GeniE
The new design tool in SESAM

A White Paper
TABLE OF CONTENT

INTRODUCTION ................................................................. 4

CONCEPTUAL STRUCTURE MODELLING ......................... 6
  Reasoning behind conceptual modelling ....................... 6
  Closing the design loop ................................................. 7
  Example of conceptual modelling – pure beams ............... 7
  Example of conceptual modelling – segmented beams ....... 8
  Example of conceptual modelling – plate and beams ........ 8

EQUIPMENT AND LOAD MODELLING ............................. 9
  From equipment to load ............................................... 9
  Equipments can be repositioned .................................... 9
  Alternative equipment representations ........................ 10

STRUCTURAL ANALYSIS .................................................. 11
  Deriving analysis models from concept models .......... 11
  Top-down concept modelling and FEM derivation .......... 11

RESULTS EVALUATION .................................................. 12
  Analysis and Results Evaluation ................................. 12

SOFTWARE TECHNOLOGY ............................................. 13
  Building on a flexible technology basis: MOFA ............ 13
  Other applications using the technology .................... 13

REFERENCES ...................................................................... 14
Genie is the new design analysis tool in SESAM, and was initiated in the Joint Industry Project SESAM 2000 (1994 – 2000) with the following main objectives:

- Ease of operation
- Reduces time for design and analysis
- Better quality in design and analysis
- Support for Quality Assurance of analysis models and results
- Customisable
- Architecture and openness to other systems

Genie is fully owned, maintained and supported by DNV Software, an independent business unit of Det Norske Veritas AS.

By now Genie represents the latest generation design and analysis software supporting designers and engineers. Following the SESAM 2000 project, the development has been motivated and driven by end-user needs for new solutions offering significantly faster modelling, tightly integrated with advanced strength assessment. Genie supports work phases from initial concept studies to mature design and re-analysis:

- Intuitive user interface and strong features for 3D visualisation of model and results.
- Interactive modelling capabilities relevant for design of topside structures, jackets, or similar types of structures.
- Combined plate and beam modelling, intelligent tubular joint design based on user defined rules.
- Easy to interrogate the model by using browser techniques.
- Load application much more flexible by modelling equipments, their footprints, and load transfer rules.
- Flexible handling and converting of units.
- Openness to CAD systems and other analysis systems, with import of section libraries and existing weight list, as well as support for MS Office applications.
- A powerful journalling system based on the Jscript language.
- Integrated analysis and results processing.

Genie builds on DNV Software’s long time experience as a solution provider in the offshore market, as well as recent advances in IT technology. This includes

- ACIS geometry/topology modeller, from Spatial Corporation, USA
- AISC Shapes Database v3.0, from AISC Inc., USA
- DevTools for advanced 3D graphics, from Visual Kinematics, USA

A typical jacket modelled with Genie

Intuitive windows based user interface
MFC for the graphical user interface, from Microsoft Corp., USA
Objective Toolkit for grids and docking of windows, from Rogue Wave Software, USA
ObjectStore PSE Pro for data storage in object-oriented databases, from eXcelon Corporation, USA

These industry standard technologies are combined with DNV Software’s own proven and unique technologies, including

- Finite element mesh generation
- Finite element analysis
- Finite element results visualization
- Environmental loads calculation
- Code checking and rule based design
- Openness towards leading CAD vendors

Genie may be used as a stand alone tool using a direct analysis approach (all modelled in one and same finite element model) where the user can

- model structure, equipments and other loads
- calculate hydrodynamic loads and run structural analyses
- visualise and postprocess results
- perform code checking based on recognised standards

Genie is also ideal for creating parts in a superelement analysis. Typically, topsides and modules are created in Genie and assembled with other parts of the structure like the hull of a FPSO, Semi-submersible, TLP, Spar or similar structure. There are many advantages of using this combined technique; one of the most important is the automatic transfer of the hydrodynamic loads and accelerations back to the topside structure prior to doing the code check.

In this White Paper the technical foundation behind Genie is described in terms of concept modelling (structure and equipment), analysis model creation, and results evaluation. For a full functional description of the program, reference is made to the on-line documentation system for SESAM.
CONCEPTUAL STRUCTURE MODELLING

This section outlines the reasons for and benefits from using conceptual structure modelling in Genie.

Reasoning behind conceptual modelling

A design vision in Genie has been the realisation of *concept modelling* techniques. In previous generation design and analysis software, the end-goal was to perform structural assessment based on the Finite Element Method (FEM). This proved to be highly effective for assessing the status of a given design. However, the associated software solutions had architectural limitations prohibiting efficient integration with CAD systems, and software for rule based capacity checking.

The main problem was a missing vehicle for communication between CAD software, structure analysis software and rule based capacity check software. These different domains had too little in common for efficient information exchange. For example, the CAD system produced a detailed geometry/topology model, the analysis system required a FEM model while the capacity check software could only use specialised capacity models. It was not possible to efficiently communicate model changes across these domain boundaries. The consequence was excessive and costly re-modelling in each domain.

Concept modelling provides a means of overcoming many of these issues, because the user’s design intent is better captured. Instead of representing, e.g. deck structures as element, nodes, faces or edges, the user can model the plates and beams explicitly. These new modelling concepts capture the design intent much more closely, because it is now possible to model such things as whole deck plates or segmented beams as single design concepts. This relieves the end user from tedious and unnecessary work, and makes the model richer since plates with holes, supports, equipments etc. can be modelled explicitly.

The concepts hold information about attributes (for example section profiles or hydrodynamic properties) as well as connectivities to other structural members. When moving a structural part, the connectivity (topology) is automatically updated.

Architecturally, the concept model based software is characterised by its 3 main layers (Concepts, Transfer and Analysis). Another characteristic feature is the subdivision into information domains where the 3 layers are represented in each domain.

These capabilities simplify redesign work significantly, leading to more use of iterative design, and hence to more optimised and cheaper final solutions.
Closing the design loop

Another central vision in Genie is to provide facilities for supporting fast design iterations. This is achieved by offering features for design, modelling, analysis and results evaluation within the same user interface.

The use of concept models is central to performing design iterations, as the FEM analysis results are mapped back to the design concepts, thereby facilitating direct feedback to the next design iteration.

These capabilities simplify redesign work significantly, leading to more use of iterative design, and hence to more optimised and cheaper final solutions.

Example of conceptual modelling – pure beams

The picture to the left shows a K-joint configuration comprising a total of 3 structural members (Bm1, Bm2, Bm3). When creating the model the vertical member (the chord) has been modelled as one member while the inclined beams (the braces) have been modelled by simply typing in coordinate values somewhere along the chord member and at each brace end. The analysis model that is automatically created from the concept model consists of 4 finite elements and 5 finite element nodes.

The structural concepts know their connectivity to other members (and other objects like equipments). Thus, when moving the joint downwards the connectivity and length of braces are automatically updated and the names remain the same. When creating an analysis model, the new finite element model will reflect the changes done to the model.

In other words, the changes are performed at the conceptual model and the analysis model – which is derived from the conceptual model – automatically inherits the changes.

This example shows beams only, but the methods employed are also used for other types of concepts like e.g. plates or equipments. Furthermore the example illustrates use of 3 members only – when moving larger parts of a structure the technique still apply. For example when moving a horizontal middle deck up or down, the lengths of the beams in the vertical plane will either extend or reduce.

The main benefit is that the user only needs to relate to the physical model and not the analysis model.
Example of conceptual modelling – segmented beams

This example illustrates the use of conceptual modelling techniques for segmented beams. There are several ways of creating a segmented member, the one to the left have been created using a top-down modelling approach by splitting the beams into segments and changing section properties afterwards.

The analysis model created consists of one finite element for each segment belonging to the main member.

When changing e.g. the length of a segment, the program will automatically adjust neighbouring segments (the user is in control of which ones will change) to ensure that the main member length is maintained. Similarly, the analysis model will inherit the changes automatically.

The example to the left shows that one of the segments has received a new length simply by typing in the new length – all other changes are automatically accounted for.

Again, the benefit of using conceptual modelling is that the user only needs to relate to the physical model and not details of the finite element model.

Example of conceptual modelling – plate and beams

A topside structure often consists of both beams and plates. The use of conceptual modelling technique will benefit the user to a great extent; some of them are automatic offset of beams and automatic update of connectivity between plates and beams when moving a beam.

The picture shows one plate connected to three beams and the automatically created analysis model. In this case the mesh density has been specified for the plate – the beams will inherit the same mesh density to ensure connectivity between plate and beam finite elements.

When moving the longitudinal beam the program will automatically recalculate the connectivity between plate and beams and create a new analysis model. The user may of course control the mesh configuration in several ways ranging from the simplest feature to specify mesh density of an object(s) through the more advanced option of mesh control using a Jacobian determinant.
Gravitational and inertia loads relevant for a topside structure normally have their background in equipments placed on the structure. Thus, the designer needs to calculate these loads prior to applying them to the analysis model. Genie offers the possibility to model the equipments and to place these to the conceptual structural model for given load conditions. The program will compute the necessary line loads or masses automatically.

This section describes the benefits of using this approach compared to the more traditional load application based on explicit analysis loads.

### From equipment to load

Structural assessment cannot be limited to dealing with structural concepts. Often, the design of the structure will be influenced by other conceptual information, such as heavy equipment. Equipments may be seen as non-deformable objects that generate loads, but do not contribute to structural stiffness (if this assumption does not hold, the equipment should be modelled as a structural entity directly).

In Genie, equipment items are independent concepts that may be modelled explicitly, or imported via weight lists. The equipments have mass and dimensions, and via gravity or other acceleration this mass induces loads. The loads must be distributed over the load carrying interface between the equipment footprint and the structure, and can be represented as line loads applied to the beams part of the load carrying interface.

The footprints may be modelled exactly to ensure a correct load transfer down to the structure. The user may also specify a load pattern rule, or in other words specify parts of the structure that shall not receive loads.

### Equipments can be repositioned

One and the same equipment may be placed in different locations at different times, or simply be repositioned as part of an iterative design process. This requirement is met in Genie, where the equipment is always positioned within the context of a load case, and because it takes advantage of the geometry engine to perform boolean operations, i.e. calculate the load carrying interface between structure and equipment. This is a fully automatic operation.

Once the load carrying interface is established, the load generated by the equipment can be calculated. First, the total equipment force is calculated as $F_{\text{equipment}} = M_{\text{equipment}} \times \text{gravity}$. This force is then distributed over the load carrying interface, while ensuring force and moment equilibrium. Thus, an intermediate load representation of the equipment has been established, suitable for discretisations in Genie’s finite element mesh generator.
The equipment now ensures a correct mass matrix (mass and centre of gravity) – no moments transferred to structure when subjected to accelerations.

Another example where automatic load re-calculation is of importance is when inserting extra supporting members beneath equipments. The load carrying interface then changes and new loads are calculated to ensure a correct force and moment equilibrium.

When the acceleration contains a horizontal part, shear forces as well as a force pair is automatically calculated – the vertical location for the centre of gravity plays an important role.

The final step in transforming the structure with equipment into a FEM representation is performed in the mesh generator, where the structure is subdivided into discrete elements and the load is subdivided into corresponding discrete loads.

With this process, we have achieved the goal of transforming the conceptual representation of the design model into a Finite Element representation with accurate loads. It is straightforward to assess the structural integrity of a given equipment design choice, paired with the current structure design, since the results evaluation is performed in the same user environment as used for structure and equipment modelling.

**Alternative equipment representations**

Representing equipment as loads is not always suitable. Sometimes, it is necessary to represent the equipment as a mass even in the FEM model, e.g. when the structure is subject to dynamic loads or large rigid body motions. Genie allows the user to select a mass representation when required, and the mesh generator will then generate suitable mass elements in the FEM model. Such models may then be used to calculate e.g. proper structural eigenvalues or proper rigid body motions if subject to environmental wave loads.

This feature is often used when a Genie model is connected to a hull in a hydrodynamic and structural analysis. The analysis model created from Genie now contains a correct mass model rather than a force model – the mass and location of centre of gravity are important for a dynamic structural or hydrodynamic analysis.

The equipment is now represented by a mass element connected to the structure with beam members – hinges are automatically inserted at the beam ends to ensure transition of vertical and lateral acceleration forces only.

Using the superelement analysis features of SESAM, the acceleration forces are automatically applied to the topside simplifying the work to do code checking or fatigue analysis. Of equal importance is the increased quality built into the analysis since manual input of loads is highly error prone.

There may be other reasons for applying loads different than using equipments. For this Genie supports a full range of explicit loads like point loads, line loads, surface loads, temperature loads, and prescribed displacements.
Analysis models are derived automatically from the conceptual model. The user may influence the finite element representation by specifying the mesh density or via other means of controlling the quality of the finite element mesh.

**Deriving analysis models from concept models**

Genie provides concept modelling and features for automatic generation of Finite Element analysis models. The user may control the mesh generation through mesh density parameters and feature edges, but often this is not required. The translation involves conceptual information from several domains, such as for example:

- Plates and segmented beams
- Shared properties such as sections and materials
- Supports are translated into boundary conditions
- Equipment objects are translated as loads or mass
- Load cases

The translation of a concept model into a FEM model must be complete and meaningful. This means that for example adjacent plates must be connected via common in the FEM model. The mesh generation software achieves this through tight integration with the geometric/topological description of the concepts.

**Top-down concept modelling and FEM derivation**

The figures to the left illustrate how a simplified top-down approach to modelling a structure can be employed in Genie. Here, plate concepts are used, and the resulting model is realised by automatic intersection calculations and hole-punching operations, which are supported by the underlying geometry engine.

This example also illustrates how the FEM mesh generation features allow plates with holes to be handled without need for direct user intervention (although such intervention is possible). All plate-to-plate connectivities are automatically taken care of through mesh generation. These principles still apply for more complex and mixed beam/plate models.

Top-down modelling techniques offer several advantages over more traditional bottom-up techniques. These advantages include speed (many detail tasks can be delegated to the software), quality (same data entered only once) and a migration path from initial to more mature design since details can be added later in the design process.

Even if top-down modelling is supported in Genie, it is not enforced. In situations where a bottom-up approach is more appropriate, it can be employed also in Genie. In either case, a FEM mesh can be derived.
Given a consistent Finite Element model the user may run a linear static analysis (direct analysis approach) directly from the Genie user interface. The analysis module Sestra /1/ is used as a background service and the user has direct access to analysis details for verification purposes. Alternatively the Finite Element Model may be used in a superelement analysis (hydrodynamic or structural analysis) or by other programs, such as Usfos /2/ or imported to Patran-Pre /3/.

**Analysis and Results Evaluation**

Strength assessment implies being able to visualise the structural response of a given design with regard to applied loads generated by the environment, or equipment or other loads.

Genie provides integrated FEM results visualisation with coupling to the concept model, i.e. fundamental results such as displacements, stresses and forces, but also derived results such as principal stresses and beam moment diagrams.

These results can be presented for any selected load case as contour plots, vector plots, and numerical annotations on un-deformed or deformed model. Such visualisation can be done for any concept model subset.

These capabilities form the fundamental results evaluation features required for providing feedback to the next design cycle. Coupling to rule-checking software (Framework /4/) exists today, and further integration will be provided in the form of integrated rule checks.

**RESULTS EVALUATION**

*Results may be presented for any concept, any model subset, or the whole model.*

- Presenting contour plots of displacements for whole model
- Presenting contour plot of VonMises stresses at bottom plate surface
- Presenting beam shear forces as
- Presenting beam moment diagram for whole model
- Presenting beam moment diagram for single member – numerical values included
DNV Software has developed its own software foundation for developing software tools dealing with design analysis of offshore and marine structures. This section explains the basic principles behind the foundation MOFA.

**Building on a flexible technology basis: MOFA**

The previous sections assumed that the modelling operations were performed within Genie, where a meshable topology is maintained by default. This assumption does not always apply when the conceptual model is defined elsewhere, e.g. in a CAD system. In this case the technology used in Genie can be employed to support such scenarios.

Genie is an application built on the MOdel Foundation Architecture (MOFA), offering conceptual modelling capabilities and a state-of-the-art geometric and topological engine, as well as Finite Element mesh generation and results retrieval/visualisation. The geometric and topological engine is built on ACIS (from Spatial Technology), the industry standard CAD technology, also used in IntelliShip.

The architecture is well positioned to implement the non-trivial transition from CAD to FEM and back. For this purpose, the key capabilities can be summarised as:

- A state of the art geometry/topology engine (based on ACIS) with strong support for geometric Boolean operations, which makes it possible to resolve issues such as non-connectivity in the original CAD model, essential for mesh generation.
- State-of-the-art mesh generation capabilities with support for load application, boundary conditions and complex geometries with internal holes.
- Direct integration with standard SESAM structural analysis software.
- Coupling between concept model details and FEM model details, allowing per-concept results assessment.

**Other applications using the technology**

The MOFA technology is also used as the basis of DeepC /5/, an application integrating riser and marine operation software to provide fully coupled riser and mooring analyses.

MOFA is also being employed in ongoing development work to provide faster, better and tighter integration between NAUTICUS Hull /8/ concept models and FEM analysis.

Wasim /6/, a program for computation of wave loads and seakeeping based on a fully three-dimensional solution, also uses MOFA.

The ongoing development work to provide a graphical front-end to Wadam /7/ is also based on MOFA.
REFERENCES

1. DNV Software “Sestra User Manual”
2. DNV Software “Usfos User Manual”
3. DNV Software “Patran-Pre User Manual”
5. DNV Software “DeepC User Manual”
6. DNV Software “Wasim User Manual”
7. DNV Software “Wadam User Manual”
8. DNV Software “Nauticus Hull User Manual”
Visit us at:

Head office:
Oslo
DNV Software
Veritasveien 1
NO-1322 Havik, Norway
Tel: +47 67 57 76 50
Fax: +47 67 57 72 72

DNV Software regional offices:

Busan
Det Norske Veritas
DNV Software
Nambusan P.O. Box 120
Busan 613-011
Republic of Korea
Tel: +82 51 610 7700
Fax: +82 51 611 7172

Houston
DNV Software
16340 Park Ten Place
Suite 100
Houston, Texas 77084-5132
USA
Tel: +1 (281) 721 6700
Fax: +1 (281) 721 6880

Kobe
Det Norske Veritas
DNV Software
Port P.O. Box 77
Kobe 651-0191
Japan
Tel: +81 78 291 1305
Fax: +81 78 291 1330

Kuala Lumpur
Det Norske Veritas
DNV Software
24th Floor, The Weld Tower
Jalan Raja Chulan
50200 Kuala Lumpur
Malaysia
Tel: +60 3 2050 2888
Fax: +60 3 2031 8080

London
DNV Software
Palace House
3 Cathedral Street
London SE19DE
United Kingdom
Tel: +44 (0) 20 7716 6525
Fax: +44 (0) 20 7716 6738

Marseille
Det Norske Veritas
DNV Software
16 Impasse Blancard
13007 Marseille
France
Tel: +33 (0) 4 91 13 71 66
Fax: +33 (0) 4 90 54 46 89

Rio de Janeiro
Det Norske Veritas
DNV Software
Rua Sete de Setembro
111/12 Floor
20050006 Rio de Janeiro
Rio de Janeiro
Brazil
Tel: +55 21 2517 7232
Fax: +55 21 2221 8758

Shanghai
Det Norske Veritas
DNV Software
House No. 9,
No. 1591 Hong Qiao Road
Shanghai 200336
China
Tel: +86 21 6278 8076
Fax: +86 21 6278 8090

Taiwan
Det Norske Veritas
5F-3 No.160 Sec. 6,
Minquan E. Rd
114 Taipei
Taiwan
Tel: +886 2 2792 5352
Fax: +886 2 2792 5357

DNV Software is the commercial software house of DNV serving more than 3,000 customers in the marine, offshore and process industries.
DNV Software is a market leader in software development of design, strength assessment, risk and information management.