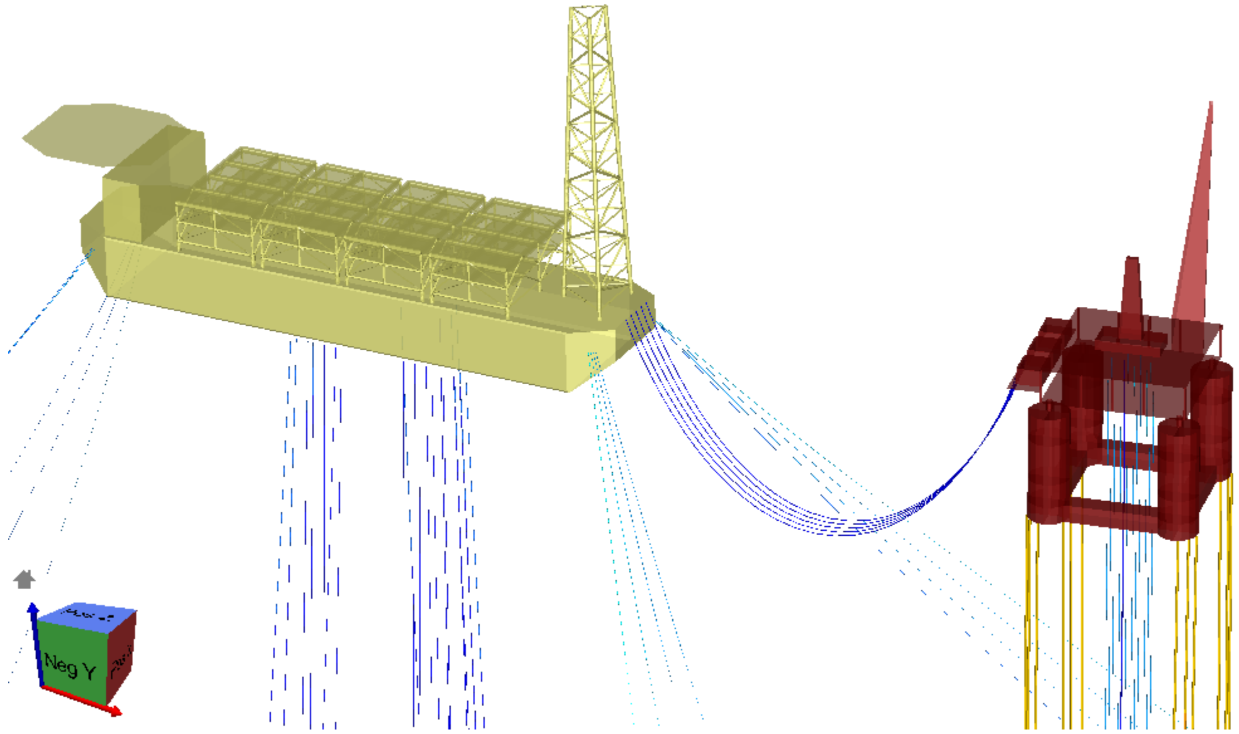
Workflows 	Workflows can be set up in Sima to prepare input, run Sima models, do post-processing and store results in various formats. A workflow can contain other workflows (nesting), post-processors and external programs, visualized as flowcharts.

DeepC

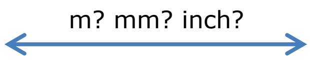
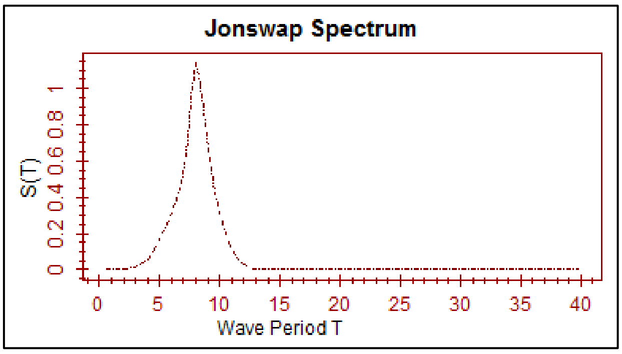
DEEP WATER COUPLED FLOATER MOTION ANALYSIS

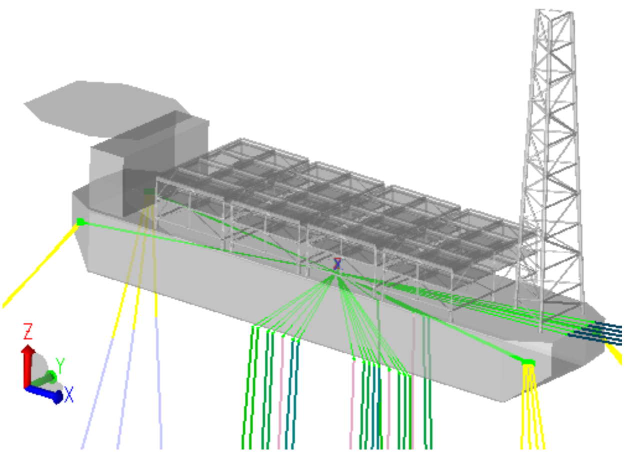
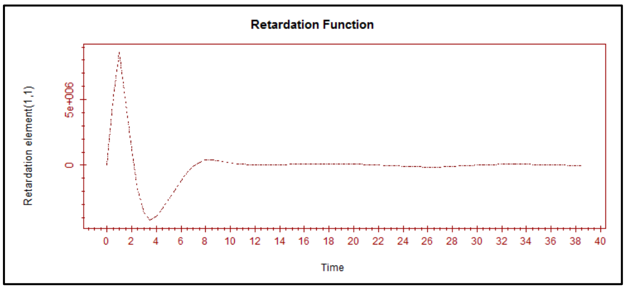
Last revised: April 23, 2018. Describing version 5.2-02 (64-bit).

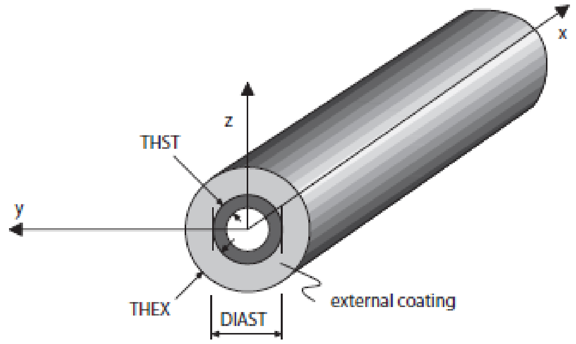
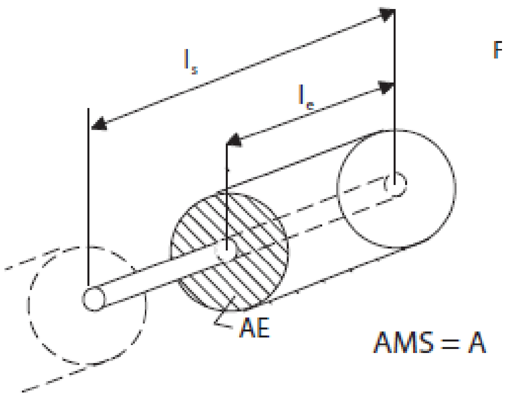
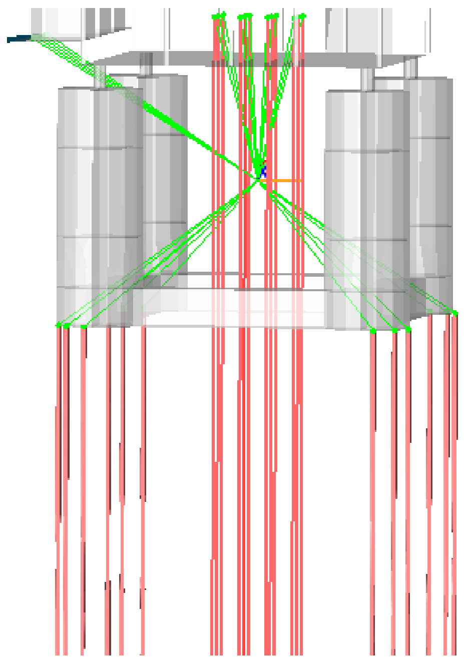
DeepC is an interactive program used to model floating configurations attached to the seabed with mooring lines, tension legs, risers etc. It employs the Marintek developed programs Simo and Riflex to perform the non-linear time domain finite element simulations. The results of the analysis can be viewed within the DeepC environment.



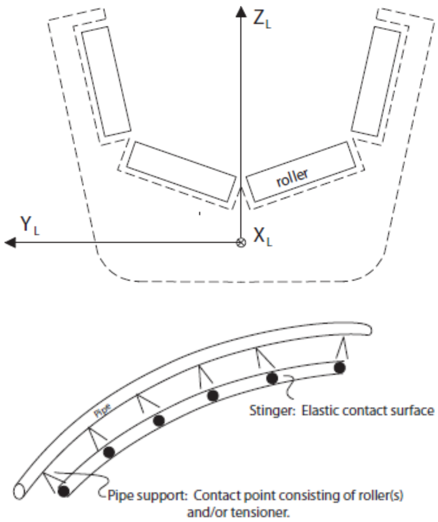
FEATURES OF DEEPC

FEATURE	DESCRIPTION
Unit support  	The user may mix units throughout the modelling. The data logging (scripting) ensures that re-creating the model gives the same result.
Locations and environments  	Environment modelling includes: • Current and wind profiles • Wave and wind spectra • Seafloor properties (stiffness, friction) • Locations (site specific data such as water depth, gravity, water density, seabed etc.) • Regular time conditions including current • Irregular time conditions including wind data, wind generated wave data, swell and current • Scatter diagrams and discretization

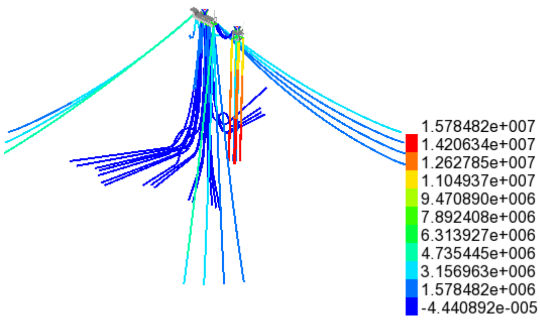
Floaters definition

 	

 	
	Any number of vessels may be modelled and employed in the analysis. Main parameters include:
	• Hydrodynamic coefficients read in from HydroD
	  ○ Added mass and potential damping
	  ○ 1st and 2nd order wave excitation
	  ○ RAOs
	  ○ Etc.
	• Wind and current coefficients
	• Vessel damping coefficients
	• Additional viscous force elements used to capture the Morison forces on the beam structure
	• Specified point forces
	DeepC automatically calculates retardation functions from added mass and damping coefficients.

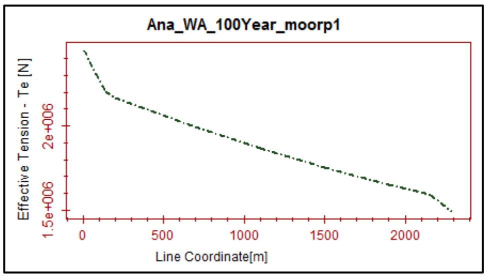
Slender structure properties  	Structure properties describe common attributes on geometrical entities. Sections: Different kinds of cross sections for segments and stress joints (pipe, axi-symmetric etc.) Load Interfaces: Contains Morison coefficients, segment external wrapping and segment marine growth properties. Materials: Contains material definitions. Mesh Densities: Contains mesh density properties for segments and stress joints. Slender Components: Internal fluid, buoy common properties, rotation hinge properties, stress concentration factors, SN-curves and section stress parameters. Combined Loading: Contains material, pipe cross section and fluid properties for combined loading analyses only. Fatigue: Contains SN-curve, stress concentration factor and cross sectional properties for fatigue analyses only.
Slender structures 	Various slender structures, including mooring and riser, tendon, jumper, etc. can be modelled by: • Segmented lines (any number of segments) • Stress joints (lines with piecewise linearly varying tubular cross section) • Ball joints, buoys, flex joints

<table>
<tr>
<td>Pipe-in-pipe contact
</td>
<td>Pipe-in-pipe is used to model contact between inner and outer pipes.

A pipe-in-pipe pair consists of a master pipe and a slave pipe.
The master pipe will automatically be equipped with tubular contact components at all FE nodes belonging to the pipe.
The slave pipe will experience contact with the master pipe as discrete element loads.
</td>
</tr>
<tr>
<td>Roller and tensioner contact

</td>
<td>Roller and tensioners are used when modelling the stinger and pipe contact during pipe-laying analysis.

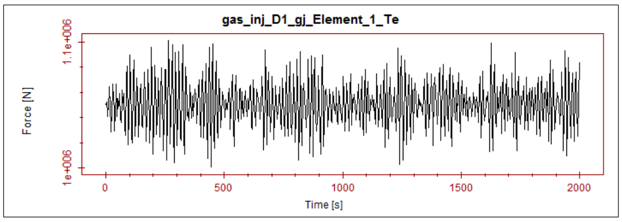
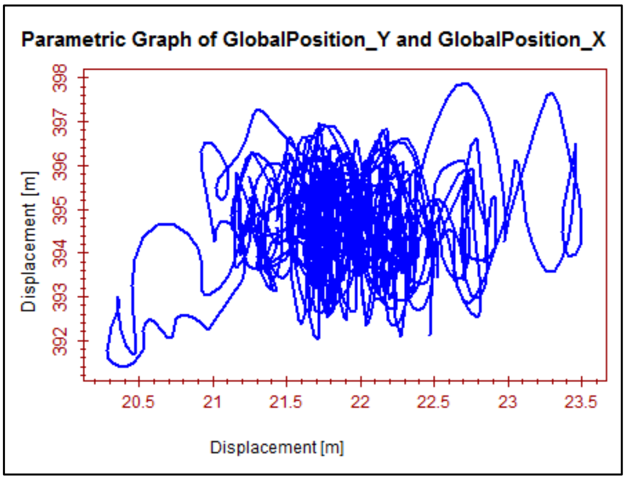
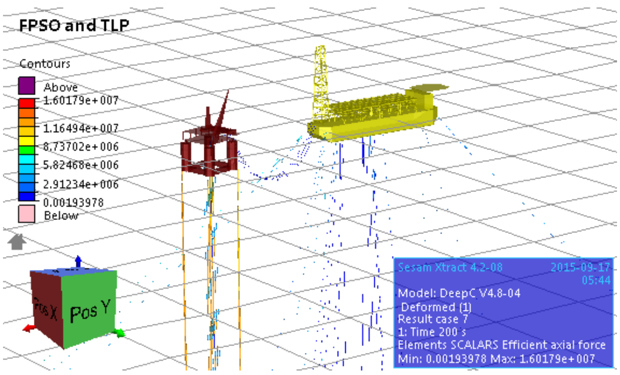
The rollers can be also used in other elastic contact surface modelling such as middle water tether arc.

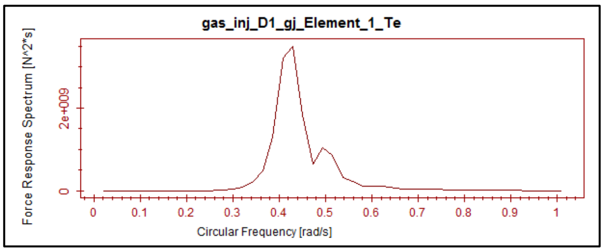
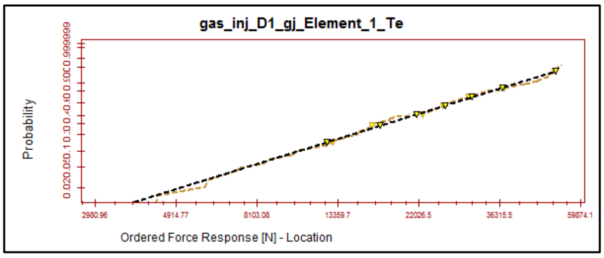
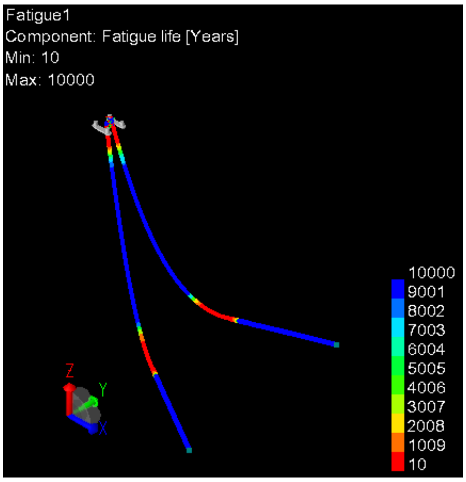
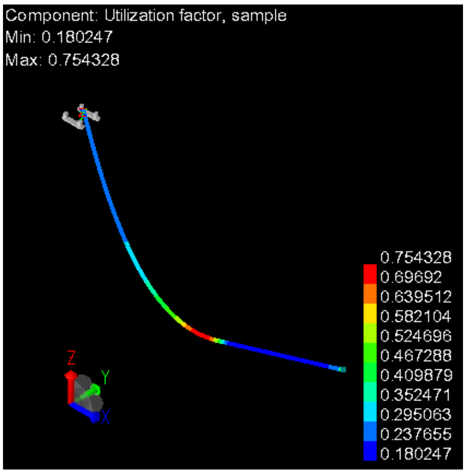
Contact between roller and pipe is modelled by a bi-linear or non-linear spring and a bi-linear dashpot damper. The contact force acts normal to the pipe and the roller. It is treated as a discrete element load acting on the pipe, while the contact load acting on the roller is transferred as a nodal force to the stinger. The last includes possible torsional moment.</td>
</tr>
<tr>
<td>Static analysis results



</td>
<td>All the load steps of the static analysis can be viewed as contour plots on the deformed structure in the 3D.

In addition, one can view 2D graphs of the following results:

Axial force and elongation
Bending moment My and Mz
Shear force Sy and Sz
Torsional moment and deformation
Curvature Cy and Cz
X-, Y- and Z-displacements
</td>
</tr>
</table>

Dynamic analysis results   	Users can plot and export time-series results of different forces and displacement, including: • Vessel 6-DOF motions • Wave forces • Effective tension • Bending moment about local y- and z-axes • Resulting bending moment • Decomposed line end forces in global X-, Y- and Z-directions • Nodal displacement in global X-, Y- and Z-directions • Element curvature • Etc.
Animation  	3D animations of vessels and lines displacements with contour colour of forces or bending moments on the lines. These are displayed in Xtract.

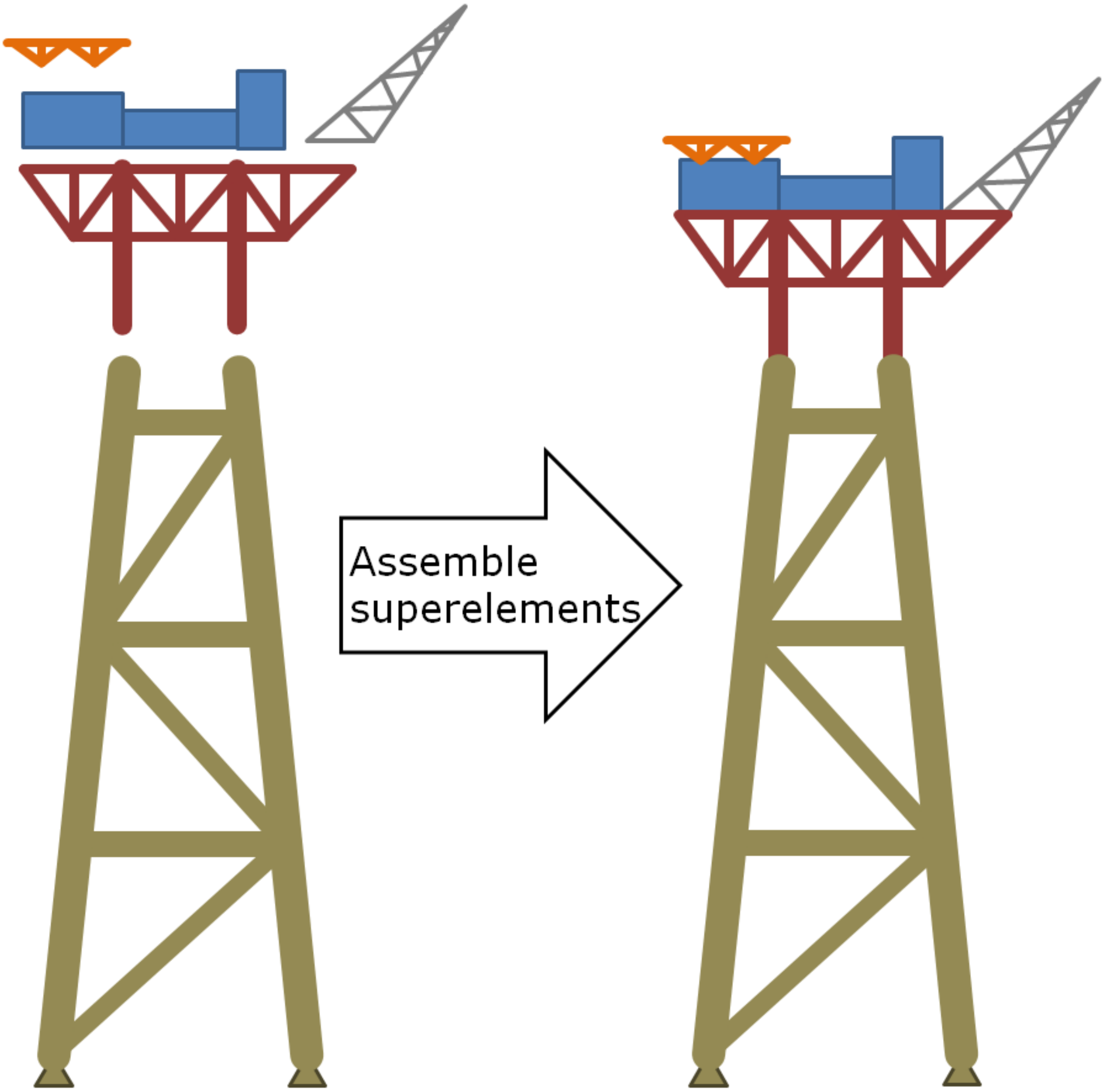
Post-processing  	DeepC also provides in-built post-processing calculation for all kinds of time series including: • Energy spectra • Cumulative distributions • Cumulative distribution function on Weibull paper • High and low frequency filtering • Range curves of effective tension showing max/min/mean and standard deviation along the lines
Fatigue evaluation 	For both regular and irregular time condition analyses DeepC can perform fatigue calculations by rainflow counting and Miner-Palmgren summation. Fatigue results include 3D contour plots of fatigue life and 2D graphs of fatigue life along the lines.
Code check for metallic risers 	Based on regular or irregular time domain riser analysis combined loading capacity analysis may be performed. Available codes are DNV OS-F201 and ISO 13628-7. Combined loading results include 3D contour plots and 2D graphs of utilization factor along riser.

Presel

PREPROCESSOR FOR ASSEMBLING SUPERELEMENTS

Last revised: August 22, 2017. Describing version 7.5-01.

Presel is Sesam's tool for assembling superelements to form the complete model. A superelement is basically a finite element (FE) model of a part of the complete structure. The Sesam preprocessors GeniE and Patran-Pre are used for creating first level superelements, i.e. part models comprised of finite elements like beams, shells, etc. Presel puts these parts together to form superelement assemblies. First level superelements are typically assembled to form second level superelement assemblies, these are in turn assembled to form third level superelement assemblies, and so on until the complete model — the top level superelement assembly — is formed.

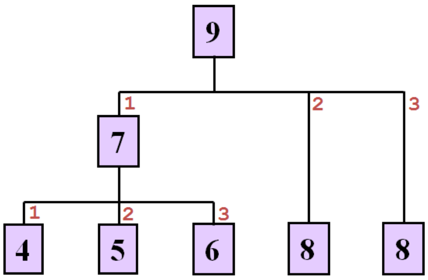
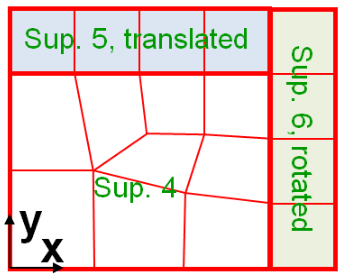
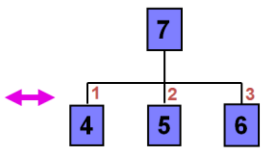
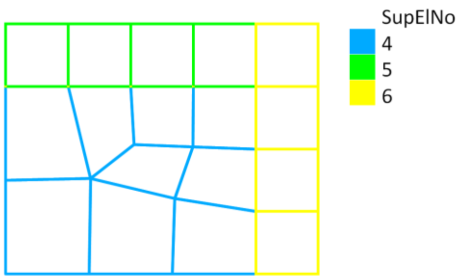


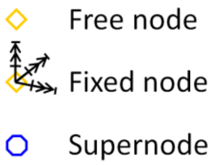
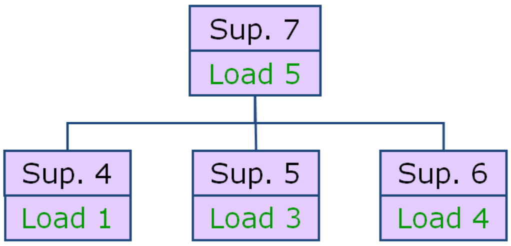
FEATURES OF PRESEL

Assembling a superelement consists of two steps:

- Assemble superelements (assemble geometry)
- Assemble or combine loads

These two steps are repeated for all higher level (from 2 and up) superelements created in Presel.

<table>
<tr><th>FEATURE</th><th>DESCRIPTION</th></tr>
<tr><td>Assembling superelements

</td><td>All 1st level superelements (created by GeniE and Patran-Pre) may be assembled into a 2nd level superelement.
Or they may be assembled through any number of levels to form the complete model.
Any superelement at any level may be repeatedly included in an assembly. This may be done for geometrically identical parts.</td></tr>
<tr><td>Include by transformation

</td><td>When including a superelement into an assembly it may be:
• Translated
• Rotated
• Mirrored
• Fitted by referring to two sets of three nodes to be matched</td></tr>
<tr><td>Print superelement hierarchy
<pre>SUPER ELEMENT LEVEL
       2   1
-----1:7.1-----------1:4.1
                 !
                 !-----2:5.1
                 !
                 !-----3:6.2</pre>
</td><td>Assembling superelements to form the complete model results in a superelement hierarchy. This may be printed for verification.</td></tr>
<tr><td>Display of superelement

</td><td>Any superelement at any level from 1st level superelements created by GeniE and Patran-Pre to the top level superelement, the complete model, may be displayed with colours distinguishing the individual superelements.</td></tr>
</table>

<table>
<tr><td>Boundary conditions

</td><td>Boundary conditions may be added to higher level superelements (from 2nd level and up). 1st level superelements may not be modified in Presel:
Fixed
Prescribed
Super</td></tr>
<tr><td>Load combination

</td><td>The load combination may be done in two ways:
Manually by specifying all loads of included superelements contributing to the combination
A group of loads combined one-to-one into a group of load combinations. This is useful for loads created by Wajac and Wadam.</td></tr>
<tr><td>Loads</td><td>Nodal loads may be added to any higher level superelement.</td></tr>
<tr><td>Sets</td><td>Sets of nodes may be defined. These are available in the postprocessors (Framework, Stofat, and Xtract).</td></tr>
<tr><td>Node triplet
◇ 5.1.7
Superelement 5
Index 1 (1st time used)
Node 7</td><td>Nodes are referred to by so-called triplets, i.e. three numbers being the superelement number, the superelement index and the node number. The superelement index is a number starting at 1 and incremented each time the same superelement is included.</td></tr>
<tr><td>Label</td><td>Node symbols and node numbers may be added to the display:
Boundary conditions
Coupled nodes (where superelements join)
Non-coupled nodes
Node symbols (yellow diamond for free, blue octagon for super)
Node numbers
Origin symbol</td></tr>
</table>

Linear dependency	Linear dependencies by making one or more degrees of freedom (dofs) linearly dependent on one or more other (independent) dofs.
Plot CGM →	The display may be sent to a file in alternative graphics formats: • CGM-Binary (Computer Graphics Metafile which may be imported into MS Office) • Postscript Or the display may be sent directly to an on-line printer.

Submod

DISPLACEMENTS FROM GLOBAL MODEL TO SUB-MODEL

Last revised: April 23, 2018. Describing version 3.2-02.

The sub-modelling technique allows a part of a (global) model to be re-analysed to produce more accurate results locally. The procedure is:

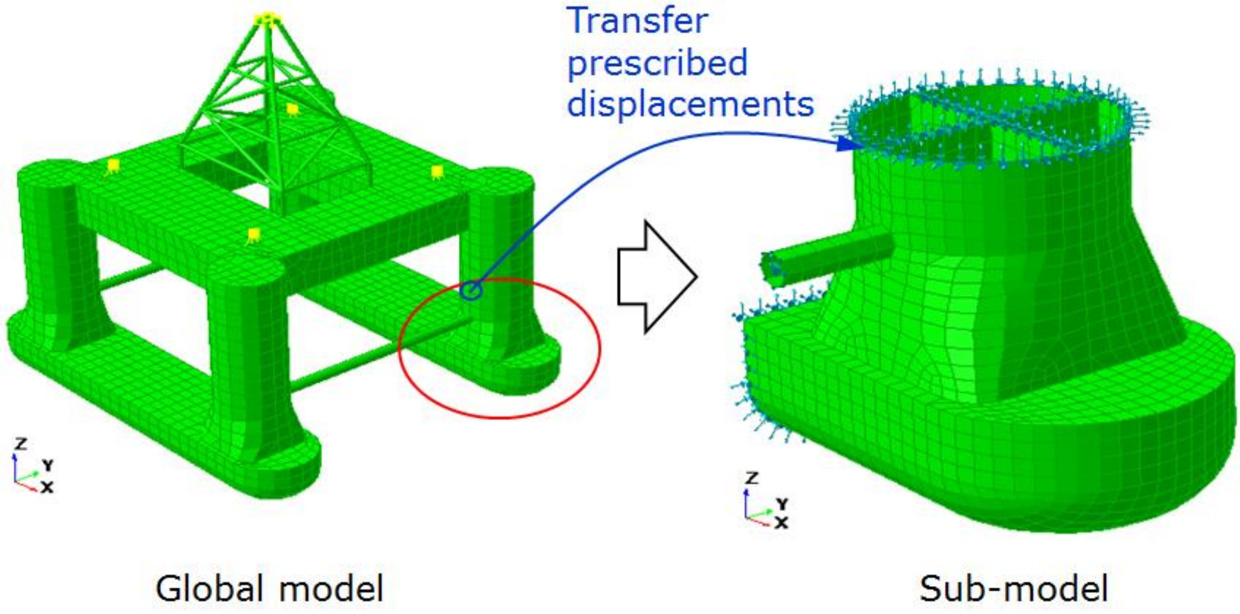
- Perform a (global) analysis of the structure.
- Create a sub-model with refined mesh of a region of interest.
- Fetch displacements from the global model and transfer these to the boundary of the sub-model. This is the task of Submod.
- Analyse the sub-model using the interpolated displacements as prescribed (forced) displacements and obtain a more accurate solution for the region.

To take advantage of the sub-modelling technique no preparations are required in the global analysis. Doing a sub-modelling analysis is an option after having performed an analysis of the whole structure. If the global results reveal a need for detailed analyses in certain areas, then the sub-modelling technique is an appealing alternative to a new global analysis.

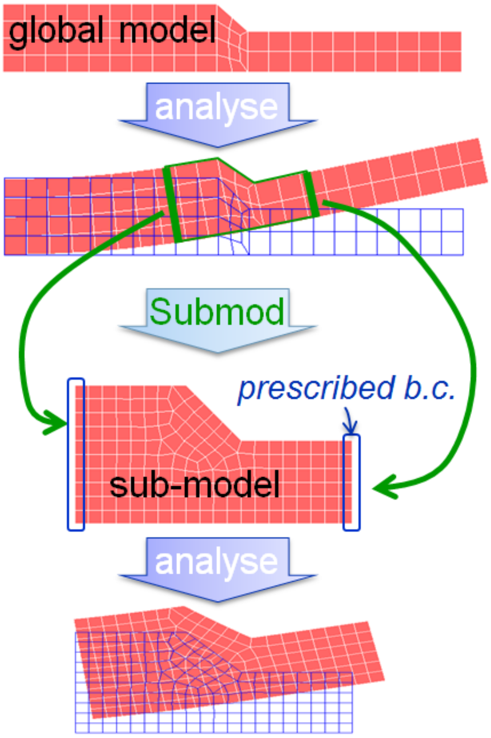
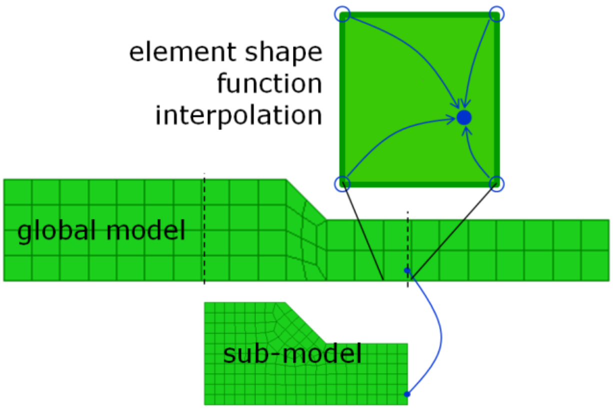
Minor changes to the geometry of the sub-model may be done to study the effect of alternative designs or to model more accurately details that were neglected in the global analysis. This can only be done provided the changes within the sub-model region have negligible effects on the global solution.

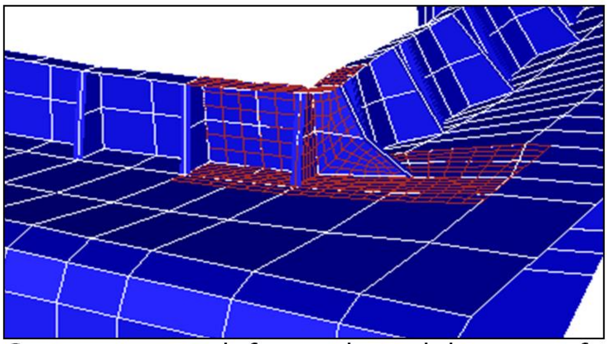
Both the global model and sub-model may be superelement models.

For a plate/shell fatigue analysis a sub-modelling analysis is close to mandatory as the requirement to mesh density in a fatigue analysis cannot normally be met in a global analysis.



FEATURES OF SUBMOD

FEATURE	DESCRIPTION
Fetch and transfer displacements  	The sub-model must have prescribed type of boundary condition at its edges where it is cut out of the global model. Submod compares geometrically the global model and sub-model to determine the position of prescribed nodes within the elements of the global model.
Node-to-element match  	Submod fetches displacements by interpolation inside the elements using the shape functions of the elements: • Any plate/shell element (3, 4, 6 and 8 node) ○ Interpolation may be in the thickness direction • Any solid element • 2 and 3 node beam elements but only along neutral axis The global model and sub-model may have different element types.
Node-to-node match	Alternatively to interpolating within elements, the displacements may be fetched from the nearest node. This is normally not the preferred option.

Verification of results  Superimpose deformed models to verify transfer of displacements	Submod's transfer of displacements may be verified using Xtract.

Wadam

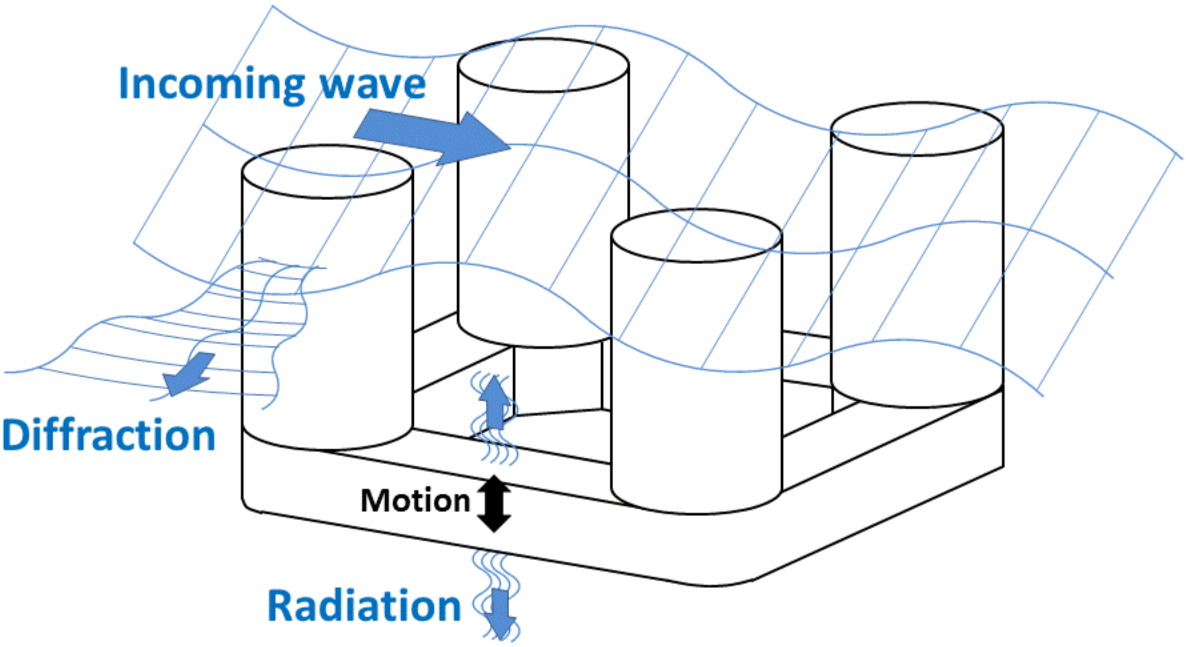
WAVE ANALYSIS BY DIFFRACTION AND MORISON THEORY

Last revised: January 9, 2019. Describing version 9.5-03 (64-bit).

Wadam is an analysis program for calculation of wave-structure interaction for fixed and floating structures of arbitrary shape, e.g. semi-submersible platforms, tension-leg platforms, gravity-base structures and ship hulls. Wadam performs hydrodynamic analysis in the frequency domain. The program is based on the radiation diffraction methodology for large volume structures. For slender structures the Morison formulation is used to account for the drag effects of such structures. Wadam can handle a very small forward speed/current. Wadam uses input data generated by HydroD.

The Wadam results are presented directly as complex transfer functions. The loads on the finite element model can also be given in this way, but they may alternatively be converted to deterministic results for a specified sequence of phase angles of the incident wave. For fixed structures Morison's equation may also be used with deterministic output option to calculate drag forces due to time independent current.

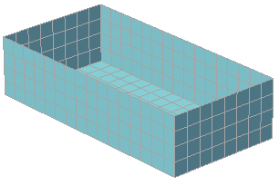
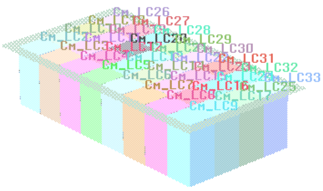
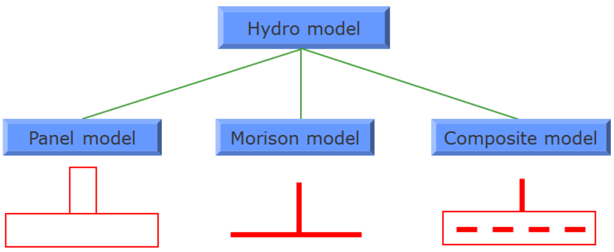
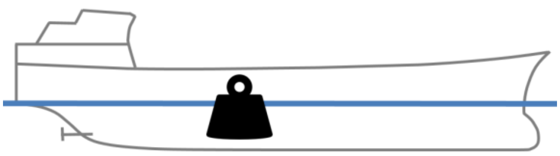
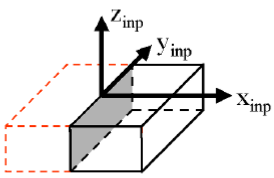
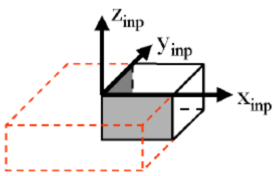
The same analysis model may be applied to both the calculation of global responses in Wadam and the subsequent structural analysis. For shell and solid element models Wadam also provides automatic mapping of pressure loads from a panel model to a differently meshed structural finite element model.

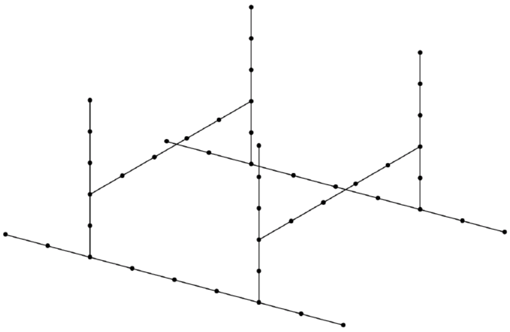
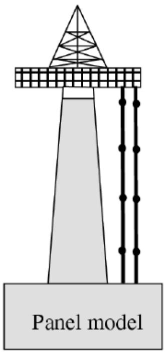


FEATURES OF WADAM

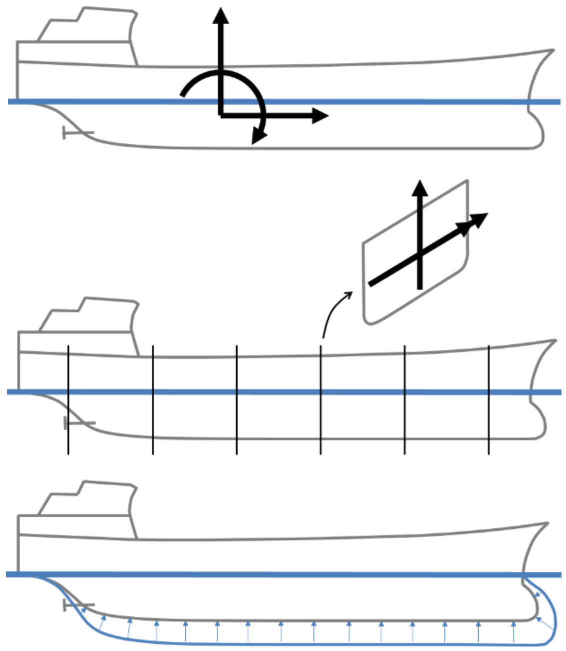
The features of Wadam are summarised below.

Model types

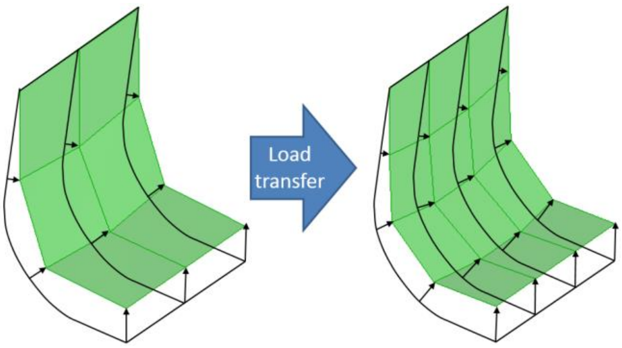
FEATURE	DESCRIPTION
Model types  	There are three main model types: • Hydro model used to calculate hydrodynamic forces • Structural model onto which hydrodynamic and hydrostatic loads are transferred • Mass model for floating structures being either a model or a mass matrix
Hydro models 	The hydro model may be: • Panel model for potential theory computations • Morison model for computation by the Morison equation • Composite model being a combination of a panel and a Morison model for use when potential theory and the Morison equation are applied for different parts of the hydro model
Mass model 	The mass can be provided in any of the two forms • Global mass data • A file describing the mass distribution A description of the mass distribution is needed for computation of sectional loads.
Panel model  y-z plane of symmetry  x-z and y-z plane of symmetry	The panel model may be a single superelement or a hierarchy of superelements. It may describe either the entire wet surface or take advantage of one or two planes of symmetry.

Morison model 	Morison elements: • 2D Morison elements for loads on 2 node beam elements • 3D Morison elements for loads in three directions in nodes • Pressure area elements for loads at the ends of 2D Morison elements • Dry Morison elements for beams on which no load is to be computed • Anchor and TLP elements for restoring contributions in nodes. These elements can be used to provide a simplified model of the moorings and risers. The element is a linear spring with pretension.
Composite model 	Hydrodynamic forces on a composite model are computed from potential theory for the panel model and from the Morison equation for the Morison model. The hydrodynamic exciting forces and matrices from both theories are accumulated in the equations of motion for the composite model.
Multi-body 	Hydrodynamic and mechanical interaction between several structures can be analysed. The hydrodynamic interaction is computed from the potential theory as applied for a single structure with the extension that the number of degrees of freedom is increased from 6 to 6N where N is the number of structures. A multi-body model can contain both fixed and floating bodies.

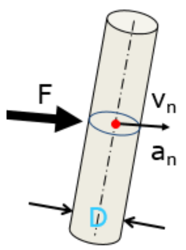
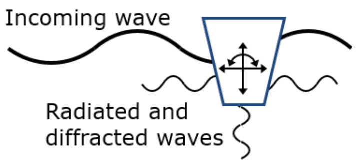
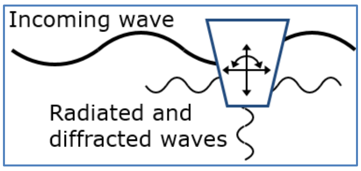
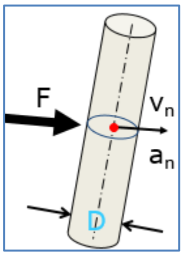
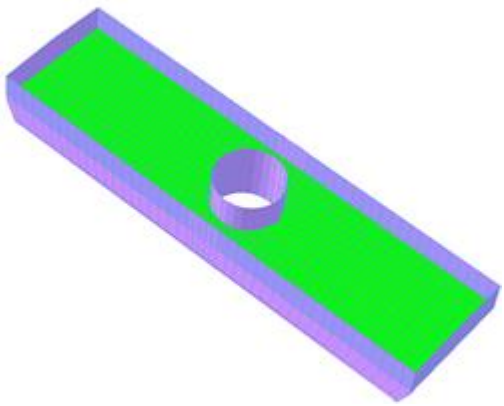
Analyses

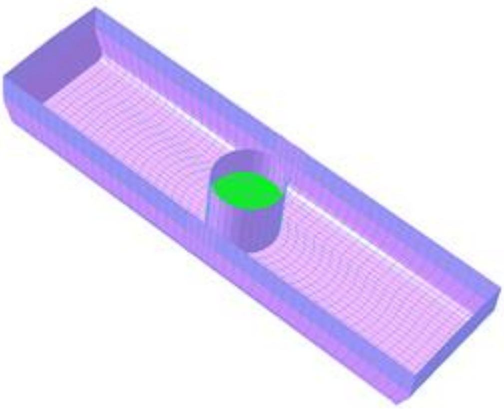
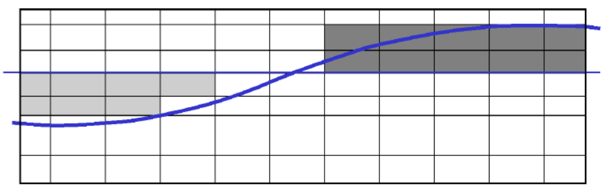
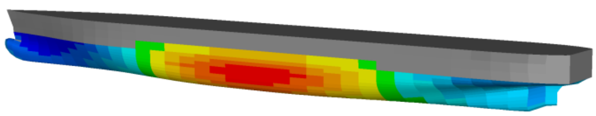
FEATURE	DESCRIPTION
Forward speed/Wave current interaction	The effect of a small current or forward speed can be accounted for. The speed limit is case dependent, but is typically in the range 1-2 m/s. The current can be in any direction.
Hydrostatic	Calculation of hydrostatic data and inertia properties
Global response 	Global response is calculated including: • First order wave excitation forces and moments • Second order wave excitation forces and moments (used to model springing effects, low frequency forces etc.) • Hydrodynamic added mass and damping • First and second order rigid body motions • Sectional forces and moments • Steady drift forces and moments • Wave drift damping coefficients • Sectional load components (mass, added mass, damping and excitation forces) • Panel pressures • Fluid particle kinematics (for gap calculations and free surface animation) • Second order free surface elevations

Transfer of load to structural analysis

FEATURE	DESCRIPTION
Load transfer  Panel mesh FE mesh	Automatic load transfer to a finite element model for subsequent structural analysis including: • Inertia loads • Line loads on beam elements from Morison model • Point loads from pressure areas, anchor elements etc. from Morison model • Pressure loads on plate/shell/solid elements • Internal tank pressure in compartments
Load transfer in frequency domain	The computed load transfer functions are mapped onto a finite element model. For the shell/solid part the mesh need not be the same as the mesh used in the hydrodynamic analysis (the panel model). For the beams the model must be identical in the hydrodynamic and structural analysis.
Deterministic load transfer	The loads may alternatively be extracted as real load cases at specified phase angles during the wave cycle. In this case a wave amplitude must also be specified and the load transfer function is multiplied with this wave amplitude. The loads will be used in a quasi-static structural analysis.

Theory and formulation

FEATURE	DESCRIPTION
Morison equation  $F_{Inertia} = \rho \pi D^2/4 \, C_m \, a_n$ $F_{Drag} = \rho \, D/2 \, C_d \, v_n \, \lvert v_n \rvert$	The Morison equation is used for slender (beam) structures.
Potential theory  $\nabla^2 \varphi(x, y, z, t) = 0$	First and second order 3D potential theory is used for large volume structures. The solution is based on using the Green's function formulation.
The Morison equation and potential theory combined  + 	The forces from Morison equation and potential theory are added when the structure comprises of both slender and large volume parts.
Removal of irregular frequencies 	Irregular frequencies may be removed from the radiation-diffraction solution. This method is based on a modified integral equation obtained by including a panel model of the internal water plane.

Damping lids 	The wave elevation in areas where resonance may happen can be controlled by including a damping lid on the free surface in those areas.
Tank pressures	Tank pressures may be computed • Quasi-statically • Dynamically
Roll damping	Viscous roll damping included in different ways: • By using the roll damping models of Tanaka and Kato • By using the ITTC roll damping model • By prescribing a linear and a quadratic damping coefficient • By using a Morison model
Additional damping and restoring matrices	Additional damping and restoring matrices can be specified. This can be used for including damping or restoring from moorings and risers and/or viscous damping
Pressure loads up to free surface 	Panel pressures calculated by first order potential theory may be extrapolate to the free surface. Correspondingly, dry elements below the still water level receive no loads. This is done by stretching pressures above the still water level. This option is only for deterministic load transfer.
Reduced pressure around the free surface 	This is a recommended option for fatigue analysis and follows DNV GL Class not 30.7. The pressure is modified around the waterline. The method should only be applied on that part of the vessel which has vertical intersection with the free surface.

Wasim

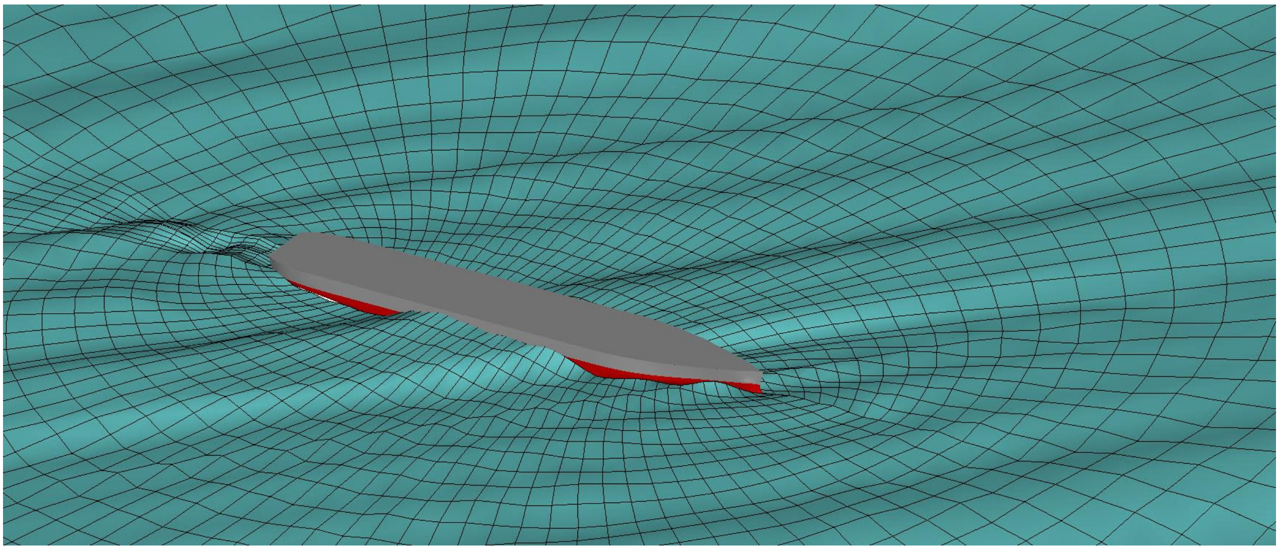
LINEAR AND NON-LINEAR SEA-KEEPING AND WAVE LOADS ON VESSELS WITH FORWARD SPEED

Last revised: January 9, 2019. Describing version 5.5-09 (64-bit).

Wasim is an analysis program for calculation of wave-structure interaction for fixed and floating structures. Wasim performs hydrodynamic analysis in time domain, but also includes an option for conversion to frequency domain. The program is based on the radiation diffraction methodology for large volume structures. For slender structures the Morison formulation is used to account for the drag effects of such structures. Wasim can handle any forward speed from zero to very high as long as the vessel is not planing. Wasim uses input data generated by HydroD. The forward speed problem may alternatively be regarded as a wave/current interaction problem with the forward speed replaced by a uniform current in the opposite direction.

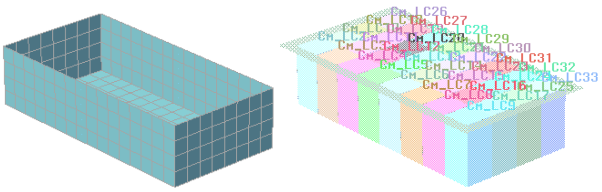
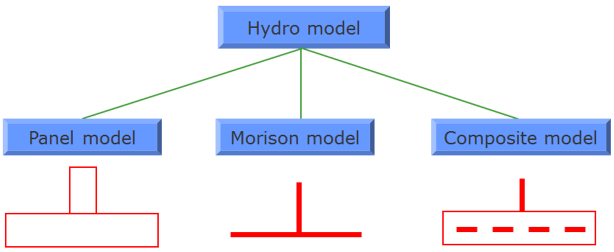
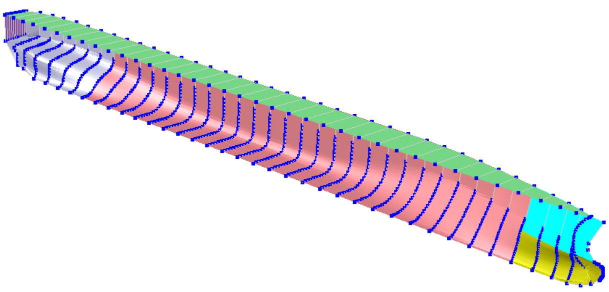
The same analysis model may be applied to both the calculation of global responses in Wasim and the subsequent structural analysis. For shell and solid element models Wasim also provides automatic mapping of pressure loads from a panel model to a differently meshed structural finite element model.

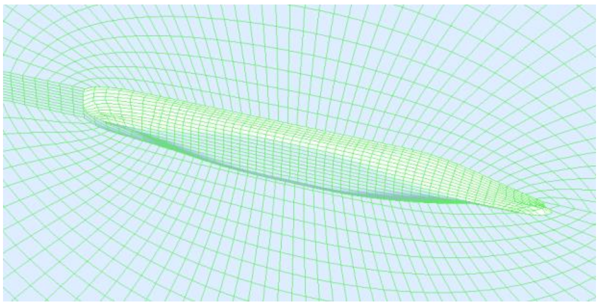
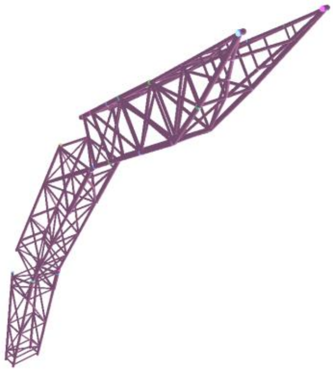
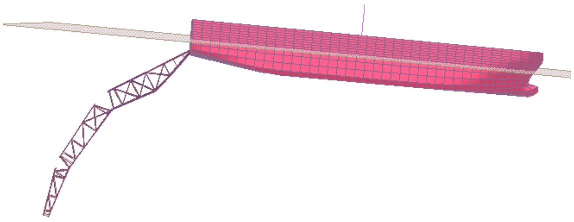
The 3D potential theory in Wasim is based on a Rankine formulation. This means that both the hull and free surface must be meshed. HydroD has tools for meshing the free surface.



FEATURES OF WASIM

Model types

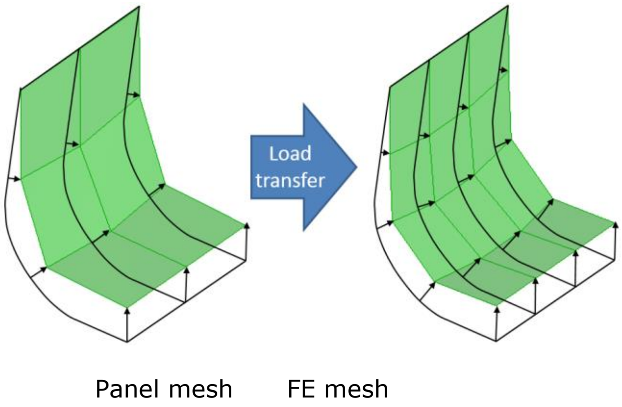
FEATURE	DESCRIPTION
Model types 	There are three main model types: • Hydro model used to calculate hydrodynamic forces • Structural model onto which hydrodynamic and hydrostatic loads are transferred • Mass model for floating structures, the mass model is either a model or a mass matrix
Hydro models 	The hydro model may be: • Panel model for potential theory computations • Morison model for computation by the Morison equation • Composite model being a combination of a panel and a Morison model for use when potential theory and the Morison equation are applied for different parts of the hydro model
Mass model	The mass can be provided in any of the two forms: • Global mass data • A file describing the mass distribution A description of the mass distribution is needed for computation of sectional loads.
Section model 	The section model is a set of curves describing the vessel geometry. From the section model HydroD will make a mesh of both the hull and the free surface. These meshes are then used by the Wasim solver. The section model can be the complete model or it may use symmetry about the xz-plane.

Panel model 	The Rankine solver requires a mesh on both hull and free surface. Both meshes are created by HydroD from the section model. The free surface mesh can be automatically or interactively created.
Morison model 	Morison elements: • 2D Morison elements for loads on 2 node beam elements • Pressure area elements for loads at the ends of 2D Morison elements • Dry Morison elements for beams on which no load is to be computed • Anchor and TLP elements for restoring contributions in nodes. These elements can be used to provide a simplified model of the moorings and risers. The element is a linear spring with pretension.
Composite model 	Hydrodynamic forces on a composite model are computed from potential theory for the panel model and from the Morison equation for the Morison model. The hydrodynamic excitation forces and matrices from both theories are accumulated in the equations of motion for the composite model.

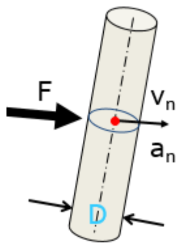
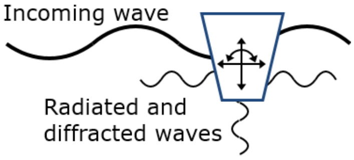
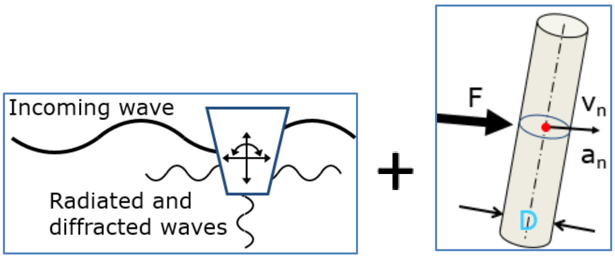
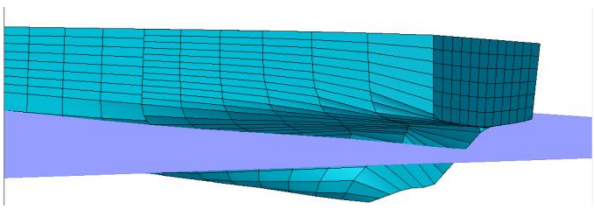
Analyses

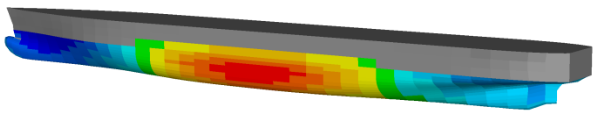
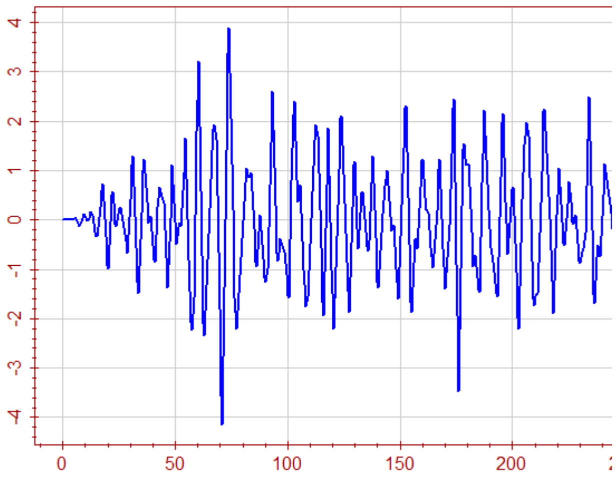
FEATURE	DESCRIPTION
Forward speed/Wave current interaction	Wasim can handle any value of forward speed/current as long as the vessel is not planing. The current must be in the positive or negative x-direction.
Calm sea analysis	Computation of the static loads on the hull. If there is a forward speed the loads due to that will be included.
Global response	Global response is calculated including: • Wave excitation forces and moments • Hydrodynamic added mass and damping • Rigid body motions • Sectional forces and moments • Steady horizontal drift forces and moments. If there is a forward speed, the x-component of the drift force is the added resistance. • Panel pressures • Fluid particle kinematics and wave elevation (for gap calculations and free surface animation) • Relative motion

Transfer of load to structural analysis

FEATURE	DESCRIPTION
Load transfer 	Automatic load transfer to a finite element model for subsequent structural analysis including: • Inertia loads • Line loads on beam elements from Morison model • Point loads from pressure areas, anchor elements etc. from Morison model • Pressure loads on plate/shell/solid elements • Internal tank pressure in compartments
Load transfer in frequency domain	The time histories of the loads are converted into frequency domain by Fourier analysis and then applied to the finite element model to be used in a quasi-static or frequency domain structural analysis.
Load transfer in time domain	A selected part of the time histories is applied as time domain loads on the finite element model to be used in a time domain (dynamic) structural analysis.
Snapshot loads	Loads at selected points in time are applied as static loads on the finite element model to be used in a quasi-static structural analysis.

Theory and formulation

FEATURE	DESCRIPTION
Morison equation  $F_{Inertia} = \rho\ \pi\ D^2/4\ C_m\ a_n$ $F_{Drag} = \rho\ D/2\ C_d\ v_n\ \lvert v_n \rvert$	The Morison equation is used for slender (beam) structures.
Potential theory  $\nabla^2\varphi(x,y,z,t) = 0$	3D potential theory is used for large volume structures. The solution is based on using the Rankine formulation.
The Morison equation and potential theory combined 	The forces from Morison equation and potential theory are added when the structure comprises of both slender and large volume parts.
Non-linear effects 	The following non-linear effects can be included in the analysis: • Hydrostatic and Froude-Krylov pressure on exact wetted surface • Exact treatment of inertia and gravity • Quadratic terms in Bernoulli equation • Quadratic roll and pitch damping • Stokes 5^{th} order or Stream function wave
Tank pressures	Tank pressures may be computed using the quasi-static approximation.

Viscous damping	Viscous damping in roll and pitch can be modelled by prescribing a linear and a quadratic damping coefficient. Alternatively, the effect can be included by using a Morison model.
Additional damping and restoring matrices	Additional damping and restoring matrices can be specified. This can be used for including damping or restoring from moorings and risers and/or viscous damping.
Pressure loads up to free surface	In a non-linear analysis loads are computed on the exact wetted surface.
Reduced pressure around the free surface 	This is a recommended option for fatigue analysis and follows DNV GL Class note 30.7. The pressure is modified around the waterline. The method should only be applied on that part of the vessel which has vertical intersection with the free surface.
Time series input  	Wasim can read and utilize any combination of the following time series inputs: • Wave elevation at a prescribed point • Motion, velocity and acceleration of the structure • Forces (and optionally also moments) acting in specified points

Waveship

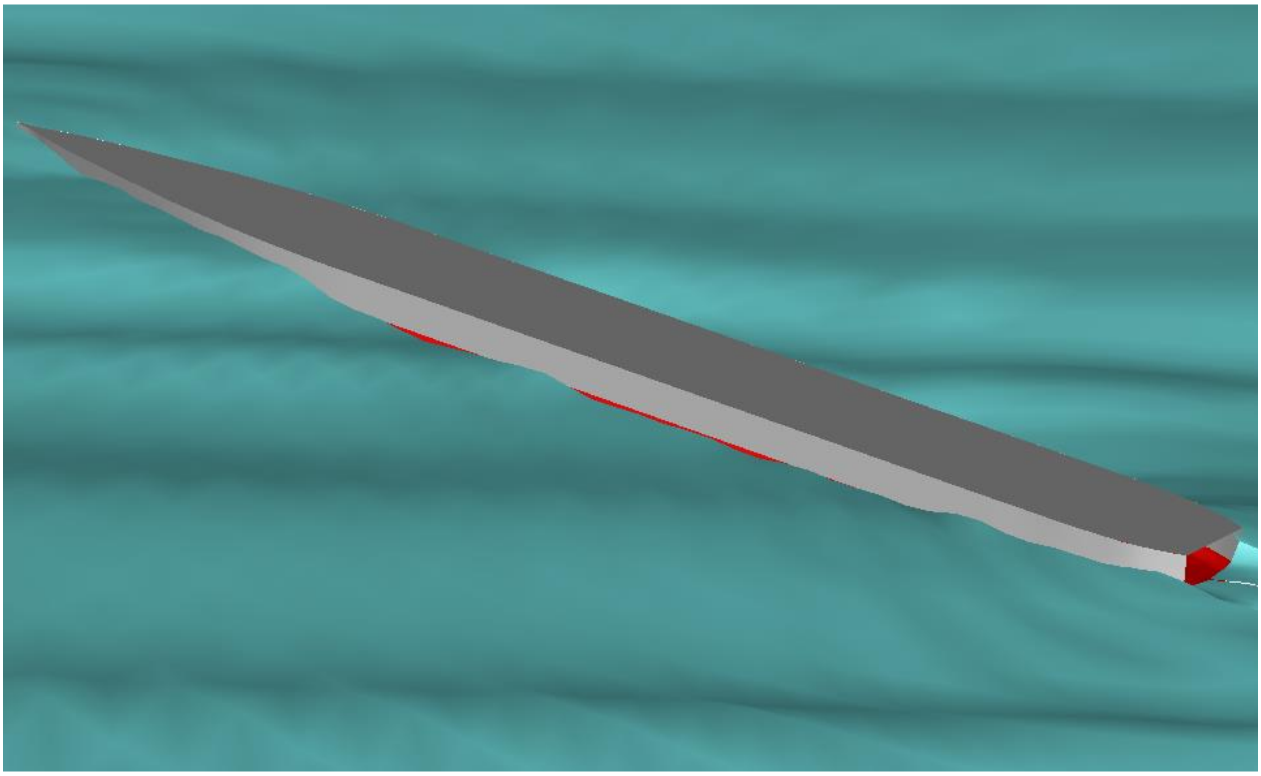
SEA-KEEPING OF SLENDER VESSELS

Last revised: August 22, 2017. Describing version 6.2-05.

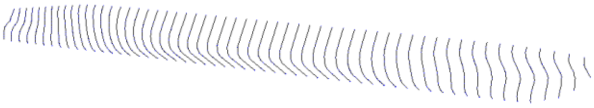
Waveship is a frequency domain hydrodynamic analysis program for calculation of wave loading and response for ships and other slender marine structures, with and without forward speed. The Waveship results are presented directly as complex transfer functions.

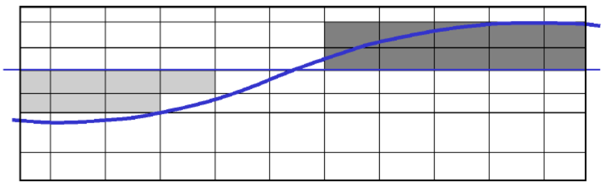
Waveship is using strip theory. This is in many cases a good approximation for obtaining global responses but is less reliable for local responses (i.e. pressure distribution).

The most typical approximation of Waveship is to compute global results on slender structures at zero to moderate speed. When applicable these results can be obtained with a very small computational effort.



FEATURES OF WAVESHIP

FEATURE	DESCRIPTION
Strip model 	The strip model is the basis for the hydrodynamic solver. A 2D problem is solved in the planes defined by each curve (yz-plane). The strip model covers one half of the vessel so only vessels with xz-symmetry can be analysed.
Mass model	The mass can be provided in any of the two forms: • Global mass data • Sectional mass matrices Sectional mass matrices are needed for computation of sectional loads.
Mooring elements	A simplified model of moorings and risers can be given in the form of linear springs with pretension.
Forward speed	Waveship can handle moderate forward speeds. The results are best at zero speed and becomes gradually less reliable with increasing speed.
Global response	Global response is calculated including: • Wave excitation forces and moments • Hydrodynamic added mass and damping • Rigid body motions • Mean drift force • Sectional forces and moments Mean drift force and sectional loads are less reliable than the global quantities since it requires computation of the pressure distribution.

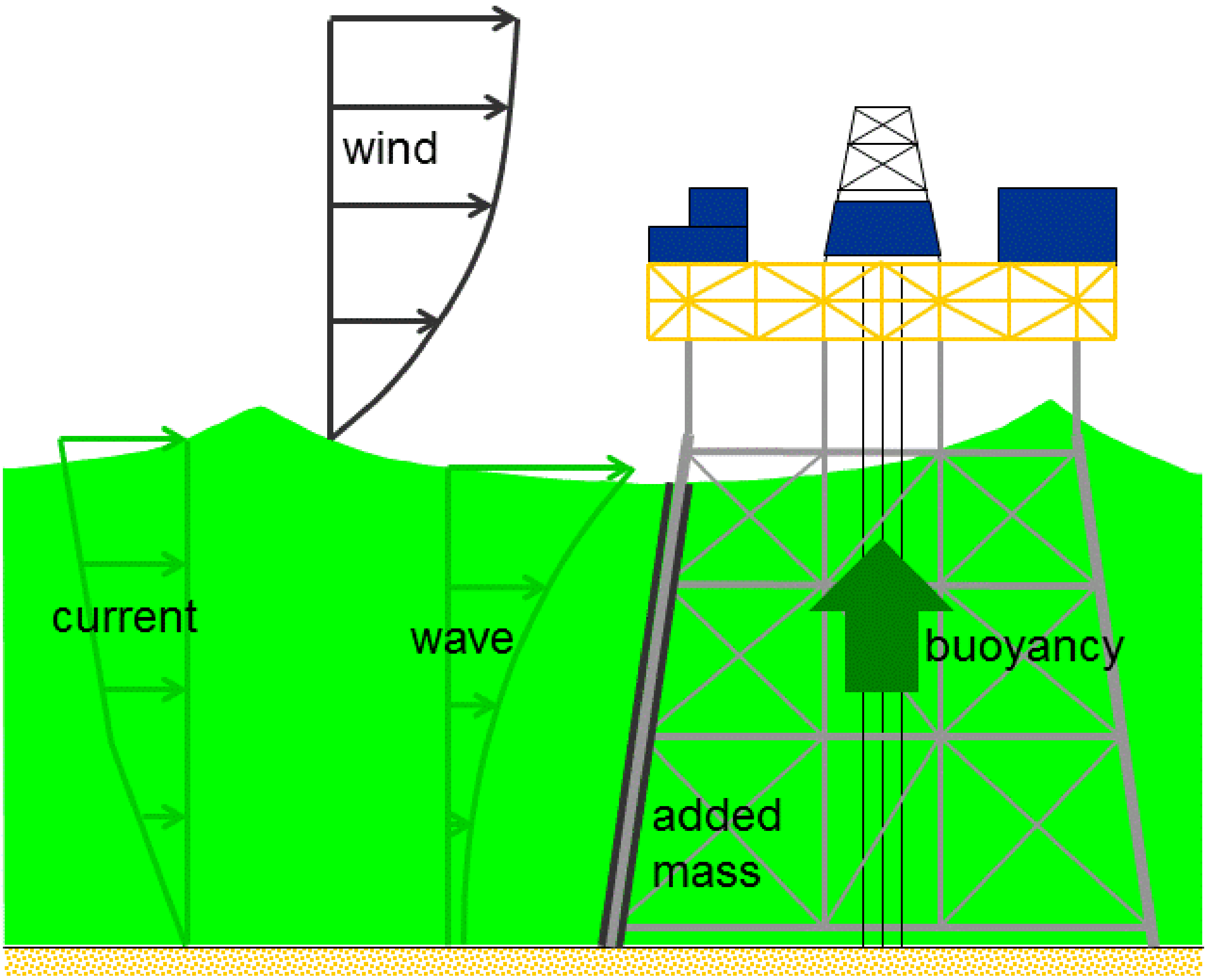
Load transfer	Automatic load transfer to a finite element model for subsequent structural analysis including: • Inertia loads • Pressure loads on plate/shell/solid elements The pressure loads may have significant uncertainty and should be used with care. Load transfer from Waveship is therefore in general only recommended for load transfer to the cargo in a transportation analysis, i.e. transfer only of inertia loads.
Tank pressures	Tank pressures may be computed using the quasi-static approximation.
Roll damping	Viscous roll damping from hull and bilge keel can be included by using the roll damping models of Tanaka and Kato.
Pressure loads up to free surface 	Panel pressures calculated by first order potential theory may be extrapolate to the free surface. Correspondingly, dry elements below the still water level receive no loads. This is done by stretching pressures above the still water level.

Wajac

WAVE AND CURRENT LOADS ON FIXED RIGID FRAME STRUCTURES

Last revised: February 1, 2019. Describing version 7.4.

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) in the time domain (deterministic), frequency domain or in a short-term time domain simulation. Loads are transferred to structural analysis in Sestra and statistical post-processing in Postresp.

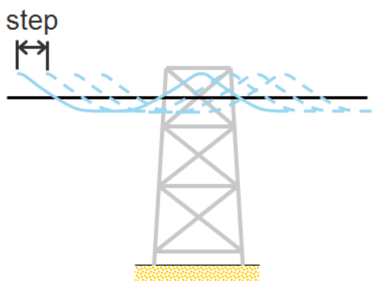
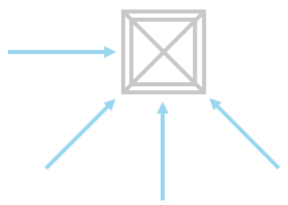
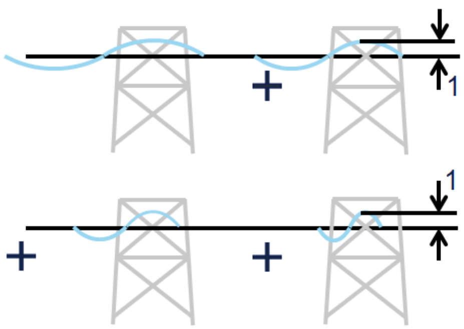
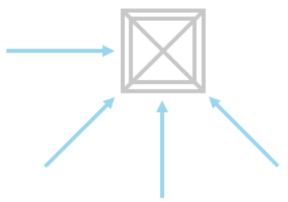


FEATURES OF WAJAC

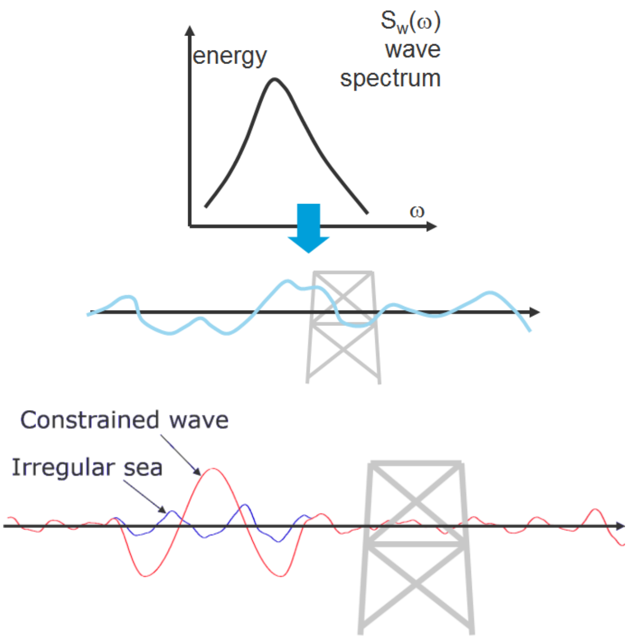
The features of Wajac are summarised below in sections:

- Types of Analysis
- Details on Certain Features

Types of analysis

FEATURE	DESCRIPTION
Deterministic load calculation  	Deterministic load calculation is performed in the time domain (wave stepping through structure) and is generally used for design purposes in an ultimate limit state analysis. Several wave theories are available including Stokes 5th, Stream Function and Cnoidal. Multiple water depths may be specified. Such an analysis is the basis for: • Static analysis in Sestra • Deterministic fatigue analysis (FLS) in Framework • Code checking (ULS) in GeniE Print of transfer functions for base shear and overturning moment allows graphing in Excel.
Spectral (frequency domain) load calculation  	This involves calculation of wave force transfer functions in the frequency domain. Such an analysis is the basis for: • Frequency domain dynamic or quasi-static analysis in Sestra • Stochastic (spectral) fatigue analysis (FLS) in Framework Print of transfer functions for base shear and overturning moment allows graphing in Excel.

<table>
<tr>
<td>
Time domain simulation

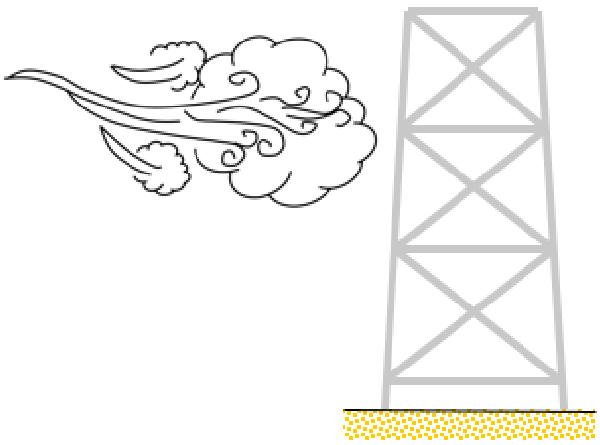


</td>
<td>
Loads are calculated for a random short-term (irregular) seastate. The seastate is generated based on a wave spectrum and a seed.

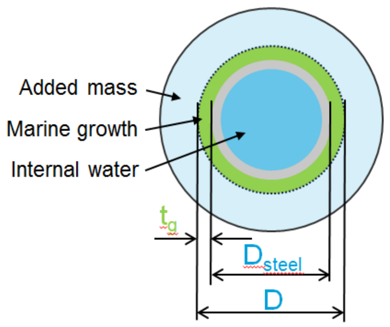
A constrained wave (a given wave profile) may optionally be embedded in an irregular seastate. This allows combining an irregular seastate with a large wave with e.g. a 100-year return period.

The time domain wave loads are used in:

- Time domain static or dynamic analysis in Sestra
- Time history fatigue analysis (FLS) in Fatigue Manager
- Code checking (ULS) in GeniE
</td>
</tr>
<tr>
<td>
Static wind loads


</td>
<td>
Static wind loads are computed for a stationary wind and are used in:

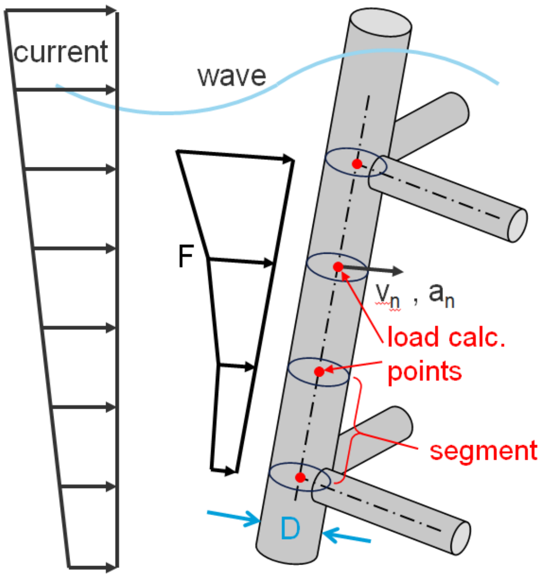
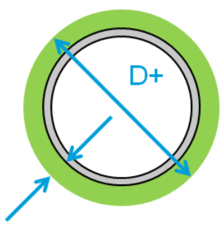
- Combination with wave loads in a deterministic load calculation
- Wind fatigue analysis in Framework
</td>
</tr>
<tr>
<td>
Added mass

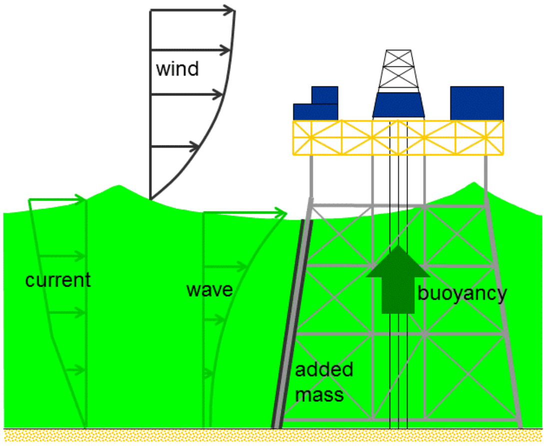


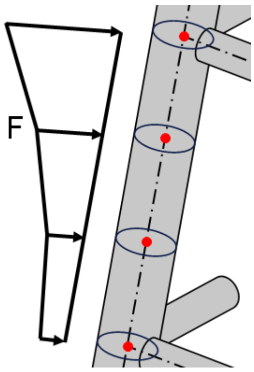
</td>
<td>
Added mass for beams is computed and used in:

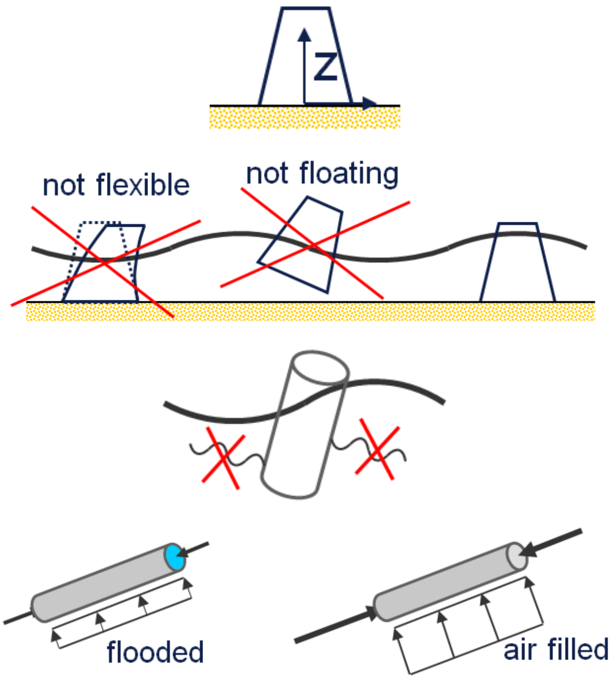
- Combination with deterministic, spectral and time domain analyses
- Eigenvalue (free vibration) analysis in Sestra
</td>
</tr>
</table>

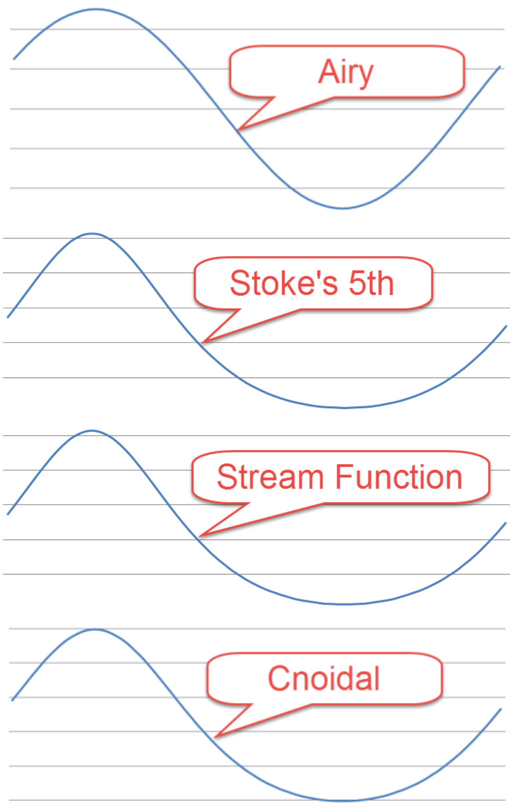
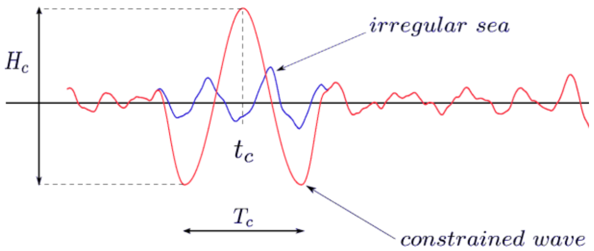
Details on certain features

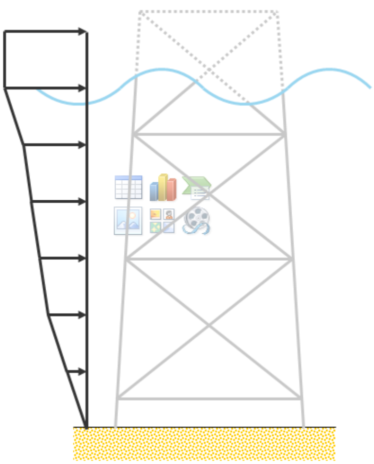
FEATURE	DESCRIPTION
Load calculation 	All loads calculated by Wajac are based on Morison's equation: $F = \rho \pi D^2/4\, C_m\, a_n + \rho\, D/2\, C_d\, v_n\, \|v_n\|$
Hydrodynamic coefficients $F_{Inertia} = \rho\, \pi\, D^2/4\, C_m\, a_n$ $F_{Drag} = \rho\, D/2\, C_d\, v_n\, \|v_n\|$	The hydrodynamic coefficients C_m (inertia) and C_d (drag) may be specified in alternative ways: • Constant ○ Different constant values to different parts of structure • Function of diameter • Function of roughness and Reynolds number • Function of roughness and Keulegan-Carpenter number • By API rule
Marine growth 	Marine growth may contribute to: • Drag force • Inertia force and added mass • Weight and buoyancy

Loads 	Wajac computes:
	• Wave loads
	  ○ Alternative wave theories, see below
	• Wind loads
	• Current loads
	• Buoyancy loads
	  ○ Rational and marine methods
	• Added mass

Load transfer 	Piecewise linear loads (two or more segments per beam) are stored on the Loads Interface File to be read by Sestra.

Fundamentals 	Fundamental assumptions in Wajac are:
	• Z-axis must point upwards
	• Origin preferably in centre of jacket
	• Only 2 node beams considered
	• Fixed and rigid structure
	• Hydrodynamic force:
	  ○ Morison equation for beams
	  ○ MacCamy-Fuchs for vertical tubes
	• Water particle motion undisturbed by presence of structure (diffraction from nearby tank may be accounted for)
	• Buoyancy included, with or without flooding, see more details in the GeniE feature description

Wave theories


	Wave theories included are:
	• Airy – linear harmonic theory
	    ○ Wave kinematics defined up to still water level
	    ○ Constant, extrapolation and Wheeler stretching to wave crest
	• Stokes 5th order – steep waves, deep waters
	    ○ Wave kinematics defined up to wave crest
	• Dean's Stream Function – numeric approximation of given wave
	• Cnoidal – shallow waters
	• NEWWAVE – theory introduced by Shell
	• Constrained wave embedded in an irregular sea

Current

	Current defined in Wajac:
	• Contributes to drag force for deterministic wave
	• Contributes to drag force for time domain simulation of short term sea state
	• Is used in equivalent linearization of drag force for spectral wave
	• Direction of current:
	    ○ X-, Y- and Z-component
	    ○ Horizontal and given direction
	    ○ Horizontal and parallel with wave

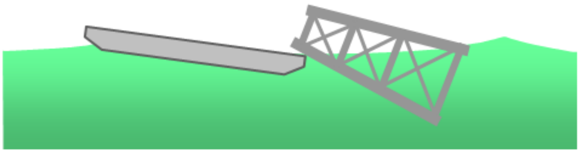
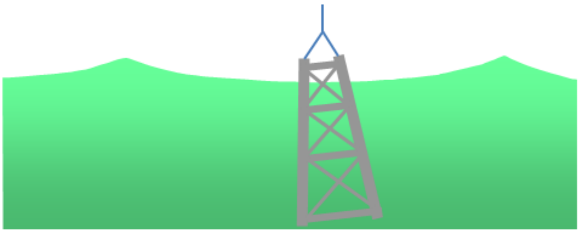
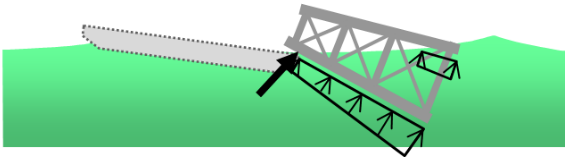
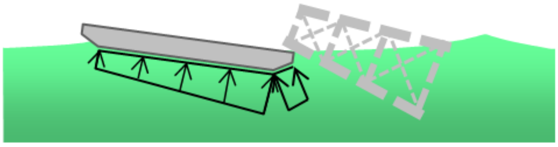
Installjac

LAUNCHING AND UPENDING ANALYSIS

Last revised: August 22, 2017. Describing version 8.2-00.

Installjac simulates the installation launch and upending of a jacket. The program provides a comprehensive assessment of the hydrostatic and hydrodynamic properties of the jacket during the installation simulations.

FEATURES OF INSTALLJAC

FEATURE	DESCRIPTION
Launching from barge 	The jacket launching from a single or multiple hinged barge may be simulated.
Free floating of the jacket and barge 	Free floating of the jacket and barge after launching may be simulated.
Upending of jacket 	The upending of the jacket may be done using cranes, hooks and member flooding.
Launch loads 	The member launch loads for a jacket stress analysis may be generated.
Hydrodynamic forces on barge 	The hydrodynamic and hydrostatic forces on a barge or multiple hinged barges may be modelled

Simo

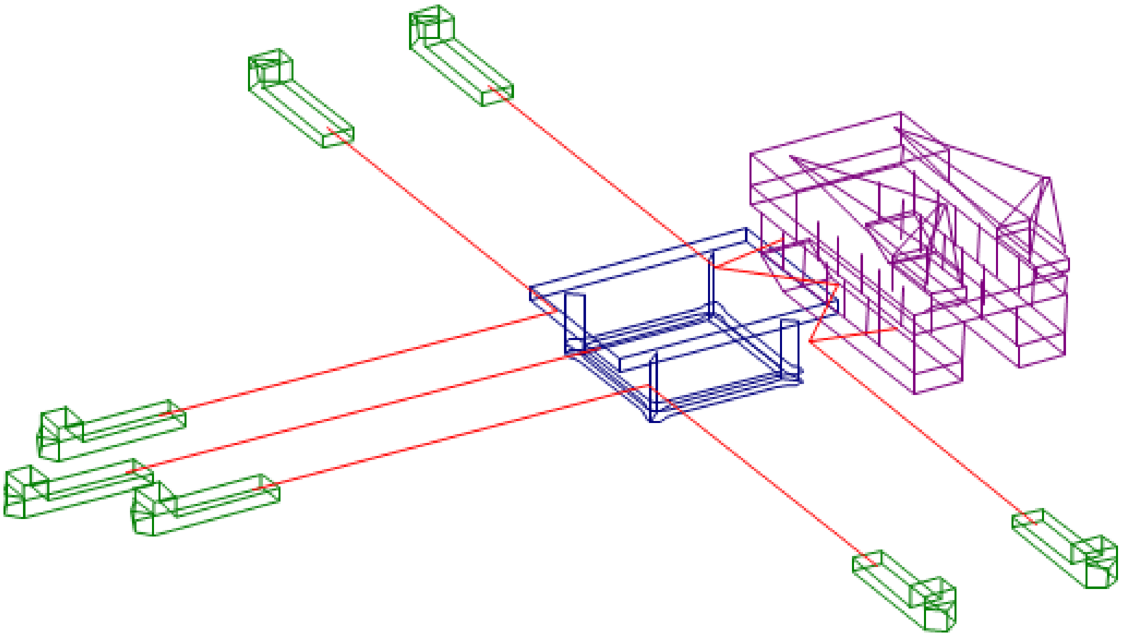
COMPLEX MULTIBODY CALCULATIONS

Last revised: March 12, 2019. Describing version 4.14-00 (64-bit).

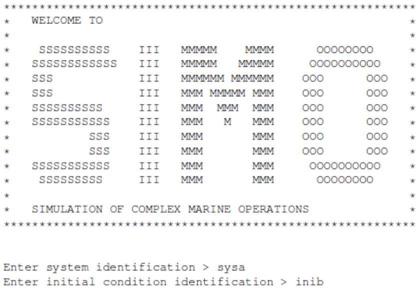
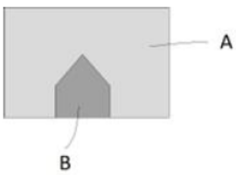
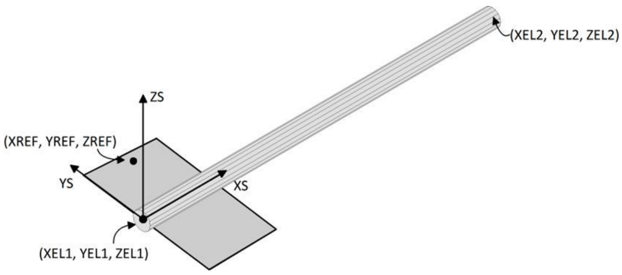
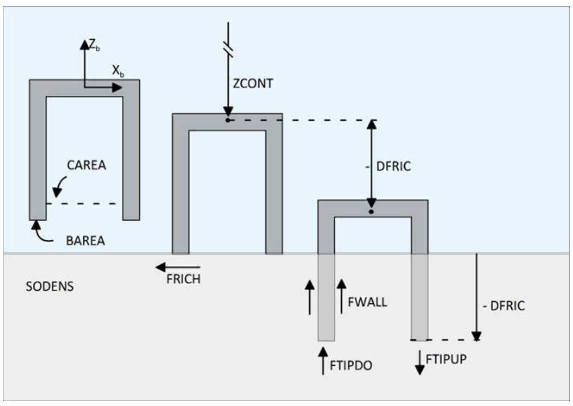
Simo is for simulation of motions and station-keeping behaviour of complex systems of floating vessels and suspended loads. Essential features are:

- Flexible modelling of multimode systems
- Non-linear time domain simulation of wave frequency as well as low frequency forces
- Environmental forces due to wind, waves and current
- Passive and active control forces
- Interactive or batch simulation

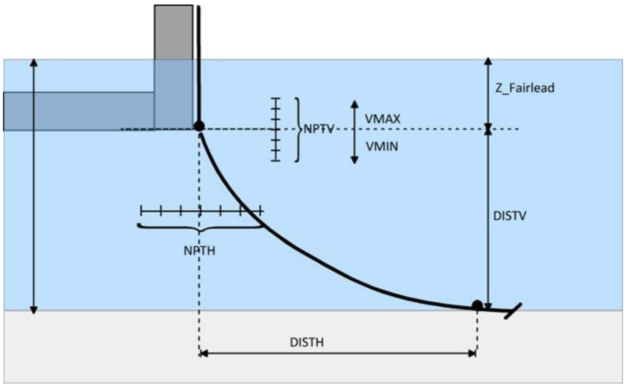
Simo is run from the Sima GUI.



FEATURES OF SIMO

FEATURE	DESCRIPTION
Running Simo 	Simo can be run independently with DOS commands and batch files.
Time dependent mass  Possible tank	Time dependant mass can be directly defined in Simo, which is useful in upending analysis. It could also be used to simulate the ballasting tanks for ships or offshore platforms.
Morison equation for slender elements 	Slender elements can be defined to capture the loads calculated with Morison equation. As an additional modelling option, the user may specify depth dependent scaling of hydrodynamic coefficients for the slender structures.
Soil penetration 	Soil penetration parameters for friction model with soil fracture can be used to simulate suction piles of subsea manifolds.

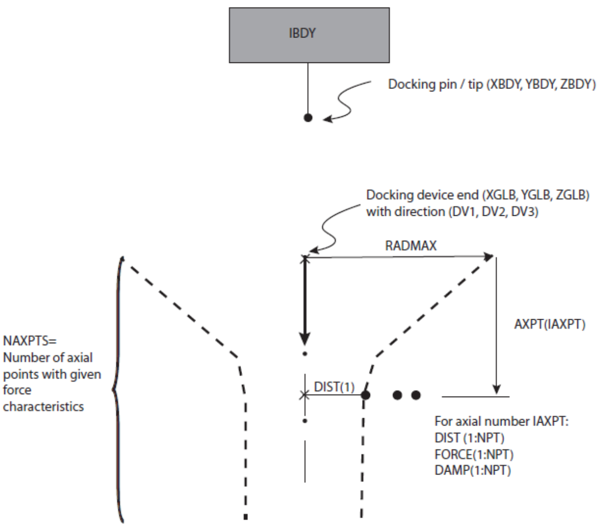
<table>
<tr>
<td>
Positioning elements



</td>
<td>
The positioning elements are divided into the following groups:

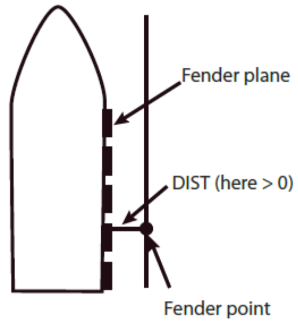
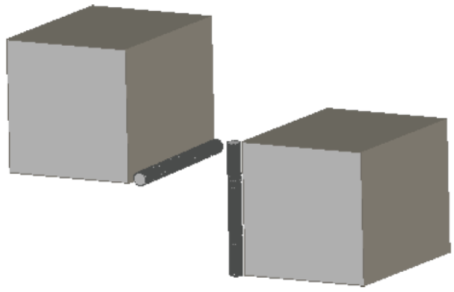
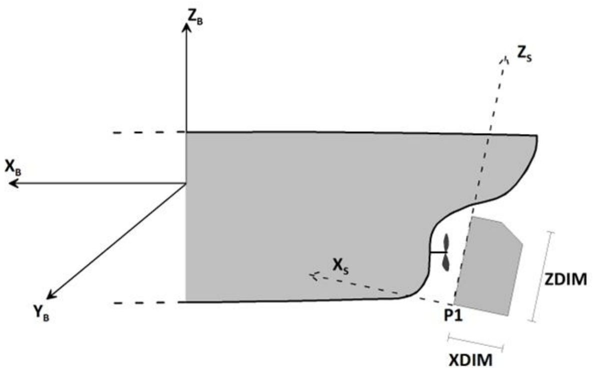
- Catenary anchor lines described by line properties. This is the recommended method to specify mooring lines. Line characteristics will be calculated within the program.
- Directly specified catenary anchor line characteristics. Horizontal and vertical components can be specified for different vertical positions of fairlead.
- Force-elongation relationship with fixed attack points. The force is directed between the end points.
- Docking cone, giving a radial force at offset from target
- Force-elongation relationship with sliding attack points, fender
- Thrusters

If the positioning system includes thrusters, these may be controlled by a dynamic positioning system (DP) which is specified separately.
</td>
</tr>
<tr>
<td>
Docking cone



</td>
<td>
A pair of docking funnel and docking post is a device assisting during the final precision manoeuvre before landing or connection of a structure to a fixed point in a body or on the seabed. These devices are arranged in pairs (guide funnel and guide posts or docking cone/cylinder and guide pin).

The orientation is vertical for all typical load-handling cases. However, in order to utilise the model also for e.g. horizontal coupling between bodies the orientation can be defined by the direction cosines of the axis of the guiding cone/cylinder.
</td>
</tr>
</table>

Fender model 	The fender can either be point symmetric, giving friction in all directions along the sliding plane, or it can be defined as a roller. In the latter case, only motion parallel to the rotation axis results in friction. More details are found in Sima feature description.
Bumper model 	The bumper element model is used to model contact forces between a body and a globally fixed cursor or bumper, or contact forces between bodies. The model is particularly useful in the analysis of offshore installation operations where deflectors/bumper bars are used to guide a module to its correct position and to protect existing equipment from impact damages. The bumper element model can be applied for global positioning of a body, or as a contact coupling between bodies.
Lift and drag forces 	The intention with this option is, in a simplified way, to model the drag and lift forces, for instance on a rudder as response to main propeller actions.

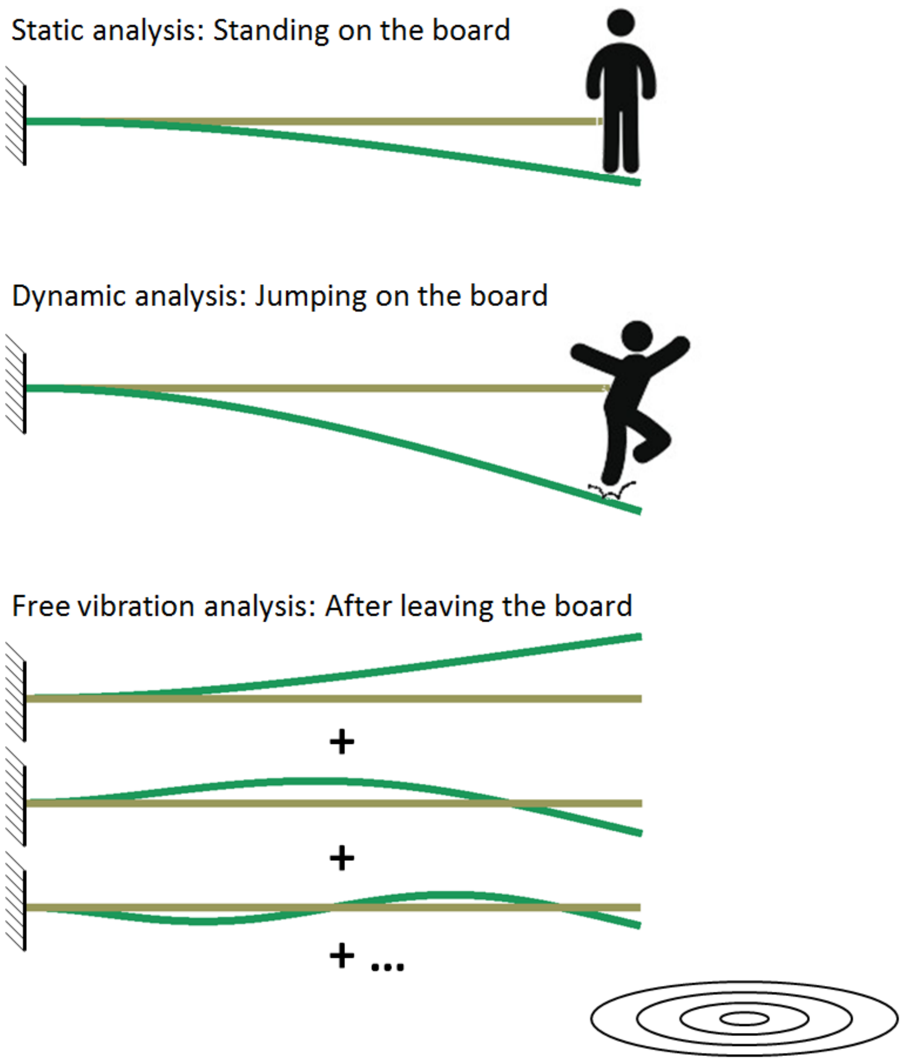
Sestra

COMPUTE STRUCTURAL RESPONSE TO STATIC AND DYNAMIC LOADING

Last revised: January 9, 2019. Describing versions 8.8-02 (64 bit) and 10.6-00 (64-bit).

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.

Sestra exists in two versions: 8.8 and 10. The latter will over time replace the former. Currently Sestra 10 offers, with some limitations, static analysis, free vibration (eigenvalue) analysis, dynamic analysis in time and frequency domain, dynamic analysis by direct and modal superposition methods and analysis of tension/compression only members.

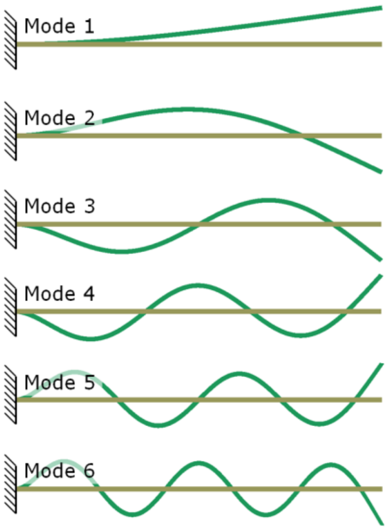


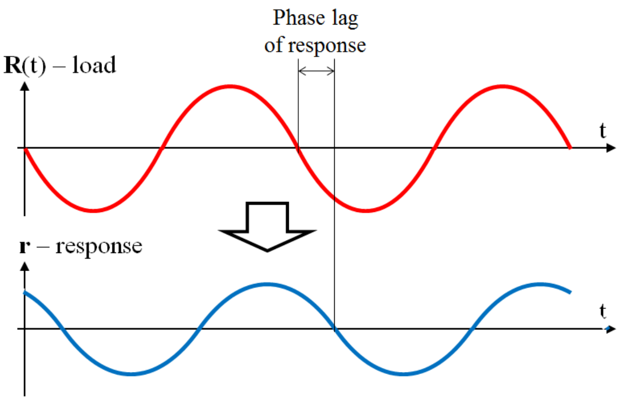
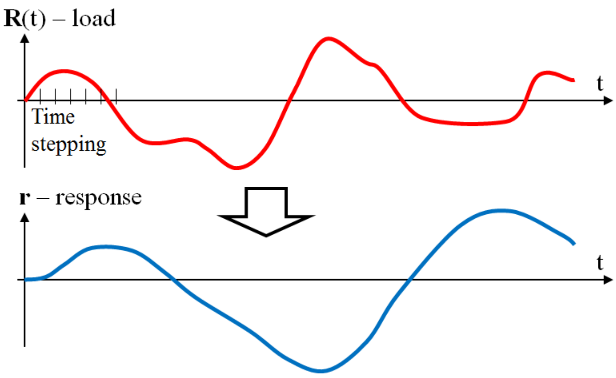
FEATURES OF SESTRA

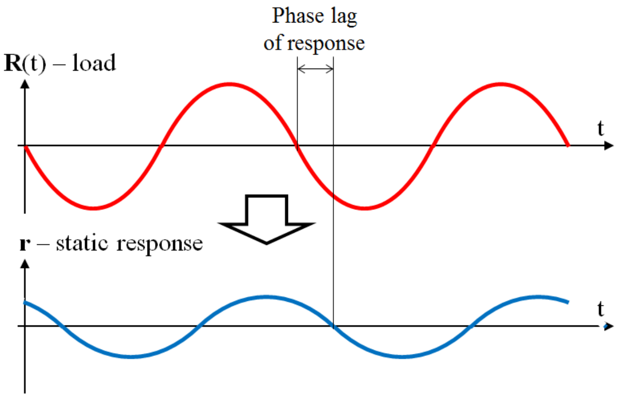
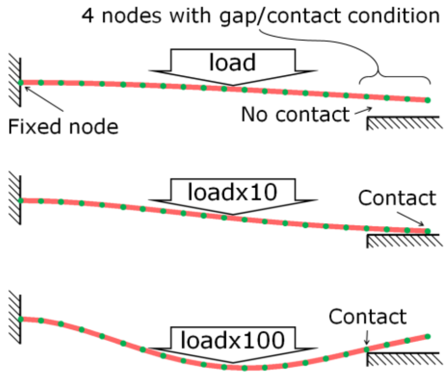
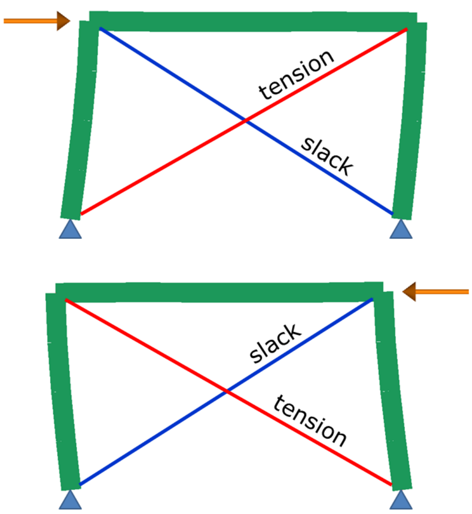
Types of analysis

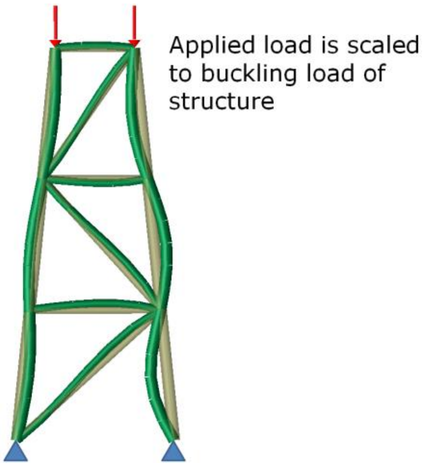
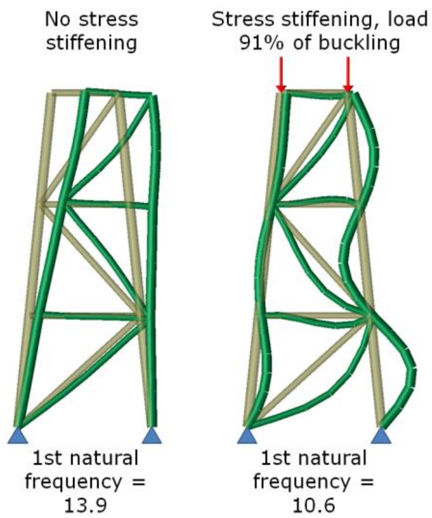
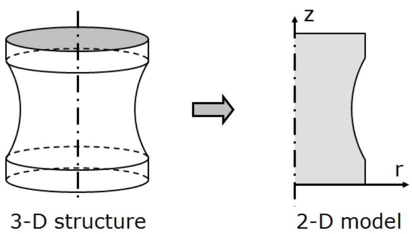
The types of analysis illustrated and explained in the table below are:

- Linear static analysis
- Linear free vibration analysis
- Linear dynamic analysis in frequency as well as time domain
- Linear frequency domain static analysis
- Gap/contact analysis
- Tension/compression only member analysis
- Linear buckling analysis
- Stress stiffening analysis
- Axi-symmetric analysis

FEATURE	DESCRIPTION
Linear static analysis 	The loads are (more or less) constant and the structural response is linear. Linear response means that it is proportional to the load: Double the load and the displacements are doubled. Equation of equilibrium being solved: **Kr** = **R** **K** is stiffness matrix, **r** is displacement vector, **R** is load vector
Linear free vibration analysis 	An analysis to determine the free vibration of the structure when there are no loads and no damping. The mode shapes (eigenmodes) and natural frequencies (eigenfrequencies) are computed. Equation of equilibrium being solved: **Ma** + **Kr** = **0** **M** is mass matrix, **a** is acceleration vector, **K** is stiffness matrix, **r** is displacement vector Assuming **r** = **Φ** sin(ωt) the equation turns into the eigenvalue problem: $(\mathbf{K} - \omega^2\mathbf{M})\mathbf{\Phi} = \mathbf{0}$ ω is angular frequency (=2πf), **Φ** is mode shape (eigenvector) See below for information on eigenvalue solvers offered by Sestra.

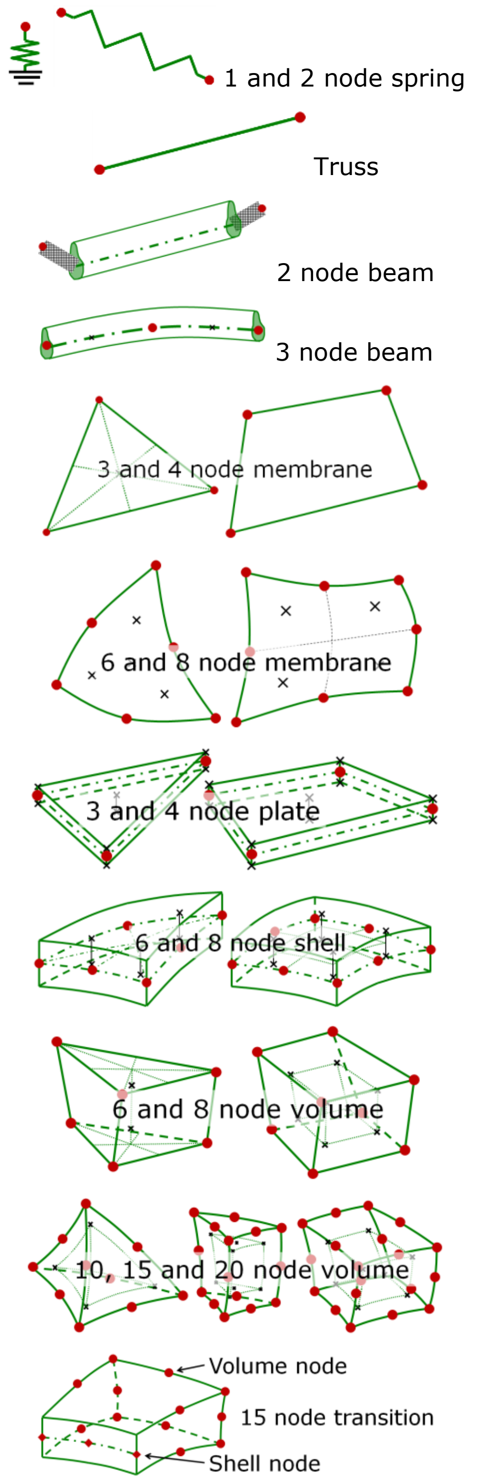
Linear dynamic analysis 	When the loads vary quickly there will be inertia forces due to the mass of the structure. Assuming the structural response is proportional to the dynamic loads a linear dynamic analysis will solve the problem. Equation of equilibrium being solved: **Ma** + **Cv** + **Kr** = **R**(t) **M** is mass matrix, **a** is acceleration vector, **C** is damping matrix, **v** is velocity vector, **K** is stiffness matrix, **r** is displacement vector, **R**(t) is time varying load vector Linear dynamic analysis falls into two categories: • Frequency domain • Time domain
Linear frequency domain dynamic analysis 	The load variation is harmonic, i.e. sinusoidal. The response will also be harmonic. The equation of equilibrium is solved using complex numbers, i.e. the loads and the solution are expressed in two parts: a real part and an imaginary part. The analysis is performed in the frequency domain meaning that a main differentiator between the loads is their frequency. In a frequency domain wave analysis in Sesam (on which the stochastic fatigue analysis is based) the wave amplitude is 1 (in whichever unit the analysis is based: m, mm, inch, …).
Linear time domain dynamic analysis 	The load variation is arbitrary. The analysis is performed in time domain meaning that time is stepped forward and the equation of equilibrium is solved at each time increment. Sestra 10 offers this type of analysis. Time domain analysis demands in general more computer resources than other linear analyses. However, the introduction of Sestra 10 has reduced the analysis time compared with Sestra 8.8 considerably thereby making such type of analysis feasible for fairly large models. Features for time domain analysis: • Repetition of periodic load (e.g. a wave) • Automatic detection of steady state • Store total reaction forces on csv file • Compute and store reaction force RAO

Linear frequency domain static analysis 	This is a variant of the linear frequency domain dynamic analysis. The difference is that the inertia forces due to the mass of the structure are neglected. The inertia forces are small and can be neglected when the loads are varying significantly slower than the first natural period (eigenperiod) of the structure.
Gap/contact analysis 	Two parts of a structure may be in contact or there may be a gap between them depending on the load on the structure. A gap/contact condition is defined for the nodes in question. A structure may also have a gap/contact condition with an external fixed object, e.g. the ground. This is a non-linear problem that cannot be solved directly by Sestra. Rather SestraGap is used. It is a separate program solving the gap/contact problem by repeatedly running Sestra in the background. A gap/contact analysis is available only for static analysis.
Tension/compression only member analysis 	Beam members may be defined as truss (no bending stiffness) of type "tension only" or "compression only". Additionally to assigning such properties to members, a Tension/Compression type of analysis must be specified (from GeniE or Sesam Manager). This is a non-linear problem that is solved by an iteration procedure. A tension/compression analysis is available only for static analysis.

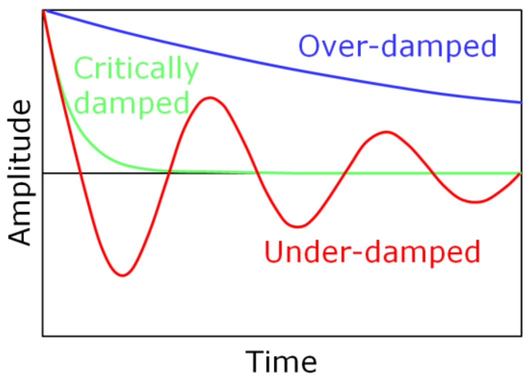
Linear buckling analysis 	A linear buckling analysis determines the scaling of a load required to reach linear buckling. This is an automatic two step procedure: • Static analysis to find forces and stresses. A geometric stiffness matrix $\mathbf{K}_g$ is computed based on these results. • Solve eigenvalue problem: $(\mathbf{K} - \lambda \mathbf{K}_g)\mathbf{\Phi} = \mathbf{0}$ λ is eigenvalue or stability factor, $\mathbf{\Phi}$ is buckling modes. Only the first buckling mode is of interest. Load times λ is the critical buckling load. Second order (P-Δ) effects are not accounted for. This limitation plus the fact that structural imperfections may not be properly accounted for involves that a linear buckling analysis overestimates the real buckling load.
Stress stiffening analysis 	A stress stiffening analysis takes the stiffening effect of tension and softening effect of compression into account when computing e.g. the free vibration of the structure. This is an automatically performed two step procedure: • Static analysis to find the initial stress level. A geometric stiffness matrix $\mathbf{K}_g$ is computed based on these initial stresses. $\mathbf{K}_g$ is added to the stiffness matrix $\mathbf{K}$ thereby adding/subtracting stiffness to the ordinary stiffness. • A new analysis is done based on the updated stiffness matrix. This analysis may be a static, free vibration or forced response dynamic analysis.
Axi-symmetric analysis 	A rotational symmetric model may be analysed as a 2D problem thereby reducing the size of the problem to a fraction compared to a full 3D analysis.

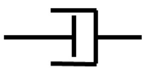
Elements, properties and loads

Presented below are essential features related to the types of analysis presented above.

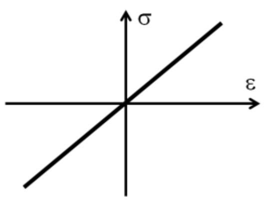
<table>
<tr><th>FEATURE</th><th>DESCRIPTION</th></tr>
<tr><td>Element types

</td><td>The library of elements in Sestra covers all types of structures:
• 1 and 2-node spring
• Damper
• Mass
• Truss
• 2 node beam, with offset (eccentricities) in nodes
• 3 node beam
• 3 and 4 node membrane
• 6 and 8 node membrane
• 3 and 4 node plate
• 6 and 8 node shell
• Axi-symmetric volume
• 6 and 8 node volume (solid)
• 10, 15 and 20 node volume (solid)
• Transition element between 6/8 node shell and 15/20 node volume (solid)</td></tr>
</table>

<table>
<tr>
<td>Hinge

</td>
<td>Hinge, optionally with flexibility, is available for 2 node beam elements. The hinge is general in that it can be defined for all six degrees of freedom, i.e. translational as well as rotational hinges.</td>
</tr>
<tr>
<td>Damping properties


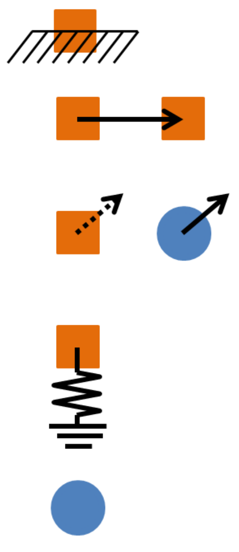

Dashpot</td>
<td>Depending on the type of dynamic analysis being performed there are alternative damping models available:

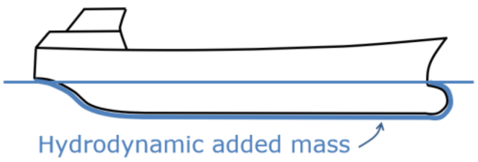
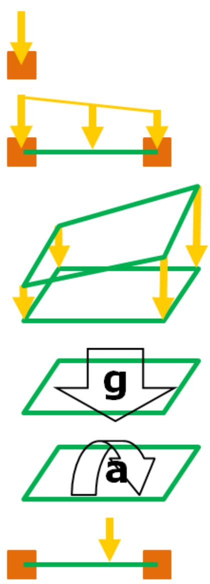
Modal damping as fraction of critical damping
A viscous damping where fractions of critical damping λ for the modes used in Modal Superposition are given.
Proportional damping or Rayleigh damping
A viscous damping where a damping matrix is specified as a linear combination of the stiffness and mass matrices:
$\mathbf{C} = \alpha_1\mathbf{M} + \alpha_2\mathbf{K}$
Structural damping
This is a damping proportional to the displacements and in phase with the velocity. It is relevant only for frequency domain analysis.

Dashpots and axial dampers
Dashpots and axial dampers may be specified by the preprocessors for inclusion in the viscous damping matrix. It is relevant only for direct methods, i.e. not Modal Superposition.</td>
</tr>
<tr>
<td>Material properties

</td>
<td>There are two linear material properties available:

Isotropic

Anisotropic (of which orthotropic is a special case)
Different properties in two directions</td>
</tr>
</table>

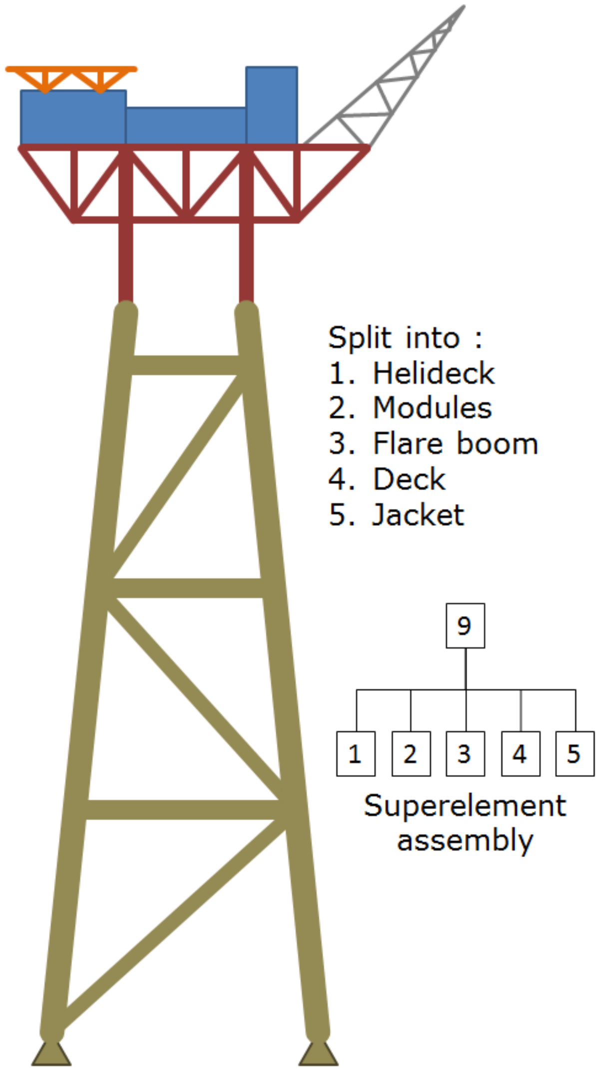
Boundary conditions

Fixed
Prescribed
Linearly coupled
Elastic support
Supernode
(also termed retained)
	The following boundary conditions may be given:
	• Fixed (at zero) displacements
	• Prescribed displacements (used by sub-modelling)
	• Multipoint constraints (or linear dependencies)
	• Elastic support
	• Super degree of freedom is considered a boundary condition for a superelement
	• Additional boundary conditions for dynamic analysis:
	  ○ Prescribed time dependent displacements and accelerations
	  ○ Viscous support
	  ○ Initial displacements and velocities in forced response analysis
	Transformations of node coordinate system may be defined allowing for skew boundary conditions.

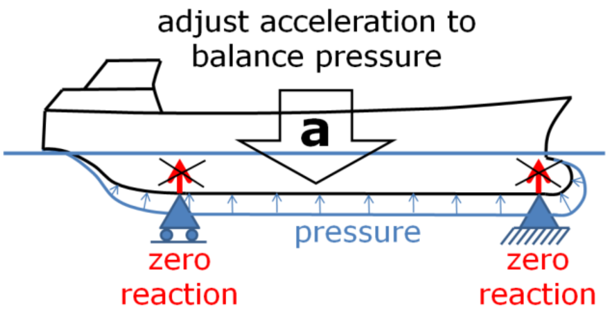
Hydrodynamic added mass  	Hydrodynamic added mass may be computed for floating large volume structures such as ships and offshore platforms.
Loads  Nodal load Line load Surface pressure Gravity Rigid body acceleration Point load on 2 node beam	The following load types may be given: • Nodal load (including moment) • Line load • Surface pressure • Surface load in a component form (X, Y, Z) • Temperature • Gravity • General inertia load • Rigid body acceleration • Point load or load linearly distributed over part of the element for 2 node beam elements

Equation solvers

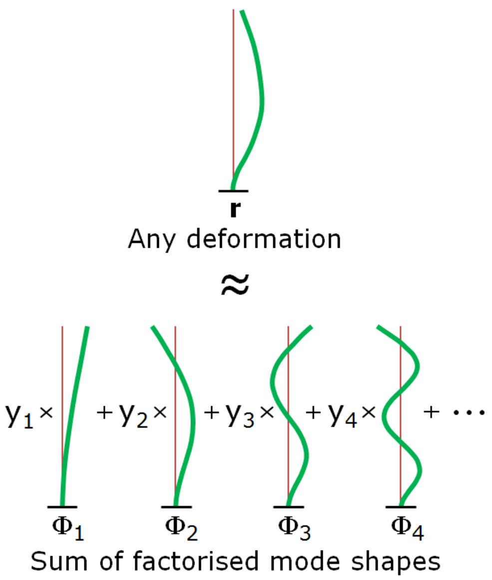
Presented below are solvers related to the types of analysis presented above.

FEATURE	DESCRIPTION

Superelement analysis

 	
	Sestra features a multilevel superelement technique applicable for static analysis. This technique involves splitting the structure to be analysed into part models, superelements, and assembling these to form the complete model. Not only Sestra but the whole Sesam suite of programs incorporates the superelement technique:
	• GeniE creates 1st level superelements
	• Presel assembles superelements through any number of levels to form the complete model
	• Wajac and Wadam computes loads and motion for superelement assemblies
	• Sestra analyses superelement assemblies
	• Postprocessors (Xtract, Framework, Stofat, etc.) handle superelement assemblies
	Whereas the computer power and technology of today no longer makes the superelement technique a necessity it is still useful:
	• Different teams can work on different superelements
	• Previously analysed model may be added as a superelement to a new model
	• A huge results file for a very big model may be split thereby facilitating postprocessing work
	• Allows highly efficient non-linear analysis when the non-linearity is limited to regions, examples:
	    ○ Structure-pile-soil analysis, Sestra for linear structure, Splice for non-linear pile-soil
	    ○ Contact/gap analysis, Sestra for linear structure, SestraGap for non-linear contact/gap region

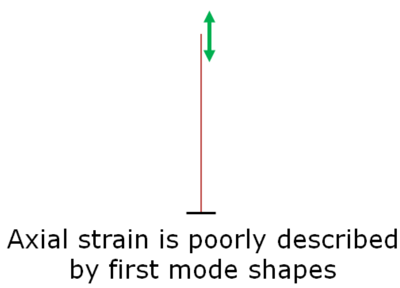
<table>
<tr>
<td>Inertia relief
</td>
<td>Inertia relief is useful in a static analysis of a floating object to perfectly balance acceleration and pressure loads, i.e. to achieve zero reaction forces in the fixed nodes.

A floating object has no fixations but a static analysis requires as a minimum a statically determined model. I.e. three nodes with three, two and one degree of freedom fixed, respectively.

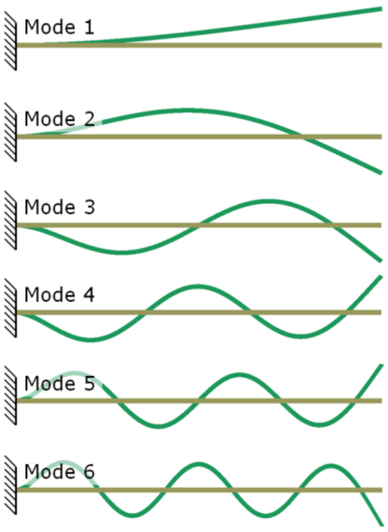
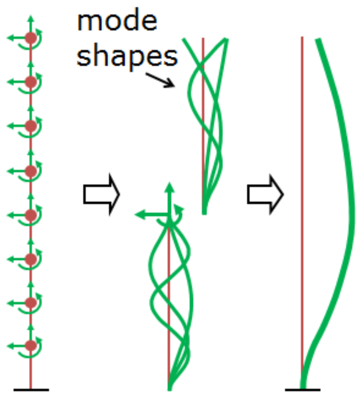
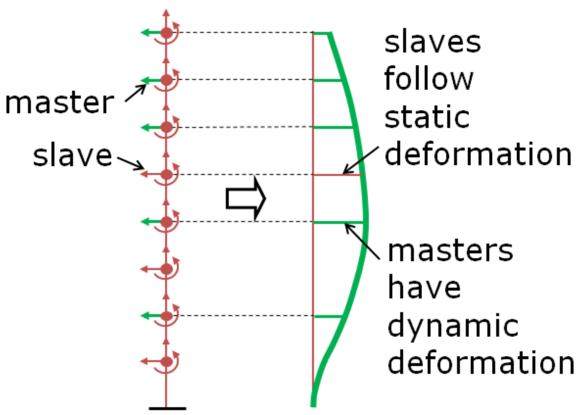
A floating object analysed by HydroD/Wadam will have acceleration and pressure loads that are approximately in balance. To improve this balance the inertia relief feature of Sestra adjusts the accelerations. Make sure an approximate balance has been achieved before using this feature!</td>
</tr>
<tr>
<td>Modal superposition
</td>
<td>Modal superposition is available for both frequency and time domain dynamic analyses.

The principle of the method is to replace the degrees of freedom (dofs) of the model with mode shapes. The first step of the method is therefore to perform a free vibration (eigenvalue) analysis. The user decides how many mode shapes to contribute to the dynamic analysis, the more mode shapes the closer approximation to the direct analysis (not using model superposition):

$\mathbf{r}(t) \approx \Sigma \mathbf{\Phi}_i y_i(t)$

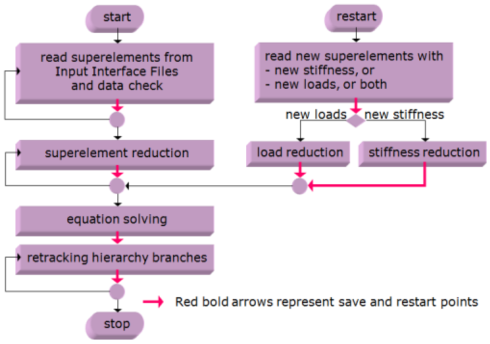
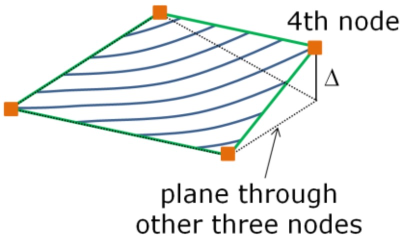
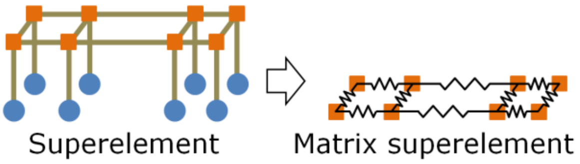
The advantage of the method is the fast solution of the dynamic problem (once the eigenvalue problem has been solved). This is due to that replacing dofs with mode shapes decouples the equations of dynamic equilibrium.</td>
</tr>
<tr>
<td>Static back substitution
</td>
<td>The modal superposition method has an inherent weakness: Axial and in-plane strains are poorly described by the first mode shapes typically selected to represent the dynamic deformation. This is because axial and in-plane stiffnesses are much higher than transverse and out-of-plane stiffnesses. Axial forces in beams and in-plane stresses in plates and shells are therefore inaccurate.

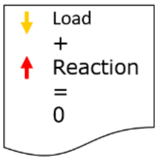
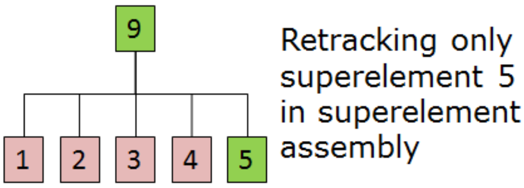
The static back substitution remedies this weakness.</td>
</tr>
</table>

Eigenvalue solvers 	The eigenvalue solvers available in Sestra are: • Multifront Lanczos An implicitly restarted Lanczos' method combined with a multifrontal solver, this is a very efficient solver • Lanczos' method Suitable for big problems but not when there are un-constrained stiff body motions • Subspace Iteration This is suitable for big problems and when the eigenvalues are dense • Householder's method This is suitable for small problems and many eigenvalues
Component mode synthesis 	Component mode synthesis is a reduction technique for dynamic analysis resembling the superelement technique for static analysis. The principle of the method is to split the model into part models (seen from a modelling point of view: superelements) and for each part replace the degrees of freedom (dofs) with mode shapes. The first step of the method is therefore to perform free vibration analyses of all part models. When proceeding from a static analysis by use of the superelement technique to a dynamic analysis then the component mode synthesis technique is an attractive choice.
Master-Slave 	Master-Slave is a reduction technique for dynamic analysis. The principle of the method is to define selected degrees of freedom (dofs) as masters (seen from a modelling point of view: super-dofs) and then do the same condensation of the mass and damping matrices as done for a stiffness matrix in a static condensation. The consequence of this is a lumping of mass and damping to the master-dofs. The Master-Slave technique also provides a way of eliminating soft areas (due to approximate modelling?) from the dynamic analysis.

Additional features

Presented below are a few additional features.

FEATURE	DESCRIPTION
Save and restart 	There are several points during the execution at which the analysis may be stopped and data saved. At a later point the analysis may be restarted based on the data saved.
Warp correction 	The 4-node plate element is based on that all four nodes lie in a common plane. This is not always possible, e.g. for doubly curved surfaces. Warped elements prevent rigid body rotations and cause false rotational reactions (moments). This in turn causes spurious stresses in the warped elements. In such cases the warp correction feature improves the results.
Export and import of matrices 	A reduced superelement may be exported as a stiffness matrix and a load vector. These may be used as a matrix superelement in a subsequent analysis.

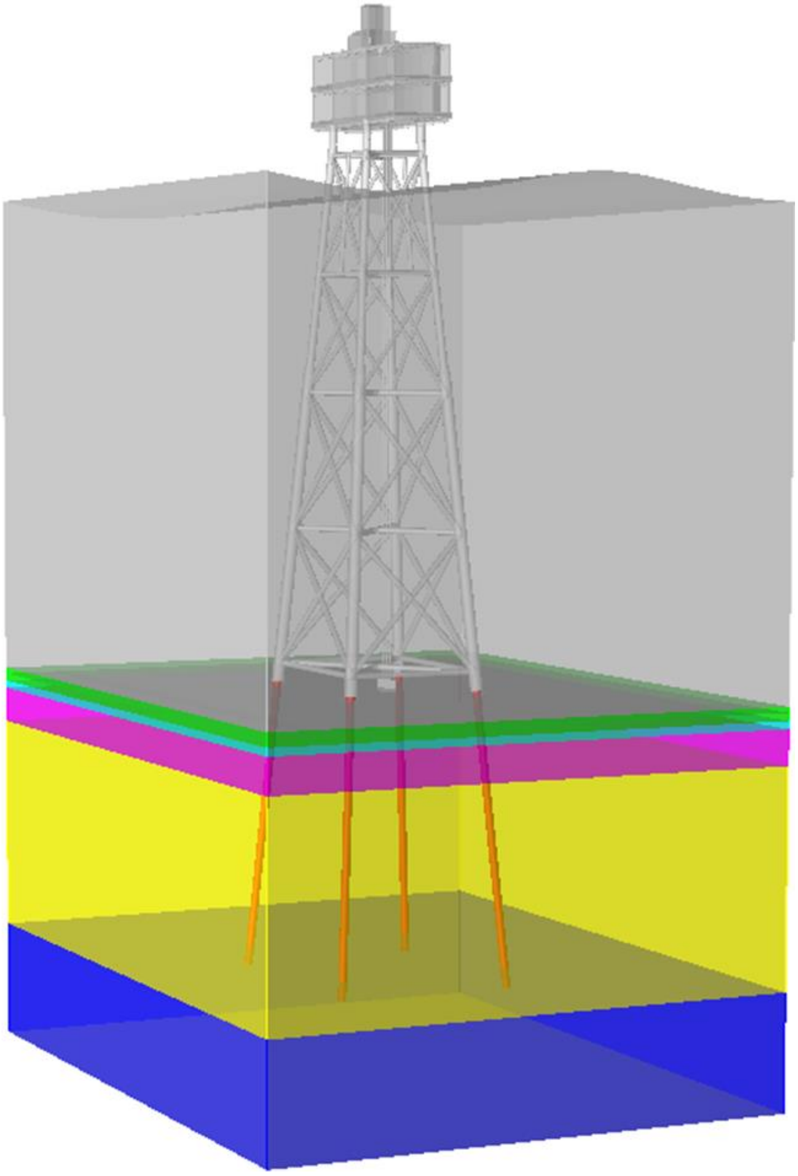
Print 	A few output text files from Sestra serve the purpose of verifying the quality of the analysis. Presentation and further processing of the results is done by the various postprocessors of Sesam. The output text files include: • Interpretation of input • Summary of model data • Possible warnings and error messages • Sum of loads • Sum of reaction forces • Sum of loads and reaction forces, should be zero for a static analysis
Retracking superelement assembly 	Retracking is the phase in the solution process computing forces and stresses in the model (based on the nodal displacements). When using the superelement technique only a part of the superelement assembly (also termed superelement hierarchy) may be retracked instead of the complete model. This saves both computation time and storage space.

Splice

STRUCTURE-PILE-SOIL INTERACTION ANALYSIS

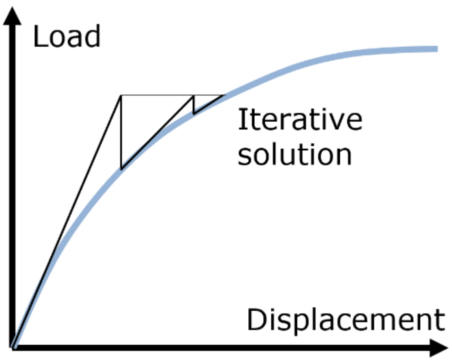
Last revised: April 23, 2018. Describing version 7.5-00 (64-bit).

Splice is a program for non-linear analysis of the structure-pile-soil interaction problem of typically a jacket supported by piles driven into the sea bed. The programs Pilgen and Gensod belong to Splice, the former for pile modelling and the latter for generation of non-linear soil stiffnesses. Pilgen is, however, normally not used since the pile modelling is done as part of the jacket modelling in GeniE. The non-linear soil stiffnesses are generated by Gensod based on soil modelling performed in GeniE. While Splice analyses the non-linear pile-soil interaction it runs Sestra in the background for analysing the linear jacket.



FEATURES OF SPLICE

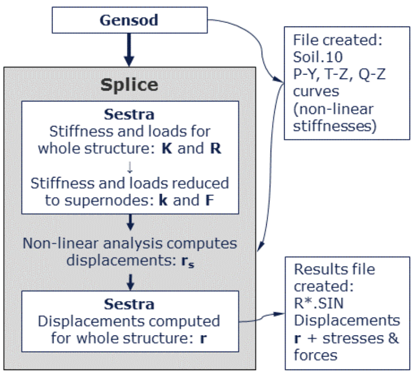
<table>
<tr><th>FEATURE</th><th>DESCRIPTION</th></tr>
<tr><td>Non-linear analysis

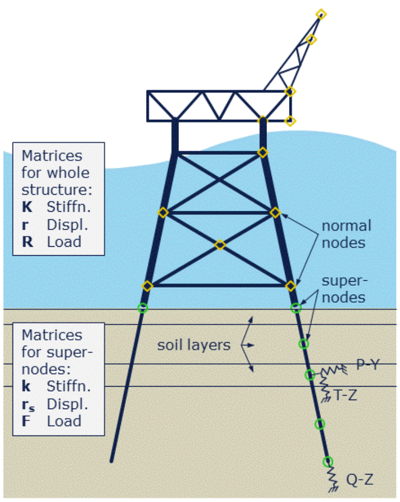


</td><td>Splice provides a fully non-linear analysis:

- Non-linear sand and clay layers
- Including past load displacement history
- Pile material non-linearities
- Pile second order moments
- Pile-soil-pile interaction (group effect)
- Temperature effects (e.g. oil in conductors)
- Imposed soil displacements (mudslides, nearby structures, etc.)

For each of several load combinations an iterative process solves the non-linear problem and computes displacements of nodes along the piles.</td></tr>
<tr><td>Structure-pile-soil interaction

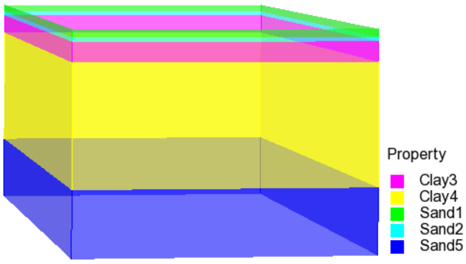
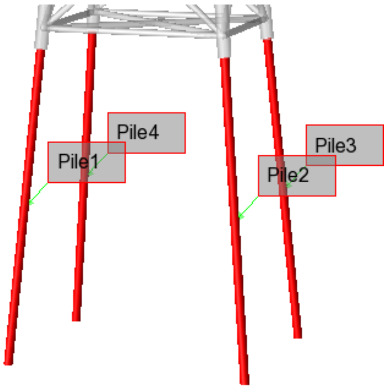
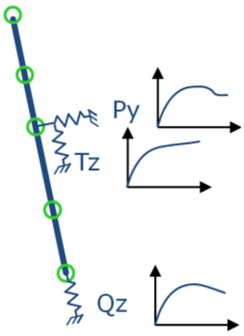
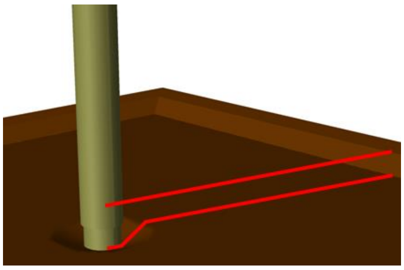


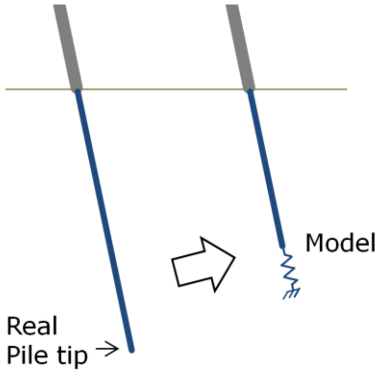
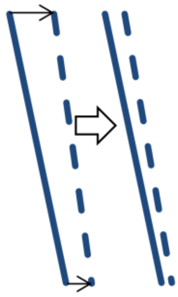
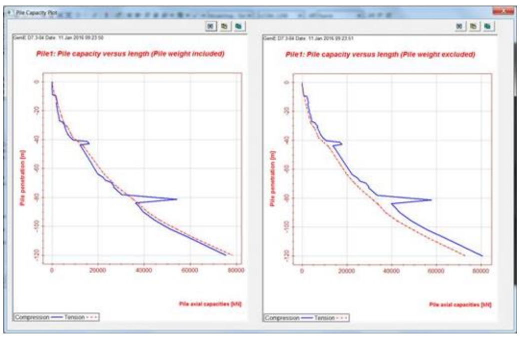


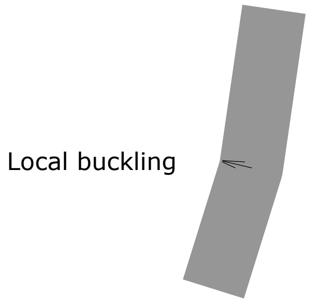
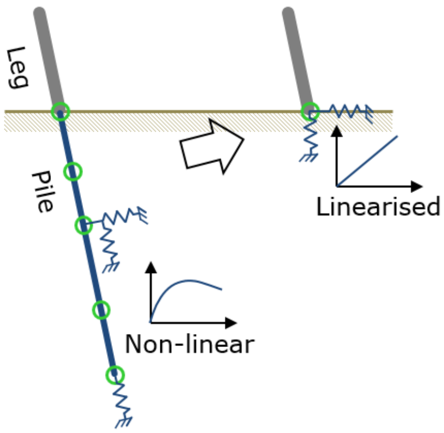
</td><td>By running Sestra in the background the stiffness of and loads on the linear structure (e.g. jacket) are reduced to the pile heads (supernodes).

These reduced stiffness and loads contribute to the non-linear stiffness of and loads on the pile-soil. Splice computes the pile displacements.

By running Sestra once more in the background the displacements, forces and stresses throughout the model are computed by back-substitution.</td></tr>
</table>

Soil modelling 	The soil consisting of sand and clay layers may be modelled by up to 100 layers. For each layer the geotechnical parameters are given: • For sand: angle of friction • For clay: undrained shear strength
Piles 	Up to 100 tubular straight piles may be modelled.
Py, Tz and Qz curves 	The Py, Tz and Qz curves representing the soil's non-linear lateral, friction and tip resistance, respectively, are determined in alternative ways: • Based on given soil data • Predefined curves according to API, DNV, ISO, etc. • Manually given
Scour 	General (for whole structure) and local (for each leg) scour may be given.

Pile tip modelling 	There are alternatives for handling the pile tip: • Free • Fixed • Pile is assumed to be infinitely long beneath real tip or beneath given level
Pile group 	Pile group effects (one pile influences a neighbouring pile) are handled based on the Mindlin equation.
Second order effects 	Second order effects for the piles may be accounted for
Pile capacity 	Calculation of the capacity curves based on allowable maximum deflections • Pile penetration depth based on maximum axial capacity Compare with geotechnical report

Pile code check 	Piles may be code checked according to: • Norsok N-004 2013 • ISO 19902 2007 • API RP 2A 21st & 22nd editions
Linearization of pile-soil 	The linearization of the pile-soil foundation required for free vibration analysis of the structure as well as for structural fatigue analysis may be done automatically based on a selected loading condition and pile.

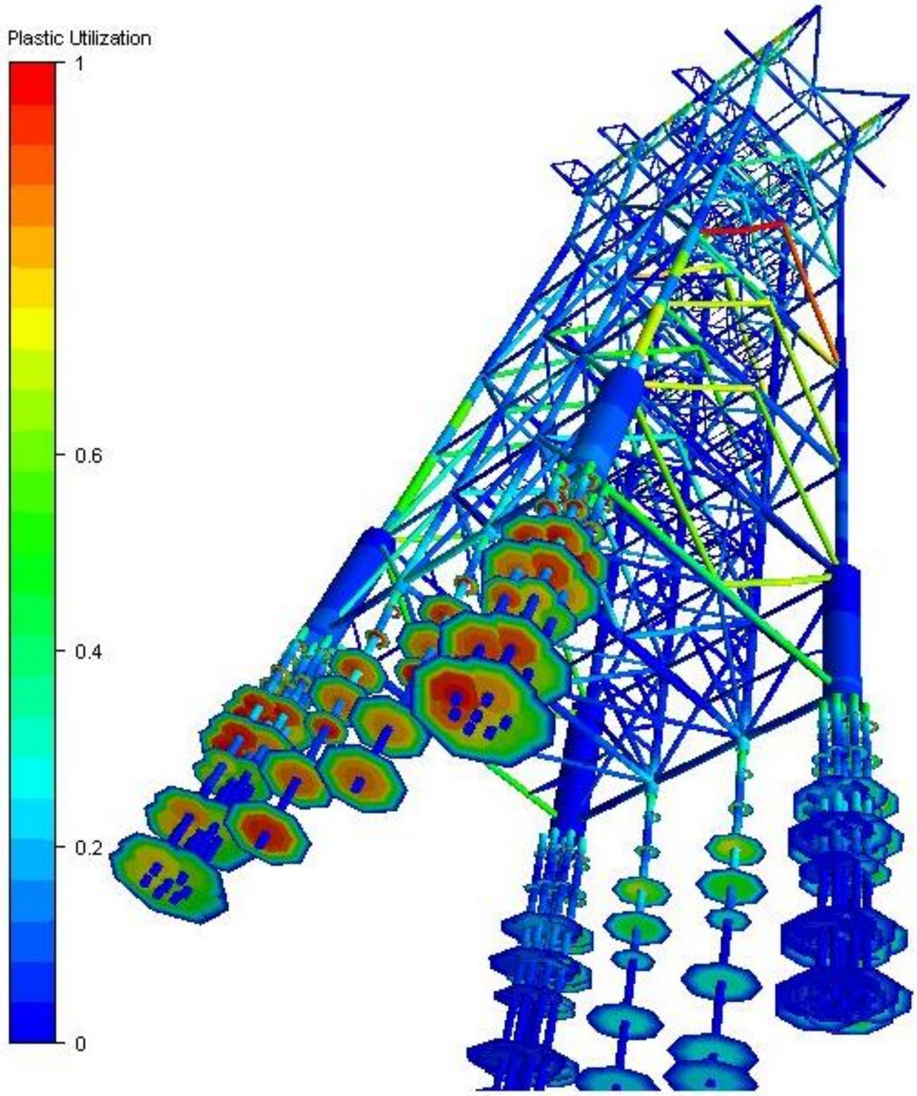
Usfos

NON-LINEAR STATIC AND DYNAMIC ANALYSIS OF SPACE FRAME STRUCTURES

Last revised: April 23, 2018. Describing version 8.8-01 (64-bit).

Usfos is used for ultimate strength, progressive collapse and accident analysis of space frame structures, e.g. jackets in intact and damaged conditions. Extreme and accidental scenarios like earthquake, explosion, fire, ship collision, dropped object etc. are analysed using Usfos.

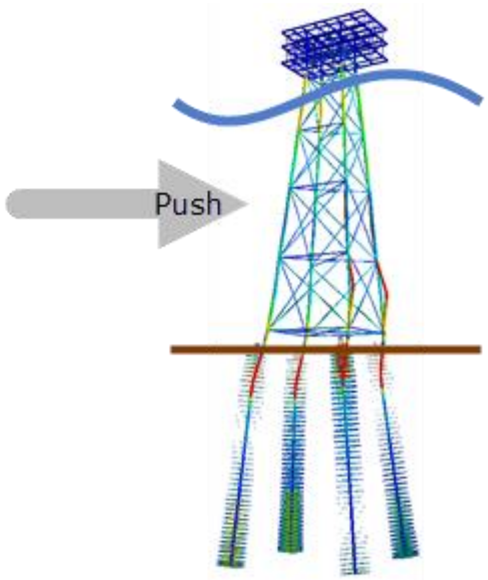
ULS (ultimate limit state) analysis in Usfos is performed using a coarse beam and plate mesh equal to the one used in linear analysis. Linear analysis models may therefore be used in Usfos with little or no modifications. Yielding, plastic hinges, buckling, local buckling, joint yielding and fracture are accounted for using a single beam element between joints.



FEATURES OF USFOS

The analysis program Usfos comes with a front end graphical user interface named Xact in which input specification and analysis control is done. The utility tool Struman for converting models from GeniE, Sacs, Staad and Abaqus is run from Xact. And the graphical presentation of results is done in Xact.

Usfos is interfaced with Fahts (Fire And Heat Transfer Simulations). Fahts has an interface to the CFD tool Kameleon FireEx KFX. Fahts prepares temperature data for structural response analysis in Usfos.

<table>
<tr><th>FEATURE</th><th>DESCRIPTION</th></tr>
<tr><td>Pushover analysis

</td><td>A pushover analysis is carried out for a combination of static loads plus wave and wind loads.

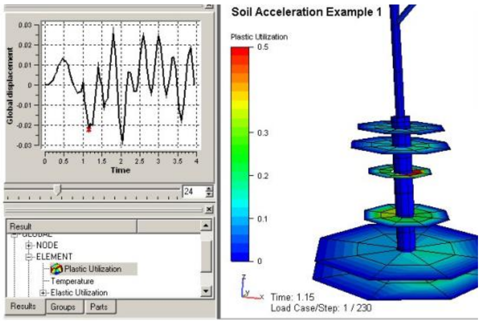
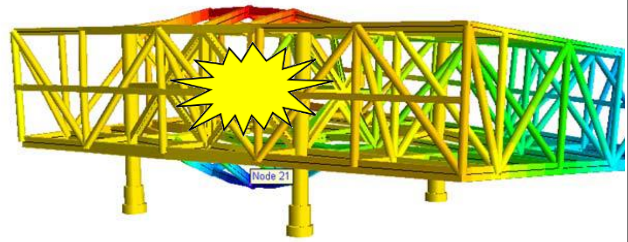
The finite element model may be imported from GeniE along with explicit loads or it may be modelled in Usfos via a text file. Wave loads and pile-soil data may be imported from Wajac and Splice or generated in Usfos.

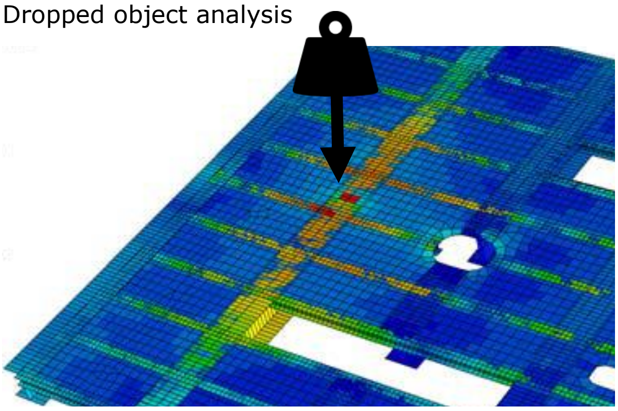
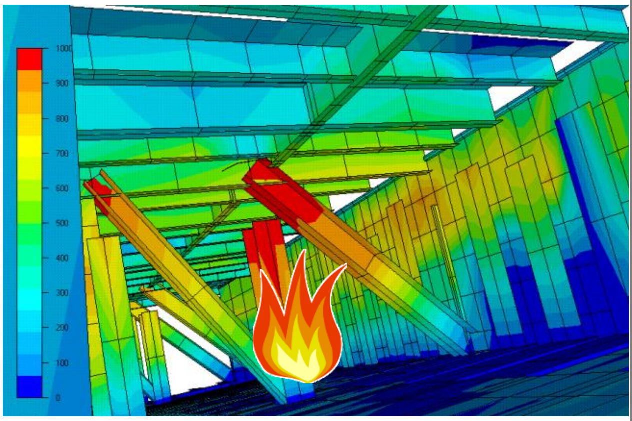
Initial imperfections may be defined according to various curves recommended by codes (standards) and by manual input data. The geometric stiffness matrix is updated after each load step based on the deflected nodal coordinates. Two surface plasticity models are used to simulate the material non-linearity.

Joint checks according to API, ISO, NORSOK etc. are integrated parts of the analysis</td></tr>
<tr><td>Boat impact analysis
</td><td>A boat impact analysis requires definition of impact energy. The impact energy is transferred into dent growth of the impacted tube, deformation of member being hit, global deformation of the structure and energy absorption of the ship.

After unloading the boat impact a 100-year design wave may be stepped through the piled jacket structure to document its strength according to different codes.

A boat impact analysis may also be simulated by a dynamic analysis in which non-linear springs transfer the kinetic energy from a point mass with a given velocity.</td></tr>
</table>

<table>
<tr>
<td>Seismic analysis

</td>
<td>A time history earthquake analysis may be carried out in Usfos for jacket models with non-linear pile-soil data. The piles need not be replaced with spring supports at the bottom of legs. Ground motion in terms of displacement/acceleration time history is applied to the soil and soil elements transfer the motion to the piles.</td>
</tr>
<tr>
<td>Explosion analysis

</td>
<td>Explosion analysis may be carried out by defining the pressure time history for the blast after subjecting the structure to static loads.

Beam and shell strains may be investigated in Xact.</td>
</tr>
<tr>
<td>Dropped object analysis
</td>
<td>Analysis of an accidentally dropped object may be simulated by different approaches in Usfos. Nodal masses with given initial velocities is one method. In case the members hit are fractured structural integrity for the global model may be checked by defining an operational pushover load. This will determine the significance of the fractured member and assist in evaluating the necessity of repair.</td>
</tr>
<tr>
<td>Fire analysis
</td>
<td>A fire analysis consists of three stages. The combustion process is simulated using a CFD tool, e.g. Kameleon FireEx KFX. The output file from the CFD tool is used by Fahts to compute the transient temperature distributions on the beam elements. The structural response is finally computed by Usfos.

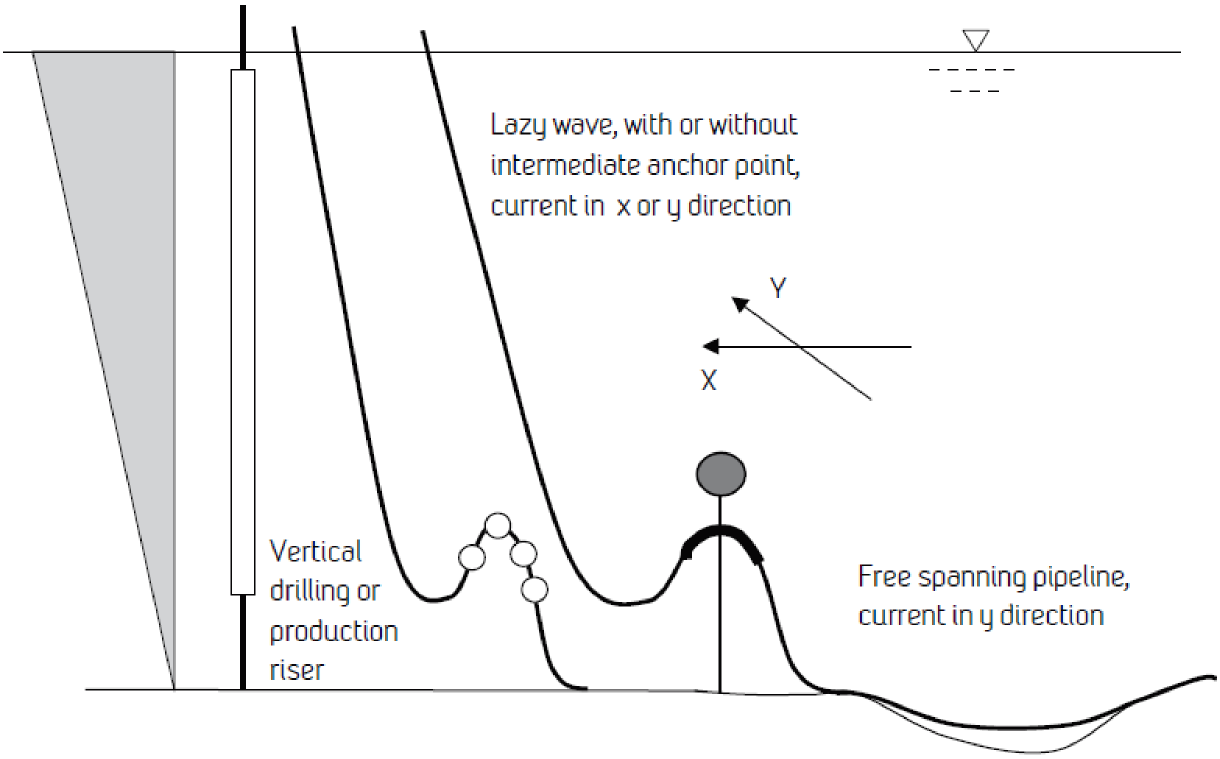
Resistance degradation effects are accounted for in Usfos.</td>
</tr>
</table>

Vivana

VORTEX INDUCED VIBRATIONS OF SLENDER MARINE STRUCTURES

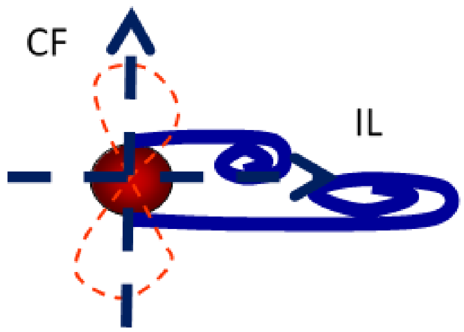
Last revised: March 12, 2019. Describing version 4.14-00.

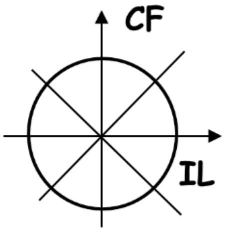
Vivana is a computer tool for calculation of vortex induced vibrations (VIV) of slender marine structures such as risers, free span pipelines and cables subjected to ocean current. This response type may in many cases be decisive for the design and operation of marine systems.



FEATURES OF VIVANA

FEATURE	DESCRIPTION

Typical slender marine structures: e.g. Risers, umbilicals, pipelines, subsea jumpers, etc. 	• Coefficient based
	  ○ added mass coefficient
	  ○ excitation coefficient
	  ○ damping coefficient
	• Finite element model, 3D response, arbitrary geometry
	• Frequency domain, response at eigenfrequencies
	• Pure CF (cross flow), pure IL (in line) or CF + IL excitation

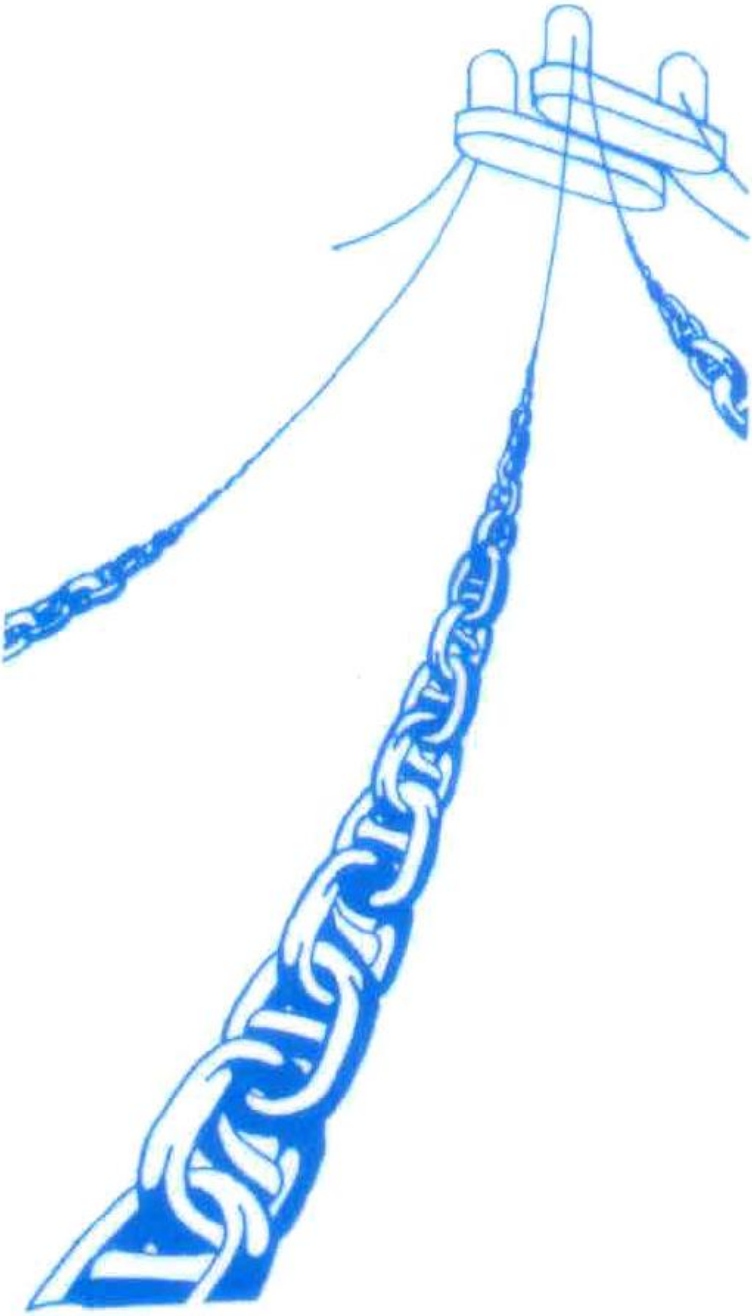
Fatigue analysis 	• All active frequencies will contribute • Miner-Palmgren damage accumulation • 8 points around the cross section in order to account for axial force variation, 3-D structure and CF/IL response

Mimosa

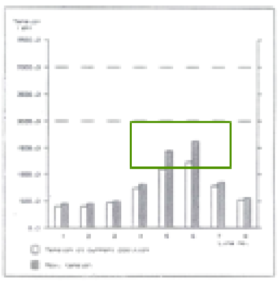
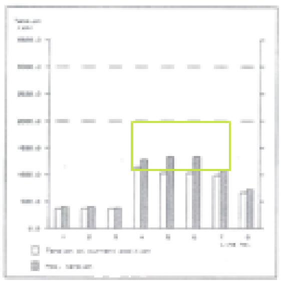
MOORING LINES ANALYSIS

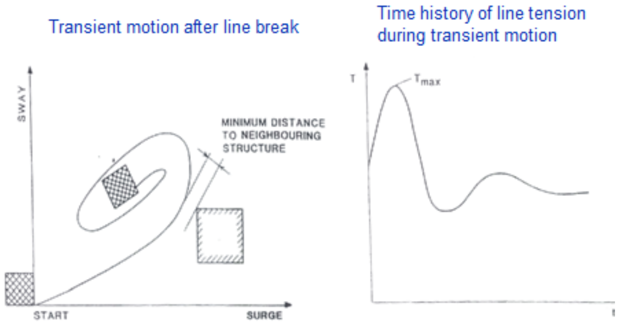
Last revised: April 23, 2018. Describing version 6.3-08.

Mimosa is an interactive program for static and dynamic analysis of moored vessels. It computes static and dynamic environmental loads, corresponding displacements and motions of the vessel and static and dynamic mooring tensions.



FEATURES OF MIMOSA

FEATURE	DESCRIPTION
Static and dynamic mooring system analysis	Mimosa offers a variety of options such as calculation of the vessel’s wave frequency and low-frequency motions and mooring line tensions. Several options are available for analysis of the properties of the mooring system and individual mooring lines.
Static and dynamic environmental forces due to wind, waves and current	• 4 wind spectra, incl. ISO 19901-1 (NPD) and API • 5 wave spectra, wave spreading • Vertically varying current profile
Tension optimization Non-optimised  Optimised 	Optimum distribution of tension based on either minimising the maximum tension in the mooring system or least squares minimisation including thrusters.
1st-order wave-induced motions (wave frequency, WF)	Wave induced motion in the WF range solved in frequency domain, using transfer functions for vessel and wave spectrum, ensuring computer efficient computations.
Slow-drift motions (low frequency, LF)	Maximum slow drift motion and maximum dynamic mooring tension based on non-Gaussian statistics.
Gaussian and non-Gaussian statistics of extremes	Combining WF and LF motion to compute the extremes of the combined WF and LF motion
Dynamic line models	1. Simplified analytic model 2. Finite element model 3. External motion-to-tension transfer functions

Transient motion after line breakage  	Transient motion after a line breakage or thruster failure in terms of time traces of motion and tension, motion trajectory, maximum excursion of any given point on the vessel, and maximum mooring tension. This also includes free drift (DP blackout).
Automatic thruster assistance	Static and dynamic forces from thrusters under dynamic positioning control.
Stability analysis	Stability of the vessel in single-point mooring or turret mooring checked by eigenvalue analysis.
Long term simulation	Long term simulation results based on using a macro command facility for running a set of environmental conditions and producing corresponding sets of results, covering e.g. 5-20 years of operation

Riflex

RISER ANALYSIS

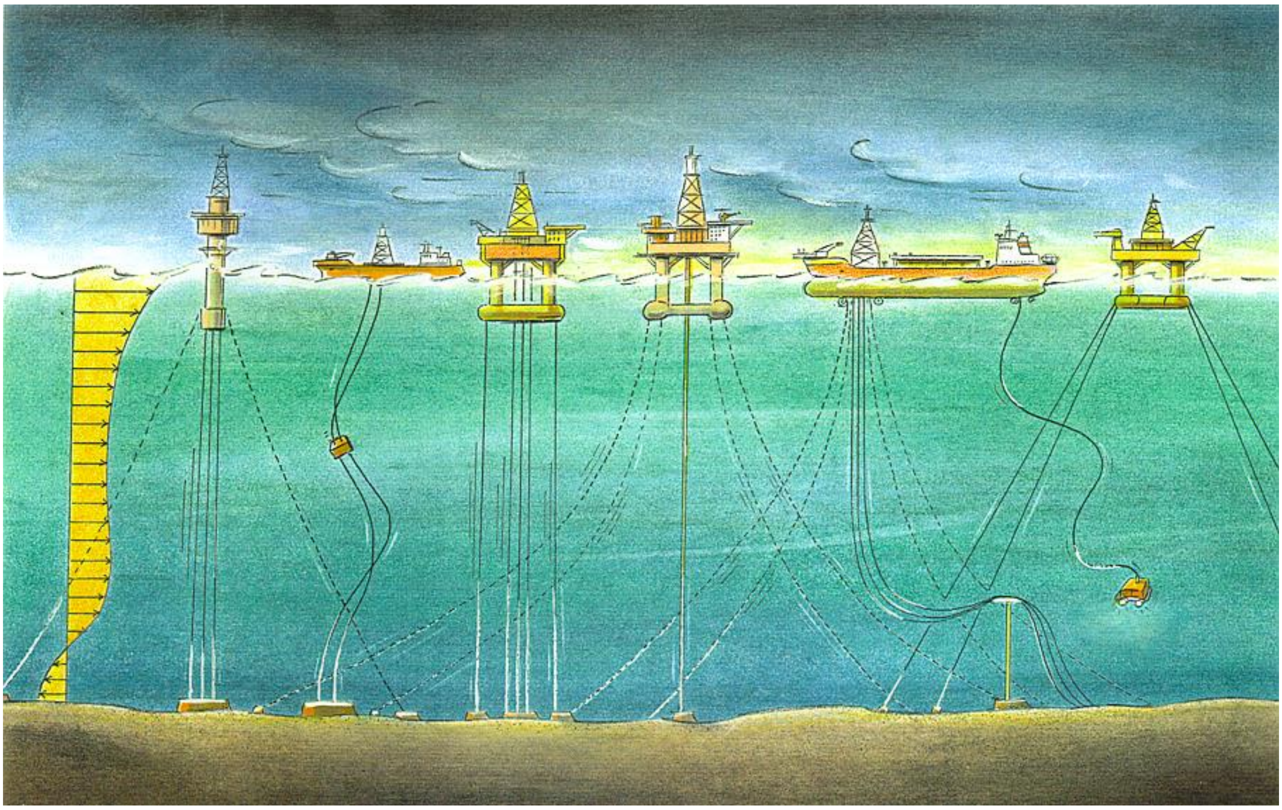
Last revised: March 12, 2019. Describing version 4.14-00 (64-bit).

Riflex was developed as a tool for analysis of flexible marine riser systems but is just as well suited for any type of slender structure such as mooring lines, umbilicals, steel pipelines and conventional risers.

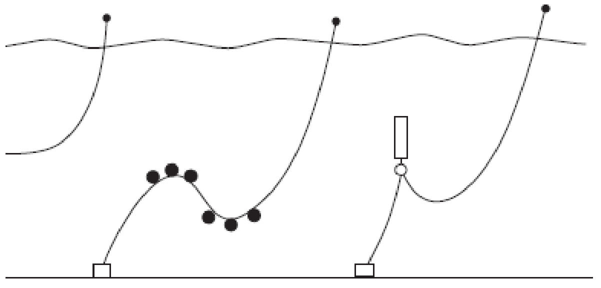
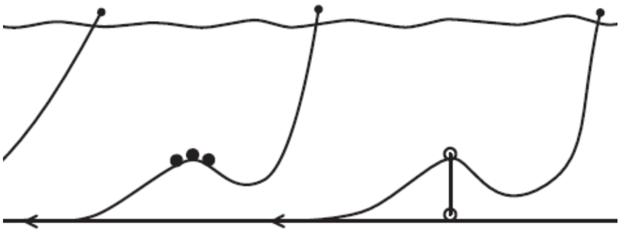
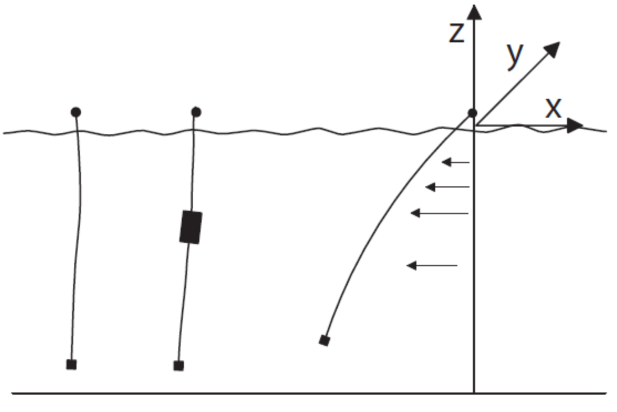
These slender structures are characterized by:

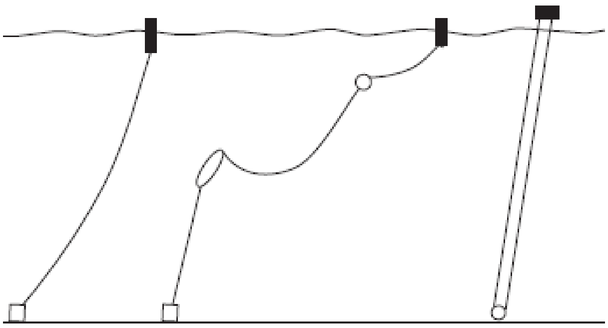
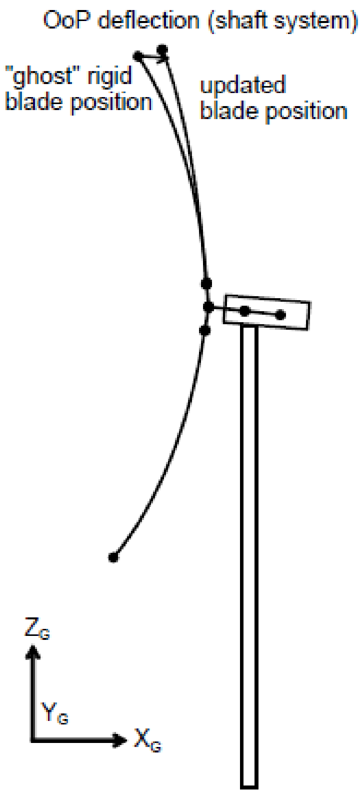
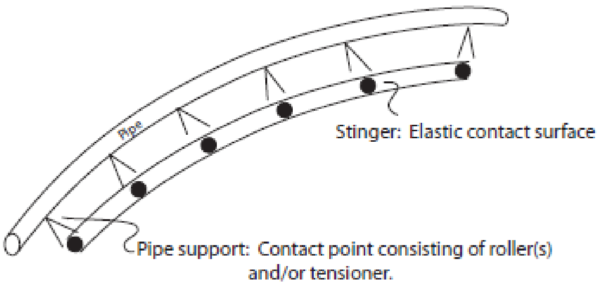
- Small bending stiffness
- Large deflection
- Large upper end motion excitation
- Nonlinear cross section properties
- Complex cross section structure

Riflex is run via Sima.



FEATURES OF RIFLEX

FEATURE	DESCRIPTION
Seafloor to surface vessel, one-point seafloor contact 	In this topology, the riser is suspended between two defined points. The lower end is fixed while the upper end is connected to the surface vessel.
Seafloor to surface vessel, seafloor tangent 	Compared to the above topology this system includes additional features: • Seafloor tangent boundary condition • Buoyancy guide at one point The seafloor contact is modelled by bilinear stiffness. The stiffness is discretized and implemented as springs at the nodal points that may touch the seafloor.
Free lower end, suspended from surface vessel 	This topology is characterized by a free lower end and all degrees of freedom being specified at the upper end. This configuration represents typical installation phases but, as indicated by the figure, towing configurations can be analysed as well.

<table>
<tr>
<td>Free upper end, single line
</td>
<td>In this topology, a single line is connected to the seafloor at its lower end and has a free upper end.</td>
</tr>
<tr>
<td>Wind turbine with floating substructure

</td>
<td>A wind turbine with floating substructure can be calculated in Riflex. More details please refer to Sima chapter.</td>
</tr>
<tr>
<td>Elastic contact

</td>
<td>Elastic contact can be calculated in Riflex. For more details please refer to the DeepC feature description.</td>
</tr>
</table>

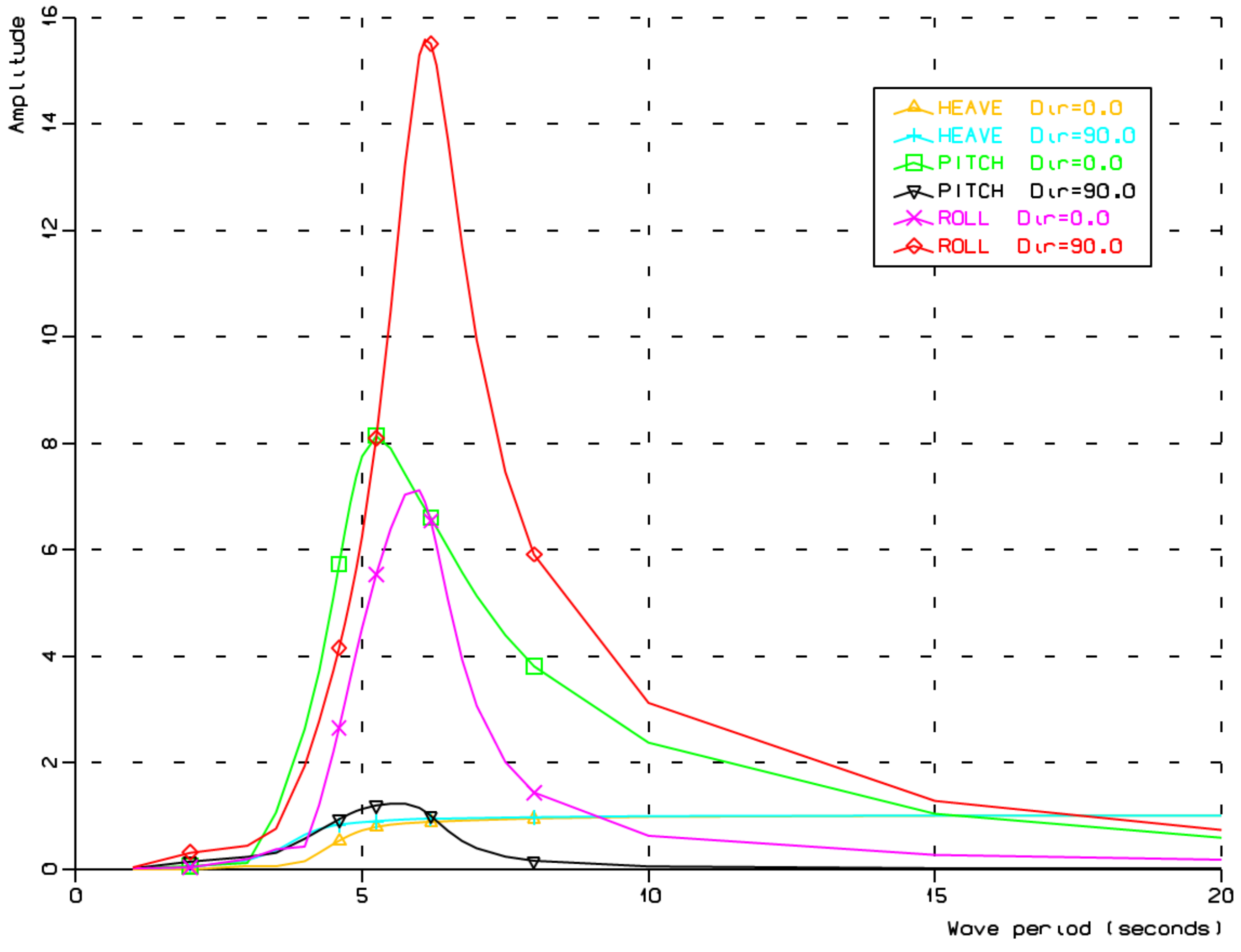
Postresp

POSTPROCESSOR FOR STATISTICAL RESPONSE CALCULATIONS

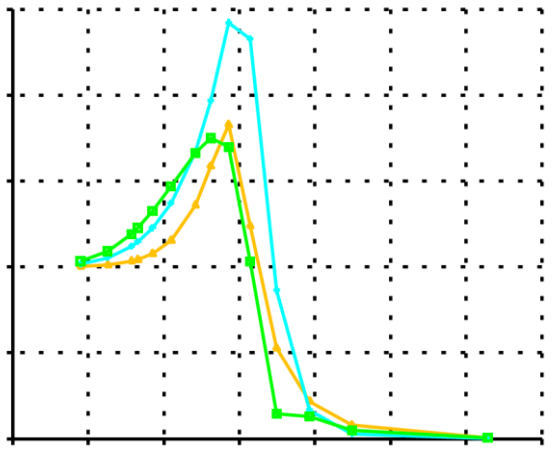
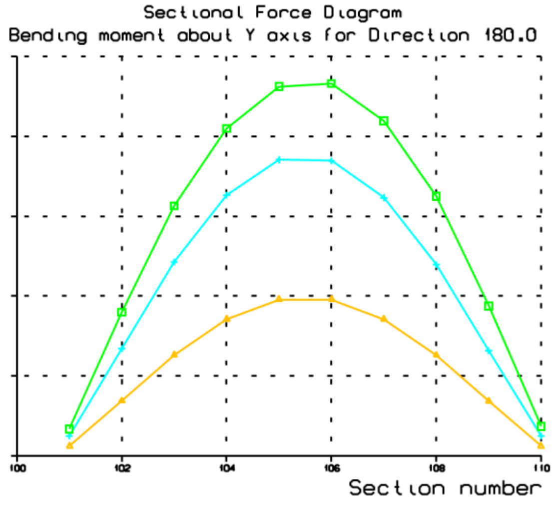
Last revised: January 08, 2019. Describing version 6.7-04.

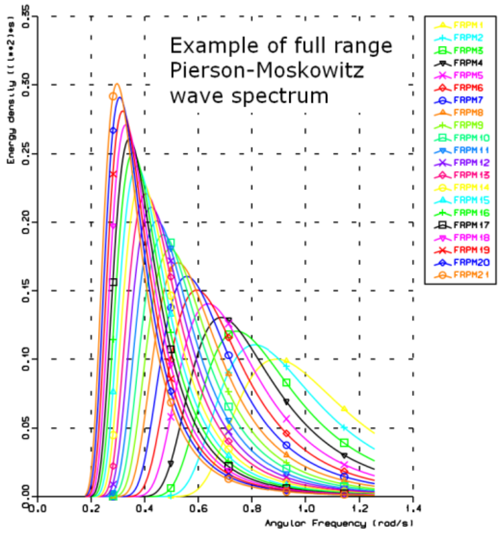
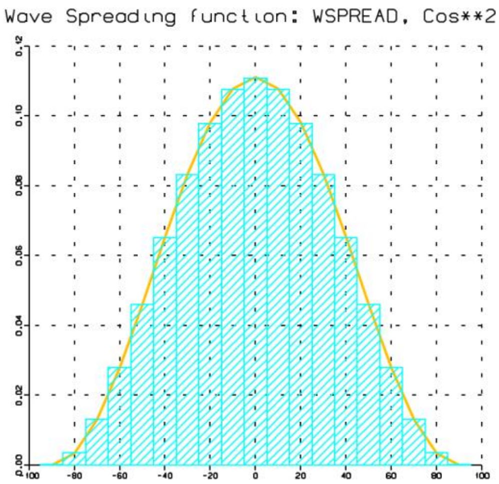
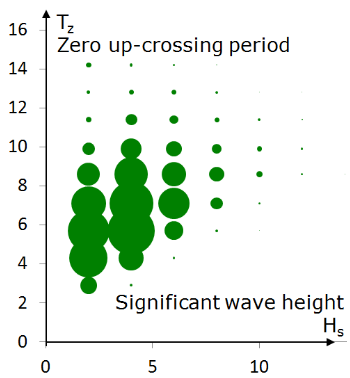
Postresp is an interactive graphic postprocessor for processing and presentation of responses in terms of transfer functions in the frequency domain. The transfer functions are usually generated by one of the hydrodynamic analysis programs in Sesam but they may also be transfer functions for any other kind of response.

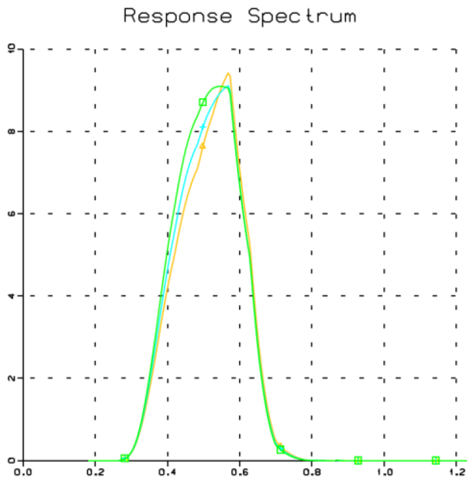
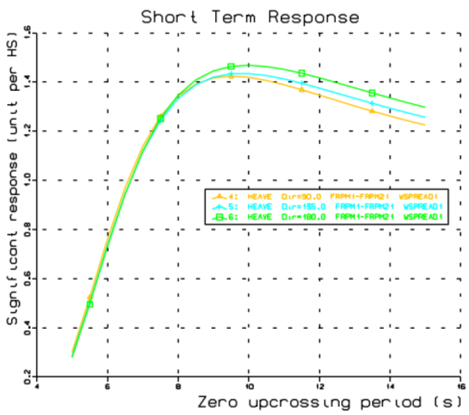
In this description, the term “transfer functions” is used. In Sesam the terms “response variables” and “response amplitude operators” are also used with the same meaning.

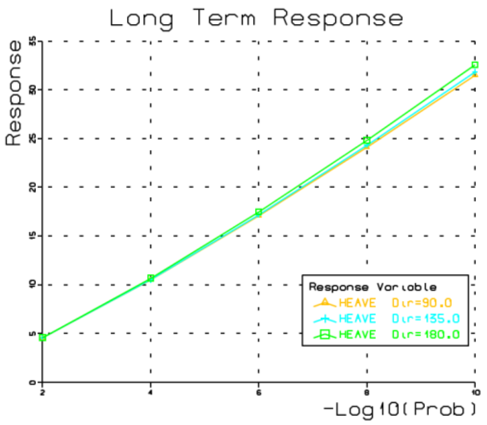
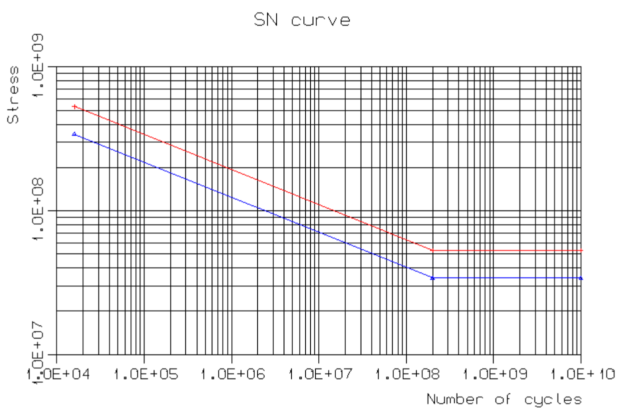


FEATURES OF POSTRESP

<table>
<tr><th>FEATURE</th><th>DESCRIPTION</th></tr>
<tr><td>Main features</td><td>The main features are:
• Display of transfer functions
• Calculation and display of response spectra
• Calculation and display of short term responses
• Calculation of short term statistics
• Calculation and display of long term responses
• Stochastic fatigue calculation</td></tr>
<tr><td>Response variables
</td><td>The transfer functions are normally read from a file produced by a Sesam program but they may also be typed in directly.
1st or 2nd order transfer functions may be combined either as standard motion combinations for displacement, velocity or acceleration, or by special combinations where the user is free to add transfer functions with scaling factors in any way.
The transfer functions may be printed, displayed and saved to a plot file.</td></tr>
<tr><td>Sectional forces

</td><td>Sectional forces and moments may be presented as a diagram, i.e. the force/moment variation along the axis of the structure.
This diagram corresponds to the force/moment diagram along the “beam” axis as presented by Cutres subsequent to a structural analysis.</td></tr>
</table>

<table>
<tr>
<td>Wave spectra

</td>
<td>Different wave spectra may be defined:
• Pierson-Moskowitz
• ISSC
• JONSWAP
• Torsethaugen
• Ochi-Hubble
• General-Gamma
• User specified
Wave spectra are used to calculate response spectra, short term response, long term response, workability analysis, second order statistics and calculation of stochastic fatigue.</td>
</tr>
<tr>
<td>Wave spreading

</td>
<td>The wave energy spreading functions are used when statistical calculations are required for short crested sea. The wave energy spreading function may be a $\cos^n(\Theta)$ function, where n is an integer value, e.g. $\cos^2(\Theta)$. A user specified wave spreading function may also be given.</td>
</tr>
<tr>
<td>Wave statistics – scatter diagram

</td>
<td>The wave statistics model describes the sea state conditions during a long-term period, and consists of zero upcrossing periods, T_z, and significant wave heights, H_s, and their probability of occurrence. The values may be given through an approach based on Nordenstrøm's theory or by specifying a scatter diagram directly.
Two standard scatter diagrams may automatically be generated:
• DNV-NA (DNV North Atlantic)
• DNV-WW (DNV World Wide trade)</td>
</tr>
</table>

Response spectra  	A response spectrum may be either an auto-spectrum or a co-spectrum. A response spectrum is generally a wave spectrum multiplied by the square of the transfer function for an auto-spectrum and multiplied by the cross function for a co-spectrum. There will be one response spectrum for each response available, main wave heading and each wave spectrum used.
Short term response  	The short-term response is calculated as the response of the structure based on an energy spectrum for a stationary sea state and the transfer function for the structure. It is defined as the mean of one third of the largest responses in the response spectrum and it is divided by the significant wave height.
Short term statistics	The short-term statistics may be tabulated. Two distribution methods are implemented: Rayleigh and Rice. Short term statistics are calculated for a given response spectrum based on three different inputs: • Probability of exceedance given a response level • Probability of exceedance and an estimate of the most probable largest response level given a sea state duration • Response level given a probability of exceedance

Long term response 	The long-term response calculation is based on either Nordenstrøm's model or a scatter diagram. Speed-reduction can be taken into account in long term response calculations. The print from the long-term calculation includes response levels for given probability levels, the Weibull parameters estimated when fitting the short-term parameters to a Weibull distribution and the response levels for up to 5 return periods. All of these are printed for each wave direction calculated and, if requested, with all wave directions included.
Equation of motion	The response variables for the motion of the structure are obtained by solving the equation of motion.
Workability analysis	Workability analysis may be done for a given response variable, wave direction and allowable double-amplitude response level.
Second order statistics	Second order statistics may be printed: the first four statistical moments, the mean, standard deviation, skewness and kurtosis of the system output. In addition, extreme levels satisfying a given probability of exceedance can be printed.
SN-curves 	Several SN-curves may be selected for the stochastic fatigue analysis: • API-X and API-XP • DNV-X and DNVC • NS

Stochastic fatigue calculation	Stochastic fatigue calculations are available: • Short term fatigue calculation based on short term duration of a given sea state and assuming Rayleigh distribution of the stress ranges • Long term fatigue calculation either based directly on a scatter diagram where Rayleigh distributions are assumed for each cell or based on a Weibull-fit from a long-term response calculation of the significant responses (stress ranges) of the cells.

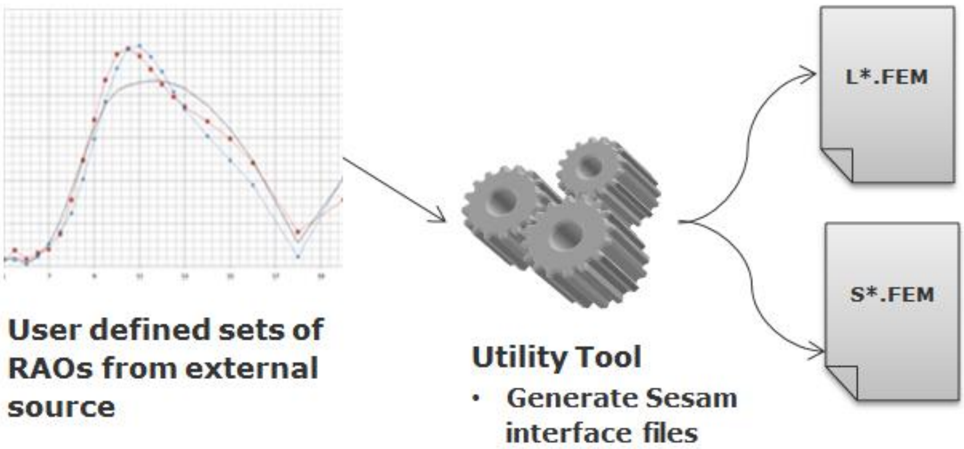
RAO

RESPONSE AMPLITUDE OPERATORS

Last revised: December 20, 2016. Describing version 1.0-00.

The console program RAO reads sets of user defined response amplitude operators, calculates the corresponding accelerations (real part and imaginary part) and writes the results to Sestra input files, S#.FEM and L#.FEM. Fluctuating gravity and forward speed (in terms of the Froude number) are accounted for.

The data on the output file have the same format and structure as the corresponding loads computed by the hydrodynamic solvers (Wadam and Wasim).



FEATURES OF RAO

FEATURE	DESCRIPTION
Load file creation 	Provided motion RAO’s are converted into accelerations and combined with gravity. The result is written on BRIGAC cards in the L-file
Sestra file creation 	The frequencies and headings corresponding to the load case numbers on the L-file are written to the S-file in the same way as what is done by the hydrodynamics solvers
Forward speed	The effect of forward speed is included in the creation of the loads and the data on the S-file

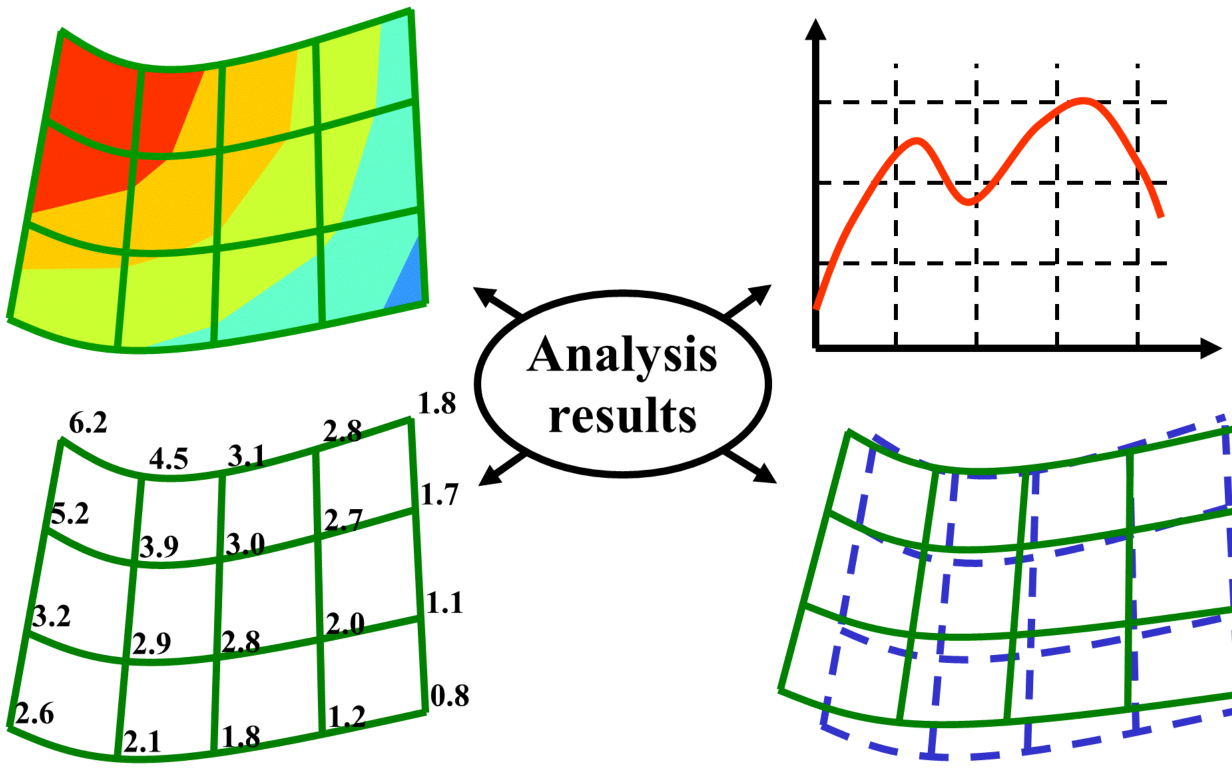
Xtract

POSTPROCESSOR FOR PRESENTATION, ANIMATION AND REPORTING OF RESULTS

Last revised: January 28, 2019. Describing version 5.2 (64-bit).

Xtract is the model and results visualisation program of Sesam. Xtract is owned and developed by Ceetron AS and marketed by DNV GL - Software. 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. Its intuitive and high-performance 3D graphics enables easy and efficient interactive rotation, zooming and panning of the model with results.

The user interface operations are logged (journaled) as command equivalents. This log may be edited and used as input to a new session for automated extraction and presentation of results.



FEATURES OF XTRACT

Structural analysis results

Xtract presents results for truss, beam, membrane, plate, shell and solid models. These results may be:

- Displacements, velocities and accelerations
- Forces
- Stresses

Xtract may create combinations and scan over result cases and combinations for highest and lowest values.

The graphic results presentation may be printed directly or saved to various graphic file formats for inclusion in reports. Tabulated data may be exported to file for importing into spreadsheets for customised results manipulation.

A 3D model with (animated) results may even be exported and opened in free tools for interactive 3D viewing:

- Xtract Viewer (free program)
- PowerPoint slideshow (with free GLview 3D Plugin embedded)
- Word (with free GLview 3D Plugin embedded)

This allows a Sesam user to prepare data and send to a non-Sesam user who may view the data in an Xtract-like environment.

Hydrodynamic analysis results

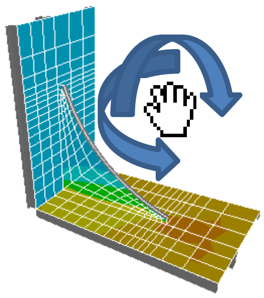
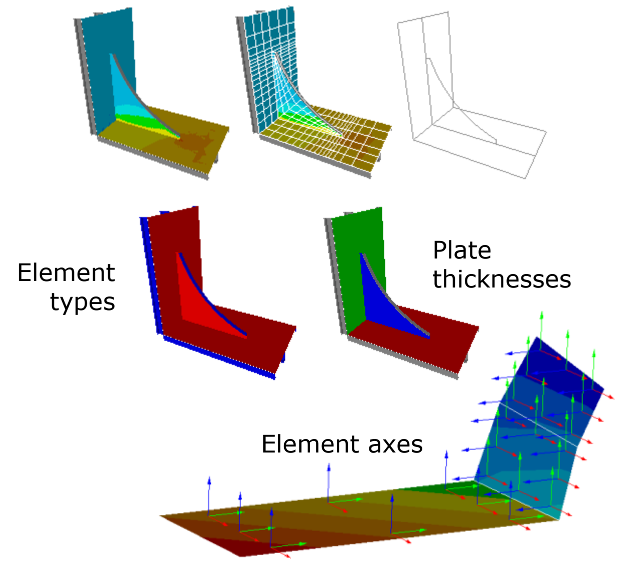
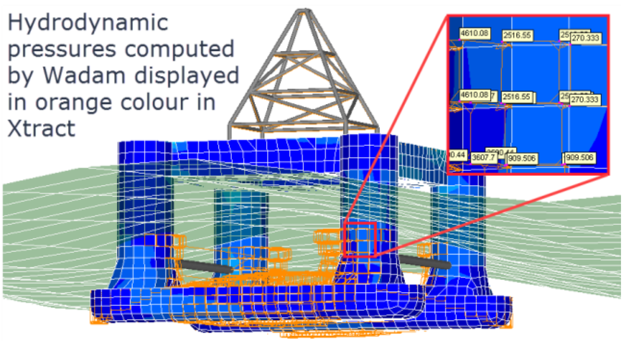
The animation feature of Xtract is especially useful for presenting results from hydrodynamic analyses. The motion of a vessel in waves may for example be animated superimposed by the corresponding hull stresses.

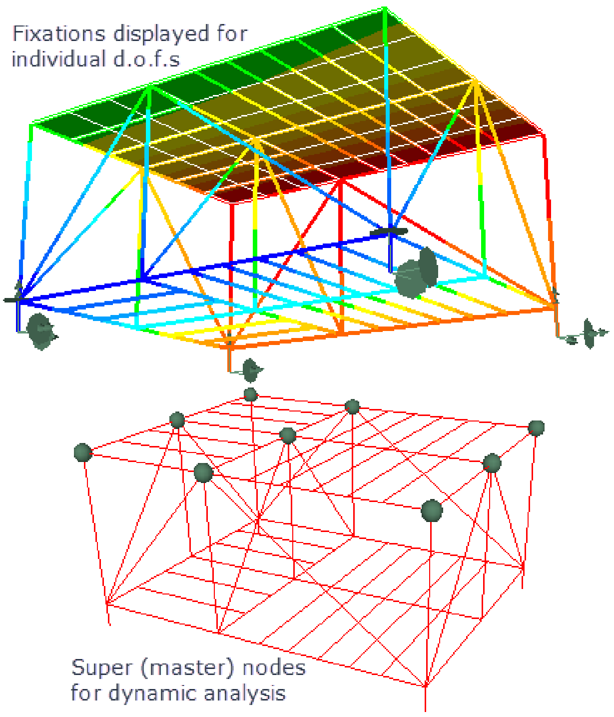
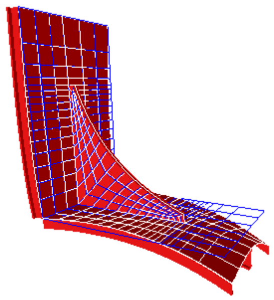
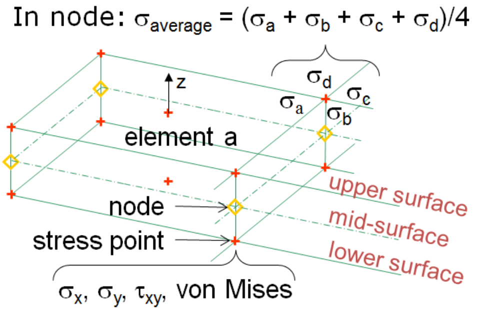
Other results

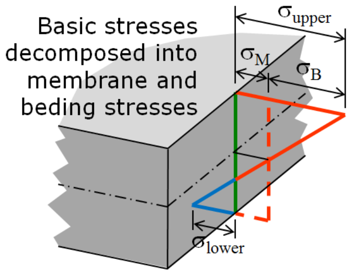
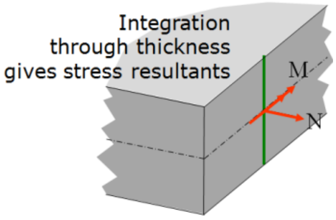
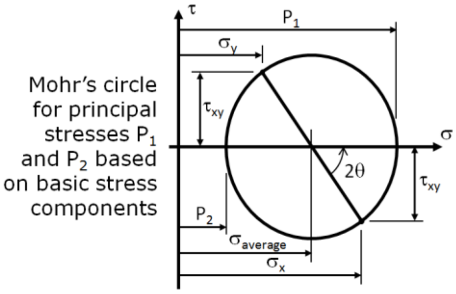
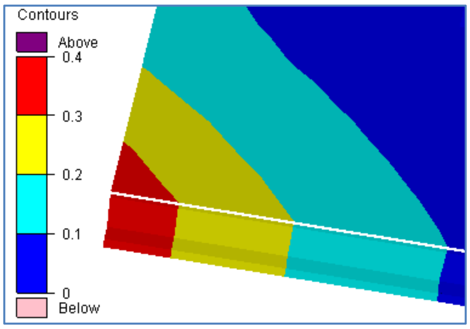
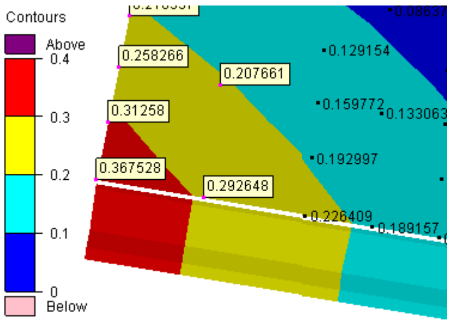
Various programs can store results on a VTF file that may be opened for postprocessing in Xtract:

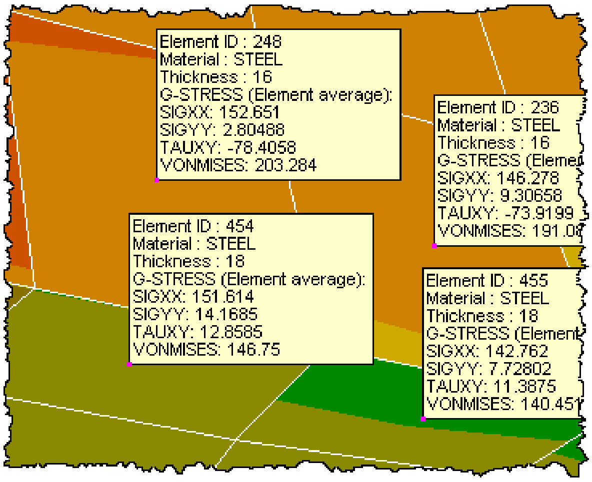
- Wasim: Time domain results for a vessel and sea surface
- Installjac: Time domain results for a launching/floating stability/upending process of a jacket
- Riflex: Time domain results for conductors, risers and anchor lines
- Stofat: Stochastic fatigue results for a stiffened plate
- ShellDesign (a product of Dr. Techn. Olav Olsen): Steel reinforced concrete shell design results

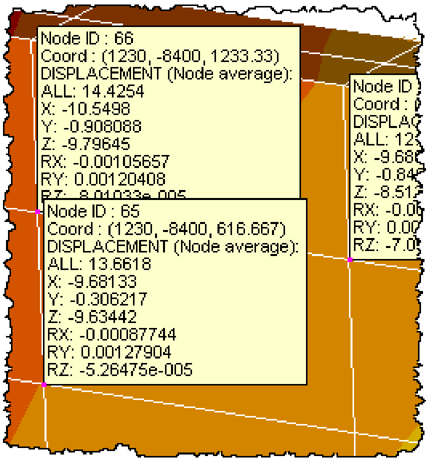
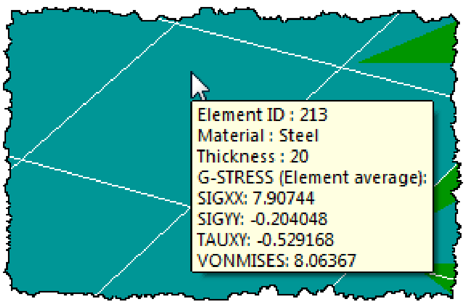
Main features

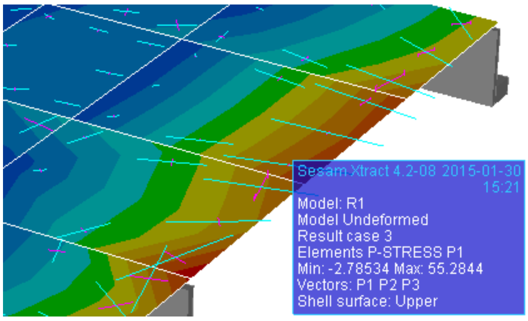
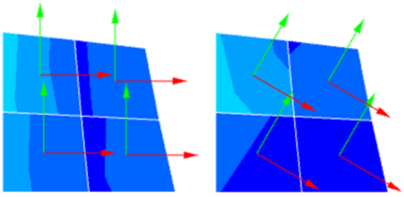
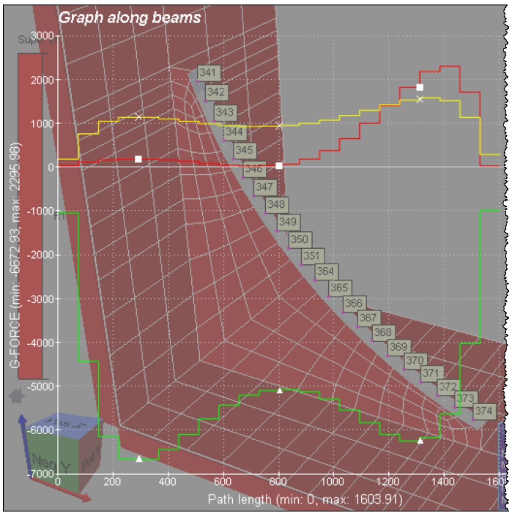
FEATURE	DESCRIPTION
Interactive rotation/zooming 	Quick and easy interactive rotation and zooming of model with results, during animation if relevant: • Rotating, optionally with fixed vertical axis • Zooming with buttons, mouse and wheel • Rubberband zooming • Panning • Fitting to frame • Function keys F5 - F9 for predefined view positions as in GeniE
Model display 	Model display and manipulation features include: • Surface with and without mesh plus outline view • Light sources giving shading effects or ambient light • Orthographic and perspective views • Colouring of elements according to element type, material type and shell/plate thickness • Adding model coordinate system, element axes, labels for node and element numbers • Retrieving sets created by the preprocessor and creating new sets
Load display 	Manually defined loads and hydrodynamic loads computed by Wajac and Wadam may be added to the displayed model. Optionally labelled with value. Dymic loads may be shown together with the animated response. • Point loads • Line loads • Surface loads, normal pressure and in component form

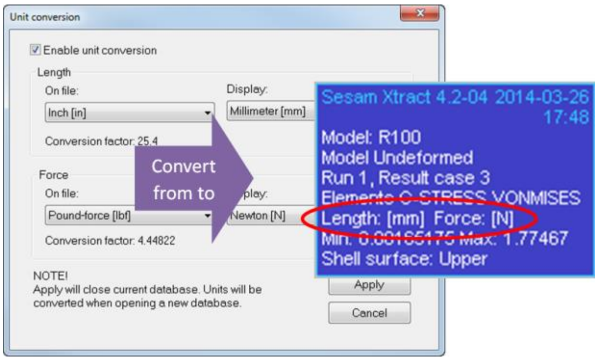
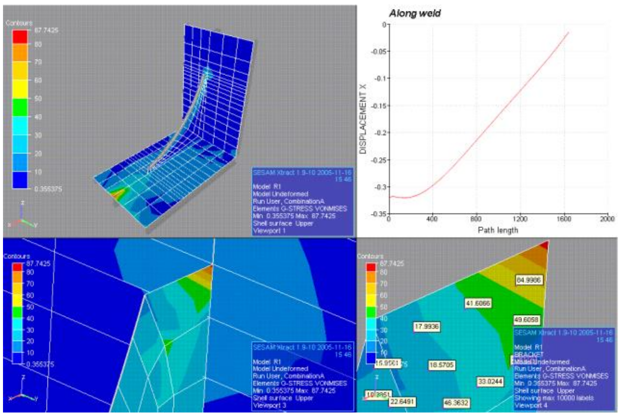
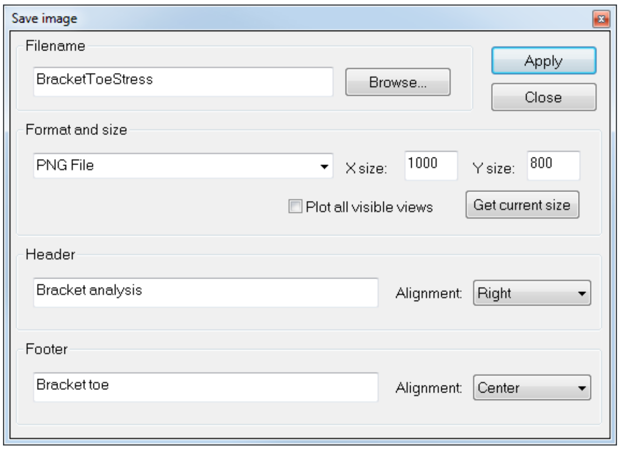
Bounadry condition display  	Boundary conditions may be added to the displayed model: • Fixed • Prescribed • Linearly dependent • Supernode (masternode for dynamic analysis) • Spring-to-ground
Deformed model 	Deformed model • Optionally together with undeformed wireframe • Optionally manual setting of scale factor
Basic stress components  	All basic stress components (σ_x, σ_y, τ_{xy}) plus von Mises stress. Stresses in: • Stress points • Averaged within elements, i.e. single value per element • Averaged in nodes • All above in upper (positive element z) and lower surface

Derived stresses   	Derived stresses are based on the basic stress components: • Basic stress components decomposed into membrane (σ_M) and bending (σ_B) stresses • Basic stresses integrated through thickness gives stress resultants (also termed direct stresses) • Principal stresses, P_1 is the highest stress in main stress direction, P2 is lowest stress perpendicular to main stress direction
Colour contouring 	Colour contouring (filling) for all types of results User control of the contour levels and legend: • Number of levels between min and max • Specified min and max • Selected values
Numerics in display 	Numeric values superimposed on whole model: • All values • Limited to ranges (min – max) • Only peak (maximum and minimum) values • Label with and without background box • Font size control • Number of digits control

<table>
<tr>
<td>Numerics in display for selected elements/nodes


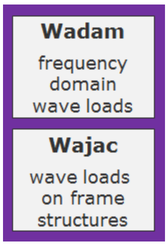

</td>
<td>Numeric values superimposed for selected:
Elements (Shift+Ctrl+click)
Nodes (Ctrl+click)</td>
</tr>
<tr>
<td>Mouseover/hover box

</td>
<td>Feedback label with node/element information including results shown by hovering the mouse.</td>
</tr>
</table>

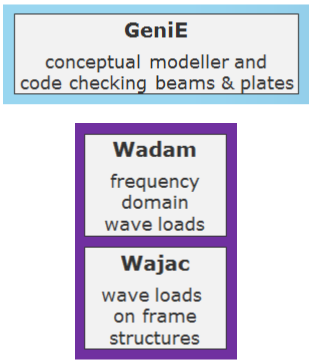
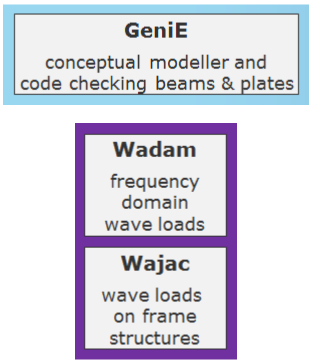
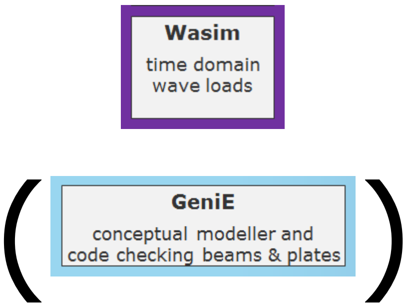
<table>
<tr>
<td>Vector presentation

</td>
<td>Vector presentation of.
Principal stresses P1, P2, P3 on top of colour contouring of one of them
Deformation</td>
</tr>
<tr>
<td>Stress transformation

Unrotated Rotated</td>
<td>Transformation of stresses to a user defined coordinate system may be done for shell, plate and membrane elements. The user gives a rotation about the normal axis.</td>
</tr>
<tr>
<td>Graph

</td>
<td>A graph in Xtract may be shown in a separate view or transparently superimposed on the model.
Graph can be shown for:
Nodes – results graphed over a series of nodes
Beams – results graphed along several beams
Occurrences – results in a node graphed over result cases, e.g. time</td>
</tr>
</table>

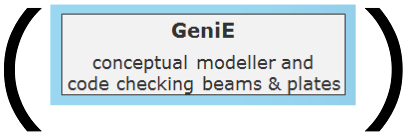
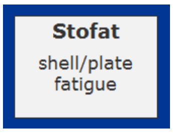
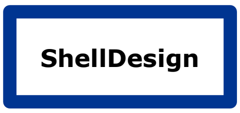
Unit conversion 	The consistent set of units on the results file may be converted into other consistent set, e.g. American units to SI units.
1, 2 and 4 viewports 	The display area may be split into one, two and four viewports.
Plot (saving graphics) 	The display may be saved to a graphics file: • Colour or grey shaded • Annotated with header and footer • Freely positioned text and pointers • Black (or coloured) background on screen optionally replaced by white in graphics file • Alternative graphics formats: PNG, GIF, TIFF and more

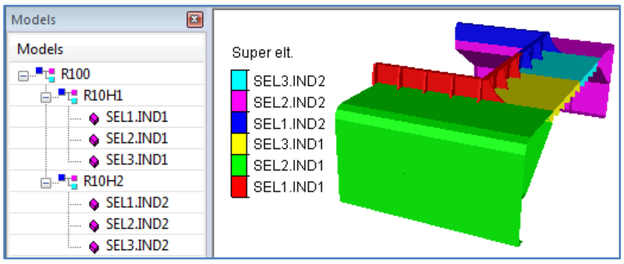
Models and results for presentation in Xtract

Xtract may present analysis results from different programs. A list of files for processing is provided below. The required program extension is also given.

FEATURE	DESCRIPTION
Structural Results Postprocessing R#.SIN file 	The R#.SIN (or R#.SIU or R#.SIF) file is typically generated by Sestra. Its contents are: • FE model • Nodal displacements • Beam forces • Element stresses Program extension required: STRU
Hydrodynamic & Structural Results Postprocessing R#.SIN + Gn.SIF 	The R#.SIN file: see above. The Gn.SIF file is typically generated by Wadam, Wajac or Prepost. Its contents are: • Transfer functions for rigid body motion of floating structure (Wadam) • Sea surface elevation (Wadam) • Transfer functions for base shear and overturning moments for fixed frame structure (Wajac) • Transfer functions for forces and stresses in selected elements (Prepost) Program extension required: STRU

Model & Loads Presentation T#.FEM + L#.FEM 	The T#.FEM file is generated by GeniE, Patran-Pre or Presel. Its contents are: • FE/panel model with nodes, elements, material, boundary conditions and loads (GeniE and Patran-Pre) • 2nd or higher level superelements (Presel) The L#.FEM file is typically generated by Wadam, Wasim, Wajac or Installjac. It is an appendix to the T#.FEM file and not a self-contained file. Its contents are: • Hydrodynamic line and pressure loads Program extension required: none
Hydrodynamic Results Postprocessing T#.FEM + L#.FEM + Gn.SIF 	All three files are described above. Program extension required: ANIM (to allow animation)
Animating Wasim Results file.VTF (+ 2D-series.VTF) (+ T#.FEM) 	VTF file produced by Wasim. The file contains: • Model of a vessel • Sea surface • Time domain motion results An additional VTF file containing 2D series data (component versus time) may optionally be opened. An additional T#.FEM file containing a FE model (extra geometry) may optionally be opened. Program extension required: ANIM

Animating Installjac Results file.VTF (+ 2D-series.VTF) 	VTF file produced by Installjac. The file contains: • Models of jacket, barge and sea surface • Time domain motion results of the launching/floating stability/upending process An additional VTF file containing 2D series data (component versus time) may optionally be opened. Program extension required: ANIM
Animating Riflex Results file.VTF (+ 2D-series.VTF) (+ T#.FEM)  	VTF file produced by Riflex. The file contains: • Time domain results for conductors, risers and anchor lines An additional VTF file containing 2D series data (component versus time) may optionally be opened. An additional T#.FEM file containing a FE model (extra geometry) may optionally be opened. Program extension required: ANIM
Presenting Stofat Results file.VTF 	VTF file produced by Stofat. The file contains: • Stiffened plate model with stochastic fatigue analysis results An additional VTF file containing 2D series data may optionally be opened. Program extension required: ANIM
Presenting ShellDesign results file.VTF 	VTF file produced by ShellDesign (a product of Dr. Techn. Olav Olsen). The file contains: • Steel reinforced concrete shell design results Program extension required: SHDS

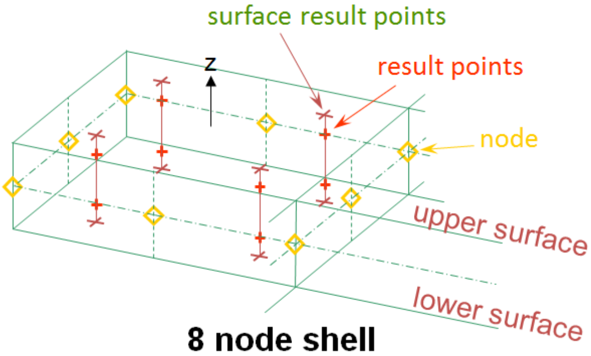
Superelement models 	Large superelement hierarchies as well as single superelement models are handled. All results may be presented in the same way for all superelements at any level. The way in which the model and results are presented may be controlled for the superelements individually. A tree overview of the superelement hierarchy makes it easy to select and display the desired superelement whether this is a first, top or intermediate level superelement.

Hierarchical organisation of results

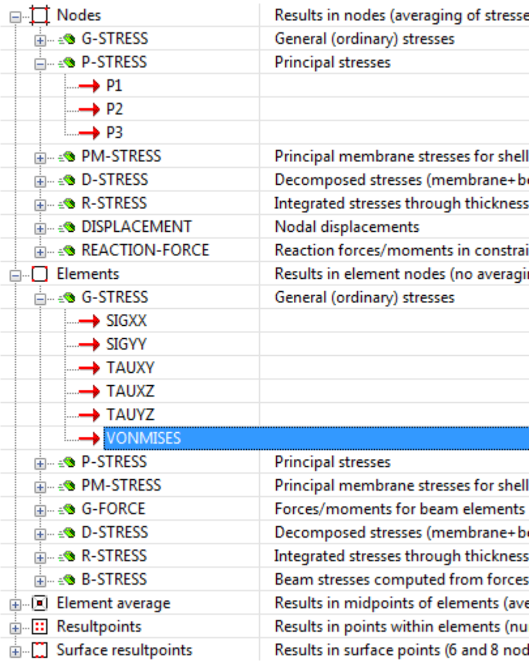
The structural analysis results stored on the Results Interface File are:

- Displacement (and optionally velocity and acceleration) components in the nodes
- Force and moment components for beam elements
- Stress components for membrane, shell and volume elements

Based on the above results data Xtract computes and presents results organised in a hierarchical manner: result positions > result attributes > result components. See details below.

FEATURE	DESCRIPTION
Result positions 	There are normally four or five results positions: • Nodes – the nodes of the model • Elements – the nodes of the individual elements • Element average – the midpoint of the elements • Resultpoints – the points in which stresses and forces are found on the results file • Surface resultpoints – available for 6 and 8 node shell elements only, these elements have their result points located inside the elements in the thickness direction, linear extrapolation to the surfaces is therefore provided

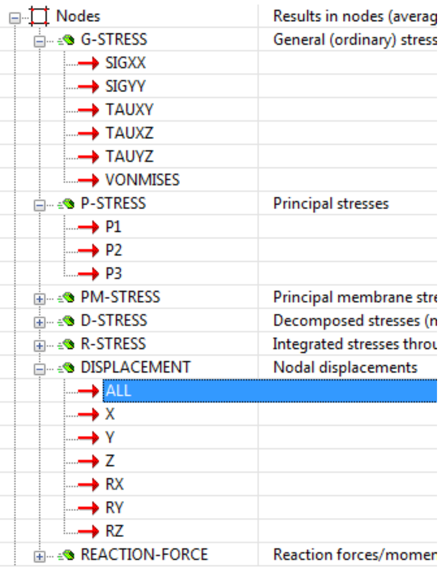
Result attributes



Each result position has its own set of result attributes being a selection of the following:

- DISPLACEMENT – nodal displacements
- VELOCITY – nodal velocities
- ACCELERATION – nodal accelerations
- REACTION-FORCE – reaction forces in supported (fixed or prescribed) nodes
- G-STRESS – general stresses (found on the results file)
- P-STRESS – principal stresses
- PM-STRESS – principal membrane stresses
- D-STRESS – decomposed stresses for shell elements
- R-STRESS – stresses integrated through the thickness
- G-FORCE – general forces for beam elements
- B-STRESS – beam stresses

Result components

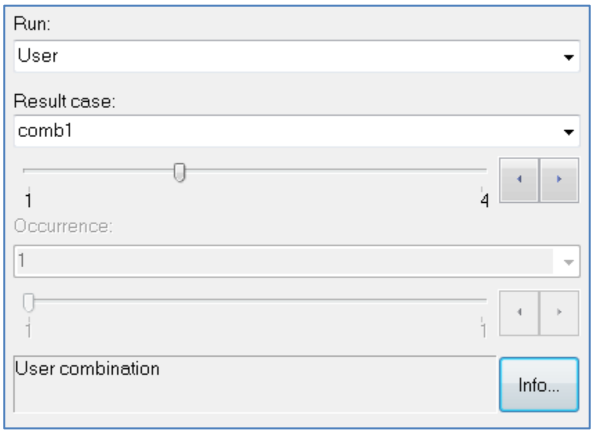
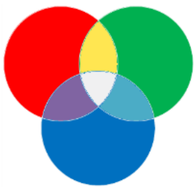
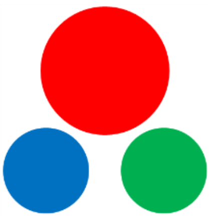


Each attribute has its own set of result components. Examples of components:

- DISPLACEMENT: X, Y, Z, RX, RY, RZ and ALL
- G-STRESS: SIGXX, SIGYY, TAUXY, etc.
- P-STRESS: P1, P2 and P3.
- PM-STRESS: P1 and P2.
- D-STRESS: SIGMX, SIGMY, TAUMXY, etc.
- G-FORCE: NXX, NXY, NXZ, MXX, etc.
- REACTION-FORCE: X-FORCE, Y-FORCE, Z-FORCE, RX-MOMENT, etc.

Result cases

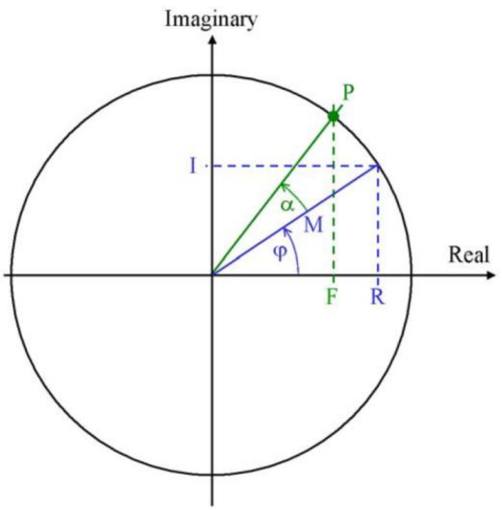
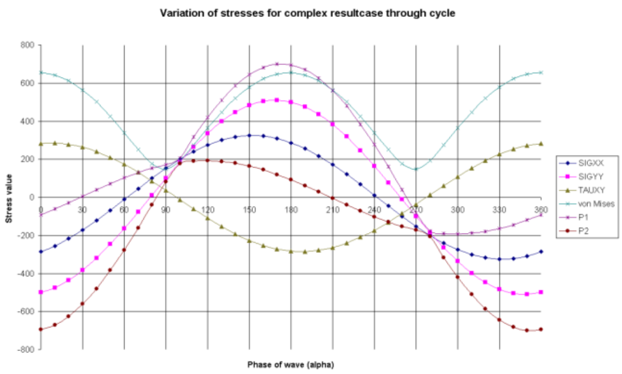
The structural analysis results stored on the Results Interface File are available as result cases and result combinations. Result combinations are one or more superimposed factorised result cases. The organisation is hierarchical: run number > result case > occurrence. See details below.

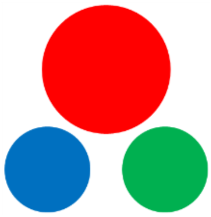
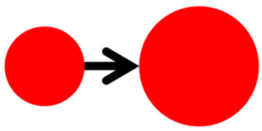
FEATURE	DESCRIPTION
Hierarchical organisation of result cases 	Result cases are organised in: • Run numbers: In most cases there will initially be only one run number (=1). Combinations, scan result cases and scaled complex result cases created in Xtract will be put in a run 'number' termed User. • Result cases: ○ Correspond to load cases or combinations for a static analysis ○ Correspond to wave directions for frequency domain analysis • Occurrences Correspond to frequencies for frequency domain analysis
Combinations 	Any number of result cases (but only one complex result case) may be included in a combination. The combination may comprise any selection of result positions and attributes (being available in the result cases included in the combination). All presentation options for a result case are also available for a combination.
Scanning 	Any number of result cases may be scaled and scanned for the (absolute) maximum or minimum of a selected result component. The data is put into a scan case available in parallel with result cases and combinations. For each result value put into the scan case an accompanying component (termed SCANINDEX) is created: the result case/occurrence owning the result value, i.e. the "winner" of the scan.

Complex results

There are specialised features for handling results from a frequency domain analysis. The complex results from such an analysis cannot be manipulated and presented in the same way as results from a plain static analysis. When presenting results for a complex case it needs to be evaluated for a specific phase of the wave. Furthermore, special considerations must be made when combining and scanning complex result cases.

Complex results are handled as follows:

<table>
<tr><th>FEATURE</th><th>DESCRIPTION</th></tr>
<tr><td>Evaluation of linear components

</td><td>When a linear result component is presented the complex result case must be evaluated:
The real (R) value (phase 0)
The imaginary (I) value (phase -90 = phase 270)
Magnitude = SQRT(R**2+I**2)
Phase shift = ATAN(I/R)
Given phase</td></tr>
<tr><td>Evaluation of non-linear components

</td><td>The components DISPLACEMENT ALL, von Mises stresses and principal stresses are non-linear meaning that they are not harmonic. While the linear components are harmonic these are not since they are non-linear combinations of the linear components. These may be evaluated:
Given phase
Maximum through cycle – stepping through 360 degrees to find maximum value
Phase angle of maximum</td></tr>
</table>

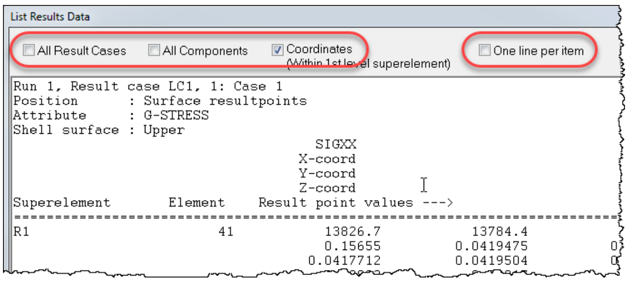
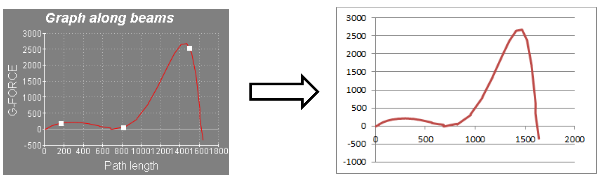
Combination 	A combination may include a single complex result case and any number of static (non-complex) result cases. The complex result case may be evaluated for the (absolute) maximum or minimum of any result component as well as for any user chosen phase of the wave.
Scanning 	A scanning for the (absolute) maximum or minimum value of a component may be done for any number of complex result cases and any number of static (non-complex) result cases. In this process, each complex result case is evaluated. The evaluation may be done for a given phase or for the same criterion as the scanning over the result cases. The latter evaluation is normally the desired one and involves that if you want to scan several result cases for the highest von Mises stress then the complex result cases need to be evaluated for the highest von Mises stress also.
Scaling 	A complex result case may be scaled. I.e. the real and imaginary parts are simply multiplied by a given factor.

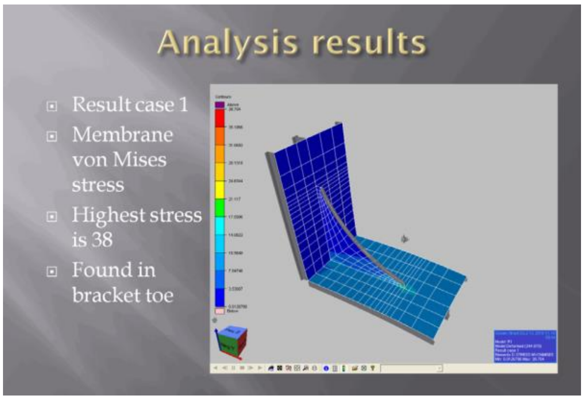
Animation of dynamic behaviour

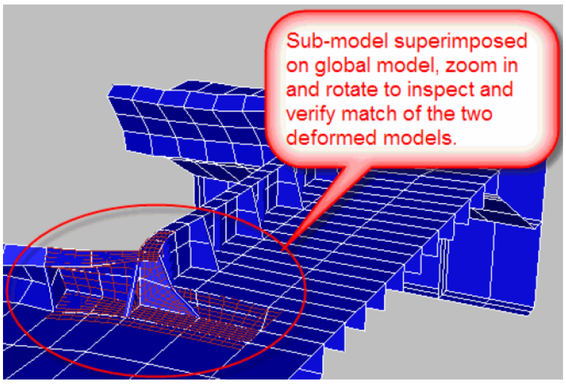
FEATURE	DESCRIPTION
Mode shape animation	The mode shape (eigenmodes) may be animated. The number of frames (time steps) through the 360-degree cycle is specified. The more frames the smoother animation.
Frequency domain animation	The harmonic motion of a structure may be animated. The step interval in degrees is specified. The shorter step the smoother animation.
Time domain animation	The time domain motion of a structure may be animated. Which time steps to include in the animation is specified.
Animation speed	The animation speed may be adjusted by setting frames per second.
Saving animation	An animation may be saved to alternative formats for replaying outside Xtract.

Exporting data for further processing and reporting

By exporting data from Xtract you have access to even more presentation features.

FEATURE	DESCRIPTION
Save tabulated data to file 	Tabulated nodal and element results may be saved to file for import into another program, e.g. Excel. Optionally the table may be unwrapped to one line per item with tab delimiters to ease further processing in Excel.
Export graph data 	Data graphed in Xtract may be exported to file for import into e.g. Excel.

<table>
<tr>
<td>3D model viewing outside Xtract


Xtract-like GUI in PowerPoint</td>
<td>A 3D model with results (and animation) may be exported to a VTFx file. The VTFx file may be opened for interactive 3D viewing in:
• Xtract Viewer (free program)
• PowerPoint slideshow (with free GLview 3D Plugin embedded)
• Word (with free GLview 3D Plugin embedded)
A Sesam user may thus prepare data and send to a non-Sesam user who may view the data in an Xtract-like environment.</td>
</tr>
<tr>
<td>Global model + sub-model

</td>
<td>Two 3D models may be exported to a common VTFx file. The feature is general but it is particularly useful for comparing a sub-model with the global model in a sub-modelling type of analysis, i.e. involving use of the program Submod. The deformed shape of the two models may for instance be overlaid to verify that the deformations at the outer boundary of the sub-model match those of the global model. Such match of deformations is an absolute demand in a sub-modelling analysis.</td>
</tr>
</table>

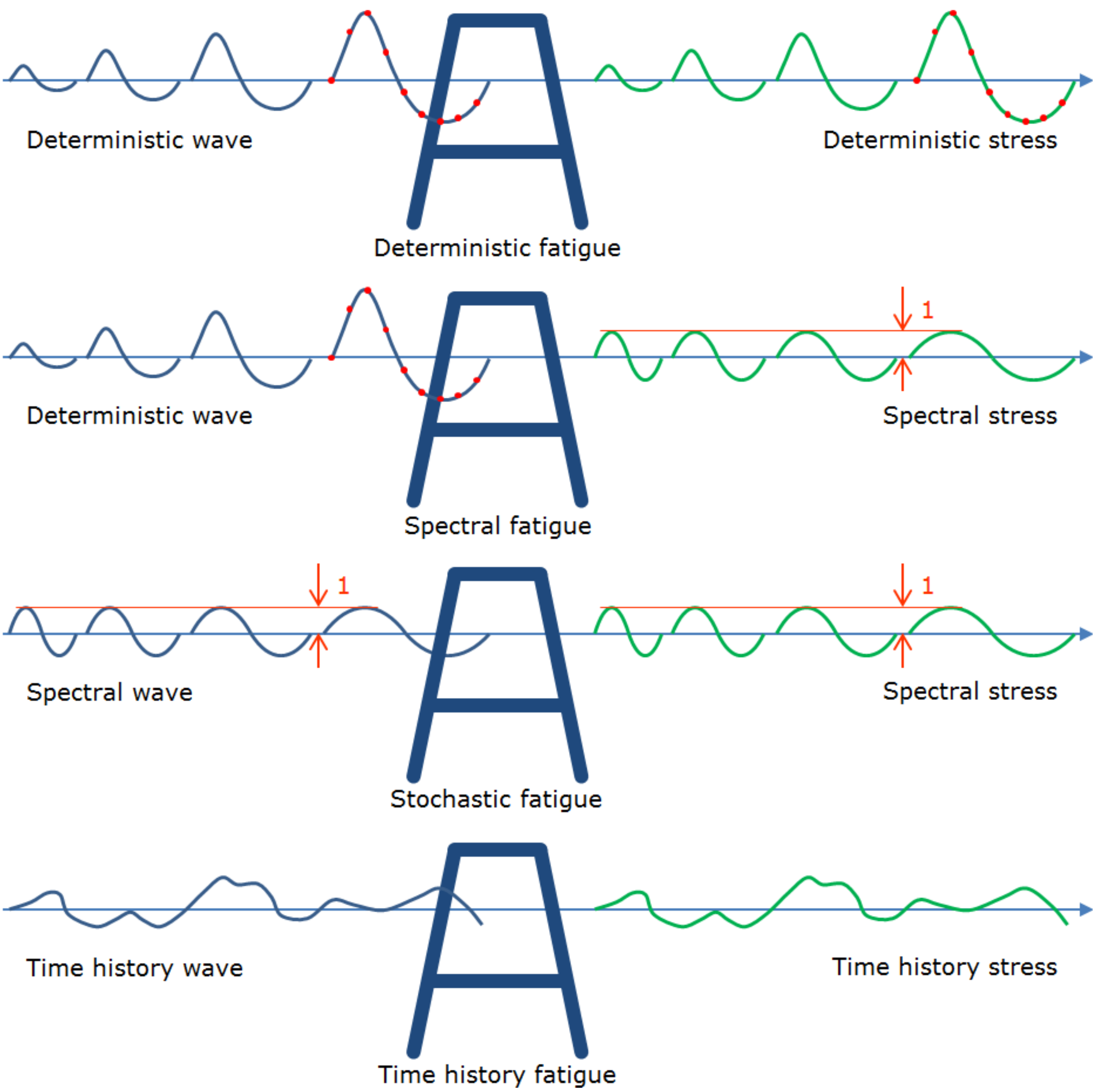
Framework

STEEL FRAME DESIGN

Last revised: October 5, 2018. Describing version 4.0-01.

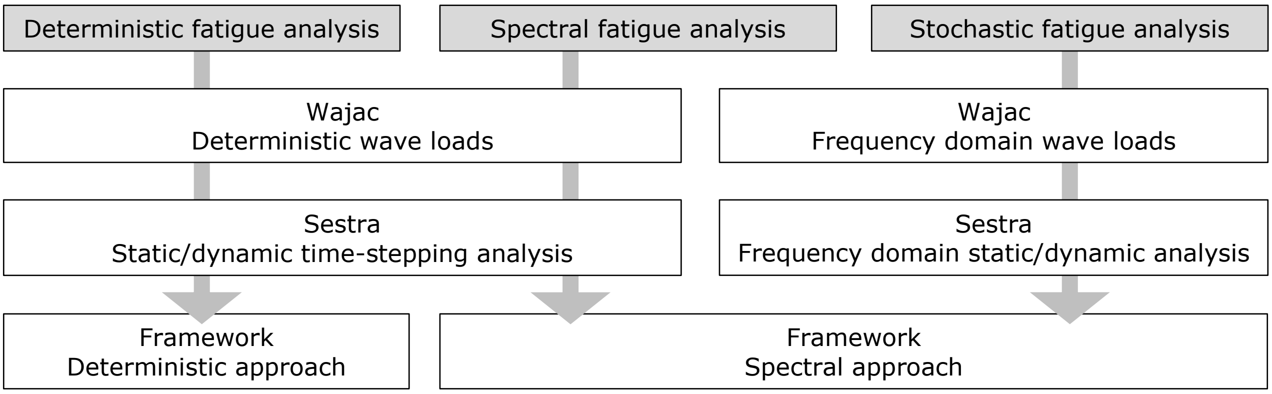
Framework is a postprocessor for frame structures, i.e. 2 node beam elements, with the following capabilities:

- Wave induced fatigue analysis
- Wind induced fatigue analysis
- Earthquake analysis including code checking

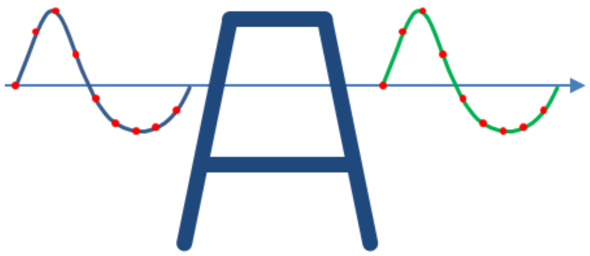


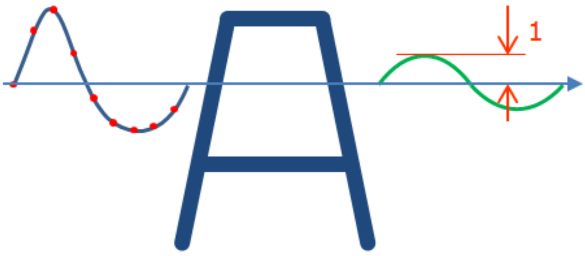
FEATURES OF FRAMEWORK

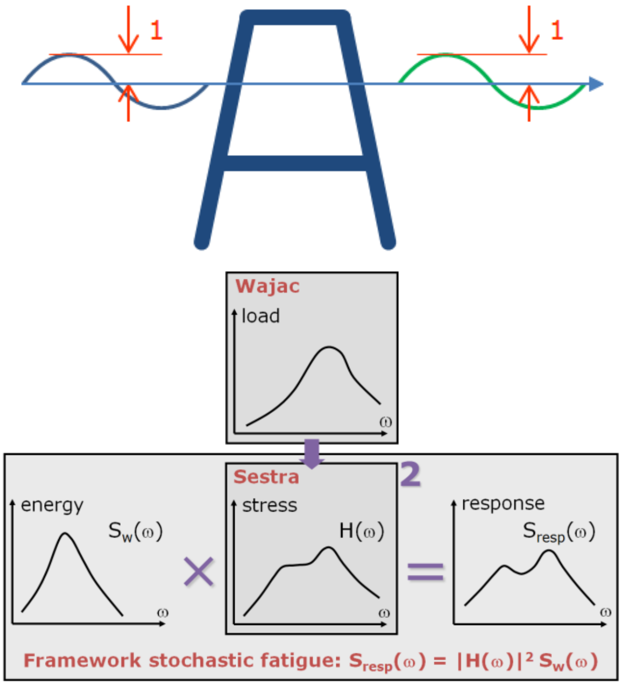
The processes in Sesam involved in the three most important wave induced types of fatigue analysis – the deterministic, spectral and stochastic – is illustrated by the figure below.



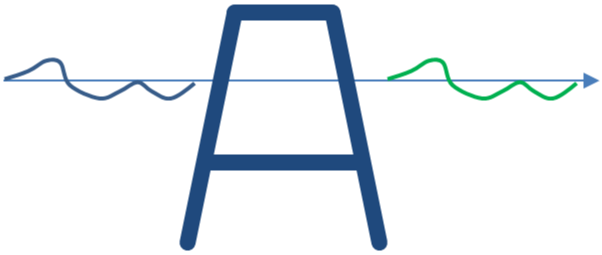
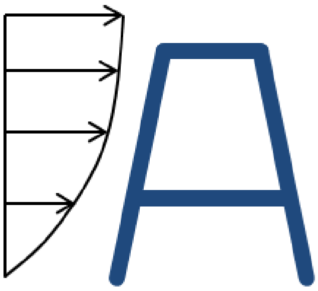
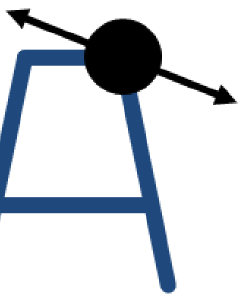
A description of the main features of Framework follows below.

FEATURE	DESCRIPTION
Wave induced deterministic fatigue 	Required analyses in: • Wajac: Several deterministic waves of different heights, lengths and directions • Sestra: Static analysis 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 is required in selecting the discrete waves to include in the analysis.

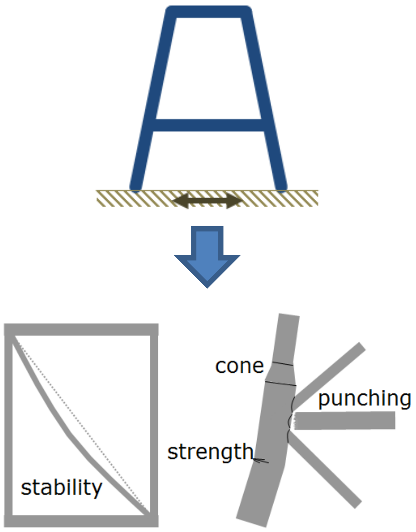
<table>
<tr>
<td>Wave induced spectral fatigue

</td>
<td>Required analyses in:
Wajac: Several deterministic waves of different heights, lengths and directions
Sestra: Static or dynamic analysis
Spectral stress transfer functions are established based on deterministic stress variations followed by a spectral fatigue analysis.

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.</td>
</tr>
<tr>
<td>Wave induced stochastic fatigue

</td>
<td>Required analyses in:
Wajac: Spectral (frequency domain) waves, unit amplitude waves of different frequency and direction
Sestra: Dynamic or quasi-static analysis
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 method properly represents the energy content of the sea-states.

This analysis is equivalent to the wave induced stochastic fatigue analysis available for welded plate and shell structures in Stofat.</td>
</tr>
</table>

Wave induced time history fatigue 	Required analyses in: • Wajac: Short term time domain simulation • Sestra Dynamic or static analysis 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. The rainflow counting method is employed.
Wind gust and VIV fatigue 	Required analyses in: • Wajac: Static wind loads • Sestra: Static analysis of wind loads • Sestra: Free vibration (eigenvalue) calculation • Merging of two results files from Sestra The wind fatigue analysis includes contribution from long term gust (buffeting) wind loads and vortex induced vibrations (VIV).
Direct deterministic (cyclic load) fatigue 	Fatigue analysis of a cyclic load of constant amplitude, e.g. caused by a rotating machinery. The number of cycles over a period of time is known.
Initial fatigue damage – accumulate fatigue damage $\text{Damage}_{\text{Transportation}} + \text{Damage}_{\text{In-place}} + \text{Damage}_{\text{Machinery}} + \ldots$	The fatigue analysis may be done with initial or part damage thereby computing accumulated damage. This allows fatigue damage for different stages such as transportation and in-place to be accumulated into a final run.

<table>
<tr>
<td>
Earthquake analysis



</td>
<td>
Required analyses in:

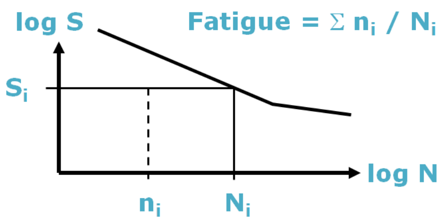
- Wajac: Added mass calculation if submerged in water
- Sestra: Eigenvalue analysis
- Sestra: Static analysis
- Merging of two results files from Sestra

The earthquake analysis is based on linear earthquake response techniques using modal combination rules. Modal combination rules available:

- Complete Quadratic Combination method; CQC
- Square Root Sum of Squares method; SRSS
- Naval Research Laboratory method; NRL
- ABSolute sum of each modal response; ABS
- The method recommended in API RP-2A; APIC

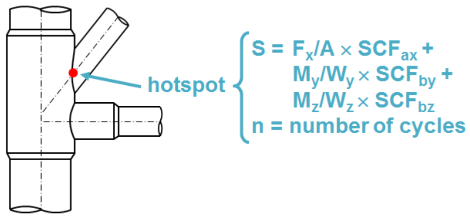
Earthquake results may be code checked according to:

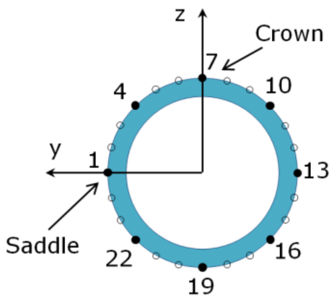
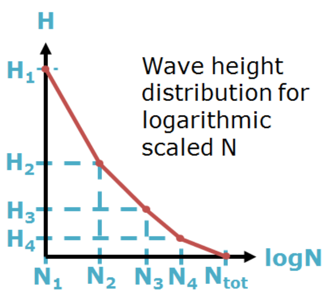
- API-AISC WSD and LRFD
- NPD
- NORSOK
- EUROCODE
</td>
</tr>
<tr>
<td>
SN-curves

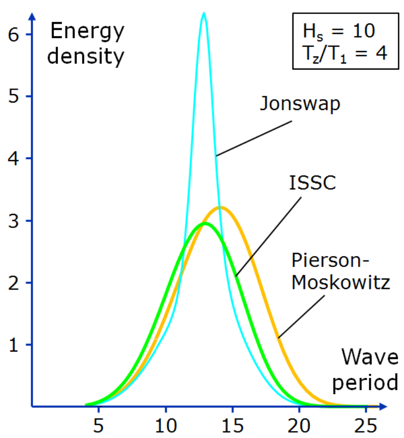


</td>
<td>
Library of SN-curves:

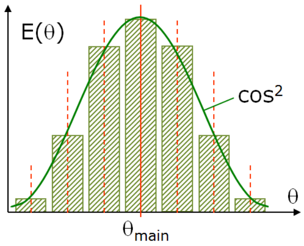
- API-X and API-X’
- DNV-X and DNVGL-RP-C203-2016
- NS3472
- NORSOK
- HSE
- ABS
- ISO
</td>
</tr>
</table>

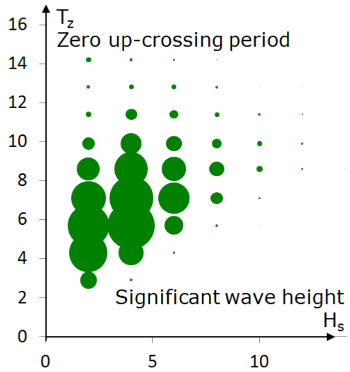
<table>
<tr>
<td>SCF – Stress Concentration Factors

</td>
<td>Manual specification and parametric (formulas):

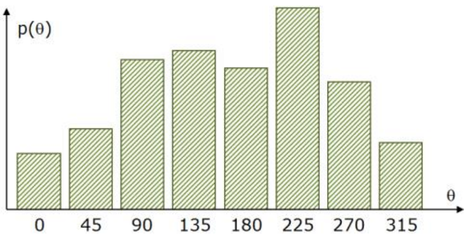
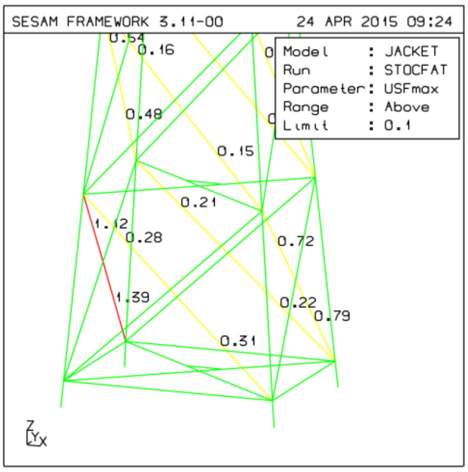
Kuang for YT, K, KT / Wordsworth and Smedley for X
Efthymiou for X, YT, K and KT
Lloyd’s Register for gap K and KT
Smedley and Fisher for SCF ring stiffened tubular joints
NORSOK for SCFs at butt welds and conical transitions
</td>
</tr>
<tr>
<td>Fatigue check points

</td>
<td>Eight points around the circumference of the tubular sections are checked for fatigue, both on the chord and brace sides of the weld. For each point an SCF may be manually or parametrically specified.</td>
</tr>
<tr>
<td>Wave height distribution

</td>
<td>The long-term wave height distribution is defined for deterministic fatigue analysis as:

Linear
Piecewise linear
</td>
</tr>
<tr>
<td>Wave spectra

</td>
<td>The wave spectra available for spectral and stochastic fatigue are:

Pierson-Moskowitz
Jonswap
General Gamma
ISSC
Ochi-Hubble
Torsethaugen
</td>
</tr>
</table>

<table>
<tr>
<td>Wave spreading
</td>
<td>Wave energy spreading functions may be defined to account for short crested sea in spectral and stochastic fatigue analyses:

- Cosine power, e.g. $\cos^2$
- User defined</td>
</tr>
<tr>
<td>Wave statistics – scatter diagram
</td>
<td>Wave statistics (scatter diagram) for long term sea state conditions are defined for spectral and stochastic fatigue:

- Manually
- Predefined (DNV-NA, DNV-WW)</td>
</tr>
<tr>
<td>Wave direction probability
</td>
<td>Wave direction probabilities are defined for spectral and stochastic fatigue to calculate contributions from the wave directions to the fatigue damage.</td>
</tr>
<tr>
<td>Graphic results presentation
</td>
<td>Results may be presented graphically:

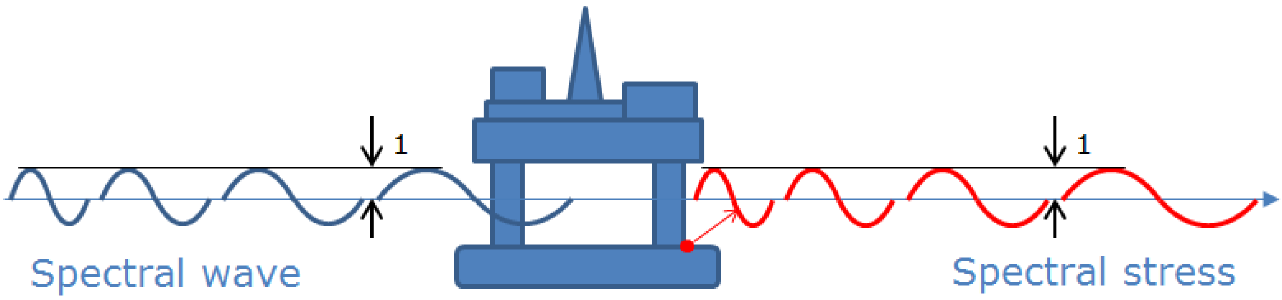
- Fatigue
- Code check (of earthquake)</td>
</tr>
</table>

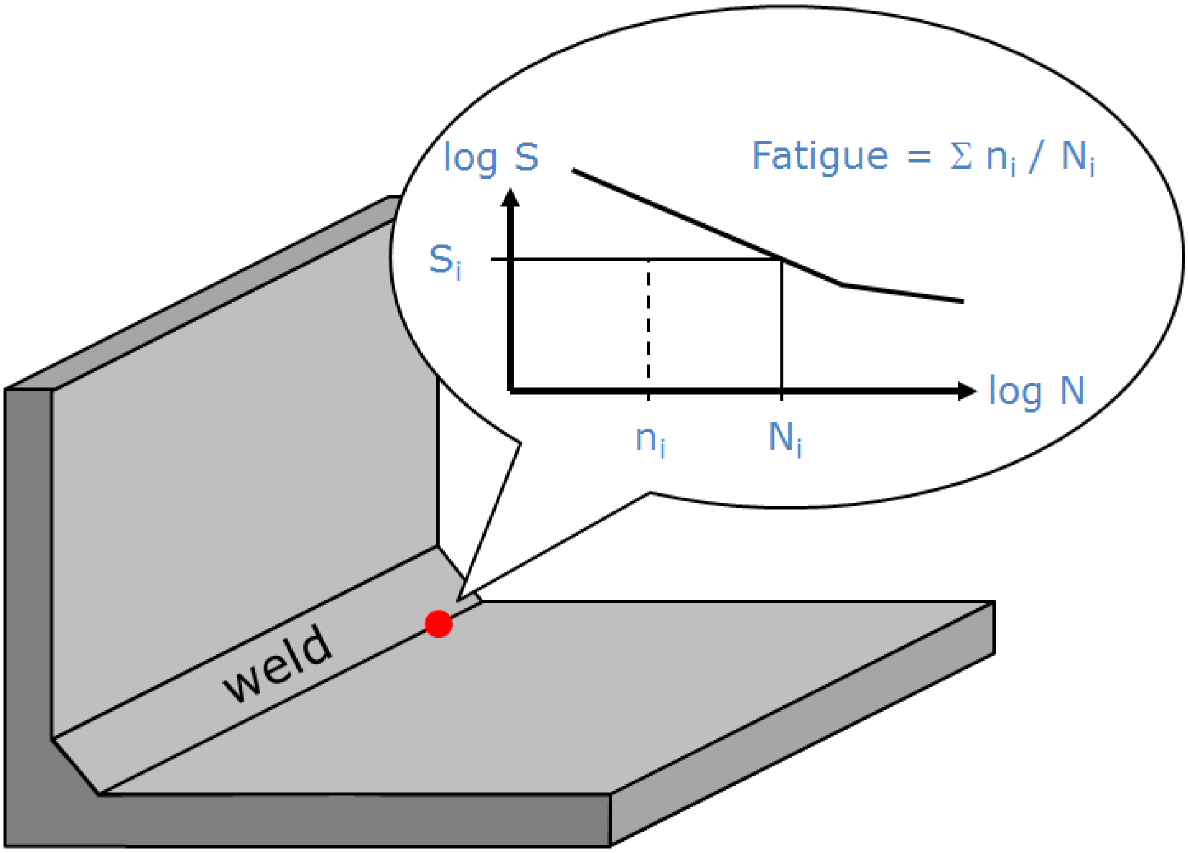
Stofat

FATIGUE ANALYSIS OF WELDED PLATES AND SHELLS

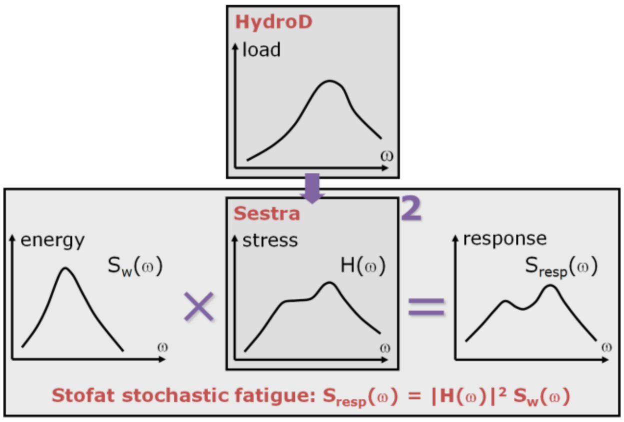
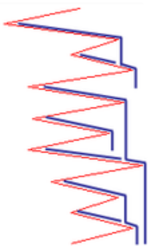
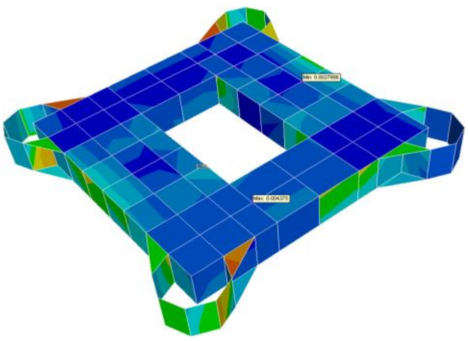
Last revised: January 14, 2019. Describing version 4.0-03.

Stofat is a postprocessor for wave induced stochastic fatigue analysis of welded shell and plate structures. The fatigue calculations are based on stress transfer functions in the frequency domain resulting from hydrodynamic pressure loads. Typically, HydroD and Sestra are involved in the process leading up to a Stofat fatigue analysis.

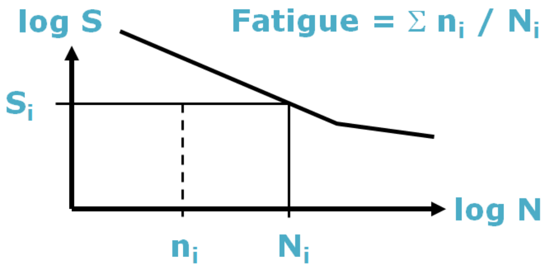




FEATURES OF STOFAT

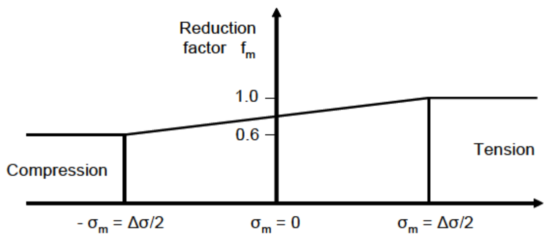
FEATURE	DESCRIPTION
Wave induced stochastic fatigue  	Required analyses in: • HydroD/Wadam: Spectral (frequency domain) waves, unit amplitude waves of different frequency and direction • Sestra: Dynamic or quasi-static analysis The stochastic fatigue analysis method is for dynamically sensitive and insensitive 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. This analysis is equivalent to the wave induced stochastic fatigue analysis available for frame structures in Framework.
Time domain fatigue based on time history load 	Required analyses in: • HydroD/Wasim: Time domain wave load analysis • Sestra: Time domain dynamic analysis The fatigue analysis is based on stress reversals determined by a rainflow-counting algorithm.
Time domain fatigue based on FFT	Based on Fast Fourier Transform (FFT) of spectral data and rainflow-counting.
Long term stress calculation 	The long-term stress calculation: • Maximum and minimum stress • Return period • Probability level • Exceedance • Etc.

<table>
<tr>
<td>SN-curves

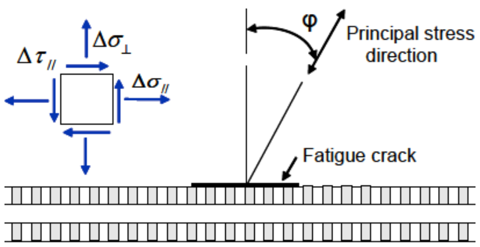


</td>
<td>Library of SN-curves:

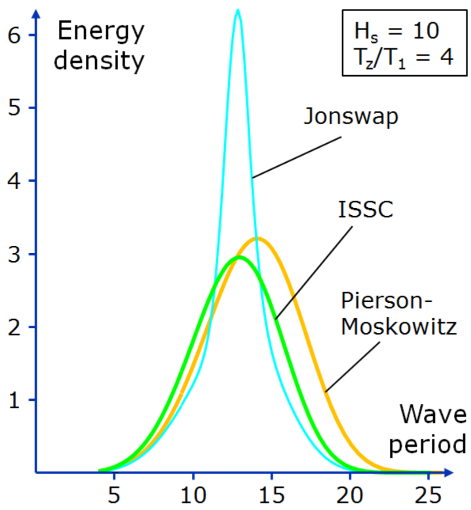
- API-X and API-X’
- DNV-X and DNV RP-C203-2010, etc.
- NS3472
- NORSOK
- HSE
- ABS
</td>
</tr>
<tr>
<td>Static stress accounted for



</td>
<td>Static stress from still water is accounted for according to DNV Classification Notes No. 30.7. I.e. tension increases and compression reduces fatigue damage.</td>
</tr>
<tr>
<td>Principal stress direction

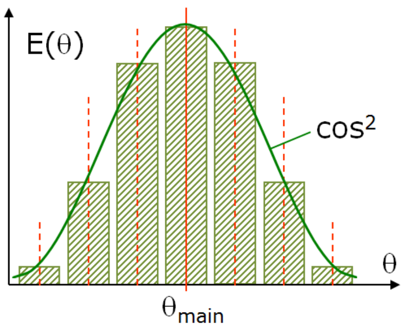
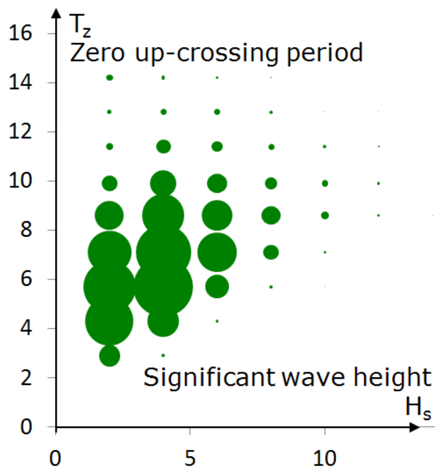
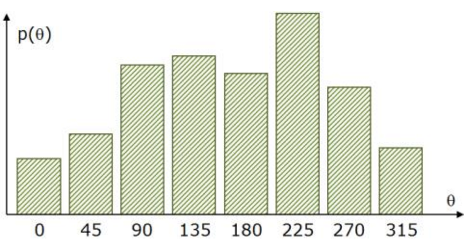
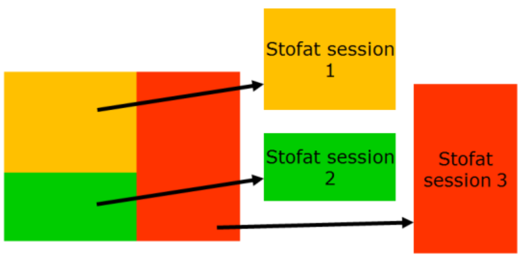


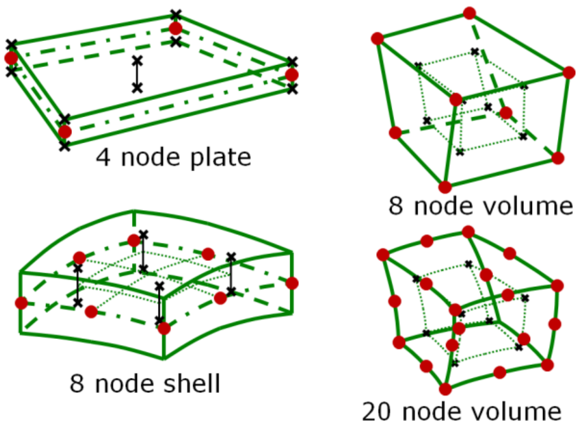
</td>
<td>Fatigue analysis is based on the principal stress within ±45º of the weld normal line.</td>
</tr>
<tr>
<td>Wave spectra

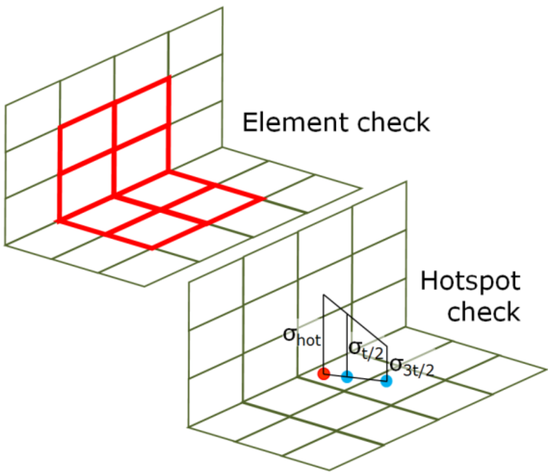


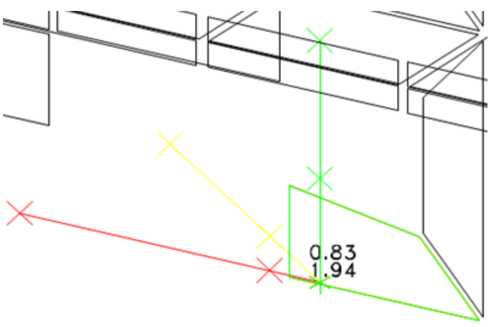
</td>
<td>The wave spectra available are:

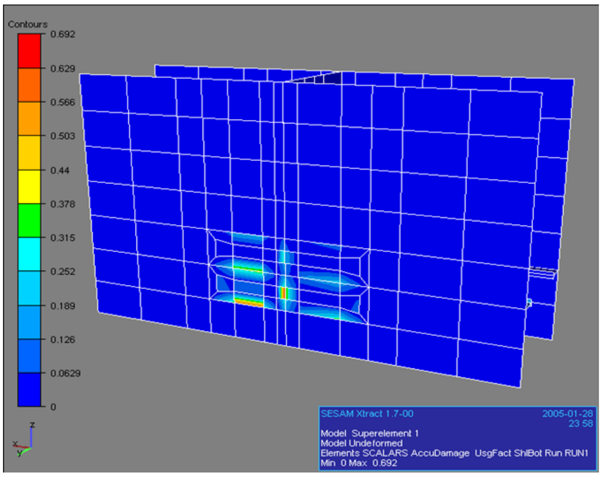
- Pierson-Moskowitz
- Jonswap
- General Gamma
- ISSC
- Ochi-Hubble
- Torsethaugen
</td>
</tr>
</table>

Wave spreading 	Wave energy spreading functions may be defined to account for short crested sea: • Cosine power, e.g. $\cos^2$ • User defined
Scatter diagram 	Wave statistics (scatter diagram) for long term sea state conditions are defined: • Manually • Predefined (DNV-NA, DNV-WW)
Wave direction probability 	Wave direction probabilities are defined for spectral and stochastic fatigue to calculate contributions from the wave directions to the fatigue damage. This also accounts for the effect of forward speed of a moving body relative to the sea.
Superelement model accounted for 	A model split into superelements may be handled by analysing the different superelements in different sessions.

Plate/shell and solid elements 	The following elements may be handled:
	• Plate/shell elements
	  ○ Flat 4 node plate
	  ○ Curved 8 node shell
	• Solid elements
	  ○ 8 node brick
	  ○ 20 node brick

Fatigue check types 	There are two fatigue check types available:
	• Element check – screening type of check
	  ○ Stress points
	  ○ Surface points
	  ○ Corner points
	  ○ Membrane points
	  ○ Centre points
	• Hotspot check – detail check
	  ○ Based on interpolation points defined by coordinates or nodes

Graphic results presentation in Stofat 	Both element and hotspot fatigue check results may be presented graphically in Stofat.

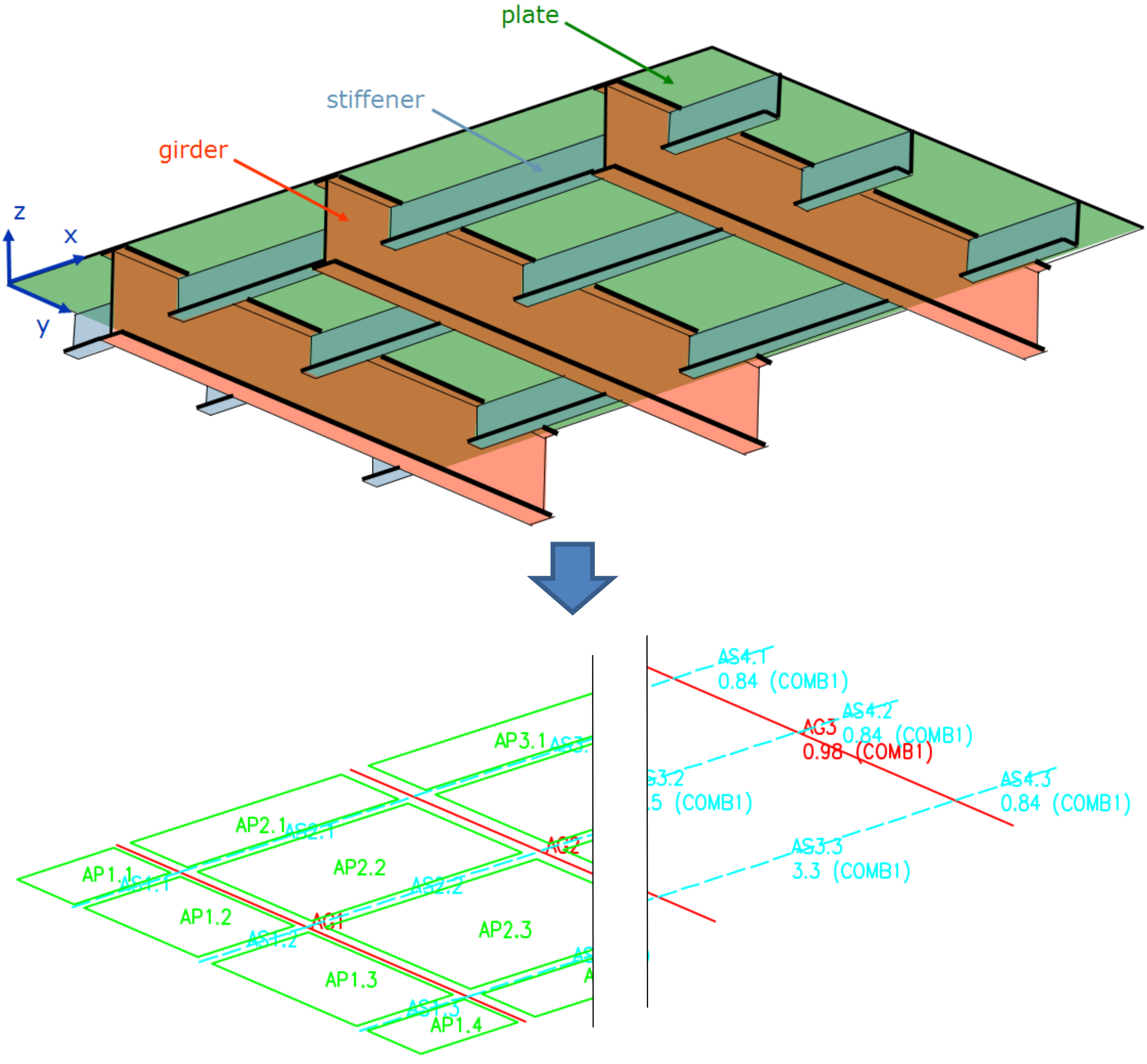
<table>
<tr>
<td>Graphic results presentation in Xtract

</td>
<td>Through a VTF file element fatigue check results as well as long term response results may be sent to Xtract for graphic presentation.</td>
</tr>
</table>

Platework

STIFFENED STEEL PLATE CODE CHECKING

Last revised: January 8, 2018. Describing version 1.9-00.

Platework is an interactive program with for code checking of stiffened plate structures according to API, DNV GL and NPD rules.

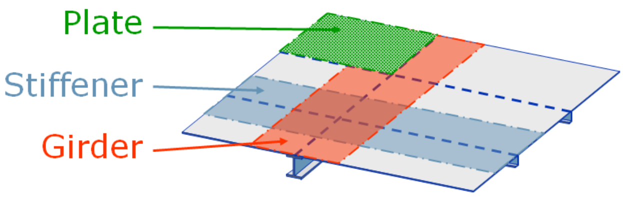


Display of capacity model

Display of code check results

FEATURES OF PLATEWORK

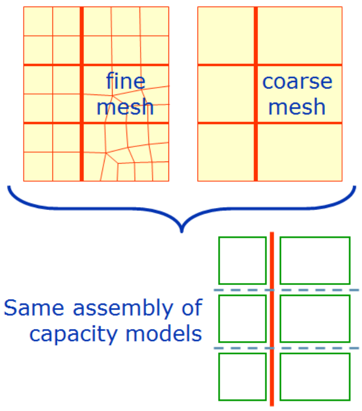
<table>
<tr><th>FEATURE</th><th>DESCRIPTION</th></tr>
<tr><td>Independent or FE use</td><td>In addition to use in conjunction with a FE analysis the program can be used as an independent tool to assess stiffened plates with given loads.</td></tr>
<tr><td>Capacity models



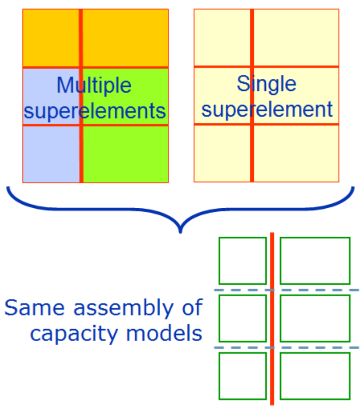
</td><td>The FE model is fully or semi-automatically split into an assembly of capacity models:

- Simple unstiffened plate
- Stiffener
- Girder
- Uniaxially and orthogonally stiffened panel (API)

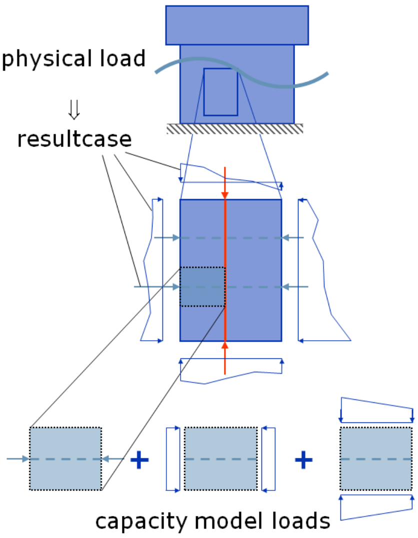
Capacity models may be displayed as shown on the previous page.</td></tr>
<tr><td>Capacity models for any FE mesh



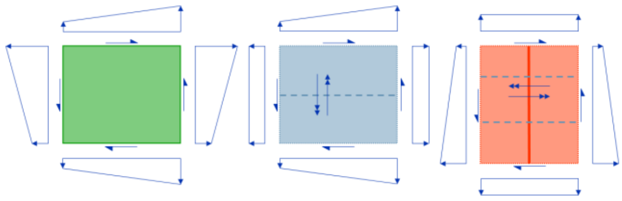
</td><td>The capacity model creation and code checking is independent of the FE mesh. The only difference is that a finer mesh will produce more accurate code checking results.</td></tr>
<tr><td>Single or multiple superelements



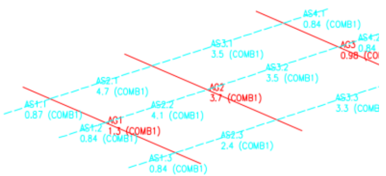
</td><td>The capacity model creation and code checking is independent of division into superelements.</td></tr>
</table>

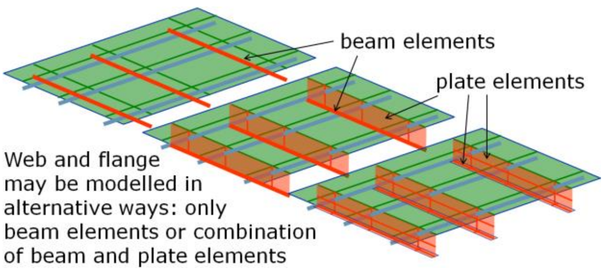
<table>
<tr>
<td>Capacity model loads
</td>
<td>Each capacity model, plate, stiffener and girder, has its set of capacity model loads on which the code checking is performed.

The capacity model loads are automatically established for all result cases.

Capacity model loads:
</td>
</tr>
<tr>
<td>Code checking standards</td>
<td>API – American Petroleum Institute

DNV – DNV GL

NPD – Norwegian Petroleum Directorate</td>
</tr>
<tr>
<td>Graphic results presentation
</td>
<td>The code check results may be presented graphically</td>
</tr>
<tr>
<td>Tabulated results presentation
+--+
! Sorting Parameter: UCMAX ! Max Entries: UNLIMITED !
! Sorting Order : DECREASING ! Max Value : UNLIMITED !
! ! Min Value : 5.000E-01 !
+--+
Status UCmax LBstat Res-Name L-stat Phas Capacity-Model Type

**-PIF2 3.08 2 E-ULS AS2.2 STF
**-PIF2 2.94 2 E-ULS AS2.1 STF
**-PIF2 1.77 5 E-ULS AS2.2 STF</td>
<td>Tabulated results may be sorted in several ways to focus on the results of interest.</td>
</tr>
<tr>
<td>Alternative modelling of stiffeners
</td>
<td>In the FE model the stiffeners may be modelled as:

• Beams
• Plate elements for web and beams for flange
• Plate elements for both web and flange

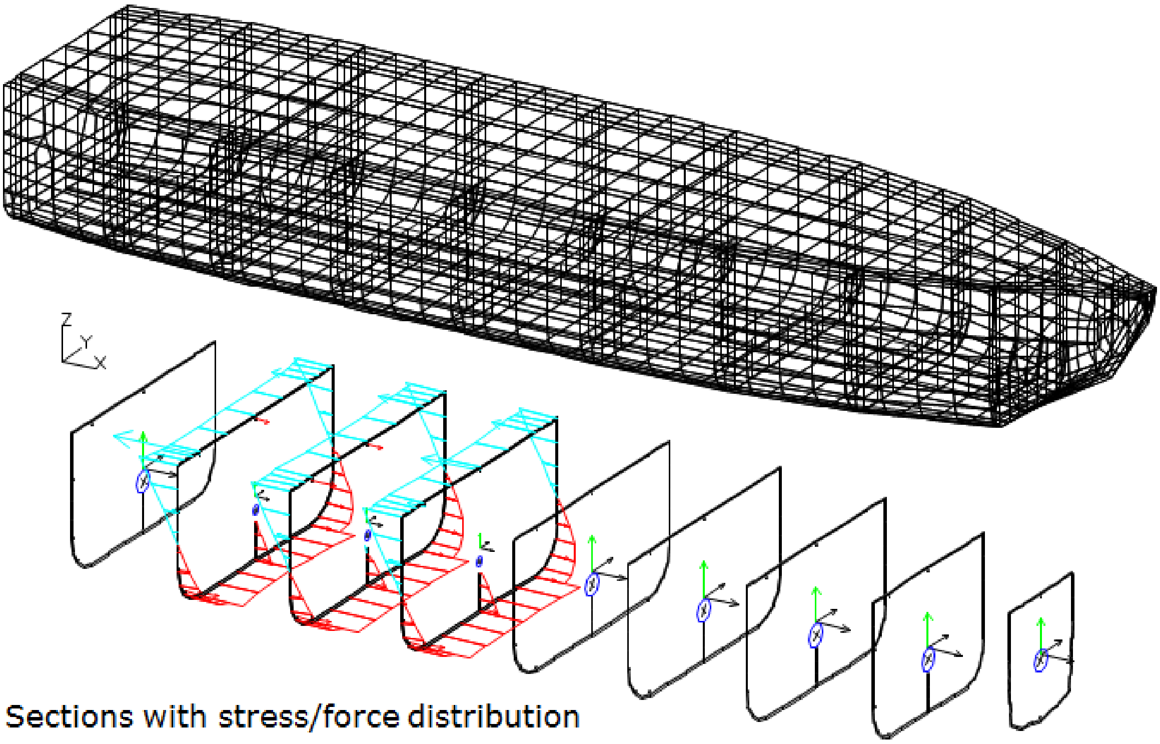
The same capacity model may be created and code checked for all three cases.</td>
</tr>
</table>

Cutres

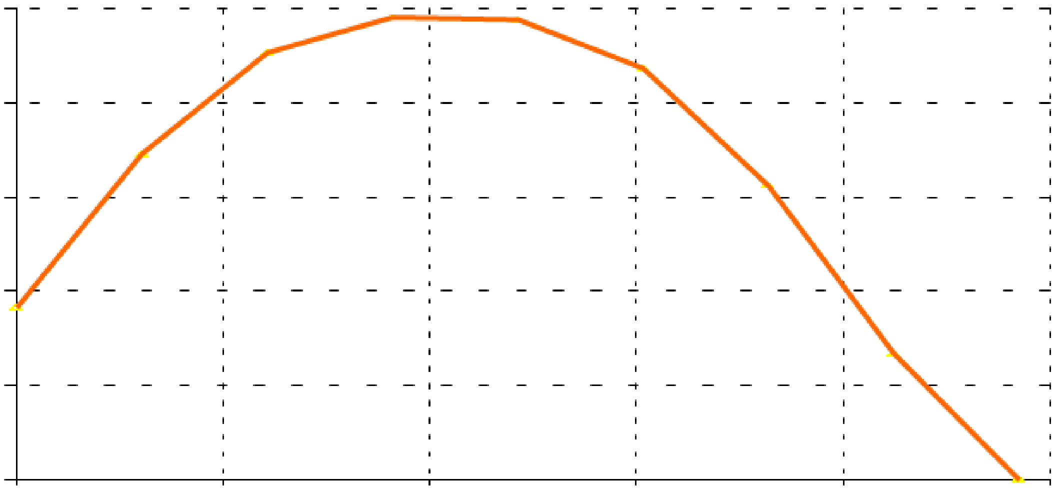
PRESENTATION OF SECTIONAL RESULTS

Last revised: April 23, 2018. Describing version 1.5-03.

Cutres is a postprocessor for presentation of results in terms of stresses, forces and moments in user defined sections through a FE shell/plate model with beam stiffeners. This is primarily relevant for oblong structures that may be regarded as beams, e.g. ship type structures and pontoons of semi-submersibles.

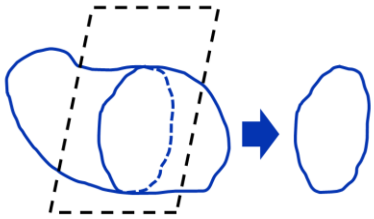
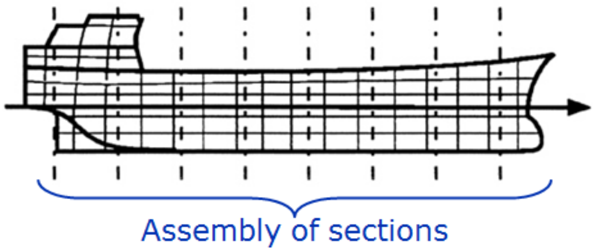
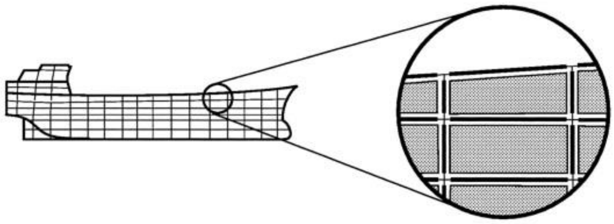
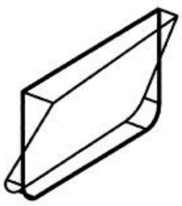
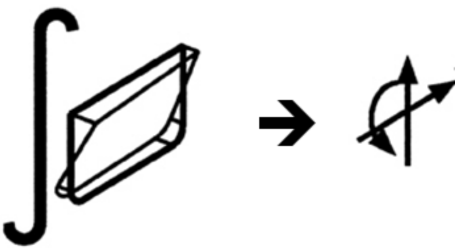


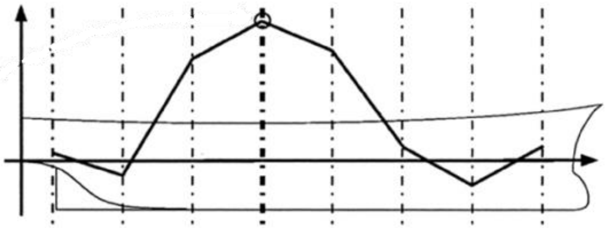
Sections with stress/force distribution

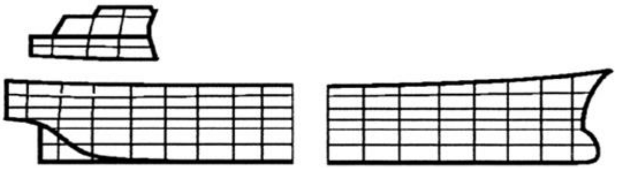
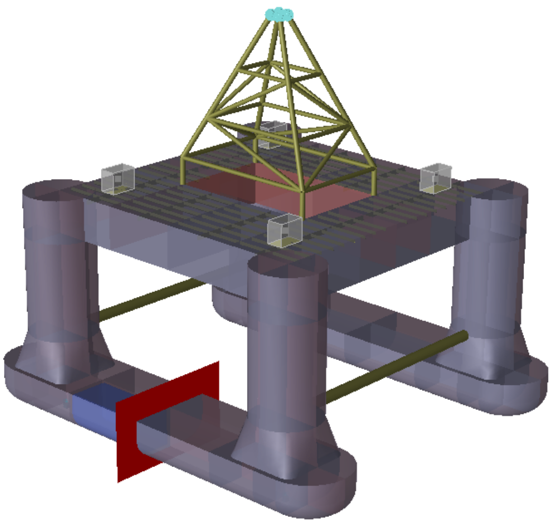


Force/moment diagram along "beam" axis

FEATURES OF CUTRES

FEATURE	DESCRIPTION
Arbitrary section 	An arbitrary section through a model may be created.
Assembly of sections 	An assembly of equally spaced sections through a model may be created.
Element types 	The model may consist of beam, truss, shell, plate and membrane elements.
Stress/force distribution over section 	By interpolating within each intersected element, the stress/force distribution over each section is found and graphed.
Integration over each section 	The stresses/forces are integrated over each section to find "beam" forces and moments, i.e. axial force and two shear forces plus torsional moments and two bending moments.

<table>
<tr><td>“Beam” forces/moments along axis
</td><td>The “beam” forces/moments may be graphed and tabulated along the “beam” axis.</td></tr>
<tr><td>Static or complex</td><td>Complex results from a frequency domain analysis may be handled as well as results from a static analysis.</td></tr>
<tr><td>Direct or superelement analysis

Direct analysis

Superelement analysis</td><td>The model may be a multilevel superelement model.</td></tr>
<tr><td>Handling part of a model
</td><td>By specifying limiting coordinates or by selecting sets a part of the model may be selected for creation of sections.

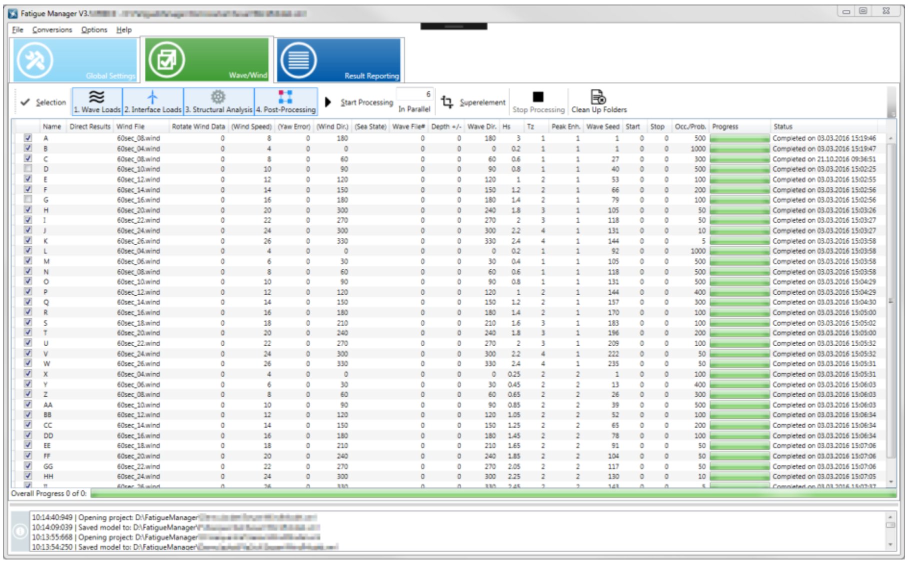
This allows calculating “beam” forces/moments for e.g. only a pontoon of a full semi-submersible model and only one hull of a double-hull ship.</td></tr>
</table>

Fatigue Manager

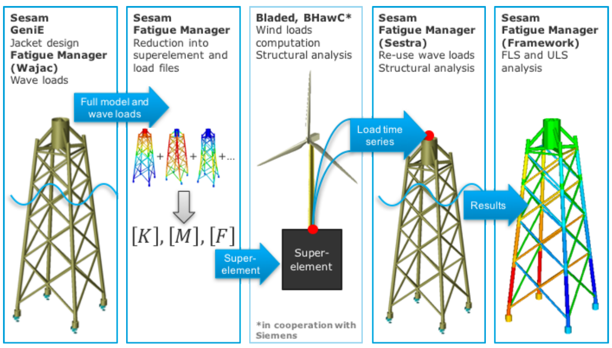
TIME DOMAIN ANALYSIS OF OFFSHORE FRAME STRUCTURES INCLUDING WIND TURBINE SUB-STRUCTURES

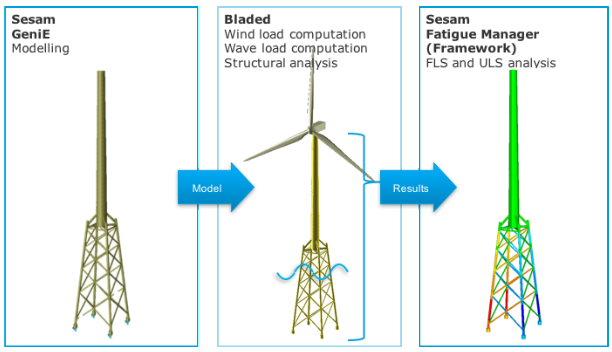
Last revised: February 1, 2019. Describing version 4.2 (64 bit).

Fatigue Manager is a tool for time domain fatigue and ultimate strength analysis of offshore frame structures subjected to wave and (optionally) wind turbine loads. Examples of structures are jackets, tripods and monopiles. It is typically used for fixed offshore wind turbine support structures.



Fatigue Manager user interface – wave/wind grid

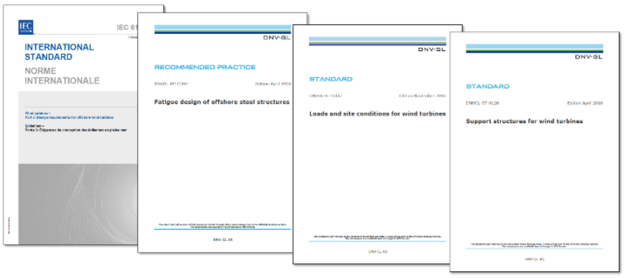
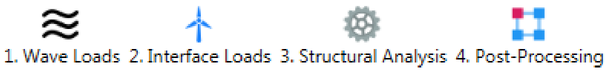
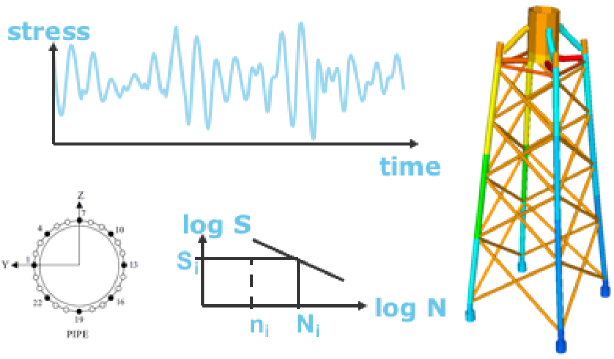


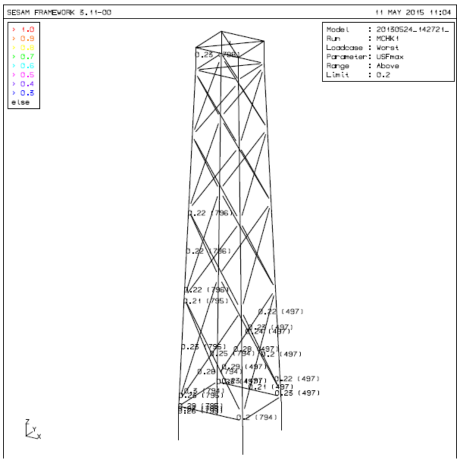
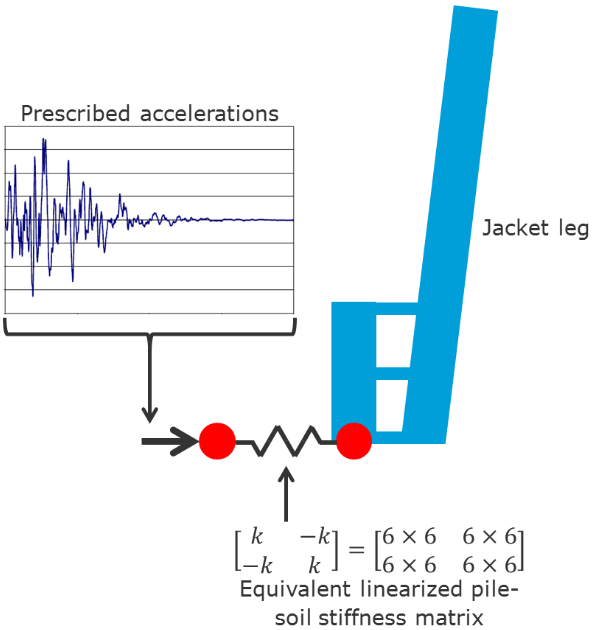


Superelement approach – received time series of wind turbine interface loads are applied at interface point and combined with wave loads in Sesam.

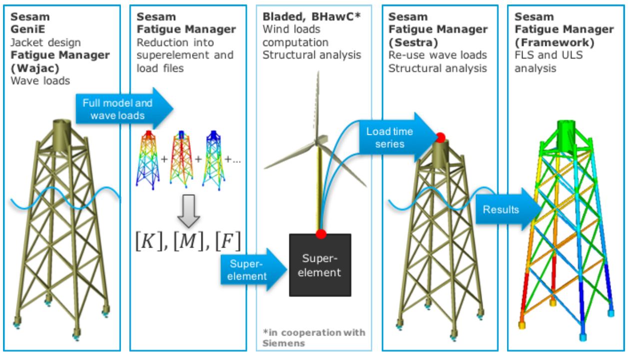
Integrated approach – full stress history of support structure is converted into Sesam's results file format for post-processing.

FEATURES OF FATIGUE MANAGER

FEATURE	DESCRIPTION
Design load cases 	The program is developed with the requirements of IEC61400-3, DNVGL-ST-0126, DNVGL-ST-0437 and DNVGL-RP-C203 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 for the included DLCs, taking into account the relative occurrence of each DLC over the life-time. Similarly, ultimate strength code checks can be performed.
Sesam programs included 	Wajac is used for the wave load computation. Wind turbine interface loads can be included in Fatigue Manager via a simple text file with load per time step. Sestra is used for structural analysis and optionally Splice for non-linear pile-soil analysis. Framework is used as a postprocessing tool for fatigue and/or ultimate strength analysis.
Fatigue analysis in time domain 	Fatigue analysis for all design load cases can be performed using Framework. Results are summed over all fatigue load cases for each hotspot in each member.

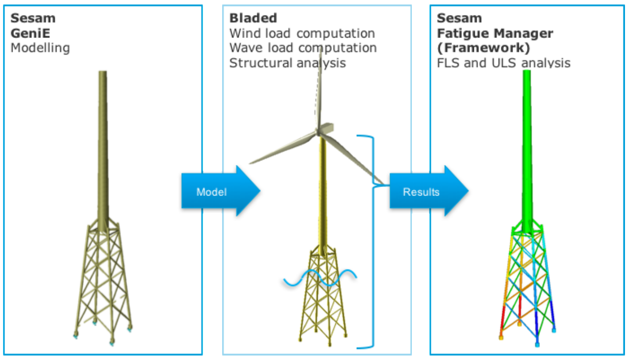
Ultimate strength analysis in time domain 	Ultimate strength analysis for all design load cases can be performed using Framework. The results for all design load cases are combined into a single overview, showing the worst design load case and utilisation factor for each beam in the model.
Seismic (earthquake) analysis in time domain 	Prescribed ground accelerations in time domain may be added into the analysis and will contribute to the total loading together with wave and wind turbine loads.

Superelement analysis



The modelling of the jacket is done in Sesam. The model (and optionally the wave loads) are then imported/converted into a superelement, and linked to a wind turbine and tower model in Bladed, BHawC, VTS/Flex5, or another wind turbine load calculation tool. A structural analysis is then performed in Bladed (or similar), after which the forces and moments are extracted at the interface point. These interface loads are then applied to the model in Sesam, together with the wave loads, and the structural analysis is run to obtain the loads in the jacket. Fatigue and extreme analysis is subsequently performed in Sesam.

Integrated analysis



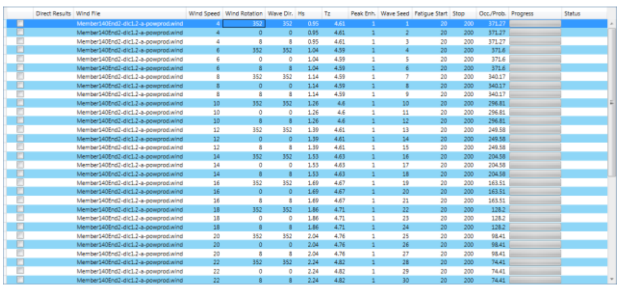
The modelling of the jacket and tower is done in Sesam. The model is then imported/converted into Bladed format and linked to a wind turbine model in Bladed. A structural analysis is then performed in Bladed, after which time series of the resulting forces and moments are extracted for every beam in the structure. These results are then converted into Sesam format. Fatigue and extreme analysis is subsequently performed in Sesam.

Import and export functionalities for data exchange

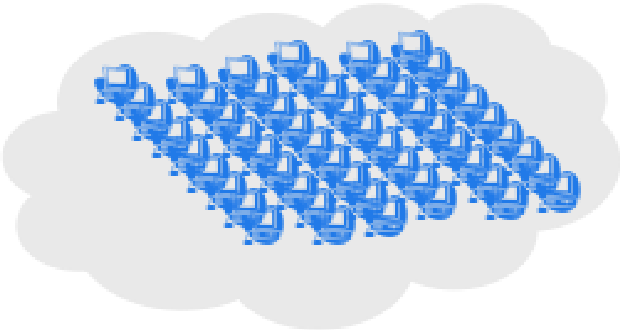
Converters exist for load import from Bladed, BHawC, VTS/Flex5 and HawC2. In addition, time series of wind turbine loads from any 3rd party tool can easily be used in Sesam by outputting the loads as a simple text file with columns of time and loads in 6 degrees of freedom.

Besides this, export functionalities exist to Bladed (integrated model or superelement+wave loads), BHawC (superelement+wave loads) and VTS/Flex5 (superelement+wave loads).

Parallel computing



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

Cloud computing 	In addition to local parallel computing, Sesam Fatigue Manager is cloud-enabled. This enables analysing all DLCs in the Sesam Cloud allowing fast and simultaneous running of many DLCs. The status of the analyses can be monitored online and results are downloaded automatically at run completion.
Verification reports 	The interfaces between Sesam and wind turbine load calculation tools Bladed and BHawC have been verified. Verification reports describing the interfaces and verifying correct implementation are available on request.

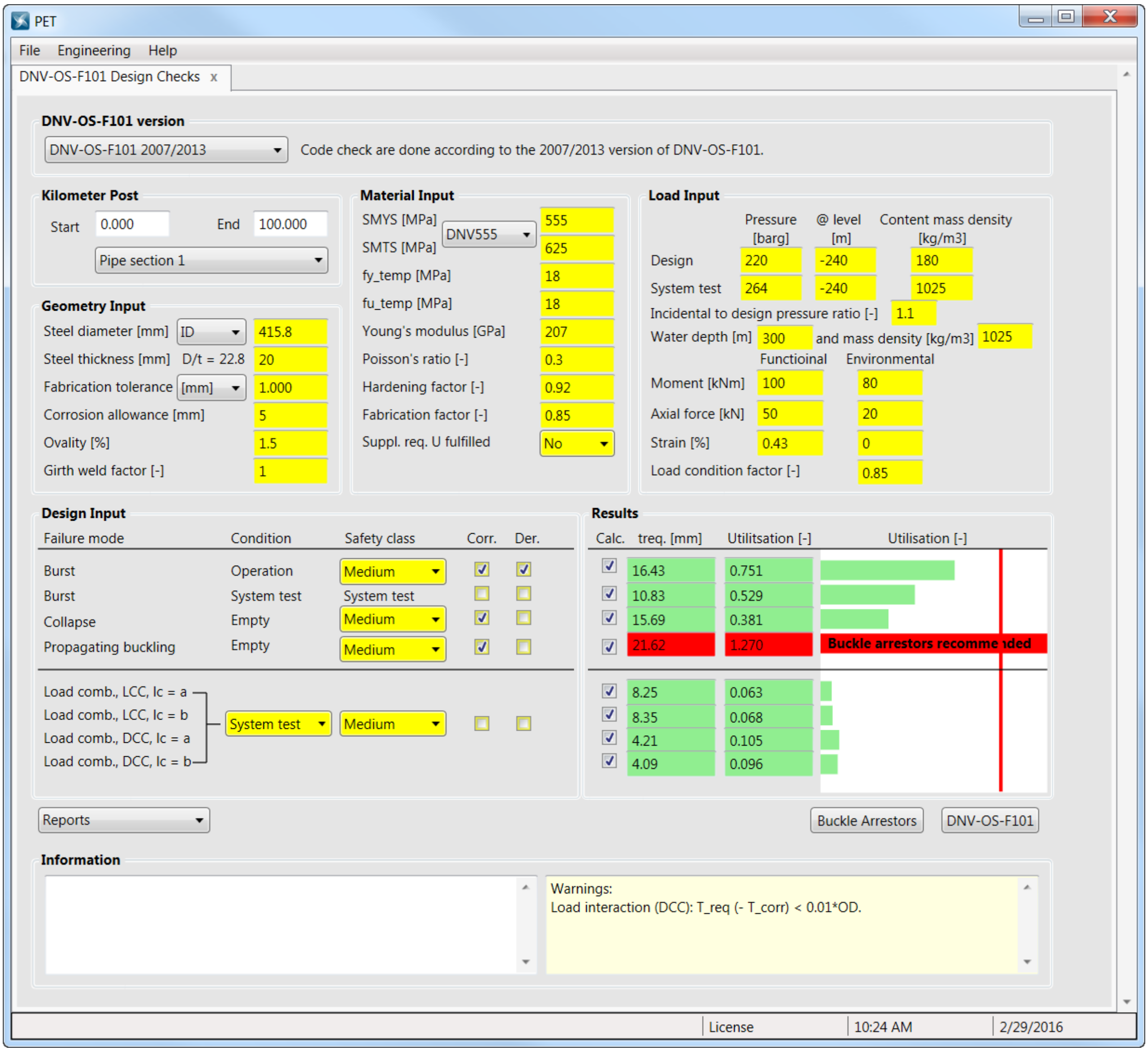
PET

PIPELINE ENGINEERING TOOL – THE DEFINITIVE TOOL FOR EARLY PHASE PIPELINE DESIGN

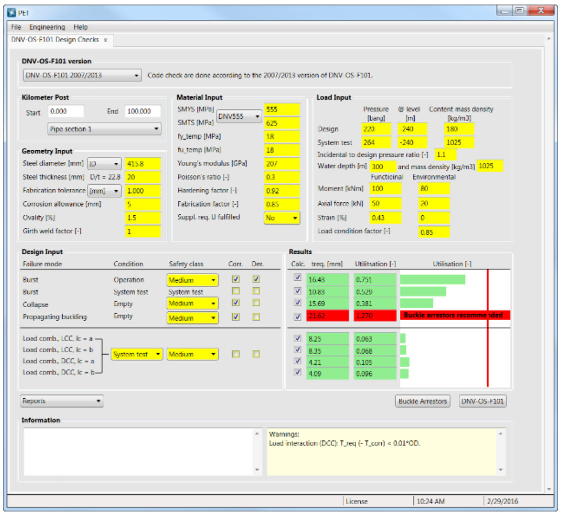
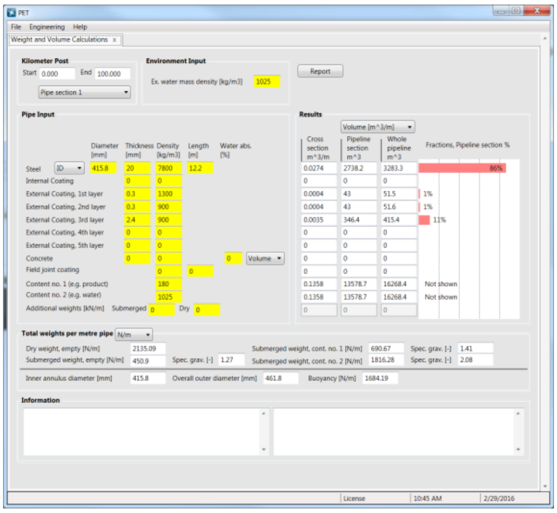
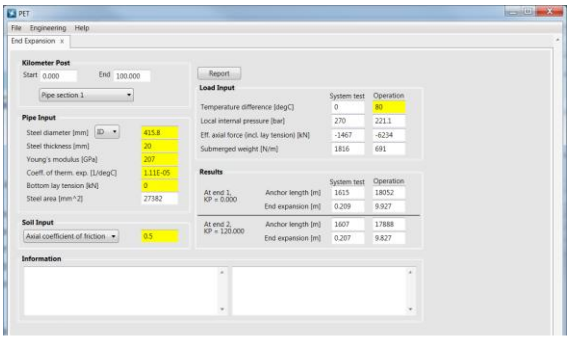
Last revised: October 5, 2018. Describing version 4.1-00.

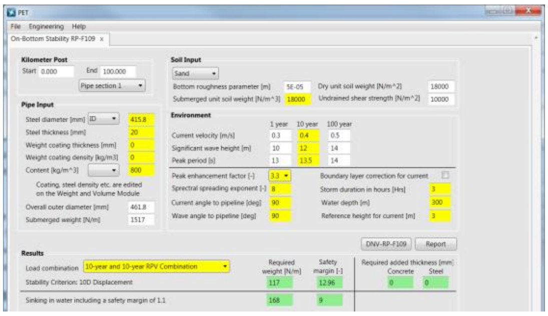
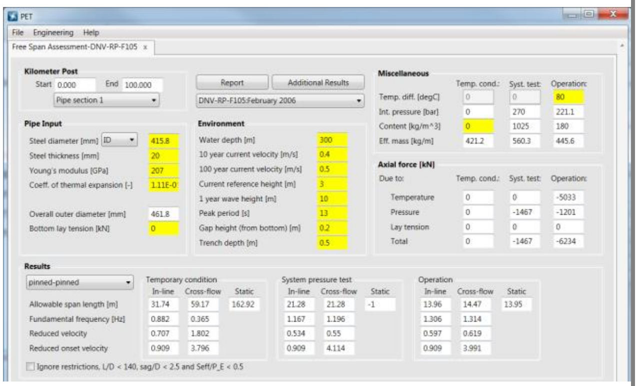
PET is a tool for the early design stage covering all significant design decisions in a simplified manner. PET contains a wide set of easy to use calculation modules for quick assessments of offshore pipeline designs.

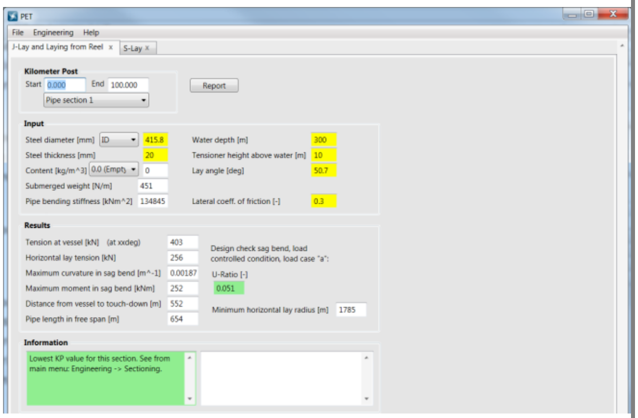
In general, PET could be used for early design where all design checks are performed in a simplified way. Later the other Pipeline Tools or other advanced tools can be used for more detailed design.



FEATURES OF PET

FEATURE	DESCRIPTION
Design checks in accordance with DNV GL ST F101 	The calculation module performs calculation for the following limit states (failure modes): • Burst – during operation as well as during system pressure test • Collapse • Propagating buckling • Combined loading
Weight and volume 	Calculates the volume, mass and dry weight of the components that constitute a pipeline, i.e. steel, coating layers and content. Volume, mass and dry weight are calculated individually and totally, per metre pipeline and totally for a given length of the pipeline.
Expansion 	Calculates end expansion due to temperature and internal pressure. The virtual anchor length is also calculated. These two results are presented for the system pressure condition and the design condition.

Upheaval buckling 	Estimates the safety level with respect to upheaval buckling for the given set of input, predicts the temperature, internal pressure and imperfection height that will trigger upheaval buckling and estimates the cover height to prevent upheaval buckling for a given safety level.
Stability calculations according to DNV-RP-F109 	Estimates the safety level with respect to stability for the given set of input, added weight coating and wall thickness required to ensure stability for 10D displacement criterion.
Free span calculations according to DNV-RP-F105 	Calculates the allowable free span length considering in-line and cross-flow vortex induced vibrations. The module also gives the buckling length (pinned-fixed condition) for the given effective axial force.
Reel straining 	Calculates maximum bending strain on the reel including a code check according to DNV-OS-F101, corresponding ovality and accumulated plastic strain during reeling, unreeling, aligning and straightening.

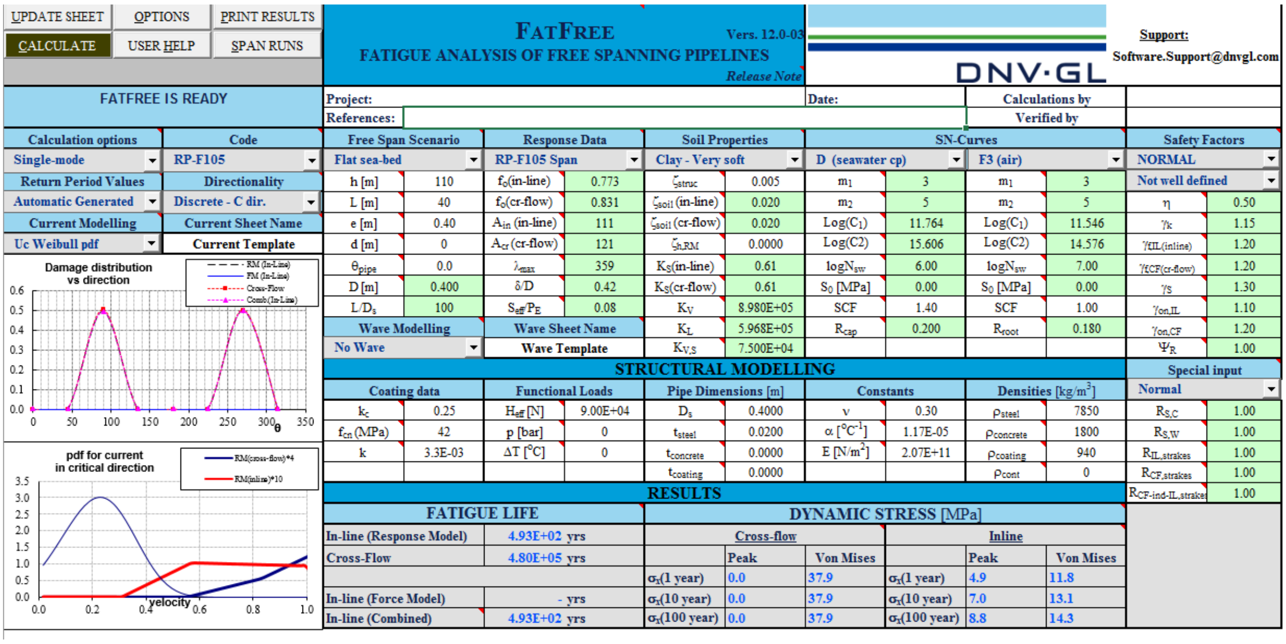
Reel packing 	Calculates the amount of pipe that can be packed on a reel/carousel.
J-lay and S-lay 	Calculates the following during pipe J-lay and S-lay: • Actual top tension during laying • Horizontal top tension • Maximum curvature and moment in the sag bend including utilisation ratio according to DNV-OS-F101 • Horizontal distance from touch down to barge • Length of pipe in the free span and minimum horizontal lay radius

FatFree

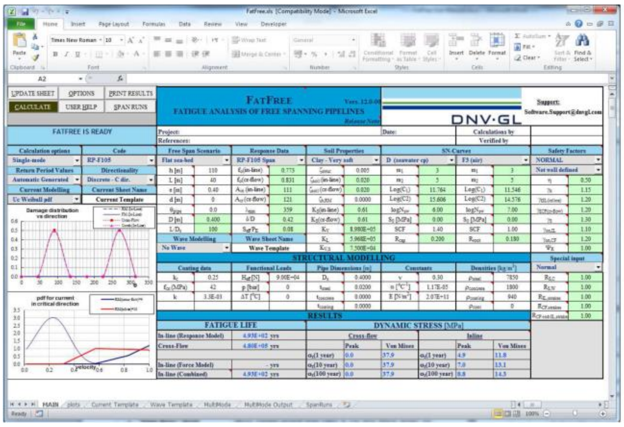
FATIGUE ANALYSIS OF FREE SPANNING PIPELINES

Last revised: April 23, 2018. Describing version 12.0-03.

FatFree is a Microsoft Excel VBA spreadsheet for design and (re-)assessment of submarine pipeline spans in compliance with DNV-RP-F105 "Free Spanning Pipelines", issued February 2006.



FEATURES OF FATFREE

FEATURE	DESCRIPTION
Fatigue life	FatFree calculates the fatigue life due to**:** • Combined direct wave action and in-line vortex induced vibrations (VIV) • Cross-flow VIV based on environmental description, i.e. directional long term distribution for current and wave (in terms of height and period) • Free span scenario (water depth, span geometry, soil conditions, etc.) • Pipe characteristics (material, geometry, SN-curve, etc.) • Natural frequency and mode shape from FE analyses or simplified beam theory expressions
Main sheet 	The main sheet contains all important input and output except environmental data. It allows definition of calculation modes and links to the environmental data.

Soil properties

Soil Properties	
Clay - Very soft	
ζ_{struc}	0.005
ζ_{soil} (in-line)	0.020
ζ_{soil} (cr-flow)	0.020
$\zeta_{h,RM}$	0.0000
K_S(in-line)	0.61
K_S(cr-flow)	0.61
K_V	8.980E+05
K_L	5.968E+05
$K_{V,S}$	7.500E+04

Different soil models are available for automatic calculation of damping properties, or these can be defined explicitly in the 'User defined' soil:

- User Defined
- Clay - Very soft
- Clay - Soft
- Clay - Firm
- Clay – Stiff
- Clay - Very stiff
- Clay - Hard
- Sand - Loose
- Sand – Medium
- Sand – Dense

SN-curves

SN-Curves			
D (seawater cp)		F3 (air)	
m_1	3	m_1	3
m_2	5	m_2	5
Log(C_1)	11.764	Log(C_1)	11.546
Log(C2)	15.606	Log(C2)	14.576
$\log N_{sw}$	6.00	$\log N_{sw}$	7.00
S_0 [MPa]	0.00	S_0 [MPa]	0.00
SCF	1.40	SCF	1.00
R_{cap}	0.200	R_{root}	0.180

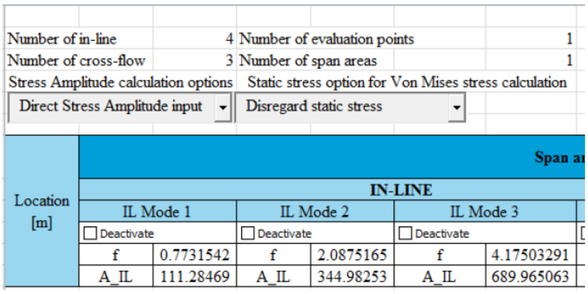
The user can define SN-curves for the weld root and weld cap. FatFree automatically presents the lowest fatigue life of the two sets. The SN-curves can be chosen from a set of predefined curves or user defined.

Environmental data

Direction relative to geographic N	Sector probability	Weibull parameters F(x)=1-exp(-((x-γ)/α)^β) Shape (β)	Scale (α)	Location (γ)
Omni	1	3.922	0.495	0.000
0	0.135	2.680	0.275	0.000
45	0.215	2.546	0.258	0.000
90	0.067	3.774	0.461	-0.184
135	0.05	3.448	0.389	-0.088
180	0.154	2.641	0.270	0.000
225	0.249	2.506	0.253	0.000
270	0.077	3.922	0.495	-0.229
315	0.054	3.509	0.403	-0.107

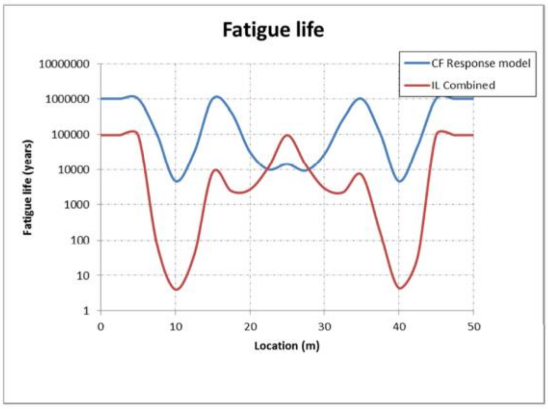
Current and wave data are conveniently defined in separated spreadsheets. Data can be inserted in the form of Weibull distribution; RPV input for Weibull distribution; histogram and, in the case of wave modelling, scatter diagram.

Single or Multi-mode analysis



The calculations can be done considering single mode vibrations or higher modes can be included using the multi-mode analysis. In both cases the mode frequencies and shapes can be estimated using response quantities according to DNV-RP-F105 or classical beam theory for pinned-pinned, pinned-fixed and fixed-fixed boundary conditions. Alternatively, the user can define the response data obtained from other methods, like FE analysis.

Multiple location analysis



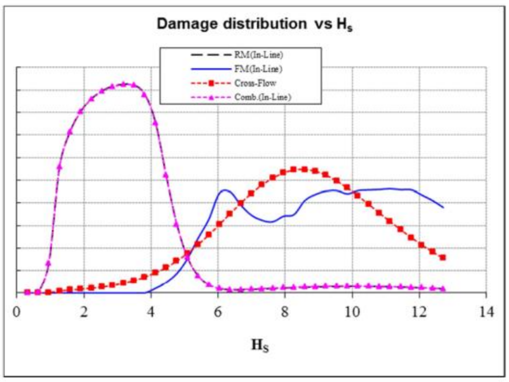
The user can define a large number of evaluation points throughout the pipeline.

Span runs

Fatigue Life (years)			
Inline Response	Inline Force	Inline Combined	Cross-flow
1.00E+06	1.00E+06	9.75E+05	1.00E+06
3.03E+04	1.00E+06	2.95E+04	2.23E+04
1.82E+02	1.00E+05	1.81E+02	3.46E+02
6.29E+01	8.06E+03	6.28E+01	2.54E+01
5.58E+01	1.10E+03	5.42E+01	1.45E+01
9.49E+01	3.30E+02	7.85E+01	2.32E+01

This option is used to calculate several span cases in one run. Thus, it can be used for screening purposes, to perform sensitivity studies or just to analyse many separate spans in one run and keep the input and results together in one data sheet. For each run all input data and results are presented in a single row. Input data can be conveniently imported from the main sheet.

Plots sheet



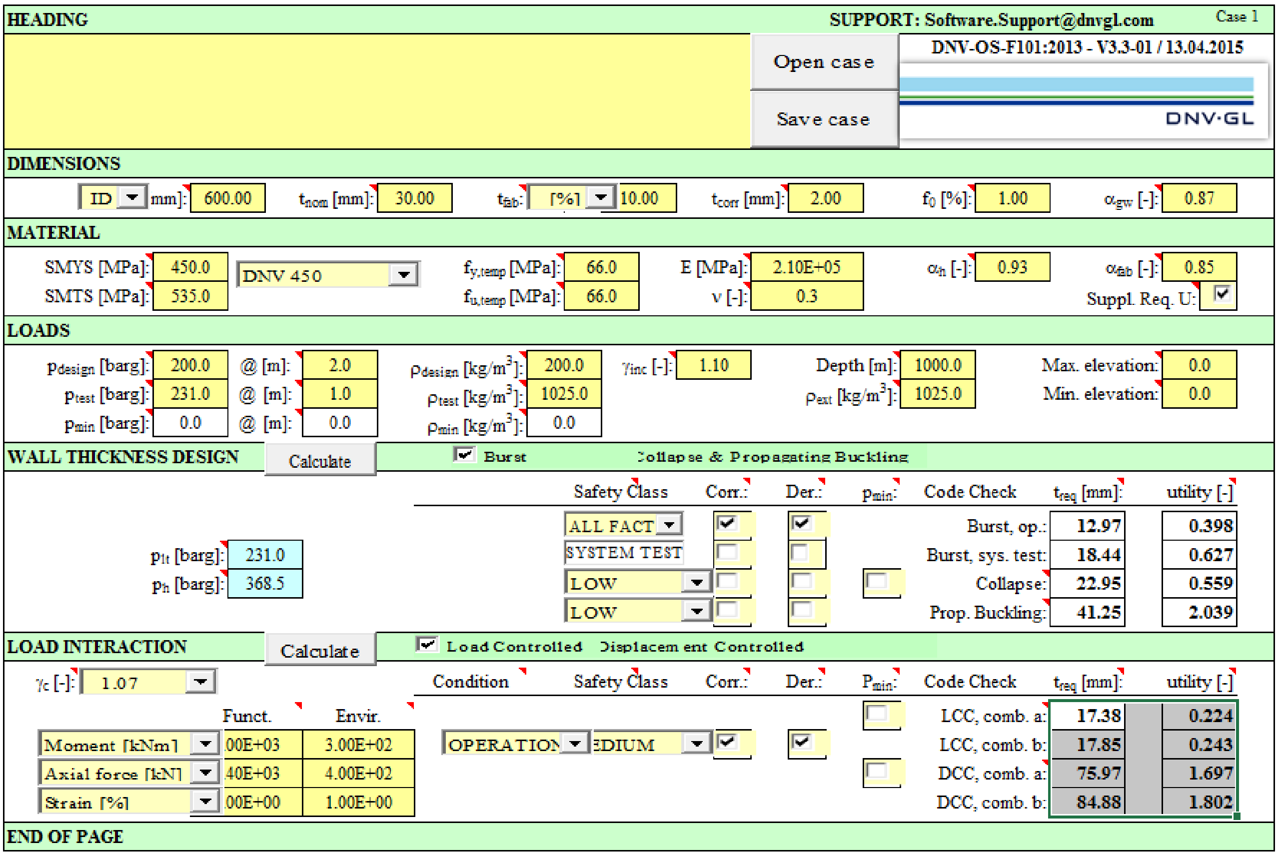
The plot sheet provides the basis for the graphical results. It also contains additional information from analysis and settings.

OS-F101

CODE COMPLIANCE DESIGN OF SUBMARINE PIPELINES

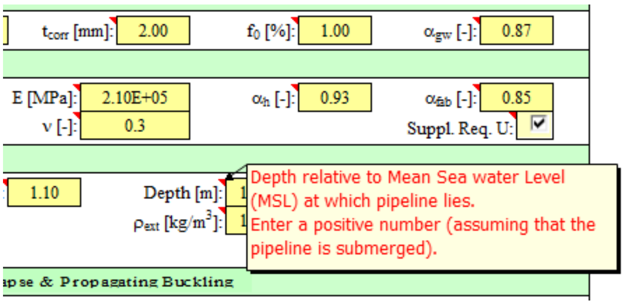
Last revised: October 5, 2018. Describing version 3.4-00.

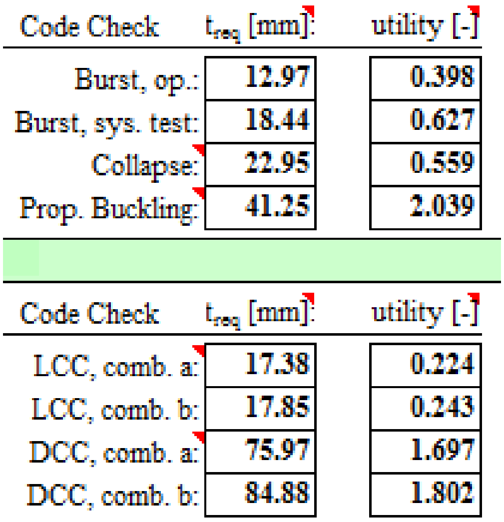
OS-F101 is a Microsoft Excel VBA spreadsheet for checking compliance with DNVGL-ST-F101.



FEATURES OF OS-F101

<table>
<tr><th>FEATURE</th><th>DESCRIPTION</th></tr>
<tr><td>Main sheet


</td>
<td>All input and results are shown in short form on the Main sheet. Explanations are given as comments in the relevant cells.</td></tr>
<tr><td>Code checks


</td>
<td>The following code checks are included:

- Burst (pressure containment) related to both system test condition and operation
- Collapse for an empty pipeline
- Propagating buckling for an empty pipeline
- Load controlled load interaction
- Displacement controlled load interaction

The program calculates:

- The minimum required wall thickness for the given conditions
- Utilisation based on a wall thickness given by the user</td></tr>
</table>

Reported sheets

OS-F101 - SUBMARINE PIPELINE SYSTEMS - 2013
Pressure Containment (bursting)
DNV-OS-F101:2013 - V3.3-01 / 13.04.2015

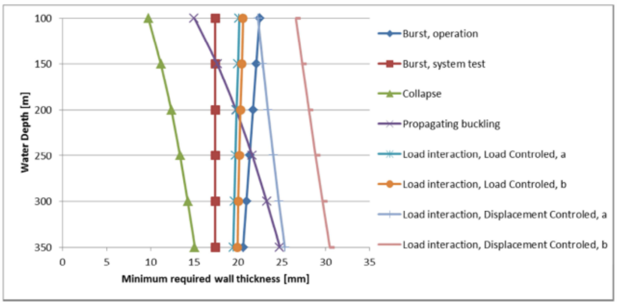
DNV·GL

Prepared: Date: Sign:		Verified: Date: Sign:	
RELEVANT INPUT PARAMETERS:		**Operation**	**System test**
Nominal inner steel diameter:	ID	600.00 mm	
Nominal steel wall thickness:	t_{nom}	30.00 mm	
Fabrication tolerance:	t_{fab}	10.00 %	
Corrosion allowance:	t_{corr}	2.00 mm	
Specified minimum yield stress:	SMYS	450.0 MPa	
Specified minimum tensile strength:	SMTS	535.0 MPa	
Derating in yield stress due to temperature:	$f_{y,temp}$	66.0 MPa	
Derating tensile strength due to temperature:	$f_{u,temp}$	66.0 MPa	
Material strength factor:	α_U	1.00	1.00
Internal pressure at reference level:	$p_{int,ref}$	200.0 barg	231.0 barg
Reference level for internal pressure:	z_{ref}	2.0 m	1.0 m
Density of internal fluid:	ρ_{int}	200.0 kg/m3	1025.0 kg/m3
Incidental to design pressure ratio:	γ_{inc}	1.10	1.00
Depth:	d	1000.0 m	1000.0 m
Density of external fluid:	ρ_{ext}	1025.0 kg/m3	1025.0 kg/m3
Safety Class:		ALL FACTORS=1	SYSTEM TEST
Corroded wall thickness:		YES	NO
Derated material properties:		YES	NO
INTERMEDIATE RESULTS:		**Operation**	**System test**
Characteristic yield stress:	f_y	384.00 MPa	450.00 MPa
Characteristic ultimate strength:	f_u	469.00 MPa	535.00 MPa
Steel wall thickness used in code check:	t_1/t_2	25.00 mm	27.00 mm
Pressure containment resistance, yielding limit state:	$p_{b,s}$	349.14 barg	443.27 barg
Pressure containment resistance, bursting limit state:	$p_{b,u}$	370.80 barg	458.26 barg
Pressure containment resistance, minimum of $p_{b,s}$ & $p_{b,u}$:	p_b	349.14 barg	443.27 barg
Material resistance factor:	γ_m	1.00	1.15
Safety class resistance factor:	γ_{SC}	1.000	1.046
Local design pressure:	p_{ld}	219.66 barg	331.65 barg
Local incidental/test pressure:	p_{lx}	239.66 barg	331.65 barg
External pressure:	p_e	100.55 barg	100.55 barg
Pressure difference:	$p_{lx} - p_e$	139.11 barg	231.10 barg
PRESSURE CONTAINMENT, BURSTING:		**Operation**	**System test**
Code check, utility:		**0.40 OK**	**0.63 OK**
Minimum required nominal wall thickness:		12.97 mm	18.44 mm

When calculation is done report sheets are created automatically for all code checks (collapse and propagation buckling are reported in a single sheet).

These report sheets are meant for paper print-out and inclusion in reports. They contain all relevant input, some intermediate results (to ease external verification) and the results from the code check.

Parametric studies



The program allows the user to perform parameter studies/sensitivity studies on each case. E.g. calculate minimum required wall thickness for varying depths.

RP-F101

SPREADSHEET FOR ASSESSMENT OF CORRODED PIPELINES IN COMPLIANCE WITH DNV-RP-F101

Last revised: March 2, 2016. Describing version 1.0-01.

The RP-F101 spreadsheet is a Microsoft Excel VBA spreadsheet developed by DNV GL for assessment of corroded pipelines in compliance with DNV-RP-F101 “Corroded pipelines” Part A.

The RP-F101 spreadsheet contains modules for assessment of:

- Single defects using relative and absolute depth measurements
- Single defect under combined loading (internal pressure and compressive stress)
- Interacting defects
- Complex shaped defects

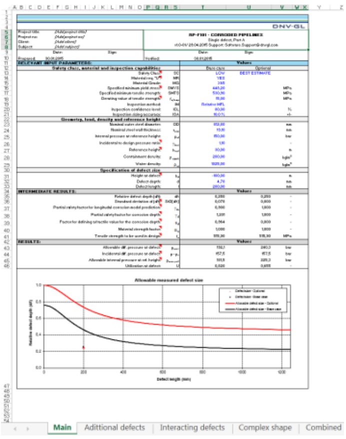
11	**RELEVANT INPUT PARAMETERS:**		**Values**		
12	**Safety class, material and inspection capabilities**		Base case	Optional	
13	Safety Class:	SC	LOW	BEST ESTIMATE	
14	Material req. "U":	MR	YES		
15	Material Grade:	MG	X65		
16	Specified minimum yield stress:	SMYS	448,20		MPa
17	Specified minimum tensile strength:	SMTS	530,90		MPa
18	Derating value of tensile strength:	$f_{u,temp}$	15,00		MPa
19	Inspection method:	IM	Relative MFL		
20	Inspection confidence level:	ICL	80,00		%
21	Inspection sizing accuracy:	ISA	10.0 %		+/-
22	**Geometry, load, density and reference height**				
23	Nominal outer steel diameter:	OD	812,80		mm
24	Nominal steel wall thickness:	t_{nom}	19,10		mm
25	Internal pressure at reference height:	p_{int}	150,00		bar
26	Incidental to design pressure ratio:	γ_{inc}	1,10		-
27	Reference height:	h_{ref}	30,00		m
28	Containment density:	ρ_{cont}	200,00		kg/m^3
29	Water density:	ρ_w	1025,00		kg/m^3
30	**Specification of defect size**				
31	Height at defect:	h_d	-100,00		m
32	Defect depth:	d	4,78		mm
33	Defect length:	l	200,00		mm
34	**INTERMEDIATE RESULTS:**		**Values**		
35	Relative defect depth (d/t):	d/t	0,250	0,250	-
36	Standard deviation of (d/t):	StD[d/t]	0,078	0,000	-
37	Partial safety factor for longitudal corrosion model prediction:	γ_m	0,900	1,000	-
38	Partial safety factor for corrosion depth:	γ_d	1,201	1,000	-
39	Factor for defining a fractile value for the corrosion depth:	ε_d	0,964	0,000	-
40	Material strength factor:	α_u	1,000	1,000	-
41	Tensile strength to be used in design:	f_u	515,90	515,90	MPa
42	**RESULTS:**		**Values**		
43	Allowable diff. pressure at defect:	p_{corr}	192,1	240,3	bar

Main | Adittional defects | Interacting defects | Complex shape | Combined loading

FEATURES OF RP-F101

<table>
<tr><th>FEATURE</th><th>DESCRIPTION</th></tr>
<tr><td>

Main sheet



</td><td>

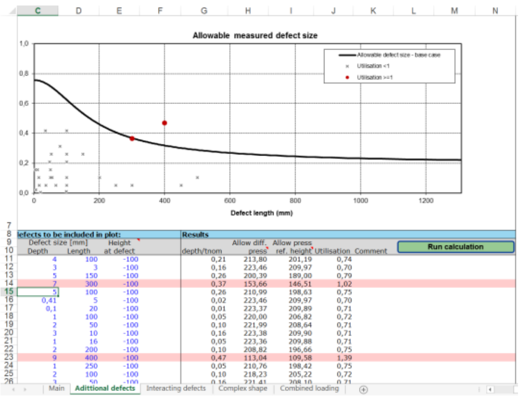
Used for assessment of single defects.

There are two set of input parameters:

- Base case: All of the base case parameters must be entered. All other features of the RP-F101 spreadsheet make reference to these values.
- Optional: This set of parameters is meant for sensitivity study by comparison to the base case input. Input that is left blank will be equal to the base case input.

</td></tr>
<tr><td>

Additional defects sheet



</td><td>

An extension of the main sheet where multiple defects can be assessed for the base case set of parameters in the main sheet.

The calculations performed in the additional defects sheet make use of the base case parameters in the main sheet, but where the defect depth, defect length and height at defect is changed to a list of defects.

</td></tr>
<tr><td>

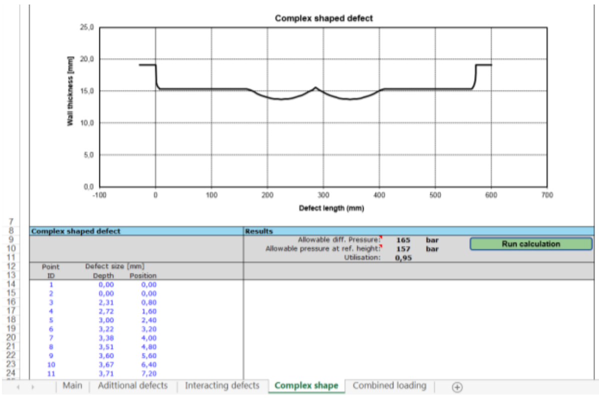
Interacting defects

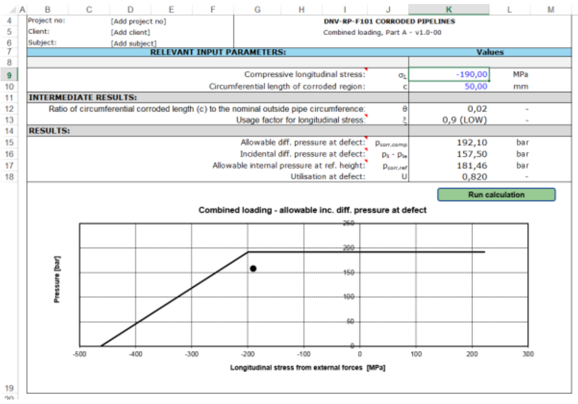
</td><td>

Used for assessment of interacting defects as per section 3.8 of DNV-RP-F101.

To assess the pressure resistance of a pipeline with a set of interacting defects. The list of defects is entered with start position, defect depth and length. All defects in the defect list are projected onto a projection line to obtain the wall thickness profile. Overlapping of defects is handled automatically by the spreadsheet.

</td></tr>
</table>

<table>
<tr>
<td>Complex shape

</td>
<td>Used for assessment of complex shaped defects as per section 3.9 of DNV-RP-F101.

To assess the pressure resistance using the complex shape methodology, the wall thickness profile is entered in the form of a list of position and defect depth.</td>
</tr>
<tr>
<td>Combined loading

</td>
<td>Used for assessment of a single longitudinal defect with internal pressure and superimposed longitudinal compressive stresses as per section 3.7.4 of DNV-RP-F101.</td>
</tr>
</table>

SimBuck

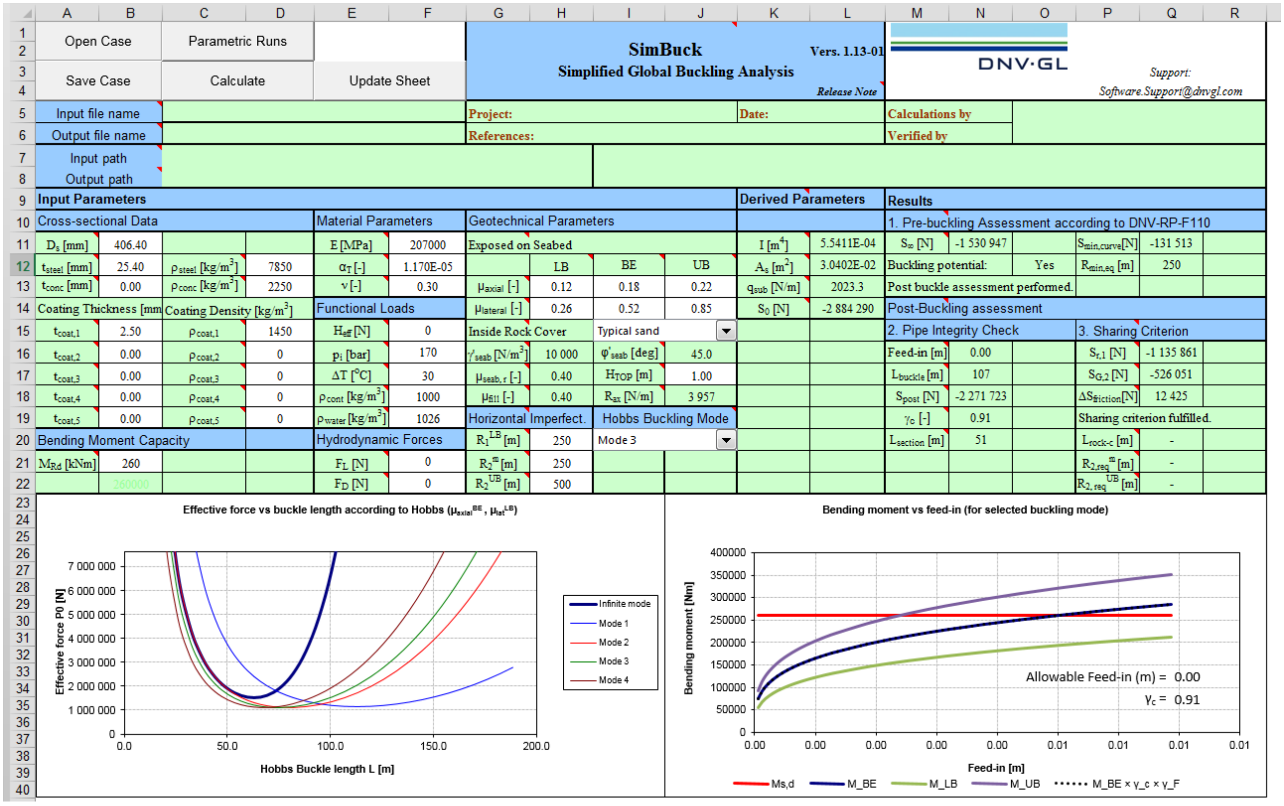
SIMPLIFIED GLOBAL BUCKLING ANALYSIS OF SUBMARINE PIPELINES

Last revised: August 22, 2017. Describing version 2.0-02.

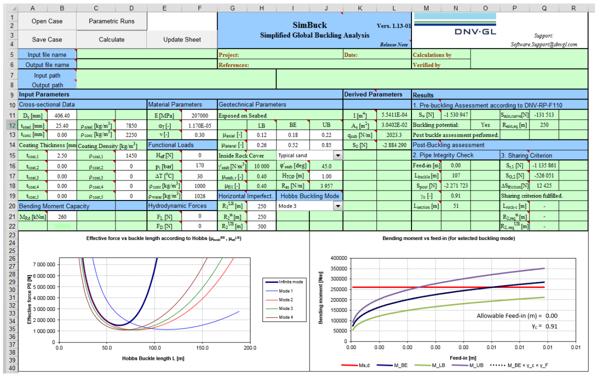
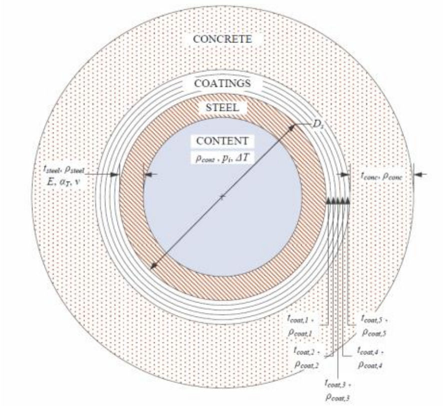
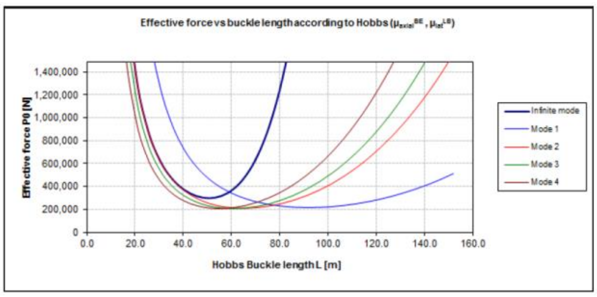
SimBuck is a Microsoft Excel VBA spreadsheet developed by DNV GL for preliminary design and verification of global buckling potential for submarine pipelines in compliance with DNV-RP-F110 ‘Global Buckling of Submarine Pipelines – Structural Design due to High Temperature / High Pressure’, issued October 2007.

SimBuck performs:

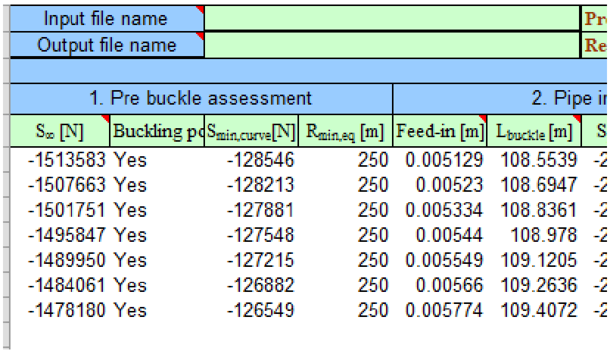
- Single buckle assessment for exposed pipeline on even seabed
- Multi-buckle assessment for exposed pipelines on even seabed
- Checks for upheaval buckling potential



FEATURES OF SIMBUCK

<table>
<tr><th>FEATURE</th><th>DESCRIPTION</th></tr>
<tr><td>Single Buckle Assessment

</td><td>All input and output for single buckle analysis can be accessed through this sheet. This will provide preliminary design solutions on pre-buckling assessment, post-buckling pipe integrity check and sharing criterion evaluation.</td></tr>
<tr><td>Pipeline cross-sectional Geometry

</td><td>Derived parameters such as pipeline's second moment of area, steel's cross-sectional area and submerged weight per metre are automatically calculated by defining diameters, wall thicknesses and densities of the different pipeline layers: steel, pipeline coatings (up to 5 layers) and concrete.</td></tr>
<tr><td>Graphical results

</td><td>Result plots are created automatically in the Single Buckle sheet and in the plots sheet.</td></tr>
</table>

Parametric Runs

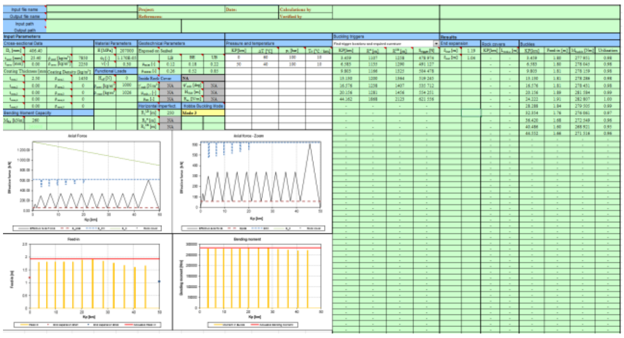


Input file name		Pr
Output file name		Re

1. Pre buckle assessment				2. Pipe in		
S_∞ [N]	Buckling p	$S_{min,curve}$[N]	$R_{min,eq}$ [m]	Feed-in [m]	L_{buckle} [m]	S
-1513583	Yes	-128546	250	0.005129	108.5539	-2
-1507663	Yes	-128213	250	0.00523	108.6947	-2
-1501751	Yes	-127881	250	0.005334	108.8361	-2
-1495847	Yes	-127548	250	0.00544	108.978	-2
-1489950	Yes	-127215	250	0.005549	109.1205	-2
-1484061	Yes	-126882	250	0.00566	109.2636	-2
-1478180	Yes	-126549	250	0.005774	109.4072	-2

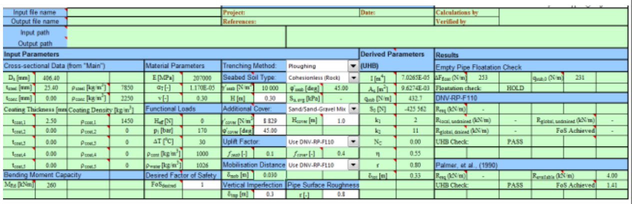
Used to run several lateral buckling cases (single buckle), ideal for screening, performing sensitivity analyses, or simply analysing many separate cases and keeping all the input and output results together on a single data sheet.

Multi Buckle Assessment



Provides additional functionality for analysing an entire pipeline. Buckling triggers can either be specified manually or calculated by SimBuck.

UHB Sheet



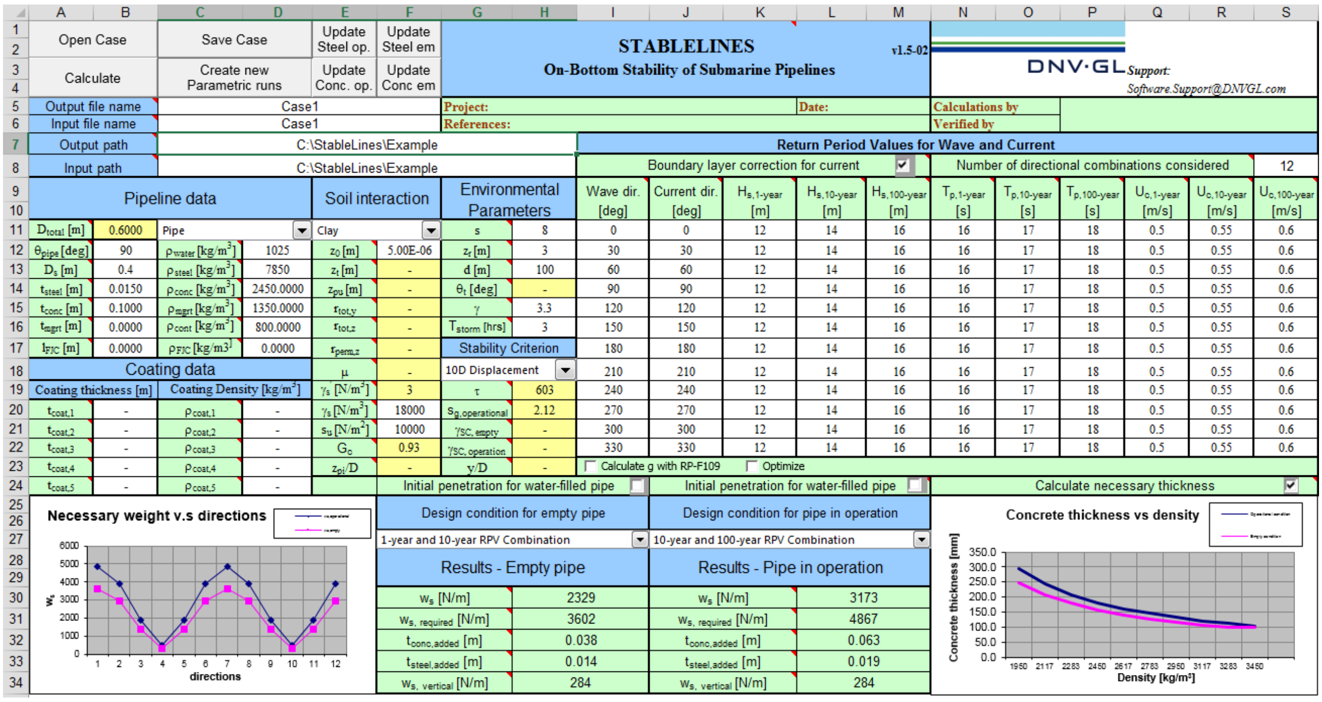
Assessment of upheaval buckling based on deterministic approach according to DNV-RP-F110.

StableLines

ON-BOTTOM STABILITY DESIGN OF SUBMARINE PIPELINES

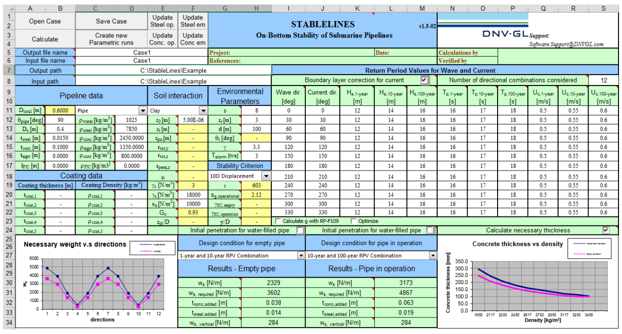
Last revised: October 5, 2018. Describing version 1.7-01.

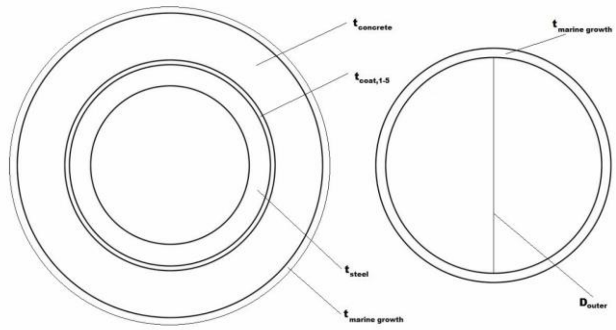
StableLines is a Microsoft Excel VBA spreadsheet for design and assessment of on-bottom stability for submarine pipelines in compliance with DNV-RP-F109 'On-Bottom Stability Design of Submarine Pipelines'.



FEATURES OF STABLELINES

<table>
<tr><th>FEATURE</th><th>DESCRIPTION</th></tr>
<tr><td>Main sheet

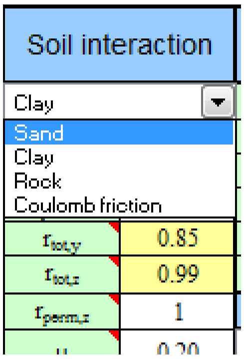

</td><td>The user can input all the relevant data, calculate and view results for a single stability case using the main sheet.</td></tr>
<tr><td>Stability for pipeline and umbilicals



</td><td>The user can check stability for either pipelines or umbilicals. Dimensions and weights are automatically calculated through the user defined thicknesses and densities.

Two stability methods are available:

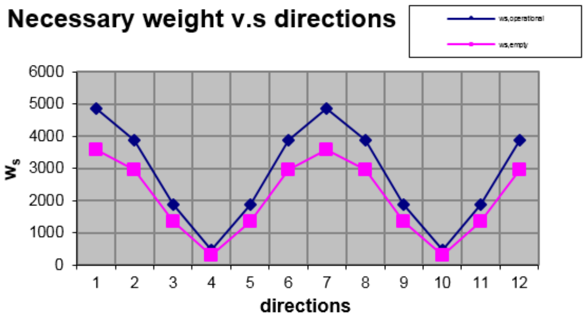
- Absolute stability
- Generalized lateral stability method with displacements up to 10D
</td></tr>
<tr><td>Soil interaction



</td><td>Four different soil models are available:

- Sand
- Clay
- Rock
- User defined Coulomb friction
</td></tr>
</table>

Results presentation



The user can check graphical representation of the stability results such as (but not restricted to):

- Required submerged weight as function of direction
- Required concrete thickness as function of concrete density (pipeline mode)
- Required weight as a function of outer diameter (umbilical mode)

StableLines also suggests added steel and concrete/armour thicknesses to make unstable pipelines stable.

Report Sheet

RELEVANT INPUT PARAMETERS:

Pipeline data		
Nominal outer steel diameter:	D_s	0.4 m
Direction of pipeline:	θ_{pipe}	90 deg.
Density water:	ρ_{water}	1025 kg/m^3
Steel thickness:	t_{steel}	0.0150 m
Concrete thickness:	t_{conc}	0.1000 m
Marine growth thickness:	t_{mgrt}	0.0000 m
Auxilliary coating layer thicknesses		
	$t_{coat,1}$	0.0000 m
	$t_{coat,2}$	0.0000 m
	$t_{coat,3}$	0.0000 m
	$t_{coat,4}$	0.0000 m
	$t_{coat,5}$	0.0000 m

Report of final and intermediate results ready for copy&paste into technical reports.

Parametric runs

w_s [N/m]	$w_{s,\ required}$ [N/m]	$t_{conc,added}$ [m]	$t_{steel,added}$ [m]	$w_{s,\ vertical}$ [N/m]	w_s [N/m]	$w_{s,\ required}$ [N/m]	$t_{conc,added}$ [m]	$t_{steel,added}$ [m]	$w_{s,\ vertical}$ [N/m]
2329	3602	0.0378	0.0142	284	3173	4867	0.0632	0.0191	284
2395	3764	0.0396	0.0137	334	3482	5080	0.0577	0.0163	334
2421	3933	0.0427	0.0138	387	3782	5305	0.0538	0.0142	387

Calculate several on-bottom stability cases in one run. Can be used for screening purposes, to perform sensitivity studies or just to analyse many separate cases in one run and keep the input and results together in one data sheet.

Helica

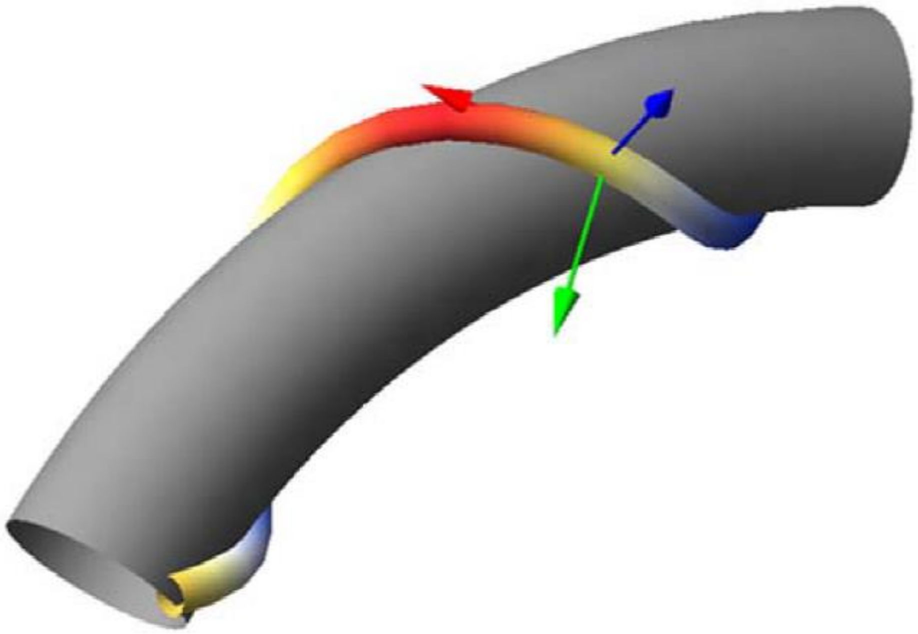
CROSS-SECTIONAL ANALYSIS OF FLEXIBLE PIPES, UMBILICALS AND POWER CABLES

Last revised: March 3, 2019. Describing version 3.0.

Helica is a stress analysis tool tailor made for cross-section analysis of flexible pipes, umbilicals and power cables.

The main functionalities of the program are:

- Cross-sectional load sharing analysis
- Short-term fatigue analysis
- Long-term fatigue analysis
- Extreme analysis
- VIV fatigue analysis

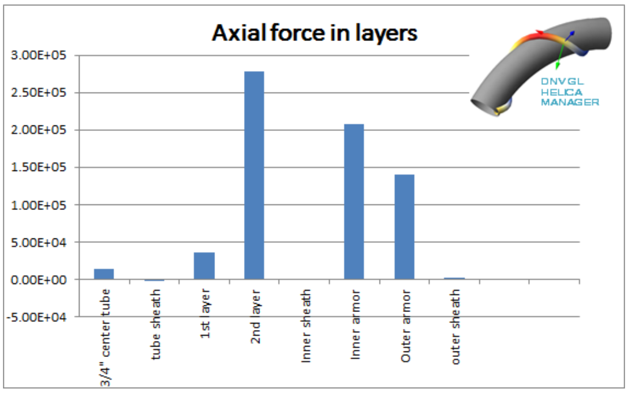


FEATURES OF HELICA

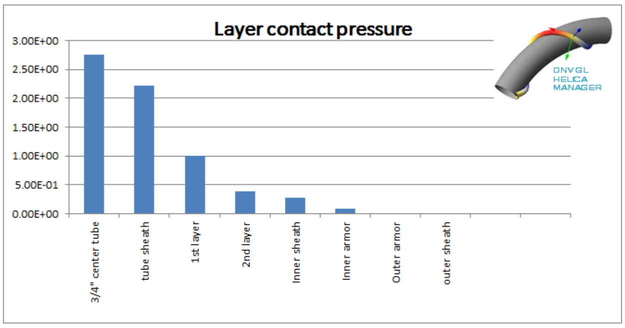
Cross-sectional load sharing analysis

<table>
<tr><th>FEATURE</th><th>DESCRIPTION</th></tr>
<tr><td>
Stiffness and twist characteristics
<table>
<tr><th>Global responses
Group</th><th>Response</th></tr>
<tr><td>BC</td><td>Boundary condition, axial displ.
Boundary condition, torsion displ.</td></tr>
<tr><td>Axial</td><td>Effective axial force [N]
End-cap force (pressure load) [N]
True axial force [N]
Axial displacement [mm]
Axial stiffness [N]</td></tr>
<tr><td>Torsion</td><td>Torsional load [Nmm]
Torsional displacment (twist) [rad]
Torsional stiffness [Nmm/rad]
Torsional/axial displacement ratio</td></tr>
<tr><td>Bending</td><td>Helix bending stiffness, axial, stick [Nmm2]
Helix bending stiffness, bending, stick and slip [Nmm2]
Cylinder layer bending stiffness [Nmm2]
Bending stiffness, full stick [Nmm2]
Bending stiffness, full slip [Nmm2]</td></tr>
</table>
</td><td>
Helica may consider one or more load cases, i.e. combinations of tension, curvature and torque (left or right) for various boundary conditions.

For each load case specified by the user, Helica calculates stiffness characteristics and/or angle of twist.
</td></tr>
<tr><td>
Axial load distribution



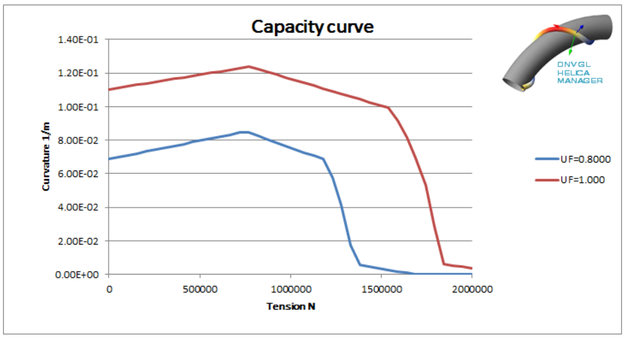
</td><td>
For a load case specified by the user, Helica determines the axial load distribution between layers in the cross-section.
</td></tr>
<tr><td>
Layer contact



</td><td>
For a load case specified by the user, Helica determines the contact pressure on each layer resulting from applied tension.

External (e.g. hydrostatic) pressure is accounted for, if applicable.
</td></tr>
</table>

Capacity curve



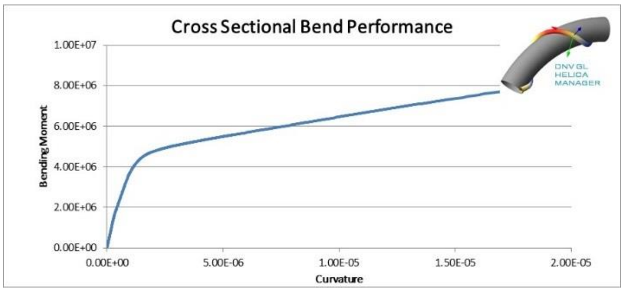
Capacity curves, considering inter-layer contact and friction, may be established for allowable utilization level(s) of material stress capacities specified by the user.

Effects of internal and (uniform) external pressures are considered, if applicable.

Magnitude and number of tension and curvature levels to be considered are specified by the user to facilitate finer discretization, if necessary.

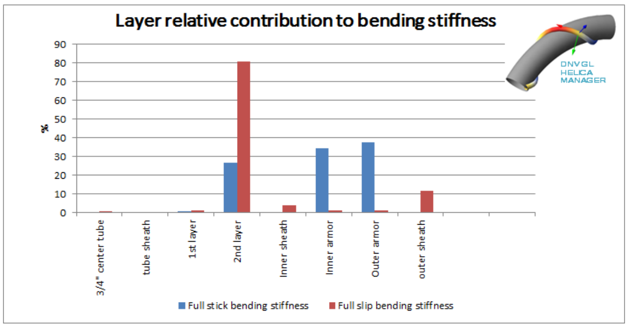
Hierarcical calulation scheme allowing for capacity curve calulation of bundle helix elements.

Non-linear bending performance



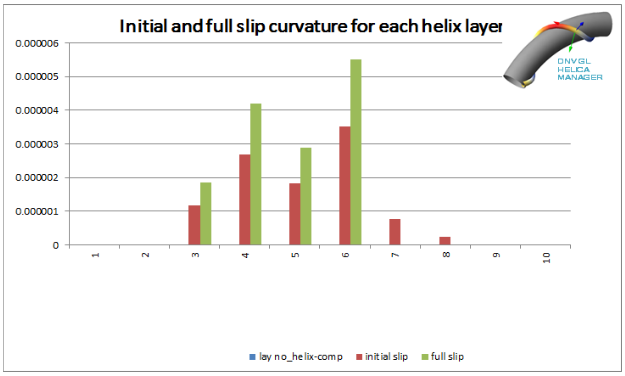
The non-linear bending characteristics of a cross-section may be determined, considering stick/slip effects, for an applied tension specified by the user.

Contribution to bending stiffness

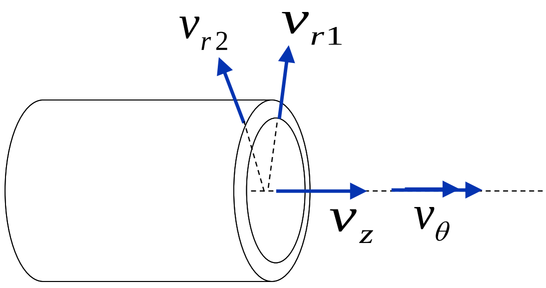
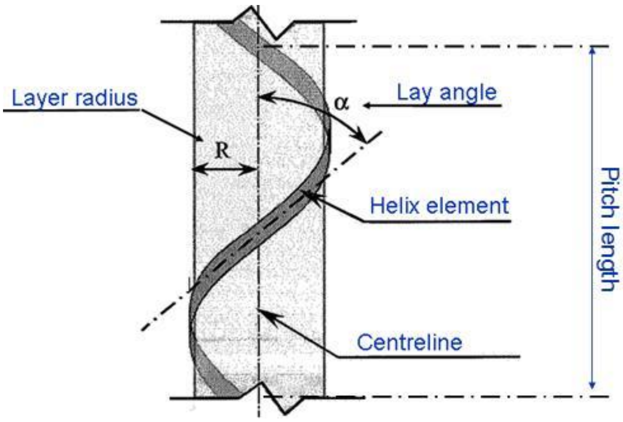


The relative contribution of each layer to the bending stiffness of a cross-section may be determined for both full stick and full slip conditions.

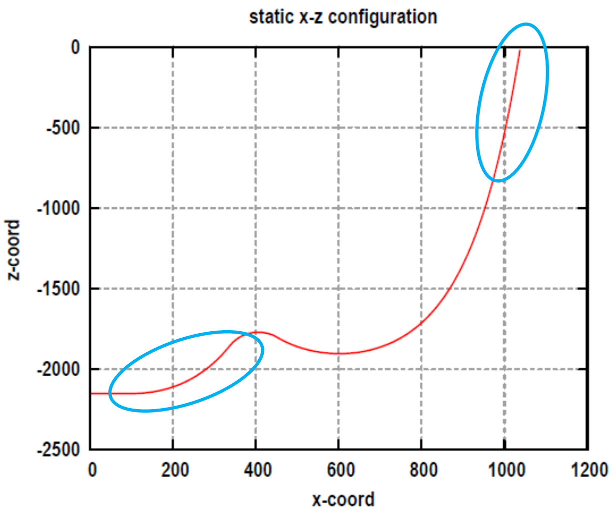
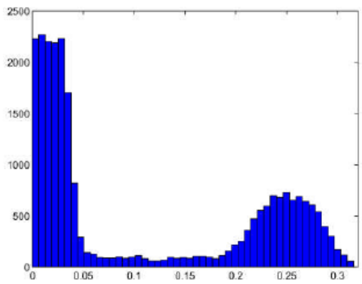
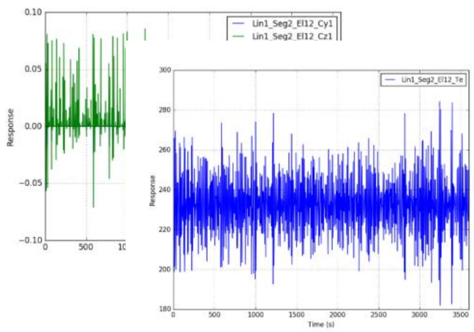
Slip curvatures

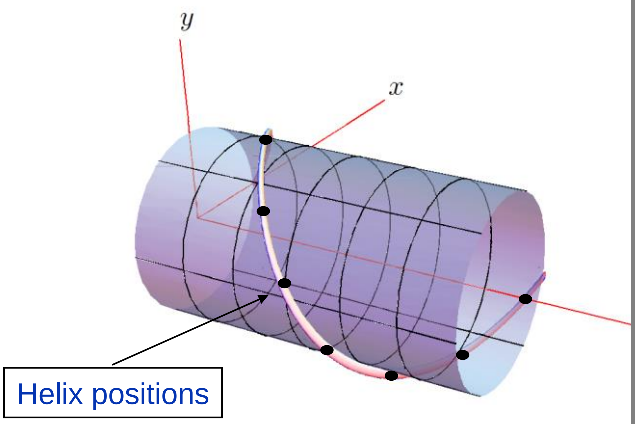
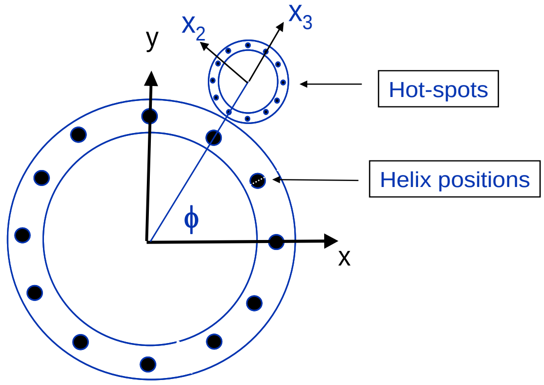
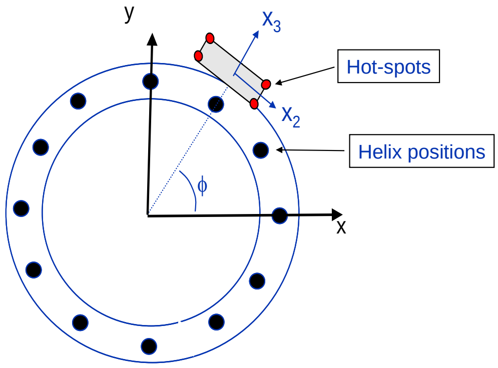


The curvature at which slip of a helix layer is initiated is determined, as well as the curvature that represents full slip of the layer.

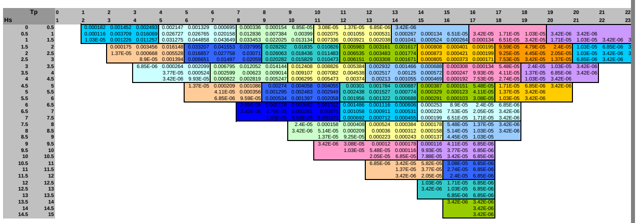
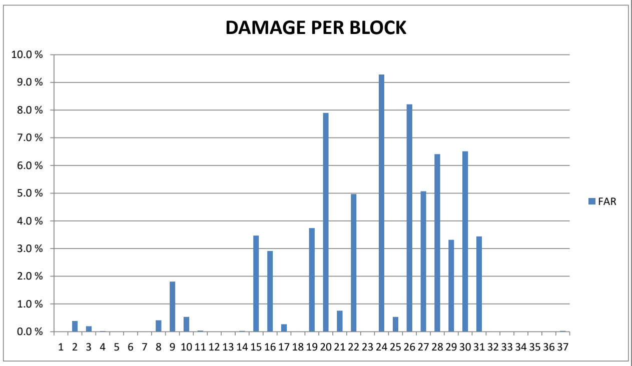
Layer types 	Layers may be modelled in Helica as: • Thin homogenous cylinder layers that do not account for radial deformation • Thick homogenous cylinder layers that account for thickness and radial deformation • General cylinder layer, specified by stiffness properties • Helix layers that may be composed of different types of helix components to model e.g. layers of umbilicals with several different elements in the same layer • Helix layers can be specified in terms of helix lay angle or helix pitch length
Component types 	The following component types are currently available in Helica: • Solid circular cross-section geometry • Circular pipe cross-section geometry • Solid rectangular cross-section geometry • Arbitrary cross-section geometry (circular cross-section with equivalent homogeneous properties)

Short-term fatigue analysis

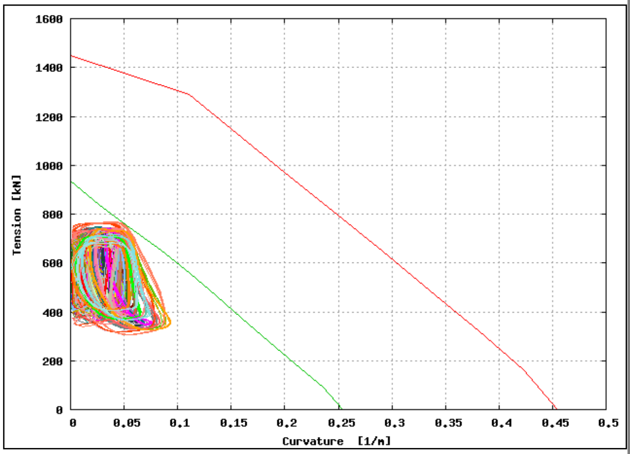
FEATURE	DESCRIPTION
Load case definition 	The following is specified by the user: • What helix element to be considered in the fatigue analysis • Hierarcical calulation scheme allowing for fatigue analysis of 2nd order stick/slip • What locations along riser/umbilical, e.g. nodes in global model, to be considered • Fatigue capacity data (S-N curve) and stress concentration factor • Method for mean stress correction • Response conversion to obtain fatigue stresses in MPa in compliance with units applied in the global response calculations • Filtering technique for processing of stress data • Method of computing fatigue stress in helix elements, e.g. friction/no friction and contact loads based on max or specified tension
Specification of global loads  	Short-term fatigue analyses may be performed based on histograms or time series of tension and curvature imported directly from global analyses. Helica can be run in batch mode, thus facilitating parallel processing if specified by the user.

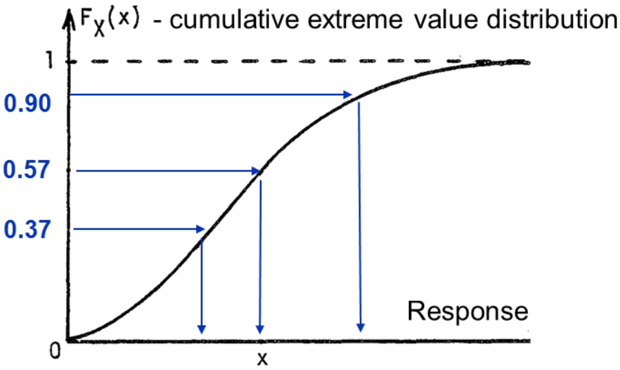
Helix positions 	Fatigue damage is evaluated at a user specified number of *helix positions* per pitch length of a component.
Hot spots  	Fatigue damage is evaluated at a user specified number of *hot-spots* in the cross-section of the helix component to capture the most critical location with respect to fatigue damage.

Long-term fatigue analysis

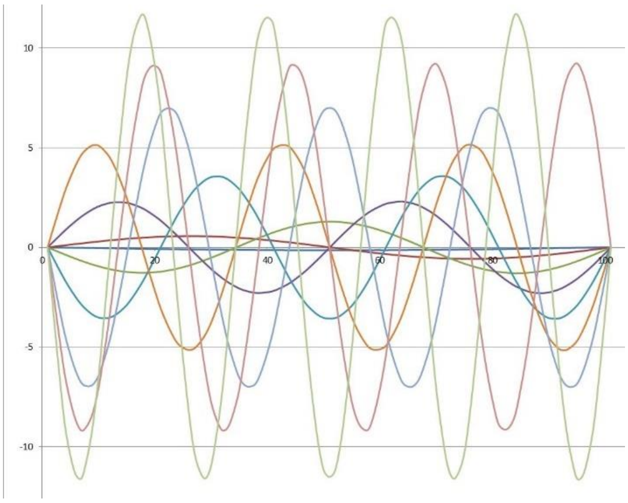
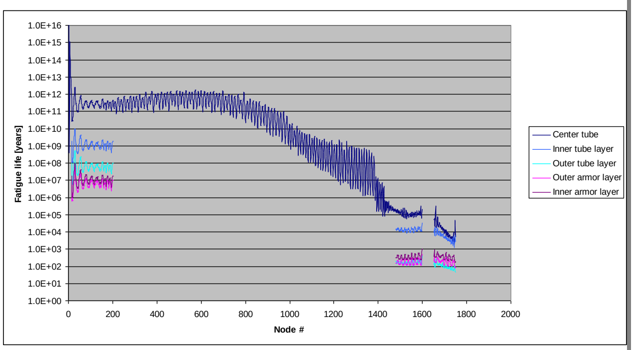
FEATURE	DESCRIPTION
Accumulation of fatigue damage 	Average long-term fatigue damage is found by weighting the damage calculated for selected short-term conditions by the probability of occurrence. The load cases (e.g. locations along the riser) and conditions (e.g. representing one or more offsets) to be considered in the long-term fatigue analysis are specified by the user.
Output 	The output from the analysis is: • accumulated long-term fatigue damage, • long-term stress cycle distribution, and • relative contributions from short-term conditions to the long-term fatigue damage.

Extreme analysis

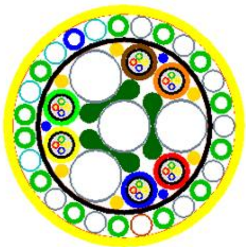
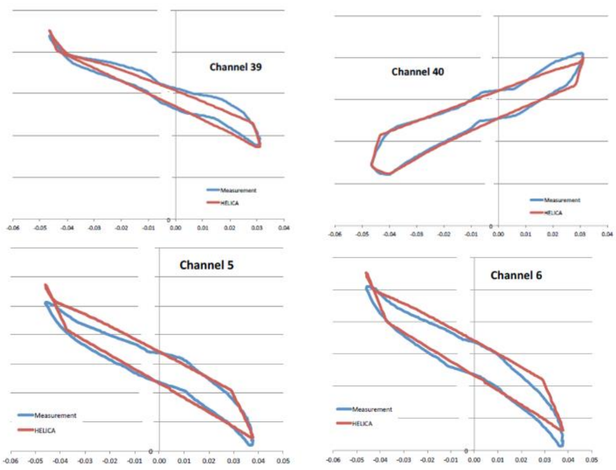
FEATURE	DESCRIPTION
Capacity check 	The response from an extreme analysis may be checked against the capacity curve of the cross-section. The cross capacity curve may be established by Helica or specified by the user. Utilization time series are established by interpolation of the cross-section capacity curve according to the cross-section loading described by time series of effective tension and curvature components imported from a global time domain dynamic analysis.

Statistical processing 	In case of irregular wave loading, statistical processing of the utilisation time series is performed to establish extreme values. Deterministic processing is applied in case of regular wave loading.

VIV fatigue analysis

FEATURE	DESCRIPTION
Short-term fatigue analysis 	Modal damping may be calculated by Helica to account for the amplitude dependant hysteretic stick/slip behaviour of the cross-section in bending. Fatigue stress calculations are performed in Helica considering stick/slip of helix elements and the exited modes and modal amplitudes determined in the VIV analysis. Result files from VIV analyses performed using acknowledged software may be imported directly to Helica.
Long-term fatigue analysis 	Long-term fatigue damage may be calculated by weighting the damage calculated for selected short-term conditions by the probability of occurrence.

Validation

FEATURE	DESCRIPTION
Full-scale validation test   	JIP has been executed with participation from ExxonMobil, Shell, Technip, Oceaneering and ABB. • Objective: validation of umbilical stress calulation by full-scale tests • Test results for 2 steel-tube umbilicals • Measurements of strain in umbilical tubes at multiple locations • Exposed to tension, bending and internal pressure in tubes
Correlation to full-scale tests 	Outstanding correlation between Helica and full-scale test result has been found. Results are documented in public domain (ISOPE 2016).

About DNV GL

DNV GL is a global quality assurance and risk management company. Driven by our purpose of safeguarding life, property and the environment, we enable our customers to advance the safety and sustainability of their business. We provide classification, technical assurance, software and independent expert advisory services to the maritime, oil & gas, power and renewables industries. We also provide certification and supply chain services to customers across a wide range of industries. Operating in more than 100 countries, our experts are dedicated to helping customers make the world safer, smarter and greener.

Digital Solutions

DNV GL is a world-leading provider of digital solutions for managing risk and improving safety and asset performance for ships, pipelines, processing plants, offshore structures, electric grids, smart cities and more. Our open industry platform Veracity, cyber security and software solutions support business-critical activities across many industries, including maritime, energy, and healthcare.