

# Nauticus Hull Prescriptive Nauticus Hull Finite Element

Sales presentation

Ragnar Eimhjellen

31 May 2023

# Content of this presentation

| Main topic                                         | Keywords                                                                                                                                                | Duration   | Target customer                                   |
|----------------------------------------------------|---------------------------------------------------------------------------------------------------------------------------------------------------------|------------|---------------------------------------------------|
| <b>DNV's Calculation tools for ships</b>           | 3D approval, OCX, NH Prescriptive package, NH Finite Element package, price policy, digital sales, training and support, NH + GeniE for container ships | 30 minutes | High level introduction to managers and engineers |
| <b>Nauticus Hull Prescriptive</b>                  | GUI, 3D geometry engine, Re-use data, Section Scantlings, Transverse bulkheads, 3D model, export to GeniE, Rule calc. tools, 3D Beam                    | 25 minutes | Introduction to managers and engineers            |
| <b>Nauticus Hull Finite Element</b>                | GUI, Concept modelling, Ship configuration, all Rule sets covered, efficient tool for CV, *.bdf import                                                  | 15 minutes | Introduction to managers and engineers            |
| <b>Container ship cargo hold model</b>             | Modelling, load templates, analysis, Rule checks, design iterations, report                                                                             | 15 minutes | Container ship designers                          |
| <b>Gas fuelled container ship with type-B tank</b> | Temperature loads, load templates, friction in vertical supports, clearances in supports, sub-models, FE assessments                                    | 15 minutes | Container ship designers                          |
| <b>Container ship global model</b>                 | ShipLoad interface, Rule checks                                                                                                                         | 5 minutes  | Mainly internal                                   |
| <b>FE Rule checks for CSR BC&amp;OT</b>            | Yield screening, fine mesh yield check, fatigue screening, automatic mesh refinement, FE Fatigue assessments                                            | 15 minutes | Designers also focusing on CSR vessels            |
| <b>Modelling features in GeniE</b>                 | Dual assembly, slot/lug and hole concepts, plate seam concept, non-prismatic outer shell, sets, clipping planes, verification tools                     | 15 minutes | Engineers with high FE knowledge                  |
| <b>Meshing in GeniE</b>                            | Conditional re-meshing, Mesher optimised for ships, idealise model inaccuracies, feature edges, mesh properties, mesh editing tools                     | 10 minutes | Engineers with high FE knowledge                  |

# DNV's Calculation tools for ships

DNV – the leading provider of classification calculation tools

Nauticus Hull – in the middle of the design process

Re-use of the full design model - OCX import

Intersect OCX imported 3D model for analysis

Nauticus Hull Prescriptive package

Nauticus Hull Finite Element package

Price policy

Digital sales of Nauticus Hull

Training and support easily available

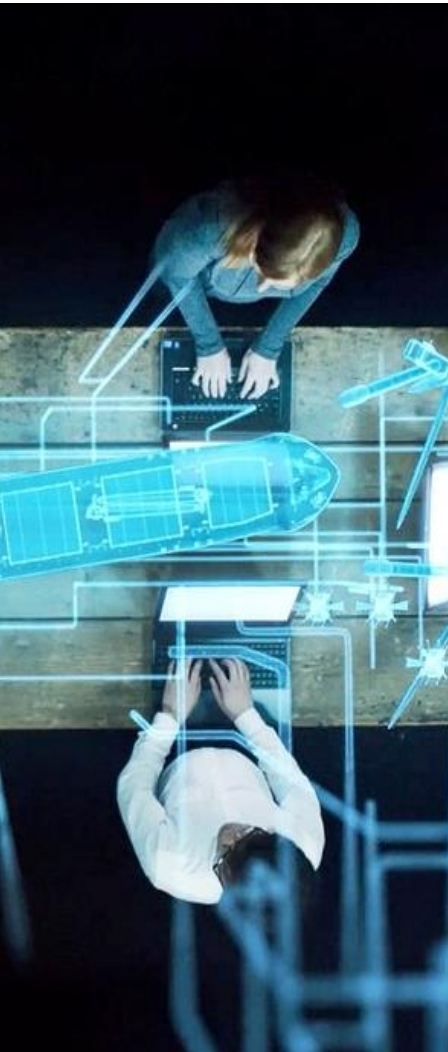
Nauticus Hull and Sesam GeniE for Container ships

# Digitalization – a game changer for all industries

A word cloud featuring various terms related to digitalization and technology. The background is a vibrant blue and purple gradient with dynamic, streaky light effects. The words are in different colors (green, blue, white) and sizes, creating a sense of movement and energy. The most prominent words are 'machine learning', 'artificial intelligence', 'digital transformation', and 'big data'. Other visible terms include 'data analytics', 'predictive analytics', 'robotics', 'cloud', 'IoT', 'blockchain', 'hadoop', 'NoSQL', 'digital twin', 'sensors', 'modeling', 'visualization', 'business analytics', 'data lake', 'streams', 'executive dashboards', 'connectivity', 'prescriptive analytics', 'data engineering', 'computing power', 'algorithm', 'spark', and 'big data'.

big data  
data analytics  
machine learning  
digital transformation  
artificial intelligence  
cloud  
IoT  
prescriptive analytics  
data engineering  
predictive analytics  
robotics  
NoSQL  
digital twin  
sensors  
modeling  
visualization  
business analytics  
data lake  
streams  
executive dashboards  
connectivity  
blockchain  
hadoop  
spark  
algorithm

# ...digital shift in ship design

A person is shown from above, working on a laptop. A translucent 3D model of a ship's hull is overlaid on the laptop screen, with lines connecting it to various data points and components, illustrating the digital design process.

## **Computational capabilities and data storage**

enables more advanced modelling and simulation tools and for design and optimisation

## **3D models in early design**

CAD models are replacing technical drawings and documentation

## **FE analysis**

trend towards more direct calculations and global FE analysis

## **Scientific based rule formulations**

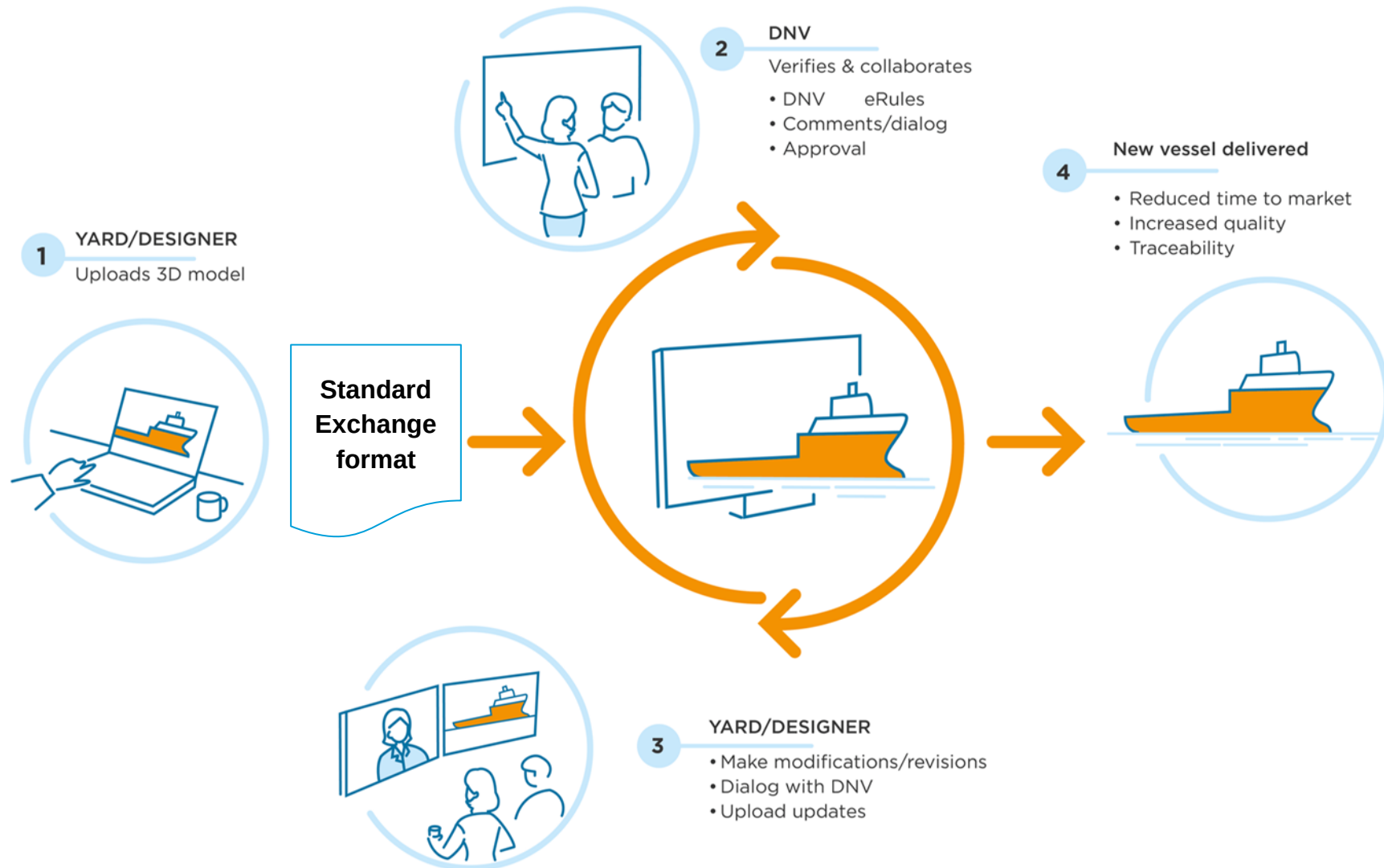
enhanced framework to optimize the design and develop new concepts

**Re-use of models, better collaboration, fast iterations, optimized design**

# Can we replace 2D drawings with a 3D model as design documentation?



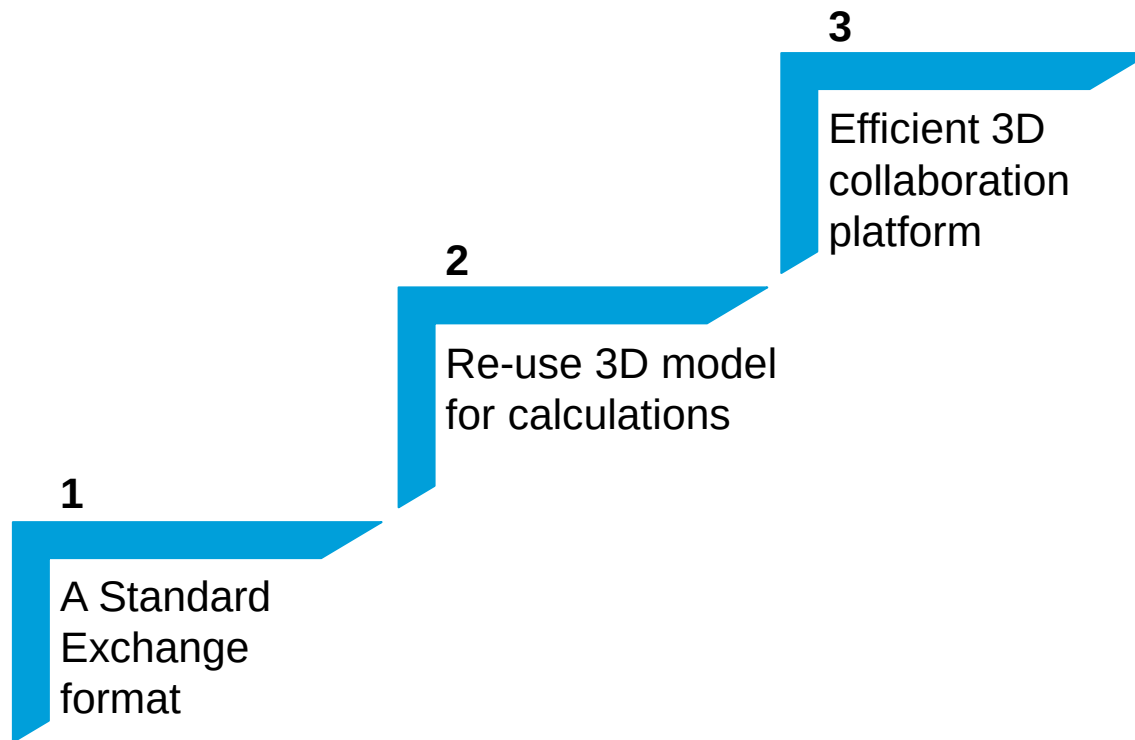
# Model based approval – a design centric work process



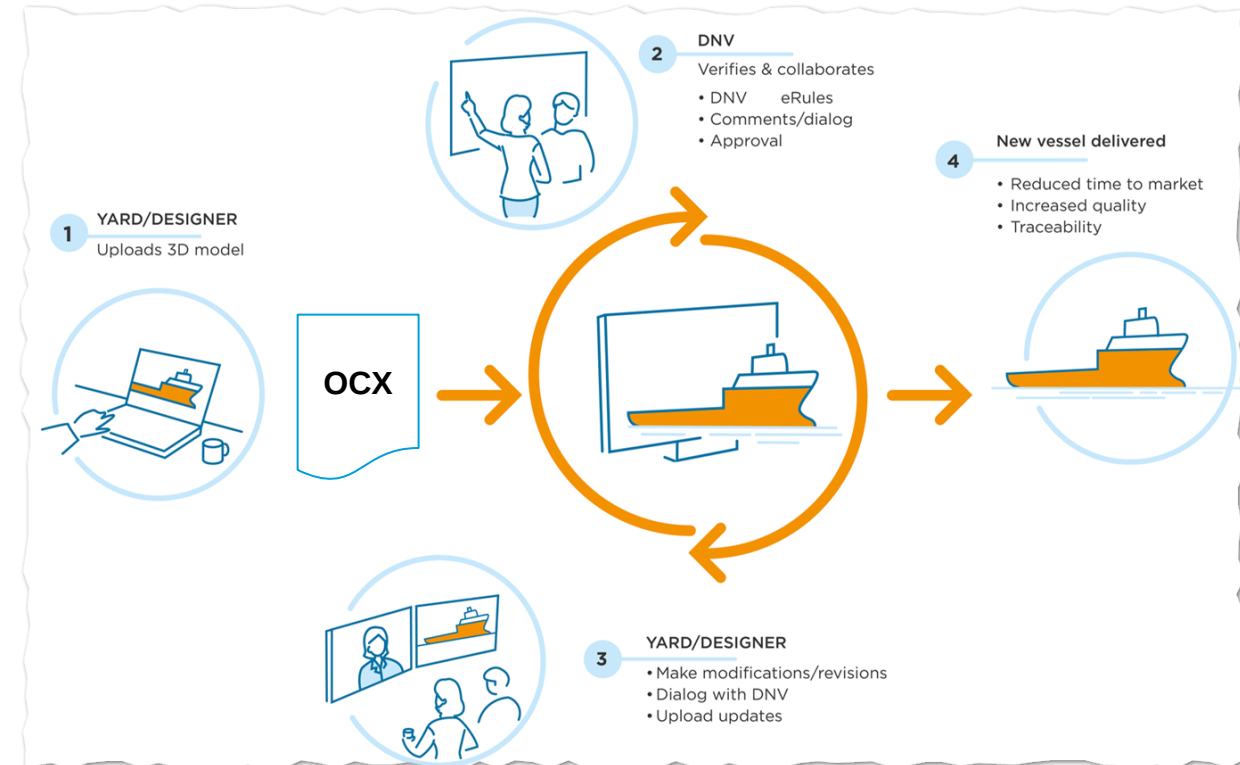
## Benefits:

- Allowing **Designers and Class to engage** in a fully digital work process
- Enable **early involvement of Class** in the design process
- Saving **customer's time** by reducing the need for 2D class drawings/documents
- **Re-use CAD models** for prescriptive and FE calculations

# Three steps are required to succeed to replace 2D drawings



## 3D model as design verification



# OCX “The Open Class 3D Exchange standard”

*A vessel specific standard addressing the information needs of the Classification Society is a key enabler to replace traditional 2D drawings with a 3D model as design documentation submitted for verification*

Enable exchange and sharing of  
3D models

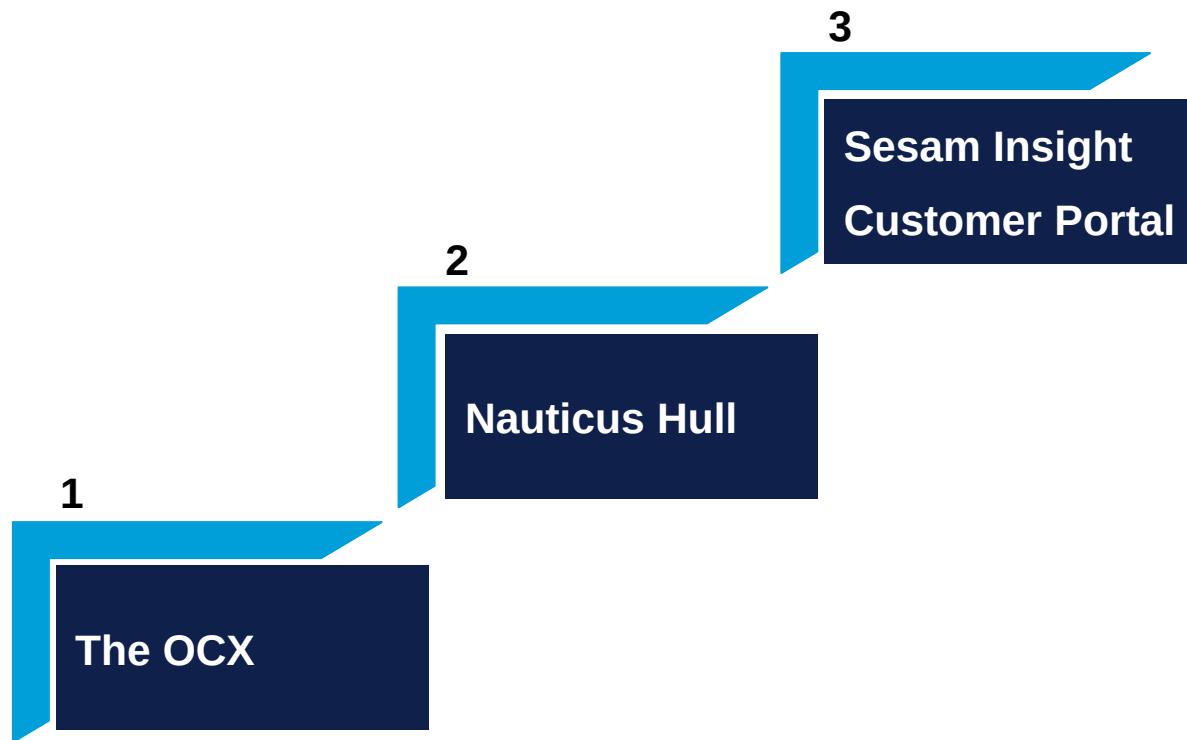
Share to scale

Speed up process of design  
verification

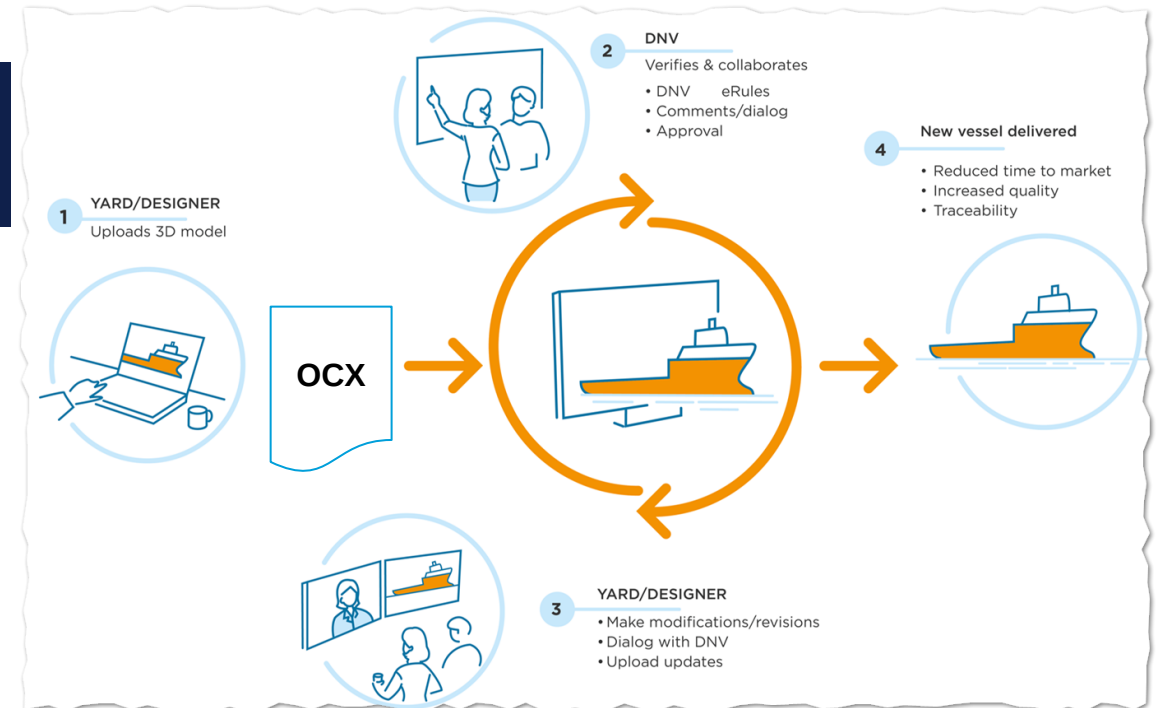
Enhance collaboration and  
improved customer experience



# The DNV 3D Approval Platform

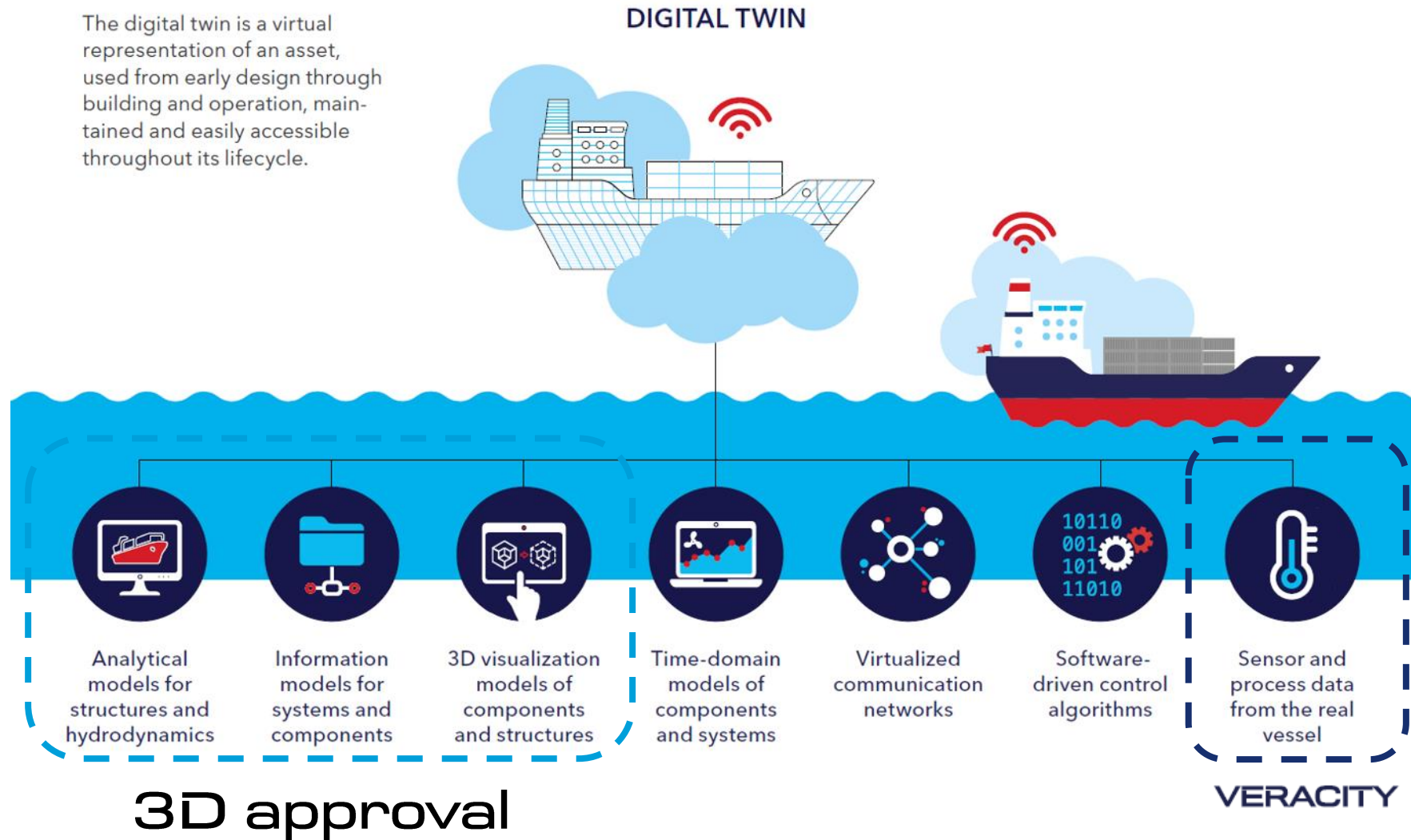


## 3D model as design verification



# 3D approval – the birth of the “Digital Twin”

The digital twin is a virtual representation of an asset, used from early design through building and operation, maintained and easily accessible throughout its lifecycle.



# DNV's Calculation tools for ship

DNV – the leading provider of classification calculation tools

Full support for **DNV ship rules** and **CSR BC & OT** with powerful rule check capabilities for **documentation of rule compliance**

Efficient **ship modelling** capabilities for prescriptive and direct strength calculations supporting customer's design process with strong capabilities for **design iterations**

**Re-use** of customer models through the **OCX exchange format** enabling integration with **industry standard CAD and CAE tools**

**Flexible solution** covering different design approaches from spot check to full finite element analysis

**Modern user interface** and architecture with ability to scale on latest technology advancements



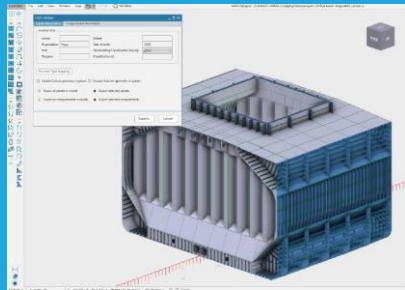
# DNV's Calculation tools for ship

## Nauticus Hull – in the middle of the design process

*Calculation Tools' development of the OCX exchange standard, closely integrated with Nauticus Hull, allows for re-used of 3D models created in leading CAD tools*

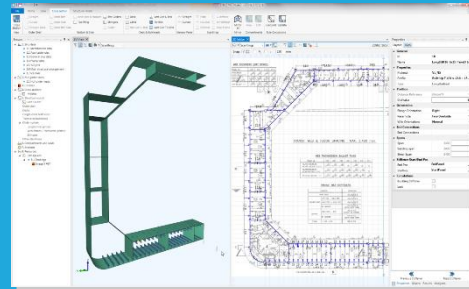
- Among leading shipbuilders, trend to create 3D Computer Aided Design (CAD) models
- DNV's Calculation tools allows for re-use of such models in Nauticus Hull for Rule checks via the OCX exchange format

### CAD system



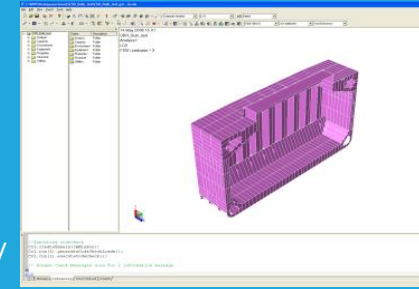
### Nauticus Hull

#### Nauticus Hull Prescriptive



#### Rule Check

#### Nauticus Hull Finite Element

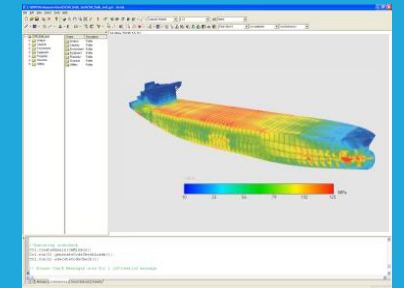


#### GeniE

Integration  
prescriptive/  
FEA

### Sesam

#### Advanced CAE analysis



### Re-use the full design model

- By Import of the [3D OCX exchange](#) format

### Utilize design data

- Import of IGES 3D hull form data
- Import of structural drawings
- Background for modelling
- Rule calculation on drawing

# DNV's Calculation tools for ship

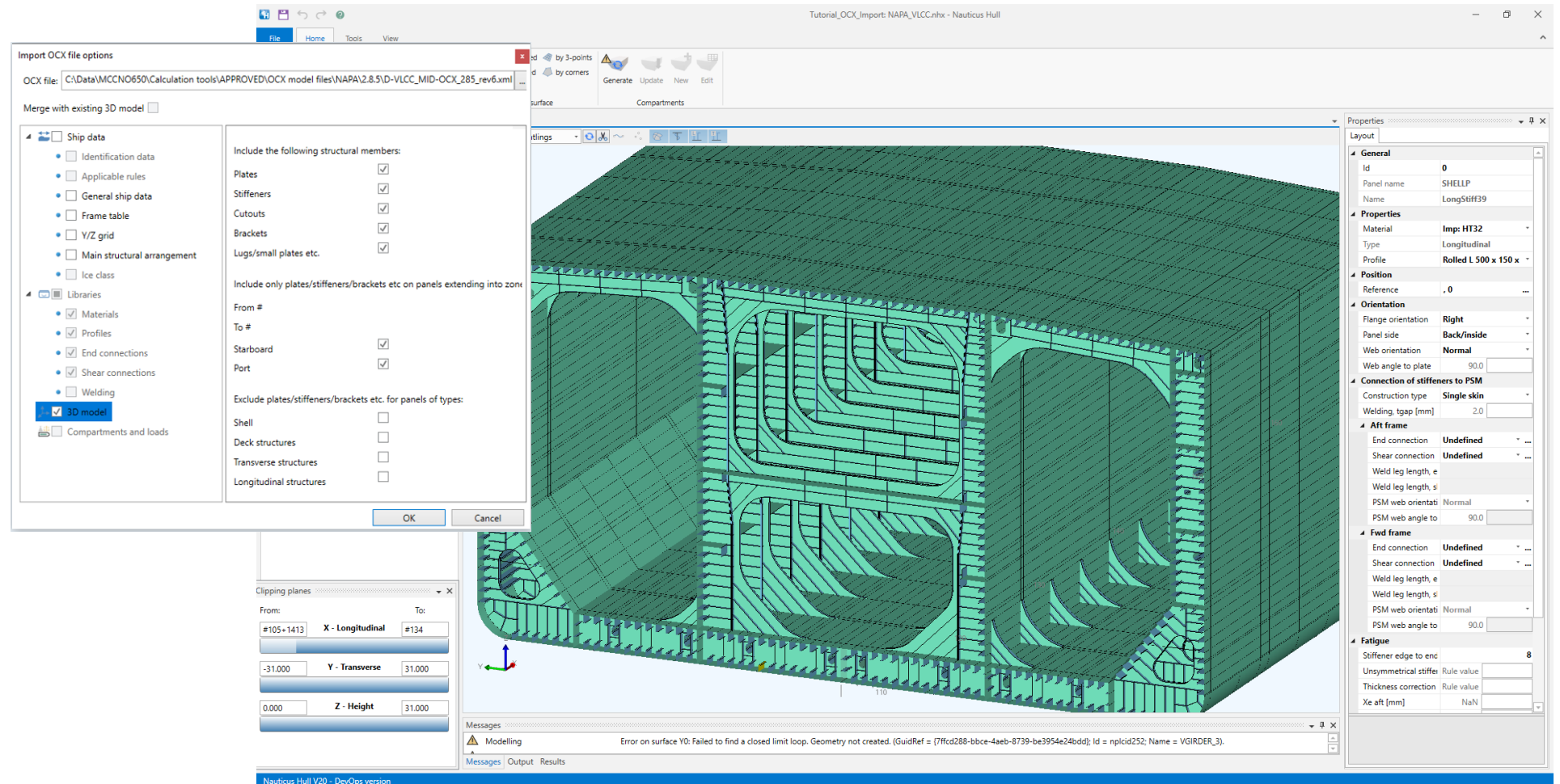
## Re-use of the full design model - OCX import

Nauticus Hull fully supports the OCX exchange format, and leading CAD vendors have been on-boarded, being able to export their models via OCX

### • Full support of OCX version 2.8.6 including:

- Main ship data
- Frame table
- Material types
- Segment types
- Geometry
- Plates
- Profiles
- Openings and cut-outs
- End connection details
- Shear connections
- Compartments

### CAD model imported to Nauticus Hull via OCX



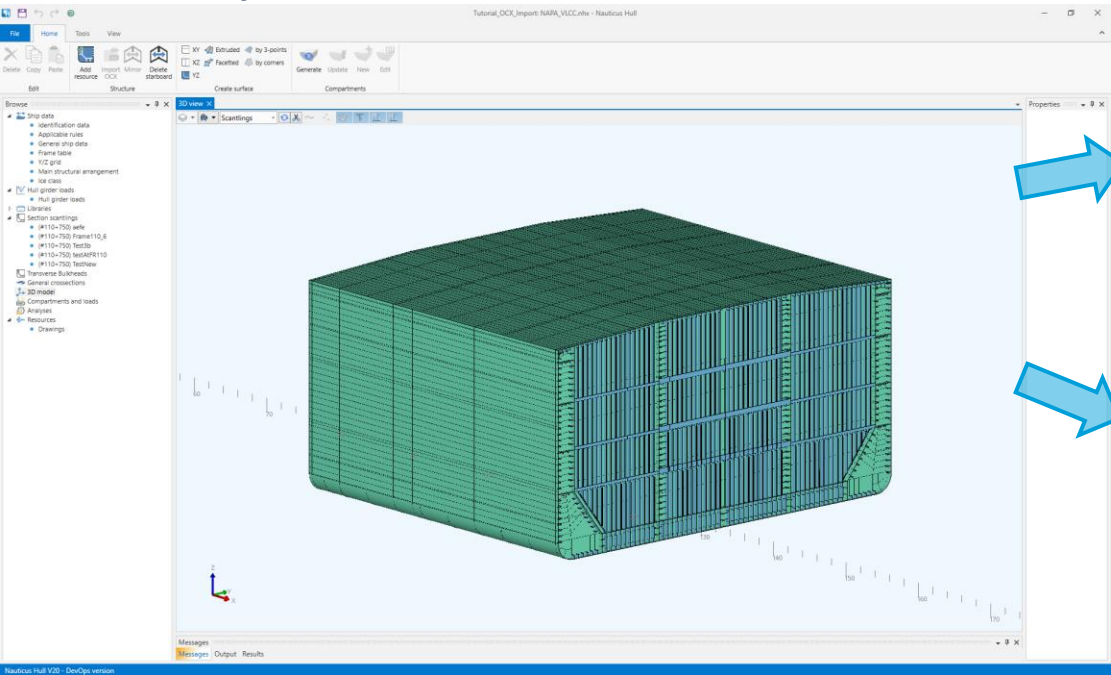
# DNV's Calculation tools for ship

## Intersect OCX imported 3D model for analysis

DNV's introduction of the **Approved project**, with the objective to use 3D CAD models as basis for calculations and approval, will significantly improve the customer interaction

- Intersect 3D model in Nauticus Hull for automatic creation of
- Cross Sections in Section Scantling tool
- Plane bulkheads in Transverse Bulkheads tool

### 3D model imported to Nauticus Hull via OCX



### Intersect 3D model

New cross section

Name: TestCSA Position #: #110+750

☒ Intersect imported hull definition

Adjacent main structural positions

2. aft: #109  
1. aft: #110  
1. fwd: #111  
2. fwd: #112

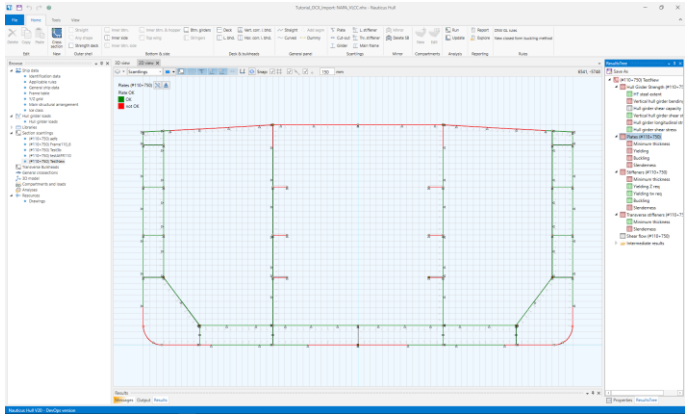
Add position

Span of longitudinals: 1500.0 [mm]  
Standard spacing between trv. stiffeners: 800.0 [mm]

☐ Midship cross section  
☐ Increased permissible hull girder stress outside midship region

Ok Cancel

### Section Scantlings analysis



### Intersect 3D model

New transverse bulkhead

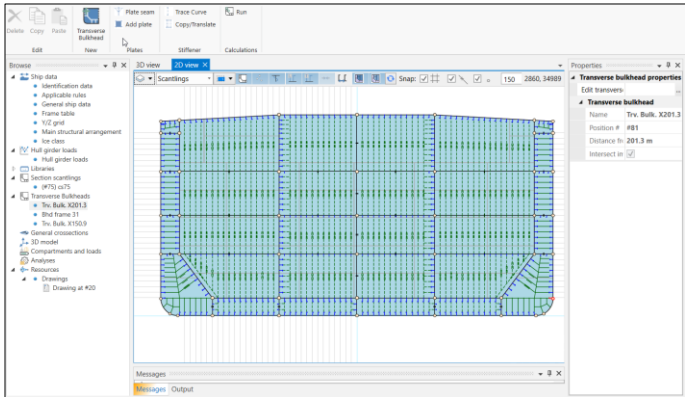
☒ Intersection 3D model

☐ Import cross section

Name: (#72) X150.9  
(#81) X201.3

Ok Cancel

### Transverse bulkhead analysis



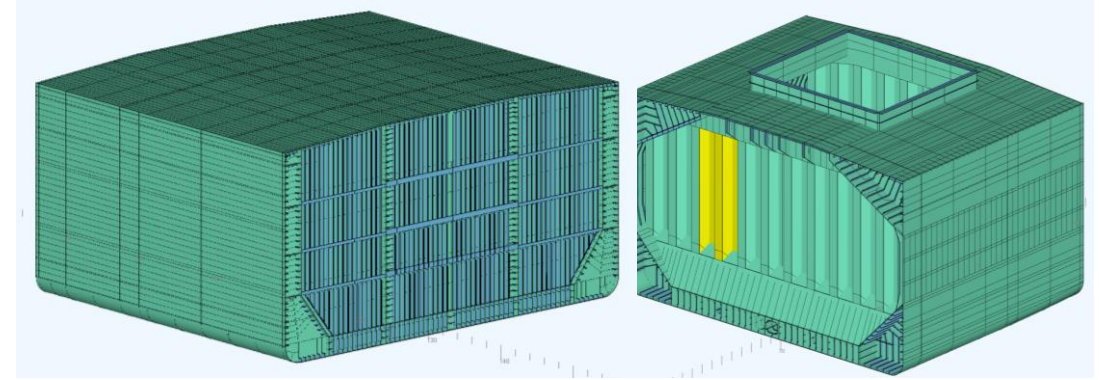
# DNV's Calculation tools for ship

## Nauticus Hull Prescriptive package

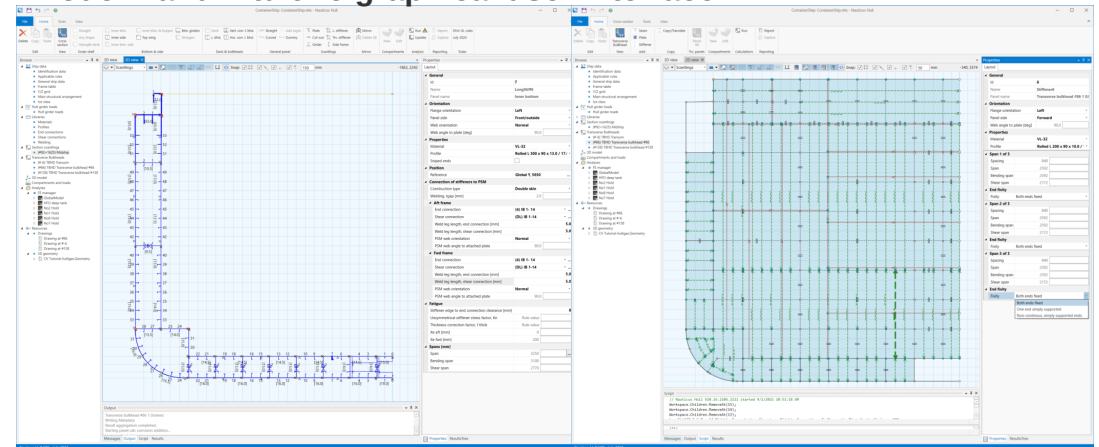
*DNV is committed to utilise the protentional efficiency gains introduced by increased digitalisation among leading shipbuilders' design processes*

- One hull calculation tool for all ship types supporting the approval process in DNV and the design process for designers and yards
- Nauticus Hull essential in the on-going Approved project (use 3D CAD models for calculations and approval)
- Re-use of 3D CAD models via the OCX exchange format
- A modern and future proof technology platform
  - New uniform user interface
  - Scripting of commands
  - Multithreading for improved performance
- Section Scantlings / Transverse Bulkhead / 3D modeller
  - Interactive modelling, undo/redo, modelling on drawing
  - Modelling, analysis and reporting in one
- New functionality for prescriptive cross section calculations
  - Bow impact reading hull shape (angles) from 3D model
  - Baltic and Arctic ice class rules
  - Connection area requirements
  - Wheel loads and UDL
- Import of drawings and hull form
  - Used in background for modelling
  - Rule calculation on drawing

**3D CAD models imported to NH via OCX exchange format**



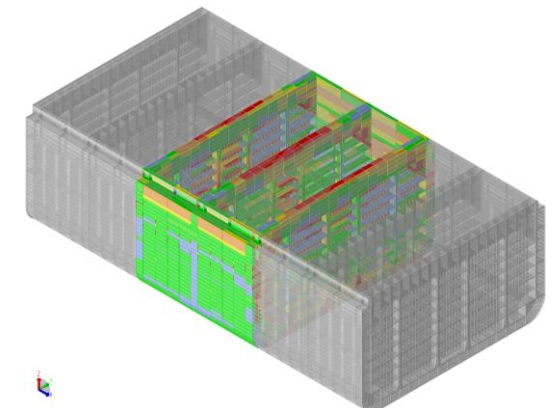
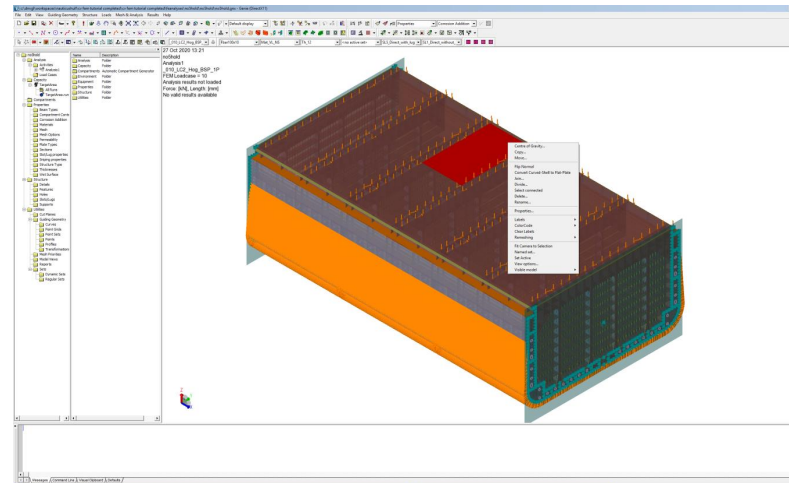
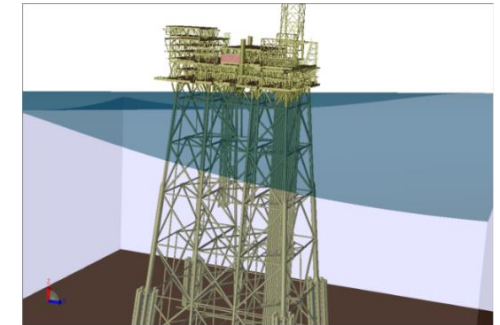
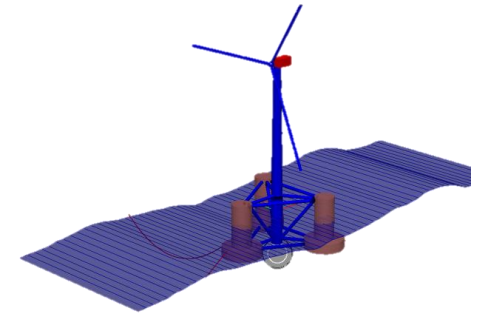
**Modern and intuitive graphical user interface**



# DNV's Calculation tools for ship

## Nauticus Hull Finite Element package - part of Sesam

- GeniE is a part of the Sesam ecosystem fully owned, maintained and supported by DNV
- Sesam with more than 50 years of experience in strength assessment, hydrostatic and hydrodynamic analysis
- More than 300 organisations use Sesam, including DNV approval engineers
- More than 70 DNV engineers develop, maintain and support Sesam
- The **Nauticus Hull Finite Element package** is a *ship configuration of GeniE*
- *Ship configuration of GeniE*, customised for the shipbuilding industry by removing not applicable features
- GeniE – A modern and future proof technology platform
  - Conceptual modelling, allowing for rapid modelling and change to a model
  - Mesh is automatically updated when updating the loads/geometry
  - Using modern architecture
  - Real 3D modeller
  - Undo / Redo – not limited to only one command
  - Scripting
  - Supports backward compatibility
  - Multithreading of analysis tasks for improved performance

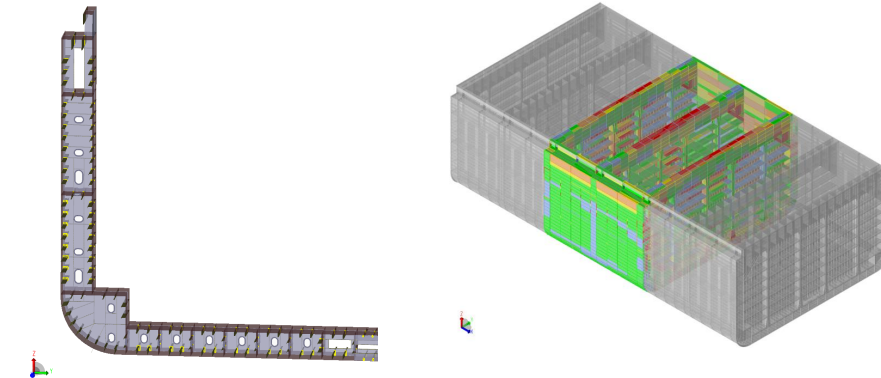


# DNV's Calculation tools for ship

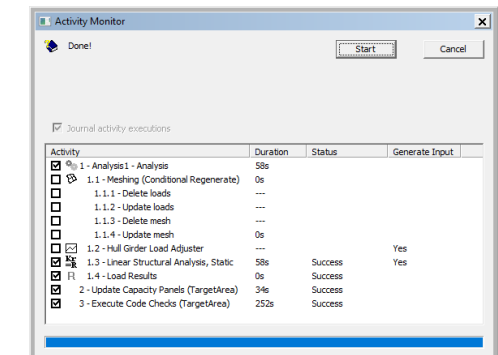
## NH Finite Element package - optimised for ships

- Only one file needed (workspace file) for storage and exchange of data
- GeniE ship configuration, for improved user-friendliness
- All FE rule checks implemented (buckling, yielding, FE fatigue, yield/fatigue screening)
- Import of FE models from other tools via the bulk data file (\*.bdf) format
- Robust 3D geometry kernel for modelling of non-prismatic hull (surfaces and stiffeners)
- Conceptual representation of holes and slot/lugs used during meshing and Rule checks
- High meshing performance by mesher developed by DNV, and conditional regeneration of mesh
- Strong integration with Nauticus Hull ensures re-use of data and geometries
- Automatic creation of loading conditions, corrosion additions and boundary conditions
- High analysis performance ensured by Sestra solver developed by DNV
- Automatic creation of buckling panels, high ABC performance and numerous result display options
- Easy workflow on design iterations (all steps controlled by GeniE's activity monitor)
- Automatic load transfer and prescribed displacements from cargo hold model to sub model
- Fully supports type-B tank design (for gas fuelled CV): initial clearances in anti-roll/pitch/floating supports, and friction in vertical supports

**Hole and slot/lug concept FE buckling results**



**Required analysis steps controlled  
by GeniE's activity monitor**



# DNV's Calculation tools for ship

## New and competitive price policy for GeniE ship configuration

### Nauticus Hull Prescriptive package

- Includes:
  - Nauticus Hull
  - 3D Beam
  - PULS
- Price policy for yearly rental, May 2023
  - Node locked: 1 515 Euro
  - Network: Node locked / 0.6 = 2 530 Euro

### Nauticus Hull Finite Element package

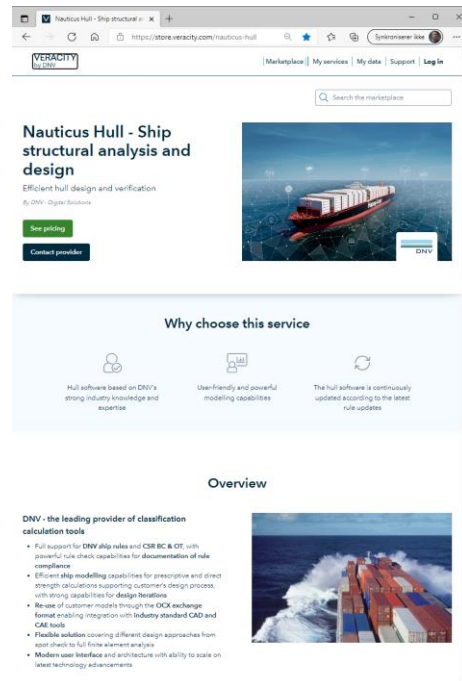
- Includes:
  - **GeniE Ship mode configuration**
  - Sestra (solver, linear and T/C analysis)
  - Submod (tool for transferring prescribed deformations from global mode to sub model)
- Price policy for yearly rental (need NH prescriptive package in addition), May 2023
  - Node locked: 7 250 Euro
  - Network: Node locked / 0.6 = 12 081 Euro

# DNV's Calculation tools for ship

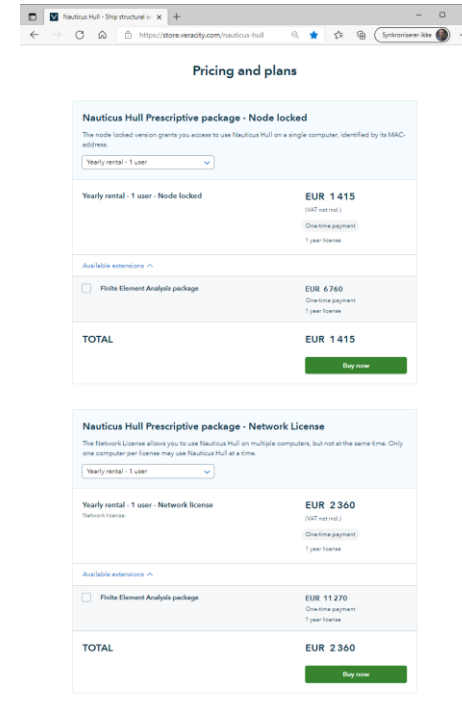
## Digital sales of Nauticus Hull

- An internet landing page for digital sales of **Nauticus Hull Prescriptive package** and **Nauticus Hull Finite Element package** is available: [Nauticus Hull - Ship structural analysis and design | Veracity by DNV](https://store.verity.com/nauticus-hull)
- Licences can be purchased by click and accept, pay by credit card or by invoice

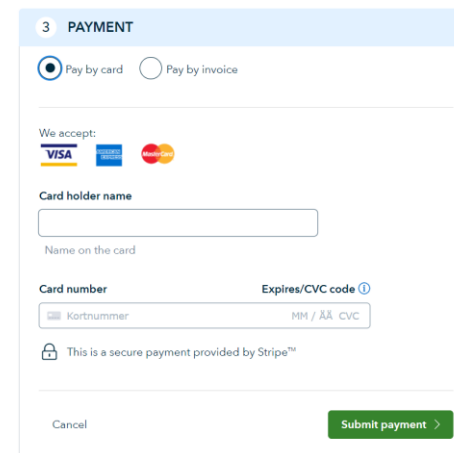
### Internet landing page for digital sales of Nauticus Hull



### Purchase on the internet



### Pay by credit card or invoice



*DNV has a world wide support organisation, with quick response, minimising the time loss when you need support*

- Tutorials (also available as training courses)
  - Container Ship cargo hold finite element analysis
  - Container Ship cross-section analysis
- 24-hour worldwide software support available
- In Nauticus Hull and GeniE, direct links to
  - User manuals
  - Customer portals with tutorials, etc

DNV-GL

---

**Sesam GeniE**

**Tutorial – cargo hold finite element analysis**

*Container ship midship model with typical hold arrangement*

---

NAUTICUS HULL

TUTORIAL

---



**Containership  
Cross-section  
analysis**

[illegible]

# Nauticus Hull Prescriptive

Nauticus Hull's modern Graphical User Interface

Nauticus Hull's 3D geometry engine

Re-using designer's design data

Sharing common data and model files

Section Scantlings

Transverse Bulkheads

Prescriptive 3D model

Export to GeniE

FE load templates for automatic load generation

Rule Calculation tools

3D Beam

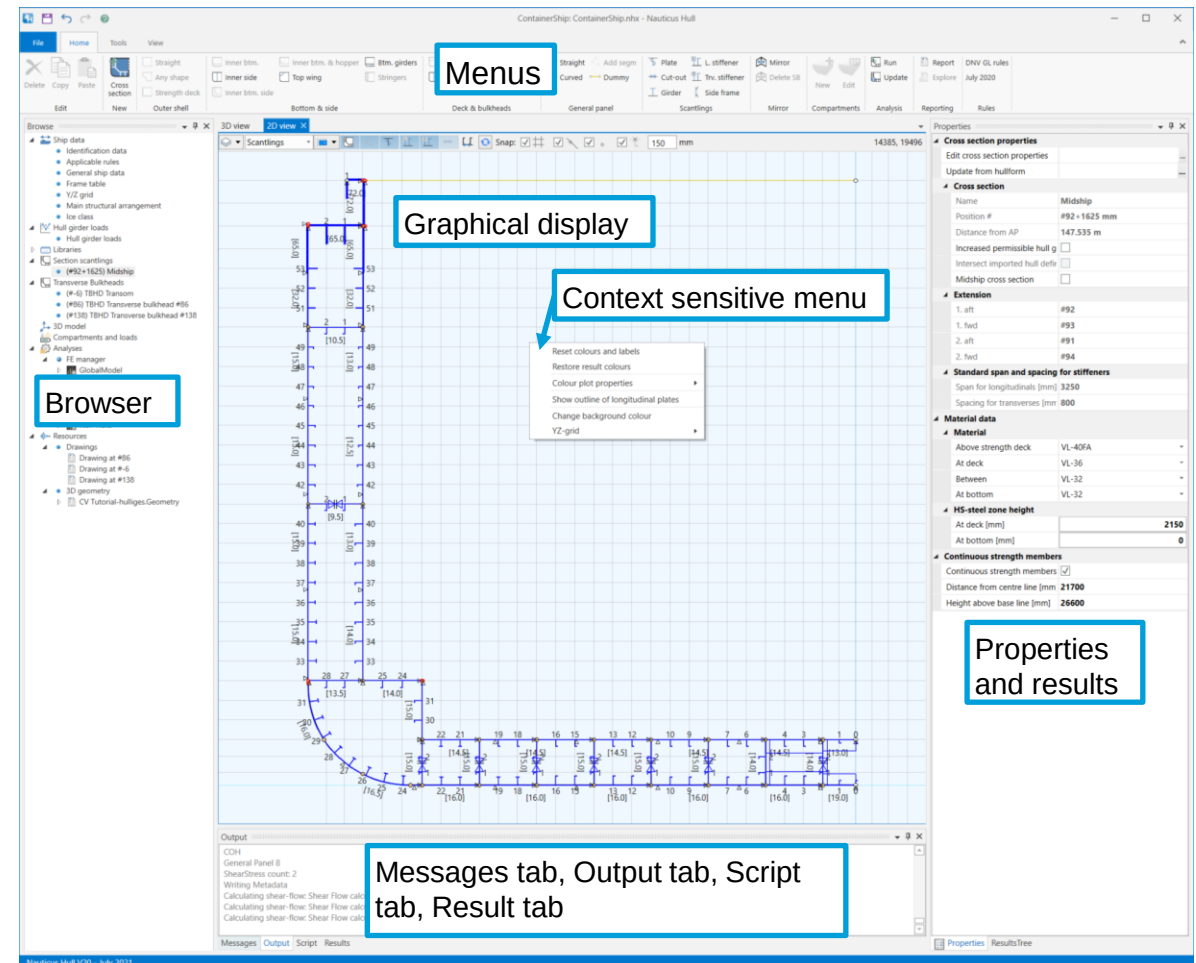
# Nauticus Hull Prescriptive

## Nauticus Hull's modern Graphical User Interface

The GUI in Nauticus Hull consists of:

1. **Browser** giving access to *main functionality*
2. **Menus** containing commands, depending on *main functionality*, for file operations, modelling, analysis, etc.
3. **Graphical display**
4. **Context sensitive menus**
5. **Properties and results**
  1. **Properties:** Modify and inspect properties for selected object(s)
  2. **Results:** Selection of various main result options
6. **Tabs** for
  1. **Messages / output:** Messages from the system
  2. **Script:** Journaling (logging) of user actions, commands, may also be typed or pasted
  3. **Result:** Detailed analysis results

The user interface is fully **dockable** meaning that the layout can be configured as preferred. The default GUI configuration can be restored.

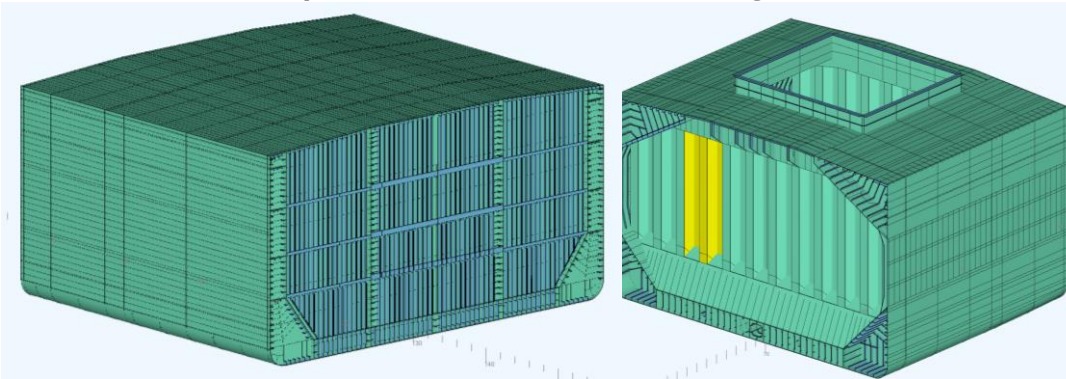


# Nauticus Hull Prescriptive

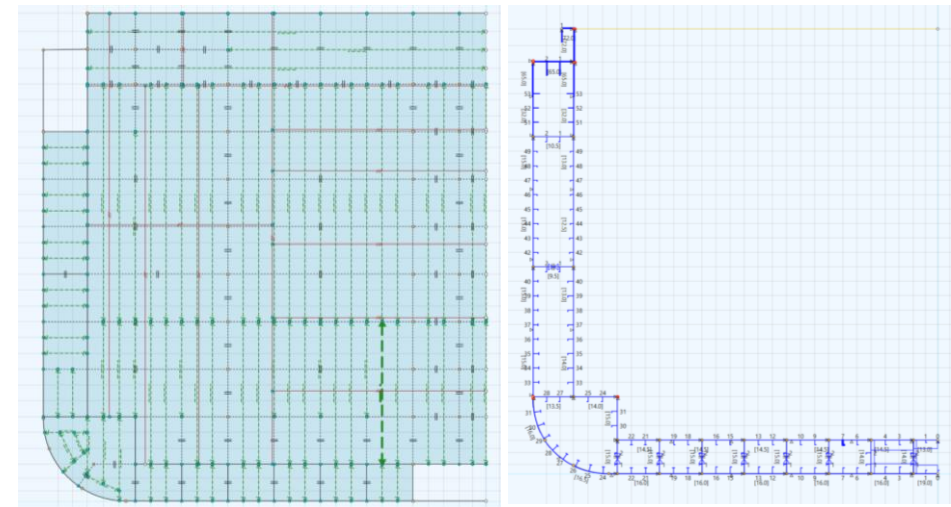
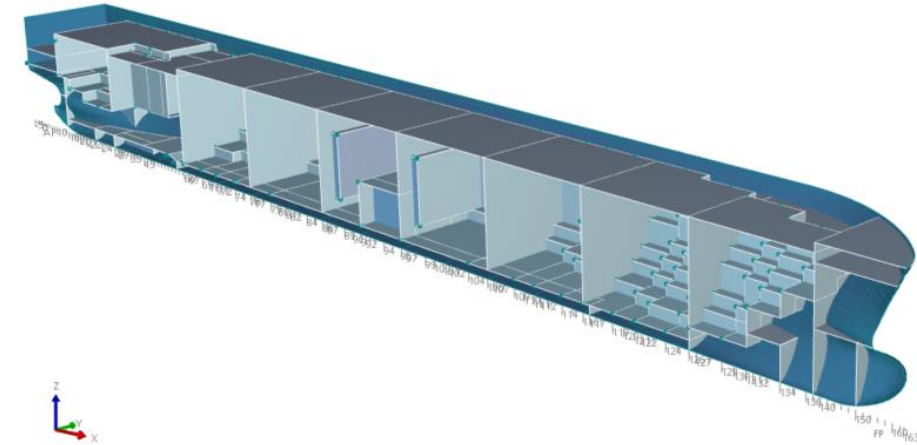
## Nauticus Hull's 3D geometry engine

- Moulded forms 3D geometry
- Geometry kernel fully developed and maintained by DNV
- Multibody data structure
- Conceptual modelling
- Same 3D geometry engine used for
  - Section Scantlings
  - Transverse Bulkheads; and
  - 3D model
- Import / export
  - Import of 3D models via OCX
  - Interface to Sesam GeniE for FEA

3D CAD models imported to NH via OCX exchange format



Same 3D geometry engine used for all NH modelling tools



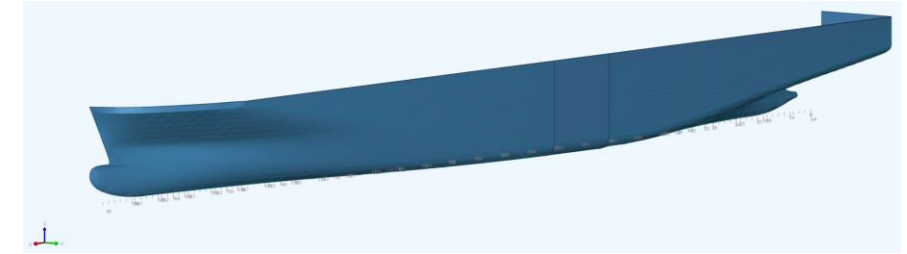
# Nauticus Hull Prescriptive

## Re-using designer's design data

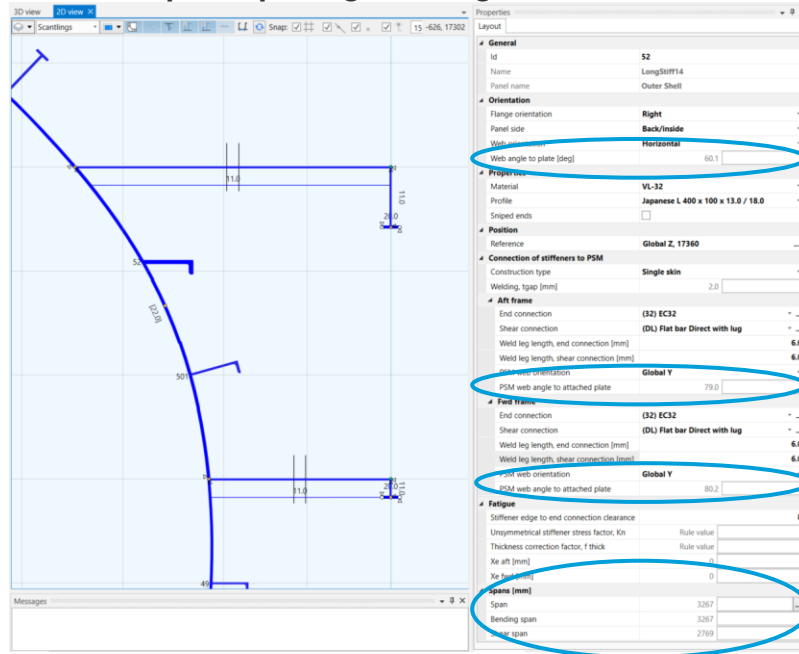
Easy import of hull from is essential for effective cross sectional analysis outside of midship region.  
Import of structural drawing, that can be displayed in the background, will considerably improve the modelling efficiently.

- Import IGES 3D hull form used in cross sectional analysis
  - Sea impact requirements: stiffener span, spacing, stiffener/PSM angle to shell
  - Ice (Baltic and Polar) regions and angles
- Import of structural drawing
  - Section Scantlings / Transverse Bulkheads: Background for modelling
  - Rule Calculator spot checks: Background for describing areas checked

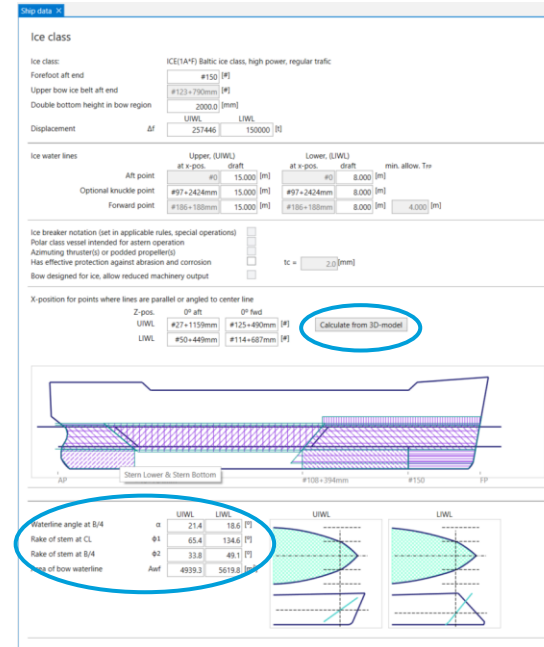
### 3D hull imported to Nauticus Hull



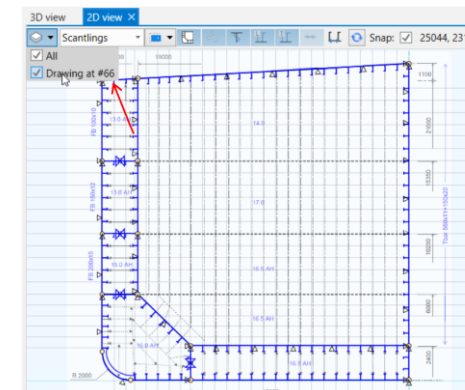
### Stiffener span, spacing and angles from 3D hull



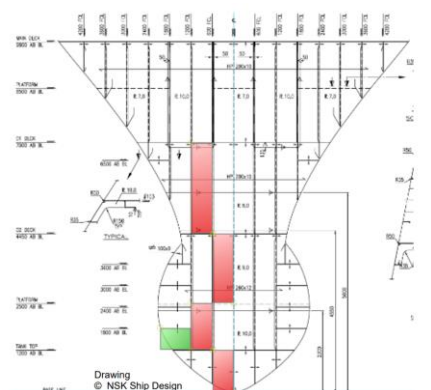
### Ice class regions/angles from 3D hull



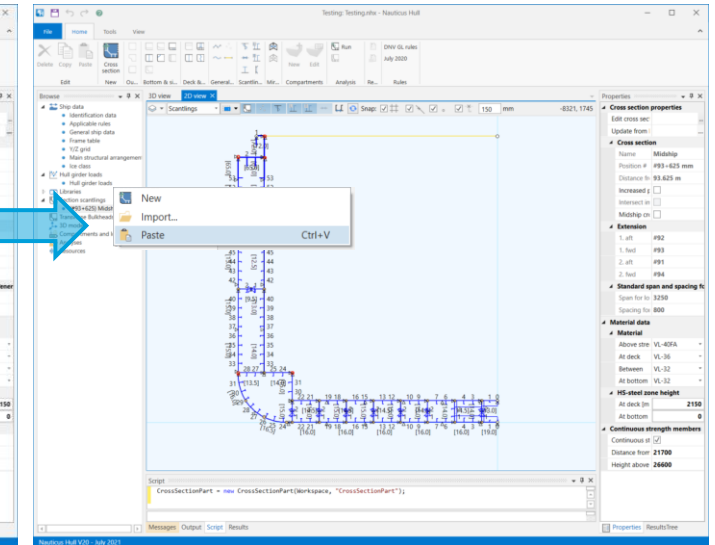
### Background drawing: modelling



### Background drawing: spot-check



- ## Import selected data from other workspace



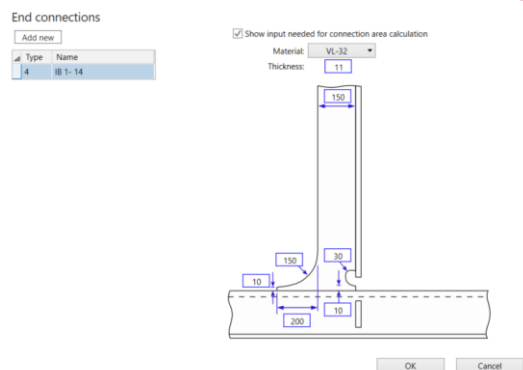
# Nauticus Hull Prescriptive

## Section Scantlings - Modelling

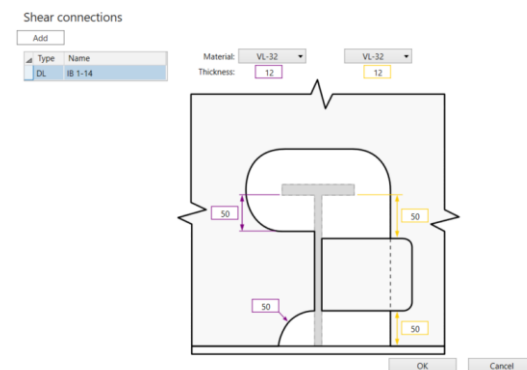
Section Scantlings has a modern and intuitive environment for efficient cross sectional modelling

- Model can be modelled from scratch, or intersect 3D model
- Wizards for effective creation of geometries, plates, stiffeners, cut-outs
- User can simply (multi)edit properties
- Conceptual representation of
  - End connection (for fatigue, span reductions, connection strength)
  - Shear connection (for connection strength)

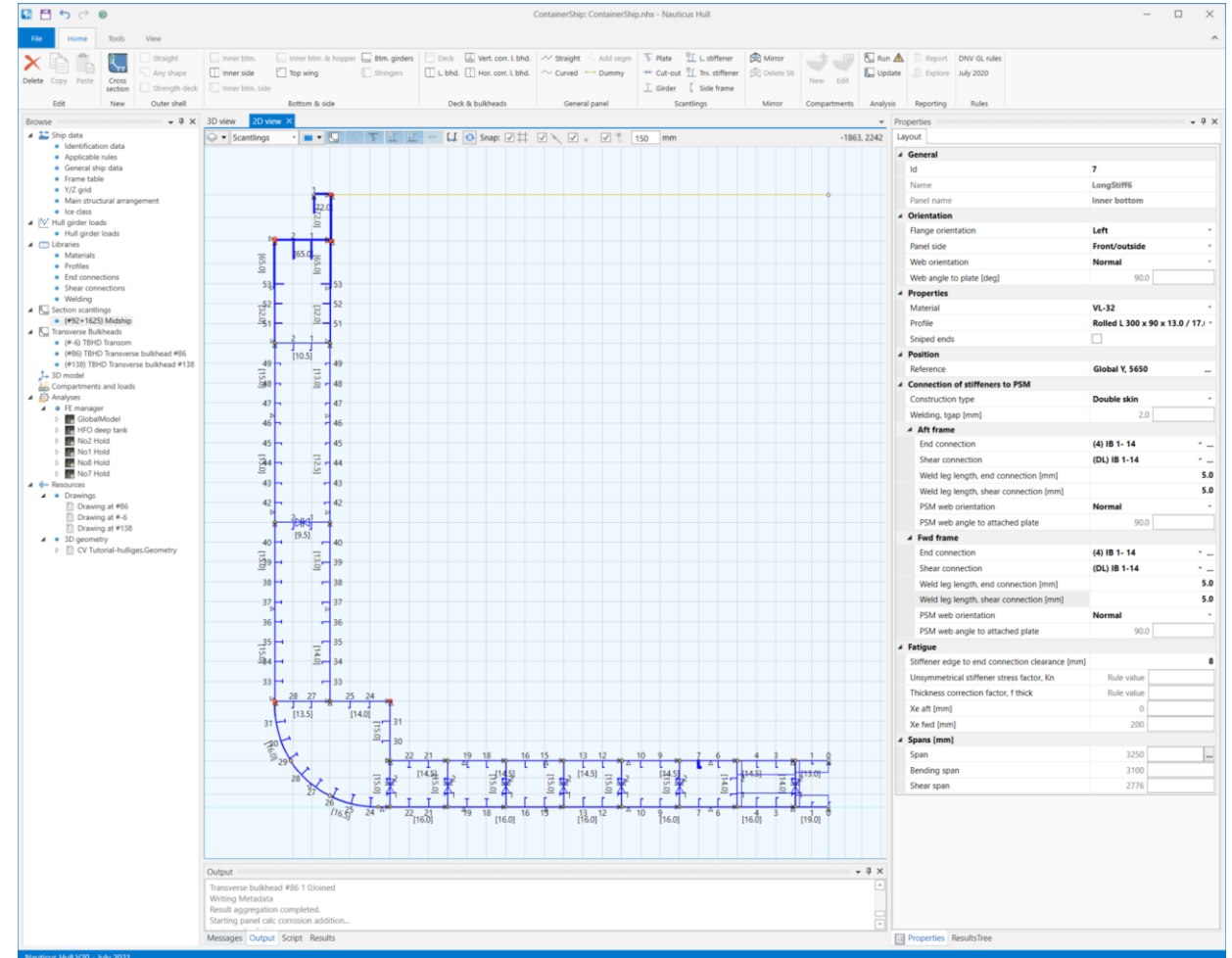
### End connection concept



### Shear connection concept



### Section Scantlings Graphical User Interface



# Nauticus Hull Prescriptive

## Section Scantlings - Analysis

Various analysis results are easily available, giving the user a good understanding in governing Rule requirements, and how to optimise the design

- Prescriptive Rule checks for (nodes in the results tree)
  - Hull girder strength
  - Plates, stiffeners, transverse stiffeners
  - Connections of stiffeners to PSM
- All Rule revisions for DNV and CSR BC&OT
- Imported hull form in analysis for
  - angles (bow impact, ice)
  - stiffener span (cord length)
- Design iterations plates and stiffeners
  - Increase scantlings to Rule requirement
  - Reduce scantlings to Rule requirement
- Report available
- Colour coding of analysis results
- Detailed plate requirements on EPP level

### Updated scantlings

Update Scantlings

Update modelled plates and stiffener scantlings

Plates

☒ Increase plate thickness to governing local thickness requirement
 ☒ Reduce plate thickness to governing local thickness requirement

Stiffeners

☒ Increase stiffener scantlings to governing local requirement
 ☒ Reduce stiffener scantlings to governing local requirement

Transverse Stiffeners

☒ Increase stiffener scantlings to governing local requirement
 ☒ Reduce stiffener scantlings to governing local requirement

Update

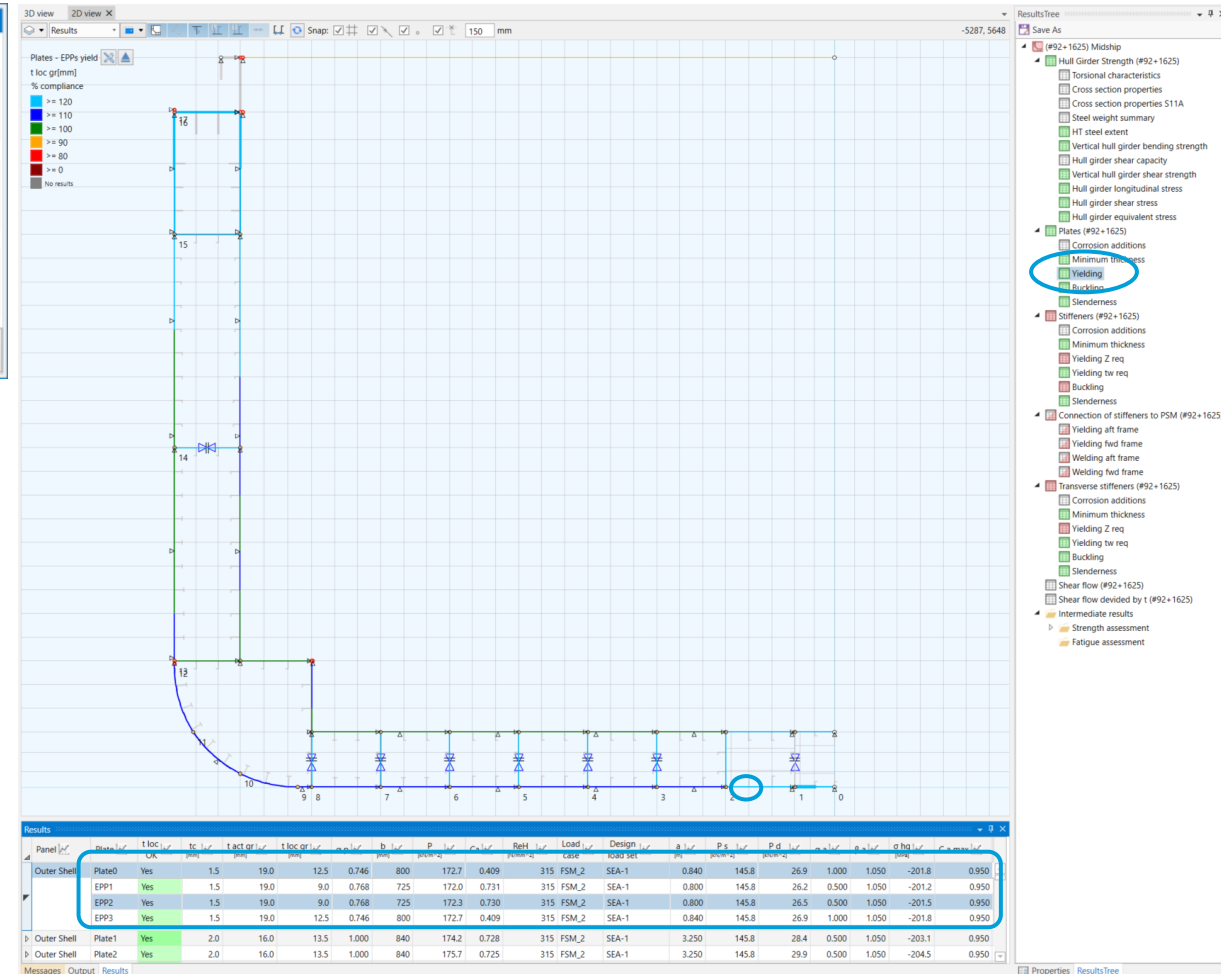
### Report



## NAUTICUS™ HULL

Cross Section Analysis according to DNV GL rules

### Analysis results: colour coding and detailed results on EPP level



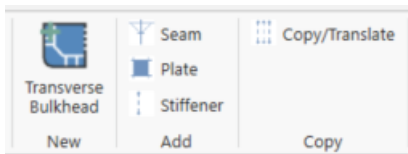
# Nauticus Hull Prescriptive

## Transverse bulkheads - Modelling

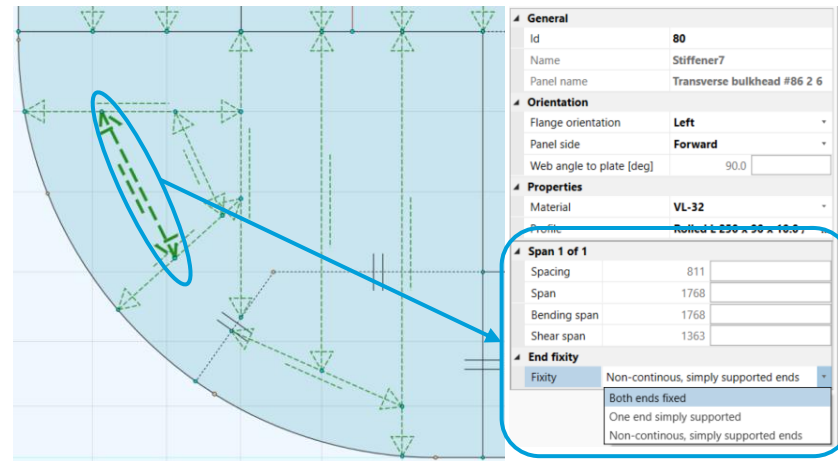
The modelling approach in the TBHD tool is based on the success of Section Scantlings, being recognised as one of the most efficient tools for prescriptive Rule checks

- Model can be modelled from scratch, or intersect 3D model
- Similar modelling approach as for Section Scantlings
- Wizards for adding sems, plates and stiffeners in any sequence
- Copy / translate of stiffeners available
- User can override stiffener spacing/span and end fixity
- Stiffeners are divided into spans between span killers (stringers/decks)

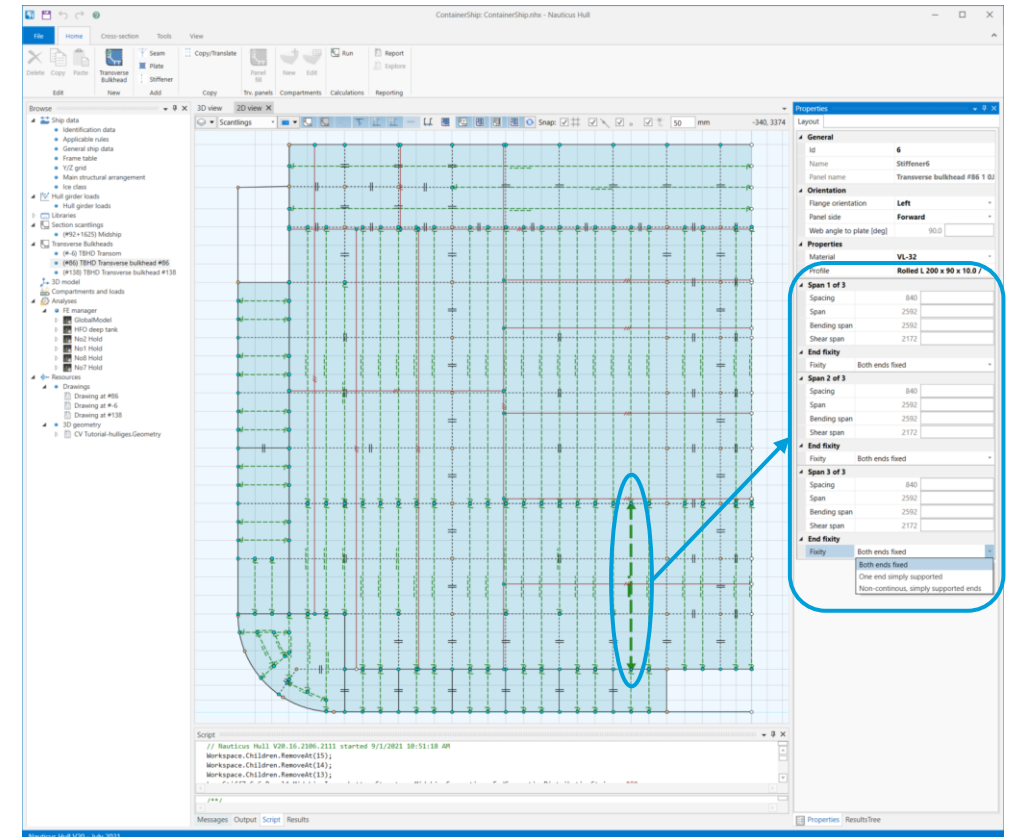
### Modelling wizards



### Stiffener properties for analysis can be edited



### Stiffener divided in to spans between stringers/decks

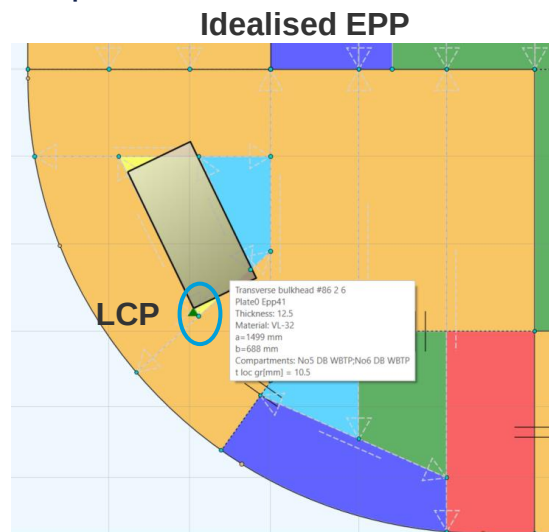


# Nauticus Hull Prescriptive

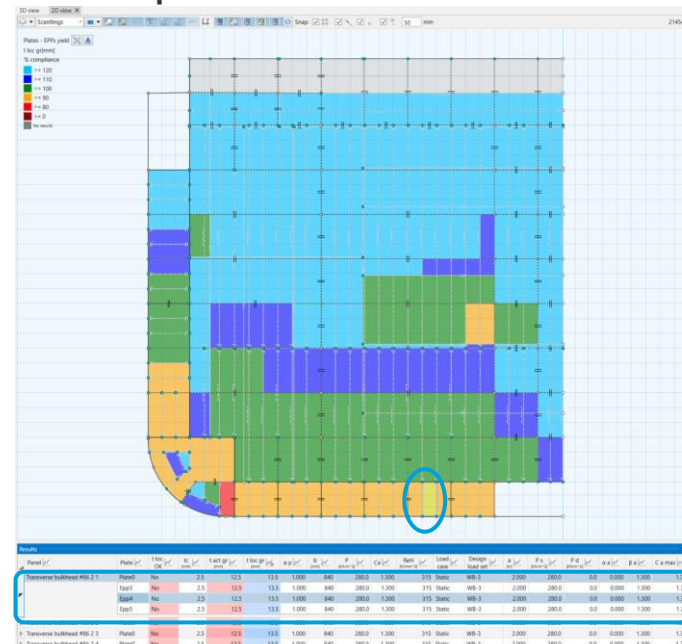
## Transverse bulkheads - Analysis

Similar to the relation between plates and EPPs, detailed results are shown for a complete stiffener span and stiffener spans between span killers

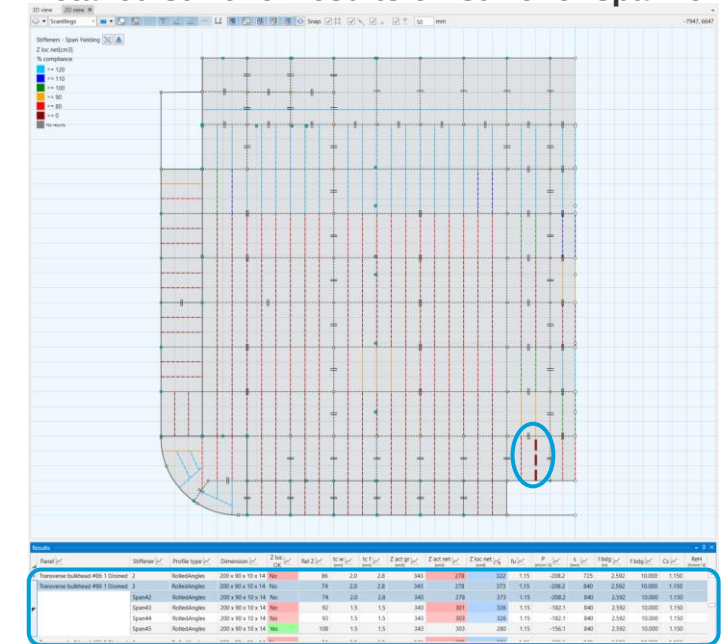
- Prescriptive Rule checks for plates and stiffeners
- Implemented for DNV Rules and CSR BC&OT
- Similar analysis approach as for Section Scantlings
- Report available
- Display of idealised EPPs with load calculation point (LCP)
- Detailed results
  - Plates on EPP level
  - Stiffener on stiffener span level



**Detailed plate results on EPP level**



**Detailed stiffener results on stiffener span level**

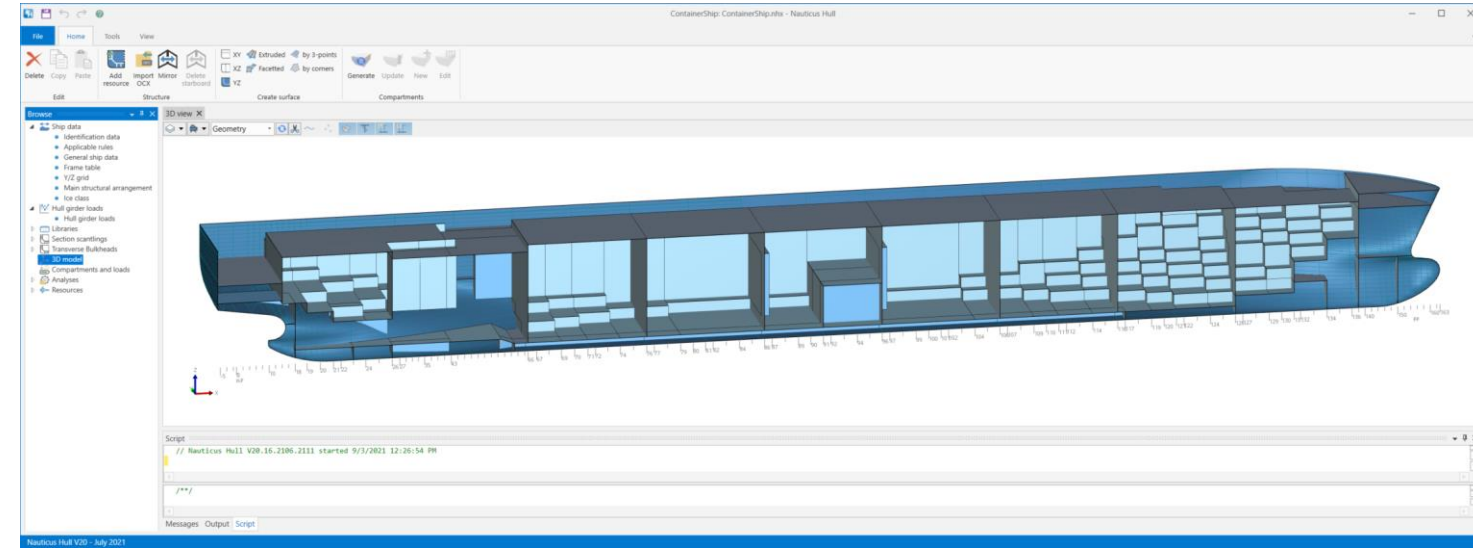


# Nauticus Hull Prescriptive

## Prescriptive 3D geometry and compartment model - Modelling

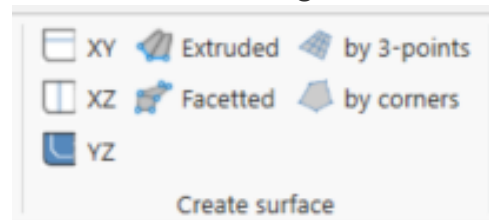
DNV has developed a 3D geometry kernel, optimised for ships, that can re-use yard's CAD models via the OCX exchange format

### 3D model in Nauticus Hull of a container ship modelled from scratch

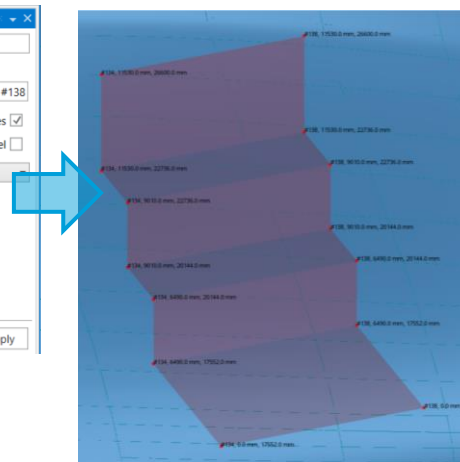
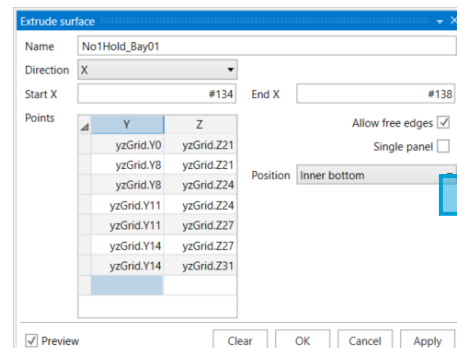


- 3D model in Nauticus Hull originating from
  - OCX import; or
  - Modelling from scratch (geometry surfaces only)
- Modelling of geometries by use of
  - Wizards; or
  - Scripting
- Intersect 3D model for Section Scantlings / Transverse Bulkheads modelling

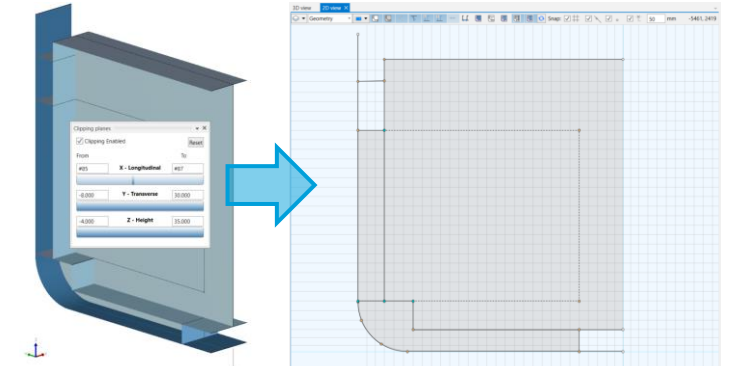
### Available modelling commands



### Extrude surface command with preview



### Intersect 3D model for Transverse Bulkheads tool



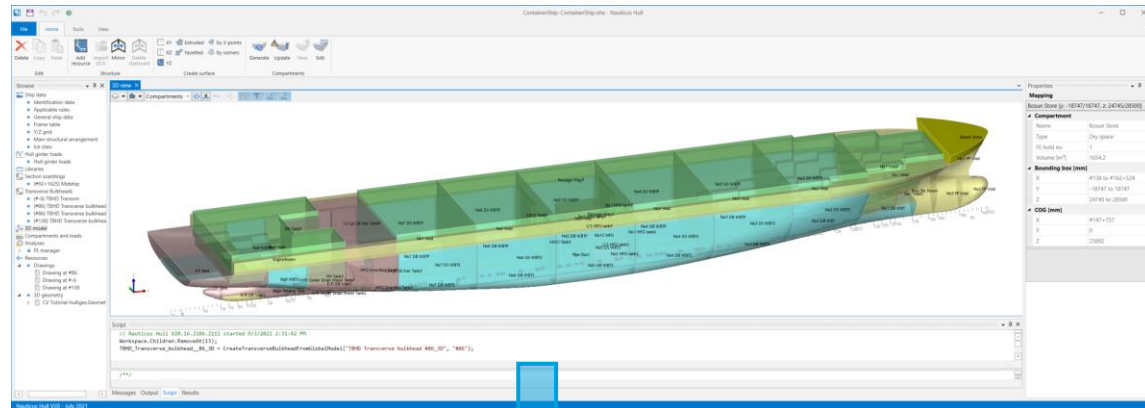
# Nauticus Hull Prescriptive

3D model compartments automatically give correct compartment pressures, as compartment attributes are based on actual compartment geometry

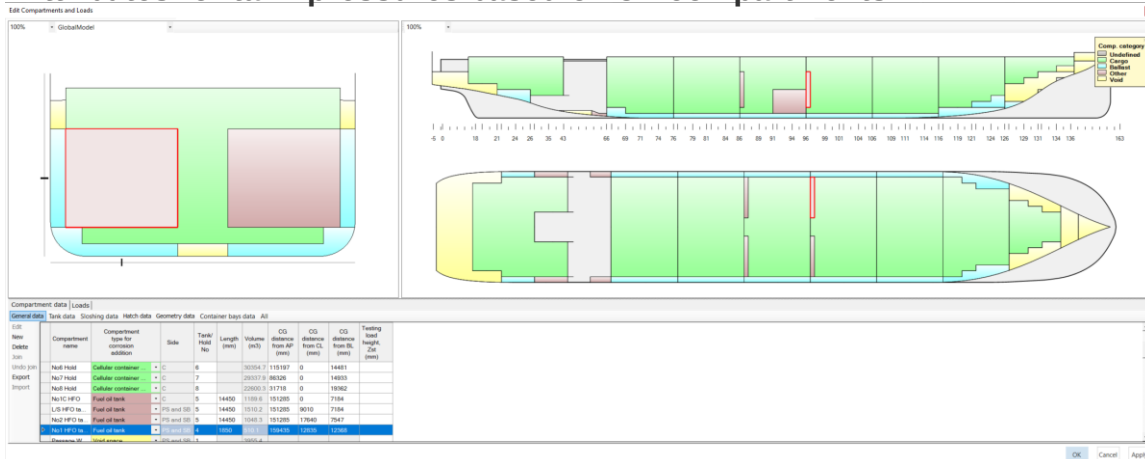
## Prescriptive 3D geometry and compartment model - Compartments

- Compartments will be generated based on modelled WT geometries
- True 3D compartment geometry for automatic determination of attributes for correct pressures, e.g. centre of gravity and reference points

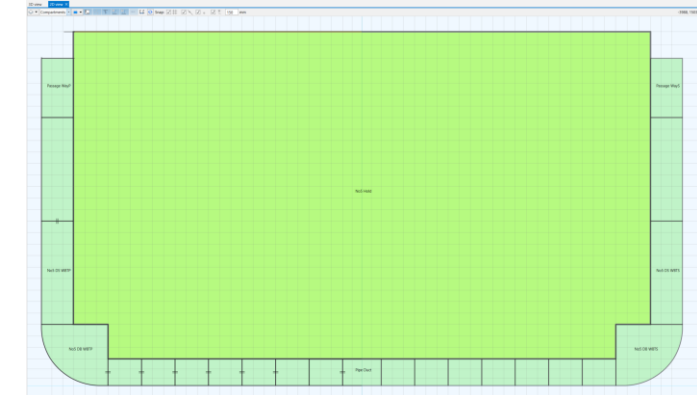
### Compartments generated from WT geometry



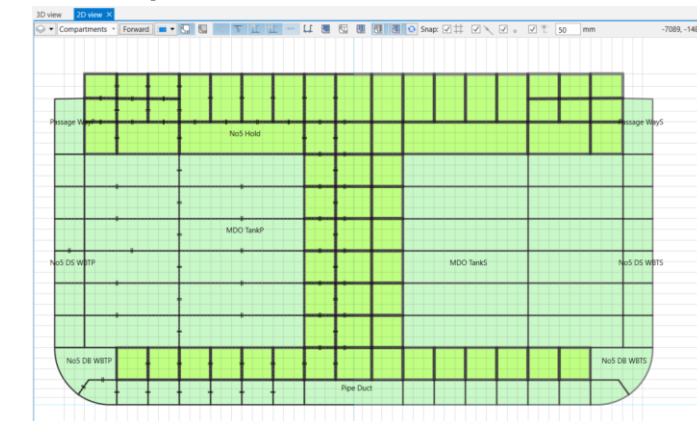
### Attributes for tank pressures based on 3D compartments



### 3D compartments in Section Scantlings



### 3D compartments in Transverse Bulkheads



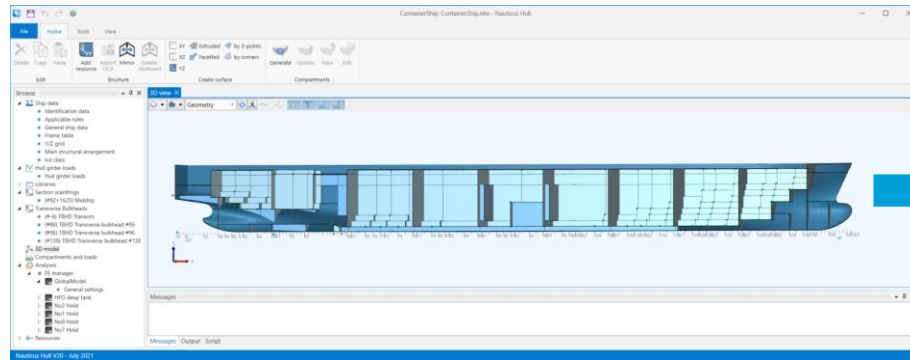
# Nauticus Hull Prescriptive

## Export to GeniE - Global model

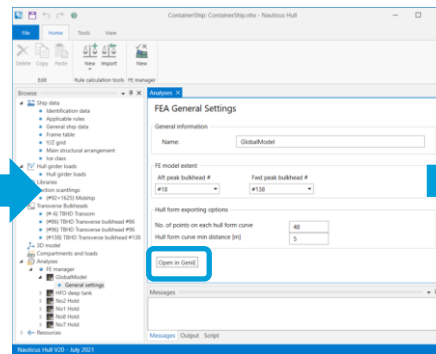
*The 3D model in GeniE has the same surface representation (same number and extent) as in Nauticus Hull, being a solid foundation for a close integration between the two tools*

- Prescriptive global 3D model can be exported to GeniE by a single click
- The export includes all surface geometries, including complex bow shape and skeg

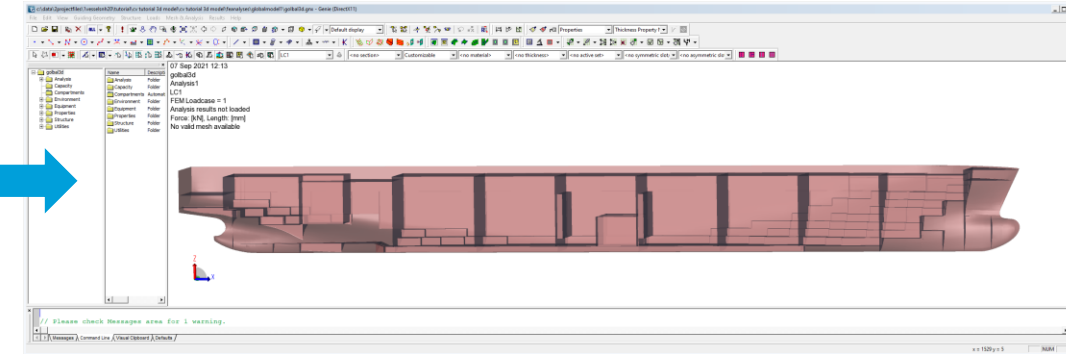
3D geometry model in Nauticus Hull



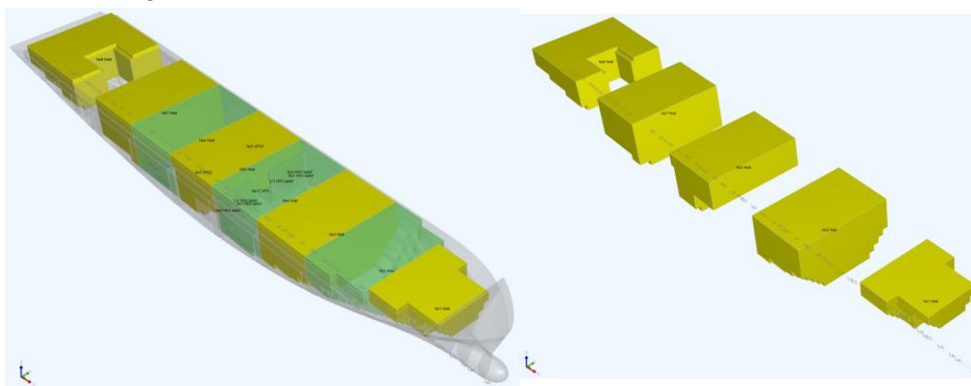
Automatic export from Nauticus Hull to GeniE



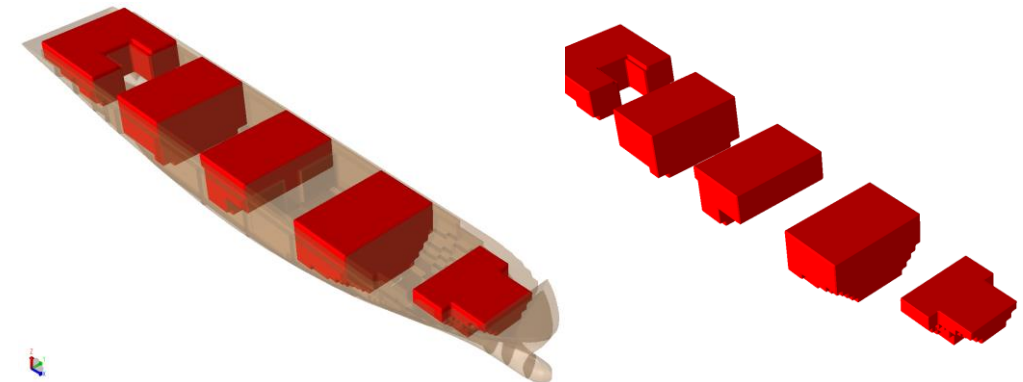
3D geometry model in GeniE



3D compartments in Nauticus Hull



3D compartments in GeniE

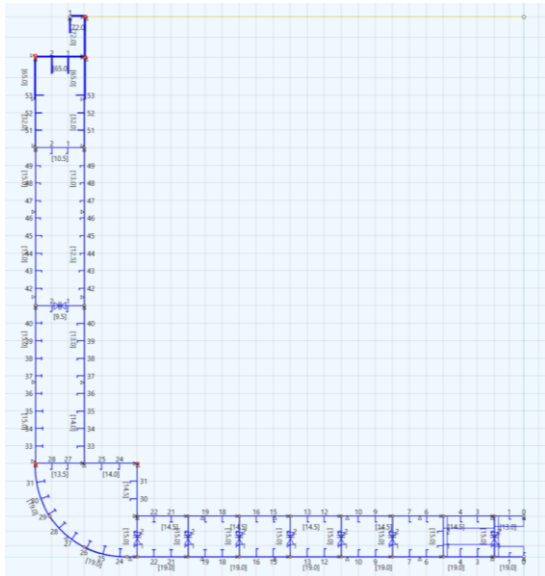


# Nauticus Hull Prescriptive

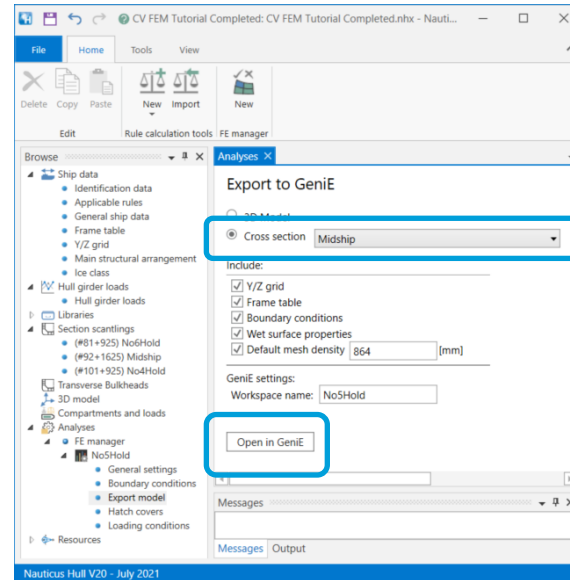
## Export to GeniE – Midship cargo hold model – Extrude Cross Section

- A selected Cross Section can be used for creation of longitudinal members in the FE model by extracting the cross section in GeniE
- Export includes surface geometries, plates, stiffeners, boundary conditions (including end beams) and wet surfaces for application of sea pressures
- The process is fully atomised: GeniE can be opened by clicking on a button, and then the cross section will be extruded

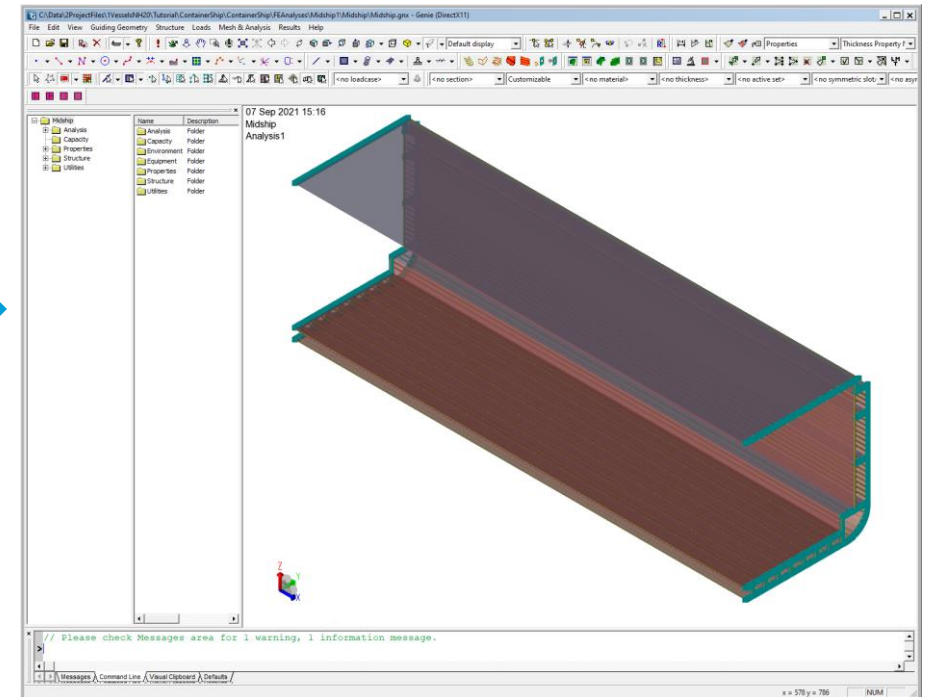
Typical midship cross section in Section Scantlings



Automatic export from Nauticus Hull to GeniE



Cross section extruded in GeniE with same scantlings as in the Cross Section



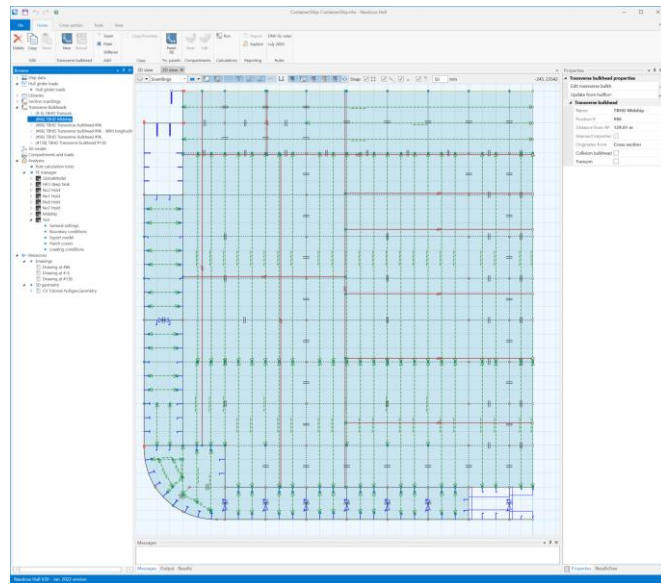
# Nauticus Hull Prescriptive

## Export to GeniE – Midship cargo hold model – Transverse Bulkhead

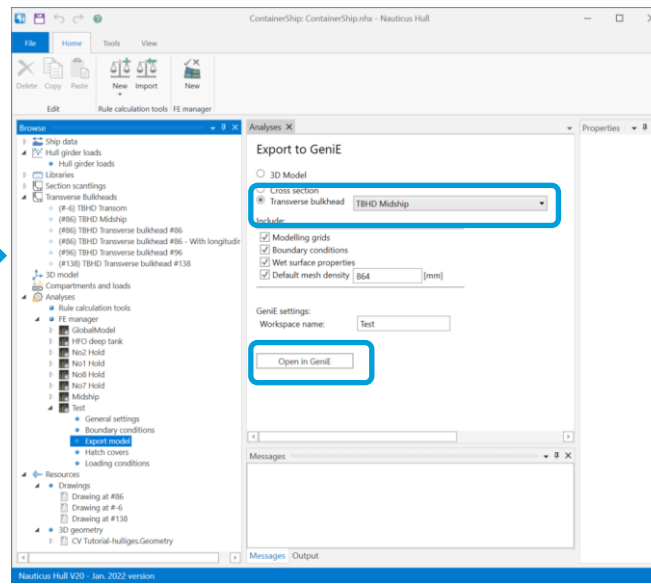
*For prismatic cargo hold models, exporting a typical transverse bulkhead modelled in Nauticus Hull to GeniE, will significantly reduce the FE modelling time*

- A selected Transverse Bulkhead can be used for creation of transverse bulkheads at specified positions in GeniE
- Longitudinal members (plates and stiffeners) will also be extruded
- Export includes surface geometries, plates, stiffeners, boundary conditions (including end beams) and wet surfaces for application of sea pressures
- The process is fully atomised: GeniE can be opened by clicking on a button

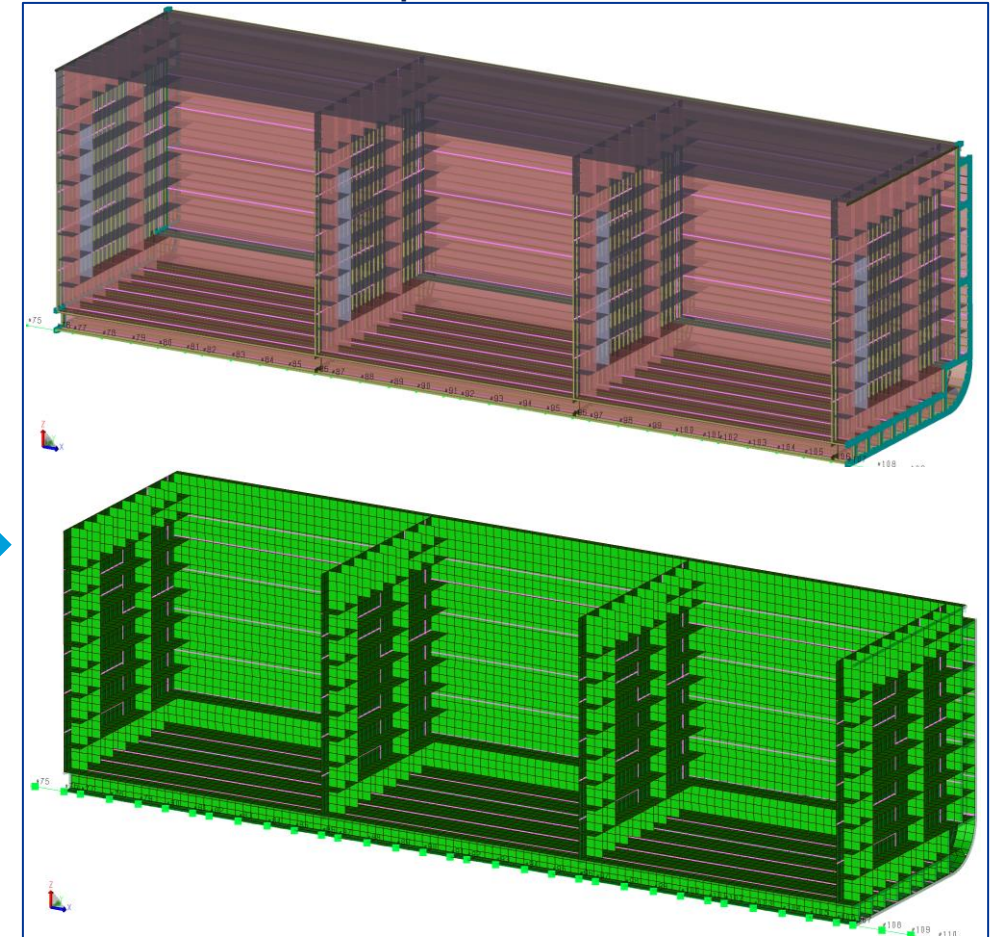
Transverse bulkhead modelled in NH



Export a selected transverse bulkhead



Model exported to GeniE



## FE load templates for automatic load generation

- ## Container stowage on hatches



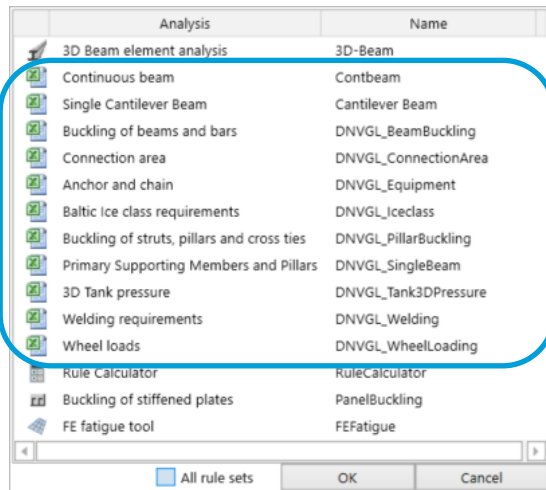
# Nauticus Hull Prescriptive

## Rule Calculation tools

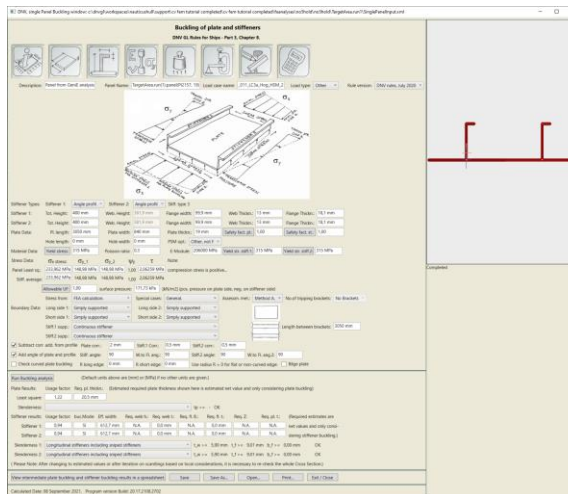
Various tools are available for spot checks and Rule checks not covered by Section Scantlings / Transverse Bulkheads

- Excel spreadsheets for e.g. anchor and chain, primary supporting members, welding
- Single Panel Buckling tool for buckling assessments (can be accessed from Section Scantlings and GeniE)
- FE Fatigue tool for FE fatigue strength calculations (integrated with GeniE)
- Rule Calculator for spot checks of plates and stiffeners (can also be combined with drawing check)

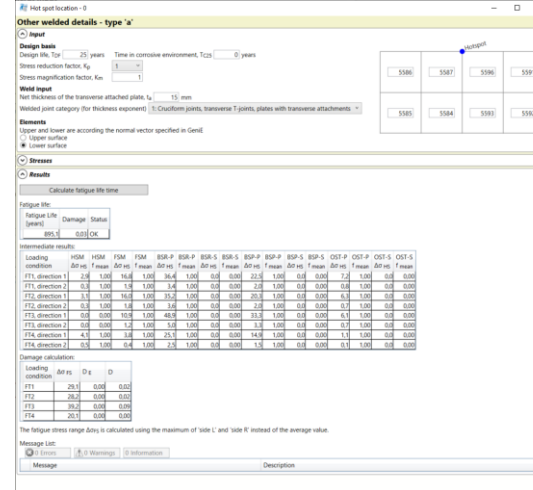
### Excel spreadsheets



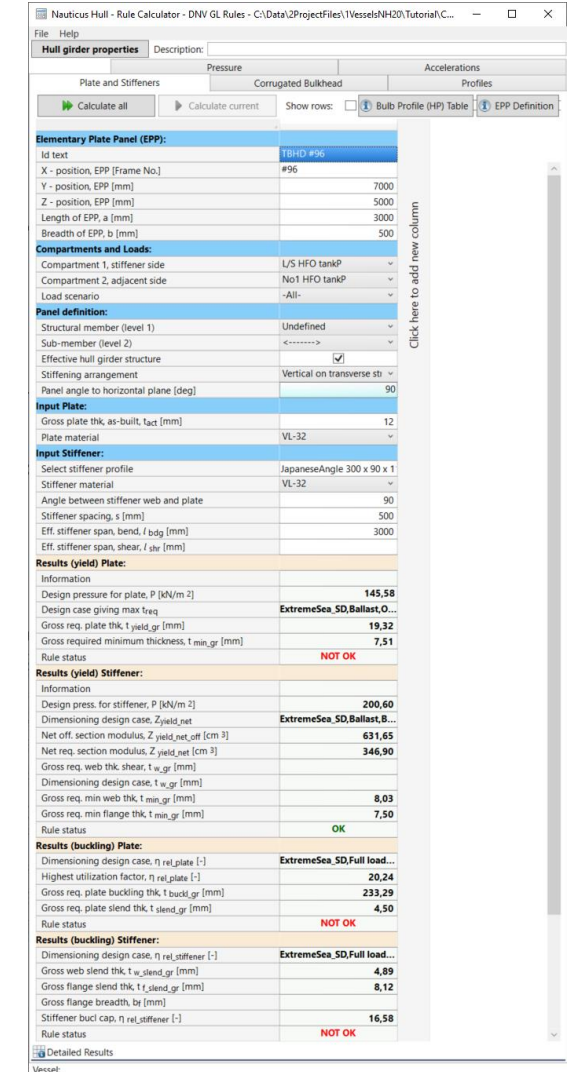
### Single Panel Buckling tool



### FE Fatigue tool



### Rule Calculator



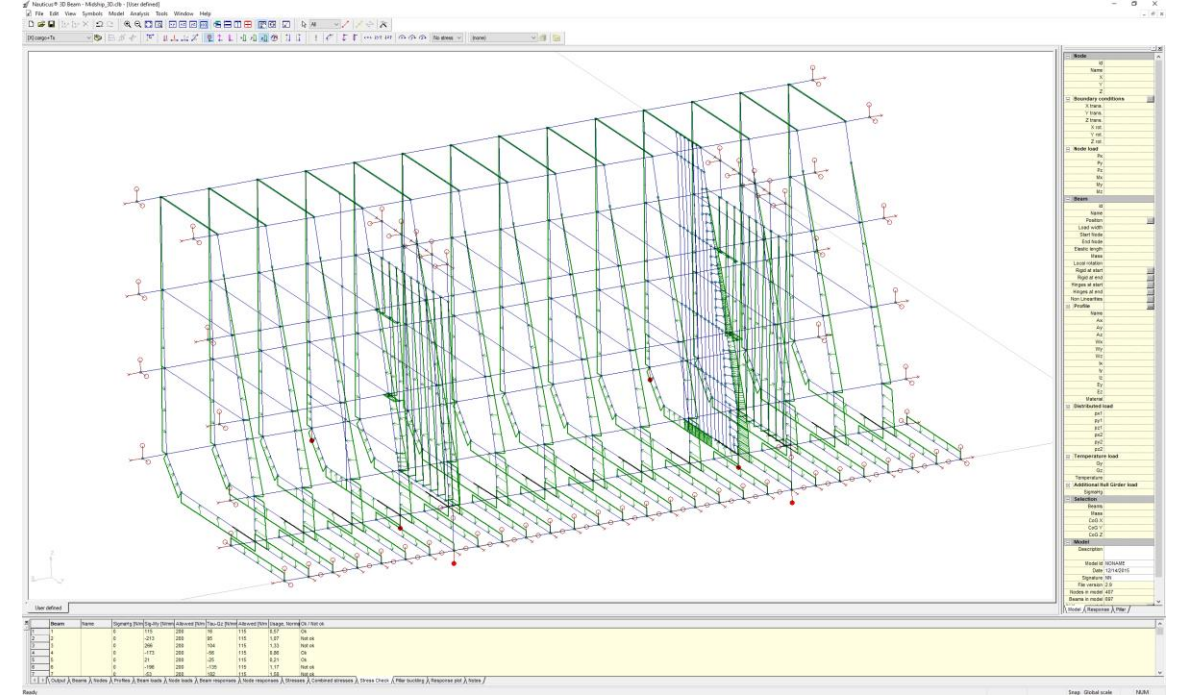
# Nauticus Hull Prescriptive

## 3D Beam

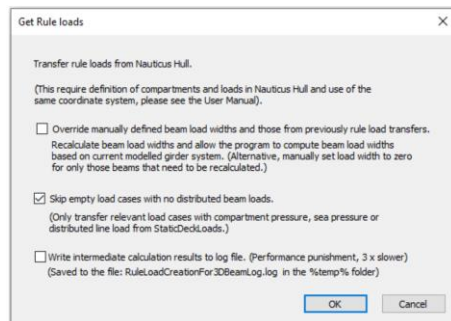
3D Beam is an efficient tool for analysing complex girder systems and deck foundations

- General beam element analysis tool for both simple 2D and complex 3D beam element analysis
- Intuitive and user-friendly interface, tailor-made for accurate, fast modelling and strength assessment of complex 3D girder structures
- Rule loading conditions (pressures and hull girder bending stress) can be automatically transferred from Nauticus Hull
- Model can be exported to GeniE for beam code checking and fatigue
- PSM Rule checks implemented
  - Yield check
  - Pillar buckling
- Stress results along length of beam can be assessed for detailed results and considering brackets

### 3D Beam's modern and user-friendly GUI



### Automatic Rule loading



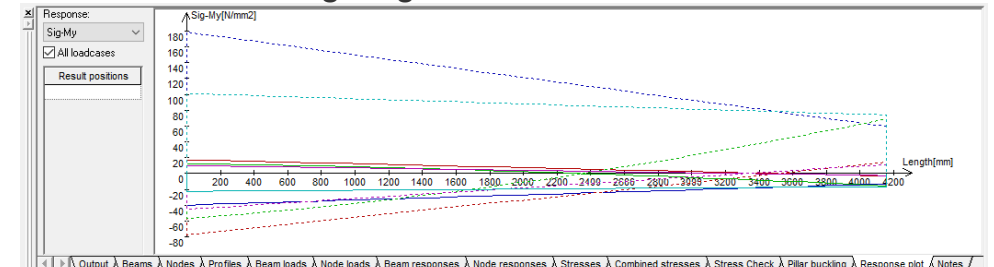
### PSM Rule checks for yield and pillar buckling

| Beam | Name | SigmaMg [N/mm] | Sig-Mly [N/mm] | Allowed [N/mm] | Tau-Qz [N/mm] | Allowed [N/mm] | Usage, Norm | Ok / Not ok |
|------|------|----------------|----------------|----------------|---------------|----------------|-------------|-------------|
| 1    | 1    | 0              | 115            | 200            | 16            | 115            | 0.57        | Ok          |
| 2    | 2    | 0              | 213            | 200            | 95            | 115            | 1.07        | Not ok      |
| 3    | 3    | 0              | 266            | 200            | 104           | 115            | 1.33        | Not ok      |
| 4    | 4    | 0              | -173           | 200            | -56           | 115            | 0.86        | Ok          |
| 5    | 5    | 0              | 21             | 200            | -25           | 115            | 0.21        | Ok          |
| 6    | 6    | 0              | -196           | 200            | -135          | 115            | 1.17        | Not ok      |
| 7    | 7    | 0              | -53            | 200            | 182           | 115            | 1.58        | Not ok      |

| Beam | Name | Usage, Axial | Usage, Comb | Usage, Allow | Usage, Norm | x-Pos [mm] | SigCr, Axial | SigCr, Bend | Sig-Nx [N/mm] | Sig-Mly [N/mm] | Sig-Mz [N/mm] | Curve   | Curve   | Ok / Not ok |
|------|------|--------------|-------------|--------------|-------------|------------|--------------|-------------|---------------|----------------|---------------|---------|---------|-------------|
| 1    | 1    | 0.08         | 0.86        | 0.75         | 1.14        | 1410       | 201          | 235         | 18            | -25            | -158          | curve a | curve b | Not ok      |
| 2    | 2    | 0.37         | 1.81        | 0.75         | 2.42        | 316.67     | 235          | 235         | -86           | -181           | -154          | curve a | curve b | Not ok      |
| 3    | 3    | 0.76         | 1.77        | 0.75         | 2.36        | 346.67     | 234          | 235         | -177          | -229           | -1            | curve a | curve b | Not ok      |
| 4    | 4    | 0.16         | 0.77        | 0.75         | 1.03        | 6225       | 36           | 235         | -8            | -132           | -7            | curve a | curve b | Not ok      |
| 5    | 5    | 0.12         | 0.17        | 0.75         | 0.23        | 206.67     | 228          | 235         | 29            | -4             | -6            | curve a | curve b | Ok          |
| 6    | 6    | 0.33         | 0.74        | 0.75         | 0.99        | 6916.7     | 31           | 235         | -10           | -59            | -8            | curve a | curve b | Ok          |
| 7    | 7    | 0.25         | 0.69        | 0.75         | 0.91        | 286.67     | 228          | 235         | 58            | -21            | -82           | curve a | curve b | Ok          |

### Stress results along length of beam



# Nauticus Hull Finite Element

Sesam GeniE as Nauticus Hull's finite element tool

FE support easily available

GeniE – A modern and future proof FE tool

Only workspace file needed for storage and exchange of FE model

Configuration of GeniE – Modes

Ship mode configuration of GeniE

All Rule sets covered

Efficient tool for Container Ship FE models required by the Rules

Efficient generic FE tool

Bulk data file \*.bdf can be imported to GeniE

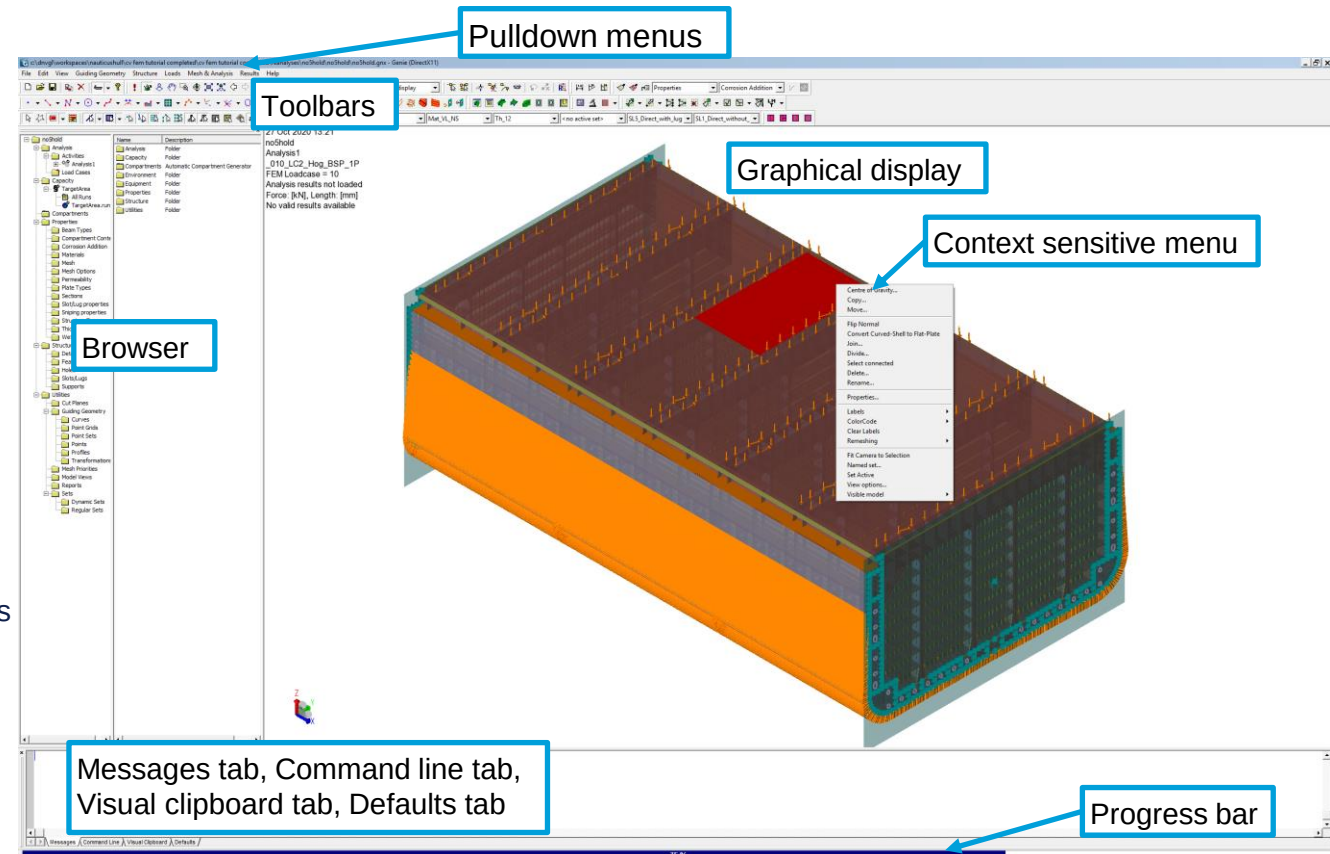
# Nauticus Hull Finite Element

## Sesam GeniE as Nauticus Hull's finite element tool

The GUI in GeniE consists of

1. **Pulldown menus** containing commands for file operations, modelling, analysis, etc.
2. **Toolbars** containing most frequently used commands that are available from the pulldown menus
3. **Browser** containing model data available for editing
4. **Graphical display**
5. **Context sensitive menus**
6. **Tabs** for
  1. **Messages:** Information (green), warning (orange) and error (red) messages issued by the program
  2. **Command:** Journaling (logging) of user actions, commands may also be typed or pasted
  3. **Visual clipboard:** User can retrieve information from the model in input fields
  4. **Defaults:** User may set default selection options and default values used during modelling
7. **Progress bar** at the very bottom of the GeniE, a window providing progress information

The user interface is fully **dockable** meaning that the layout can be configured as preferred. The default GUI configuration can be restored.



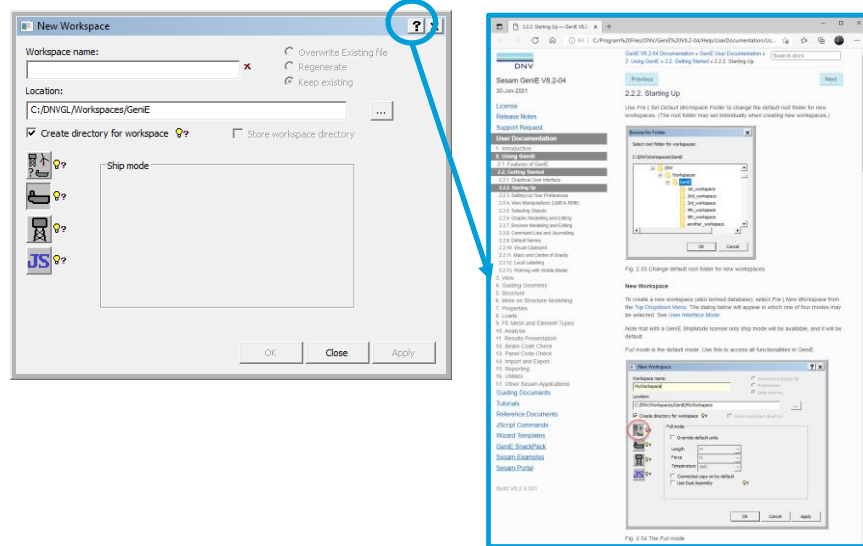
# Nauticus Hull Finite Element

## FE support easily available

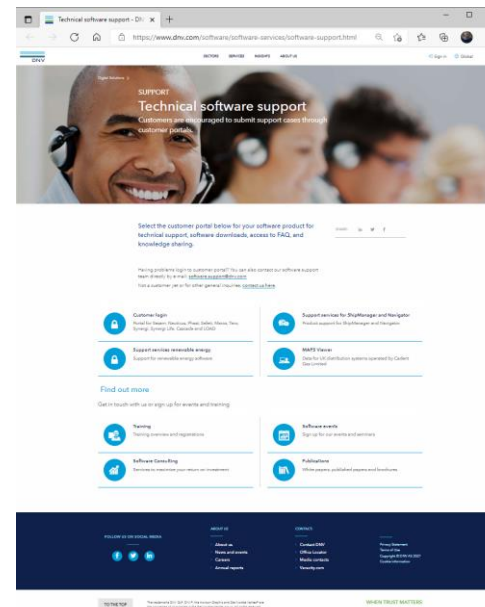
DNV has a world wide support organisation, with quick response, minimising the time loss when you need support

- Dialogues having direct link to User Documentation
- Supporting engineers in DNV available 24/5
- Tutorials on a customer portal
- Ship specific tutorials for FE models required by the Rules

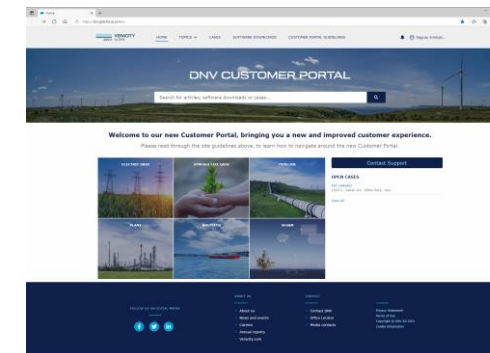
### Direct access to User Documentation



### Support 24Seven



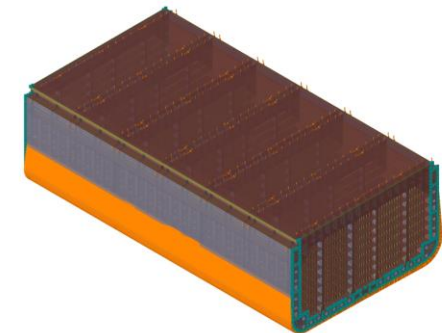
### Tutorials on the web



### Ship specific tutorials

#### Sesam GeniE

**Tutorial – cargo hold finite element analysis**  
Container ship midship model with typical hold arrangement



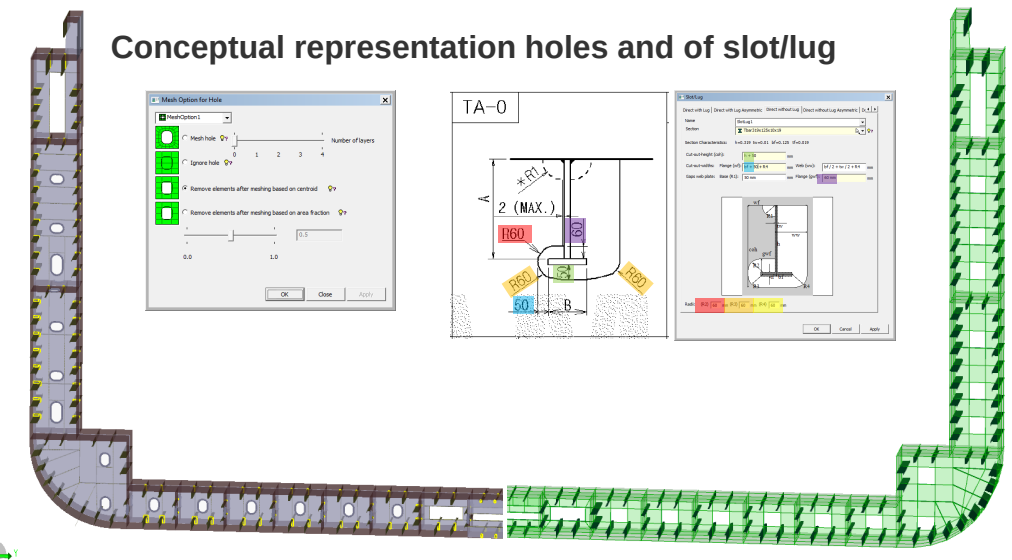
# Nauticus Hull Finite Element

## GeniE – A modern and future proof FE tool

- GeniE – A modern and future proof tool
  - Using modern architecture
  - Real 3D modeller
  - Conceptual modelling, allowing for rapid modelling and change to a model
  - Mesh is automatically updated when updating the loads/geometry
  - Undo / Redo – not limited to only one command
  - Scripting
  - Supports backward compatibility
  - Multithreading of analysis tasks for improved performance
- Benefits by conceptual modelling
  - More efficient modelling, less time needed
  - More realistic representation of the geometry
  - Design iterations more efficient by working on the geometry and not the finite element representation
  - Concept information also used in Rule checks
  - Loads are applied to the concepts and automatically mapped to the finite elements

What is **concept modelling**? Sesam GeniE is the only software on the market that offers concept modelling, which is an immense time saver for engineers.

**Concept modelling in Sesam GeniE maintains the dynamic connectivity between structural components such as beams and plates.** This means that whenever the user inserts or moves a structural component, the connectivity to other plates or beams is automatically updated. Design optimisation is often characterised by many and large changes – concept modelling is therefore the most time-efficient method of design



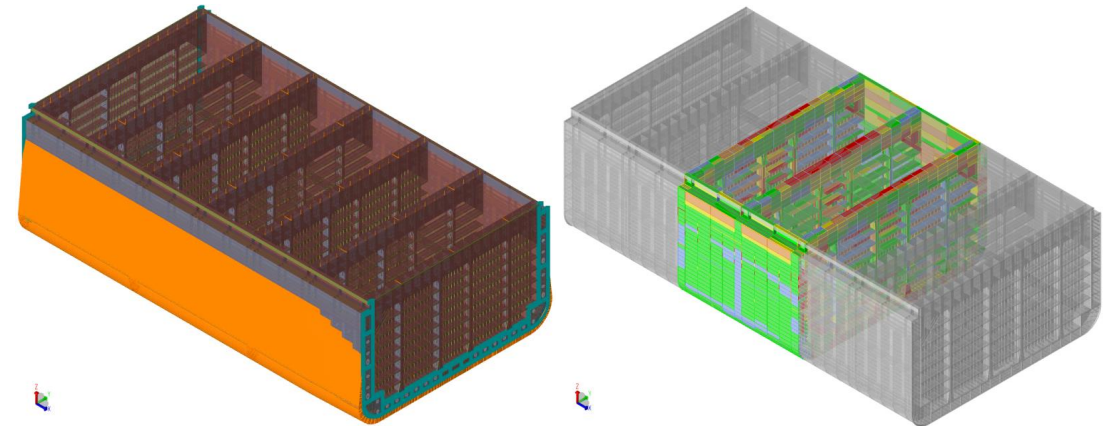
# Nauticus Hull Finite Element

Only workspace file needed for storage and exchange of FE model

- Sesam GeniE's database file format is \*.GNX
- The \*.GNX file includes:
  - Concept model, including loads
  - Mesh information
  - Analysis set-up
  - FEA Rule check set-up (including buckling panels)
  - Report template
  - JS file with scripted commands
- The \*.GNX database file format has the following benefits
  - Backwards compatible
  - Only Workspace.GNX file is needed when exchanging/filing model
  - Easy to share with others, just open GNX and run analysis to get results
  - Workspace.GNX does not include any results and has therefore limited size
  - Workspace.GNX will always create identical analysis results when re-running analysis (provided same SW versions applied)

File size for a CV cargo hold model, including  
FEA Rule check set-up

| Name                                                                                            | Type     | Size      |
|-------------------------------------------------------------------------------------------------|----------|-----------|
|  No5Hold.gnx | GNX File | 39 853 KB |

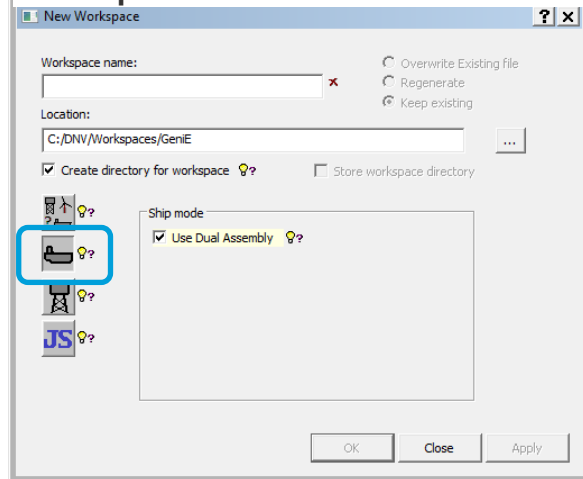


# Nauticus Hull Finite Element

## Configuration of GeniE – Modes

- GeniE is a general FE tool optimised for various applications by the introduction of the following modes
  - Full mode
  - Ship mode
  - Jacket mode
- The different models will ensure a customisation of GeniE
  - Filtering of menus, toolbars, dialogs and browser
  - Default properties and units according to industry standard
- When creating a new workspace, with a GeniE Ship Mode licence (provided by the NH Finite Element package), only the Ship Mode will be available

**When creating a new workspace, only the ship mode will be available**



# Nauticus Hull Finite Element

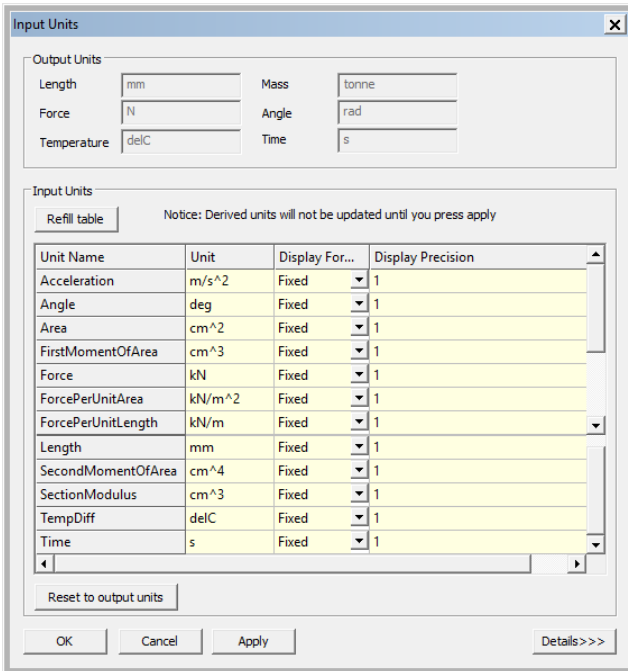
## Ship mode configuration of GeniE

Ship mode configuration improves the user-friendliness by filtering out features that are not applicable for the shipbuilding industry

Ship mode ensures default units as described in the Rules and according to shipbuilding industry standards

Ship mode filter out browser entries that are not applicable for ships

Ship mode filter out selection options in dialogues that are not applicable for ships



Input Units dialog box showing output units and input units configuration.

Output Units:

| Output Units | Unit | Mass  | Angle | Time |
|--------------|------|-------|-------|------|
| Length       | mm   | tonne | rad   | s    |
| Force        | N    |       |       |      |
| Temperature  | delC |       |       |      |

Input Units:

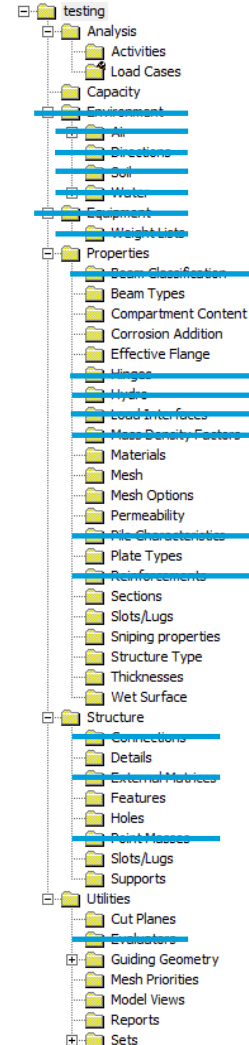
Refill table Notice: Derived units will not be updated until you press apply

| Unit Name          | Unit   | Display For... | Display Precision |
|--------------------|--------|----------------|-------------------|
| Acceleration       | m/s^2  | Fixed          | 1                 |
| Angle              | deg    | Fixed          | 1                 |
| Area               | cm^2   | Fixed          | 1                 |
| FirstMomentOfArea  | cm^3   | Fixed          | 1                 |
| Force              | kN     | Fixed          | 1                 |
| ForcePerUnitArea   | kN/m^2 | Fixed          | 1                 |
| ForcePerUnitLength | kN/m   | Fixed          | 1                 |
| Length             | mm     | Fixed          | 1                 |
| SecondMomentOfArea | cm^4   | Fixed          | 1                 |
| SectionModulus     | cm^3   | Fixed          | 1                 |
| TempDiff           | delC   | Fixed          | 1                 |
| Time               | s      | Fixed          | 1                 |

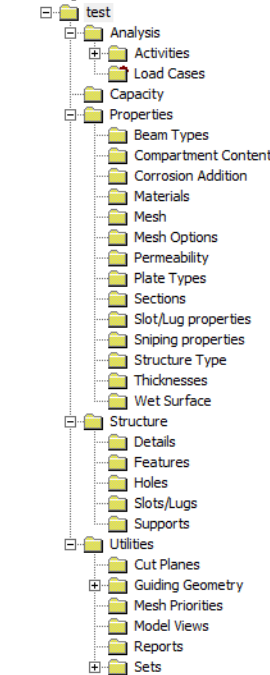
Reset to output units

OK Cancel Apply Details>>>

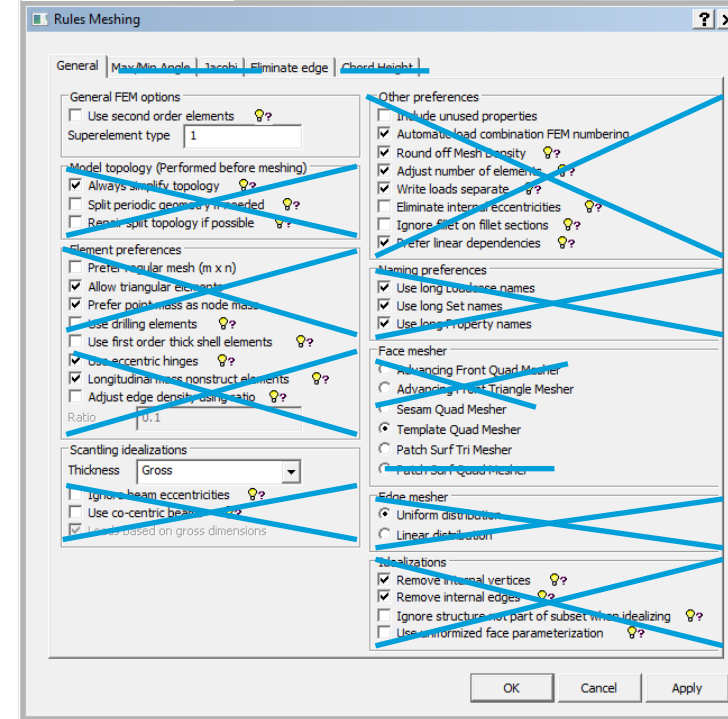
Full mode



Ship mode



Full mode



Full mode Rules Meshing dialog box showing various meshing options. Many options are crossed out with blue lines, indicating they are filtered out in ship mode.

General | Max/Min Angle | Jacobi | Eliminate edge | Chord Height

General FEM options

- ☐ Use second order elements
- Superelement type: 1

Model topology (Performed before meshing)

- ☒ Always simplify topology
- ☐ Split periodic geometry if needed
- ☐ Remove split topology if possible

Element preferences

- ☐ Prefer regular mesh (m x n)
- ☒ Allow triangular elements
- ☒ Prefer point mass as node mass
- ☐ Use drilling elements
- ☐ Use first order thick shell elements
- ☒ Use eccentric hinges
- ☒ Longitudinal mesh construct elements
- ☐ Adjust edge density using ratio

Ratio: 0.1

Scantling idealizations

- Thickness: Gross
- ☐ Ignore beam eccentricities
- ☐ Use co-centric beams
- ☒ Length based on gross dimensions

Other preferences

- ☐ Include unused properties
- ☒ Automate load combination FEM numbering
- ☒ Round off Mesh Density
- ☒ Adjust number of elements
- ☐ Write loads separate
- ☐ Eliminate internal eccentricities
- ☐ Ignore fillet on fillet sections
- ☒ Prefer linear dependencies

Naming preferences

- ☒ Use long Load case names
- ☒ Use long Set names
- ☒ Use long Property names

Face mesher

- ☐ Advancing Front Quad Mesher
- ☐ Advancing Front Triangle Mesher
- ☐ Sesam Quad Mesher
- ☒ Template Quad Mesher
- ☐ Patch Surf Tri Mesher
- ☐ Patch Surf Quad Mesher

Edge mesher

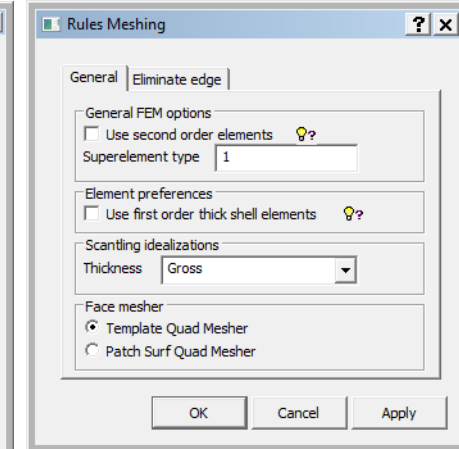
- ☒ Uniform distribution
- ☐ Linear distribution

Realizations

- ☒ Remove internal vertices
- ☒ Remove internal edges
- ☐ Ignore structure not part of subset when idealizing
- ☐ Use uniformized face parameterization

OK Cancel Apply

Ship mode



Ship mode Rules Meshing dialog box showing a filtered list of meshing options. Only options applicable for ships are visible.

General | Eliminate edge

General FEM options

- ☐ Use second order elements
- Superelement type: 1

Element preferences

- ☐ Use first order thick shell elements

Scantling idealizations

- Thickness: Gross

Face mesher

- ☒ Template Quad Mesher
- ☐ Patch Surf Quad Mesher

OK Cancel Apply

# Nauticus Hull Finite Element

All Rule sets covered

## Part 3 Chapter 7

## Part 5 Chapter X

## Class Guideline-0127

## IACS CSR BC&OT



DNV

RULES FOR CLASSIFICATION

[Ships](#)

Edition July 2021

**Part 3 Hull**

**Chapter 7 Finite element analysis**

The content of this service document is the subject of intellectual property rights reserved by DNV AS ("DNV"). The user accepts that it is prohibited by anyone else but DNV and/or its licensees to offer and/or perform classification, certification and/or verification services, including the issuance of certificates and/or declarations of conformity, wholly or partly, on the basis of and/or pursuant to this document whether free of charge or chargeable, without DNV's prior written consent. DNV is not responsible for the consequences arising from any use of this document by others.

The PDF electronic version of this document available at the DNV website [dnv.com](#) is the official version. If there are any inconsistencies between the PDF version and any other available version, the PDF version shall prevail.

DNV AS



DNV

RULES FOR CLASSIFICATION

[Ships](#)

Edition July 2021

**Part 5 Ship types**

**Chapter 2 Container ships**

The content of this service document is the subject of intellectual property rights reserved by DNV AS ("DNV"). The user accepts that it is prohibited by anyone else but DNV and/or its licensees to offer and/or perform classification, certification and/or verification services, including the issuance of certificates and/or declarations of conformity, wholly or partly, on the basis of and/or pursuant to this document whether free of charge or chargeable, without DNV's prior written consent. DNV is not responsible for the consequences arising from any use of this document by others.

The PDF electronic version of this document available at the DNV website [dnv.com](#) is the official version. If there are any inconsistencies between the PDF version and any other available version, the PDF version shall prevail.

DNV AS



DNV

CLASS GUIDELINE

DNV-CG-0127

Edition August 2021

**Finite element analysis**

The content of this service document is the subject of intellectual property rights reserved by DNV AS ("DNV"). The user accepts that it is prohibited by anyone else but DNV and/or its licensees to offer and/or perform classification, certification and/or verification services, including the issuance of certificates and/or declarations of conformity, wholly or partly, on the basis of and/or pursuant to this document whether free of charge or chargeable, without DNV's prior written consent. DNV is not responsible for the consequences arising from any use of this document by others.

The PDF electronic version of this document available at the DNV website [dnv.com](#) is the official version. If there are any inconsistencies between the PDF version and any other available version, the PDF version shall prevail.

DNV AS



Common Structural Rules for  
Bulk Carriers and  
Oil Tankers



01 JAN 2020

gives **all general rule requirements** for Cargo Hold FEA and partial models, global strength and local structural strength calculations incl. documentation requirements

defines **exceptions from Pt.3 and additional assumptions**, loading scenarios and load combinations

describes the **scope and methods required for structural analyses of ships** and the background for how such analyses should be carried out

All requirements collected in one booklet

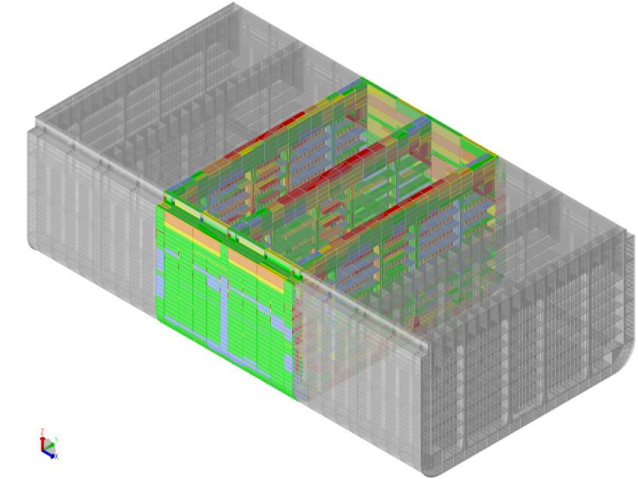


# Nauticus Hull Finite Element

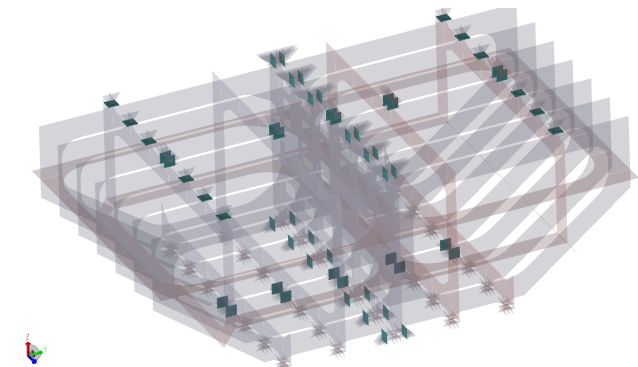
Efficient tool for container ship FE models required by the Rules

- Midship cargo hold model
  - Re-use of cross sections from prescriptive Rule checks that can be extruded in GeniE
  - Efficient modelling features
  - Automatic creation of Rule loading conditions, boundary conditions, corrosion additions
  - Automatic adjustment of hull girder loads to targets specified by the Rules
  - Automatic creation of buckling panels
  - All FE Rule checks implemented (plate buckling, stiffener buckling, plate yielding)
  - High analysis performance (meshing, solver) ensured by multithreading
  - Effective work process on design iterations, with high performance
- Gas fuelled container ships with independent Type-B tank
  - GeniE supports temperature loads
  - Automatic creation of Rule loading conditions
  - Integrated tool for considering friction in vertical supports
  - Initial clearances in anti-pitch, anti-roll and anti-lifting supports can be considered
  - Supports sub-models with effective load transfer and prescriptive displacements from cargo hold model
  - FE fatigue stresses for crack propagation analysis and fracture mechanics analysis

FE buckling results



Independent type-B tank with initial clearances



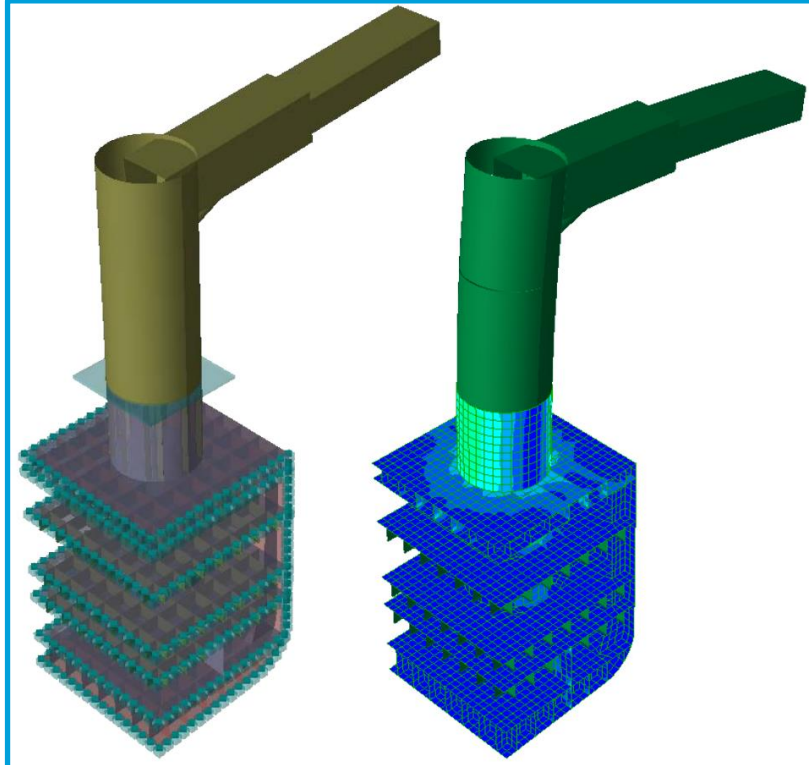
# Nauticus Hull Finite Element

## Efficient generic FE tool

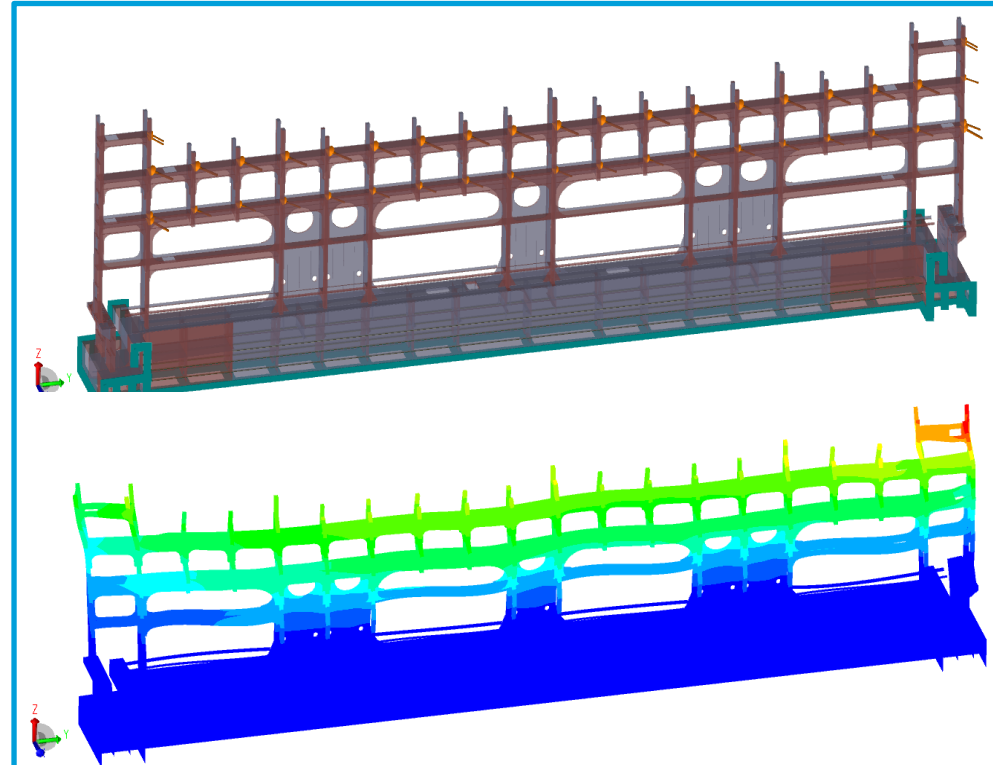
*By having GeniE, you would not need other FE tools, as GeniE is also an efficient generic FE tool*

- As a designer, you may need to create a FE model for foundations /supporting structures
- Sesam GeniE has numerous efficient general modelling features and mesh tuning options for such applications
- Other typical applications
  - Cell guide structures
  - Wave breaker
  - Rudder horn
  - Crane foundations
  - Mooring foundations
  - Etc

GeniE efficient tool for crane foundations



GeniE efficient tool for cell guide structures



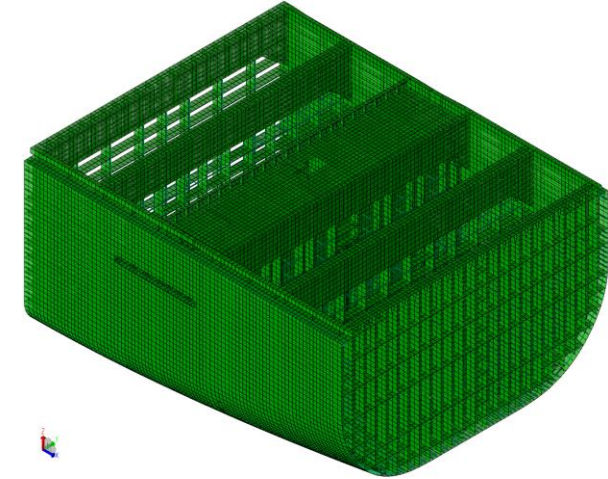
# Nauticus Hull Finite Element

Bulk data file \*.bdf can be imported to GeniE

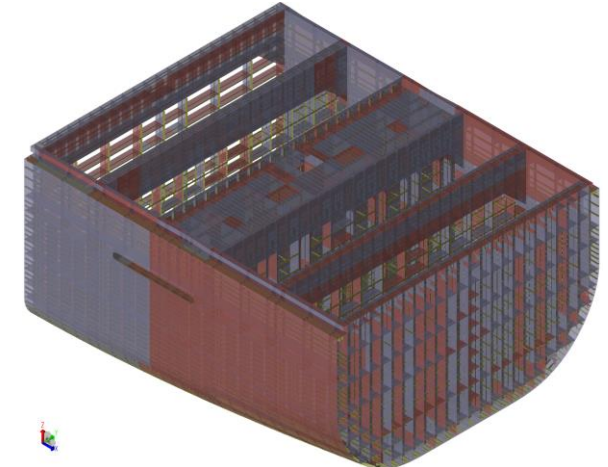
- GeniE can import a finite element model on the bulk data file format, \*.bdf
- Prior to importing the finite elements, GeniE can scan the elements, identify model issues and propose fixes
- During import, GeniE is stitching the elements into geometries (plates and stiffeners)
- On this geometry model, loading conditions and buckling panels can be automatically added, similar to a geometry model created from scratch
- The finite elements will remain during analysis, only loads and updated scantlings are assigned to the same elements
- The Sestra solver is made robust, allowing for elements with bad shape and, also singularities in the stiffness matrix

GeniE allows for re-use of models created in other FE tools by being able to import the bulk data file format \*.bdf

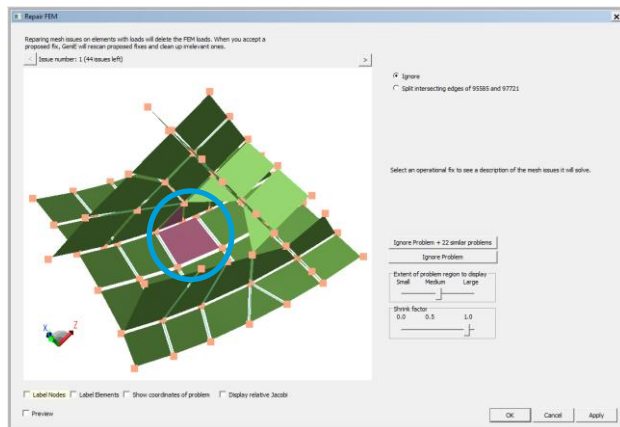
**\*.bdf scanned and repaired prior to import to GeniE**



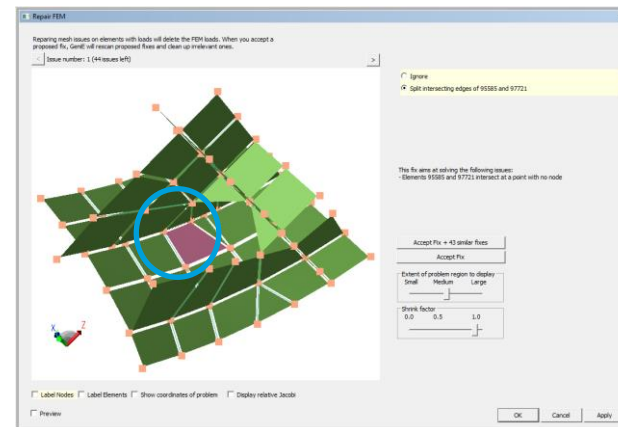
**Finite elements stitched together to geometries during import to GeniE**



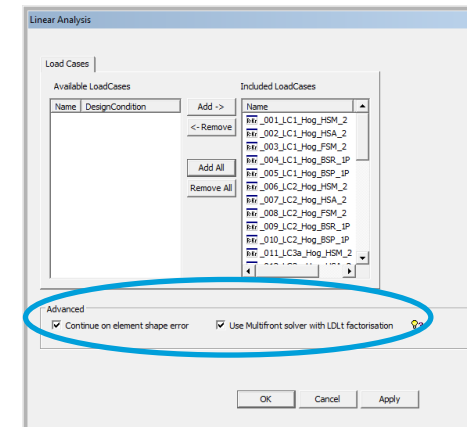
**FE issue detected by scanning tool**



**Fix proposed, also for similar issues**



**Robust Solver**



# Container ship cargo hold model

Re-use of Section Scantlings model from Nauticus Hull

Typical geometry parts that need to be modelled in GeniE

Automatic creation of Rule loading conditions

Analysis run

Visualisation of analysis results

FEA Rule checks (yielding and buckling) run

Visualisation of FEA Rule checks run

Typical design iteration – Create insert plate

Typical design iteration – Add buckling stiffener

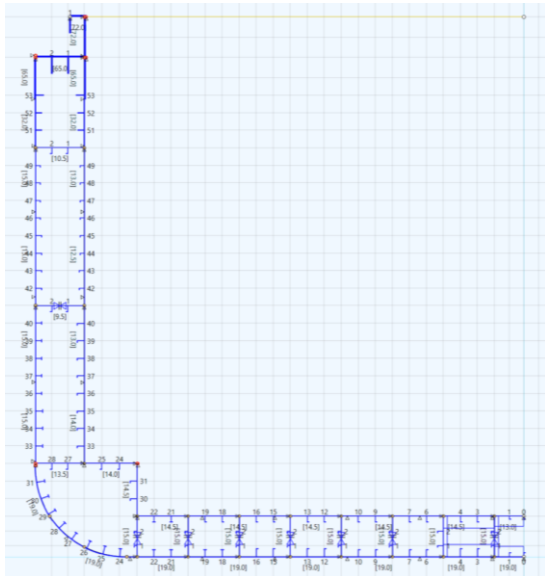
Report template for automatic report generation

# Container ship cargo hold model

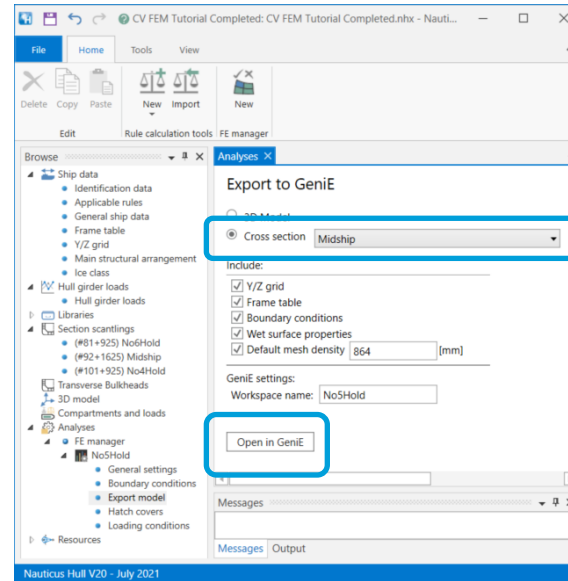
## Re-use of Section Scantlings model from Nauticus Hull

- Nauticus Hull Section Scantlings is used for prescriptive cross sectional analysis
- A selected cross section can be used for creation of longitudinal members in the FE model by extracting the cross section in GeniE
- The process is fully atomised: GeniE can be opened by a single click on a button, and then the cross section will be extruded

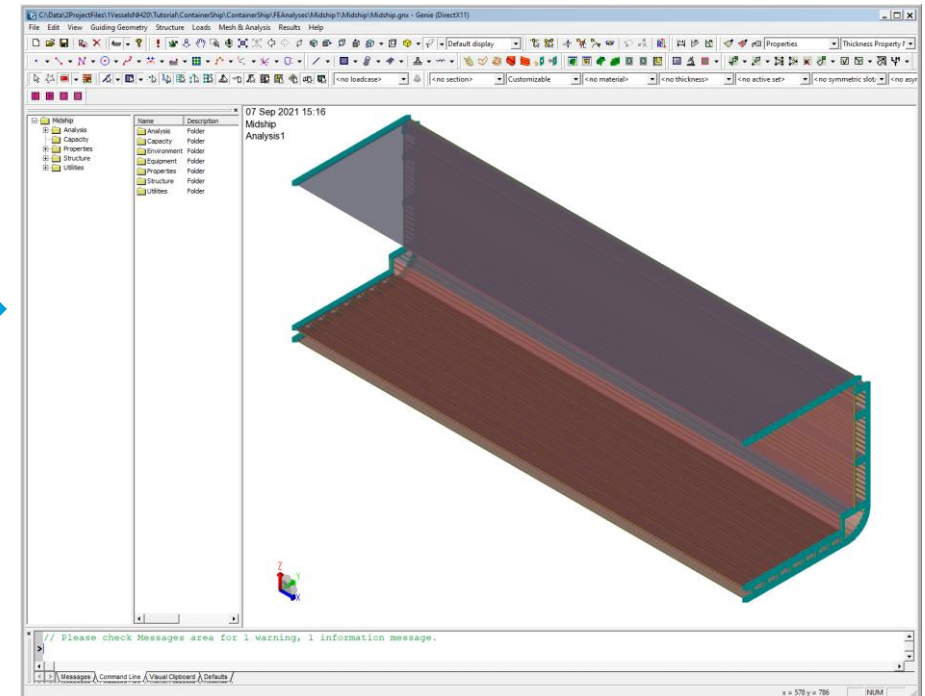
Typical midship cross section in Section Scantlings



Automatic export from Nauticus Hull to GeniE



Cross section extruded in GeniE with same scantlings as in Section Scantlings

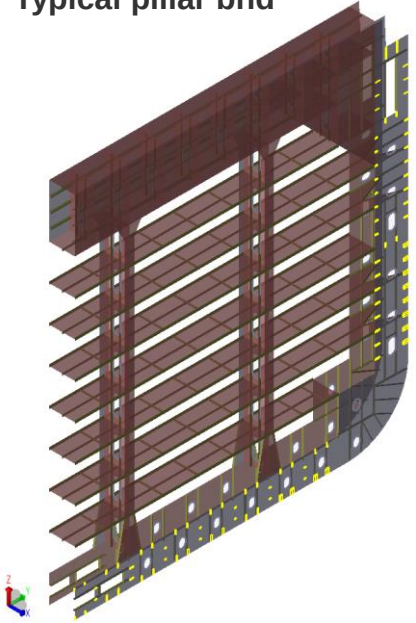


# Container ship cargo hold model

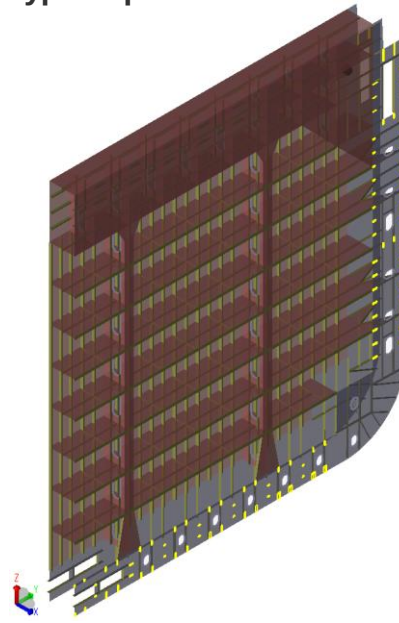
## Typical geometry parts that need to be modelled in GeniE

*Powerful features for copying and mirroring typical parts give efficient modelling of a complete cargo hold model*

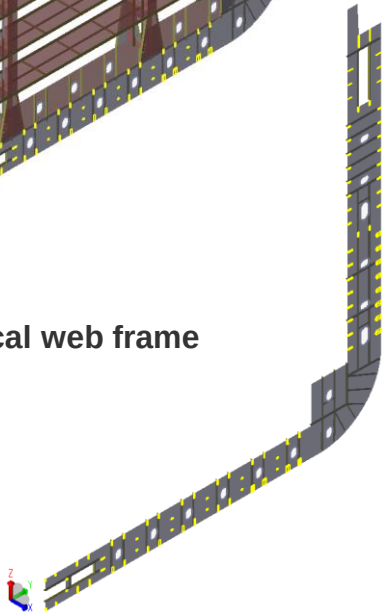
Typical pillar bhd



Typical pillar tbhd



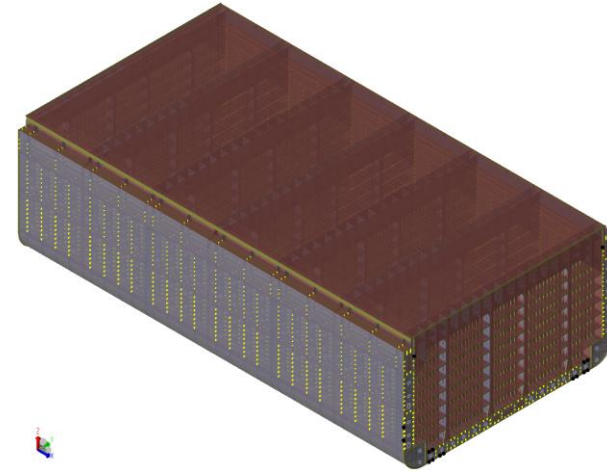
Typical web frame



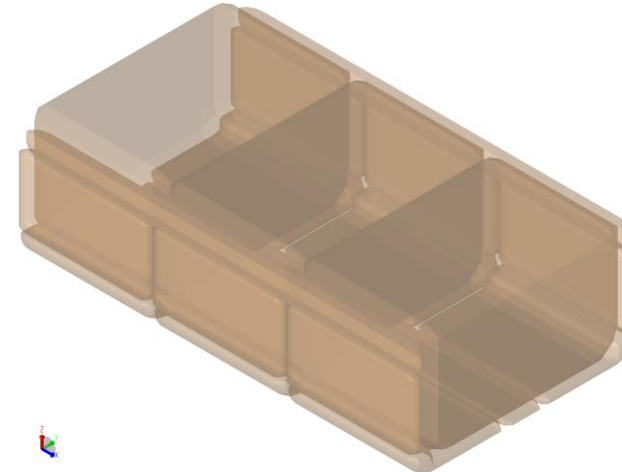
Openings iwo longitudinal girders/stringers



The typical parts are copied and mirrored in order to create a complete cargo hold model



Compartments are automatically generated, based on the modelled watertight geometry



# Container ship cargo hold model

## Automatic creation of Rule loading conditions

Loading conditions, boundary conditions and corrosion addition information are automatically created according to Rules

### Rule loading conditions

| No.                 | Description                             | Loading pattern<br>all hold size | Container weights<br>and tank content                                                                                                           | Draught      | % of<br>perm.<br>SWSP | % of<br>perm.<br>SWSP | Dynamic<br>load case                        |
|---------------------|-----------------------------------------|----------------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------|--------------|-----------------------|-----------------------|---------------------------------------------|
| Sweeping conditions |                                         |                                  |                                                                                                                                                 |              |                       |                       |                                             |
| LC1                 | 40 R Heavy <sup>(1)</sup>               |                                  | on deck: max 40 R stack weight<br>in hold: 30.5 UTEU not exceeding max 40 R stack weight<br>all tanks empty                                     | $T_{LC}$     | 100% (hog.)           | ≤ 100%                | HSM-2<br>HSA-2<br>FSM-2<br>BSR-1P<br>BSP-1P |
| LC2                 | 40 R Light <sup>(1)</sup>               |                                  | on deck: 90% of max 40 R stack weight not exceeding 17 UTEU<br>in hold: 55% of max 40 R stack weight not exceeding 16.5 UTEU<br>all tanks empty | $T_{LC}$     | 100% (hog.)           | ≤ 100%                | HSM-2<br>HSA-2<br>FSM-2<br>BSR-1P<br>BSP-1P |
| LC3a                | 40 R Heavy <sup>(1)</sup>               |                                  | on deck: max 40 R stack weight<br>in hold: 30.5 UTEU not exceeding max 40 R stack weight<br>all tanks empty                                     | $T_{LC}$     | 100% (hog.)           | ≤ 100%                | HSM-2<br>HSA-2<br>FSM-2                     |
| LC3b                | Forward bay empty                       |                                  | on deck: max 40 R stack weight<br>in hold: 30.5 UTEU not exceeding max 40 R stack weight<br>all tanks empty                                     | $T_{LC}$     | 100% (hog.)           | ≤ 100%                | HSM-2<br>HSA-2<br>FSM-2                     |
| LC4                 | Forward bay empty                       |                                  | on deck: max 40 R stack weight<br>in hold: 30.5 UTEU not exceeding max 40 R stack weight<br>all tanks empty                                     | $T_{LC}$     | 100% (hog.)           | ≤ 100%                | HSM-2<br>HSA-2<br>FSM-2                     |
| LC5                 | 40 R Heavy <sup>(1)</sup>               |                                  | on deck: max 20 R stack weight<br>in hold: max 20 R stack weight<br>all tanks empty                                                             | $0.9 T_{LC}$ | 100% (hog.)           | ≤ 100%                | HSM-1<br>HSA-1<br>FSM-1<br>BSR-1P<br>BSP-1P |
| LC6                 | Heavy deck light load <sup>(1)</sup>    |                                  | on deck: max 20 R stack weight<br>in hold: 10 UTEU<br>all tanks empty                                                                           | $0.9 T_{LC}$ | 100% (hog.)           | ≤ 100%                | HSM-1<br>HSA-1<br>FSM-1<br>BSR-1P<br>BSP-1P |
| LC7                 | Pitching <sup>(2)</sup>                 |                                  | on deck: max 20 R stack weight<br>in hold: 30.5 UTEU not exceeding max 40 R stack weight<br>all full oil tanks full at inclined side            | $T_{LC}$     | 100% (hog.)           | ≤ 100%                | HSM-1<br>HSA-1<br>FSM-1<br>BSR-1P<br>BSP-1P |
| Damaged condition   |                                         |                                  |                                                                                                                                                 |              |                       |                       |                                             |
| LC7                 | Flooded damage condition <sup>(3)</sup> |                                  | on deck: max 40 R stack weight<br>in hold: 30.5 UTEU not exceeding max 40 R stack weight<br>all full oil tanks full at inclined side            | $T_{LC}$     | 100% (hog.)           | ≤ 100%                | HSM-1<br>HSA-1<br>FSM-1<br>BSR-1P<br>BSP-1P |

### Load template for Rule loading conditions

Design load combinations - Container carriers

LC Loading pattern

Description Loads

Draught (m)  $K_r$  (m) GM (m)

Containers in hold

Container height 86" (Standard)

No of containers in stack 9

Design container loads 20 ft 30 (t/TEU)

Design container loads 40 ft 30 (t/TEU)

Containers on deck

Container height 98" (Standard)

No of containers in stack 8

No of stacks in width 19

Max stack weight 20 ft 90 (t/stack)

Max stack weight 40 ft 180 (t/stack)

Mixed storage is applicable

OK Cancel Save Default

### Detailed description of the Rule loading conditions

Design load combinations - Container carriers

Containers in hold

Container height 86" (Standard)

No of containers in stack 9

Design container loads 20 ft 30 (t/TEU)

Design container loads 40 ft 30 (t/TEU)

Containers on deck

Container height 98" (Standard)

No of containers in stack 8

No of stacks in width 19

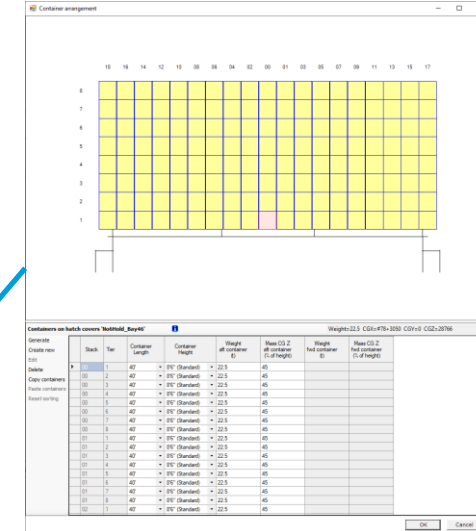
Max stack weight 20 ft 90 (t/stack)

Max stack weight 40 ft 180 (t/stack)

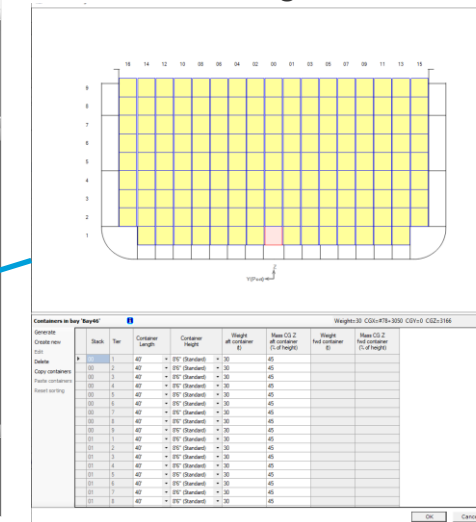
Mixed storage is applicable

OK Cancel Save Default

### Container stowage on hatches



### Container stowage in holds



### Nauticus Hull creates a loading condition file including:

- Corrosion additions
- Sets
- Boundary conditions
- Wet surfaces (for sea pressures)
- Structure types (for buckling settings)
- Compartment names (mapping)

Copy to XML file

Select the Loading Conditions to be included

Input XML file (exported from GenE)

Output XML file (to be imported in GenE)

Position of origin in GenE relative to (AP, Centre line, Base line)

X 0 (m) Y 0 (m) Z 0 (m)

Other

Include corrosion additions

Include sets

Include boundary conditions

Include wet surface properties

Include structure types

Include compartment names

OK Cancel

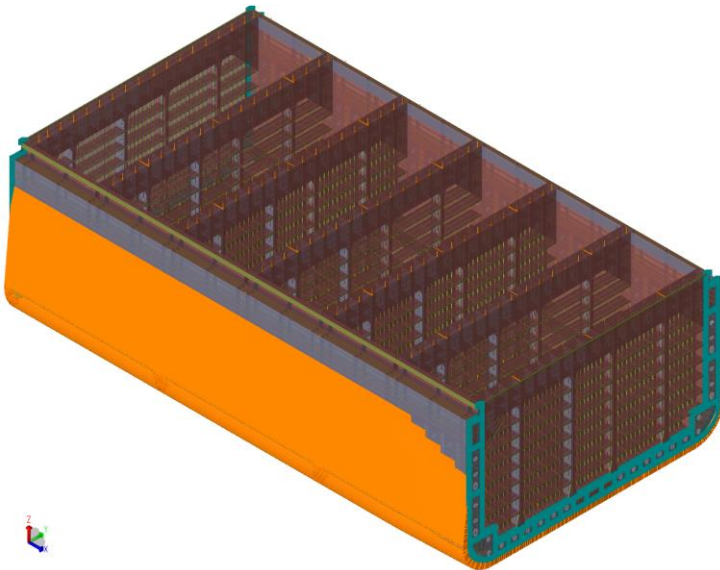
# Container ship cargo hold model

## Analysis run

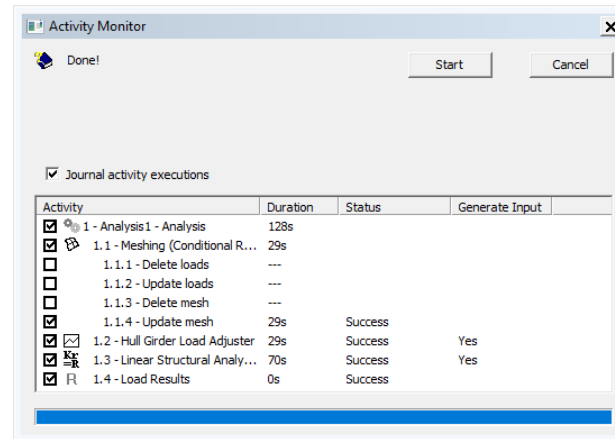
*High analysis performance is enabled by multi-threading analysis tasks as meshing and when running the solver*

- All model information is assigned to the geometry model (e.g. loads, and properties like material, thickness, corrosion additions)
- When executing the analysis, first step for GeniE is to update the finite element model (mesh and loads)
- GeniE's conditional mesher will scan the model and only update elements and FE loads that are affected by any update to the geometry model
- Adjustment of the hull girder loads will only be done, when required (e.g. loads have been updated)
- Analysis solver, Sestra, allowing for multi-threading and faster than competitors

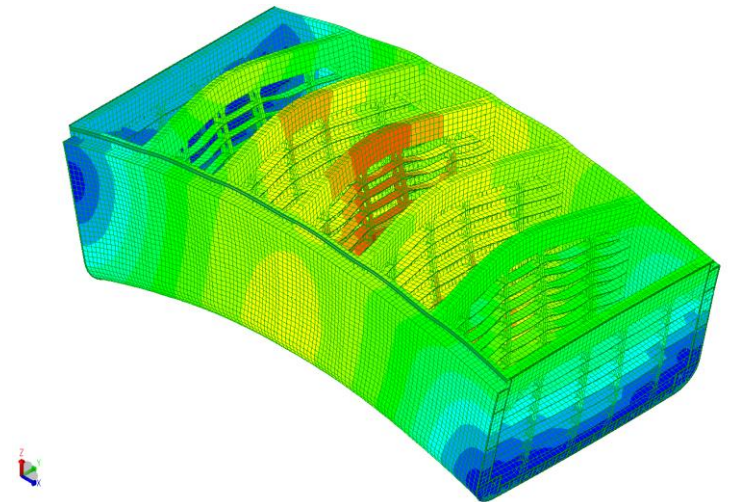
Loading conditions applied to the geometry model



Required analysis steps controlled by GeniE's activity monitor



Analysis results available in GeniE

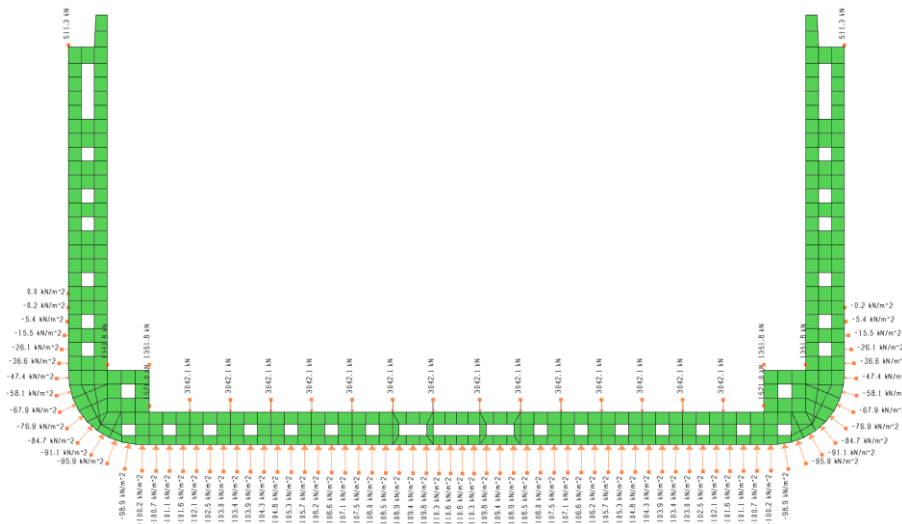


# Container ship cargo hold model

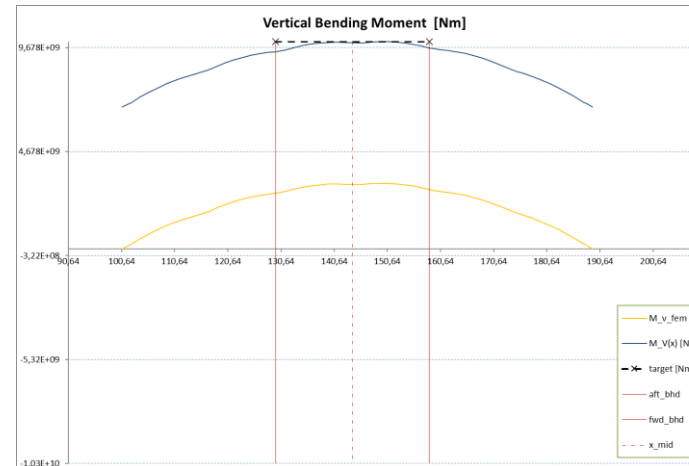
## Visualisation of analysis results

- Loads (e.g. sea pressures and container forces) and can easily be displayed with units according to Rules
- Graphical display of hull girder adjustments is available
- Following analysis results are available in GeniE
  - Displacements (all, x, y and z)
  - Gaussian membrane stresses in element centroid (von Mises, shear and normal)

Sea pressures and container forces on the FEM



Display of hull girder load adjustments



Gaussian membrane von Mises stresses

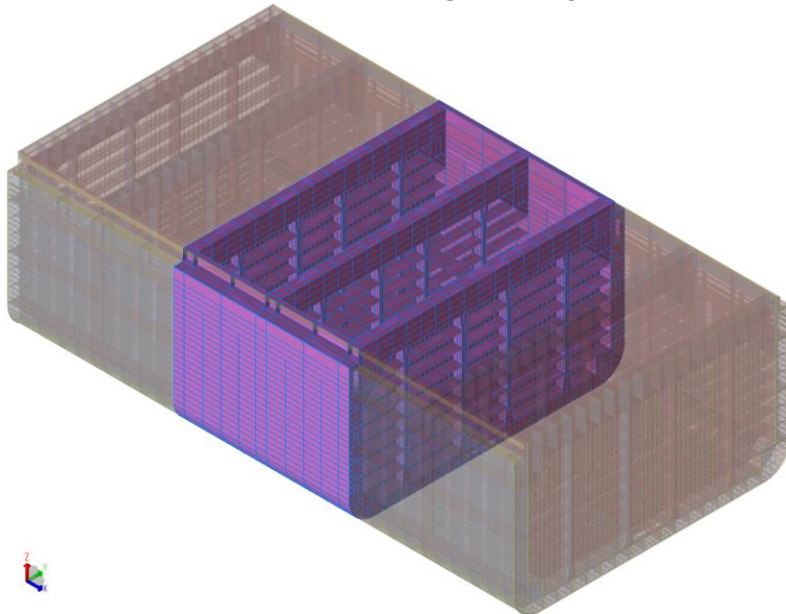


# Container ship cargo hold model

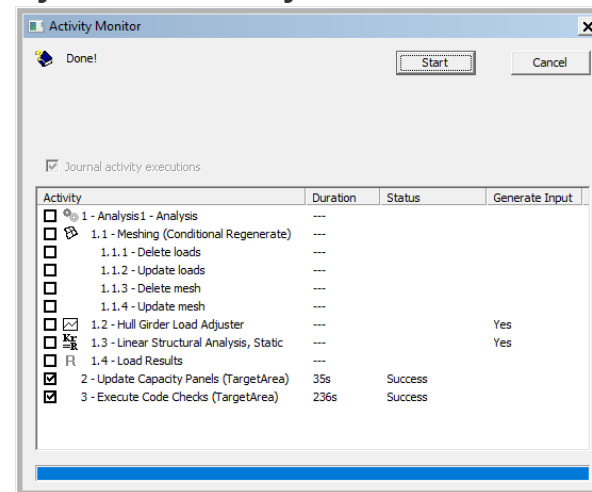
## FEA Rule checks (yielding and buckling) run

- GeniE automatically creates the buckling panels based on the modelled geometries (stiffeners and intersecting PSMs)
- User can specify relevant parts for the FEA Rule checks (either parts of the target area or all members in one run)
- User can select loading conditions and also relevant Rule checks (yielding & buckling, buckling only, or yielding only)
- The default parameters created, based on the modelled geometry, can be edited by the user (e.g. tuning panel width)

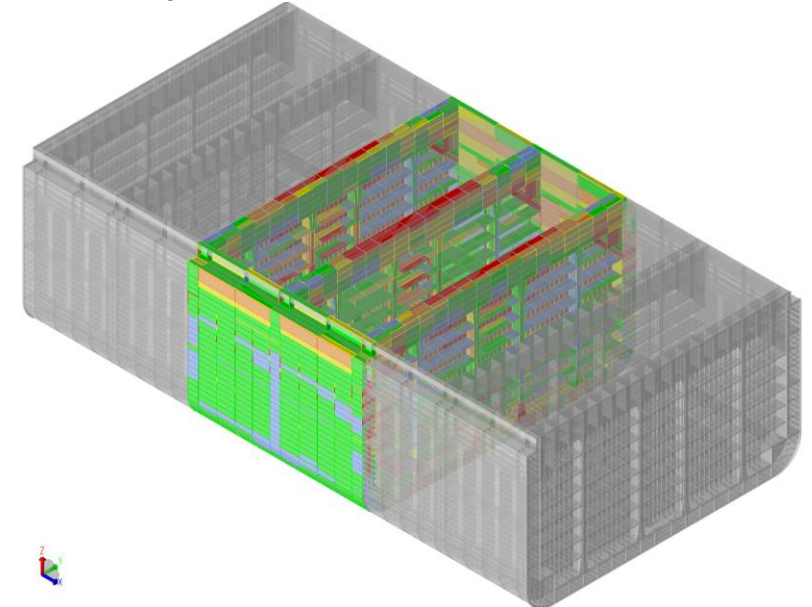
**Buckling panels automatically  
created based on modelled geometry**



**Required analysis steps controlled  
by GeniE's activity monitor**



**Several FEA Rule check  
result options are available**

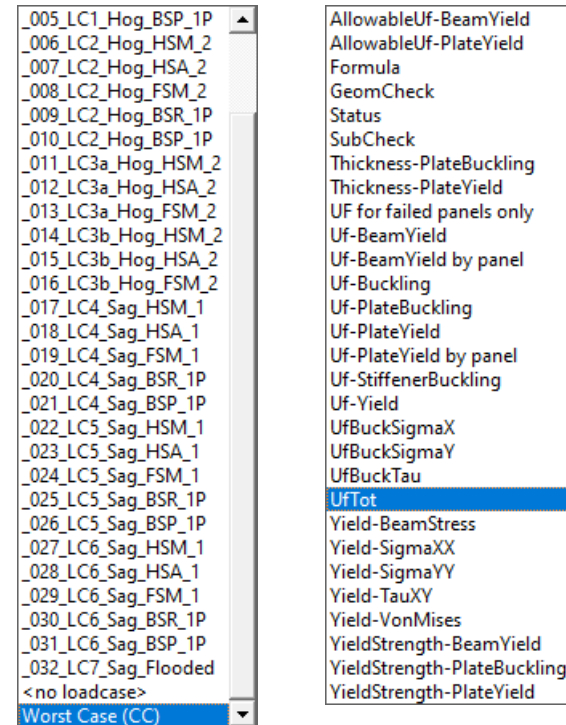


# Container ship cargo hold model

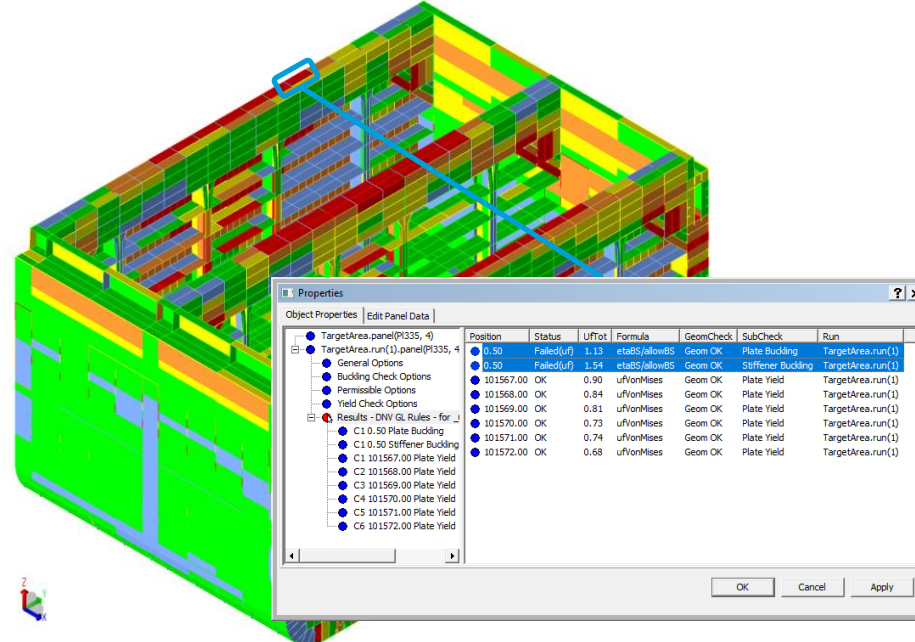
## Visualisation of FEA Rule checks run

- User can select display of results for a selected load case or worst of all load cases (scanning all load cases)
- Several graphically results are available (e.g. plate yielding, plate buckling, stiffener buckling)
- Detailed results, and input parameters applied in the Rule checks can be easily accessed
- For a selected buckling panel the user can edit buckling parameters and re-run buckling check via direct access to the Single Panel Buckling tool

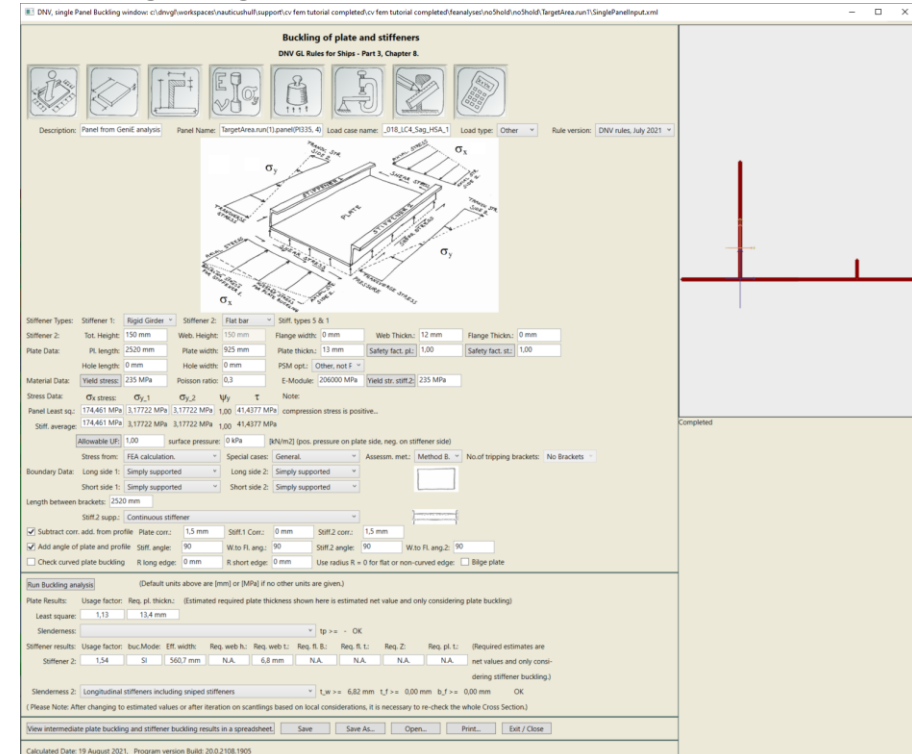
### Several selection options available for displaying FEA Rule check results



### Detailed results available when selecting a buckling panel



### Buckling design iterations available via SPBT

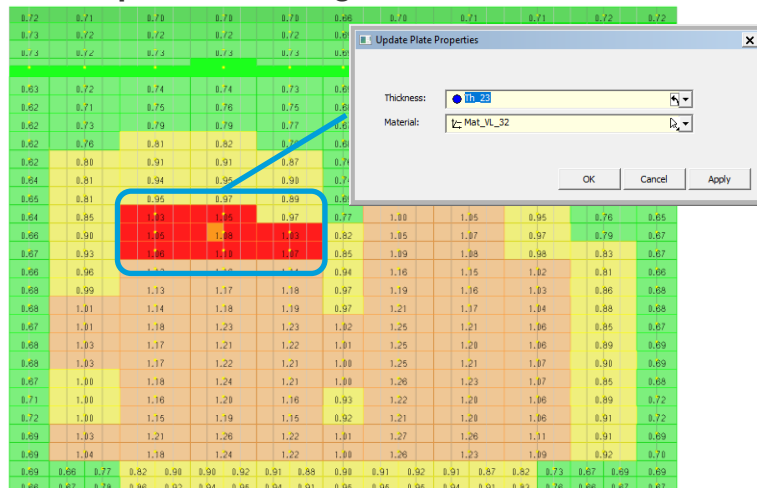


# Container ship cargo hold model

## Typical design iteration – Create insert plate

- User can interactively select multiple elements and update the scantlings (thickness and/or material)
- In the background, GeniE will automatically
  - Create a new plate with new properties; and
  - Update the finite element information
- GeniE's activity monitor will only run required analysis steps
  - Following steps will be excluded: Meshing (as the elements are directly updated), adjustment of hull girder loads (as no loading has been changed)
  - Following steps will be run: Analysis (as stiffness in the model is changed), update of buckling panels (as thickness and/or material is changed), running the FEA Rule checks

### User can interactively select elements and update scantlings

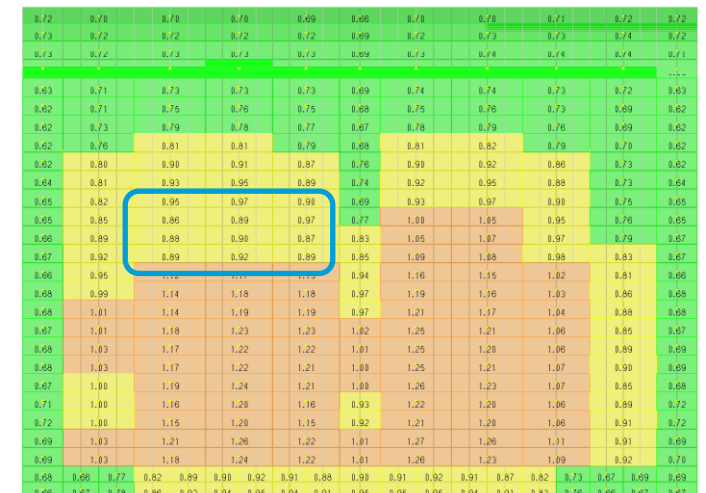


### Required analysis steps controlled by GeniE's activity monitor

| Activity                                 | Duration | Status  | Generate Input |
|------------------------------------------|----------|---------|----------------|
| 1 - Analysis 1 - Analysis                | 58s      |         |                |
| 1.1 - Meshing (Conditional Regenerate)   | 0s       |         |                |
| 1.1.1 - Delete loads                     | ---      |         |                |
| 1.1.2 - Update loads                     | ---      |         |                |
| 1.1.3 - Delete mesh                      | ---      |         |                |
| 1.1.4 - Update mesh                      | 0s       |         |                |
| 1.2 - Hull Girder Load Adjuster          | ---      |         | Yes            |
| 1.3 - Linear Structural Analysis, Static | 58s      | Success | Yes            |
| 1.4 - Load Results                       | 0s       | Success |                |
| 2 - Update Capacity Panels (TargetArea)  | 34s      | Success |                |
| 3 - Execute Code Checks (TargetArea)     | 252s     | Success |                |

The 'Activity Monitor' window displays a list of analysis steps. The '1.3 - Linear Structural Analysis, Static' step is highlighted with a blue box, indicating it is the current step being monitored. The window also includes 'Start' and 'Cancel' buttons at the top right.

### Analysis results after design iteration



*User can add buckling stiffener directly on the buckling panel without need to update geometry model and running analysis*

- User can interactively select buckling panel(s) and divide, simulating adding buckling stiffener(s)
- In the background, GeniE will automatically
  - Divide the buckling panel(s) and update the length/width
  - Update the edge stresses for buckling assessment
- GeniE's activity monitor will only run required analysis steps
  - Following steps will be excluded (as no changes to the FE stresses): Meshing, adjustment of hull girder loads, running the solver
  - Following steps will be run: Update of buckling panels (as length/width is changed), running the FEA Rule checks

- Explode Selected Panels
- Join Selected Panels
- Divide Selected Panels by a Plane
- Divide Selected Panels Along Direction
- Start Single Panel Buckling Tool - Current loadcase
- Execute Code Checks
- Delete...
- Properties...
- Labels
  - ColorCode
  - Clear Labels
- Fit Camera to Selection
- Named set...
- View options...
- Visible model

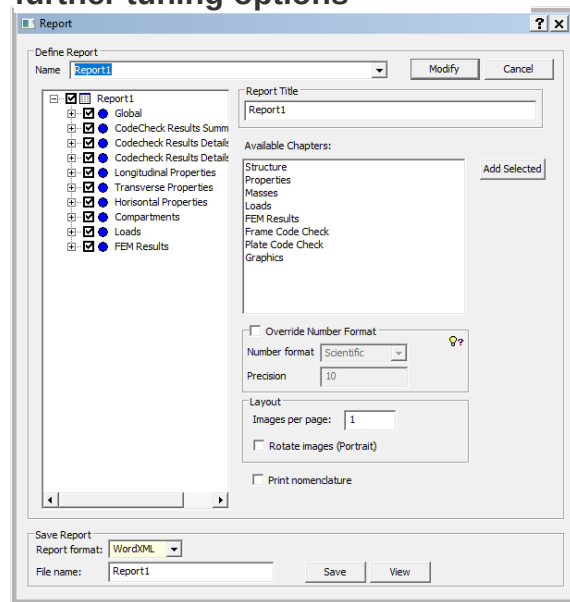
[illegible]

# Container ship cargo hold model

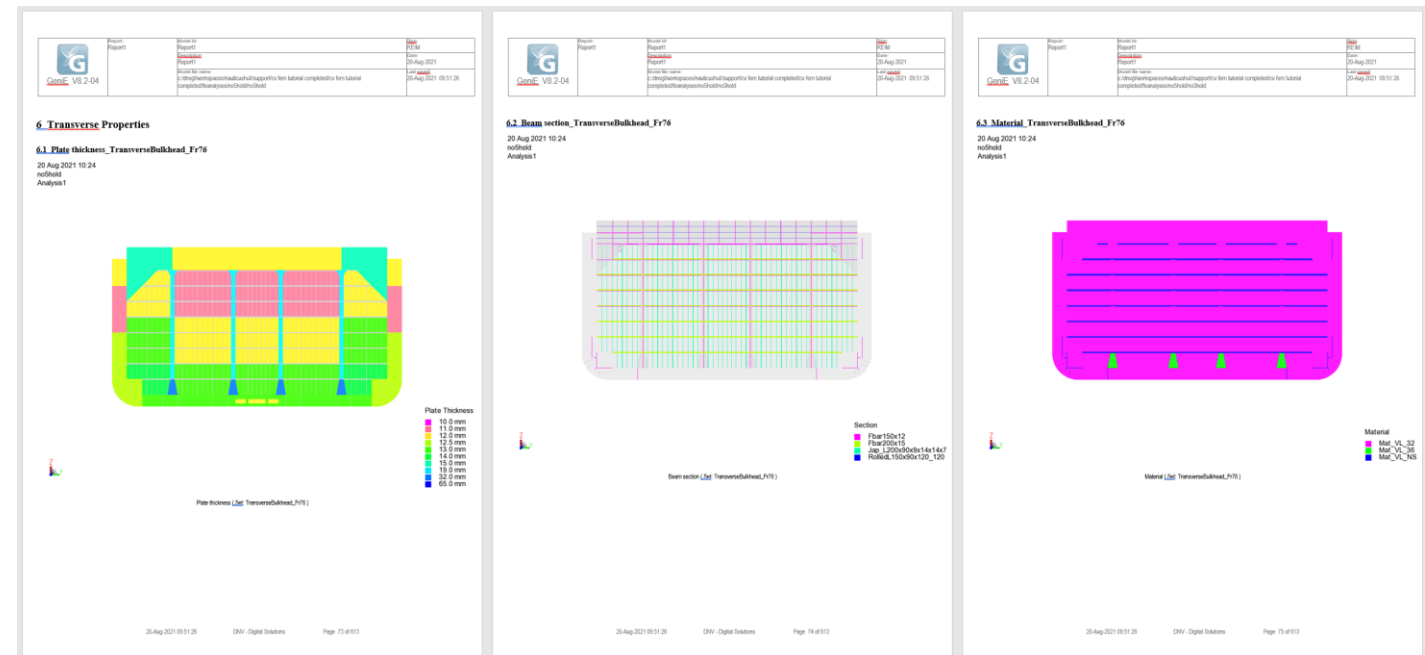
## Report template for automatic report generation

- A default report template is available
- The user can further tune the report content
- The tuned report template can be made available for other projects
- User can select applicable sets and also select display of results for selected load cases (including worst of all)

### Default report template with further tuning options



### The report can show results and properties for selected sets



# Gas fuelled container ship with type-B tank

Temperature loads as required for low cycle fatigue

FE load template for Strength assessment

FE load template for Fatigue assessment

Friction forces iwo vertical wooden supports – TSCF

Initial clearances independent tank supports

Support for sub-models – LMLA (Local Model Load Applier tool)

Support for sub-models – Local Analysis

FE Fatigue assessment by FE Fatigue tool

Tutorial available on DNV's customer portal

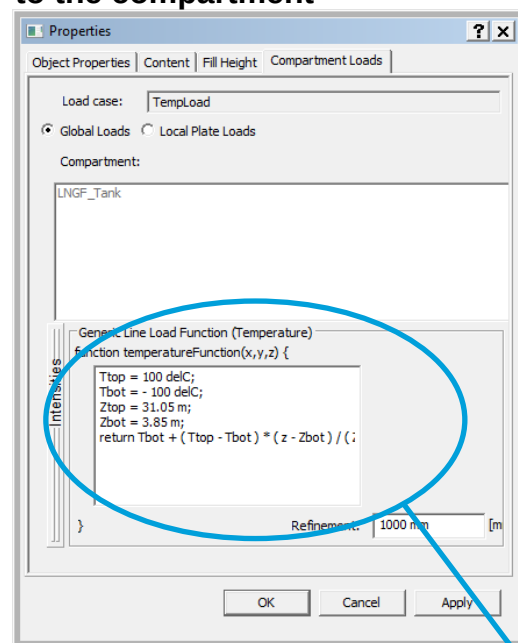
# Gas fuelled CV with type-B tank

GeniE fully supports temperature loads that are required for low cycle fatigue assessments

## Temperature loads as required for low cycle fatigue

- Temperature loads can be assigned to compartments
- The temperature loads behaviours same as for surface loads: Automatically mapped to the finite elements
- The temperature loads are automatically distributed to shell elements and beam elements

### Temperature assigned to the compartment



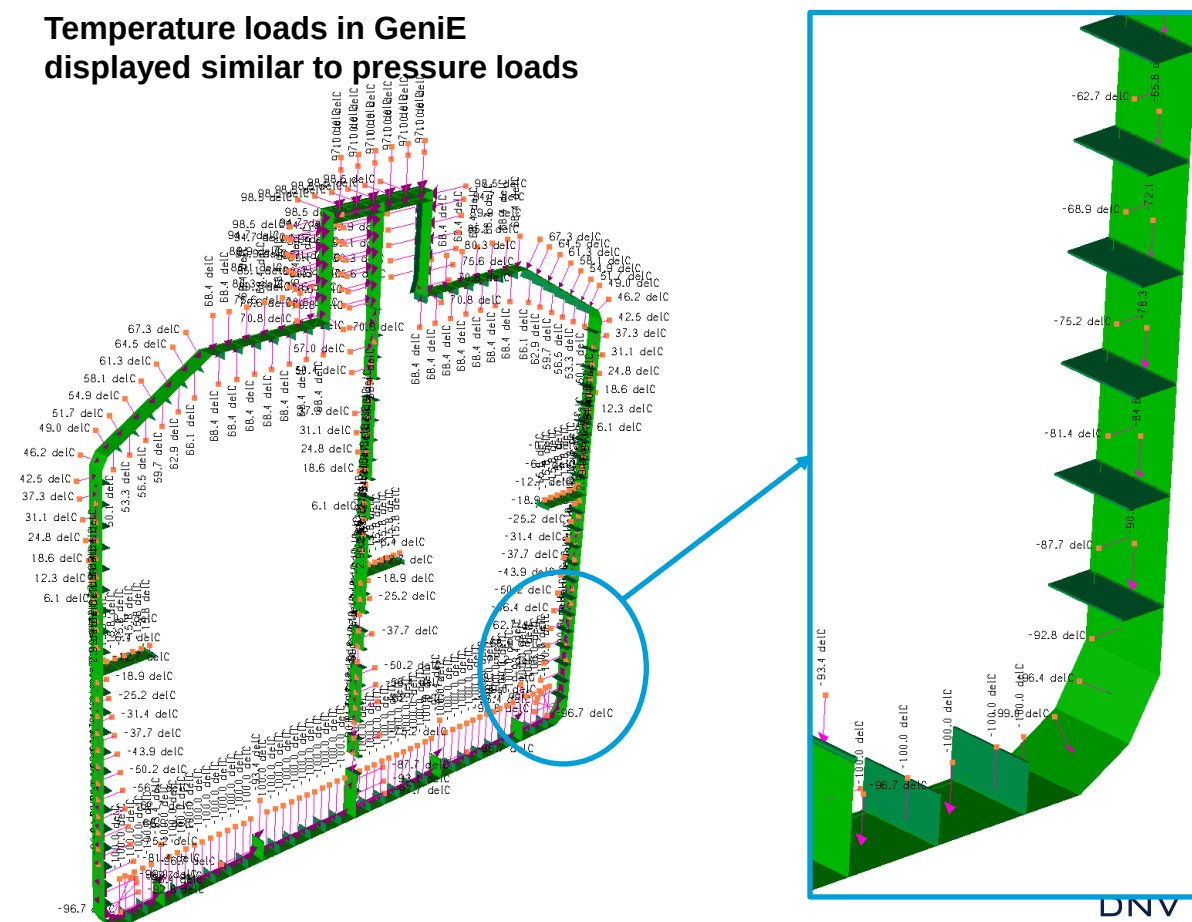
### Temperature loads represented as a JS function

In this case, bottom of tank has a temperature difference of -100°C and top of tank has a temperature difference of 100°C.

JS commands:

```
Ttop = 100 deIC;  
Tbot = - 100 deIC;  
Ztop = 31.05 m;  
Zbot = 3.85 m;  
return Tbot + ( Ttop - Tbot ) * ( z - Zbot ) / ( Ztop - Zbot );
```

### Temperature loads in GeniE displayed similar to pressure loads





- ## FE load template for fatigue assessment

## Allow for user defined motions and accelerations

## A 3D perspective view of a microchip assembly. A purple rectangular chip is mounted on an orange carrier. The carrier has numerous gold pins extending from its top surface. The chip also has pins on its top surface. The assembly is shown in a cutaway view, revealing the internal structure of the carrier and the chip.

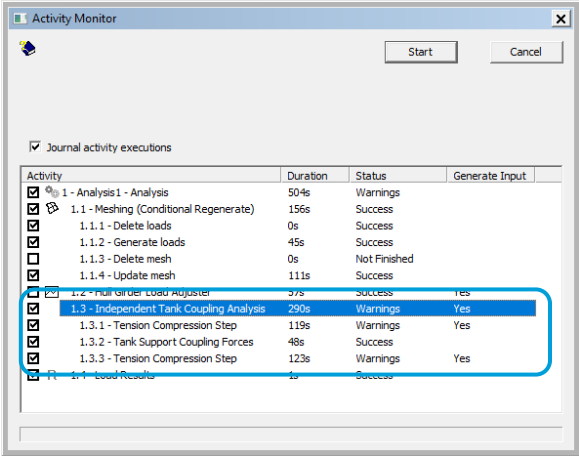


# Gas fuelled CV with type-B tank

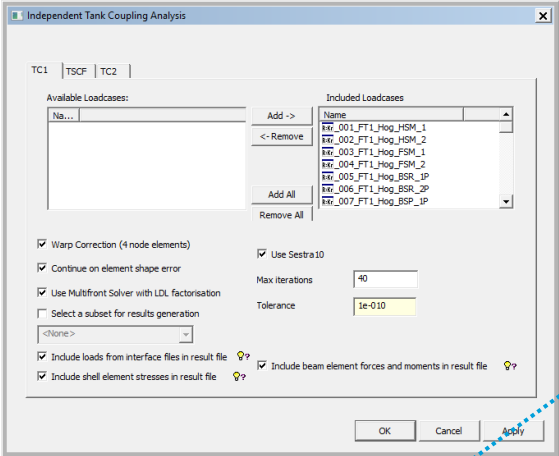
## Friction forces iwo wooden vertical supports – GUI

The advanced analysis method is controlled by GeniE's activity monitor, with an intuitive and user-friendly Graphical Use Interface

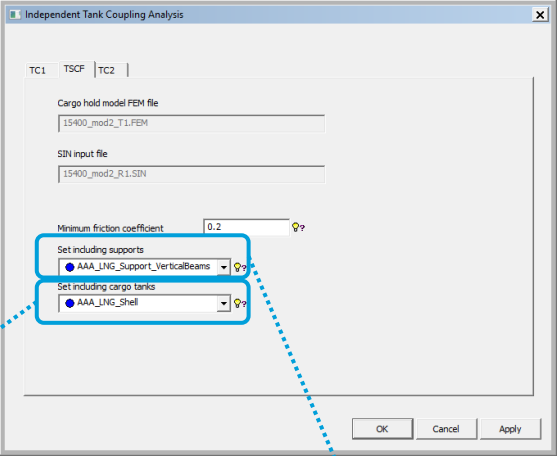
Analysis set-up in GeniE



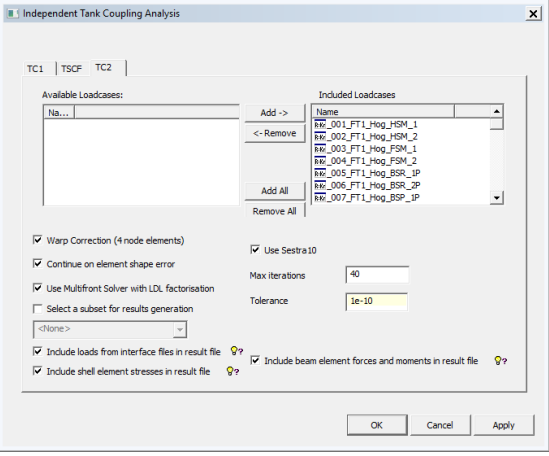
Step 1: T/C analysis



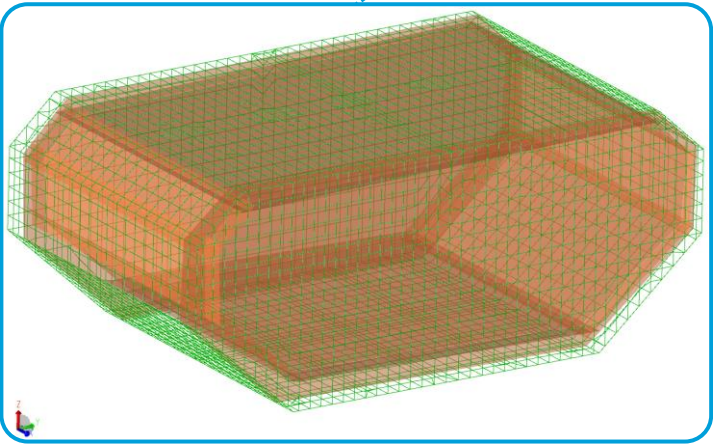
Step 2: TSCF Compute and apply horizontal friction forces



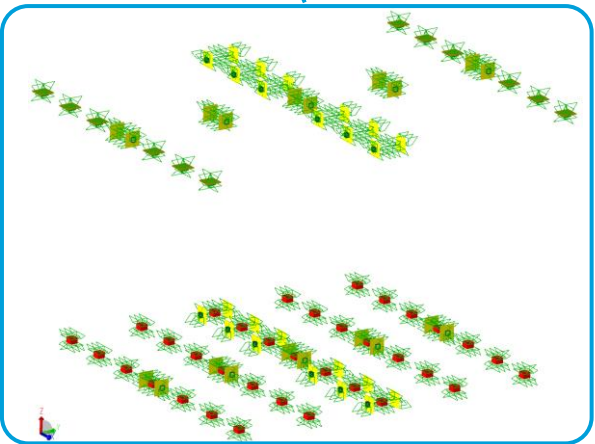
Step 3: T/C analysis, including coupling forces from TSCF



Set containing tank boundary, for computing resulting force from tank pressures



Set containing truss elements (highlighted in red), for vertical forces and distribution of friction forces

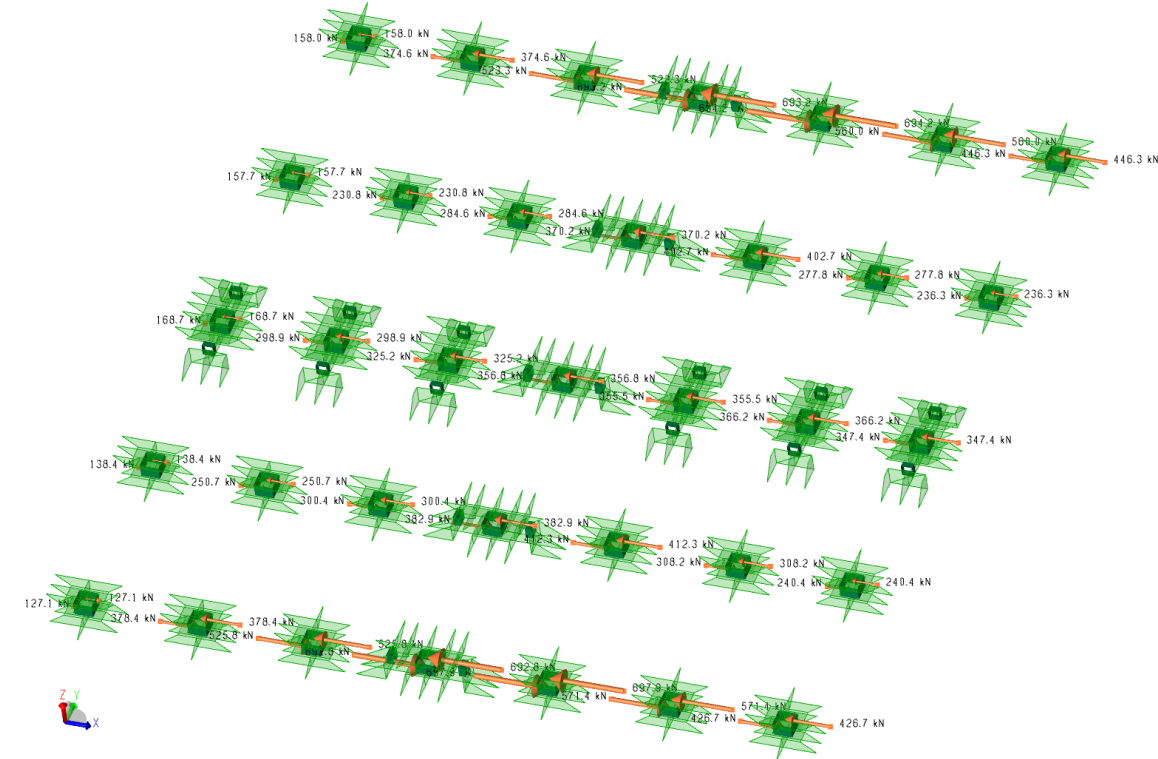


# Gas fuelled CV with type-B tank

## Computation of friction forces iwo wooden vertical supports - TSCF

1. Run T/C analysis
2. Run TSCF (Tank Support Coupling Forces) tool for all load cases
  - i. For the set containing the tank boundary, integrate the pressures and compute
    - Resulting horizontal force:  $F_{tot\_XY}$
    - Resulting force's unit vector in longitudinal and transverse direction:  $\vec{u}_x$  and  $\vec{u}_y$
  - ii. Compute sum of axial forces in way of wooden supports  $F_{tot\_Z}$
  - iii. IF  $F_{Tot\_XY} > F_{Tot\_Z} \times \mu$ , THEN (sliding will occur) in way of each vertical truss,  $i$ ,
    - $F_{i\_X} = F_{i\_Z} \times \mu \times \vec{u}_x$
    - $F_{i\_Y} = F_{i\_Z} \times \mu \times \vec{u}_y$
  - iv. IF  $F_{Tot\_XY} \leq F_{Tot\_Z} \times \mu$ , THEN (sliding will not occur) in way of each vertical truss,  $i$ ,
    - $F_{i\_X} = F_{tot\_XY} \times \vec{u}_x \times F_{i\_Z} / F_{Tot\_Z}$
    - $F_{i\_Y} = F_{tot\_XY} \times \vec{u}_y \times F_{i\_Z} / F_{Tot\_Z}$
3. Run T/C analysis, including friction force couples from TSCF tool

Friction force couples in vertical supports computed by TSCF

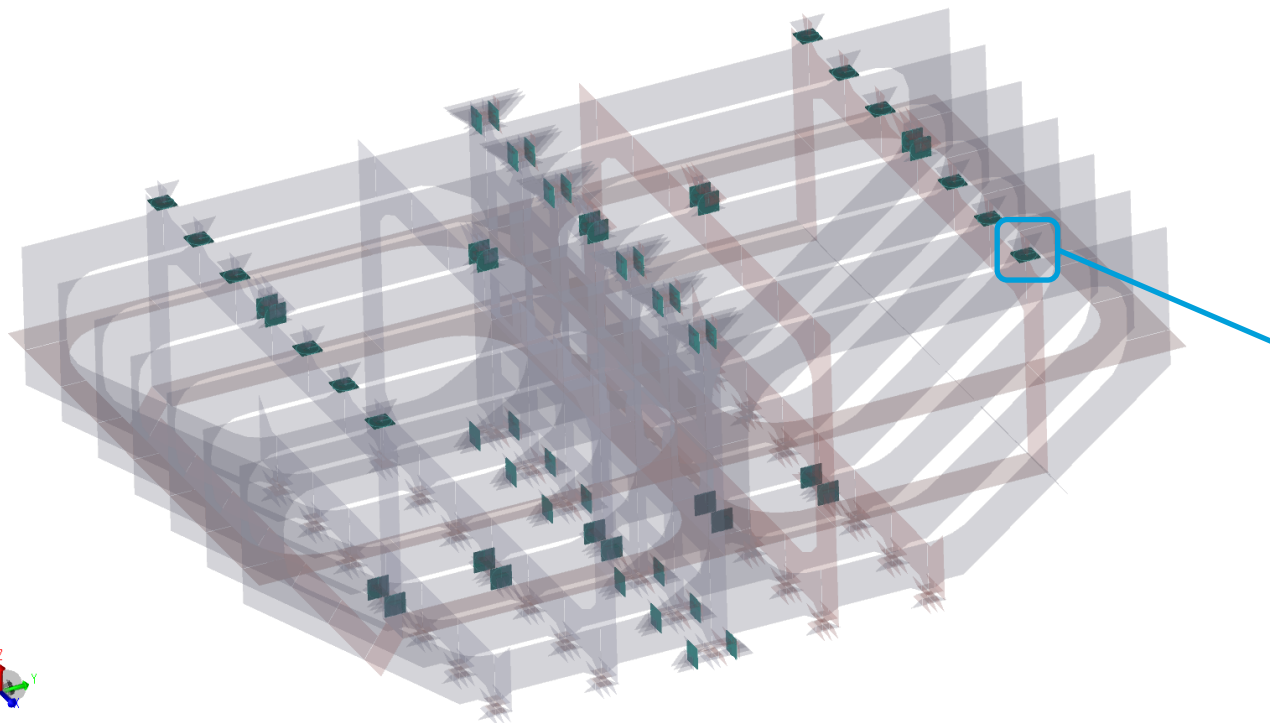


# Gas fuelled CV with type-B tank

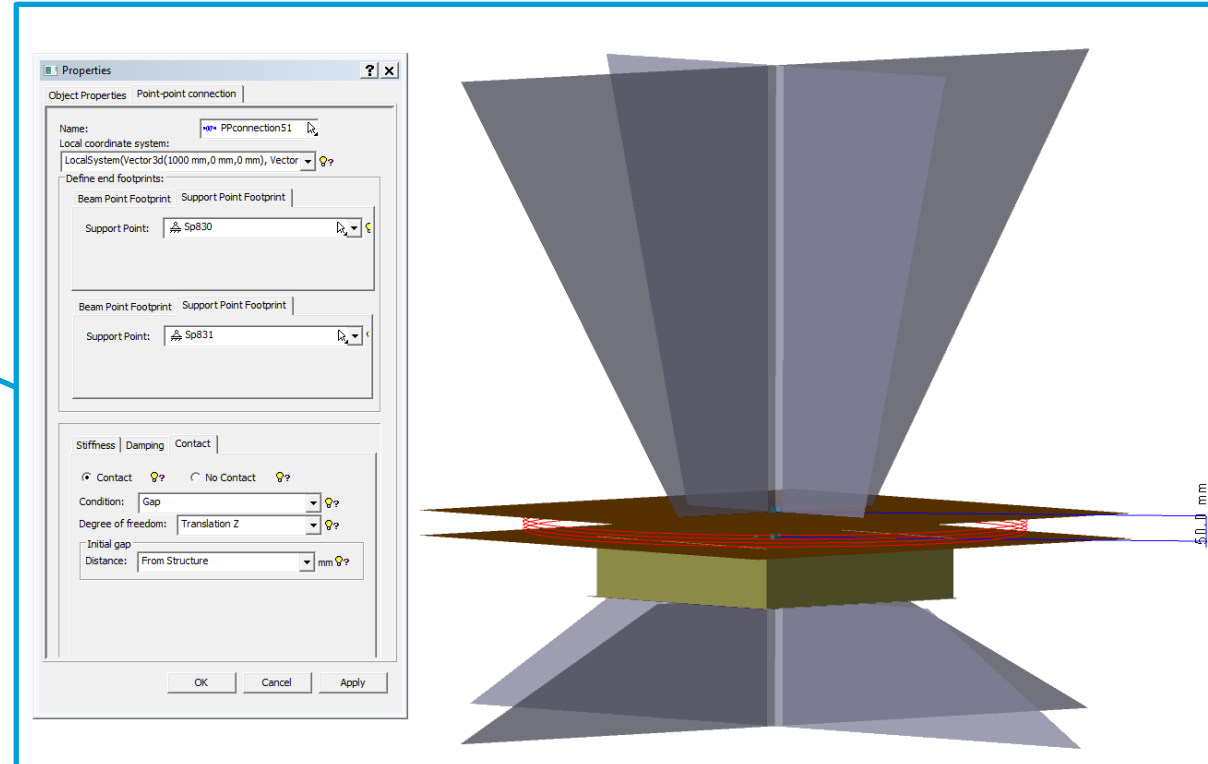
## Initial clearances independent tank supports

- Initial clearances in anti-roll, anti-pitch, and anti-floating supports are accounted for in GeniE by Point-Point connection
- In the Point-point connection dialogue, the user specifies the initial gap
- DNV's solver Sestra can consider the initial clearances when running the analysis

### Display of the Point-Point connections



### Initial gap can specified or taken from the modelled gap



# Gas fuelled CV with type-B tank

## Analysis results for 0.5g collision case – distribution of horizontal forces

- TSCF tool is creating a listing file for load transfer verification
- Sum of horizontal pressures in tank = forces taken by friction + forces in anti-pitching supports

Sum of pressures in independent tank  
**28 515 kN**

=

Forces taken in vertical supports as friction  
**13 078 kN**

+

Forces taken in anti-pitching supports  
**15 437 kN**

```
log_tscf.txt - Notepad
File Edit Format View Help

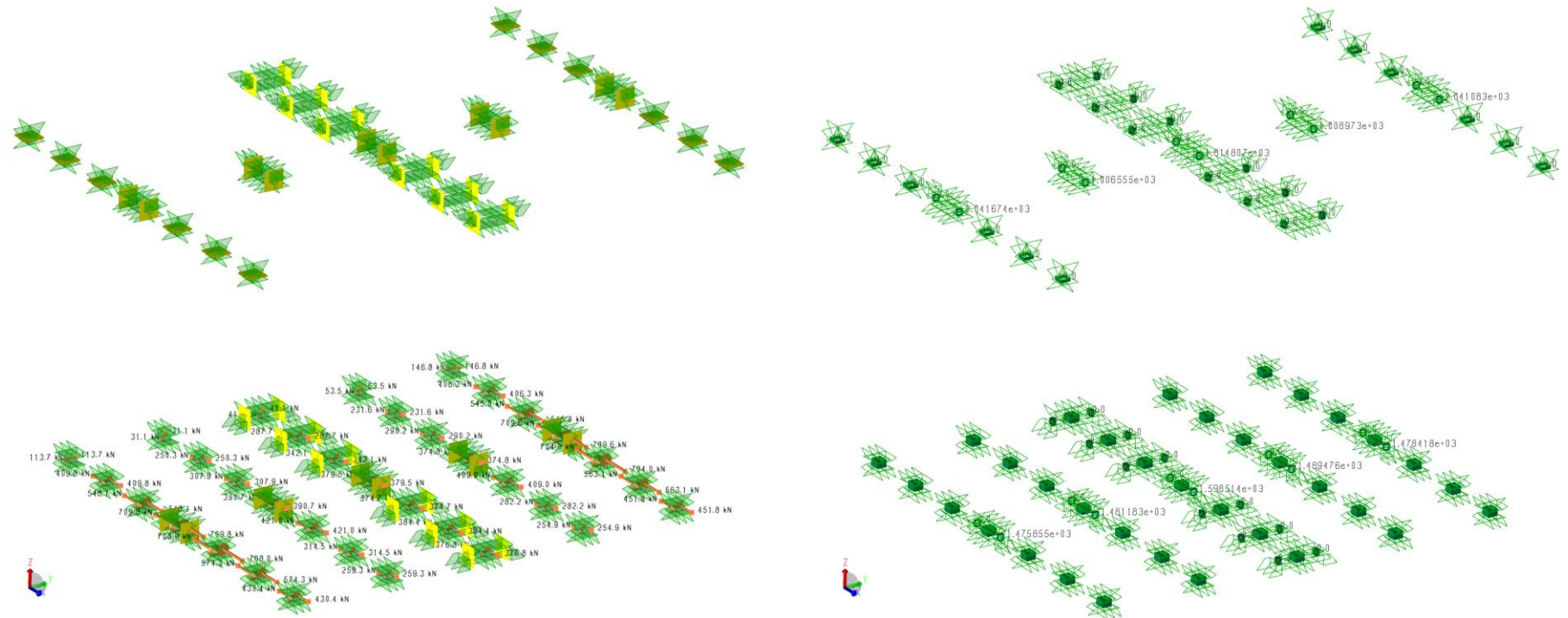
Loading combination: _008_ST3_Hog_Coll_05g

Horizontal resultant force due to tank pressures:
X: 2,85145721E+007
Y: 7,54516805E+009
XY: 2,85145721E+007

Horizontal unit vector giving direction for applied coupling forces:
X: 1,000
Y: 0,000

Vertical resultant force due to tank pressures:
Z: -5,70288485E+007

Ln 736, Col 28    100%    Windows (CRLF)    UTF-8
```

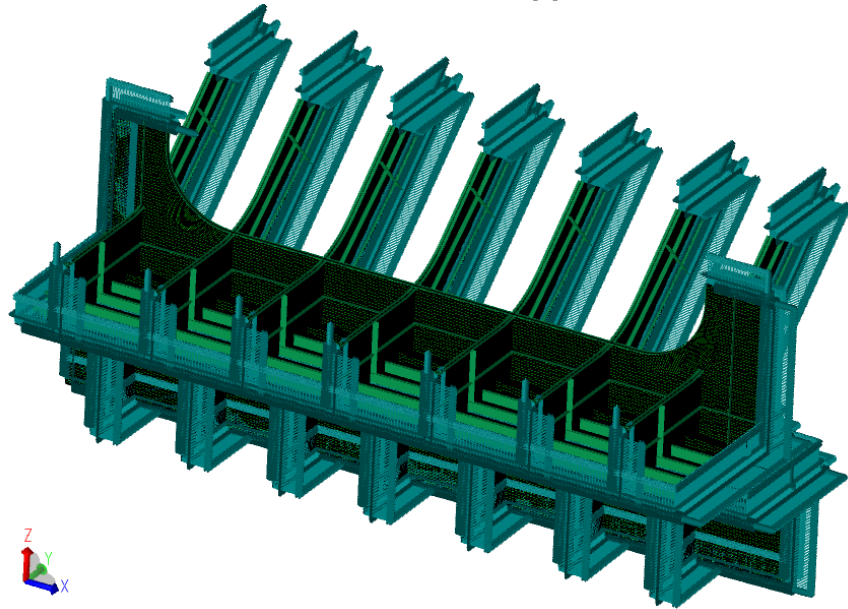


# Gas fuelled CV with type-B tank

## Support for sub-models – Local Analysis

- Sub-models can either be created in GeniE or imported via \*.bdf
- GeniE's Activity Monitor has a set-up for Local Analysis, controlling all required analysis steps in one single click
  - Automatic load transfer from the global model to the sub-model is ensured by the *Local Model Load Applier* tool (LMLA)
  - Automatic transfer of prescribed boundary conditions from the global model to the sub-model is ensured by the *Submod* tool

Sub-model of vertical supports –  
can contain more than one tank support



Required analysis steps controlled by  
GeniE's activity monitor

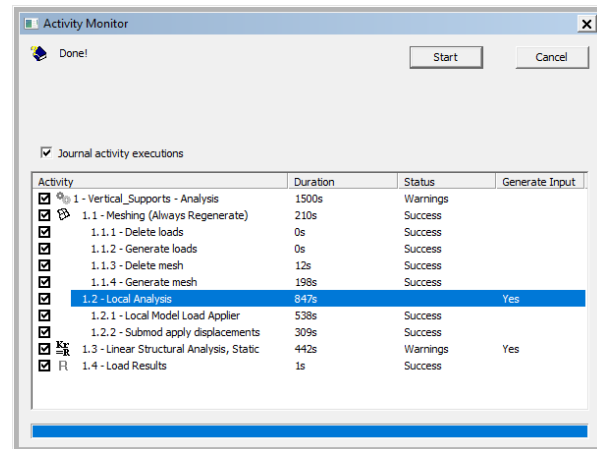
| Activity Monitor                                                             |          |              |                |
|------------------------------------------------------------------------------|----------|--------------|----------------|
| <input checked="" type="checkbox"/> Journal activity executions              |          |              |                |
| Activity                                                                     | Duration | Status       | Generate Input |
| <input checked="" type="checkbox"/> 1 - Analysis 1 - Analysis                | 37s      | Not Finished |                |
| <input checked="" type="checkbox"/> 1.1 - Meshing (Conditional Regenerate)   | 0s       | Not Finished |                |
| <input type="checkbox"/> 1.1.1 - Delete loads                                | 0s       | Not Finished |                |
| <input checked="" type="checkbox"/> 1.1.2 - Update loads                     | 0s       | Not Finished |                |
| <input type="checkbox"/> 1.1.3 - Delete mesh                                 | 0s       | Not Finished |                |
| <input checked="" type="checkbox"/> 1.1.4 - Update mesh                      | 0s       | Not Finished |                |
| <input checked="" type="checkbox"/> 1.2 - Local Analysis                     | 31s      | Success      | Yes            |
| <input checked="" type="checkbox"/> 1.2.1 - Local Model Load Applier         | 22s      | Success      |                |
| <input checked="" type="checkbox"/> 1.2.2 - Submod apply displacements       | 9s       | Success      |                |
| <input checked="" type="checkbox"/> 1.3 - Linear Structural Analysis, Static | 6s       | Warnings     | Yes            |
| <input checked="" type="checkbox"/> 1.4 - Load Results                       | 0s       | Success      |                |

- **Step 1.2.1 – Local Model Load Applier**  
distributing forces, pressures and temperature loads from cargo hold model to sub-model
- **Step 1.2.2 – Submod apply displacements**  
distributing displacements from cargo hold model to prescribed boundary conditions in sub-model
- **Step 1.3 – Linear Structural Analysis, Static**  
running Sestra solver and computing stresses and displacements

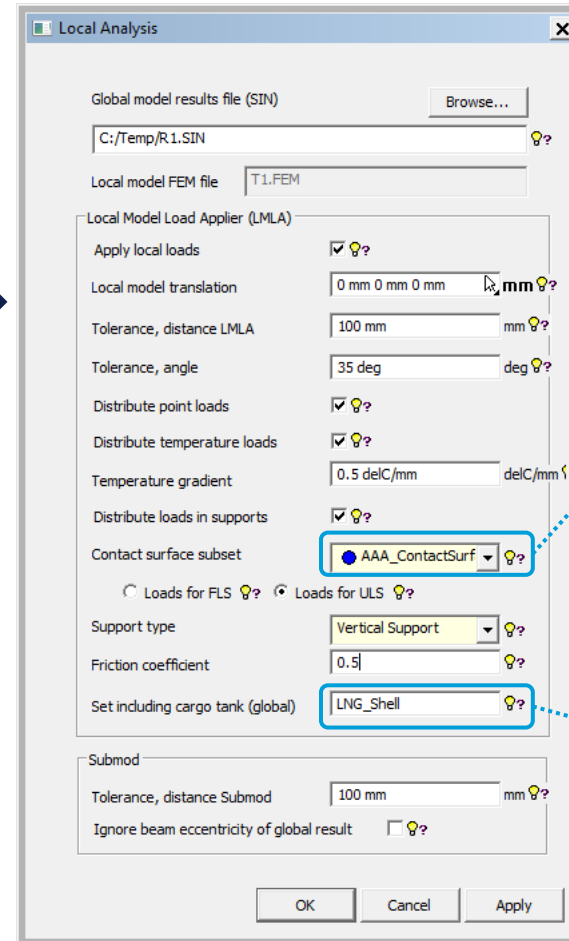
# Gas fuelled CV with type-B tank

## Support for sub-models – LMLA (Local Model Load Applier tool)

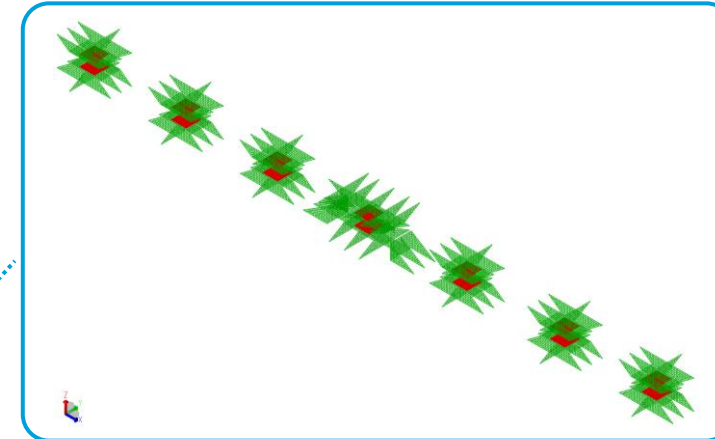
Analysis set-up in GeniE



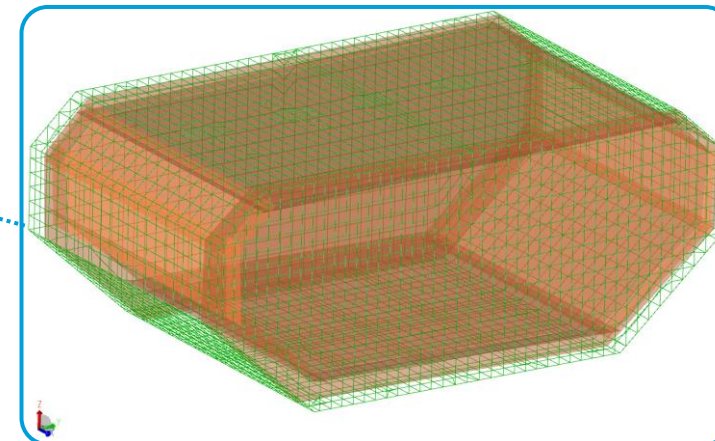
Local Analysis



Set in sub-model, containing contact area(s), for distributing pressures (lateral and traction)



Set in global model (same as for TSCF tool), containing tank boundary, for computing unit vector of horizontal pressures

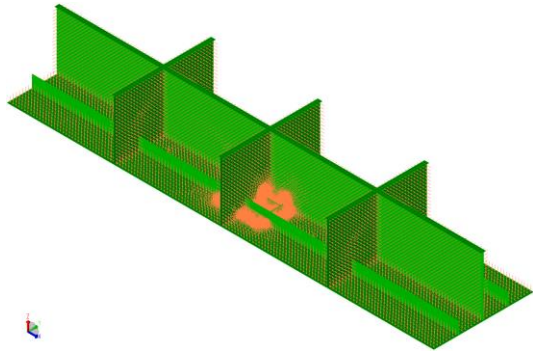


# Gas fuelled CV with type-B tank

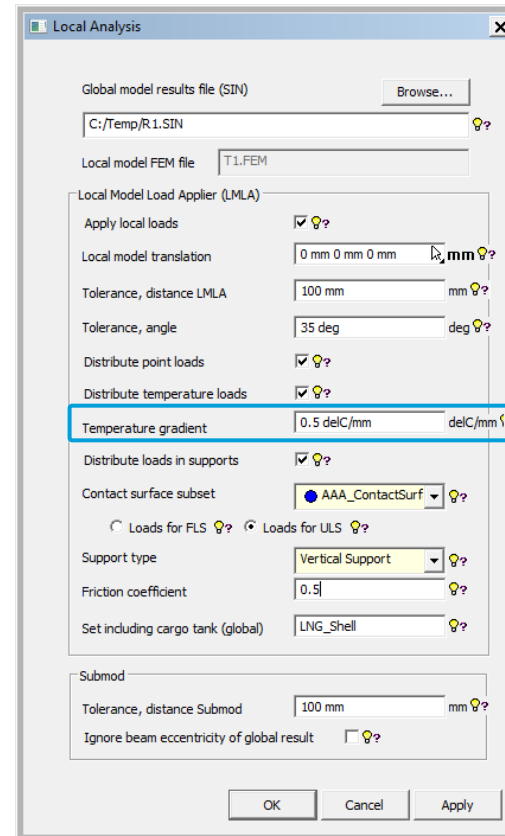
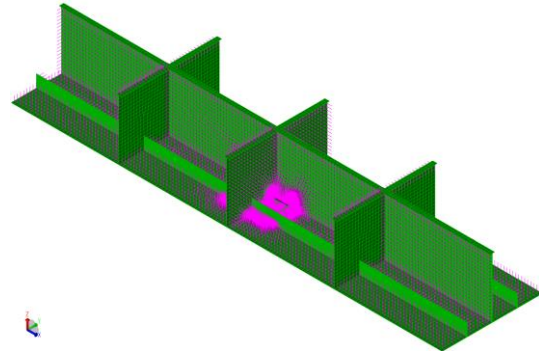
## LMLA tool – Distribution of tank loads

1. Pressures in the global model (cargo hold model with stiffener spaced mesh) will be interpolated to the sub-model (with refined mesh)
2. Temperature loads in the global model will be interpolated to the sub-model, can also be distributed to supports outside boundary

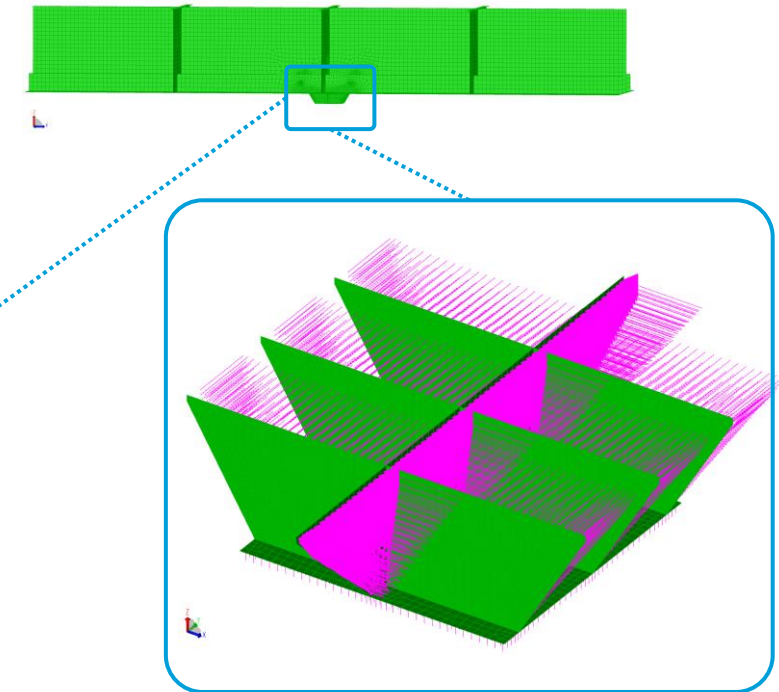
Pressures transferred from global model to sub-model



Temperature loads transferred from global model to sub-model



Temperature loads distributed to supports outside tank boundary

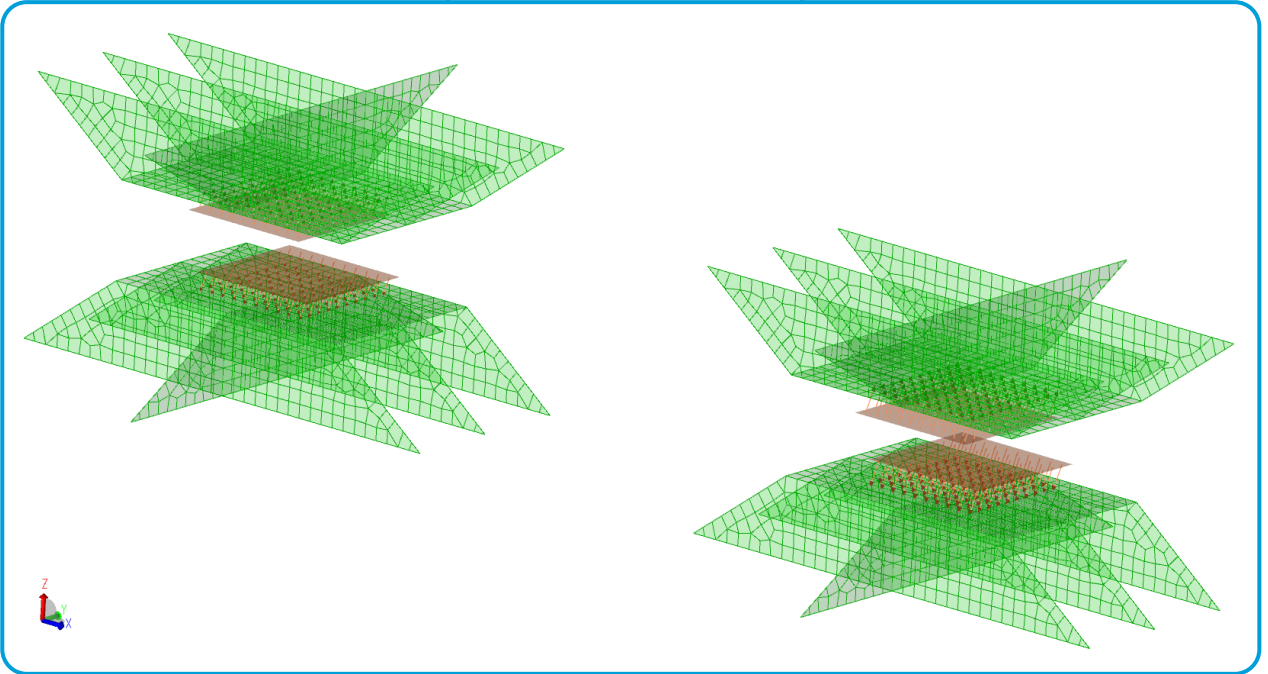


# Gas fuelled CV with type-B tank

## LMLA tool – Distribution of forces in way of tank supports

- Nodal tank support forces in the global cargo hold model will be distributed to the sub-model as pressures
- Method for distributing reaction forces and friction forces in accordance with *DNV-CG-0554*
- Robust implementation:
  - The sub-model can contain multiple contact surfaces
  - Each tank support in the global cargo hold model can be represented by more than one truss element
- All tank support types are covered
  - I. Vertical supports (ULS and FLS)
  - II. Anti-flotation supports (ULS)
  - III. Anti-roll supports (ULS); and
  - IV. Anti-pitch supports (ULS)

**Sub-model of vertical supports: Reaction forces and friction forces distributed as lateral pressures and traction pressures**



- ## FE Fatigue tool for fatigue strength calculations



## Crack propagation analysis

## Hot spot stress for fracture mechanics



# Gas fuelled CV with type-B tank

## Low cycle fatigue calculations in FE Fatigue Tool

- FE Fatigue tool also supporting low cycle fatigue
- Supports scenario where low cycle fatigue results are on a separate result file
- FE Fatigue Tool computes results for high cycle, low cycle and combined fatigue as required by the Rules

### User may include low cycle fatigue loading conditions

FE Fatigue Tool - LowerStrain.fel

File Help

**General Input**

Design basis

Design life,  $T_{90}$  25 years Time in corrosive environment,  $T_{C25}$  0 years

Stress reduction factor,  $K_p$  1

Stress magnification factor,  $K_m$  1

Number of loading and unloading cycles,  $N_{loading}$  1000

**Weld input**

Net thickness of the transverse attached plate,  $t_a$  10 mm

Welded joint category (for thickness exponent) 1: Cruciform joints, transverse T-joints, plates with transverse attachments

**Elements**

Upper and lower are according the normal vector specified in Geni

☐ Upper surface

☒ Lower surface

**Stresses**

Import data from SIN file

Specified minimum yield strength,  $R_{eH}$  235 N/mm<sup>2</sup> Thickness,  $t_{1-100}$  10.0 mm Mesh size 10.0 mm

**Results**

Calculate fatigue life time

Combined fatigue life (high and low cycle):

| Fatigue Life (years) | Damage $D_r$ | Status |
|----------------------|--------------|--------|
| 777.4                | 0.03         | OK     |

High cycle fatigue life:

| Fatigue Life (years) | Damage $D_{HCR}$ | Status |
|----------------------|------------------|--------|
| 777.4                | 0.03             | OK     |

Low cycle fatigue life:

| Fatigue Life (years) | Damage $D_{LCR}$ | Status |
|----------------------|------------------|--------|
| 0.00                 |                  |        |

Intermediate results, high cycle fatigue:

| Loading condition | HSM $\Delta\sigma_{HS}$ | HSM $f_{mean}$ | FSM $\Delta\sigma_{HS}$ | FSM $f_{mean}$ | BSR-P $\Delta\sigma_{HS}$ | BSR-P $f_{mean}$ | BSR-S $\Delta\sigma_{HS}$ | BSR-S $f_{mean}$ | BSP-P $\Delta\sigma_{HS}$ | BSP-P $f_{mean}$ | BSP-S $\Delta\sigma_{HS}$ | BSP-S $f_{mean}$ | OST-P $\Delta\sigma_{HS}$ | OST-P $f_{mean}$ | OST-S $\Delta\sigma_{HS}$ | OST-S $f_{mean}$ |
|-------------------|-------------------------|----------------|-------------------------|----------------|---------------------------|------------------|---------------------------|------------------|---------------------------|------------------|---------------------------|------------------|---------------------------|------------------|---------------------------|------------------|
| FT1, direction 1  | 12.9                    | 1.00           | 9.1                     | 1.00           | 32.5                      | 1.00             | 0.0                       | 0.00             | 37.5                      | 1.00             | 0.0                       | 0.00             | 4.8                       | 1.00             | 0.0                       | 0.00             |
| FT1, direction 2  | 2.5                     | 1.00           | 7.3                     | 1.00           | 1.3                       | 1.00             | 0.0                       | 0.00             | 3.0                       | 1.00             | 0.0                       | 0.00             | 2.2                       | 1.00             | 0.0                       | 0.00             |
| FT2, direction 1  | 7.2                     | 1.00           | 8.8                     | 1.00           | 33.3                      | 1.00             | 0.0                       | 0.00             | 33.0                      | 1.00             | 0.0                       | 0.00             | 3.5                       | 1.00             | 0.0                       | 0.00             |
| FT2, direction 2  | 0.4                     | 1.00           | 7.5                     | 1.00           | 1.3                       | 1.00             | 0.0                       | 0.00             | 2.5                       | 1.00             | 0.0                       | 0.00             | 2.7                       | 1.00             | 0.0                       | 0.00             |
| FT3, direction 1  | 1.4                     | 1.00           | 3.5                     | 1.00           | 32.5                      | 1.00             | 0.0                       | 0.00             | 29.0                      | 1.00             | 0.0                       | 0.00             | 3.1                       | 1.00             | 0.0                       | 0.00             |
| FT3, direction 2  | 0.2                     | 1.00           | 3.6                     | 1.00           | 1.2                       | 1.00             | 0.0                       | 0.00             | 0.7                       | 1.00             | 0.0                       | 0.00             | 0.8                       | 1.00             | 0.0                       | 0.00             |
| FT4, direction 1  | 0.8                     | 1.00           | 2.8                     | 1.00           | 17.0                      | 1.00             | 0.0                       | 0.00             | 14.0                      | 1.00             | 0.0                       | 0.00             | 1.8                       | 1.00             | 0.0                       | 0.00             |
| FT4, direction 2  | 0.6                     | 1.00           | 2.0                     | 1.00           | 0.7                       | 1.00             | 0.0                       | 0.00             | 0.5                       | 1.00             | 0.0                       | 0.00             | 0.6                       | 1.00             | 0.0                       | 0.00             |

Damage calculation, high cycle fatigue:

| Loading condition | $\Delta\sigma_{HS}$ | $D_r$ | $D$  |
|-------------------|---------------------|-------|------|
| FT1               | 37.5                | 0.00  | 0.07 |
| FT2               | 33.3                | 0.00  | 0.04 |
| FT3               | 32.5                | 0.00  | 0.04 |
| FT4               | 17.0                | 0.00  | 0.00 |

The fatigue stress range for high cycle fatigue,  $\Delta\sigma_{HS}$  and the combined hot spot stress ranges for low cycle fatigue,  $\Delta\sigma_{comb}$ , are calculated using the maximum of 'side L' and 'side R' instead of the average value.

Message List:

2 Warnings 0 Information

Warning, high cycle fatigue: High cycle fatigue: The stress read out point is inside the midpoint of the inner element. Read out point: 5.0, inner midpoint

Warning, low cycle fatigue: Low cycle fatigue: The stress read out point is inside the midpoint of the inner element. Read out point: 5.0, inner midpoint

### Low cycle fatigue damage with intermediate results

Hot spot location - Hotspot 1

Other welded details - type 'a'

**Input**

Design basis

Design life,  $T_{90}$  25 years Time in corrosive environment,  $T_{C25}$  0 years

Stress reduction factor,  $K_p$  1

Stress magnification factor,  $K_m$  1

Number of loading and unloading cycles,  $N_{loading}$  1000

**Weld input**

Net thickness of the transverse attached plate,  $t_a$  10 mm

Welded joint category (for thickness exponent) 1: Cruciform joints, transverse T-joints, plates with transverse attachments

**Elements**

Upper and lower are according the normal vector specified in Geni

☐ Upper surface

☒ Lower surface

**Stresses**

Import data from SIN file

Specified minimum yield strength,  $R_{eH}$  235 N/mm<sup>2</sup> Thickness,  $t_{1-100}$  10.0 mm Mesh size 10.0 mm

**Results**

Calculate fatigue life time

Combined fatigue life (high and low cycle):

| Fatigue Life (years) | Damage $D_r$ | Status |
|----------------------|--------------|--------|
| 777.4                | 0.03         | OK     |

High cycle fatigue life:

| Fatigue Life (years) | Damage $D_{HCR}$ | Status |
|----------------------|------------------|--------|
| 777.4                | 0.03             | OK     |

Low cycle fatigue life:

| Fatigue Life (years) | Damage $D_{LCR}$ | Status |
|----------------------|------------------|--------|
| 0.00                 |                  |        |

Intermediate results, high cycle fatigue:

| Loading condition | HSM $\Delta\sigma_{HS}$ | HSM $f_{mean}$ | FSM $\Delta\sigma_{HS}$ | FSM $f_{mean}$ | BSR-P $\Delta\sigma_{HS}$ | BSR-P $f_{mean}$ | BSR-S $\Delta\sigma_{HS}$ | BSR-S $f_{mean}$ | BSP-P $\Delta\sigma_{HS}$ | BSP-P $f_{mean}$ | BSP-S $\Delta\sigma_{HS}$ | BSP-S $f_{mean}$ | OST-P $\Delta\sigma_{HS}$ | OST-P $f_{mean}$ | OST-S $\Delta\sigma_{HS}$ | OST-S $f_{mean}$ |
|-------------------|-------------------------|----------------|-------------------------|----------------|---------------------------|------------------|---------------------------|------------------|---------------------------|------------------|---------------------------|------------------|---------------------------|------------------|---------------------------|------------------|
| FT1, direction 1  | 12.9                    | 1.00           | 9.1                     | 1.00           | 32.5                      | 1.00             | 0.0                       | 0.00             | 37.5                      | 1.00             | 0.0                       | 0.00             | 4.8                       | 1.00             | 0.0                       | 0.00             |
| FT1, direction 2  | 2.5                     | 1.00           | 7.3                     | 1.00           | 1.3                       | 1.00             | 0.0                       | 0.00             | 3.0                       | 1.00             | 0.0                       | 0.00             | 2.2                       | 1.00             | 0.0                       | 0.00             |
| FT2, direction 1  | 7.2                     | 1.00           | 8.8                     | 1.00           | 33.3                      | 1.00             | 0.0                       | 0.00             | 33.0                      | 1.00             | 0.0                       | 0.00             | 3.5                       | 1.00             | 0.0                       | 0.00             |
| FT2, direction 2  | 0.4                     | 1.00           | 7.5                     | 1.00           | 1.3                       | 1.00             | 0.0                       | 0.00             | 2.5                       | 1.00             | 0.0                       | 0.00             | 2.7                       | 1.00             | 0.0                       | 0.00             |
| FT3, direction 1  | 1.4                     | 1.00           | 3.5                     | 1.00           | 32.5                      | 1.00             | 0.0                       | 0.00             | 29.0                      | 1.00             | 0.0                       | 0.00             | 3.1                       | 1.00             | 0.0                       | 0.00             |
| FT3, direction 2  | 0.2                     | 1.00           | 3.6                     | 1.00           | 1.2                       | 1.00             | 0.0                       | 0.00             | 0.7                       | 1.00             | 0.0                       | 0.00             | 0.8                       | 1.00             | 0.0                       | 0.00             |
| FT4, direction 1  | 0.8                     | 1.00           | 2.8                     | 1.00           | 17.0                      | 1.00             | 0.0                       | 0.00             | 14.0                      | 1.00             | 0.0                       | 0.00             | 1.8                       | 1.00             | 0.0                       | 0.00             |
| FT4, direction 2  | 0.6                     | 1.00           | 2.0                     | 1.00           | 0.7                       | 1.00             | 0.0                       | 0.00             | 0.5                       | 1.00             | 0.0                       | 0.00             | 0.6                       | 1.00             | 0.0                       | 0.00             |

Intermediate results, low cycle fatigue:

| Name        | Static | Static $\Delta\sigma_{comb}$ | Static $\lambda_n$ |
|-------------|--------|------------------------------|--------------------|
| Direction 1 | 138.2  | 1.00                         |                    |

Damage calculation, high cycle fatigue:

| Loading condition | $\Delta\sigma_{HS}$ | $D_r$ | $D$  |
|-------------------|---------------------|-------|------|
| FT1               | 37.5                | 0.00  | 0.07 |
| FT2               | 33.3                | 0.00  | 0.04 |
| FT3               | 32.5                | 0.00  | 0.04 |
| FT4               | 17.0                | 0.00  | 0.00 |

The fatigue stress range for high cycle fatigue,  $\Delta\sigma_{HS}$  and the combined hot spot stress ranges for low cycle fatigue,  $\Delta\sigma_{comb}$ , are calculated using the maximum of 'side L' and 'side R' instead of the average value.

Message List:

2 Warnings 0 Information

Warning, high cycle fatigue: High cycle fatigue: The stress read out point is inside the midpoint of the inner element. Read out point: 5.0, inner midpoint

Warning, low cycle fatigue: Low cycle fatigue: The stress read out point is inside the midpoint of the inner element. Read out point: 5.0, inner midpoint

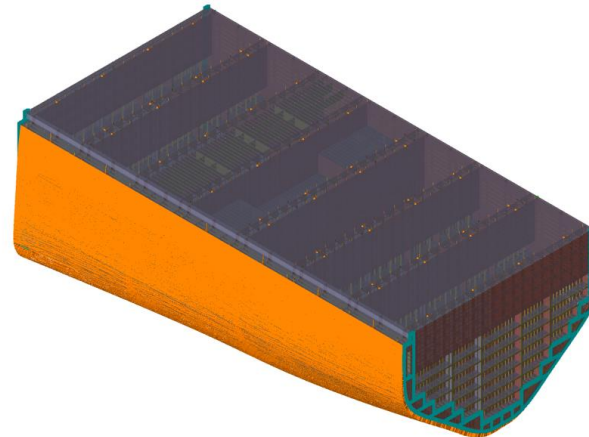
*A tutorial has been made available to external customers describing how to efficiently use GeniE and Nauticus Hull for gas fuelled container ships with independent prismatic tank*

## Tutorial model files available for download



## Tutorial – gas fuel tank finite element analysis

### Container ship model with Type-B gas fuel tank arrangement



DNV – Gas fuelled container ships tutorial

Page 1

The screenshot shows a web browser window with the address bar displaying "https://sesam.dnv.com/download/nauticushull/cargoholdmodel.mpd". The page content features a large 3D visualization of a ship's hull cross-section, specifically focusing on the cargo hold area. The model is composed of numerous green rectangular blocks representing containers, arranged in a grid-like pattern. A red dashed line runs diagonally across the container arrangement, likely indicating a structural or loading axis. To the left of the 3D view, there is a detailed tree structure listing various components and their properties, such as "Container", "Cargo", "Hull", and "Structural". At the bottom of the browser window, a console or log area displays technical messages related to the simulation process.

# Container ship global model

GeniE integrated with ShipLoad

# Container ship global model

## GeniE integrated with ShipLoad

GeniE has an interface to ShipLoad for automatic creation of loading conditions, and can run automatic buckling check

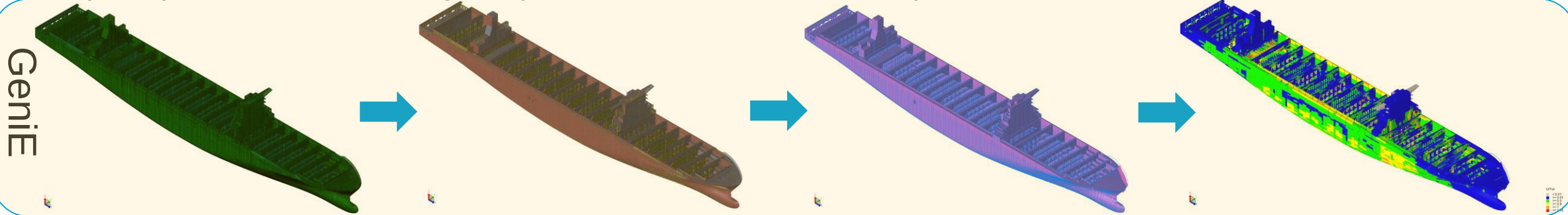
- Work process for running FE Rule checks for global container ship models in GeniE are described below
- The model can either be imported from \*.bdf, or made from scratch in GeniE
- GeniE has an interface to ShipLoad, a DNV tool providing loading conditions according to Rules

Finite element model imported to GeniE (T1.FEM)

Elements stitched together to geometry

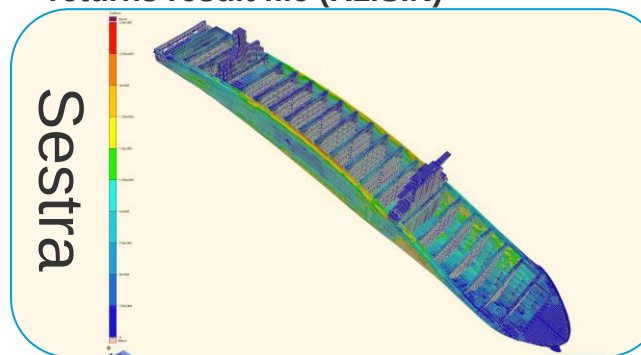
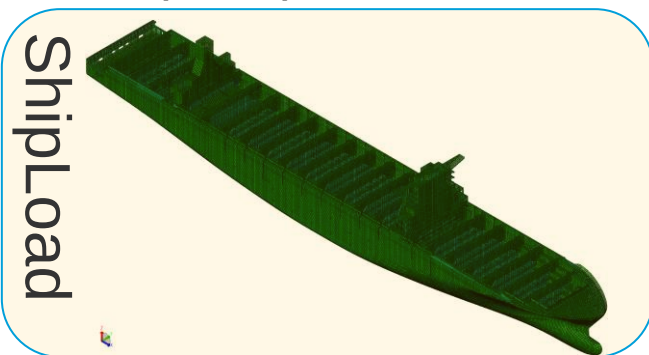
Buckling panels automatically created

GeniE FE Rule checks



ShipLoad creates finite element load file (L1.FEM)

Solver reads T1.FEM + L1.FEM and returns result file (R1.SIN)



# FE Rule checks for CSR BC&OT

Yield screening – All details required covered

Yield screening – Analysis

Fine mesh yield check - Analysis

Fine mesh iwo openings

Fatigue screening using FE Fatigue tool

Automatic mesh refinement – Cargo hold model

Automatic mesh refinement – Sub-model

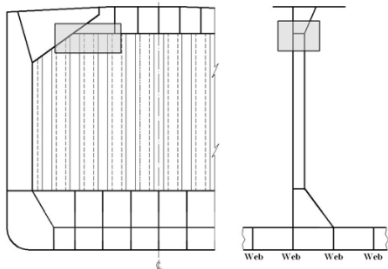
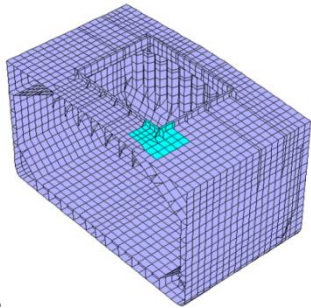
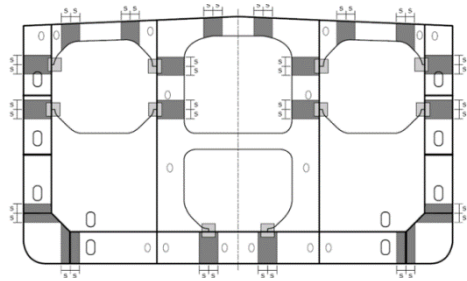
FE Fatigue assessment – Direct access to FE Fatigue tool

# FE Rule checks for CSR BC&OT

## Yield screening – All details required covered

- The extensive screening requirements are implemented in GeniE

### Areas to be yield screened



### Permissible screening factors depending on area

| Type of Details                                                                                                                                                                                                                                                                                                                                                                                                         | Screening factors, $\lambda_{sc}$                         | Permissible screening factors, $\lambda_{scperm}$ |            |
|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|-----------------------------------------------------------|---------------------------------------------------|------------|
| Within the whole cargo hold region                                                                                                                                                                                                                                                                                                                                                                                      |                                                           | S+D                                               | S          |
| Openings for which their geometry is not required to be represented in the cargo hold model in accordance with Ch 7, Sec 2, [2.4.9] in way of webs of primary supporting members, such as transverse web frame as indicated in Table 1 and Table 2, horizontal stringers as indicated in Table 3, floors and longitudinal girders in double bottom.                                                                     | Table 5                                                   | 1.70                                              | 1.36       |
| Manholes <sup>(2)</sup>                                                                                                                                                                                                                                                                                                                                                                                                 | $\lambda_y$                                               | $0.85 \lambda_{yperm}$                            |            |
| Bracket toes on transverse web frames as indicated in Table 1 and Table 2, horizontal stringers and transverse plane bulkhead to double bottom connection or buttress structure specified in Table 3.                                                                                                                                                                                                                   | Table 6                                                   | 1.50                                              | 1.20       |
| Heels of transverse bulkhead horizontal stringers specified in Table 3.                                                                                                                                                                                                                                                                                                                                                 | Table 7                                                   | 1.50                                              | 1.20       |
| Connections of transverse lower stool to double bottom girders and longitudinal lower stool to double bottom floors as indicated in Figure 5. The connection of lower hopper to transverse lower stool structure as indicated in Figure 5. The connection of topside tank to inner side as indicated in Figure 6. The connection of corrugation and upper supporting structure to upper stool as indicated in Figure 7. | $\lambda_y$                                               | $0.75 \lambda_{yperm}$                            |            |
| Hatch corner area.                                                                                                                                                                                                                                                                                                                                                                                                      | $\lambda_y$                                               | $0.95 \lambda_{yperm}$                            |            |
| Outside midship cargo hold region                                                                                                                                                                                                                                                                                                                                                                                       |                                                           |                                                   |            |
| Hopper knuckle                                                                                                                                                                                                                                                                                                                                                                                                          | $\lambda_y$                                               | $0.65 \lambda_{yperm}$                            |            |
| Side frame end bracket                                                                                                                                                                                                                                                                                                                                                                                                  |                                                           | $0.85 \lambda_{yperm}$                            |            |
| Large openings <sup>(2)</sup>                                                                                                                                                                                                                                                                                                                                                                                           |                                                           | $0.85 \lambda_{yperm}$                            |            |
| Connections of corrugation to adjoining structure                                                                                                                                                                                                                                                                                                                                                                       | $\lambda_{so} = \frac{K_{so} \cdot \sigma_o}{R_y}$<br>(1) | $1.50 f_f$                                        | $1.20 f_f$ |

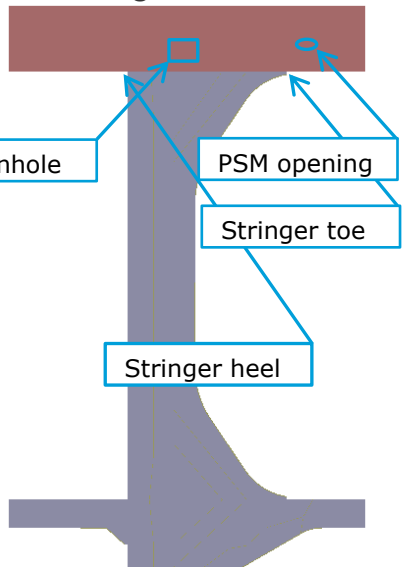
### All screening requirements implemented in GeniE

# FE Rule checks for CSR BC&OT

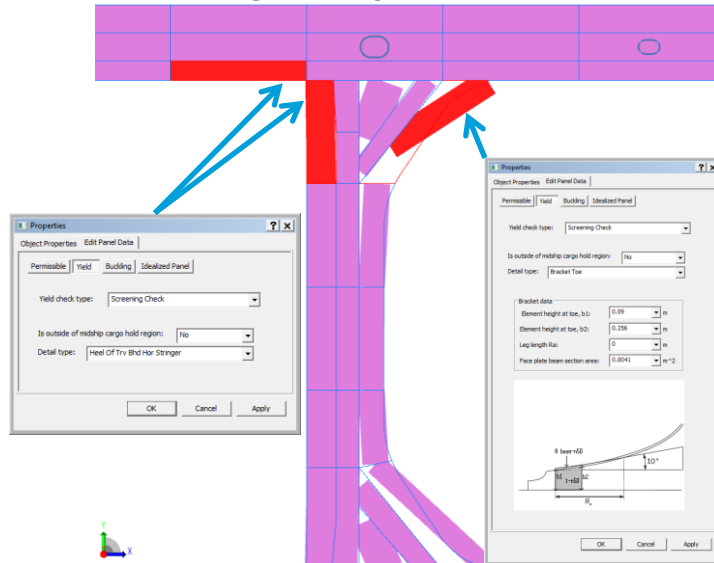
## Yield screening – Analysis

- The workflow for yield screening is similar to cargo hold Rule checks
  - Create geometry in GeniE, or import from \*.bdf
  - Import loading conditions from Nauticus Hull
  - Meshing, including adjusting hull girder loads (HGLA)
  - Give Rule check input properties
  - Run analysis in the Activity Monitor
  - Display results graphically and, when needed, detailed results

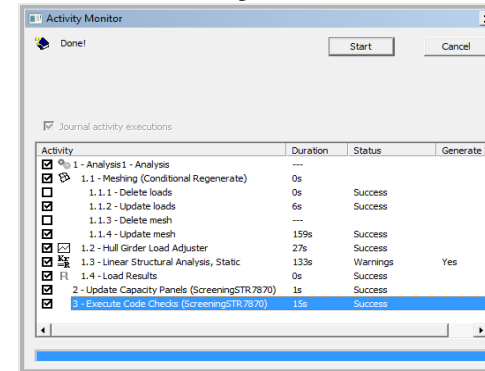
### Modelled STR geometry on a cargo hold model



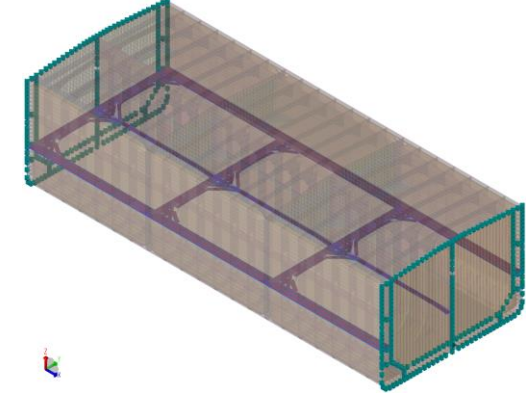
### Screening Rule check input based on modelled geometry



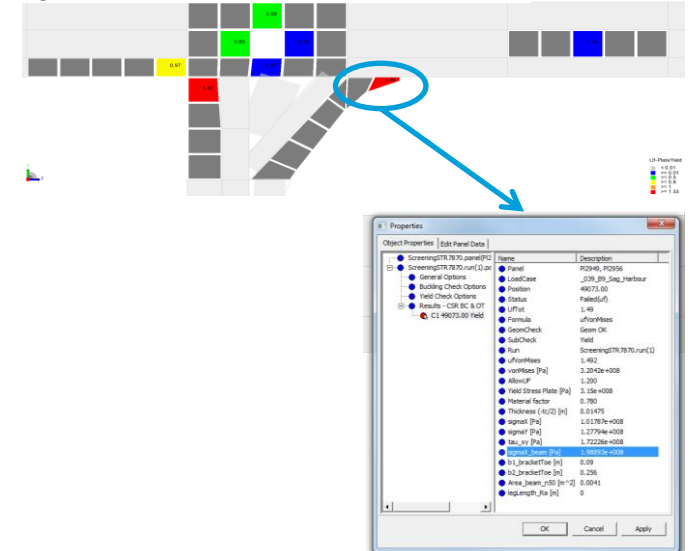
### Analysis steps controlled by GeniE's activity monitor



### Yield screening of a stringer



### Several FEA Rule check result options are available

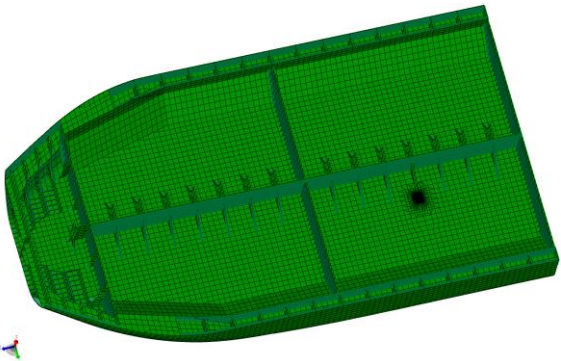


# FE Rule checks for CSR BC&OT

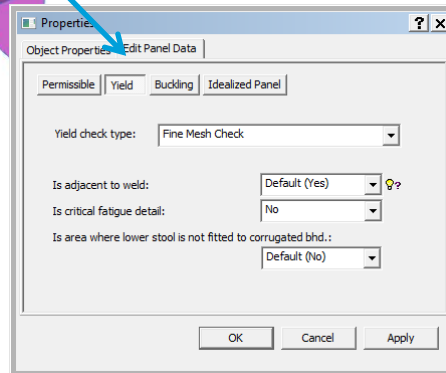
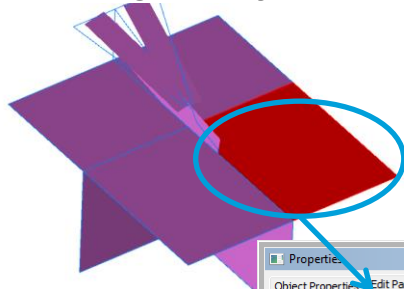
## Fine mesh yield check - Analysis

- The workflow for fine mesh yield check is similar to cargo hold Rule checks
  - Create geometry in GeniE, or import from \*.bdf
  - Import loading conditions from Nauticus Hull
  - Meshing, including adjusting hull girder loads (HGLA)
  - Give Rule check input properties
  - Run analysis in the Activity Monitor
  - Display results graphically and, when needed, detailed results

Cargo hold FE model with mesh refinement

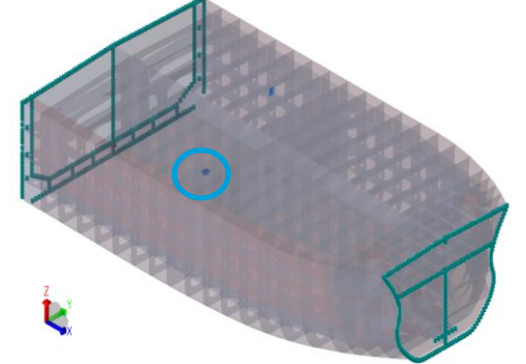


Fine mesh yield check input based on modelled geometry

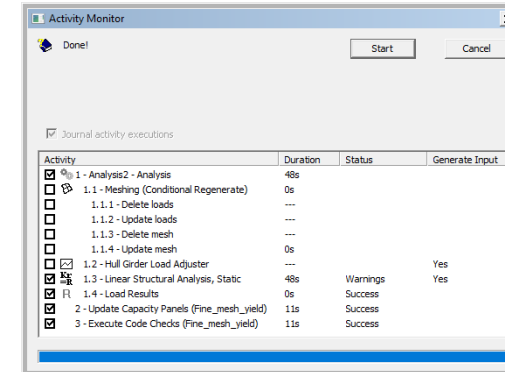


GeniE's implementation of fine mesh yield checks are based on the same efficient user experiences as for cargo hold models

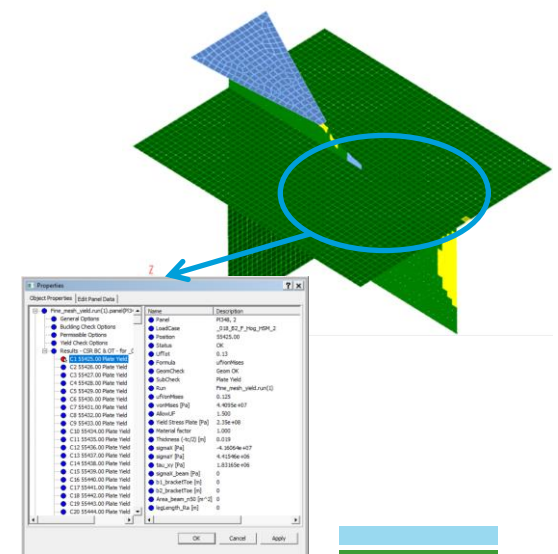
Yield screening of a web frame toe



Analysis steps controlled by GeniE's activity monitor



Several FEA Rule check result options are available

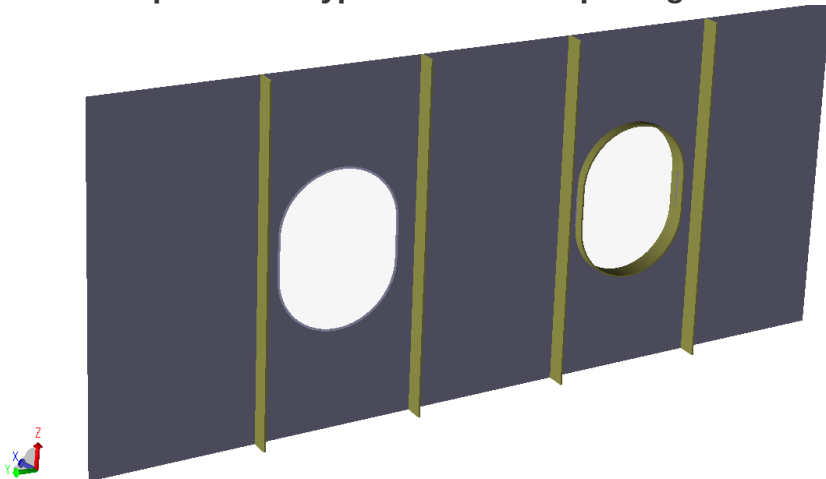


# FE Rule checks for CSR BC&OT

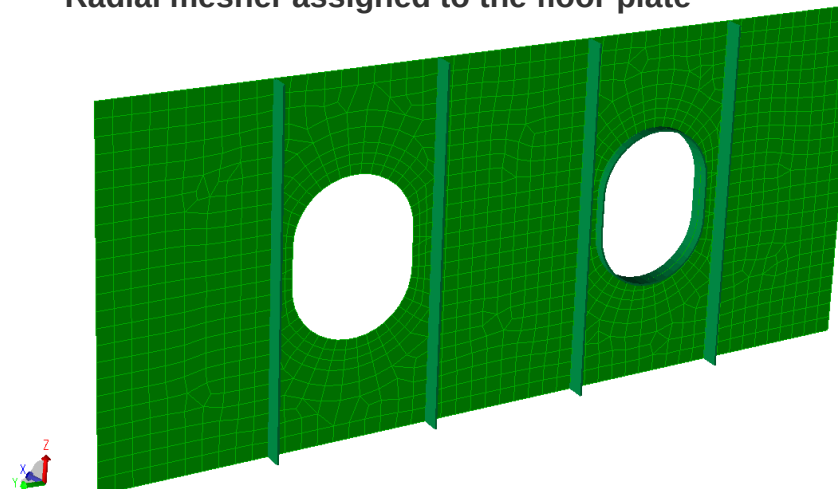
## Fine mesh in way of openings

- CSR BC&OT requires fine mesh around openings, with mesh following the opening's curvature
- GeniE has a radial mesher available where to user can specify up to 4 radial mesh layers
- The radial mesher can be assigned to
  - A hole concept; or
  - A plate where the hole is modelled
- The radial mesher also ensures radial mesh for openings with various edge stiffening

Floor plate with typical manhole openings



Radial mesher assigned to the floor plate



# FE Rule checks for CSR BC&OT

## Fatigue screening using FE Fatigue tool

- CSR BC&OT allows for fatigue screening for typical details as an alternative to very fine mesh FE models
- Fatigue screening by using stresses from the 50x50 mm mesh size is supported by the FE Fatigue tool
- Details required in the Rules
  - Bracket toe; and
  - Hopper knuckle

### FE Fatigue tool with fatigue screening implemented

FE Fatigue Tool - FEFatigue1.nhfe

File Help

**General Input**

Rules: CSR BC & OT

Ship rule length, L: 216.25 (m)

**Loading conditions:**

Name: Fraction of time:

Homogeneous: 0.5

NormalBallast: 0.5

File (.sin): C:\Data\2ProjectFiles\1Vessels\NH20\Tanker73k\FEAnalyses\No4Fatigue\Analysis1\20160513\_093611\_R1.SIN

Reload

**Load cases:**

| Name:         | HSM-1              | HSM-2              | FSM-1              | FSM-2              | BSR-P1              | BSR-P2              | BSR-S1              |
|---------------|--------------------|--------------------|--------------------|--------------------|---------------------|---------------------|---------------------|
| Homogeneous   | 001_B1_F_Sag_HSM_1 | 002_B1_F_Sag_HSM_2 | 003_B1_F_Sag_FSM_1 | 004_B1_F_Sag_FSM_2 | 005_B1_F_Sag_BSR_1P | 006_B1_F_Sag_BSR_2P | 007_B1_F_Sag_BSR_1S |
| NormalBallast | 017_B2_F_Hog_HSM_1 | 018_B2_F_Hog_HSM_2 | 019_B2_F_Hog_FSM_1 | 020_B2_F_Hog_FSM_2 | 021_B2_F_Hog_BSR_1P | 022_B2_F_Hog_BSR_2P | 023_B2_F_Hog_BSR_1S |

**Hot Spots:**

Import GenIE xml file: ...

Length unit: mm

Stress unit: MPa

| Name: | Description: | Procedure for calculation of hot spot stress: |
|-------|--------------|-----------------------------------------------|
| 0     |              | ScreeningBracketToe                           |
| 1     |              | ScreeningKnuckleDetail                        |

Open Add new Hot Spot Delete

- CruciformFlange
- CruciformWeb
- Type 'a'
- Type 'a' - hot spot stress for crack propagation analysis
- Type 'a' - hot spot stress for fracture mechanics analysis
- Free edge
- Screening - bracket toe
- Screening - knuckle detail

### Fatigue screening of a hopper knuckle

Hot spot location - 1

**Fatigue screening - knuckle detail**

**Input**

**Design basis**

Design life, Td: 25 years Time in corrosive environment, Tc: 5 years

**Structural detail category**

☒ Lower hopper welded knuckle

☐ Lower stool - inner bottom - non vertical (knuckle angle > 90°)

☐ Lower stool - inner bottom - vertical (knuckle angle = 90°)

**Cargo hold**

☒ FA (full cargo hold in alternate loading condition)

☐ EA (empty cargo hold in alternate loading condition) or C (cargo hold BC-B and BC-C)

**Weld input**

Extended fillet weld length, Xwet: 10 mm

Net thickness of the transverse attached plate, tznso: 15 mm

**Stresses**

**Results**

Calculate fatigue life time

Fatigue life:

| Fatigue Life [years] | Damage | Status |
|----------------------|--------|--------|
| 57.3                 | 0.26   | OK     |

**Intermediate results:**

| Loading condition          | HSM               | HSM    | FSM               | FSM    | BSR-P             | BSR-P  | BSR-S             | BSR-S  | BSP-P             | BSP-P  | BSP-S             | BSP-S  | OST-P             | OST-P  | OST-S             | OST-S  |
|----------------------------|-------------------|--------|-------------------|--------|-------------------|--------|-------------------|--------|-------------------|--------|-------------------|--------|-------------------|--------|-------------------|--------|
|                            | $\Delta\sigma$ HS | f mean | $\Delta\sigma$ HS | f mean | $\Delta\sigma$ HS | f mean | $\Delta\sigma$ HS | f mean | $\Delta\sigma$ HS | f mean | $\Delta\sigma$ HS | f mean | $\Delta\sigma$ HS | f mean | $\Delta\sigma$ HS | f mean |
| Homogeneous, direction 1   | 44.6              | 1.00   | 43.4              | 1.00   | 11.3              | 1.00   | 6.2               | 1.00   | 39.8              | 1.00   | 15.5              | 1.00   | 42.3              | 1.00   | 12.1              | 1.00   |
| Homogeneous, direction 2   | 73.7              | 1.00   | 69.1              | 1.00   | 4.9               | 1.00   | 15.9              | 1.00   | 39.8              | 1.00   | 47.4              | 1.00   | 73.6              | 1.00   | 16.1              | 1.00   |
| NormalBallast, direction 1 | 21.7              | 0.57   | 21.7              | 0.56   | 4.9               | 0.30   | 3.4               | 0.30   | 32.3              | 0.57   | 20.2              | 0.54   | 19.2              | 0.47   | 2.3               | 0.30   |
| NormalBallast, direction 2 | 71.5              | 0.35   | 63.2              | 0.30   | 16.8              | 0.30   | 14.0              | 0.30   | 5.1               | 0.30   | 12.5              | 0.30   | 50.6              | 0.30   | 12.5              | 0.30   |

**Damage calculation:**

| Loading condition | $\Delta\sigma$ FS | D E,air | D E,corr | D    |
|-------------------|-------------------|---------|----------|------|
| Homogeneous       | 57.4              | 0.18    | 0.55     | 0.26 |
| NormalBallast     | 19.5              | 0.00    | 0.02     | 0.01 |

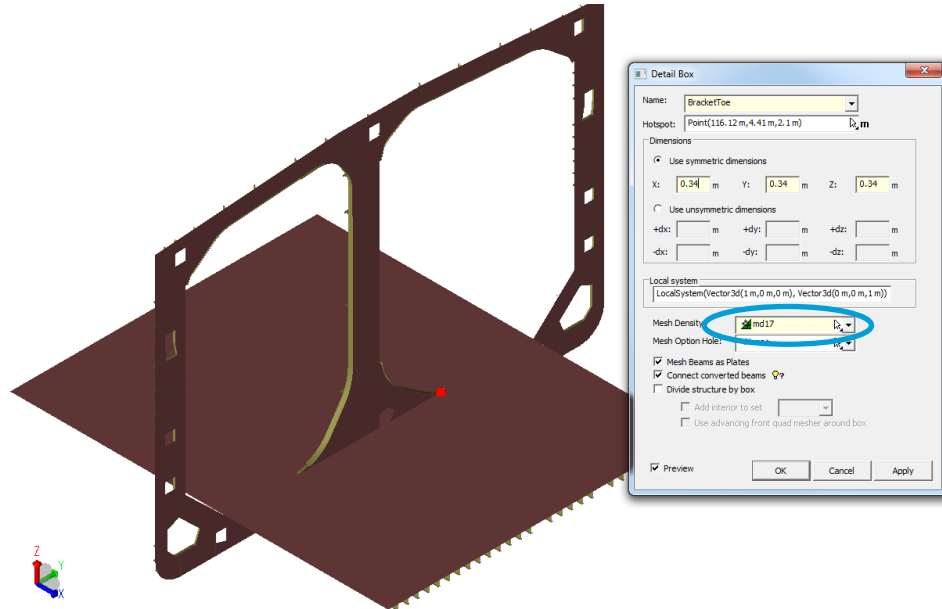
# FE Rule checks for CSR BC&OT

GeniE has effective tools for automatic mesh refinement, providing quadratic elements with size according to Rules

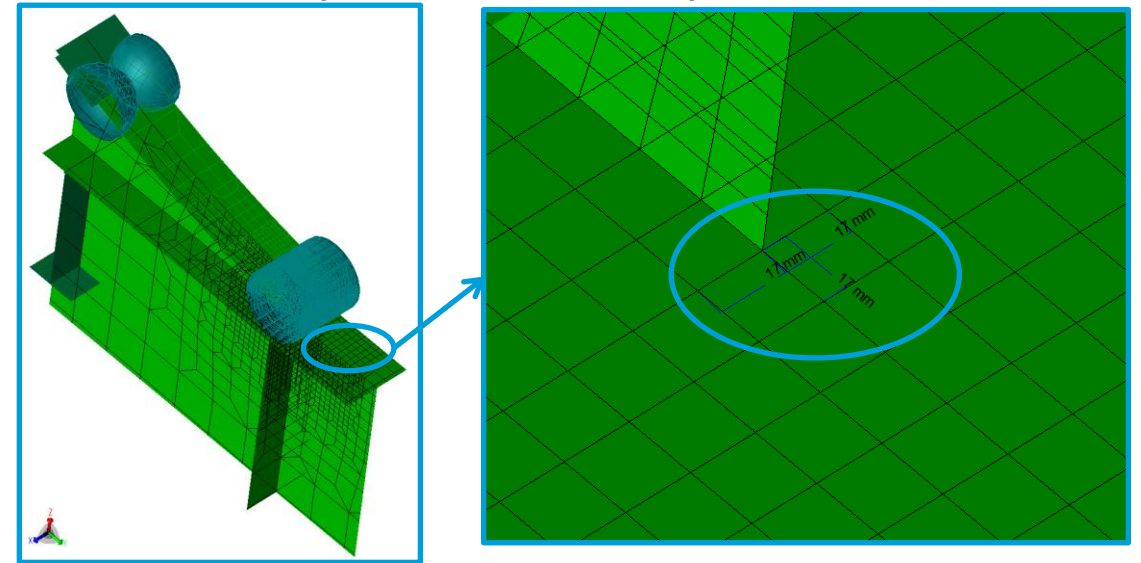
## Automatic mesh refinement - Cargo hold model

- Within a cargo hold model, the user can specify structural details where the mesh shall be refined
- The detailed box feature can also automatically convert stiffeners to shell elements
- Typical details
  - Hopper knuckles; and
  - Toes of stringers / web frames
- Auto fine mesh by use of detail box is a post-meshing operation, i.e. GeniE first run coarse meshing and then refine the mesh

### Automatic mesh refinement of a web frame toe



### Mesh refined with specified mesh size and quadratic elements

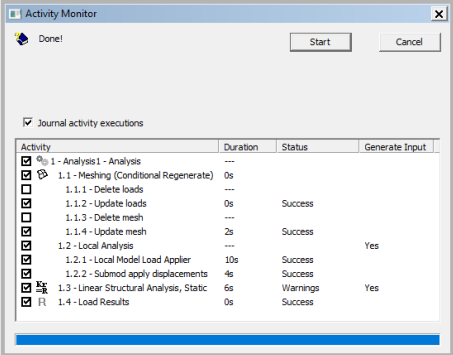


# FE Rule checks for CSR BC&OT

## Automatic mesh refinement – Sub-model

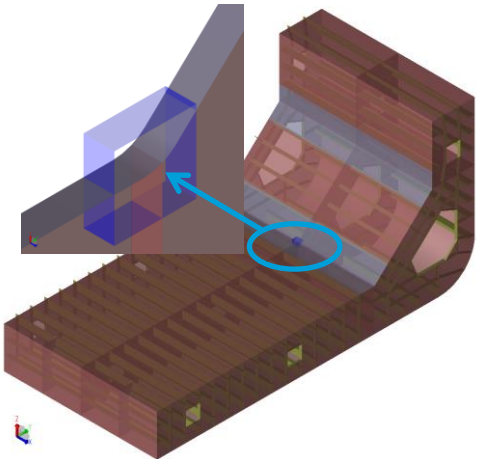
- The mesh refinement methods can also be used for a sub-model
- The local analysis in GeniE, using sub-models, have the following benefits
  - Automatic transfer of local pressure loads from cargo hold model to sub-model
  - Automatic transfer of prescribed displacements from cargo hold model to sub-model
  - No need to create compartments in the sub-model
  - Fast analysis performance
- The local model can be imported from other systems using the \*bdf format

### Analysis steps controlled by GeniE's activity monitor

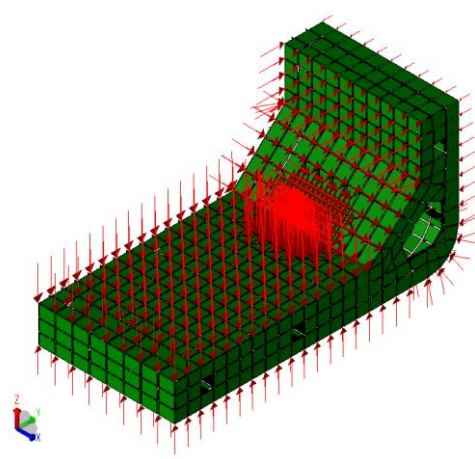


| Activity                                                                     | Duration | Status   | Generate Input |
|------------------------------------------------------------------------------|----------|----------|----------------|
| <input checked="" type="checkbox"/> 1 - Analysis1 - Analysis                 | ---      |          |                |
| <input checked="" type="checkbox"/> 1.1 - Meshing (Conditional Regenerate)   | 0s       |          |                |
| <input checked="" type="checkbox"/> 1.1.1 - Delete loads                     | ---      |          |                |
| <input checked="" type="checkbox"/> 1.1.2 - Update loads                     | 0s       | Success  |                |
| <input checked="" type="checkbox"/> 1.1.3 - Delete mesh                      | ---      |          |                |
| <input checked="" type="checkbox"/> 1.1.4 - Update mesh                      | 2s       | Success  |                |
| <input checked="" type="checkbox"/> 1.2 - Local Analysis                     | ---      |          | Yes            |
| <input checked="" type="checkbox"/> 1.2.1 - Local Model Load Applier         | 10s      | Success  |                |
| <input checked="" type="checkbox"/> 1.2.2 - Submod apply displacements       | 4s       | Success  |                |
| <input checked="" type="checkbox"/> 1.3 - Linear Structural Analysis, Static | 6s       | Warnings | Yes            |
| <input checked="" type="checkbox"/> 1.4 - Load Results                       | 0s       | Success  |                |

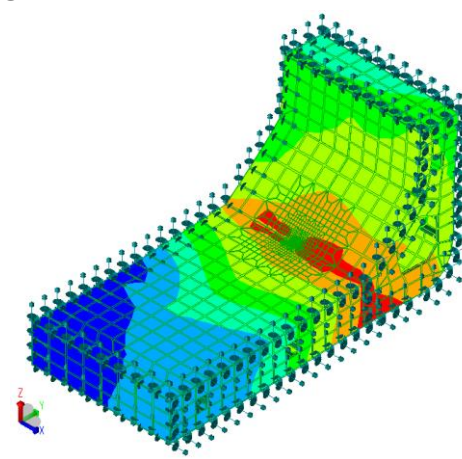
Sub-model with automatic mesh refinement



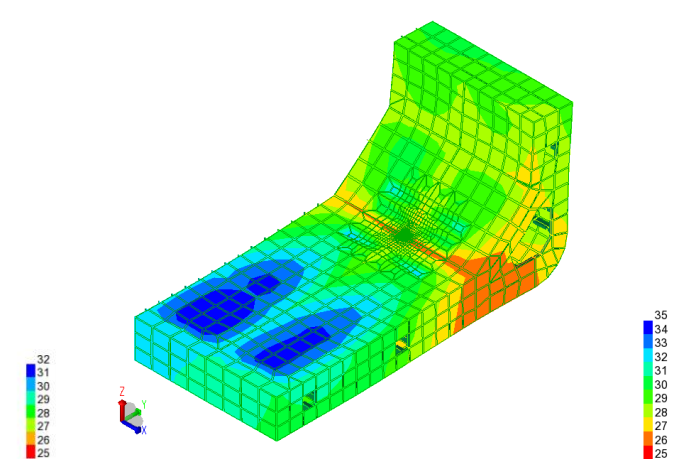
Pressures distributed from global model to sub-model



Prescribed displacements from global model to sub-model



Solver providing deformations and stresses



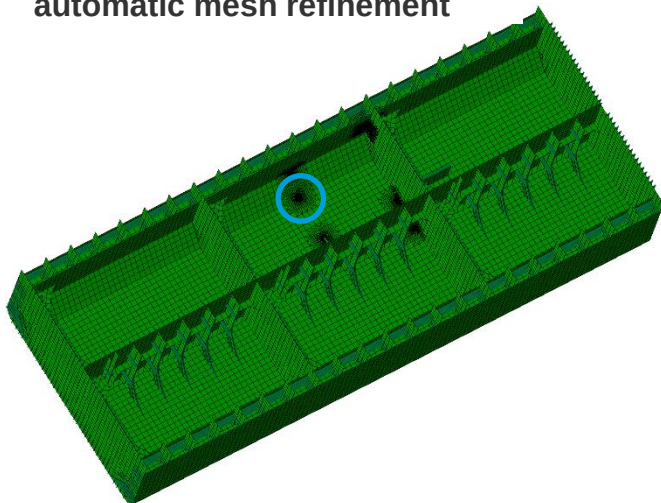
# FE Rule checks for CSR BC&OT

GeniE has a seamless interface to the FE Fatigue tool for standard details required by the Rules

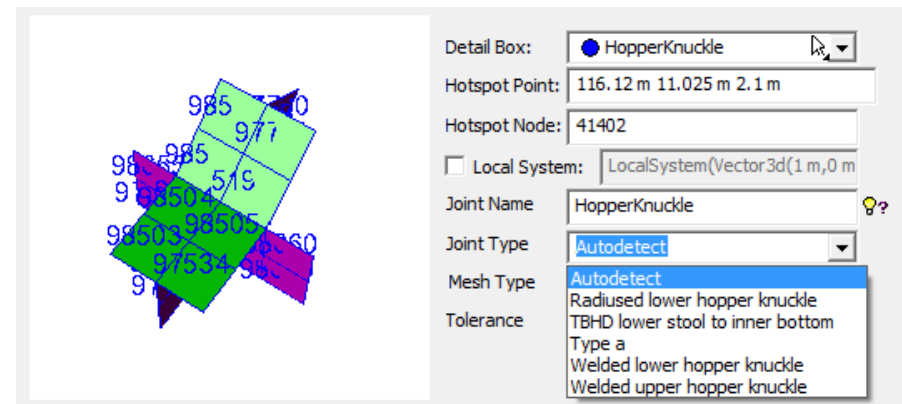
## FE Fatigue assessment – Direct access to FE Fatigue tool

- For all mandatory web stiffened cruciform joints according to CSR BC&OT, GeniE **automatically** detects all hot spots and also upper or lower surface (**Autodetect**)
  - Welded lower hopper knuckle
  - Radiused lower hopper knuckle
  - Welded upper hopper knuckle
  - Connections of TBHD lower stool to inner bottom plating iwo double bottom girders
- User may also manually specify joint type
- GeniE will export a xml file that is used by the Nauticus Hull FE Fatigue Tool

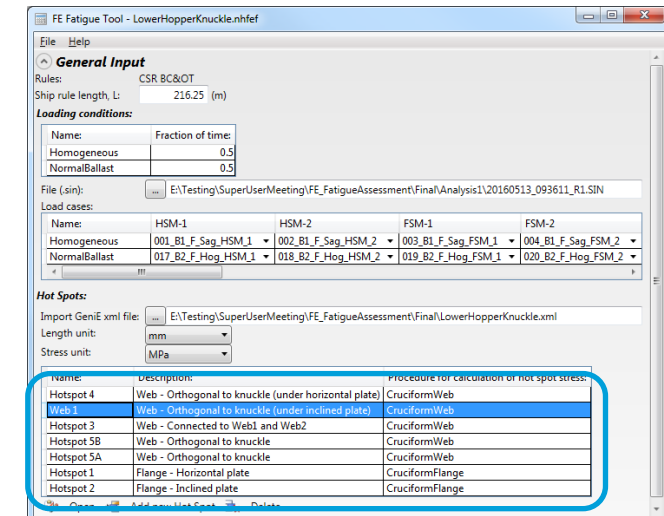
Cargo hold model with automatic mesh refinement



Export Fatigue Detail from GeniE



All hot spots automatically imported to FE Fatigue tool



# Modelling features in GeniE

Robust geometry model ensured by the Dual assembly

Slot/lug concept and hole concept used in FEA Rule checks

Plate seam concept ensuring same representation as NH

Modelling grids for easy navigation

Hull design tool for non-prismatic outer shell

Hull design tool imprinting stiffeners on non-prismatic outer shell

Active set and dynamic set

Clipping planes giving easy access to parts of the model

Default stiffener orientation for efficient modelling

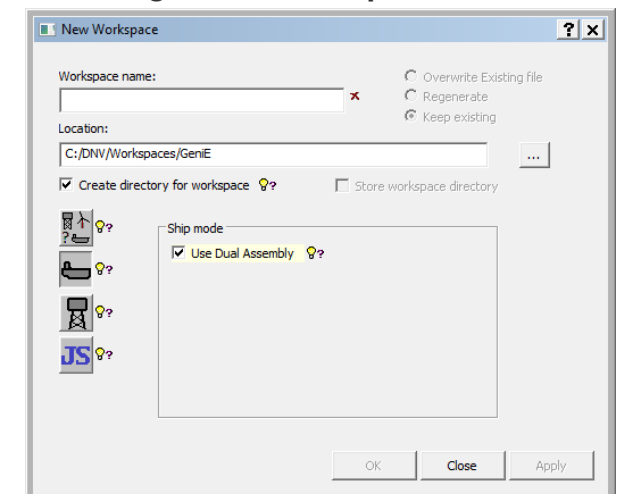
Direct access to model verification

# Modelling features in GeniE

## Robust geometry model ensured by the Dual assembly

- Dual Assembly is a new topology model in GeniE. It is based on a full re-write of the software code on how to represent the connectivity between stiffeners and plates.
- Dual assembly, consisting of two topological representations:
  - Old “Assembly A” consisting of plates (2D entities), using ACIS (GeniE’s geometry kernel)
  - New “Assembly B” consisting of stiffeners, support curves, feature edges, point masses and support points (0D and 1D entities), not using ACIS
- The main benefits of the Dual assembly is a more robust stiffener representation when they are connected to curved shells
- Other benefits (mainly due to a more effective data structure)
  - Improved performance on database operations, e.g. open workspace
  - Improved performance on modelling operations e.g. join plates/stiffeners, delete plates/stiffeners
  - Improve performance on FEM import as connectivity between plates and stiffeners are not using ACIS
- There are no differences in how to do modelling when using the Dual assembly
- The finite element mesh will be the same when using old and new topology model
- Old GeniE models (GNX, XML and js) can be imported into a new GeniE model using the Dual assembly

### Dual assembly is enabled when creating a new workspace



The screenshot shows a 3D model of a ship's hull in a software environment. The hull is primarily red, with a grey superstructure at the stern. A coordinate system (X, Y, Z) is visible in the bottom-left corner. An error dialog box is open in the center, titled "Create a set of stiffener failed". The dialog contains a yellow warning icon and the following text:

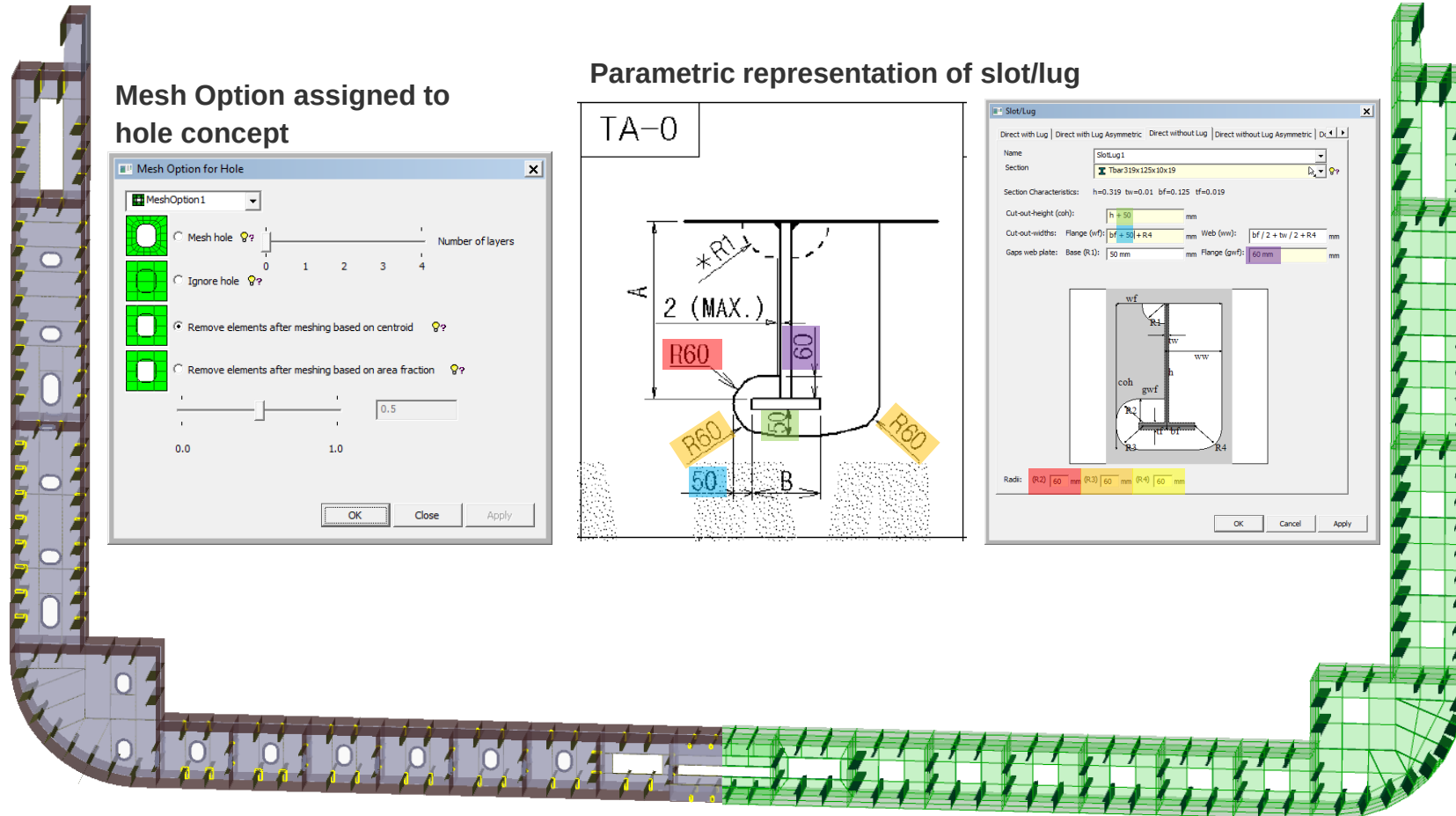
[100002] Invalid operation (line 1):  
 In function:  
 CreateSetOfStiffenerOnCircPlane@100;  
 Input: edgePlaneFunction, sctNoConstraint, 0, 0; Internal  
 ACS error:  
 ACS error: - with edgeFace (coincident) not yet  
 implemented  
 Please ONCO the previous command to revert to a valid  
 model.  
 It is recommended to verify the model using  
 ToolsStructureInventory  
 Please report to DNV GL - Digital Solutions

An "OK" button is located at the bottom right of the dialog box.

# Modelling features in GeniE

## Slot/lug concept and hole concept used in FEA Rule checks

- The hole concept can be used to represent openings in the hull structure, meshing rules can be applied
- The slot/lug concept can be used to represent detailed geometry information of stiffeners connection to PSMs

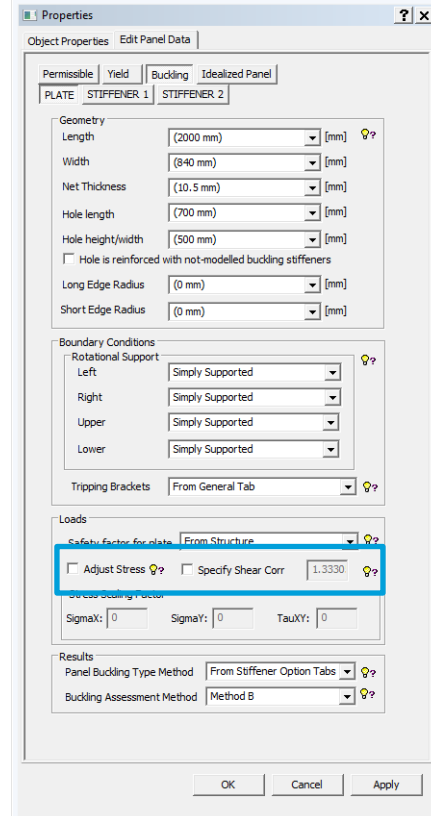
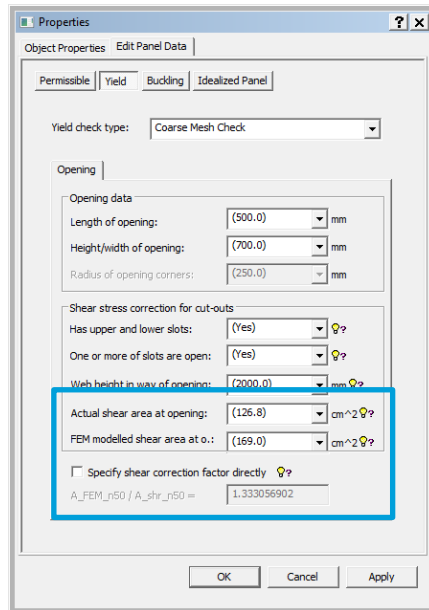


# Modelling features in GeniE

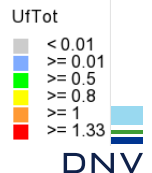
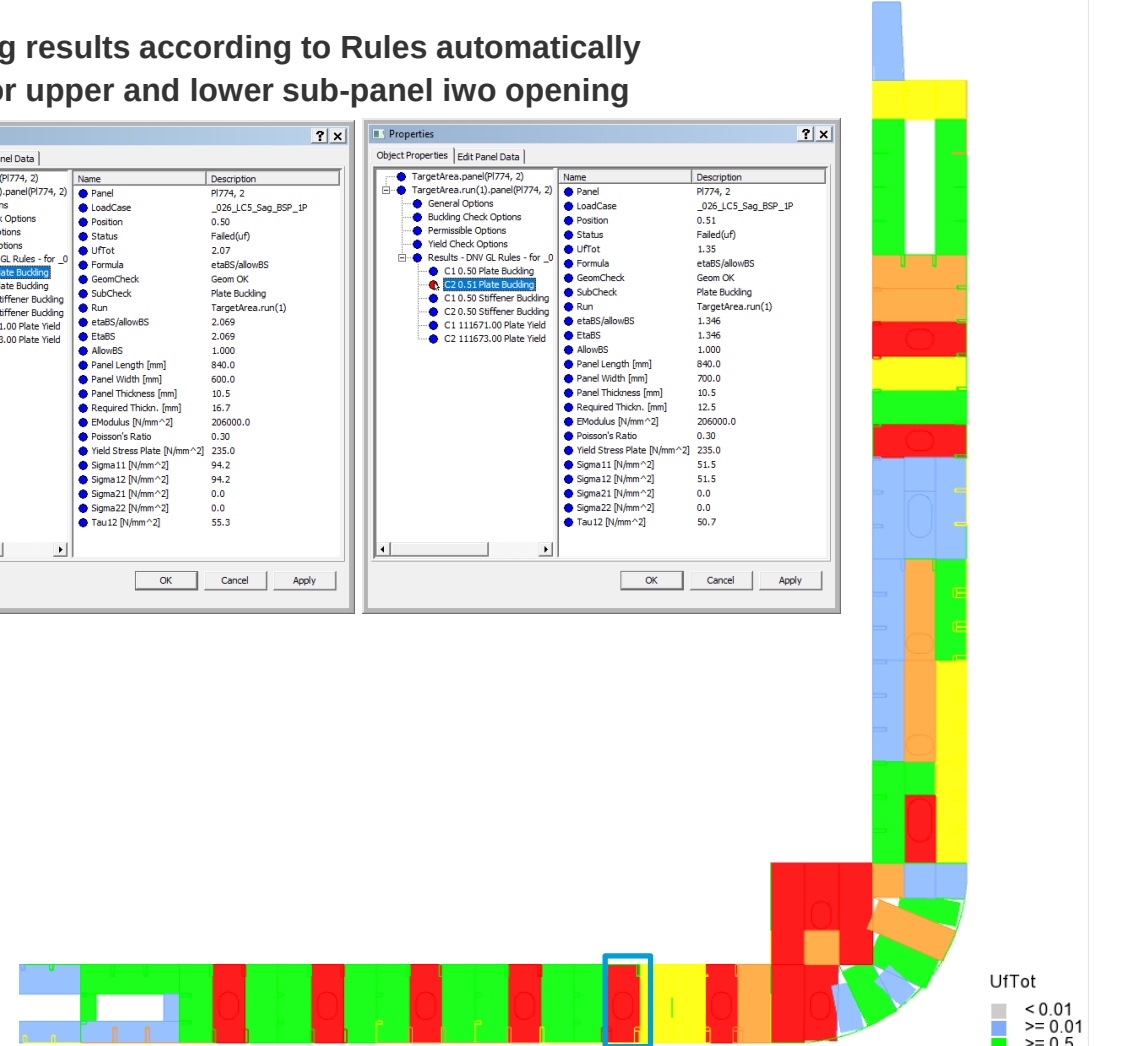
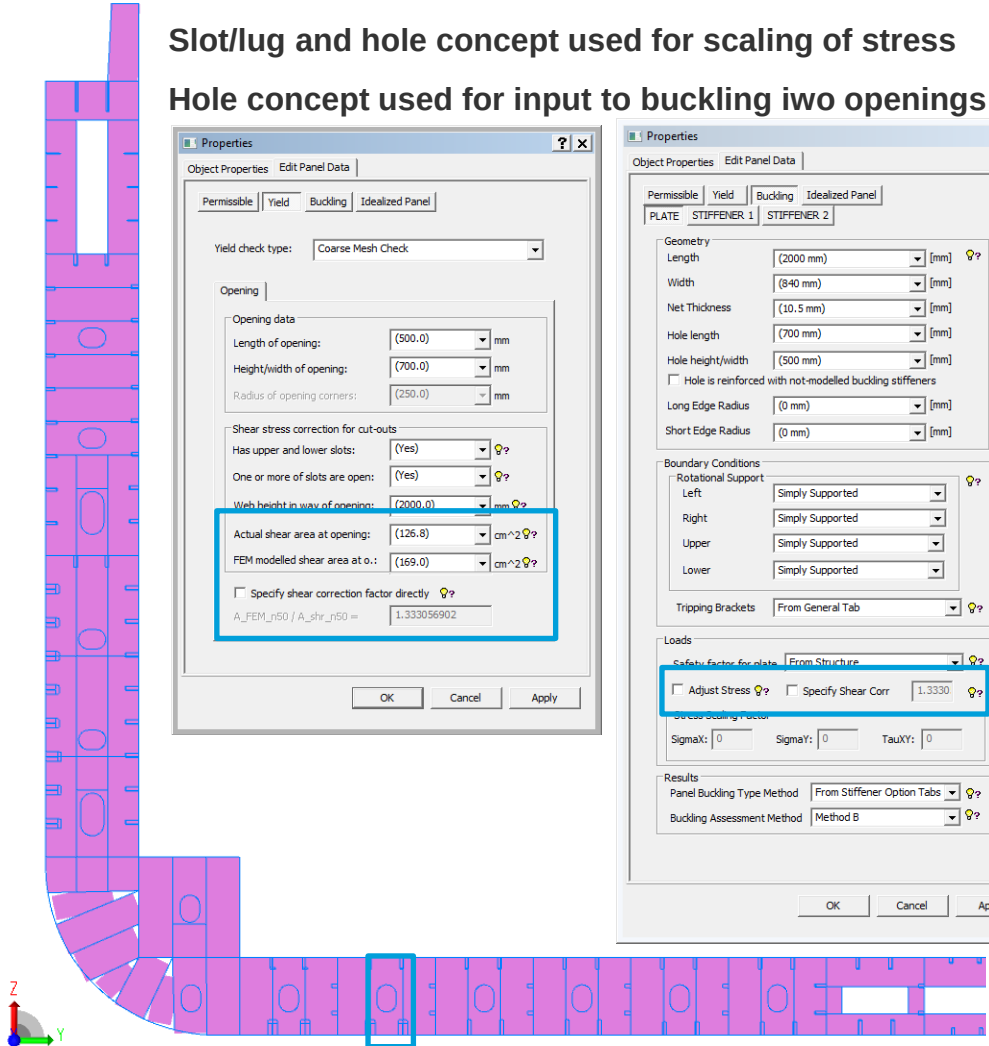
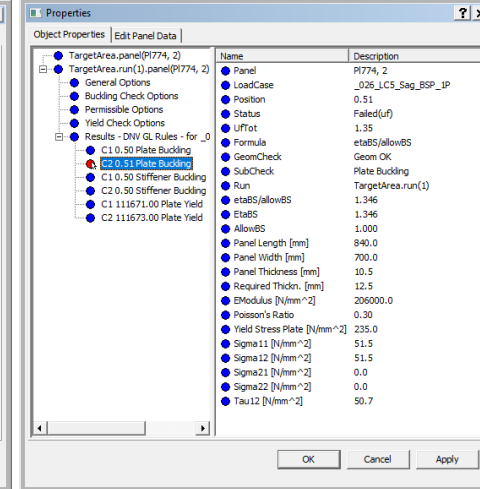
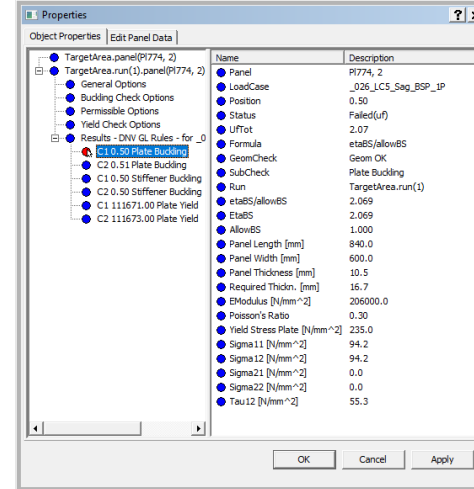
## Slot/lug concept and Hole concept used in FEA Rule checks

Slot/lug and hole concept used for scaling of stress

Hole concept used for input to buckling iwo openings



Buckling results according to Rules automatically given for upper and lower sub-panel iwo opening



# Modelling features in GeniE

## Plate seam concept ensuring same representation as NH

When using plate seams, there is not need to divide a surface to assign different properties to it (thickness, material, corrosion addition)

- Plate seams can be assigned to **surfaces**, similar to Nauticus Hull. Essential for re-use of 3D models from Nauticus Hull.
- Plate seams will **not** introduce any new geometry and therefore **not** have on impact on the layout of nodes and elements
- GeniE will only use the plate seam information when assigning **properties** to the shell elements
- The plate seam information will be used to compute the effective width of the buckling panel, as required by the Rules

$$b_{eff} = \ell_1 + \ell_2 \left( \frac{\ell_1}{\ell_2} \right)^{1.5}$$

where:

$\ell_1$  : Width of the part of the plate panel with the smaller net plate thickness,  $t_1$ , in mm, as defined in Figure 1.

$\ell_2$  : Width of the part of the plate panel with the greater net plate thickness,  $t_2$ , in mm, as defined in Figure 1.

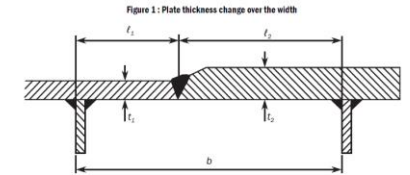


Plate seams on geometry

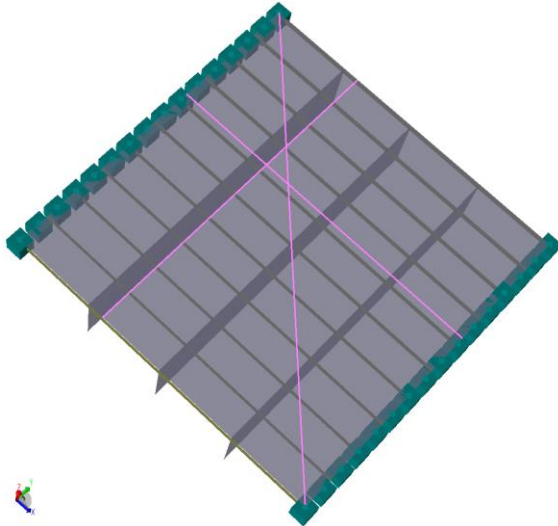
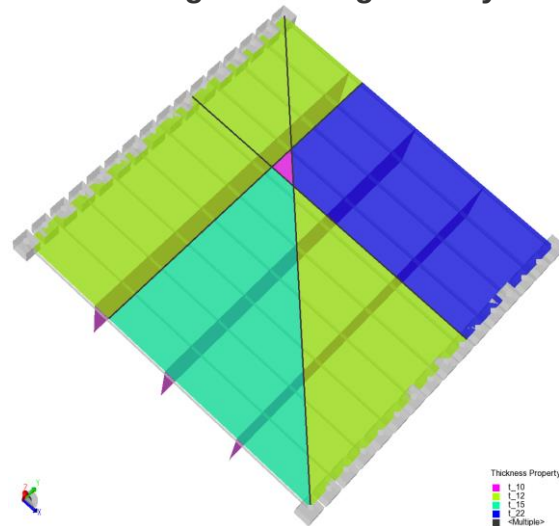
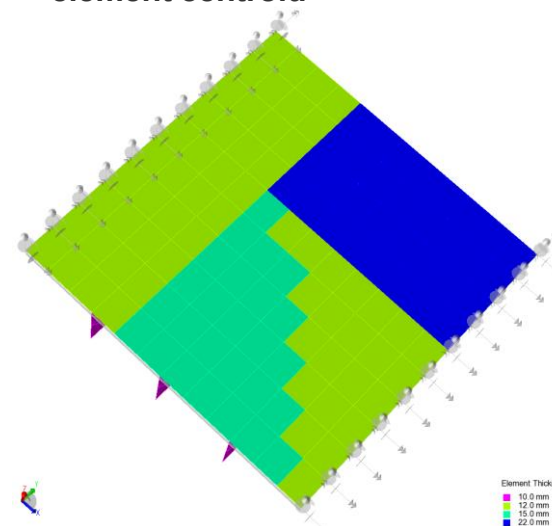


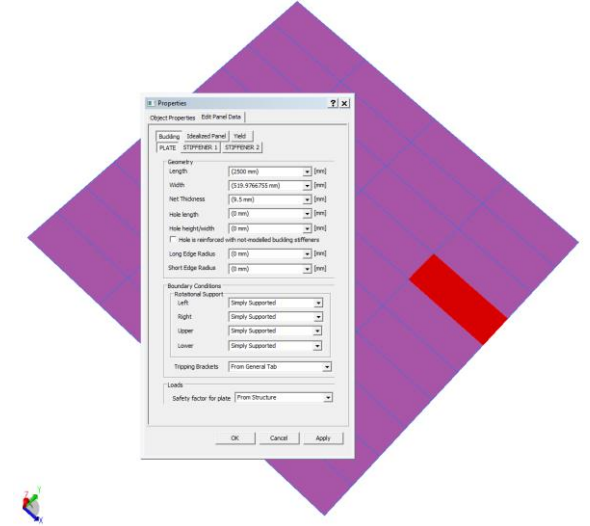
Plate segments on geometry



Element property based on element centroid



Width of buckling panel using plate seam information

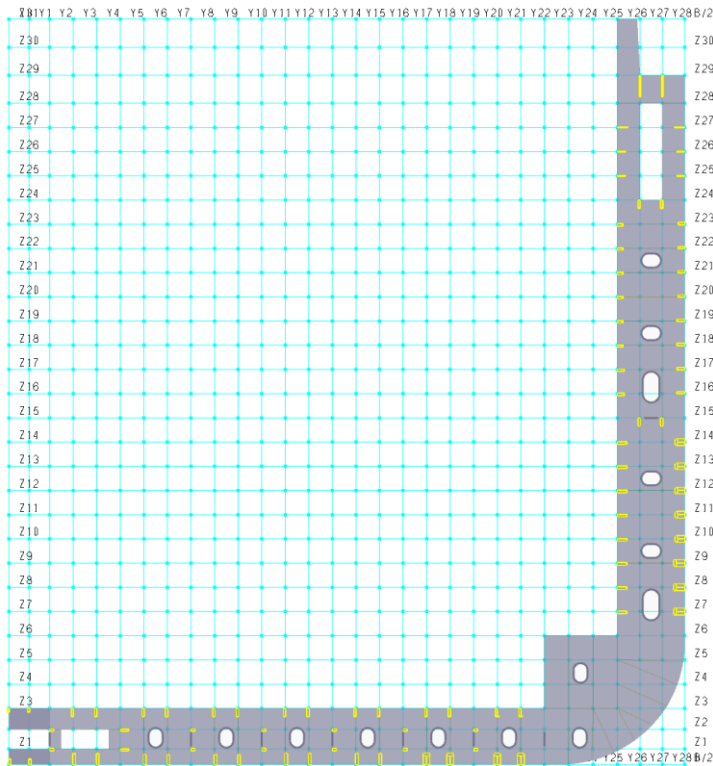


# Modelling features in GeniE

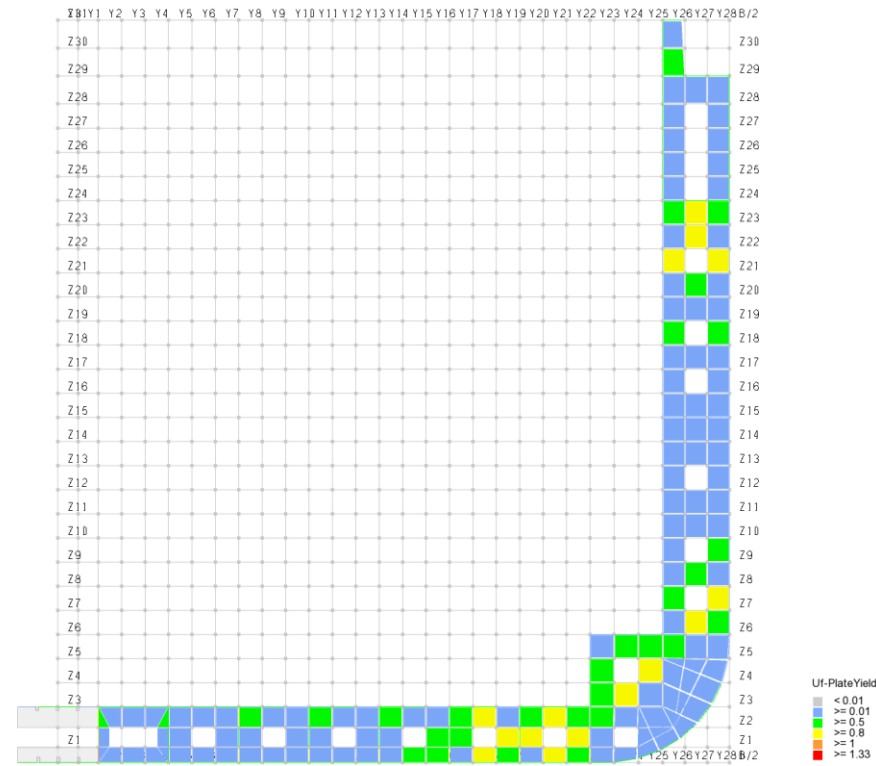
## Modelling grids for easy navigation

- By using point grids, we can include drawing references like frames, deck levels etc.
- User can create 2D grids that gives snapping points for modelling of geometry
- The point grids are also available when displaying FE Rule check results

**Modelling grids, with snapping, available during modelling**



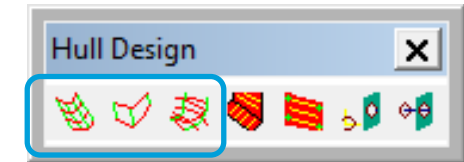
**Modelling grids when displaying FE Rule check results**



# Modelling features in GeniE

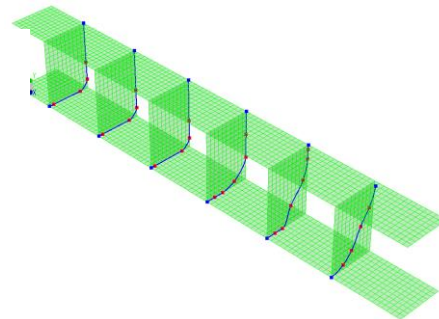
## Hull design tool for non-prismatic outer shell

- Non-prismatic hull form can be created by skinning approximation of smooth curves or polygons
- Smooth curves or polygons may be imported from other tools, e.g. NAPA DXF curves
- Hull Design Tools toolbar has three alternative methods for creation of non-prismatic hull form
  - Method 1: Create Ship Hull from Transverse Smooth Curves
  - Method 2: Create Ship Hull from Transverse Polylines
  - Method 3: Create Ship Hull from Arbitrary Set of Curves

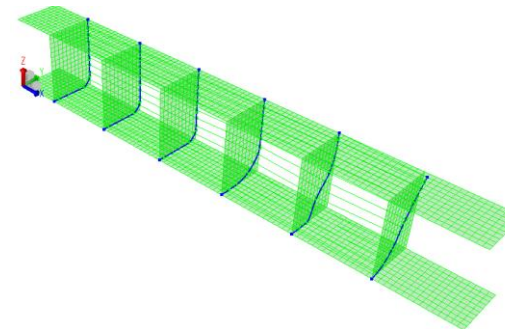


**Method 1 and method 2:  
Trans. Smooth Curves / Polylines**

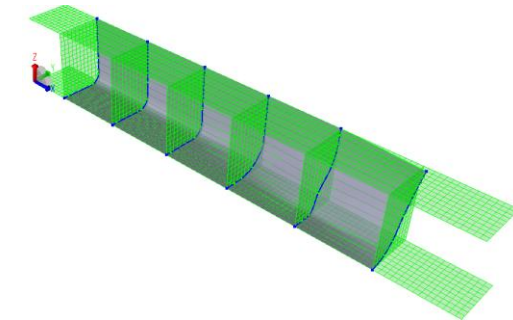
**Curves to be approximated**



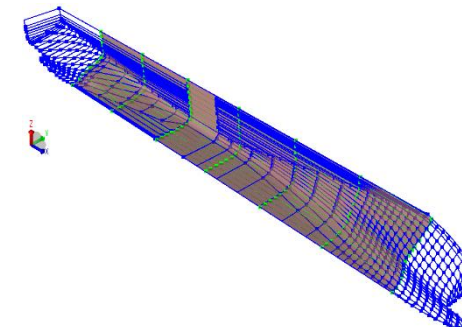
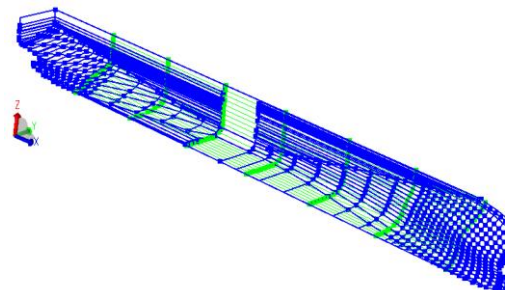
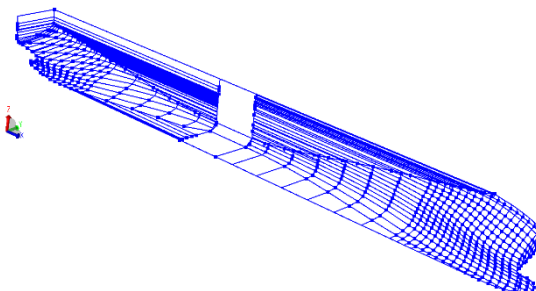
**Point grids created based on the  
approximated curves/polylines**



**The final surface created based on  
the point grids**



**Method 3:  
Arbitrary Set of Curves**

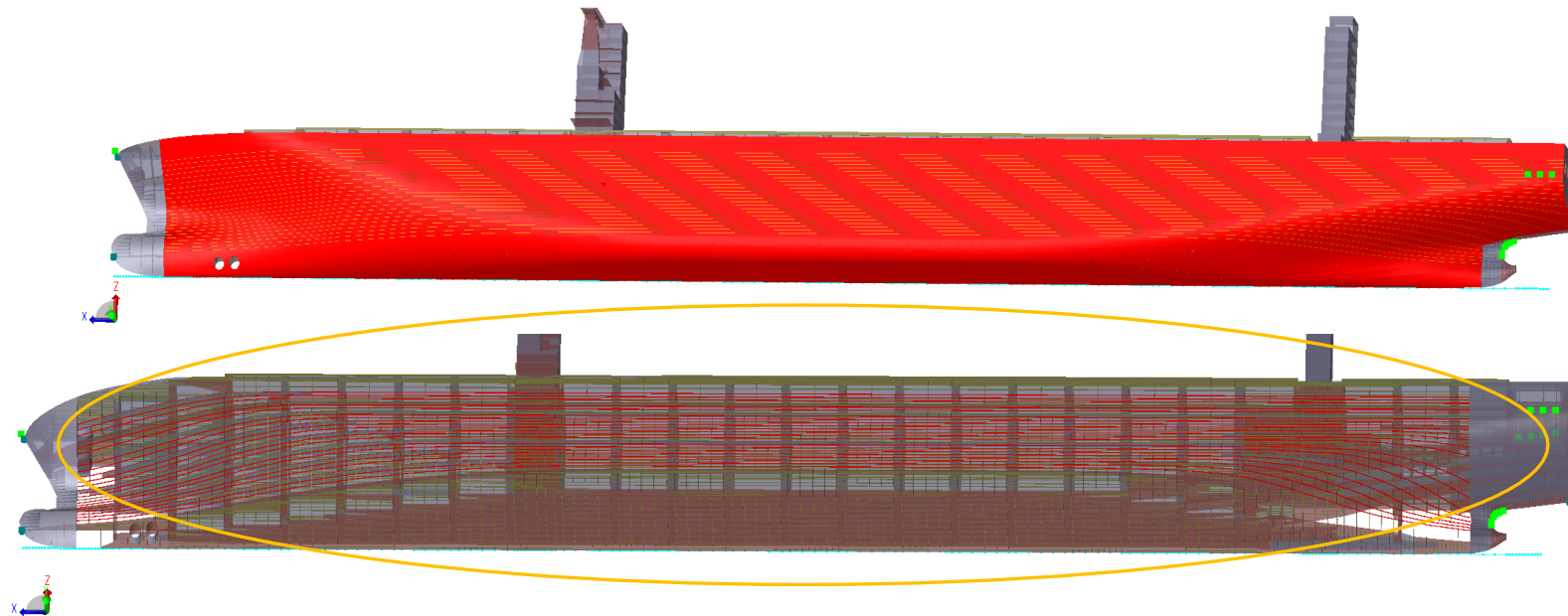
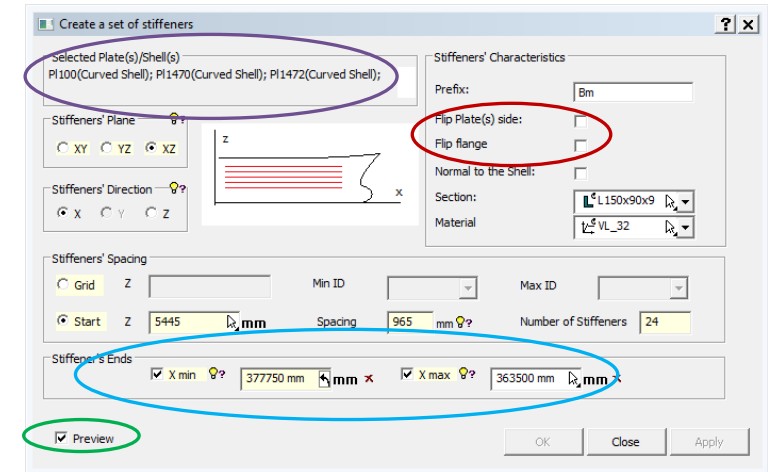


# Modelling features in GeniE

## Hull design tool imprinting stiffeners on non-prismatic shell

*This feature, with the dual assembly, will give a fast modelling of stiffeners attached to complex outer shell*

- Stiffeners on non-prismatic outer shell can be efficiently created
  - Allow for selecting more than one surface
  - Preview introduced
  - The beams that are created are not split in way of intersecting topology (e.g. web frames)
  - User can specify start and end of the stiffeners
  - Stiffeners can be flipped: Flip plate side and flip flange
- High performance: 24 stiffeners takes 0 min 55 sec to create

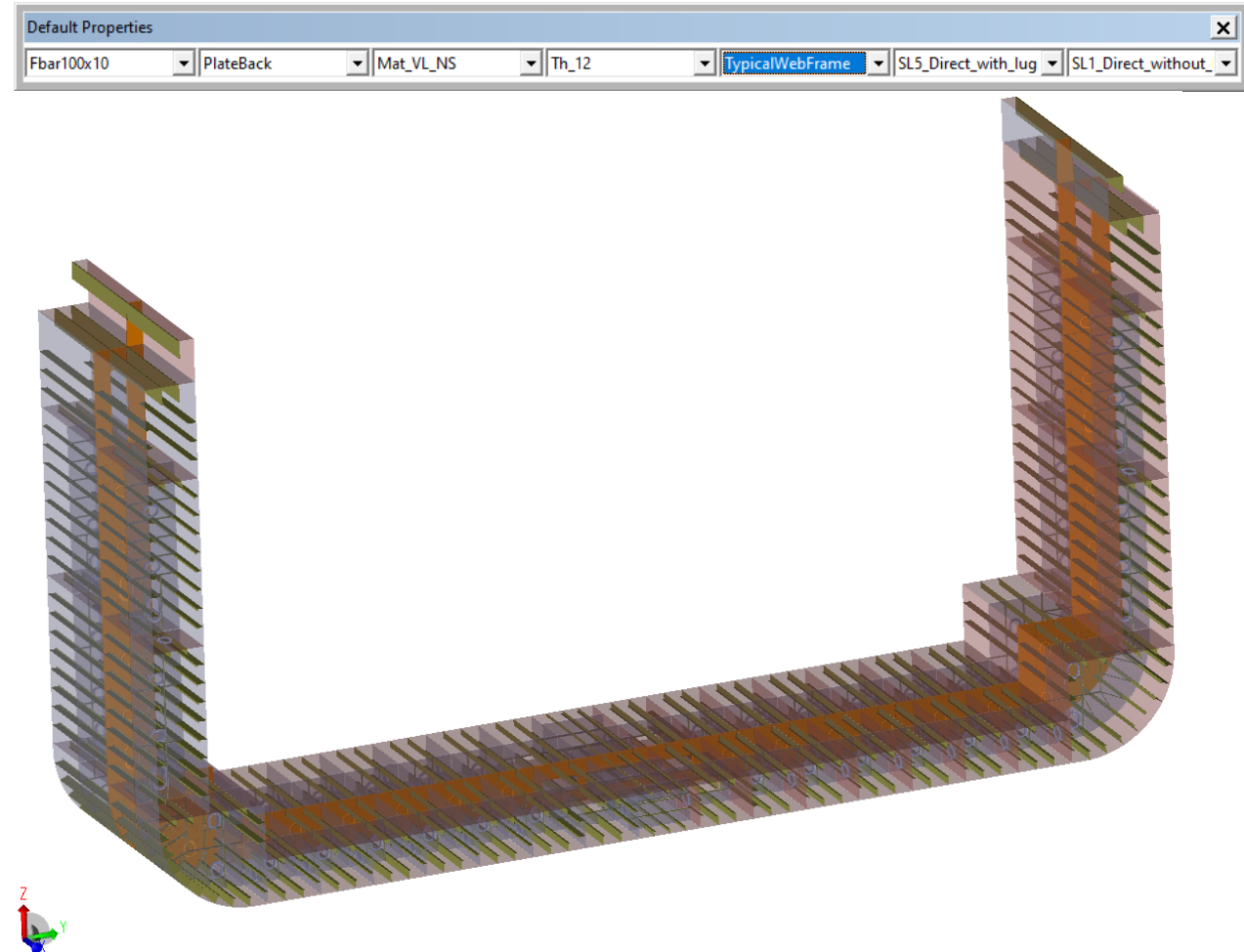


# Modelling features in GeniE

## Active set and dynamic set

- Parts of the model can be collected in sets
- Sets can be activated when modelling and when displaying results
- The user can specify an active set, meaning that all members modelled will be automatically assigned to the selected active set
- Dynamic sets are automatically updated during modelling
- Time needed for update of dynamic sets increases exponentially with the number
- Recommended to have ONE dynamic set for each typical part and complete (modelling and mesh tuning) this part before copying
  - One typical web frame
  - One typical transverse bulkhead
  - One typical pillar bulkhead

Active set can be easily selected on the Default Properties toolbar

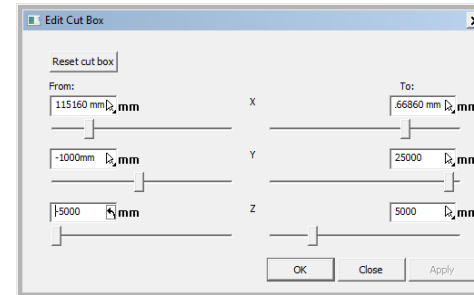


# Modelling features in GeniE

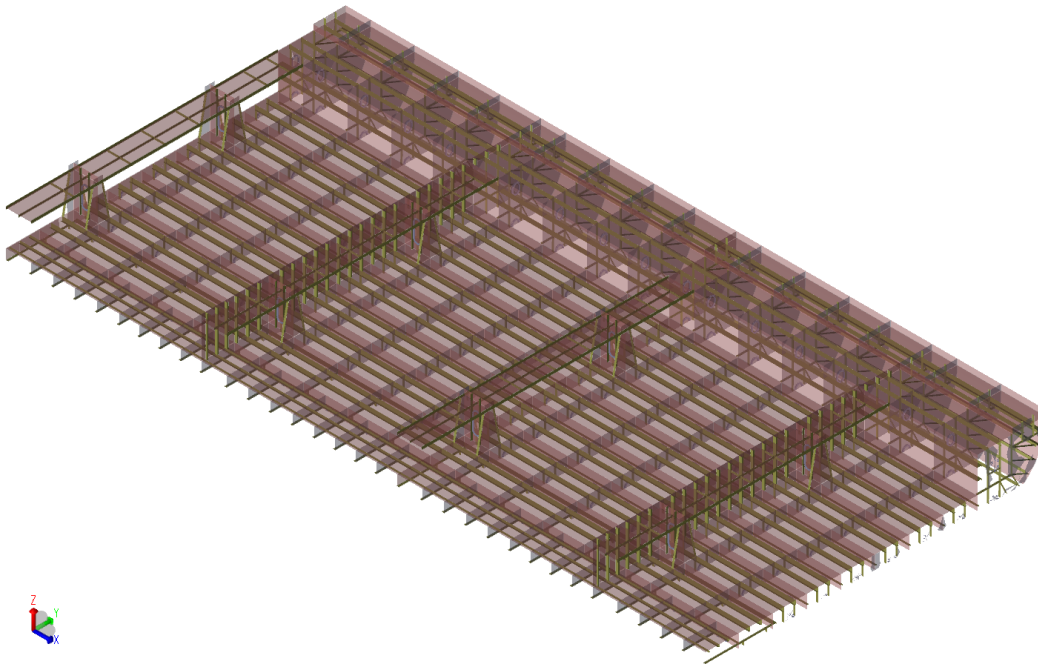
## Clipping planes giving easy access to parts of the model

- Clipping planes can be used to display parts of the model
- The clipping planes can be dynamically updated by
  - dragging sliding bars
  - Specifying coordinate; or
  - Interactively picking a point in the model

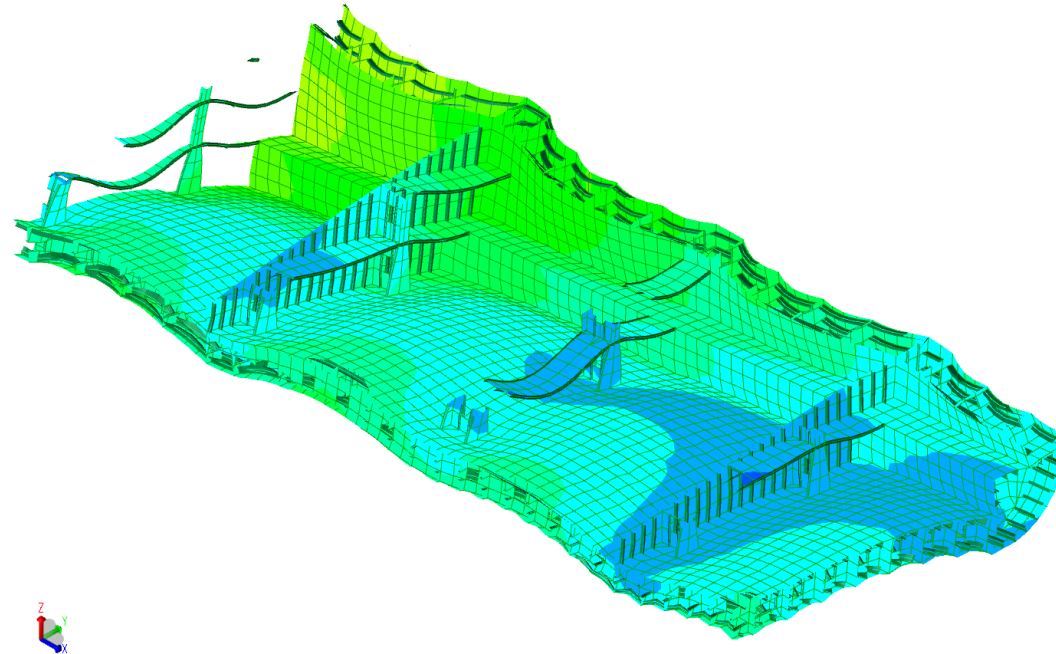
Clipping planes dialog



Clipping planes for easy access to parts of the model



Clipping planes also available when displaying deformations

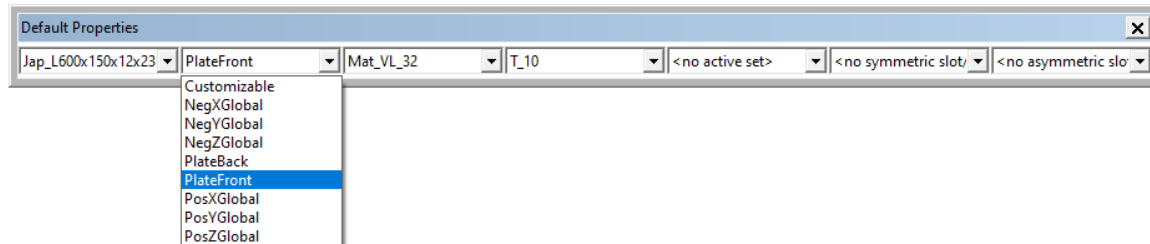


# Modelling features in GeniE

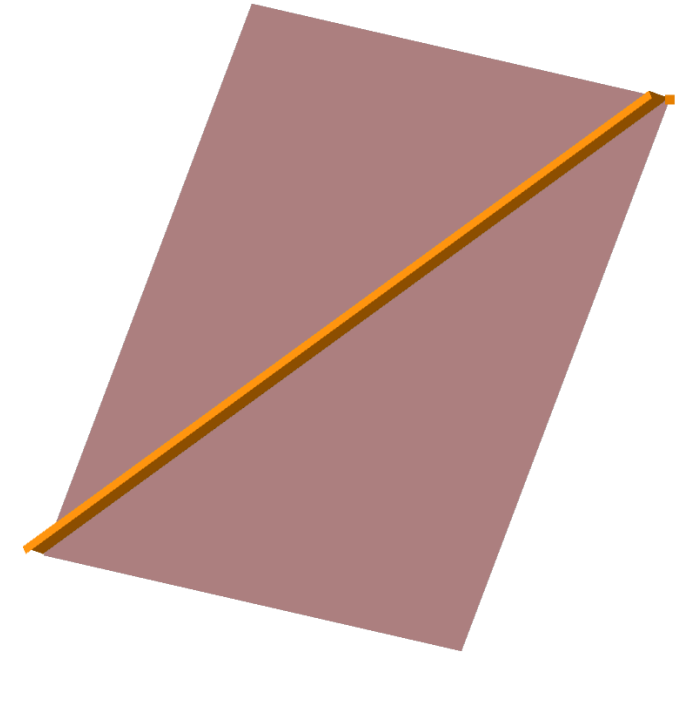
## Default stiffener orientation for efficient modelling

- The user can select a default stiffener orientation that will be applied when creating new stiffeners
- User will not need to rotate the stiffeners after creation, avoiding a step
- The default stiffener orientation includes various options, not only global coordinate system, but also normal to a plate

### Default stiffener orientation available under Default Properties toolbar



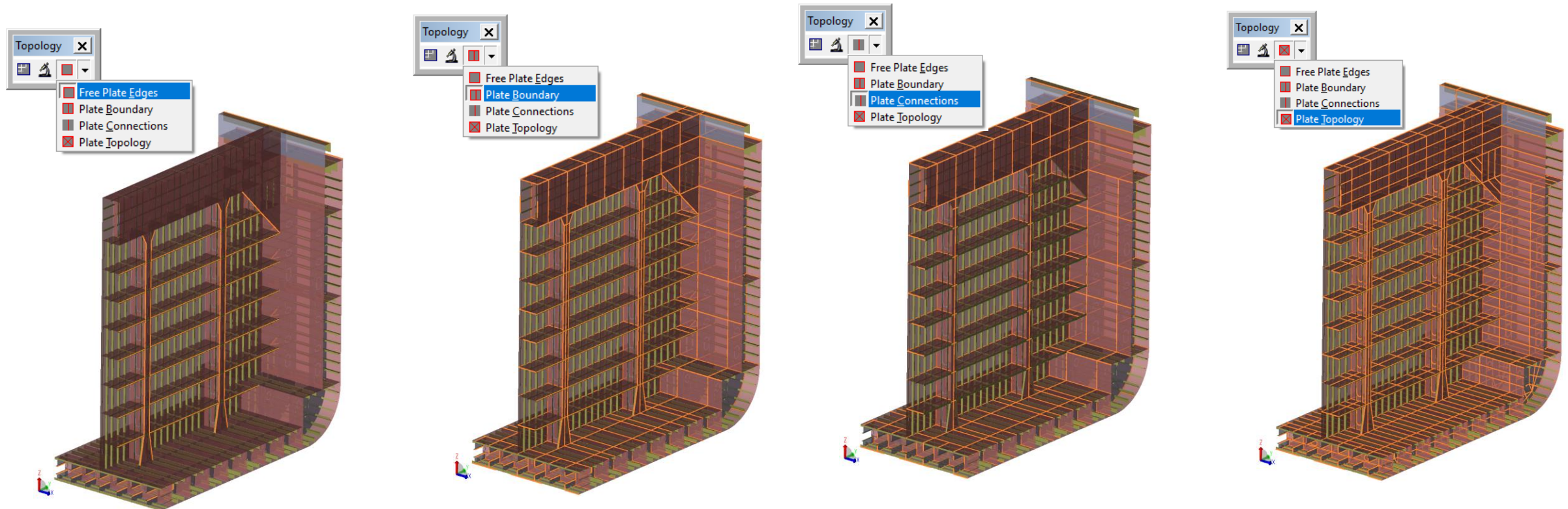
**Stiffeners can easily be created perpendicular to an inclined plate**



# Modelling features in GeniE

## Direct access to model verification

- GeniE has direct access to model verification tools
- Display Free Plate Edges is a powerful verification tool for investigation of model inconsistency, in particular sliver faces
- Sliver faces may give bad mesh and also problems when creating compartments (needed for compartment loads)
- Other useful verification tools are available (in addition to Display Free Plate Edges)



# Meshing in GeniE

Conditional Regeneration of mesh for high meshing performance

Template Quad mesher optimized for ship structures

Mesher allowing for modelling inaccuracies

Feature edges specifying meshing lines

Mesh properties ensuring mesh size according to Rules

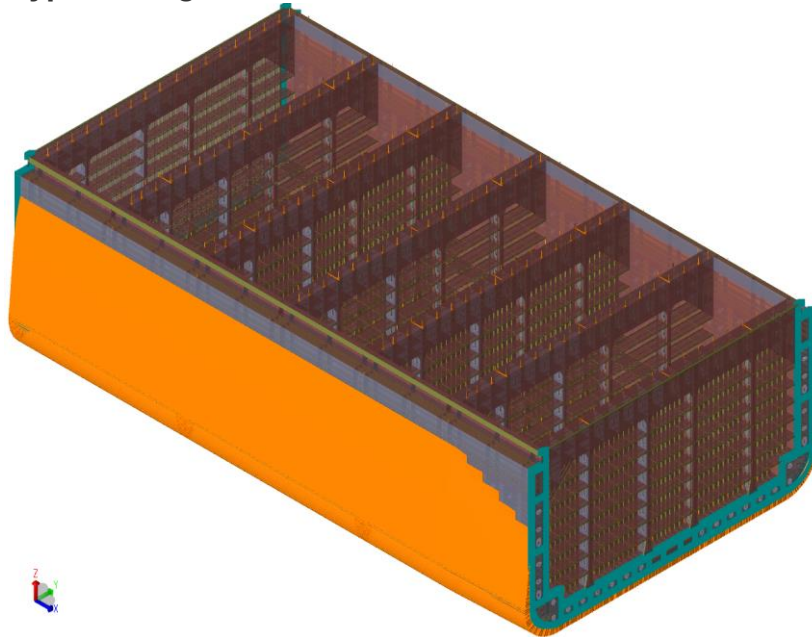
Mesh Editing tools giving control on the mesh

# Meshing in GeniE

## Conditional Regeneration of mesh for high performance

- GeniE has two mesh regeneration options available
  - Conditional Regenerate Mesh (default setting); and
  - Always Regenerate Mesh
- Always Regenerate Mesh: Full regeneration of loads and mesh, every time the meshing is executed
- Conditional Regenerate Mesh: Scans through the model and only update the mesh, where needed
- On \*.bdf models imported to GeniE, the Conditional Regenerate Mesh ensures that the mesh remains unchanged after importing loads and assigning corrosion additions, updating properties, etc.

Typical cargo hold model with 32 load cases



2 min 44 sec for a full mesh re-generation  
(Always Regenerate Mesh)

| Activity Monitor                                                      |          |         |                |
|-----------------------------------------------------------------------|----------|---------|----------------|
| Done! [Start] [Cancel]                                                |          |         |                |
| <input checked="" type="checkbox"/> Journal activity executions       |          |         |                |
| Activity                                                              | Duration | Status  | Generate Input |
| <input type="checkbox"/> 1 - Analysis1 - Analysis                     | 164s     |         |                |
| <input checked="" type="checkbox"/> 1.1 - Meshing (Always Regenerate) | 164s     | Success |                |
| <input checked="" type="checkbox"/> 1.1.1 - Delete loads              | 0s       | Success |                |
| <input checked="" type="checkbox"/> 1.1.2 - Generate loads            | 33s      | Success |                |
| <input checked="" type="checkbox"/> 1.1.3 - Delete mesh               | 4s       | Success |                |
| <input checked="" type="checkbox"/> 1.1.4 - Generate mesh             | 127s     | Success |                |
| <input type="checkbox"/> 1.2 - Hull Girder Load Adjuster              | ---      |         | Yes            |
| <input type="checkbox"/> 1.3 - Linear Structural Analysis, Static     | ---      |         | Yes            |
| <input type="checkbox"/> 1.4 - Load Results                           | ---      |         |                |

13 sec for changing a plate thickness and  
Conditional Regenerate Mesh

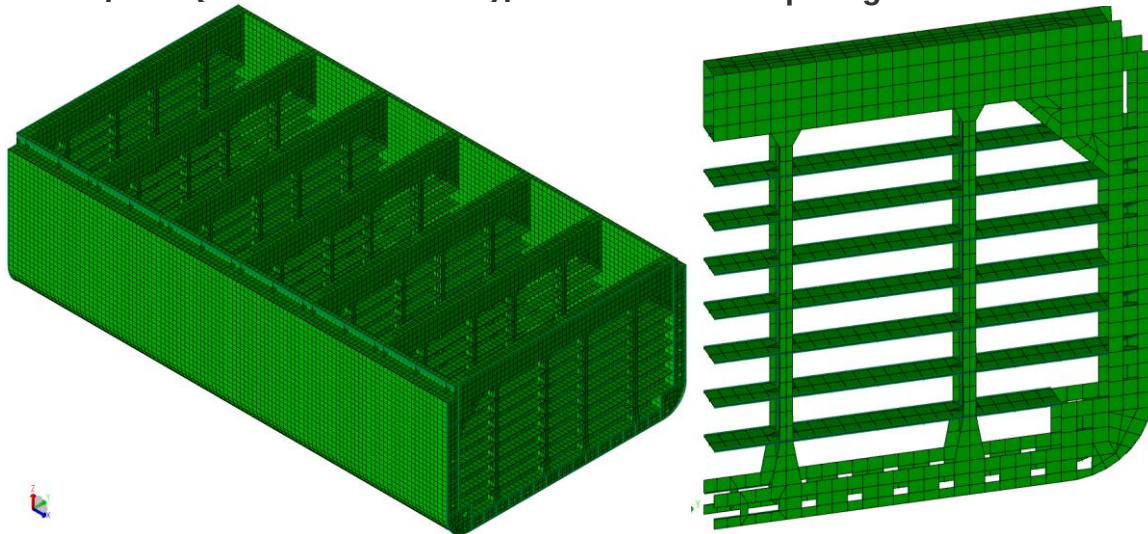
| Activity Monitor                                                           |          |         |                |
|----------------------------------------------------------------------------|----------|---------|----------------|
| Done! [Start] [Cancel]                                                     |          |         |                |
| <input checked="" type="checkbox"/> Journal activity executions            |          |         |                |
| Activity                                                                   | Duration | Status  | Generate Input |
| <input checked="" type="checkbox"/> 1 - Analysis1 - Analysis               | 13s      |         |                |
| <input checked="" type="checkbox"/> 1.1 - Meshing (Conditional Regenerate) | 13s      |         |                |
| <input type="checkbox"/> 1.1.1 - Delete loads                              | ---      |         |                |
| <input type="checkbox"/> 1.1.2 - Update loads                              | ---      |         |                |
| <input type="checkbox"/> 1.1.3 - Delete mesh                               | ---      |         |                |
| <input type="checkbox"/> 1.1.4 - Update mesh                               | 13s      | Success |                |
| <input checked="" type="checkbox"/> 1.2 - Hull Girder Load Adjuster        | ---      |         | Yes            |
| <input type="checkbox"/> 1.3 - Linear Structural Analysis, Static          | ---      |         | Yes            |
| <input checked="" type="checkbox"/> 1.4 - Load Results                     | 0s       | Success |                |

# Meshing in GeniE

## Template Quad mesher optimized for ship structures

- GeniE has different meshers available, optimised for various geometries, typically
  - Template Quad mesher; and
  - Advancing front Quad mesher (paver meshing algorithm)
- Template Quad mesher divides surfaces into patches and creates a mesh based on these
- Advancing front Quad mesher generates the mesh along the edges first and bases the surface mesh on the edges
- The user can assign different meshers to various parts of the model
- Template Quad mesher is the default mesher, with high performance, optimised for regular stiffened plate fields
- All meshers in GeniE will always ensure that beam elements and shell elements are connected in common nodes

Template Quad mesher on a typical container ship cargo hold model



Template quad:  
2 min 44 sec

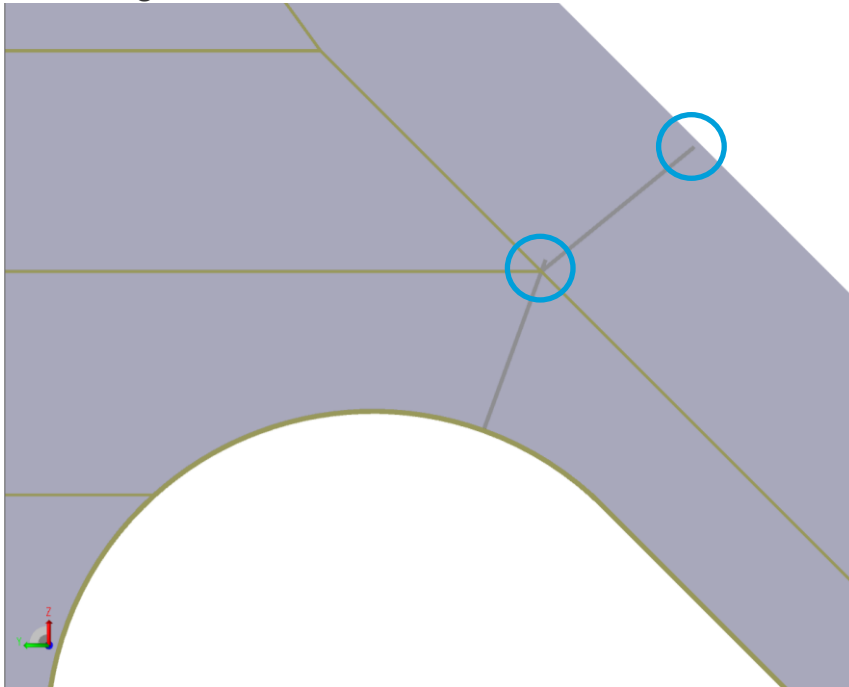
| Activity                                                                     | Duration | Status  | Generate Input |
|------------------------------------------------------------------------------|----------|---------|----------------|
| <input type="checkbox"/> 1 - Analysis1 - Analysis                            | 164s     |         |                |
| <input checked="" type="checkbox"/> 1.1 - Meshing (Always Regenerate)        | 164s     | Success |                |
| <input checked="" type="checkbox"/> 1.1.1 - Delete loads                     | 0s       | Success |                |
| <input checked="" type="checkbox"/> 1.1.2 - Generate loads                   | 33s      | Success |                |
| <input checked="" type="checkbox"/> 1.1.3 - Delete mesh                      | 4s       | Success |                |
| <input checked="" type="checkbox"/> 1.1.4 - Generate mesh                    | 127s     | Success |                |
| <input checked="" type="checkbox"/> 1.2 - Hull Girder Load Adjuster          | ---      |         | Yes            |
| <input checked="" type="checkbox"/> 1.3 - Linear Structural Analysis, Static | ---      |         | Yes            |
| <input type="checkbox"/> 1.4 - Load Results                                  | ---      |         |                |

# Meshing in GeniE

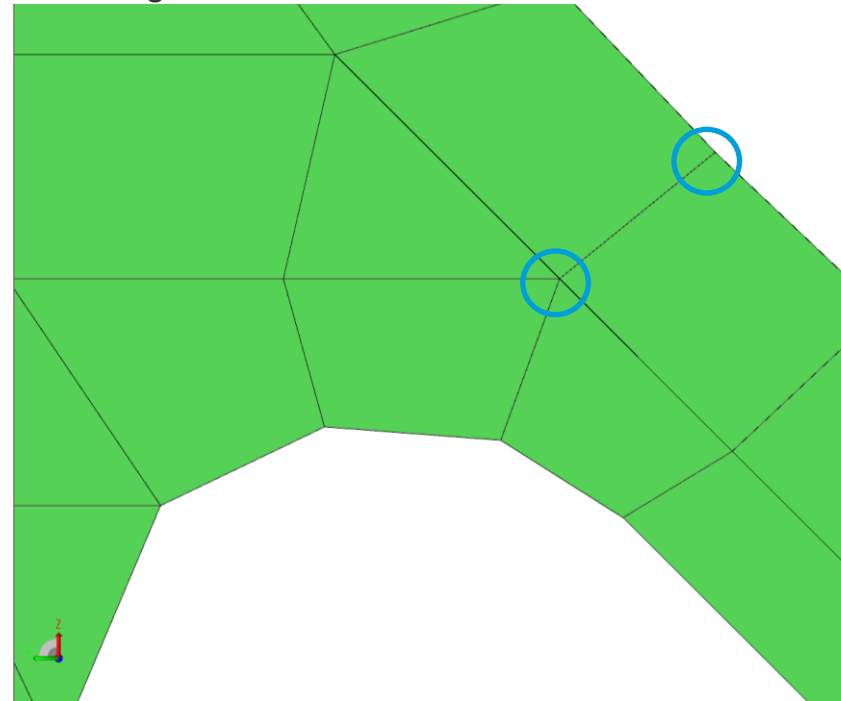
## Mesher allowing for modelling inaccuracies

- During modelling the user by mistake may create structural members that are not correctly intersecting each other
- Below example shows two typical scenarios
  - A stiffener has been modelled too long, resulting in an stiffener with a loose end, that shouldn't be there
  - A stiffener has been modelled too short, resulting in a gap, that shouldn't be there
- GeniE has powerful features, activated by default, ignoring such inaccuracies when creating the mesh

Typically modelling inaccuracies from  
modelling of stiffeners



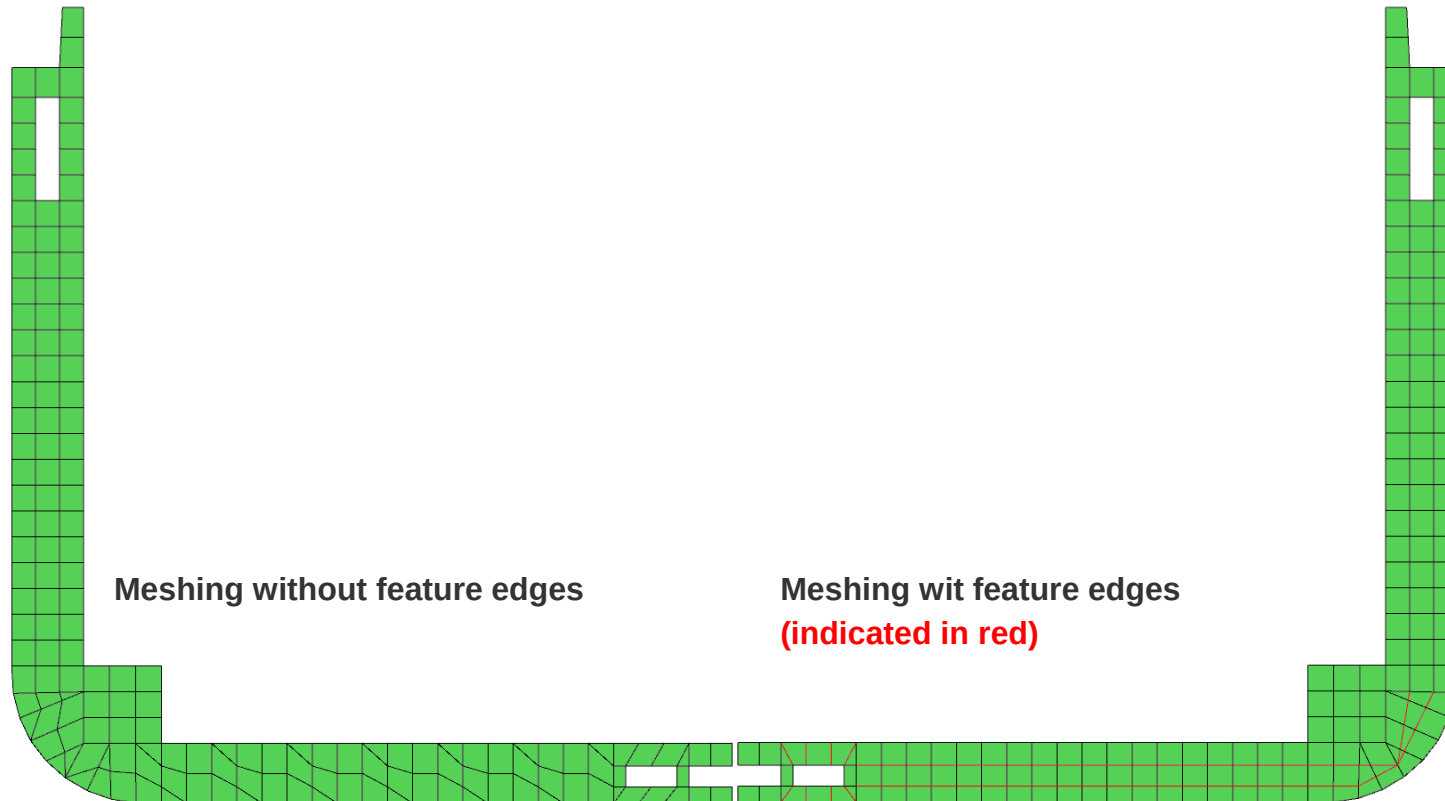
When meshing, GeniE is ignoring the  
modelling inaccuracies



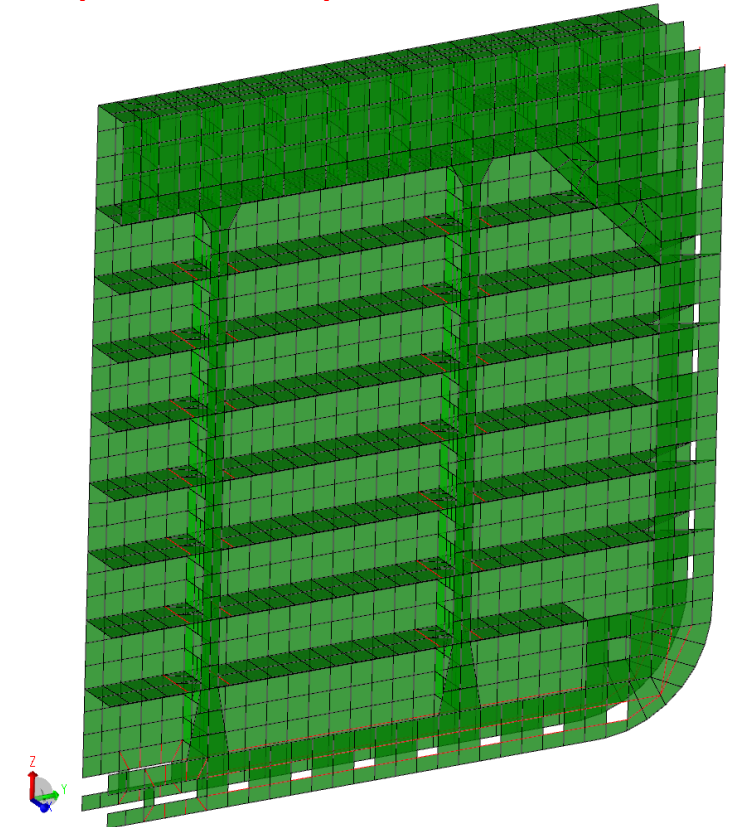
# Meshing in GeniE

## Feature edges specifying meshing lines

- Feature edge is used for controlling the mesh, the mesh lines will follow the Feature edge
- Feature edges have same modelling options as creation of other geometries, e.g. snapping, copy
- The user may also assign number of elements to a Feature edge
- Feature edges perpendicular to stiffeners will typically ensure rectangular mesh



Typical application of feature edges  
(indicated in red)



# Meshing in GeniE

## Mesh properties ensuring mesh size according to Rules

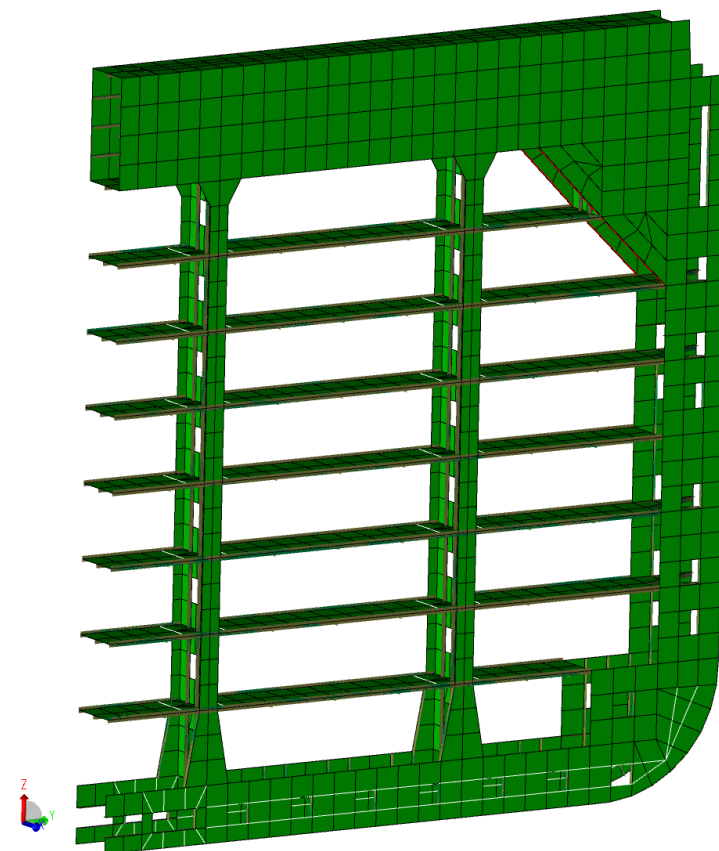
- In GeniE, the following mesh settings are available
  - Mesh density; and
  - Number of elements
- These settings can be assigned to the following members

| Mesh property      | Assigned to                                                                          |
|--------------------|--------------------------------------------------------------------------------------|
| Mesh density       | <ul style="list-style-type: none"><li>○ Plates</li><li>○ Stiffeners</li></ul>        |
| Number of elements | <ul style="list-style-type: none"><li>○ Stiffeners</li><li>○ Feature edges</li></ul> |

- The user may specify a default mesh density that is used, unless other mesh settings are assigned locally

Meshing on a typical pillar bulkhead with:

- Default mesh density 864 mm
- Feature edges (in white)
- 3 elements assigned to 4 stiffeners (in red)



# Meshing in GeniE

## Mesh Editing tools giving control on the mesh

- After generating the mesh, there are various mesh editing options available
- User can work interactively and edit the mesh without making changes to the geometry model
- The Mesh Editing tools are normally not needed for standard geometries, but may be beneficial to highly complex geometries where the user needs full control on the mesh, e.g. move node
- Numerous Mesh Edit options are available and further described in the user documentation

### Mesh editing options in GeiE

 Manipulate Triangle (Alt+Shift+T)

 Move Node (Alt+Shift+W)

 Align Nodes in Element Grid (Alt+Shift+A)

 Align Nodes Along Line and Move by Constant Vector (Alt+Shift+L)

 Align Nodes Along Line and Move by Constant Vector (Dialog) (Alt+Shift+M)

 Split Edge (Alt+Shift+E)

 Collapse an Edge (Alt+Shift+C)

 Remove Node (Alt+Shift+R)

 Add Plate Element (Alt+Shift+P)

 Remove Element (Alt+Shift+Q)

 Enhance Quality (Alt+Shift+X)

 Refine Grid (2) (Alt+Shift+2)

 Refine Grid (3) (Alt+Shift+3)

 Refine Grid Inclusive (2) (Alt+Shift+F2)

 Refine Grid Inclusive (3) (Alt+Shift+F3)

 Refine Edge (2)

 Refine Edge (3) (Alt+Shift+Del)

 Refine Box (Alt+Shift+B)

 Imprint Node (Alt+Shift+D)

 Show Limitations Dialog (Alt+Shift+V)

 Show Delete Results and Loads Dialog (Alt+Shift+F)

### Detailed descriptions in the User Manual, in this case

#### Move Node

[GeniE V8.2-04 Documentation » GeniE User Documentation »](#)

[9. FE Mesh and Element Types » 9.5. Mesh Editing » 9.5.2. Move Node](#)

[Previous](#)[Next](#)

#### 9.5.2. Move Node

The *Move Node* (Alt+Shift+W) operation changes the position of nodes. Nodes on surface edges can be moved along the edges and nodes inside surfaces can be moved inside the surface (concept).

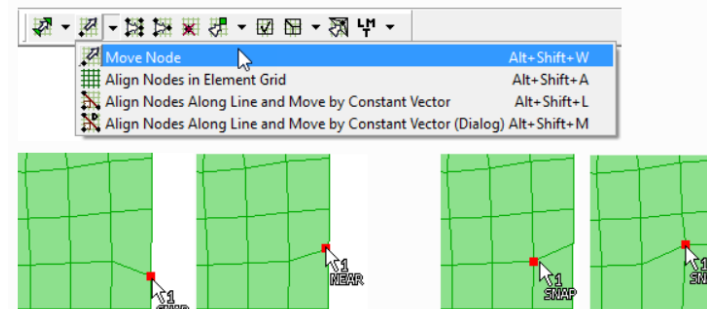


Fig. 9.47 Move a node along an edge and another node inside a plate

[Previous](#)[Next](#)

# Thank you

Ragnar.Eimhjellen@dnv.com

[www.dnv.com](https://www.dnv.com)