

Design gas fuelled container ships with independent prismatic tank

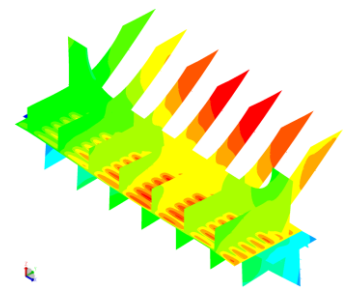
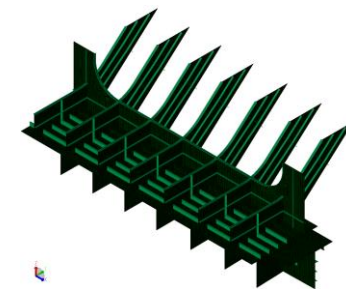
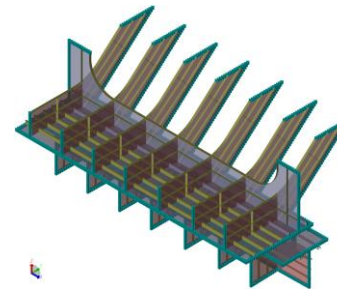
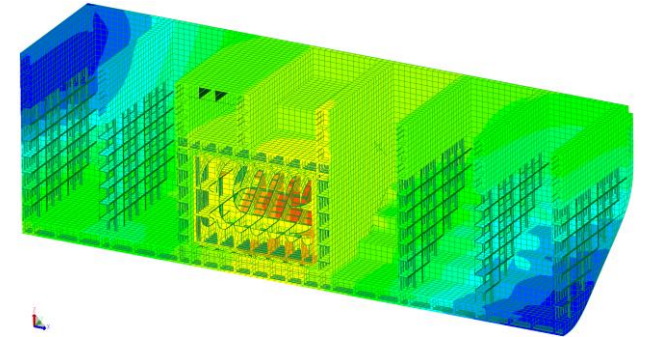
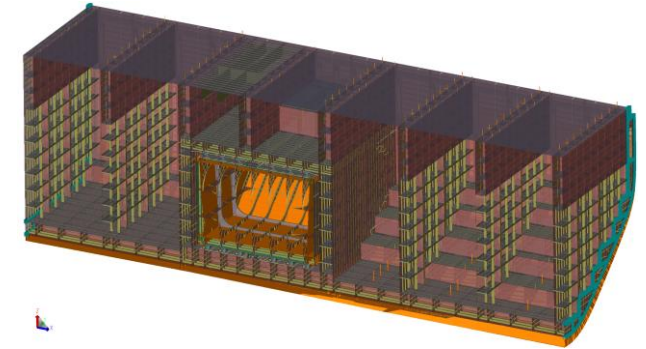
Learn how to efficiently run FE strength assessments of independent prismatic Type-B tank, as required by DNV Rules, using DNV's calculation tools Sesam GeniE and Nauticus Hull

Ragnar Eimhjellen, Product Manager Calculation Tools

25 February 2022

Agenda

- Cargo hold finite element analysis
 - Modelling of vertical supports and procedure for transmitting friction forces
 - Modelling of contact elements in stoppers accounting for initial clearances
 - Automatic creation of loading conditions as required by the Rules
 - Verification of analysis results (distribution of forces iwo tank supports)
- Local strength analysis – fine mesh model of supports
 - Creating a sub-model with refined mesh and with automatic load transfer
- FE fatigue assessment of a typical detail – bracket toe
 - Refine mesh in cargo hold finite element model
 - Creation of temperature loads for low cycle fatigue
 - Run fatigue strength assessment (combination of high cycle fatigue and low cycle fatigue)
- GeniE customized for ships
 - Gas fuelled container ships – tutorial available
 - Ship configuration of GeniE
 - Digital sales
- Q&A session

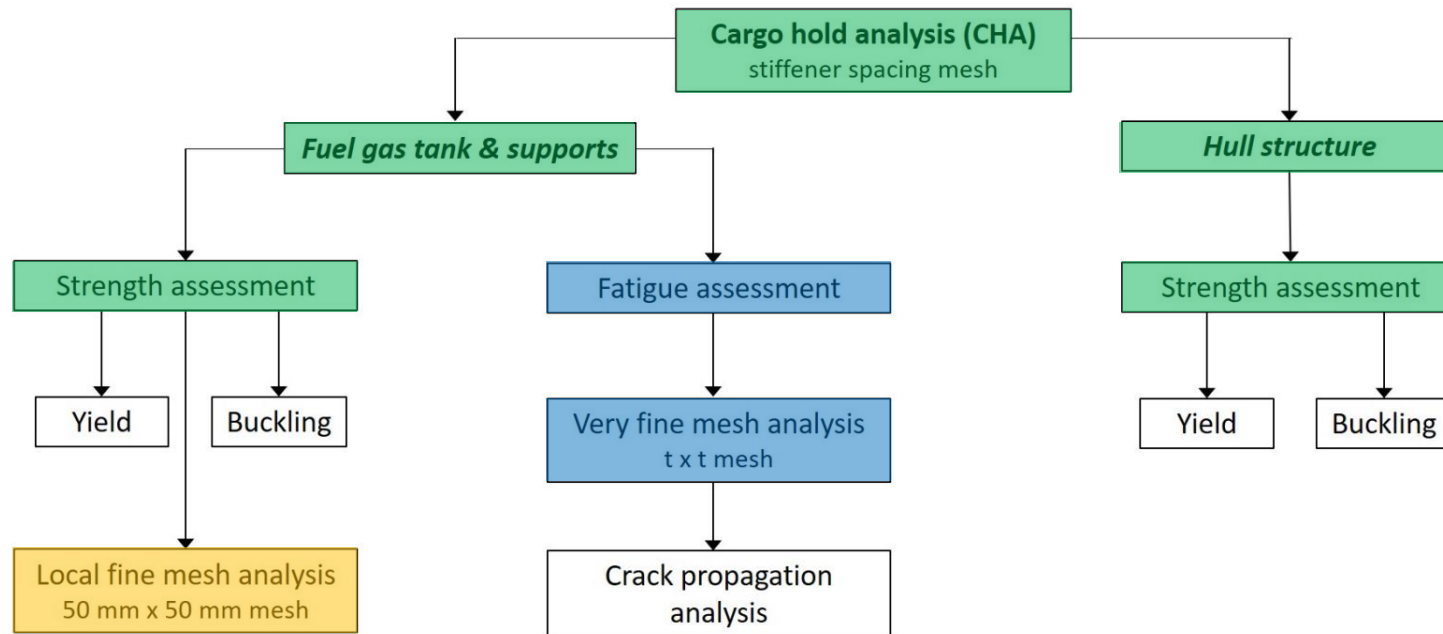


Gas fuelled CV with independent prismatic tank

Gas fuel tank finite element analysis scope

- Analysis scope described in DNV-CG-0554 *Gas fuelled container ship with independent prismatic tanks type-A and type-B*
- DNV's calculation tools Sesam GeniE and Nauticus Hull supports the minimum required analysis scope
- In this presentation, we will cover most of the analysis scope

Minimum required finite element analysis scope for a design with a type-B tank



Relation to agenda item in this presentation

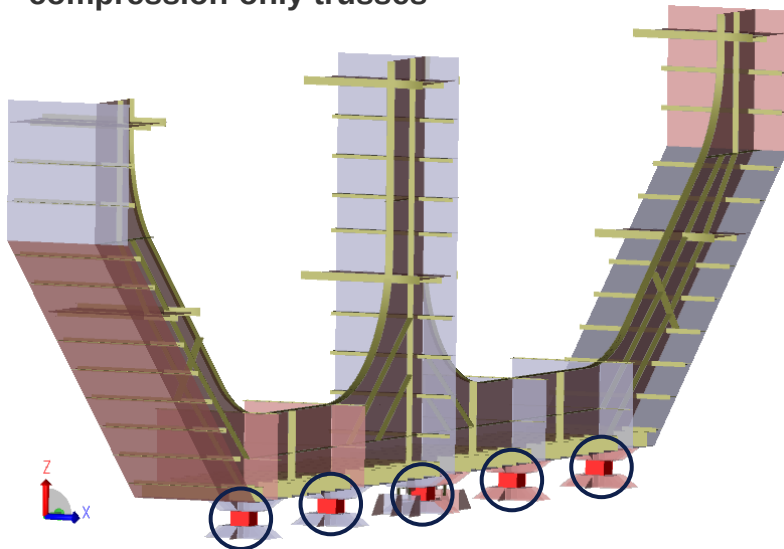
- Agenda item: Cargo hold finite element analysis
- Agenda item: Local strength analysis – fine mesh model of supports
- Agenda item: FE fatigue assessment of a typical detail – bracket toe

Cargo hold finite element analysis

Modelling of vertical supports

- Wooden chocks are modelled as compression-only truss elements (no shear stiffness, no bending stiffness, and no tensile stiffness)
- Friction forces in vertical wooden supports will be represented by a force couple in the analysis
- Method automatically accounted for in GeniE by *Tank Support Coupling Forces* tool (TSCF tool)

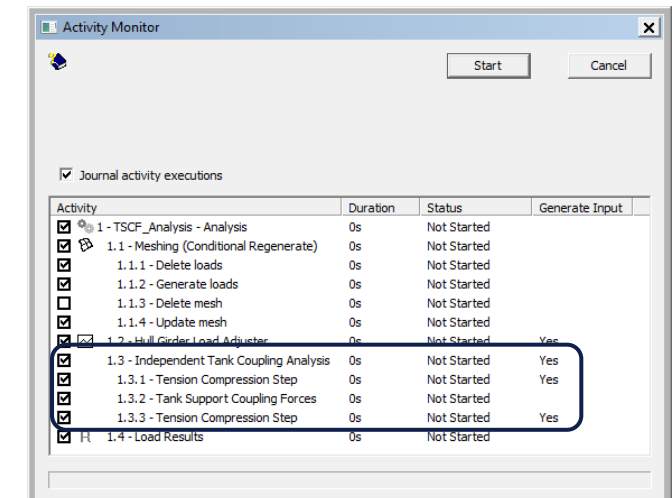
Wooden chocks:
compression-only trusses



Friction forces in vertical supports:
force couple



Tool for friction forces controlled
by GeniE's activity monitor

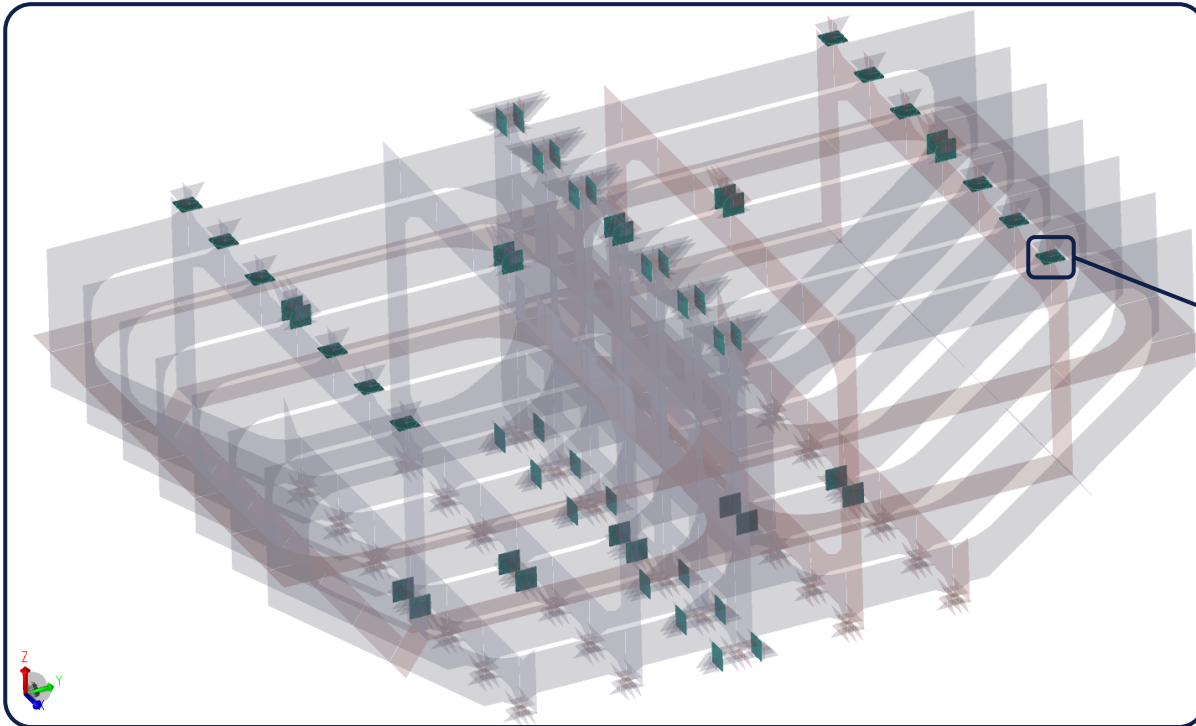


Cargo hold finite element analysis

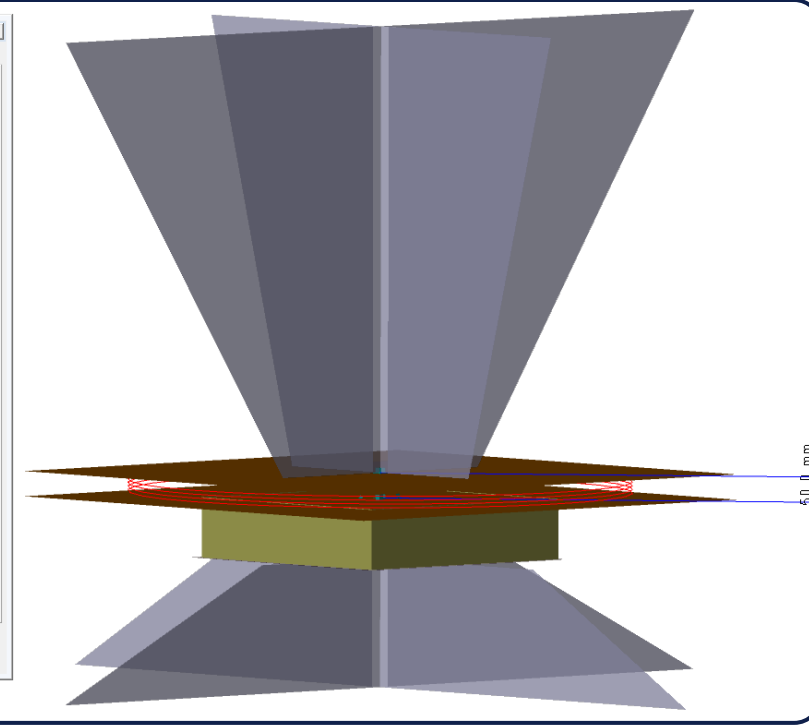
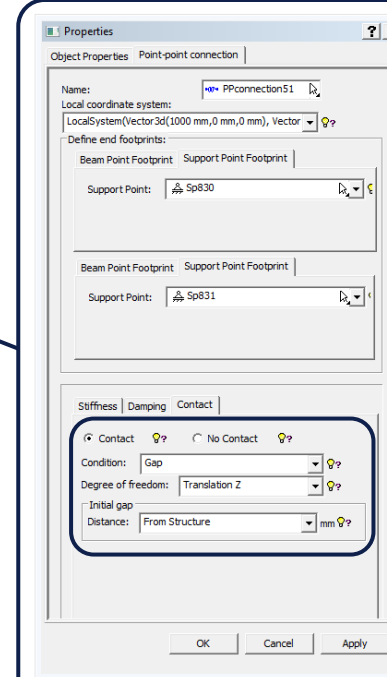
Modelling of initial clearances (gaps)

- Initial clearances in anti-roll, anti-pitch, and anti-floatation supports are accounted for in GeniE by the Point-Point connection feature
- 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



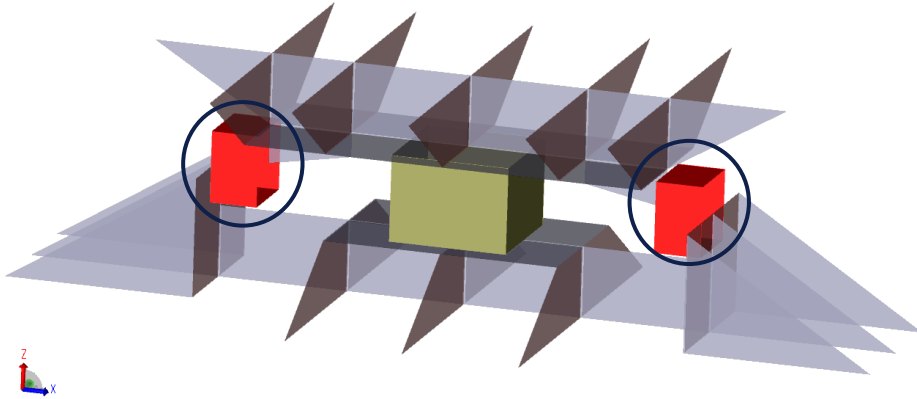
Initial clearance can be specified or taken from the modelled gap



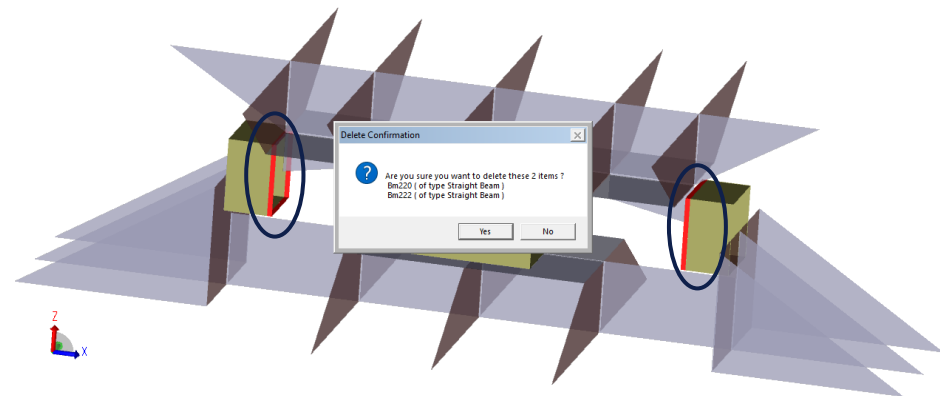
Cargo hold finite element analysis

Modelling of contact elements accounting for initial clearances

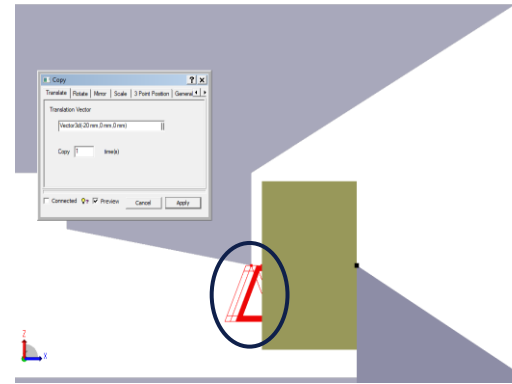
Step 1: Create a beam (not truss) connecting the tank to the supporting structure



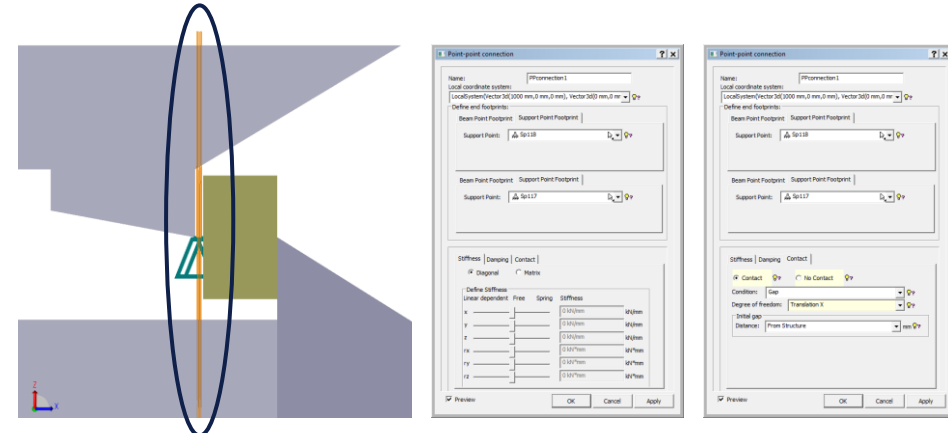
Step 2: Split the beam, representing the initial clearances



Step 3: Create support points (free in all degrees of freedom) in way of the gap



Step 4: Connect the support points by using the point-point connection feature



- ## FE load template for strength assessment

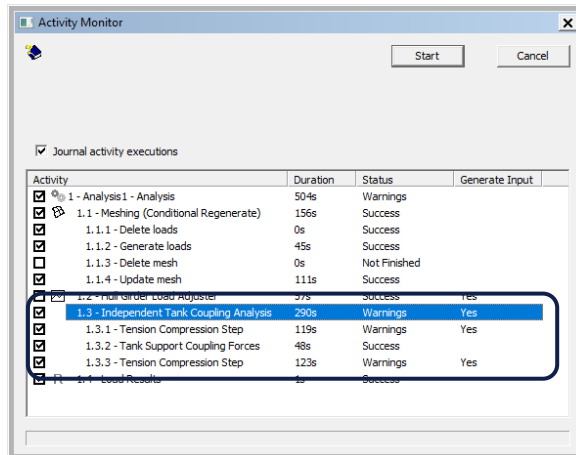
Detailed loading information can be inspected and further tuned



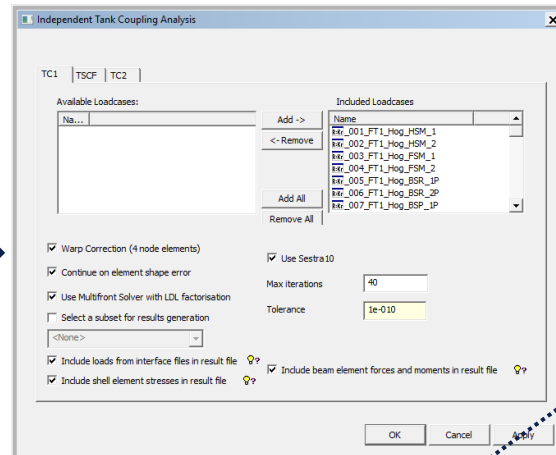
Cargo hold finite element analysis

TSCF tool – Graphical User Interface

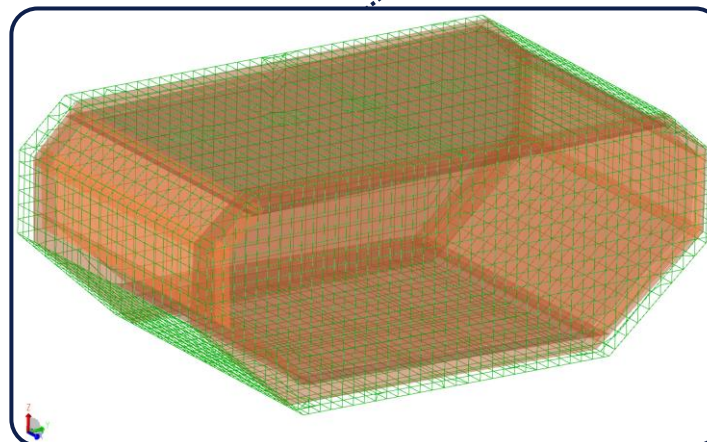
Analysis set-up in GeniE



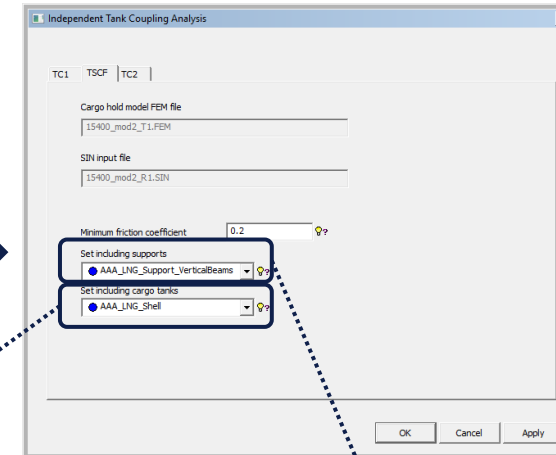
Step 1: T/C analysis



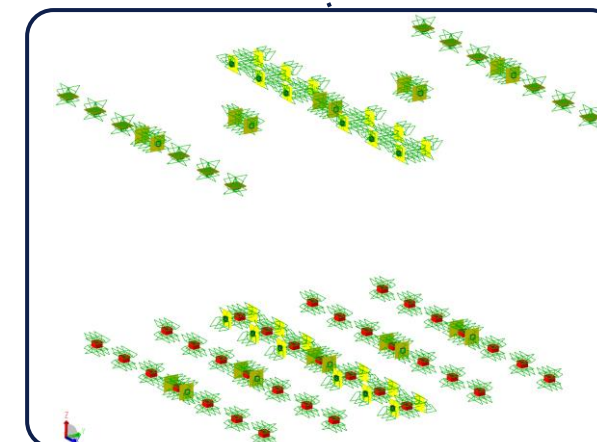
Set containing tank boundary, for computing resulting force from tank pressures



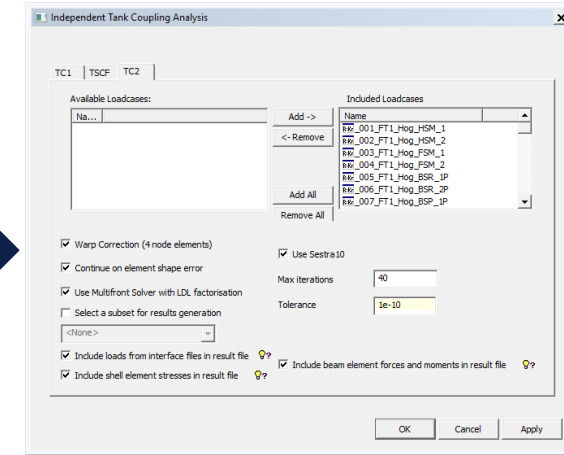
Step 2: TSCF Compute and apply horizontal friction forces



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



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

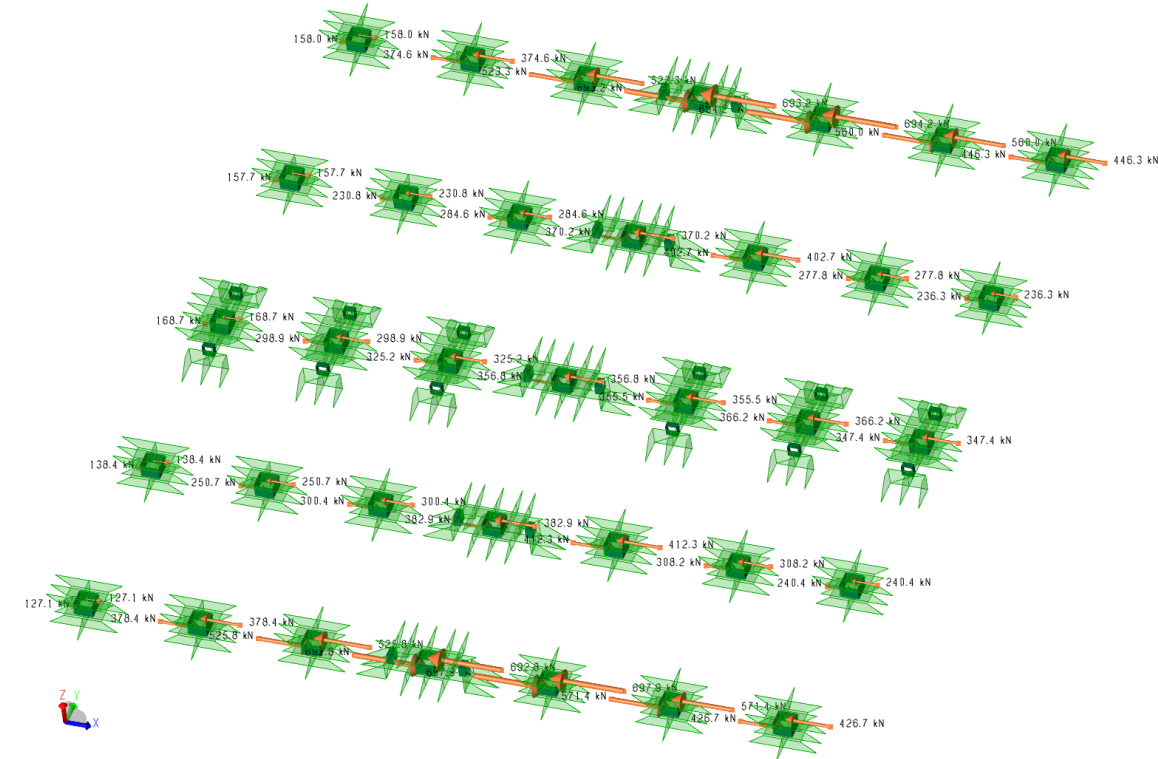


Cargo hold finite element analysis

TSCF tool – Computation of friction forces in vertical supports

1. Run T/C analysis
2. Run TSCF 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



Cargo hold finite element analysis

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**

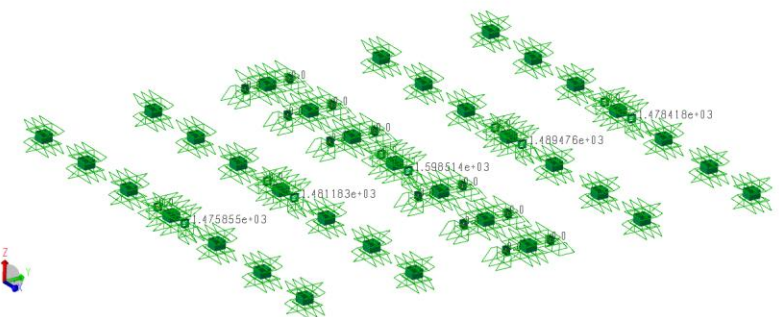
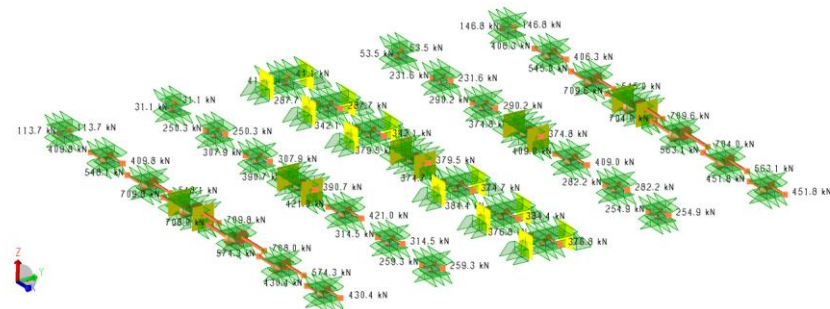
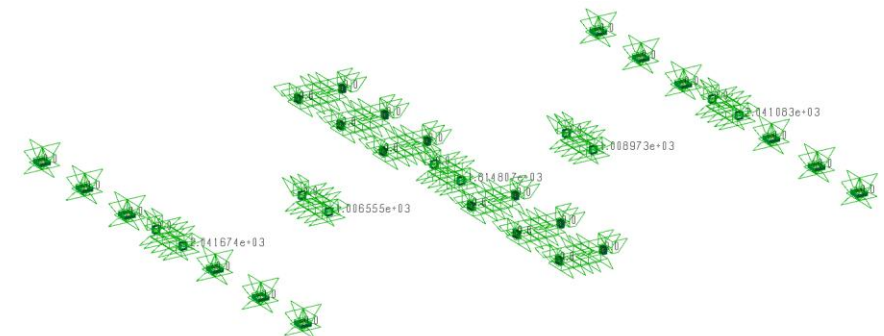
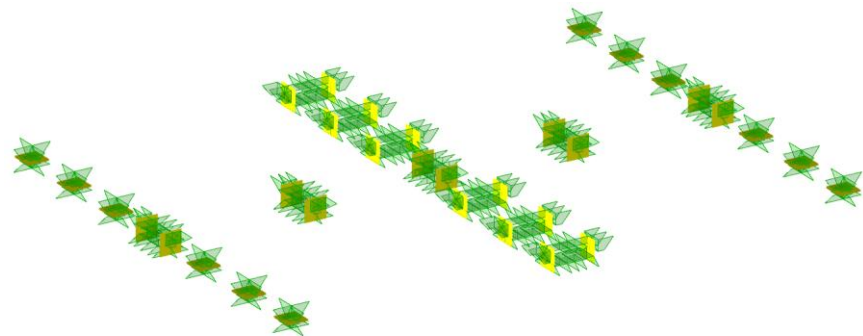
```
log_tscf.txt - Notepad
File Edit Format View Help

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
```



Cargo hold finite element analysis

Analysis results for flooded case – distribution of vertical forces

- Sum of vertical pressures acting on tank - tank's self-weight = forces in anti-floatation supports
- Self-weight can be found from a load case where the tank is not floating, being the difference between vertical forces in supports, and sum of vertical pressures in tank

Sum of pressures on independent tank
116 910 kN

Self-weight =
Vertical force in supports – sum of vertical pressures

Forces taken in anti-floating supports
108 549 kN

76 547 kN – 68 186 = 8 361 kN

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

Loading combination: _022_ST7_Hog_Flooded_2

Horizontal resultant force due to tank pressures:
X: -2,01535933E-003
Y: -3,60887498E-009
XY: 2,01535933E-003

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

Vertical resultant force due to tank pressures:
Z: 1,16989743E+008
```

```
*Untitled - Notepad
File Edit Format View Help

Loading combination: _001_ST1_Hog_HSM_1

Horizontal resultant force due to tank pressures:
X: 1,04259678E+006
Y: -1,54250301E-008
XY: 1,04259678E+006

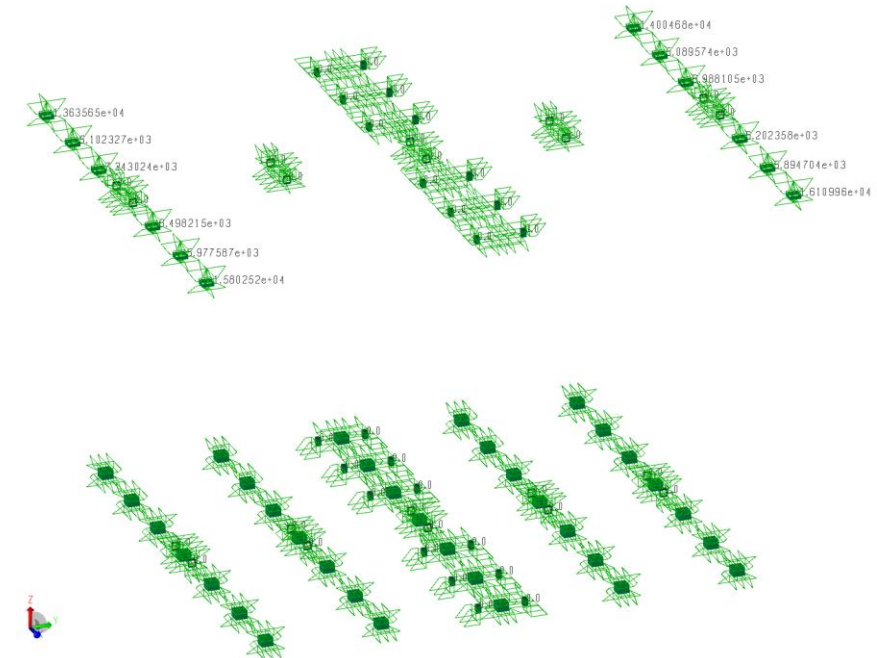
Horizontal unit vector giving direction for applied coupling forces:
X: 1,000
Y: 0,000

Vertical resultant force due to tank pressures:
Z: -6,81863088E+007

-----Tank coupling analysis result-----
Sum of horizontal forces transferred in way of vertical supports:
X: 1,04259678E+006
Y: 0,00000000E+000
XY: 1,04259678E+006

Sum of axial forces in vertical supports:
(from tank pressures and tank self weight)
Z: -7,65473520E+007

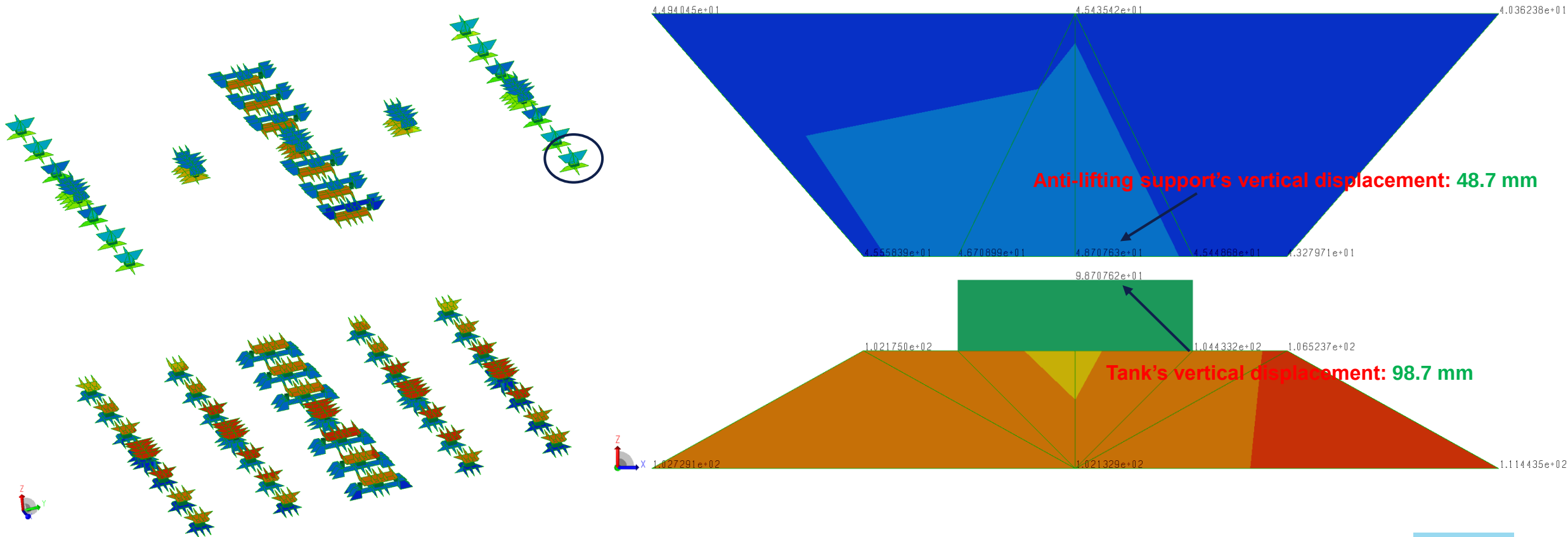
-----End of tank coupling analysis result-----
```



Cargo hold finite element analysis

Analysis results for flooded case – displacements in anti-flotation support

- Difference in vertical displacements identical to the modelled initial clearances: in this case (98.7 mm – 48.7 mm = 50.0 mm)

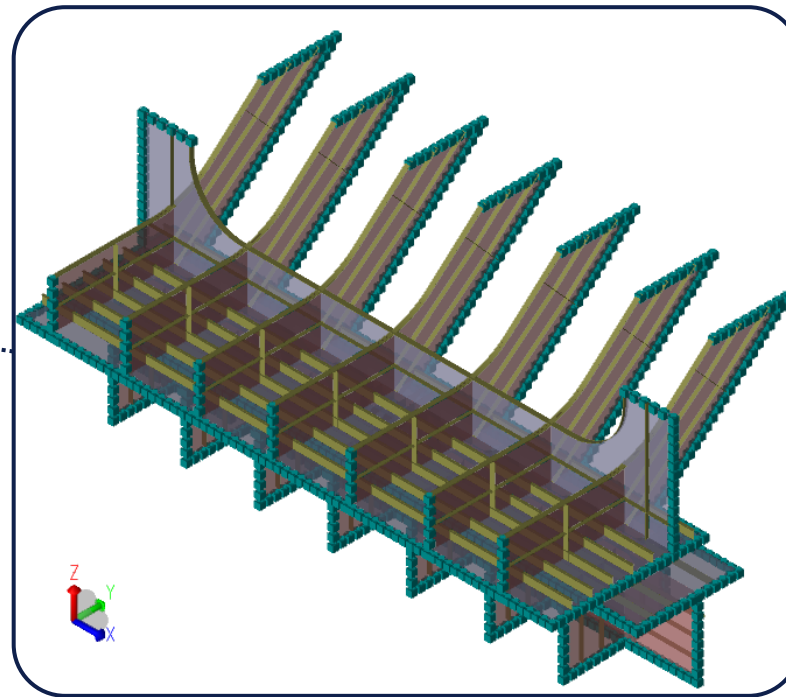
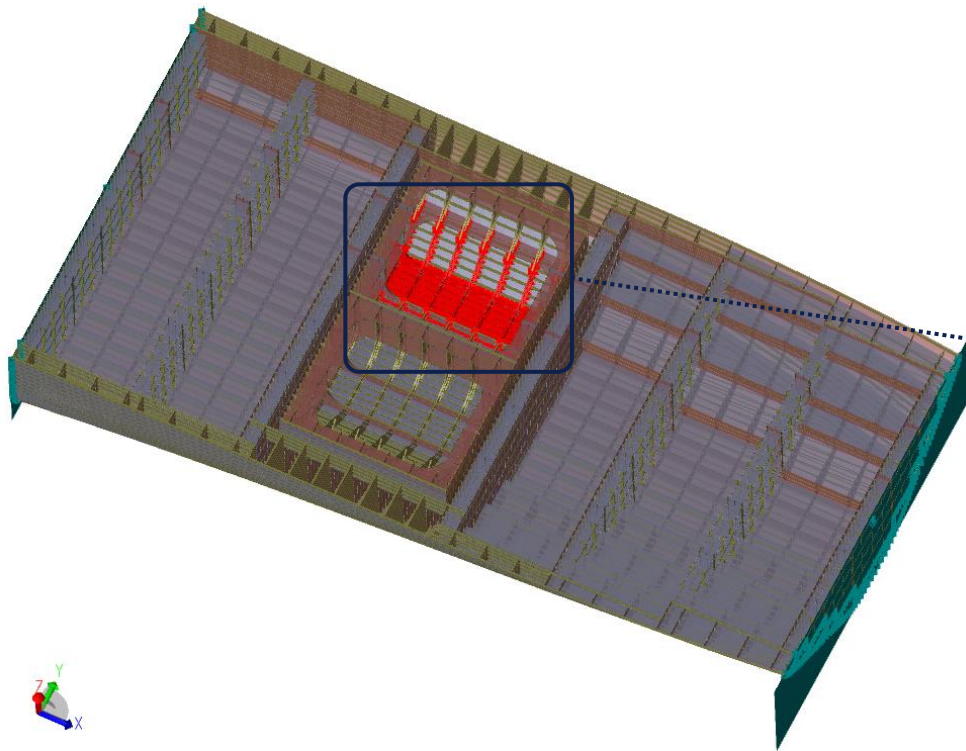


Demo: Cargo hold finite element analysis

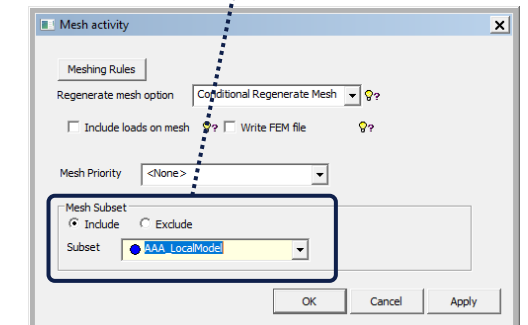
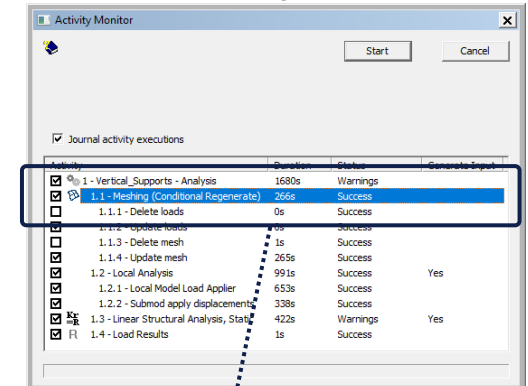
Local strength analysis – fine mesh model of supports

Create a sub-model containing vertical supports

- As a user, you can select to run parts of a cargo hold model as a sub-model (with refined mesh)
- Create a set containing the members that shall be part of the sub-model, and assign prescribed boundary conditions to the sub-model ends
- When running the analysis, you may select to mesh only the set containing the sub-model



In GeniE's Activity Monitor, mesh sub-model only

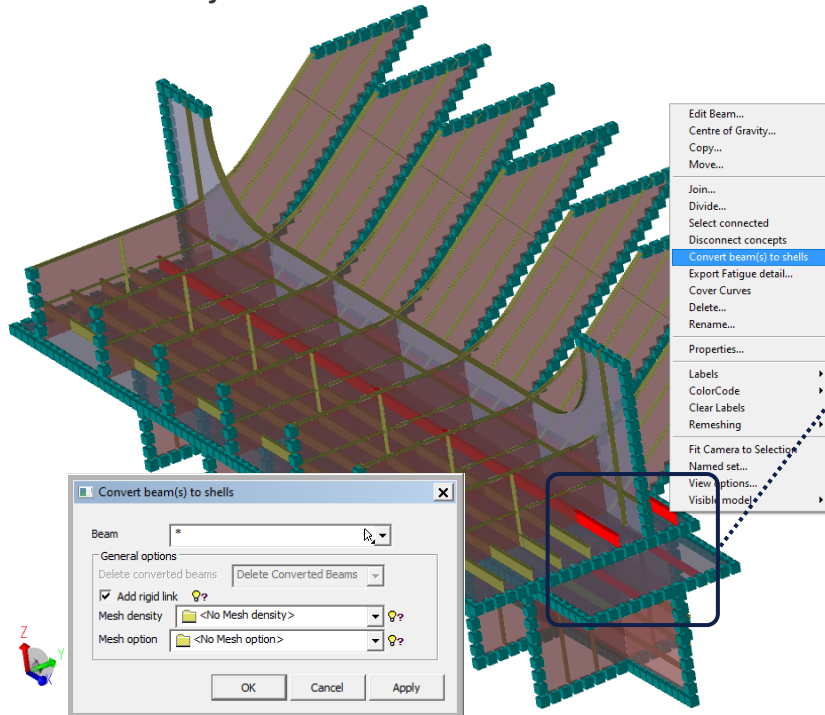


Local strength analysis – fine mesh model of supports

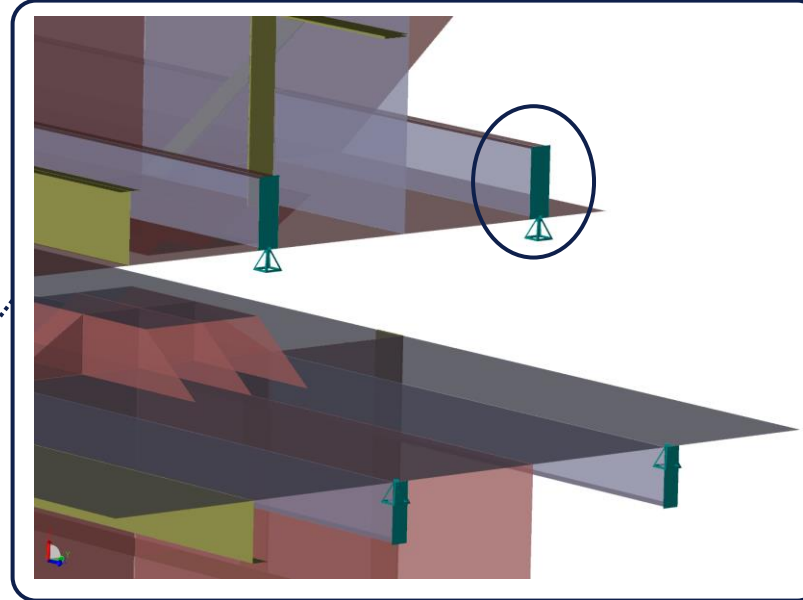
Refine sub-model's mesh

- For stiffeners close to vertical supports, convert beams to shell elements
- Specify 50x50 mm as default mesh density

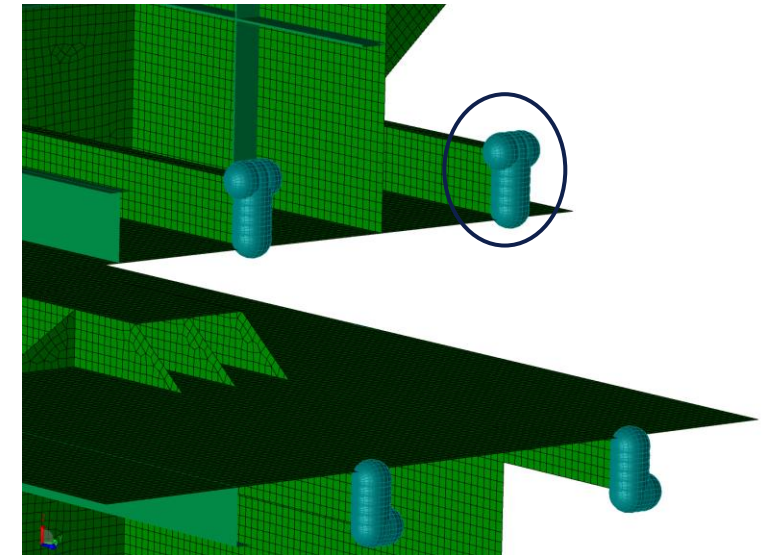
Stiffeners represented as beams can automatically be converted to shell elements



Beams converted to shell elements, with ridged link



Stiffeners represented as shell elements in way of result target area

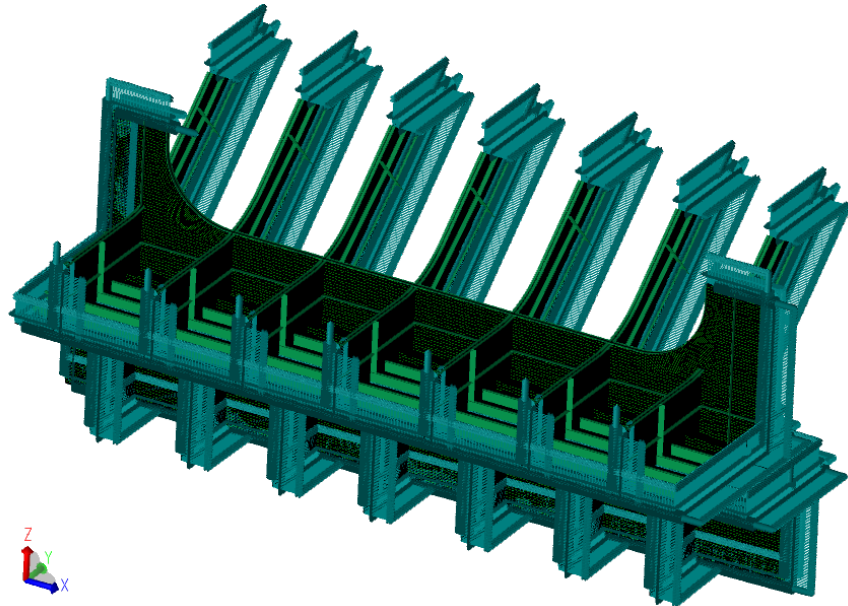


Local strength analysis – fine mesh model of supports

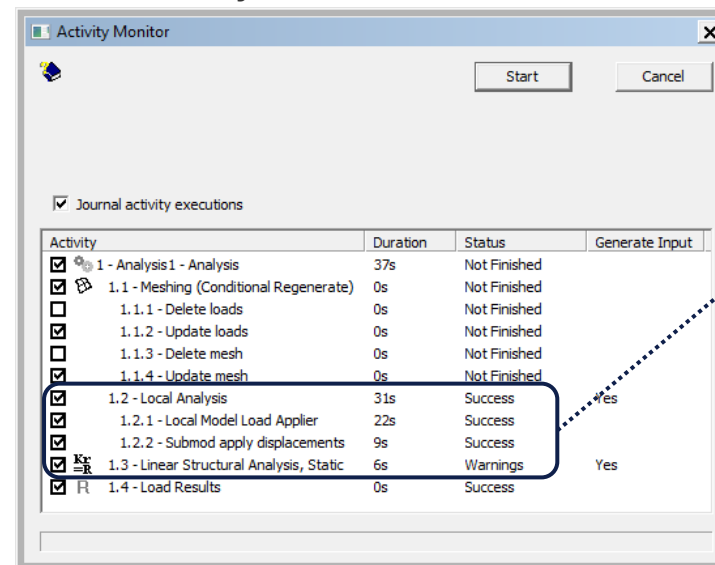
Run analysis of the sub-model

- GeniE's Activity Monitor has a set-up for Local Analysis of sub-models, 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

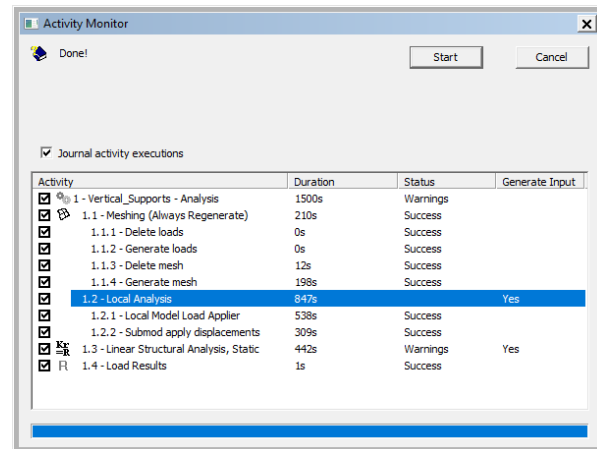


- **Step 1.2.1 – Local Model Load Applier**
distributing forces, pressures and temperature loads from cargo hold model to sub-model
- **Step 1.2.2 – Submod apply displacements**
distributing displacements from cargo hold model to prescribed boundary conditions in sub-model
- **Step 1.3 – Linear Structural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

Local strength analysis – fine mesh model of supports

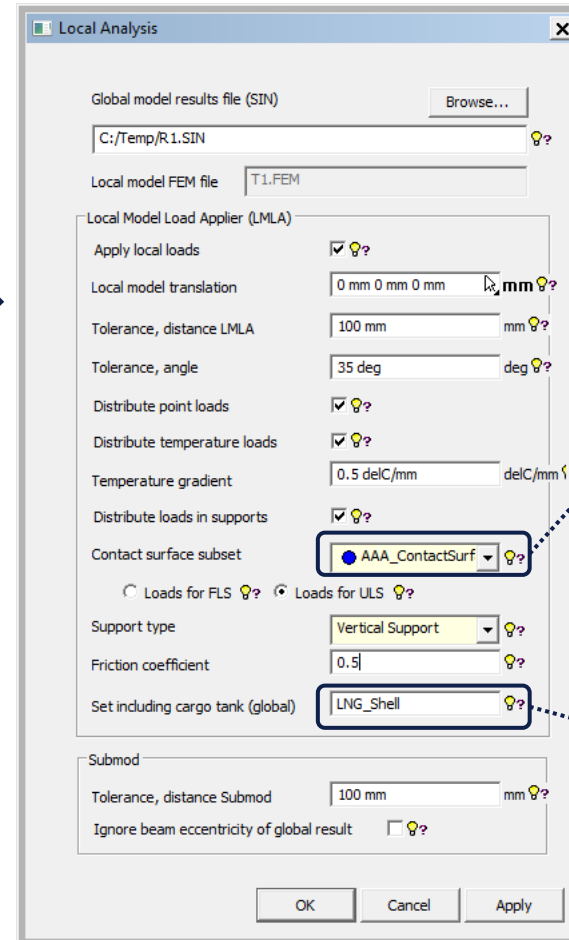
LMLA tool – Graphical User Interface

Analysis set-up in GeniE



Activity	Duration	Status	Generate Input
1 - Vertical_Supports - Analysis	1500s	Warnings	
1.1 - Meshing (Always Regenerate)	210s	Success	
1.1.1 - Delete loads	0s	Success	
1.1.2 - Generate loads	0s	Success	
1.1.3 - Delete mesh	12s	Success	
1.1.4 - Generate mesh	198s	Success	
1.2 - Local Analysis	847s		Yes
1.2.1 - Local Model Load Applier	538s	Success	
1.2.2 - Submod apply displacements	309s	Success	
1.3 - Linear Structural Analysis, Static	442s	Warnings	Yes
1.4 - Load Results	1s	Success	

Local Analysis



Global model results file (SIN)

Local model FEM file

Local Model Load Applier (LMLA)

Apply local loads ☒

Local model translation

Tolerance, distance LMLA

Tolerance, angle

Distribute point loads ☒

Distribute temperature loads ☒

Temperature gradient

Distribute loads in supports ☒

Contact surface subset

☐ Loads for FLS ☒ Loads for ULS

Support type

Friction coefficient

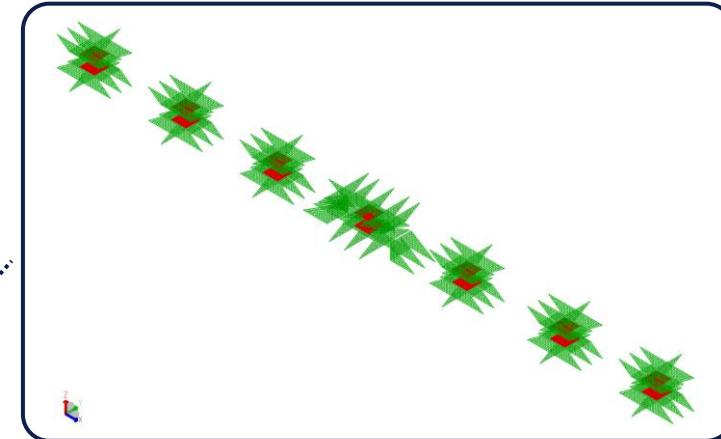
Set including cargo tank (global)

Submod

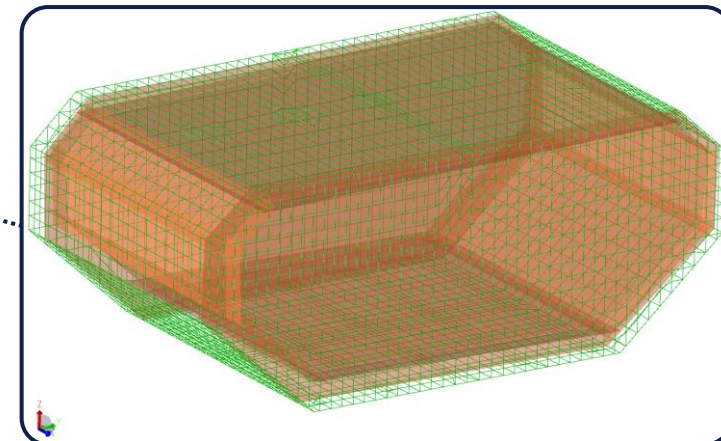
Tolerance, distance Submod

Ignore beam eccentricity of global result ☐

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

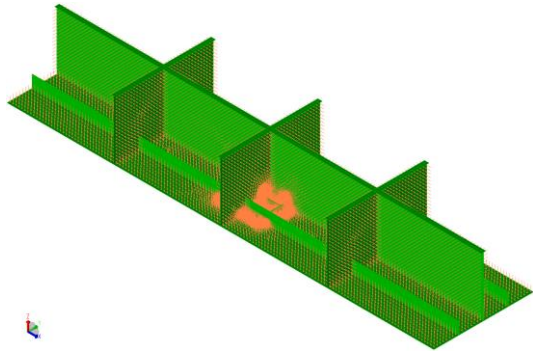


Local strength analysis – fine mesh model of supports

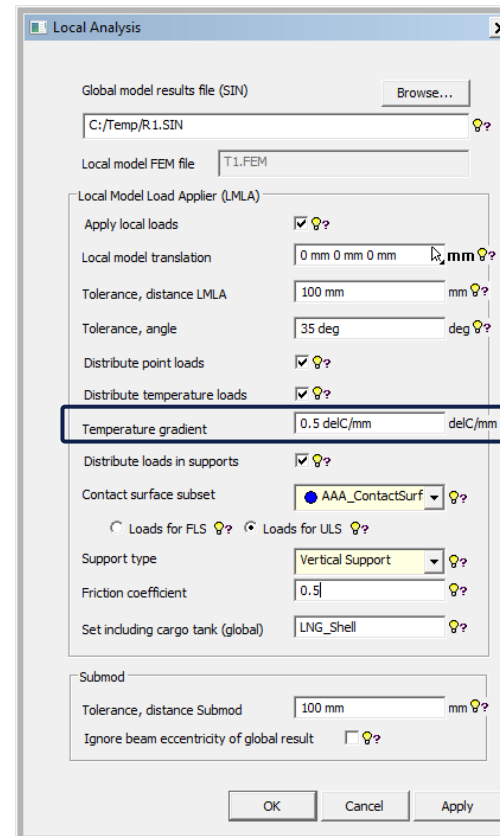
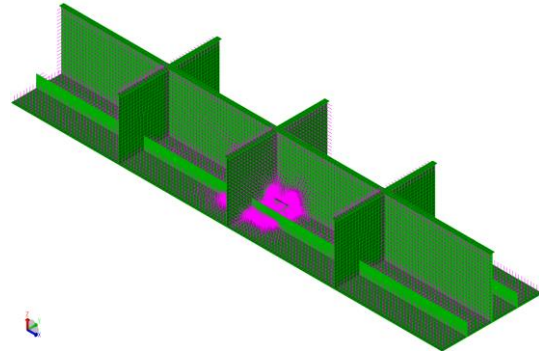
LMLA tool – Distribution of tank loads

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

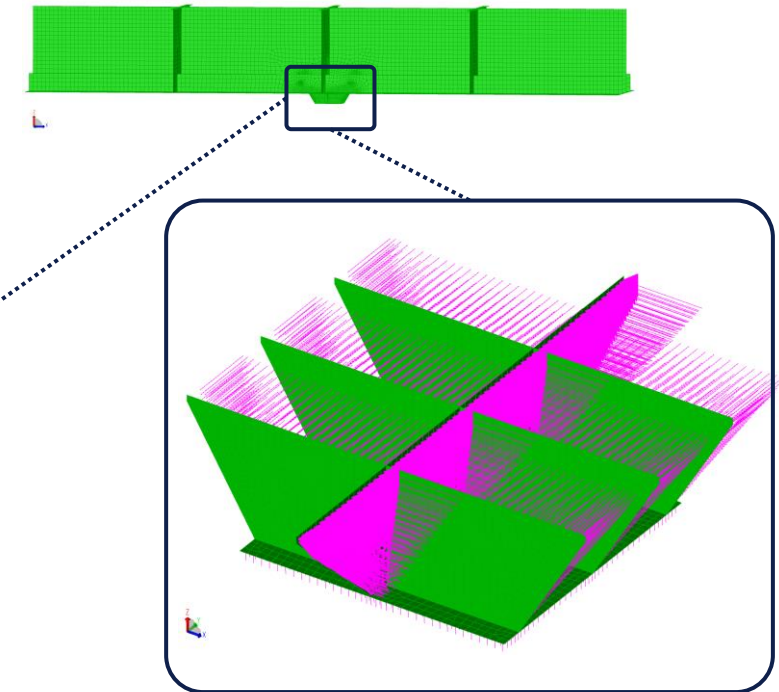
Pressures transferred from global model to sub-model



Temperature loads transferred from global model to sub-model



Temperature loads distributed to supports outside tank boundary

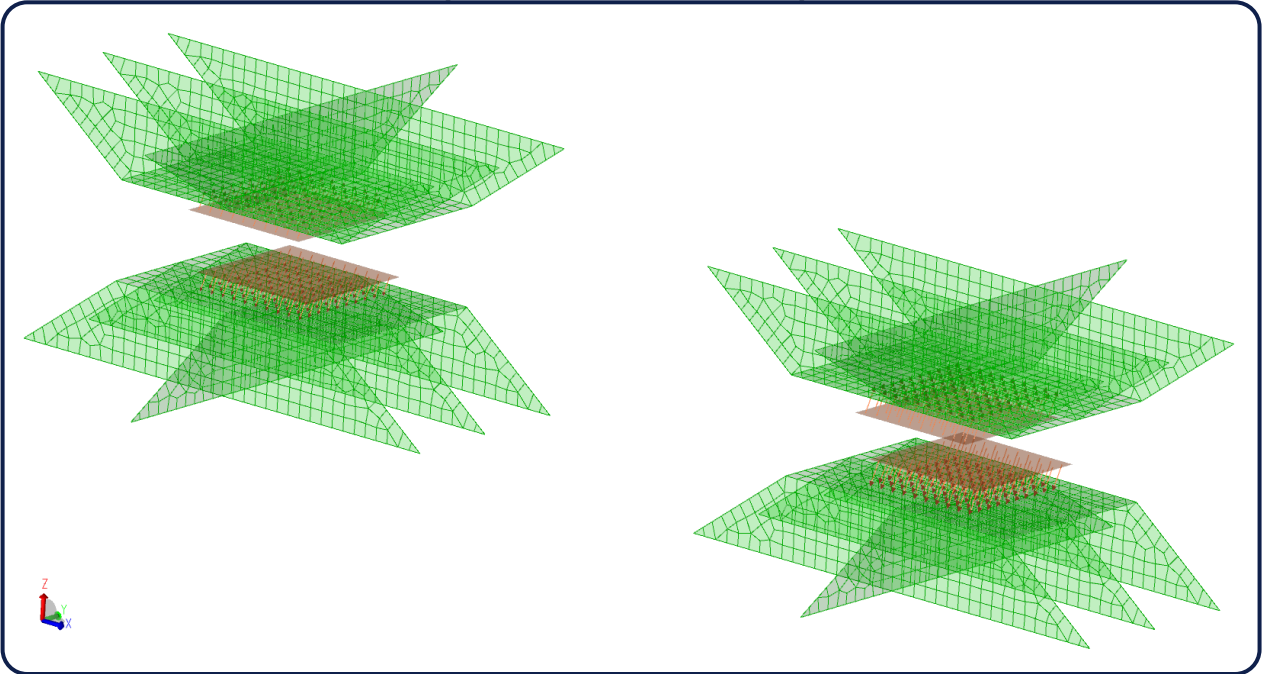


Local strength analysis – fine mesh model of supports

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

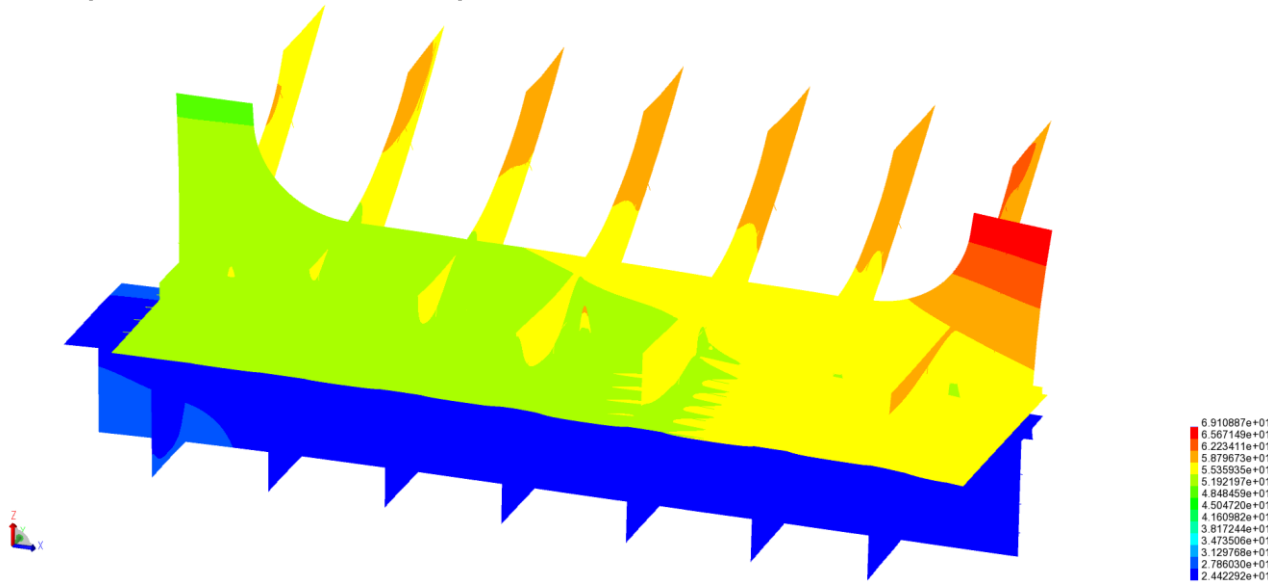


Local strength analysis – fine mesh model of supports

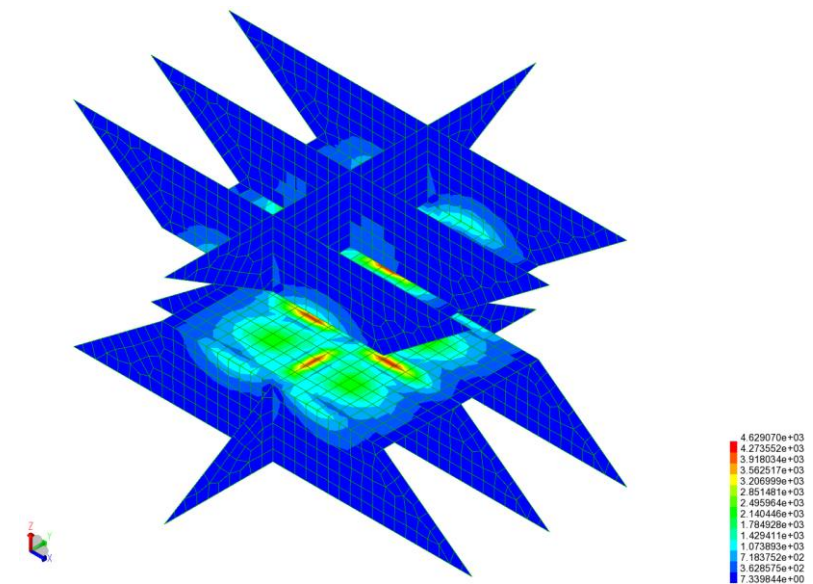
Result presentation in GeniE

- Results required for fine mesh yielding check are available in GeniE
 - Displacements, and deformation plots that can be toggled on/off
 - Gaussian von-Mises membrane stress

Displacements and deformation plots available in GeniE



Gaussian membrane von-Mises stresses in GeniE



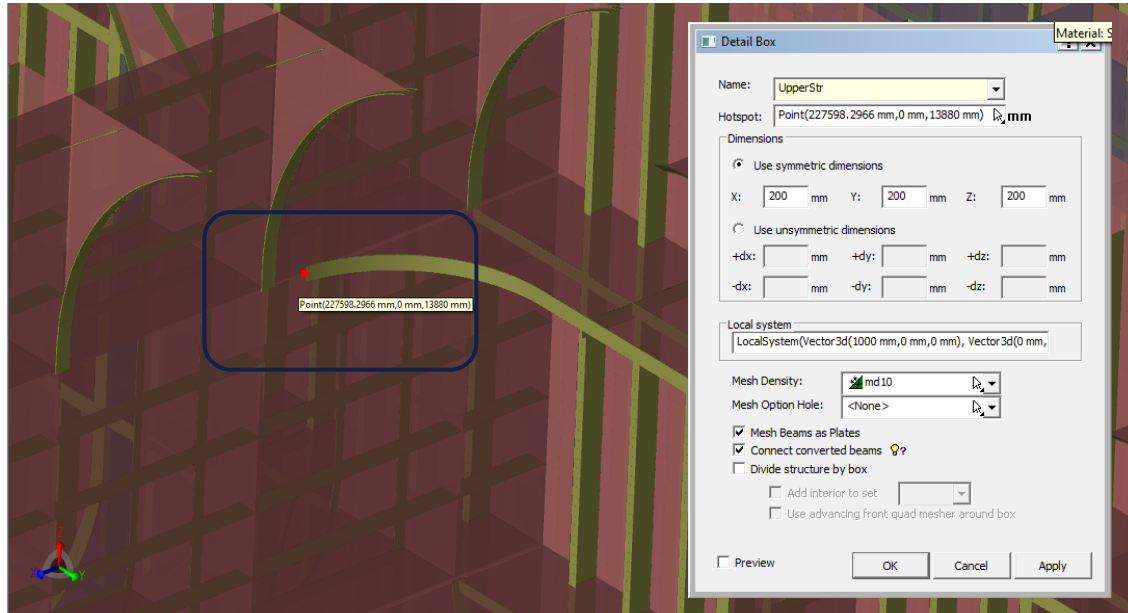
Demo: Local strength analysis – fine mesh model of supports

FE fatigue assessment of a typical detail – bracket toe

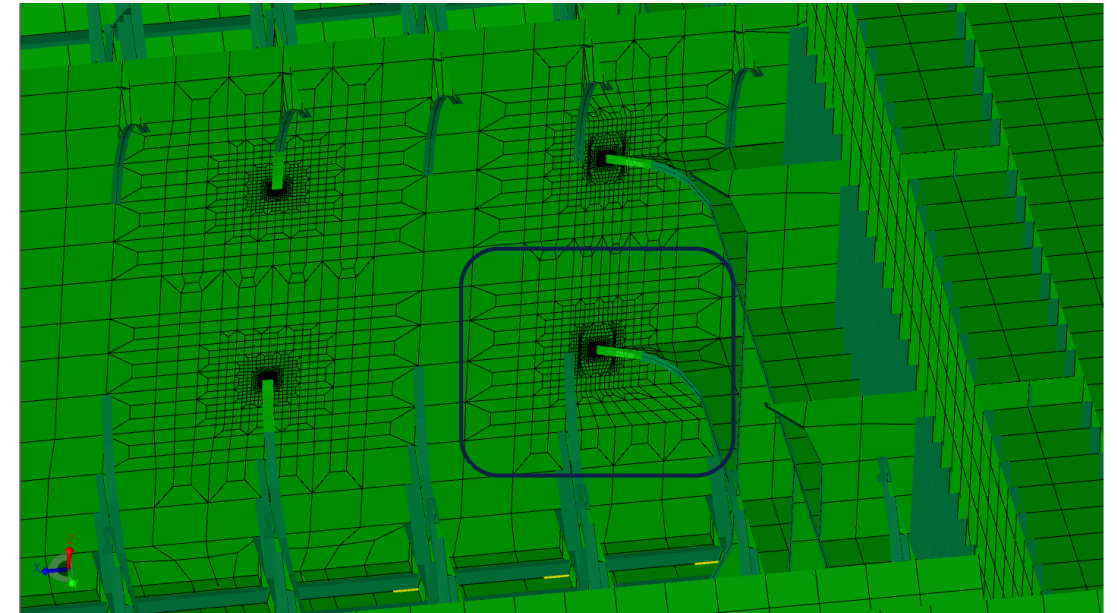
Mesh refinement

- Within a cargo hold model, the user can use the detailed box feature to specify structural details where the mesh shall be refined
- The detailed box feature can also automatically convert beams to shell elements
- 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

Select hot spot and specify refinement area and mesh size



GeniE will refine the mesh and ensure proper mesh transition



FE fatigue assessment of a typical detail – bracket toe

FE load template for Fatigue assessment

- In the FE load template for fatigue assessment, user can apply motions and accelerations from a wave load analysis
- The user can add temperature loads for the gas fuel tank, that shall be used for low cycle fatigue assessments

FE load template for fatigue assessment

Fatigue load combinations - Container carriers

LC	Loading pattern	Description	Draught (m)	Kr (m)	GM (m)
FT1		100 % filled gas fuel tank [I]	T design	0.39 B	0.11 B Load cases (10)
FT2		75 % filled gas fuel tank [II]	T design	0.39 B	0.11 B Load cases (10)
FT3		50 % filled gas fuel tank [III]	T design	0.39 B	0.11 B Load cases (10)
FT4		25 % filled gas fuel tank [IV]	T design	0.39 B	0.11 B Load cases (10)

☒ Add user defined accelerations:

a_surge (m/s ²)	a_sway (m/s ²)	a_heave (m/s ²)	a_pitch (rad/s ²)	pitch_angle (deg)	a_roll (rad/s ²)	roll_angle (deg)
[I] 0.14	0.264	0.506	0.005	1.175	0.004	2.612
[II] 0.14	0.264	0.506	0.005	1.175	0.004	2.612

Containers in hold

Container height [8'6" (Standard)]

No of containers in stack (LANG bey) 4

No of containers in stack (typical) 11

Fatigue container loads 40 ft 22 (tFEU)

Containers on deck areas and hatch covers

Container height [8'6" (Standard)]

No of containers in stack 11

No of stacks in width 20

Fatigue container loads 40 ft 22 (tFEU)

Fuel gas

Density 0.500 t/m³

Vapour pressure 70 kPa

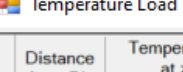
■ 20' containers ■ Water ballast □ Empty

■ 40' containers ■ Fuel gas

Detailed loading information can be inspected and further tuned

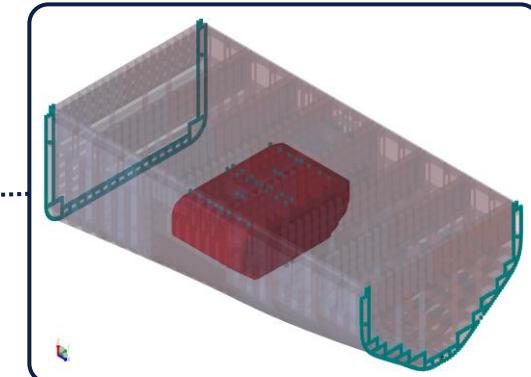


Additional Low cycle fatigue cases (pressure and temp loads) can be created in Nauticus Hull



Distance from BL (mm)	Temperature at z-position (deg Celsius)
0	-165
8090	-165
8590	-135
19670	-135
*	

OK Cancel

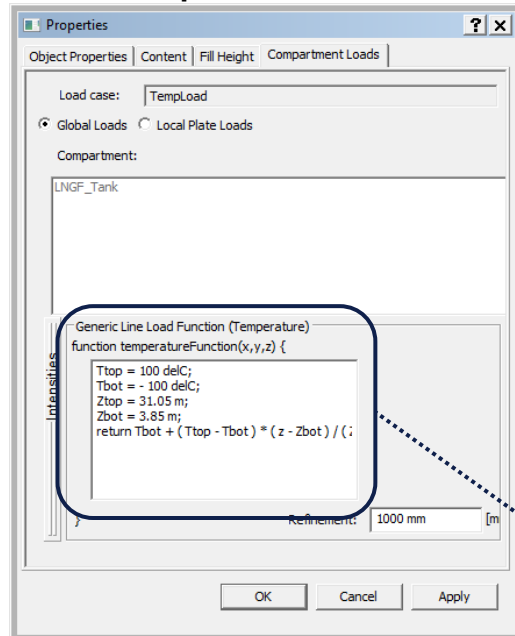


FE fatigue assessment of a typical detail – bracket toe

Temperature loads as required for low cycle fatigue assessment

- Temperature loads can be assigned to compartments
- When meshing the temperature loads are automatically mapped to the finite elements (same as for surface loads)
- The temperature loads are automatically distributed to shell elements and beam elements

Temperature assigned to the compartment



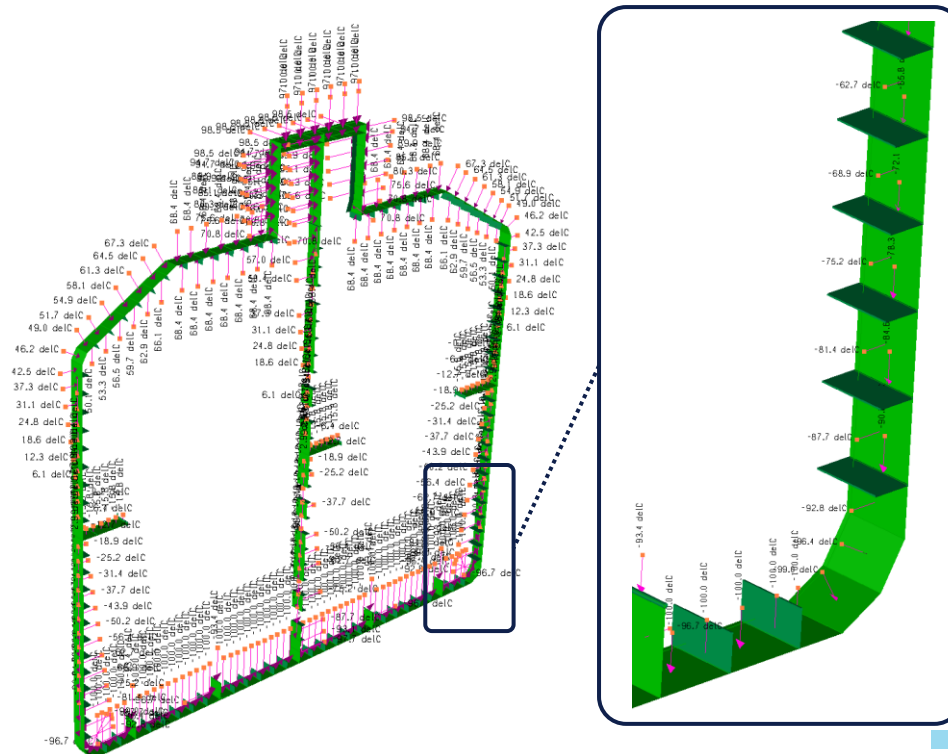
Temperature loads represented as a JS function

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

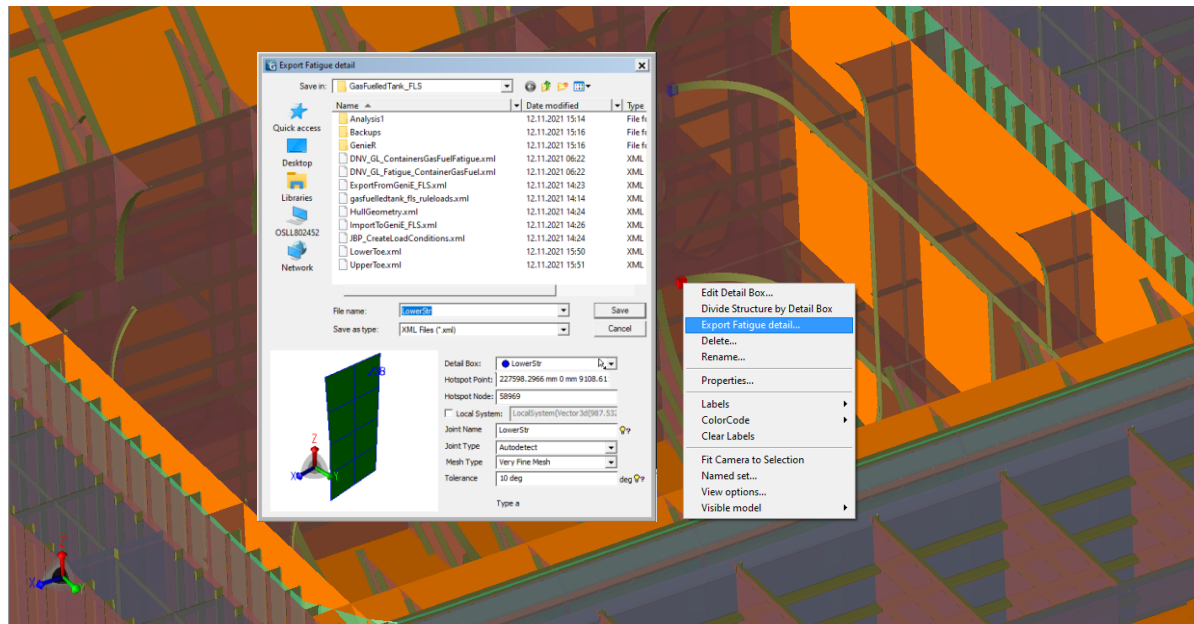


FE fatigue assessment of a typical detail – bracket toe

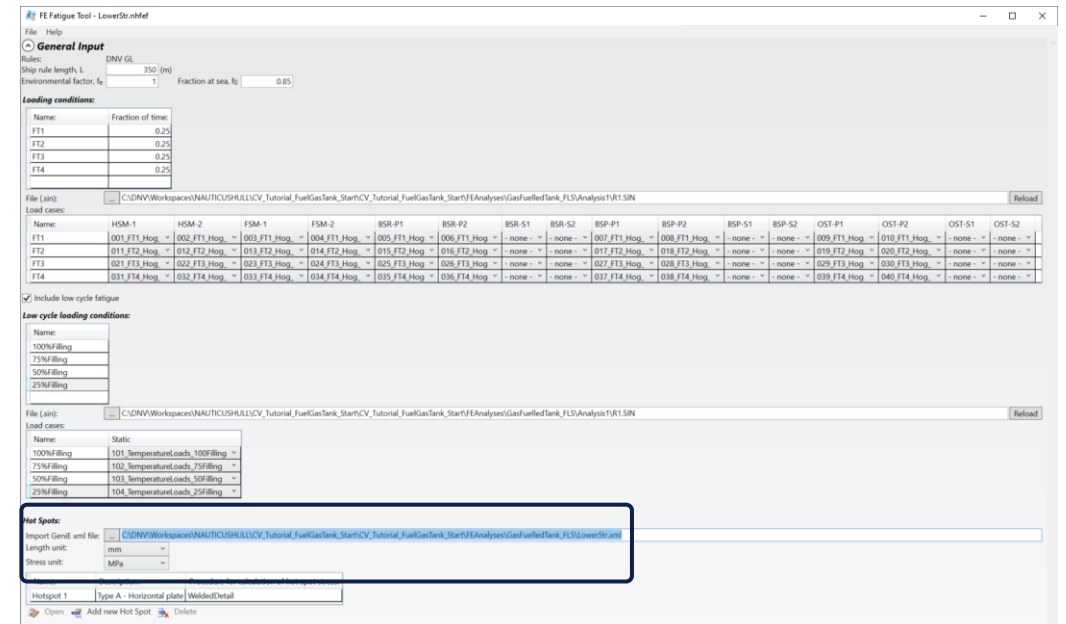
Export information to the FE Fatigue tool

- GeniE can export an XML file with finite element information (element numbers iwo hot spot, hot spot type, etc)
- This XML file can be imported by the FE Fatigue tool for effective computation of high cycle fatigue and low cycle fatigue

GeniE exports XML file with hot spot information



FE Fatigue tool imports XML file with hot spot information



FE fatigue assessment of a typical detail – bracket toe

FE Fatigue assessment by FE Fatigue tool

- Following results are available
 - I. Fatigue strength results (damage) for high cycle fatigue and low cycle fatigue
 - II. Hot spot stress for fracture mechanics
 - III. Hot spot stress for crack propagation analysis

Hot spot stress for Crack propagation analysis

Hot spot stress for fracture mechanics

FE Fatigue strength calculations

[illegible][illegible]

Hot spot location - Z

Other welded details - type 'a' - hot spot stress for fracture mechanics analysis

Input

Elements

Upper and lower are according the normal vector specified in Gen1

Upper surface

Lower surface

Stress

Result

Calculate hot spot stress

Intermediate results:

Loading condition	HSM-1	HSM-1	HSM-2	HSM-2	F5M-1	F5M-1	F5M-2	F5M-2	BSR-1P	BSR-1P	BSR-2P	BSR-2P	BSR-15	BSR-15	BSR-25	BSR-25	BSR-1P	BSR-1P	BSR-2P	BSR-2P	BSR-15	BSR-15	BSR-25	BSR-25	OST-1P	OST-1P	OST-2P	OST-2P	OST-15	OST-15	OST-25	OST-25
	α bending	α bending	α bending	α bending	α bending	α bending	α bending	α bending	α bending	α bending	α bending	α bending	α bending	α bending	α bending	α bending	α bending	α bending	α bending	α bending	α bending	α bending	α bending	α bending	α bending	α bending	α bending	α bending	α bending	α bending	α bending	
F11, direction 1	86.7	34.1	94.1	31.2	101.1	35.4	87.7	30.9	115.4	41.8	87.8	30.1	0.0	0.0	0.0	0.0	108.8	38.1	90.1	31.7	0.0	0.0	0.0	0.0	98.1	34.4	93.4	31.5	0.0	0.0	0.0	
F11, direction 2	5.8	8.4	4.8	8.3	5.2	8.8	4.5	7.7	6.2	10.2	4.2	7.4	0.0	0.0	0.0	0.0	5.7	9.5	4.4	7.8	0.0	0.0	0.0	0.0	5.8	8.6	4.7	8.1	0.0	0.0	0.0	
F12, direction 1	94.2	33.1	91.7	32.1	99.5	35.0	86.1	30.5	107.6	37.6	79.3	27.6	0.0	0.0	0.0	0.0	100.9	35.7	85.0	29.7	0.0	0.0	0.0	0.0	95.6	33.6	90.5	31.6	0.0	0.0	0.0	
F12, direction 2	4.8	8.2	4.7	8.0	5.0	8.7	4.4	7.5	5.7	9.5	3.8	6.8	0.0	0.0	0.0	0.0	5.3	8.9	4.2	7.3	0.0	0.0	0.0	0.0	4.9	8.3	4.4	7.8	0.0	0.0	0.0	
F13, direction 1	103.3	36.3	103.3	36.3	107.8	37.9	99.0	34.8	119.0	42.2	80.6	27.9	0.0	0.0	0.0	0.0	113.4	40.2	87.3	30.4	0.0	0.0	0.0	0.0	105.4	37.1	100.5	35.3	0.0	0.0	0.0	
F13, direction 2	5.2	9.0	5.2	9.0	5.5	9.4	5.0	8.6	6.3	10.5	3.8	6.8	0.0	0.0	0.0	0.0	6.0	10.0	4.2	7.4	0.0	0.0	0.0	0.0	5.4	9.2	5.1	8.7	0.0	0.0	0.0	
F14, direction 1	127.8	44.6	122.7	43.4	128.5	45.1	125.5	44.1	138.8	45.4	109.1	38.1	0.0	0.0	0.0	0.0	126.1	44.4	114.5	40.1	0.0	0.0	0.0	0.0	120.1	42.3	116.4	44.4	125.4	44.0	0.0	0.0
F14, direction 2	6.4	11.0	6.2	10.7	6.5	11.2	6.3	10.9	6.7	11.3	5.4	9.4	0.0	0.0	0.0	0.0	6.5	11.0	5.7	9.9	0.0	0.0	0.0	0.0	6.4	11.0	6.3	10.9	0.0	0.0	0.0	

Message List

Errors

Warnings

Information

Description

The stress read out point is inside the midpoint of the inner element. Read out point: 5.0, inner midpoint: 5.1

FE fatigue assessment of a typical detail – bracket toe

FE Fatigue Strength assessment by FE Fatigue Tool

- FE Fatigue tool automatically reads stresses from a finite element result file
- All stress component for high cycle fatigue and low cycle fatigue available
- Intermediate fatigue results
 - Direction 1 / Direction 2
 - For each EDW: hot spot stress range and mean stress effect
 - For each loading condition: Part damage
- Fatigue damage computation
 - High cycle fatigue
 - Low cycle fatigue
 - Combined high cycle and low cycle fatigue

FE Fatigue tool for fatigue strength calculations

Hot spot location - Hotspot 1

Other welded details - type 'a'

Input

Design basis
 Design life, T_{D} 25 years Time in corrosive environment, T_{CZS} 0 years
 Stress reduction factor, K_D 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 to the normal vector specified in GenIE
☐ Upper surface
☒ Lower surface

Stresses

Import data from SIN file
 Specified minimum yield strength, R_{eH} 235 N/mm² Thickness, t_{a50} 10.0 mm Mesh size 10.0 mm

Stresses:

Ele. No.	Loading condition	Stress	HSM-1	HSM-2	FSM-1	FSM-2	BSR-1P	BSR-2P	BSR-1S	BSR-2S	BSP-1P	BSP-2P	BSP-1S	BSP-2S	OST-1P	OST-2P	OST-1S	OST-2S	Ele. No.	Loading condition	Stress	Static
146238	FT1	σ x surface	180.3	174.5	175.7	177.0	188.4	174.4	0.0	0.0	188.4	171.9	0.0	0.0	176.3	177.6	0.0	0.0	146238	100%Filling	σ x surface	-79.1
146238	FT1	σ y surface	14.7	13.5	15.0	12.9	13.9	15.3	0.0	0.0	13.7	15.1	0.0	0.0	14.3	13.7	0.0	0.0	146238	100%Filling	σ y surface	-22.3
146238	FT1	τ xy surface	30.7	32.7	28.6	34.8	30.3	33.5	0.0	0.0	30.0	33.6	0.0	0.0	30.5	33.2	0.0	0.0	146238	100%Filling	τ xy surface	34.4
146238	FT1	σ x membrane	135.0	130.6	131.6	132.5	140.7	131.0	0.0	0.0	140.7	129.1	0.0	0.0	132.1	133.0	0.0	0.0	146238	100%Filling	σ x membrane	5.2
146238	FT1	σ y membrane	2.2	1.5	2.9	0.6	0.6	3.5	0.0	0.0	0.4	3.5	0.0	0.0	2.1	1.4	0.0	0.0	146238	100%Filling	σ y membrane	7.8
146238	FT1	τ xy membrane	50.0	51.4	47.4	53.8	50.9	52.0	0.0	0.0	50.5	51.8	0.0	0.0	49.4	52.3	0.0	0.0	146238	100%Filling	τ xy membrane	-4.6

Results

Calculate fatigue life time

Combined fatigue life (high and low cycle):

Fatigue Life (years)	Damage D_{LC}	Status	Fatigue Life (years)	Damage D_{LC}	Status	Fatigue Life (years)	Damage D_{LC}	Status
777.4	0.03	OK	777.4	0.03	OK	0.00		

Intermediate results, high cycle fatigue:

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
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	3.8	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 $\Delta \sigma_{\text{RMS}}$	Static $\Delta \sigma_{\text{RMS}}$	Static $\Delta \sigma_{\text{RMS}}$
Direction 1	138.2	1.00	

Damage calculation, high cycle fatigue:

Loading condition	$\Delta \sigma_{\text{RMS}}$	D_{LC}	D_{LC}
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, $\Delta \sigma_{\text{RMS}}$ and the combined hot spot stress ranges for low cycle fatigue, $\Delta \sigma_{\text{RMS}}$, are calculated using the maximum of 'side L' and 'side R' instead of the average value.

Message List:

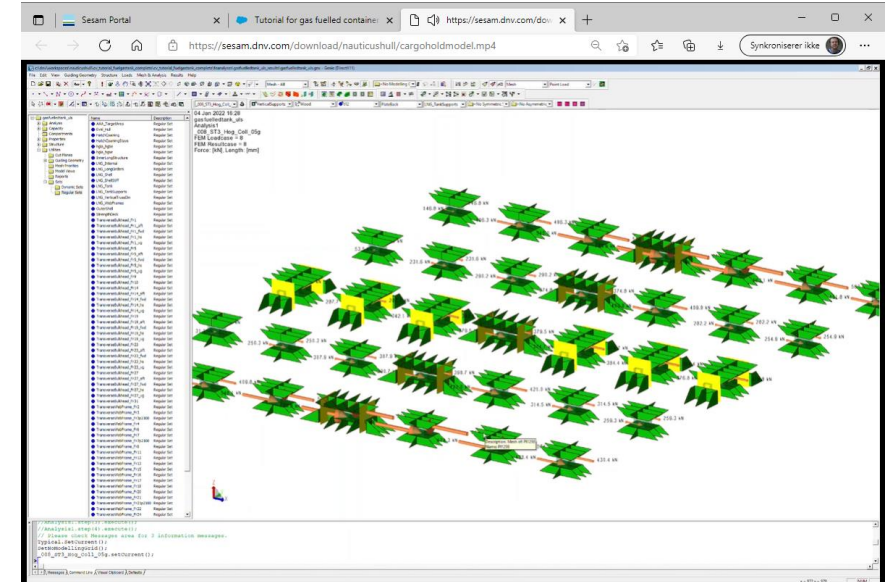
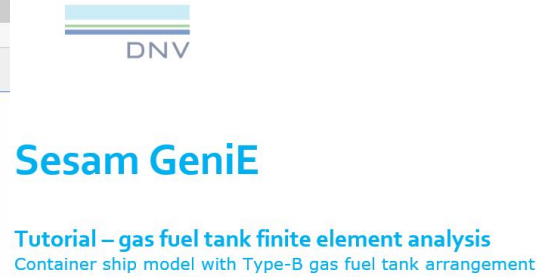
0 Errors 2 Warnings 0 Information

Message	Description
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

Demo: FE fatigue assessment of a typical detail – bracket toe

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



GeniE customized for ships

Ship mode configuration of GeniE

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

Input Units

Output Units

Length

mm

Mass

tonne

Force

N

Angle

rad

Temperature

delC

Time

s

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>>>

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

Full mode

testing

Analysis

Activities

Load Cases

Capacity

Environment

Air

Directions

Soil

Water

Equipment

Weight Lists

Properties

Beam Classification

Beam Types

Compartment Content

Corrosion Addition

Effective Flange

Hinges

Hydro

Load Interfaces

Mass Density Features

Materials

Mesh

Mesh Options

Permeability

Plate Types

Reinforcements

Sections

Slots/Lugs

Sniping properties

Structure Type

Thicknesses

Wet Surface

Structure

Connections

Details

External Matrices

Features

Holes

Point Masses

Slots/Lugs

Supports

Utilities

Cut Planes

Evaluators

Guiding Geometry

Mesh Priorities

Model Views

Reports

Sets

Ship mode

test

Analysis

Activities

Load Cases

Properties

Beam Types

Compartment Content

Corrosion Addition

Materials

Mesh

Mesh Options

Permeability

Plate Types

Sections

Slot/Lug properties

Sniping properties

Structure Type

Thicknesses

Wet Surface

Structure

Details

Features

Slots/Lugs

Supports

Utilities

Cut Planes

Guiding Geometry

Mesh Priorities

Model Views

Reports

Sets

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

Full mode

Rules Meshing

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

Rebuild 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 nonstructural elements

Adjust edge density using ratio

Ratio

0.1

Scantling idealizations

Thickness

Gross

Ignore beam eccentricities

Use co-centric beam

Use based on gross dimensions

Other preferences

Include unused properties

Automatic load combination FEM numbering

Round off Mesh Density

Adjust number of elements

Write loads separately

Eliminate internal eccentricities

Ignore fillet on fillet sections

Prefer linear dependencies

Meshing preferences

Use long loadcase 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

Idealizations

Remove internal vertices

Remove internal edges

Ignore structure not part of subset when idealizing

Use normalized face parameterization

Ship mode

Rules Meshing

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

35 DNV © 25 FEBRUARY 2022

GeniE customized for ships

New and competitive price policy for GeniE ship configuration

Nauticus Hull Prescriptive package

- Includes:
 - Nauticus Hull
 - 3D Beam
 - PULS
- Price policy for yearly rental
 - Node locked: 1 415 Euro
 - Network: Node locked / 0.6 = 2 360 Euro

Nauticus Hull Finite Element package

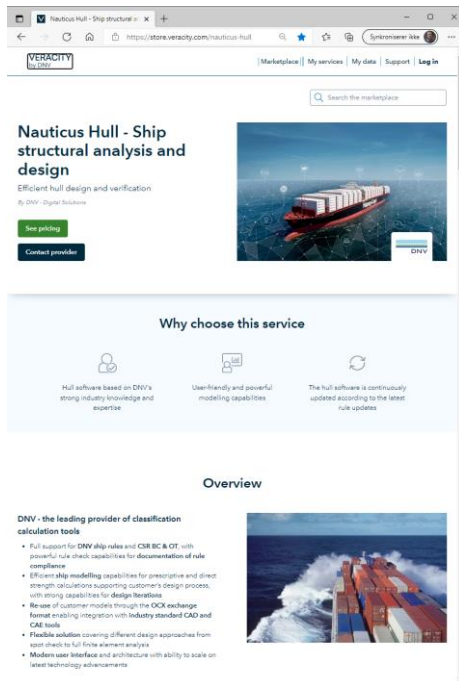
- Includes:
 - **GeniE Ship mode configuration**
 - Sestra (solver, linear and T/C analysis)
 - Submod (tool for transferring prescribed deformations from global mode to sub model)
- Price policy for yearly rental (need NH prescriptive package in addition)
 - Node locked: 6 760 Euro
 - Network: Node locked / 0.6 = 11 270 Euro

GeniE customized for ships

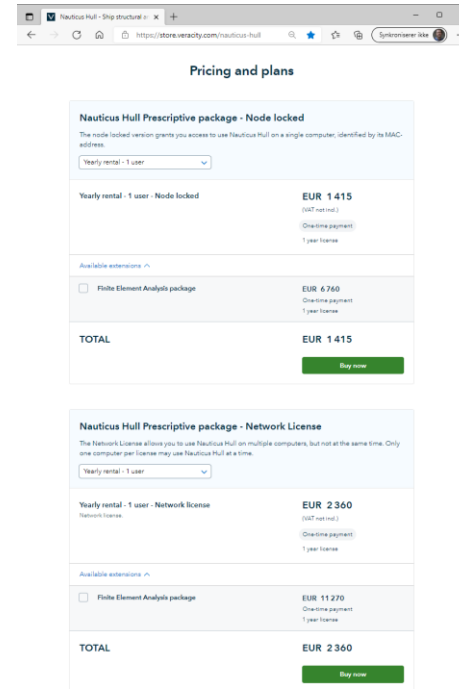
Digital sales of Nauticus Hull

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

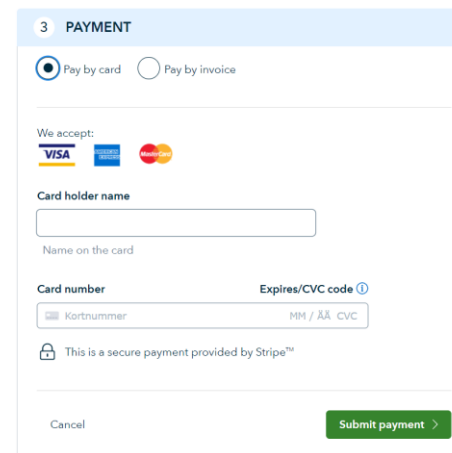
Internet landing page for digital sales of Nauticus Hull



Purchase on the internet



Pay by credit card or invoice



Thank you

Feel free to contact us if you should need any further support

Ragnar.Eimhjellen@dnv.com

www.dnv.com

