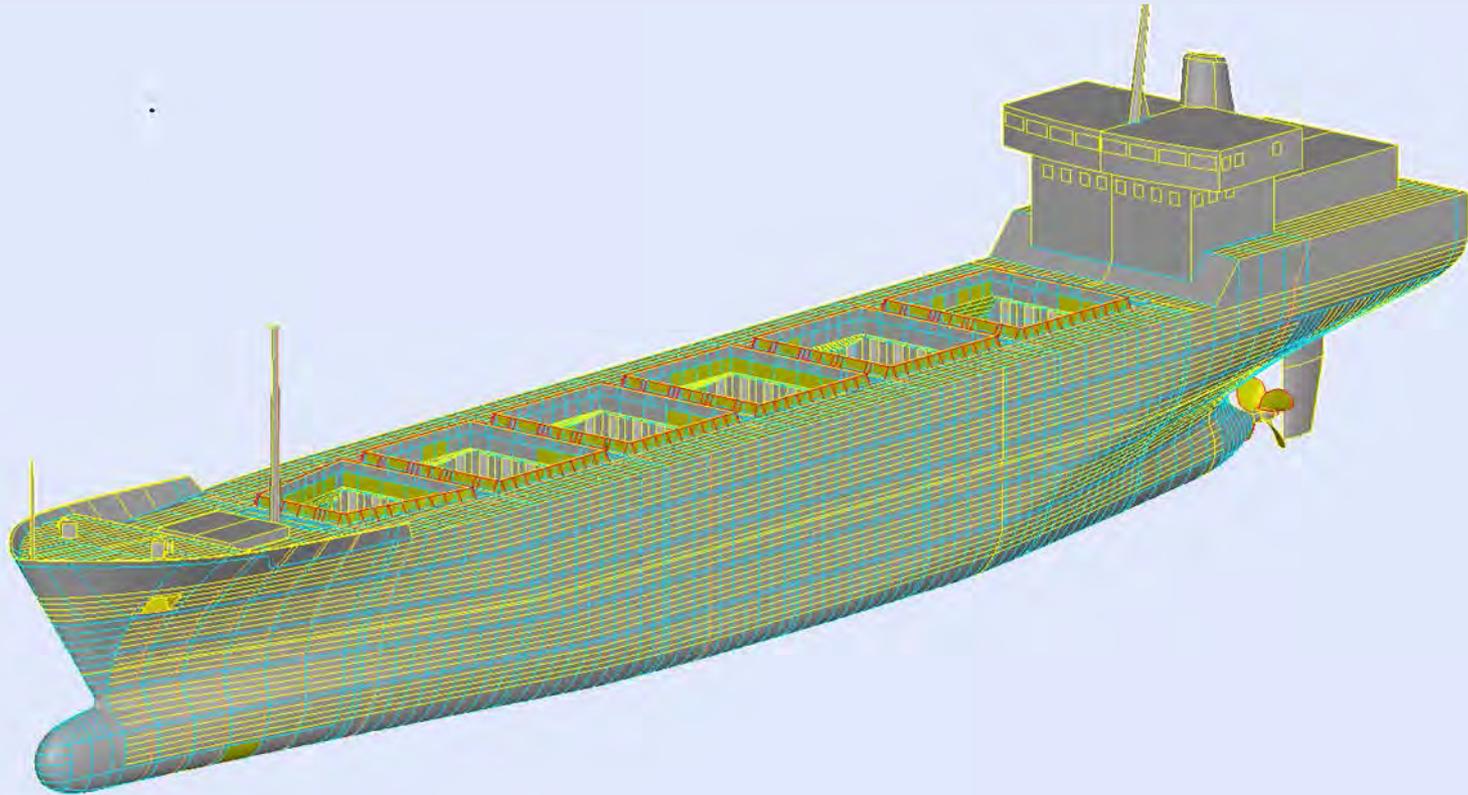
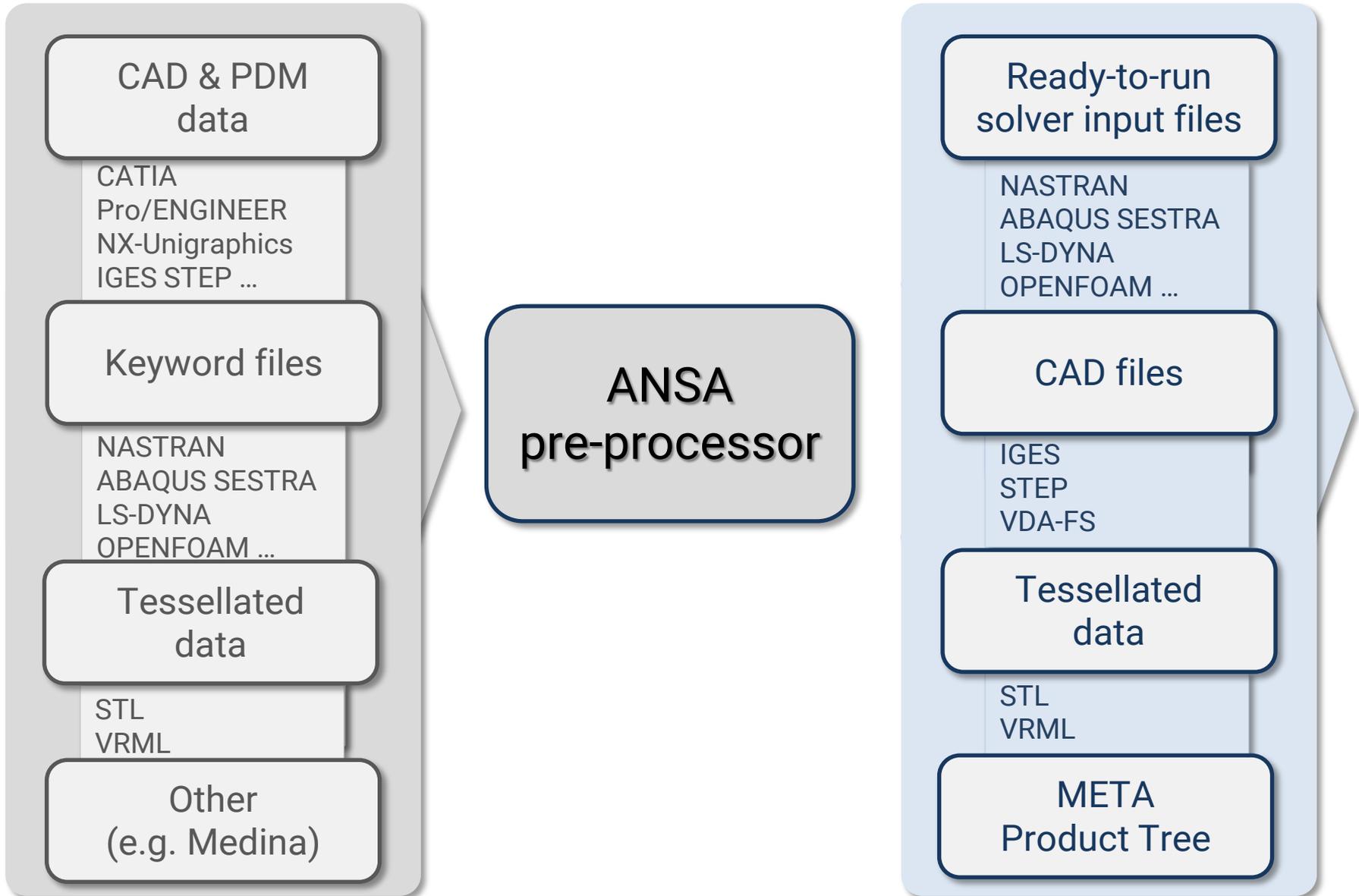


ANSA and META in Marine and Offshore structures analysis



INTERFACES

Interfaces



Model management

Database browser, Model Browser, Properties / Materials lists, DM Browser

The screenshot displays a CAD software interface with a 3D model of a ship hull. The interface includes several key panels and toolbars:

- Part Manager:** A tree view on the left showing the model's hierarchy. It lists modules (400, 300, 200, 100, 150, 140, 130, 120, 110, 80, 70, 60, 52, 50) and their names (SCIN, AFT, FORE, MIDDLE, Bottom_Longitudinal_Girders, Longitudinal_Girders, Transverse_Frames, Transverse_Bulkheads, Inner_Hull, propeller, L_Longitudinals_Stiffeners, Help_Entities, sea_level, WAVE). It shows 4 groups and 30 parts.
- Properties:** A table on the right showing details for selected parts. It includes columns for Id, Name, T, MID1, MID, and _type. The table lists 13 parts, including chimney, deck, hull sections, life boats, propeller, rudder, sea_level, storage_hatches, and superstructure.
- Materials:** A table at the bottom right showing material properties. It includes columns for Id, Name, DEFINED, E, RHO, and _type. It lists 5 material entries, all of type MAT1.
- Mesh Generation:** A panel on the right with buttons for MESH GEN, FREE, RE-MESH, BATCH, SPOT-ME, ERASE, CFD, MAP, ADV.FR, and STL.
- General Properties:** A panel on the left showing details for the selected part (Module Id: 100, Name: MIDDLE, Version, Representation, Study Version: 0, Id: 35, Hierarchy).
- Info Panel:** A panel at the bottom left showing model statistics: Current perimeter length: 0.2675 (old: 750.), Current distortion distance: 40 (old: 20.), Current distortion angle: 0 (old: 0.).

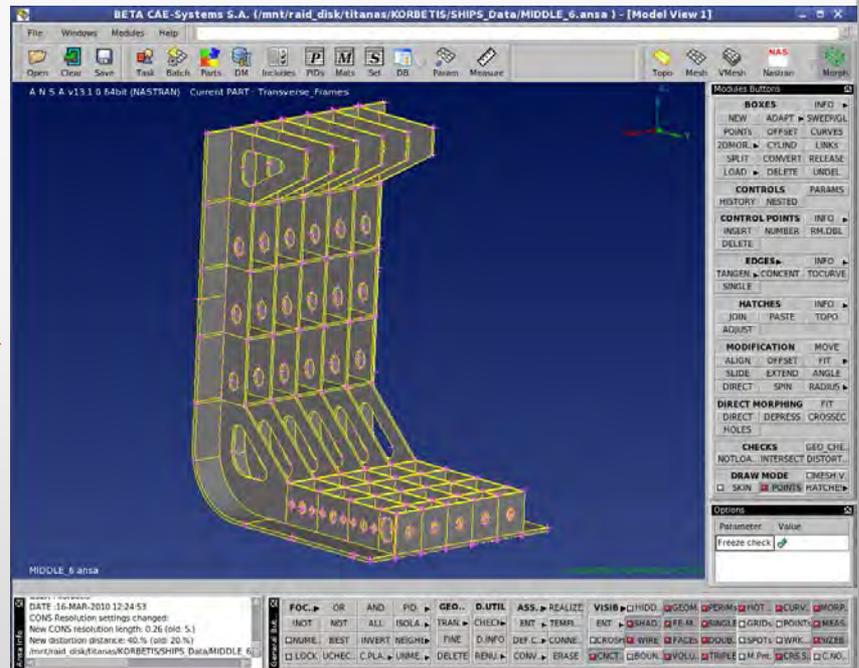
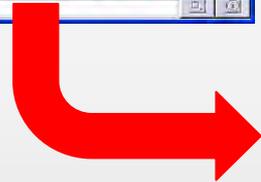
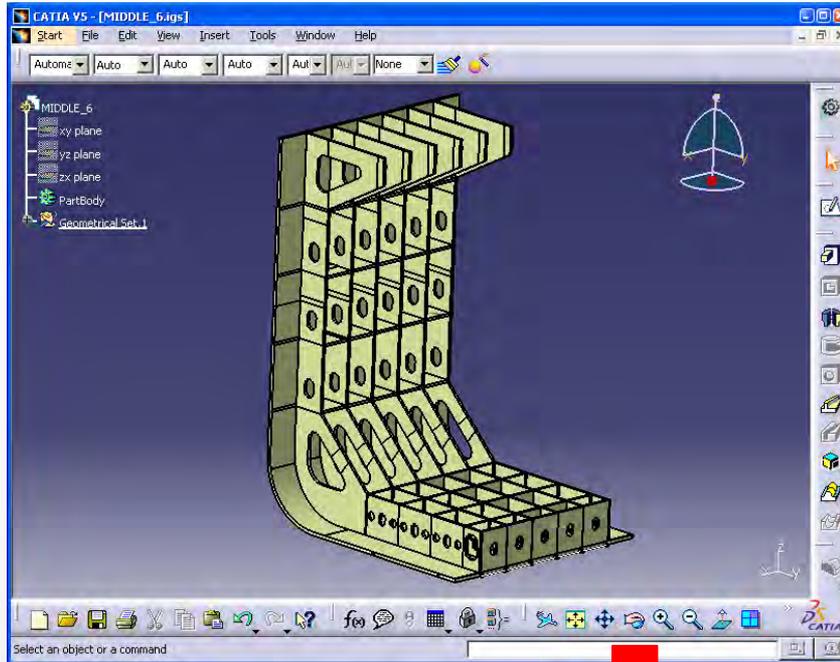
Native and Neutral CAD data input

Native CAD data input:

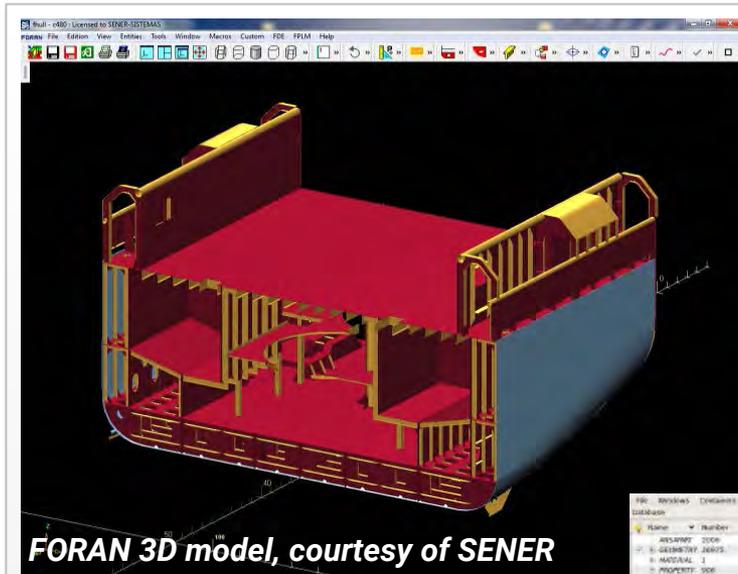
- CATIA, ProE, NX, JTOpen and more..

Direct Neutral File CAD data input:

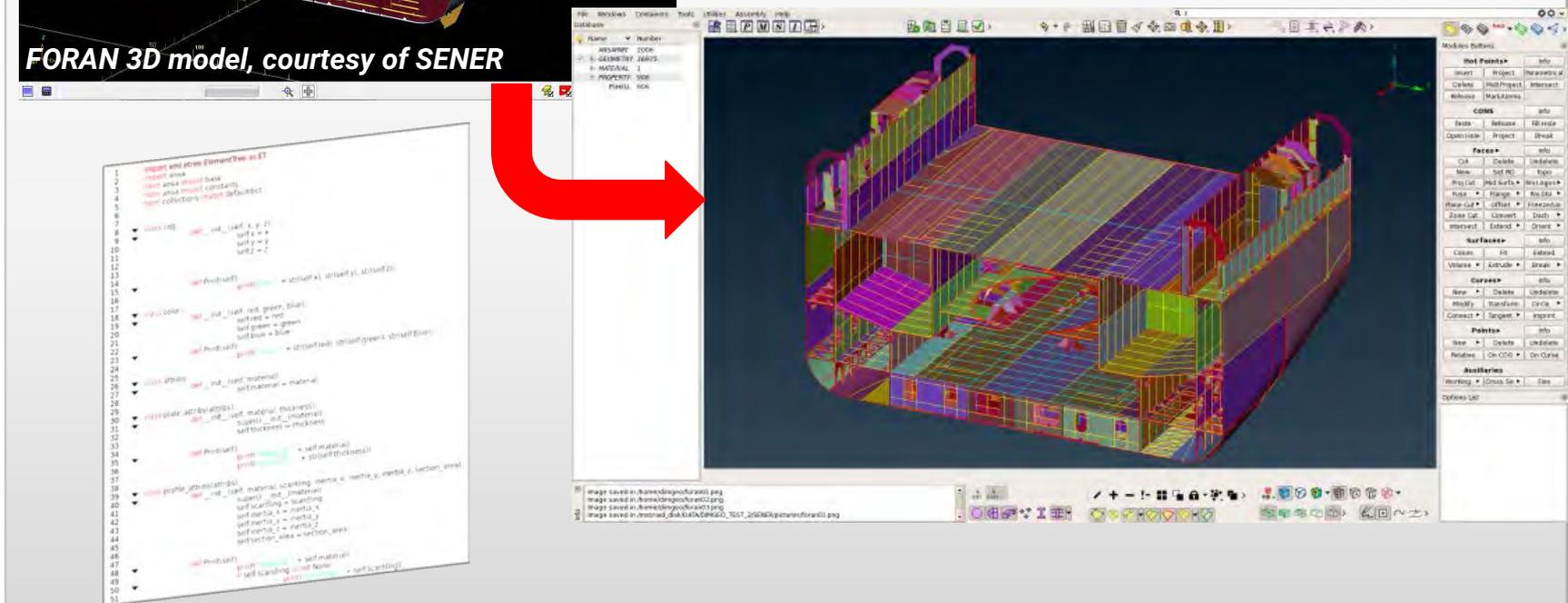
- IGES, VDA-FS, STEP



From FORAN to ANSA

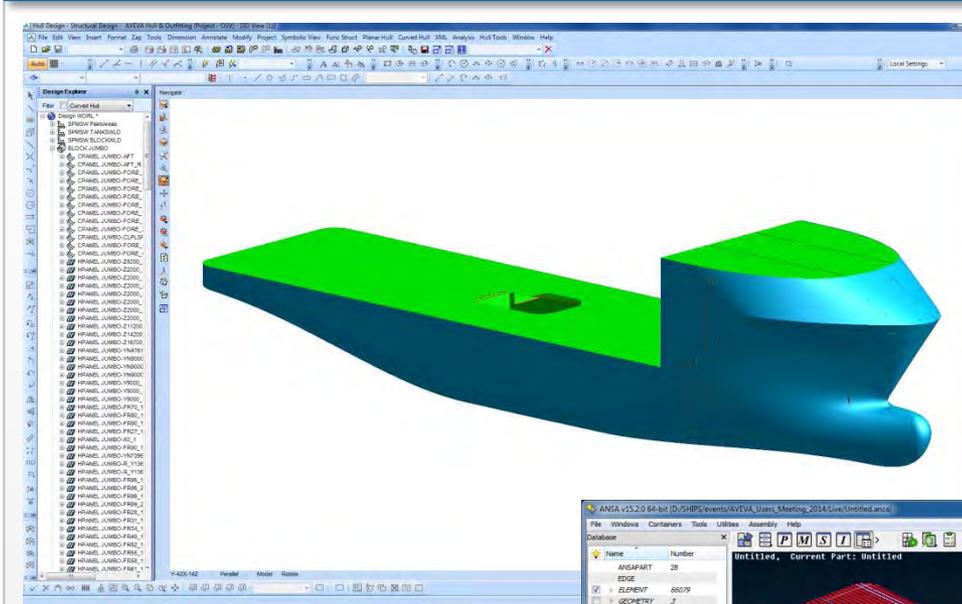


- Use a step and an xml file produced by Foran
- Open the step file
- Read the xml and apply the additional information to the geometry

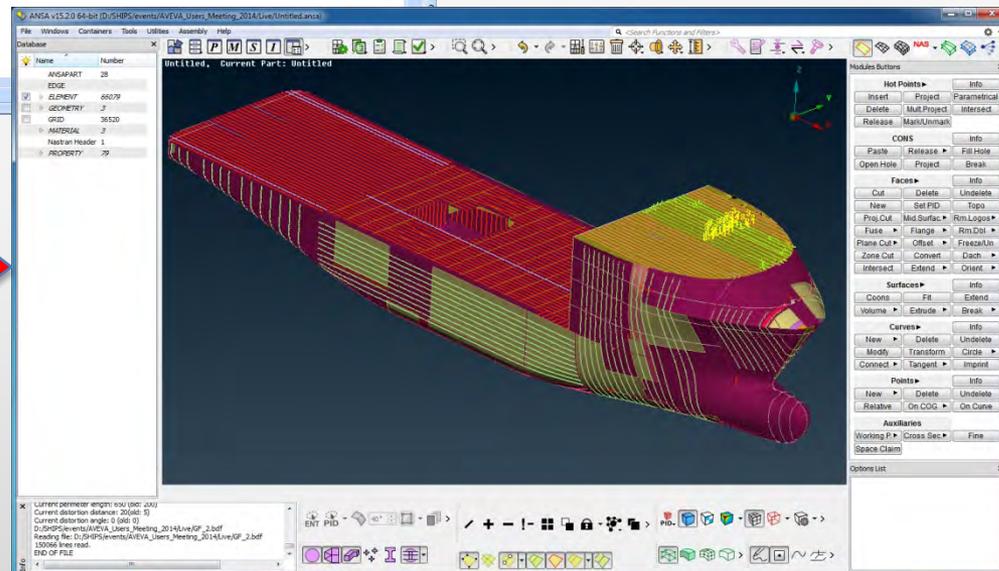
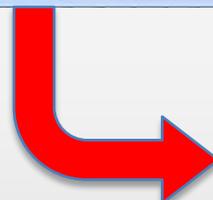


From AVEVA Marine to ANSA

Additional information is read from an AVEVA xml file and applied on the model

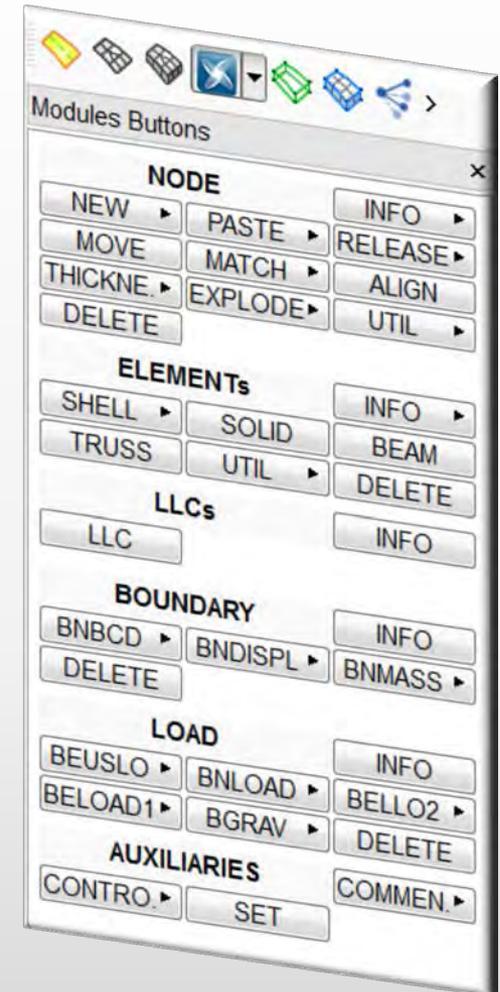
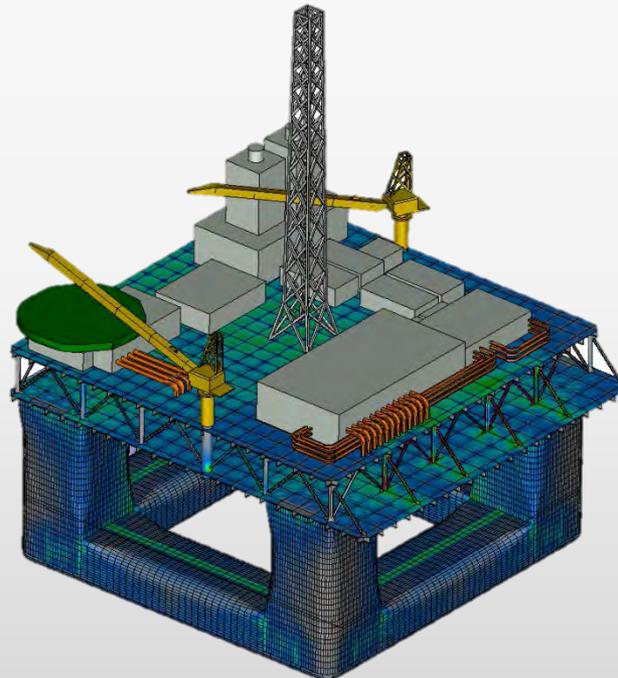
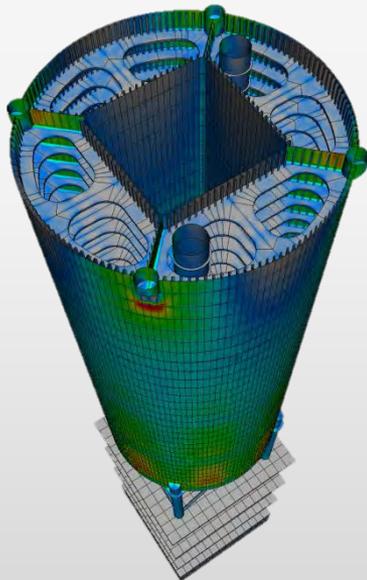
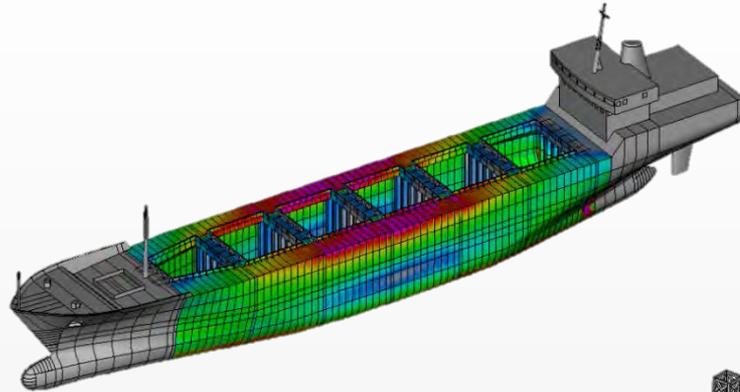


```
<Ship Version="pre 2.0">
  <Rendering Type="Idealised" />
  <Material Grade="Default" Density="7840" YoungsModulus="206000"
  PoissonRatio="0.3" YieldStress="220" UltimateStress="360" />
  <BarSection BarSectionId="BulbFlat120*23*5*6">
    <BulbFlat Height="120" Width="23" BulbRadius="5" WebThickness="6"
    />
  </BarSection>
  <BarSection BarSectionId="BulbFlat160*30*6*8">
    <BulbFlat Height="160" Width="30" BulbRadius="6" WebThickness="8"
    />
  </BarSection>
  <BarSection BarSectionId="FlatBar160*12">
    <FlatBar Height="160" Width="12" />
  </BarSection>
  <BarSection BarSectionId="FlatBar200*12">
```



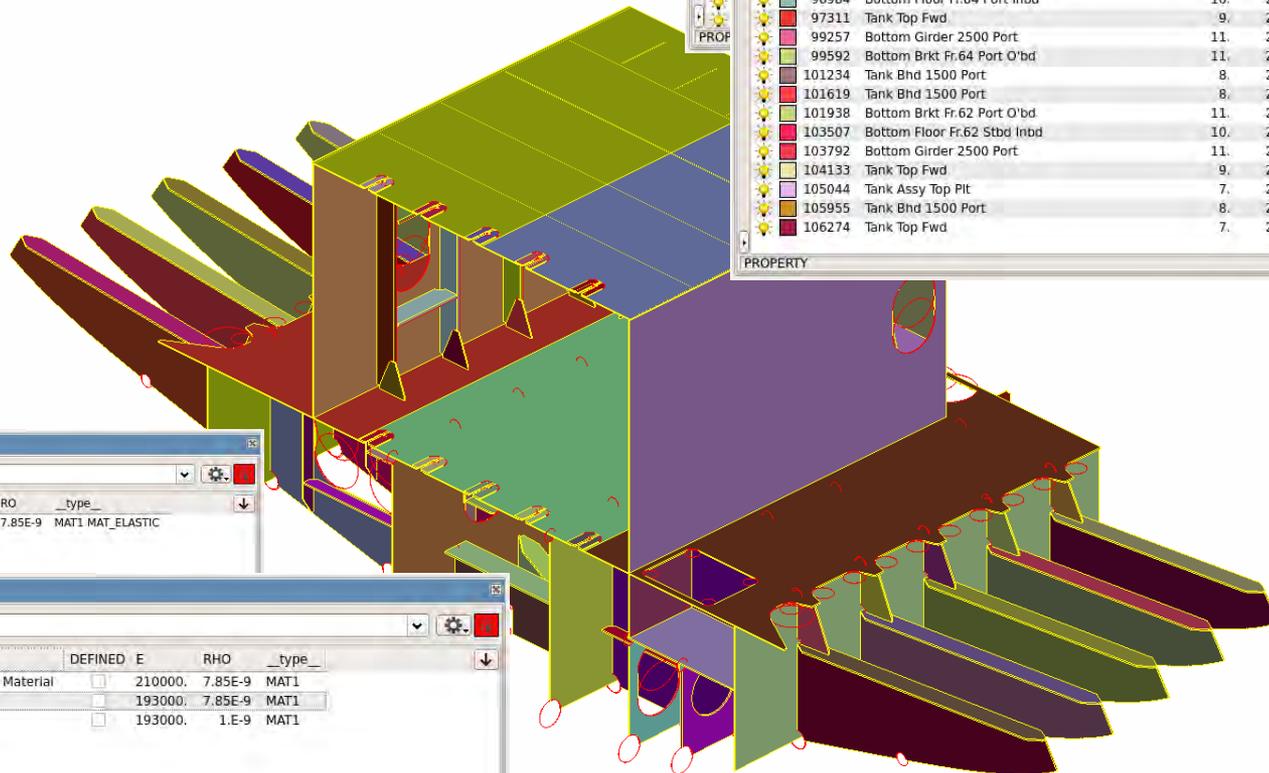
Interface for SESTR

Input, output and new DECK for SESTR



From Ship Constructor to ANSA

- Use of ship constructor data and solid description step file
- Import model data:
 - Property Thickness and
 - Material data



id	Name	T	MID1	MID	_type_
3628	{3433DF95-190C-44E0-8F47-19FF9733DD8A}	1.	1		PSHELL
3960	{F2E49BA6-3CA9-4F3C-8004-0A53200E8860}	1.	1		PSHELL
4292	{3FBD4653-9B2C-4429-B4CB-04DF02AA9FBC}	1.	1		PSHELL
4624	{C3B5889E-F3D0-41A1-94A4-04BF510DDCC8}	1.	1		PSHELL
4824	{BDD358C5-1F0E-45A7-BC48-048EB7D7158A}	1.	1		PSHELL
5024	{5ED8959D-B54E-43B2-A2E3-00ACD48B5303}	1.	1		PSHELL
5356	{80EFFD6A-5535-4B47-8B80-FE692D958893}	1.	1		PSHELL

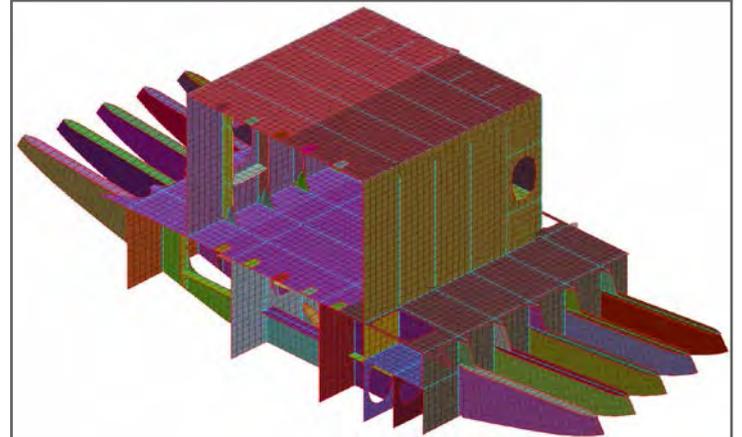
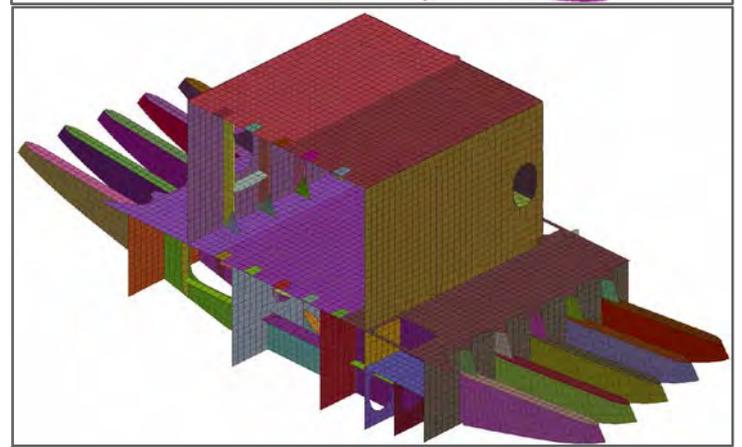
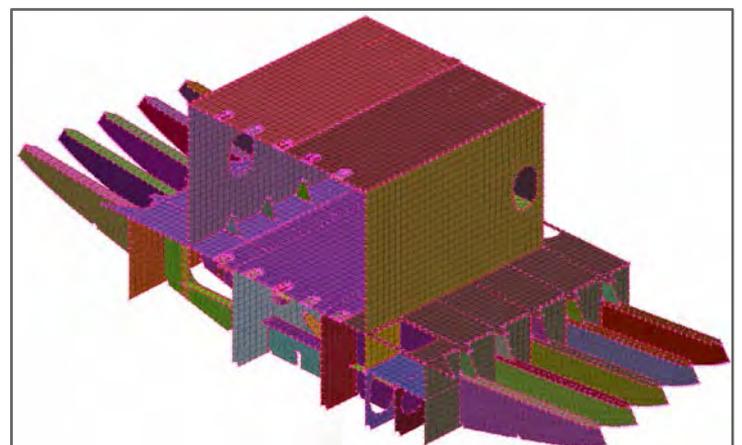
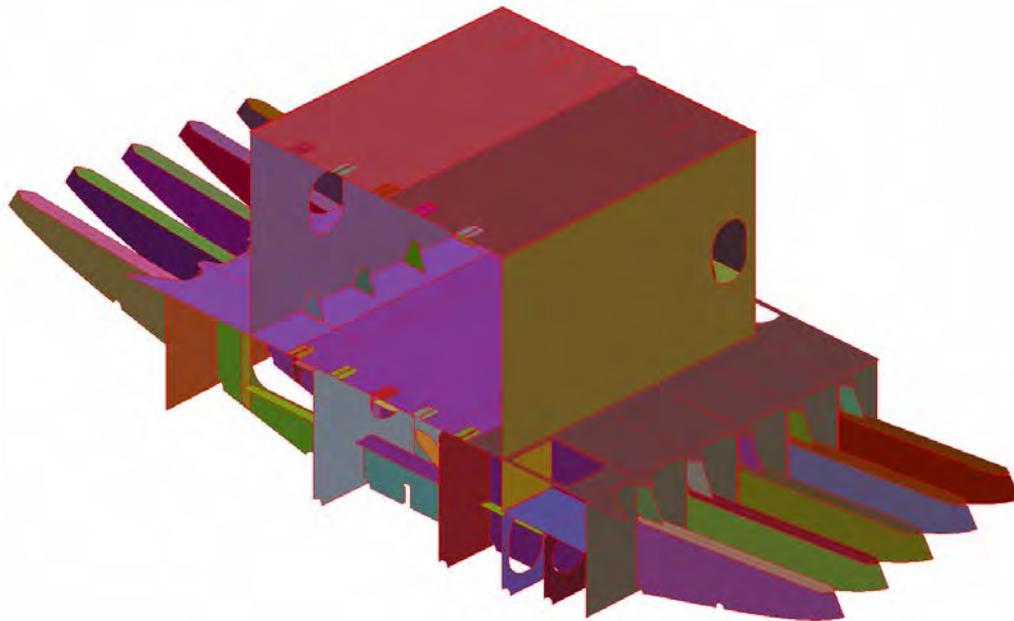
id	Name	T	MID1	MID	_type_
91150	Tank Bhd 1500 Stbd	8.	2		PSHELL
91469	Bottom Brkt. Fr.61 Port O'bd	11.	2		PSHELL
93072	Bottom Floor Fr.61 Port Inbd	11.	2		PSHELL
94374	Tank Assy Top Plt	7.	2		PSHELL
95744	Bottom Floor Fr.63 Port Inbd	11.	2		PSHELL
96984	Bottom Floor Fr.64 Port Inbd	10.	2		PSHELL
97311	Tank Top Fwd	9.	2		PSHELL
99257	Bottom Girder 2500 Port	11.	2		PSHELL
99592	Bottom Brkt. Fr.64 Port O'bd	11.	2		PSHELL
101234	Tank Bhd 1500 Port	8.	2		PSHELL
101619	Tank Bhd 1500 Port	8.	2		PSHELL
101938	Bottom Brkt. Fr.62 Port O'bd	11.	2		PSHELL
103507	Bottom Floor Fr.62 Stbd Inbd	10.	2		PSHELL
103792	Bottom Girder 2500 Port	11.	2		PSHELL
104133	Tank Top Fwd	9.	2		PSHELL
105044	Tank Assy Top Plt	7.	2		PSHELL
105955	Tank Bhd 1500 Port	8.	2		PSHELL
106274	Tank Top Fwd	7.	2		PSHELL

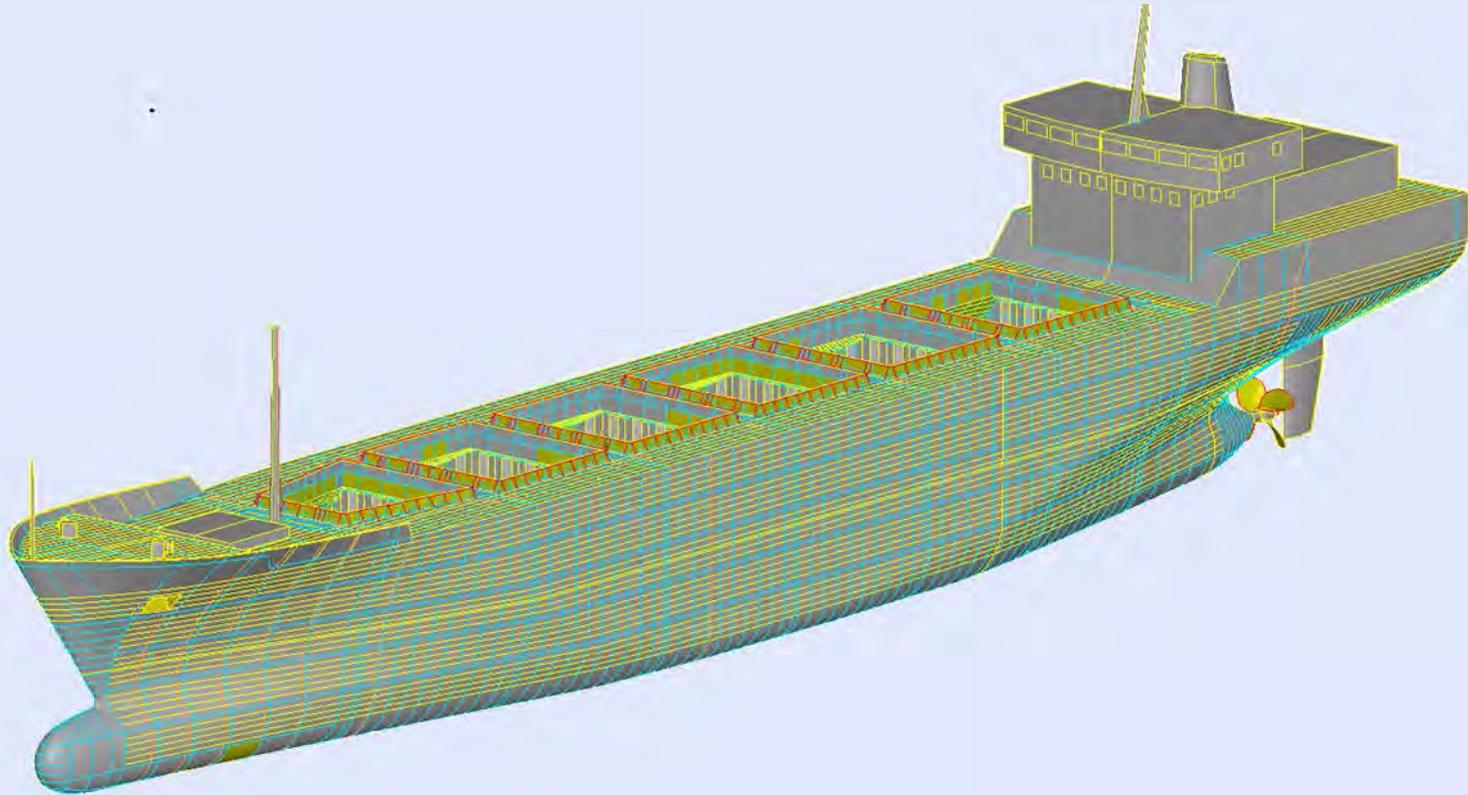
id	Name	DEFINED	E	RO	_type_
1	Default MAT1 Material	<input type="checkbox"/>	210000.	7.85E-9	MAT1_MAT_ELASTIC

id	Name	DEFINED	E	RHO	_type_
1	Default MAT1 Material	<input type="checkbox"/>	210000.	7.85E-9	MAT1
2	Gr. A	<input type="checkbox"/>	193000.	7.85E-9	MAT1
3	MDF	<input type="checkbox"/>	193000.	1.E-9	MAT1

From Ship Constructor to ANSA

- Batch Middle skin extraction
- Auto mesh
- Reconstruct and auto connect
- Output as solver format



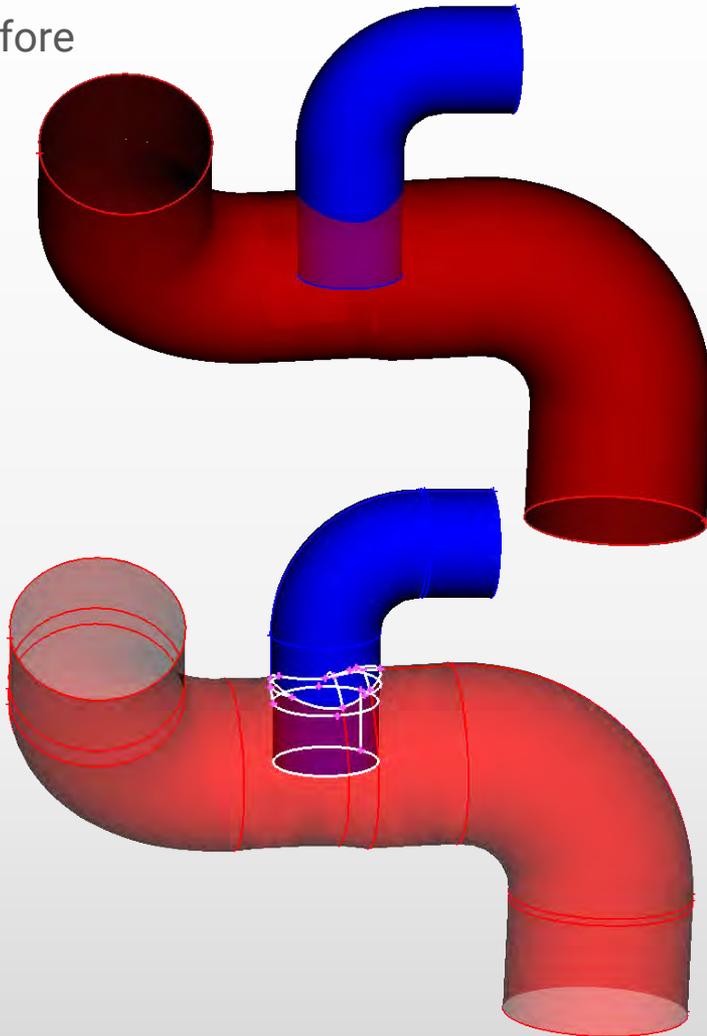


GEOMETRY

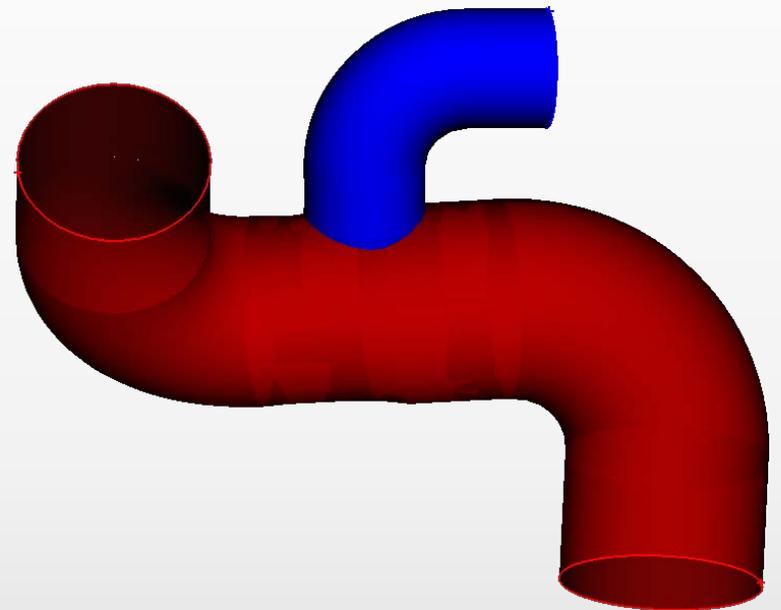
Intersections

Geometry trimming

Before

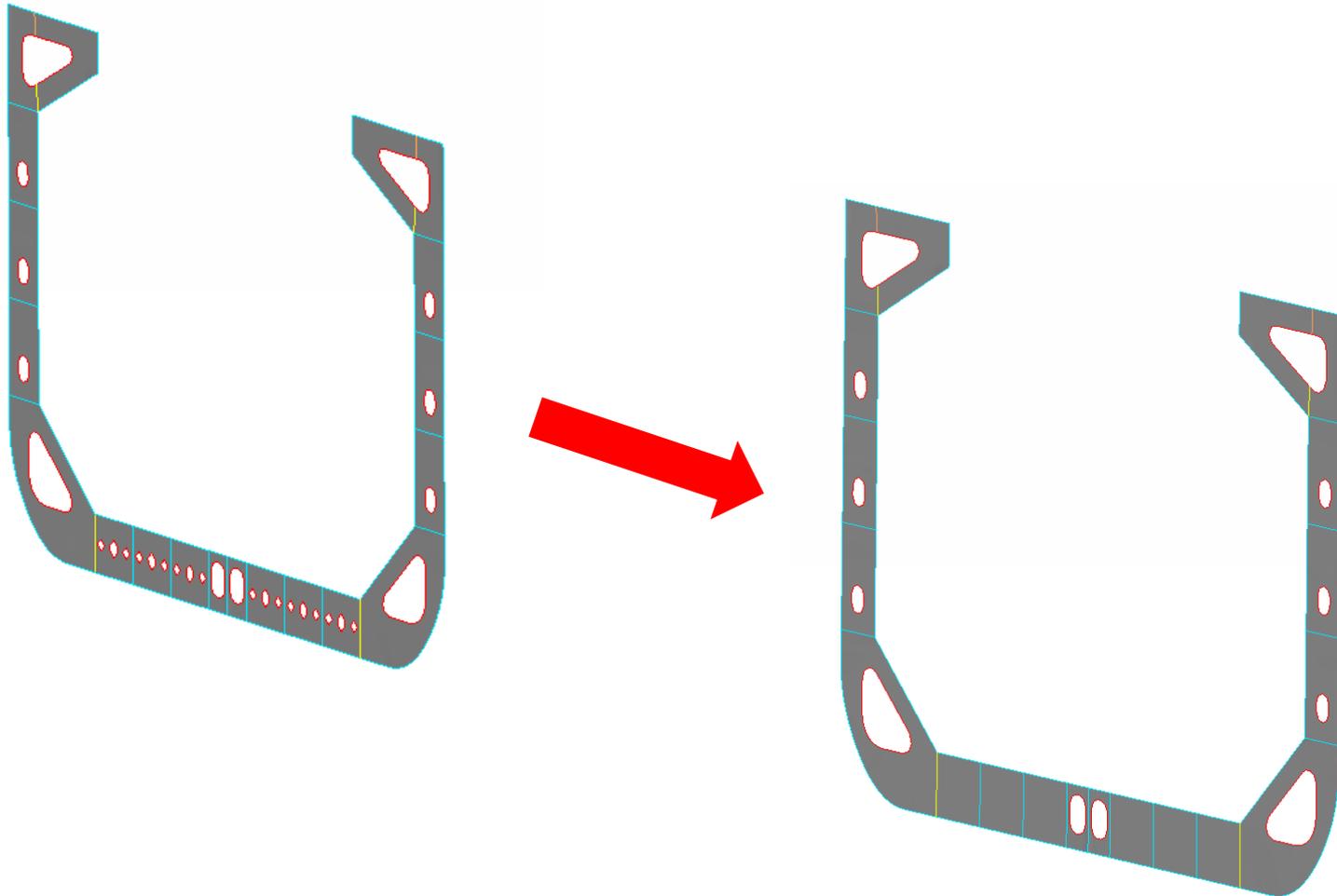


After



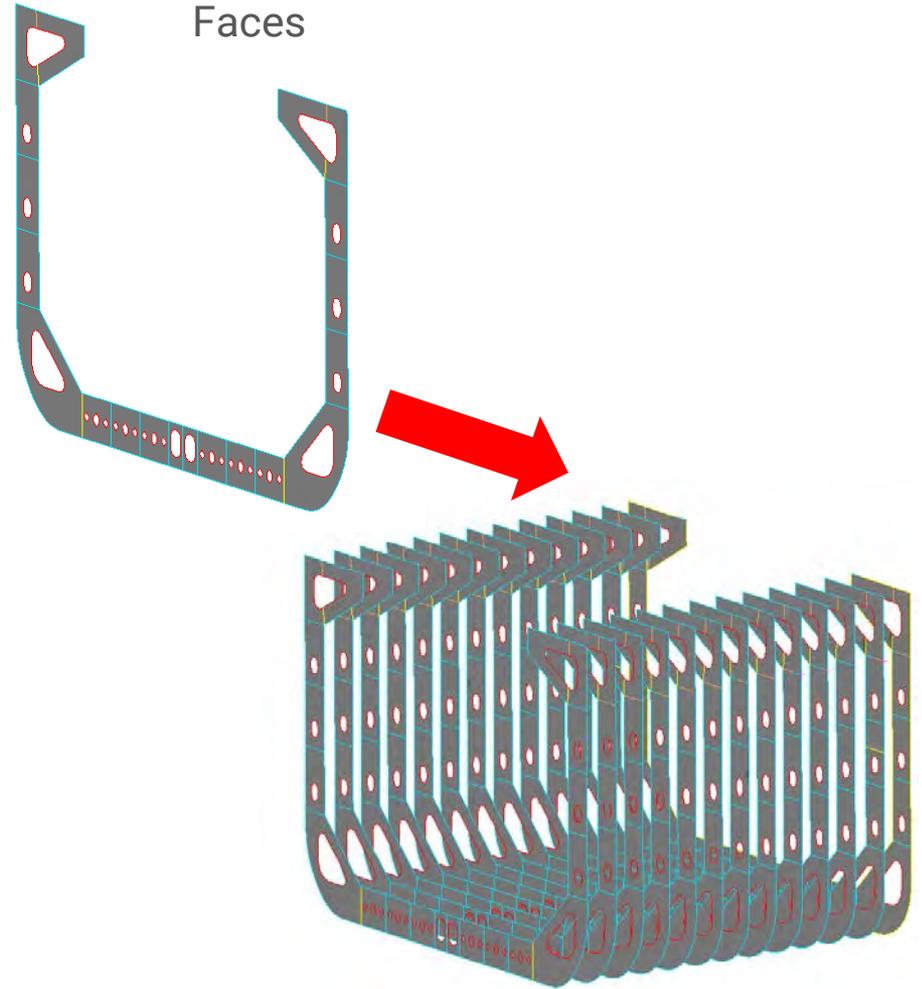
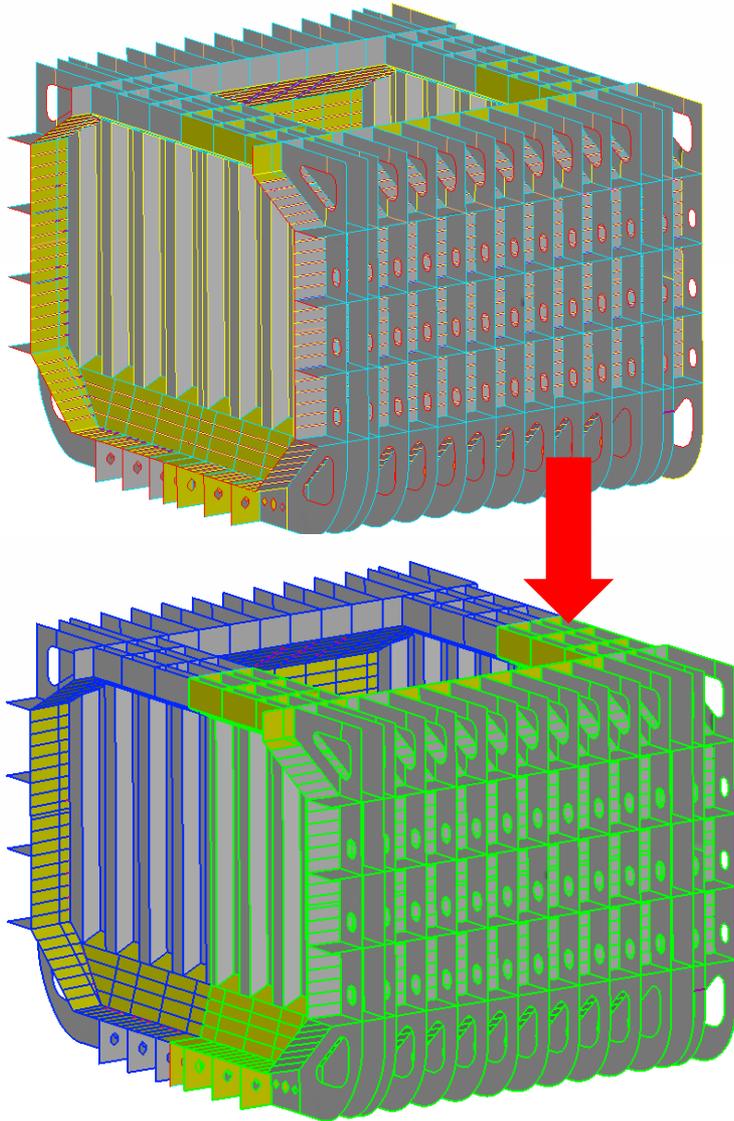
Model simplification

- Identifying unchecked faces, needle faces, collapsed CONS, triple bounds, overlaps, cracks
- Treatment of holes, fillets, chamfers, features



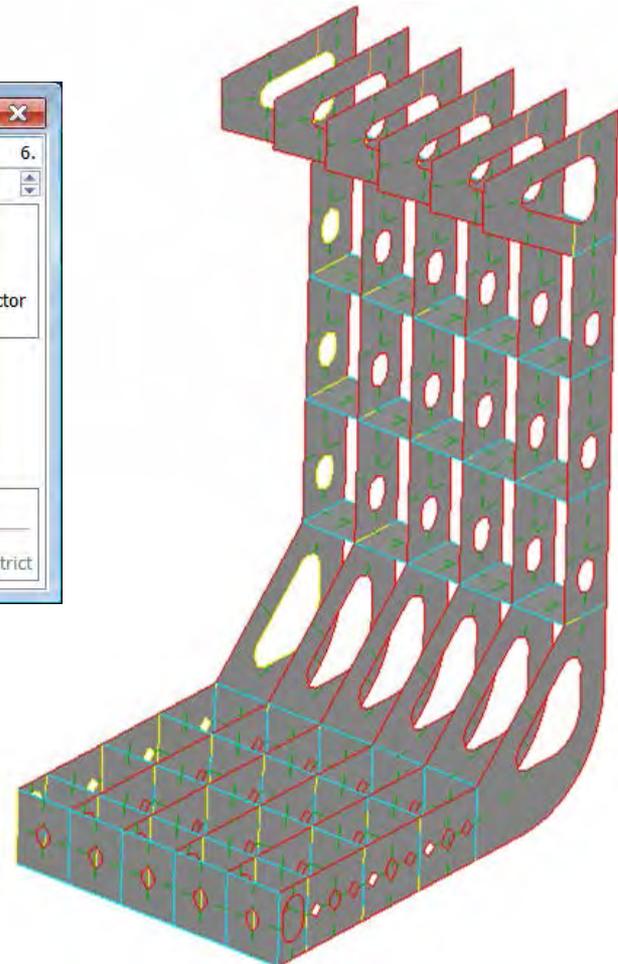
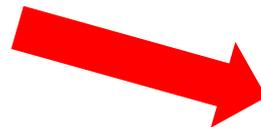
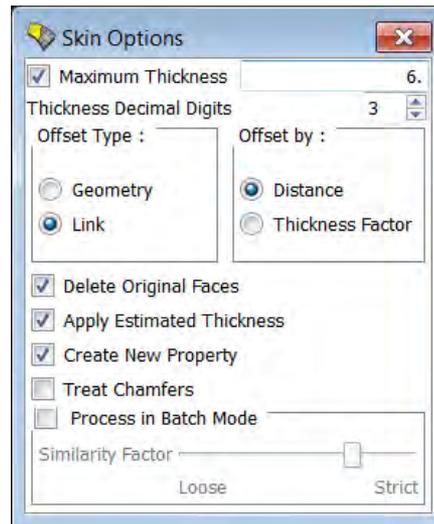
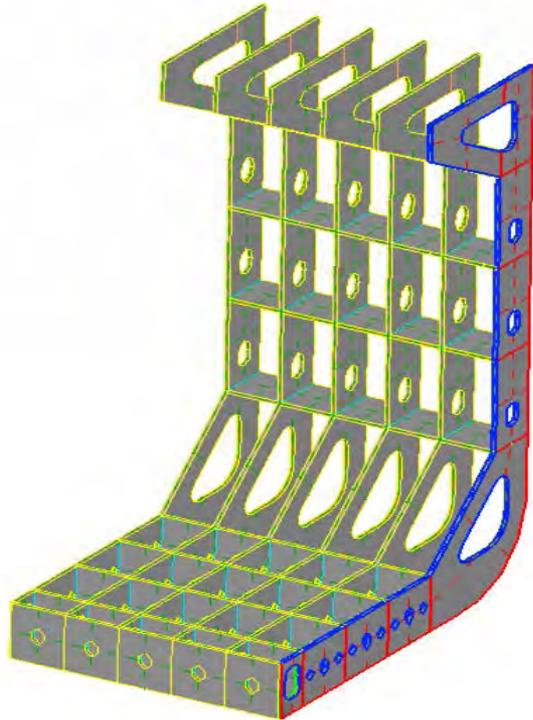
Handling pattern-wise faces & parts

- Substitute geometry with Linked Faces
- Create symmetry, mirror or translation Faces



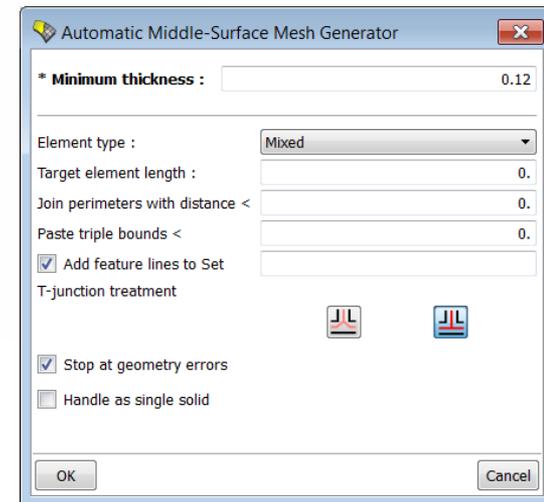
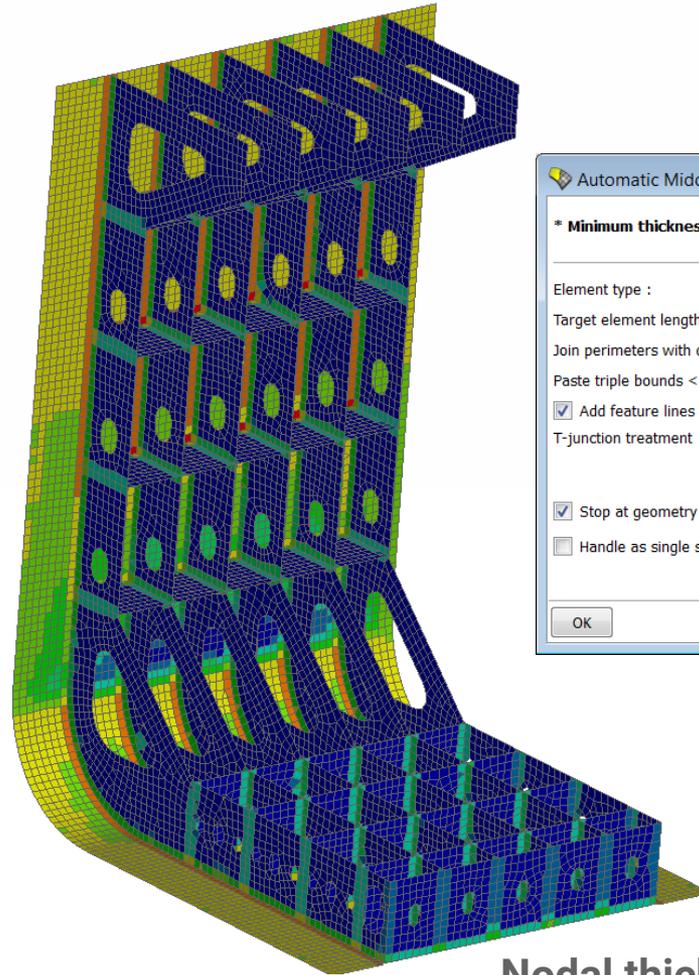
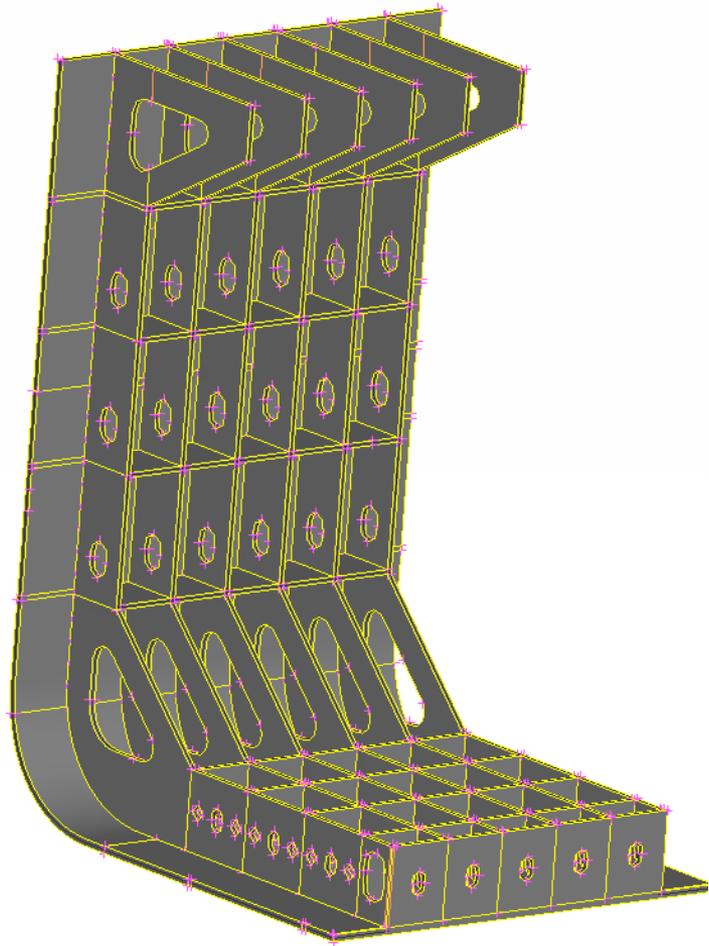
Middle skin extraction

Special tool for middle skin extraction creates new geometry



Middle skin extraction

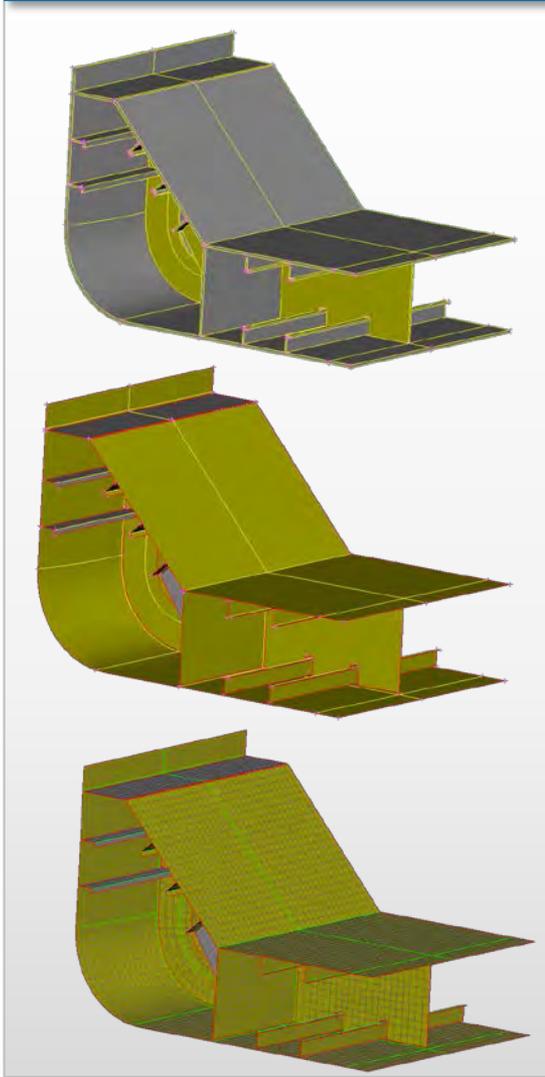
Fully automatic middle surface extraction creates FE Model



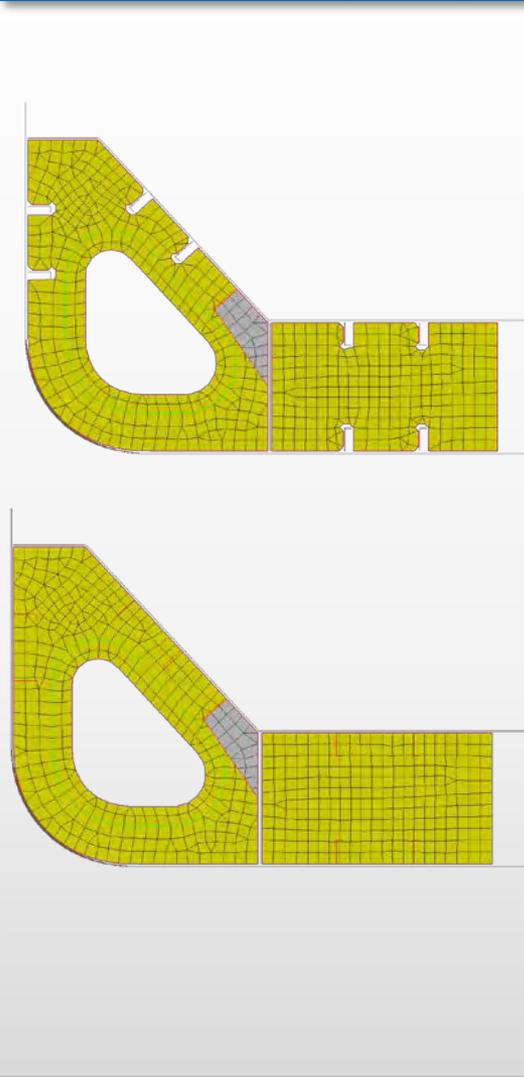
Nodal thickness assignment

Automatic middle skin extraction and assembly

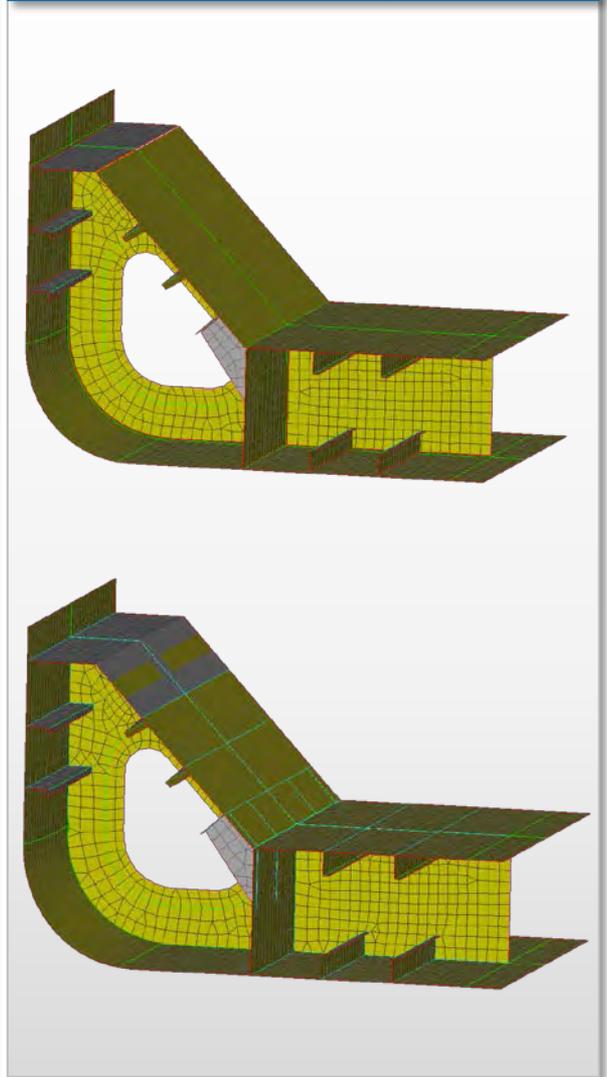
Create the middle skin

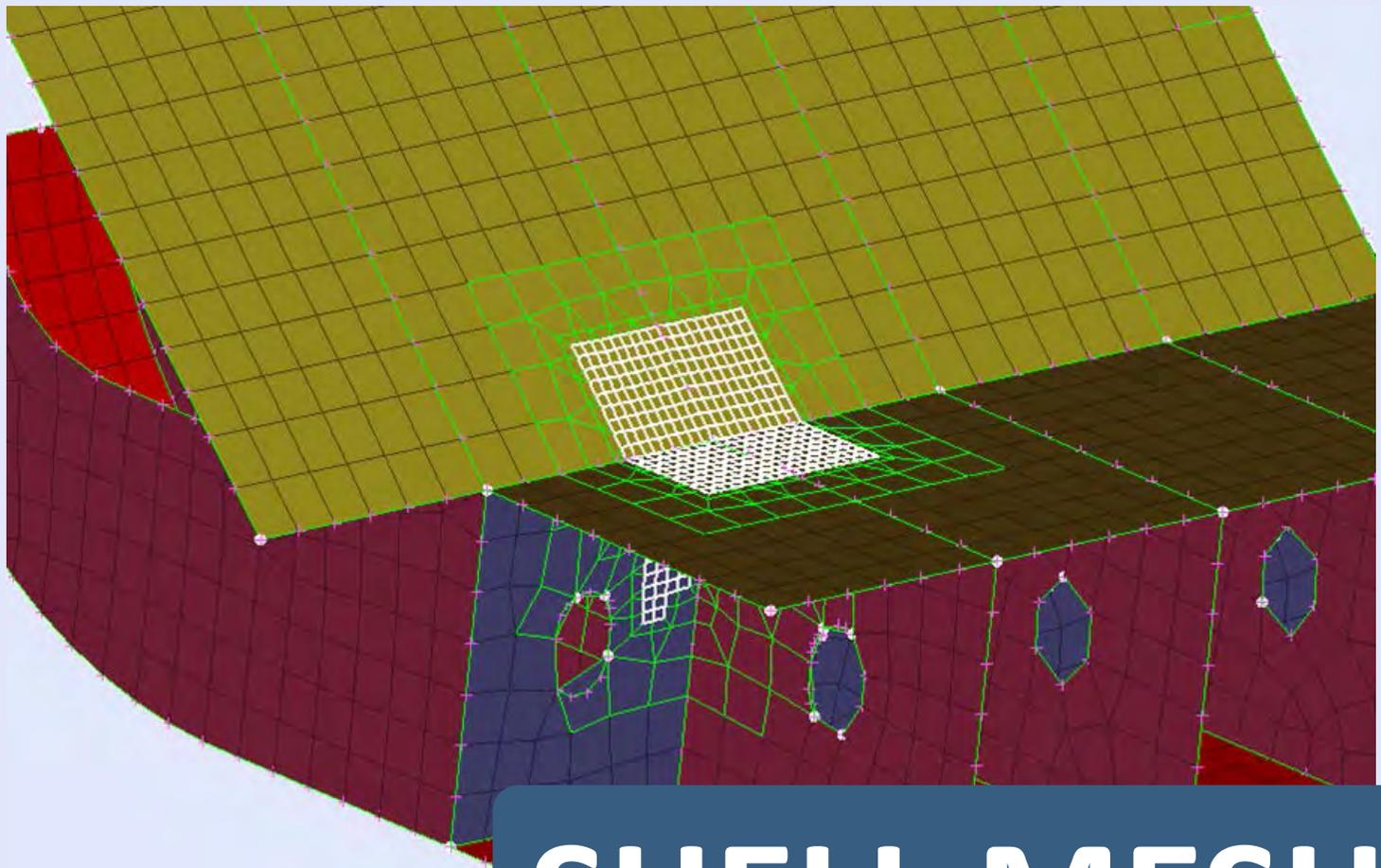


Fill unwanted openings



Connect the parts

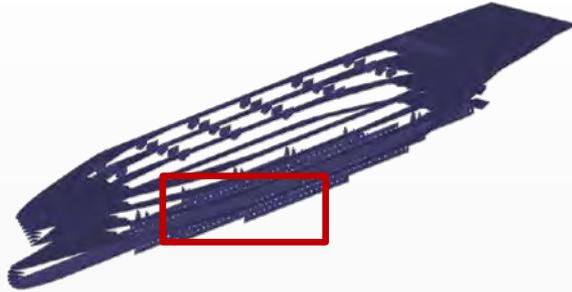




SHELL MESH

Shell Meshing

Mesh for Structural Applications: Sheet-metal components

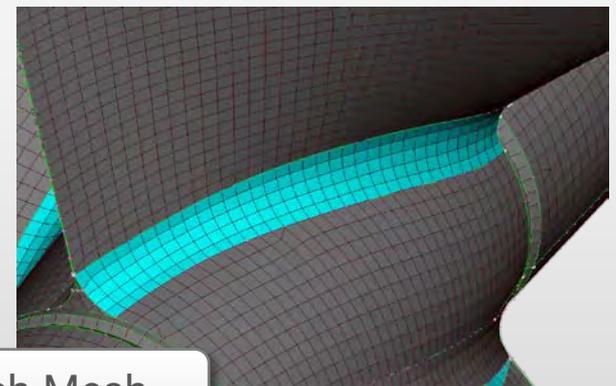
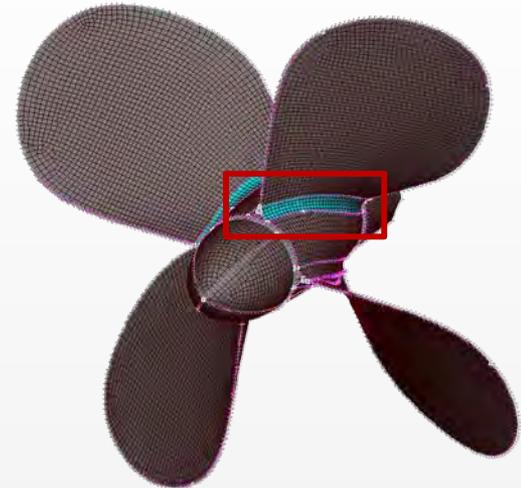
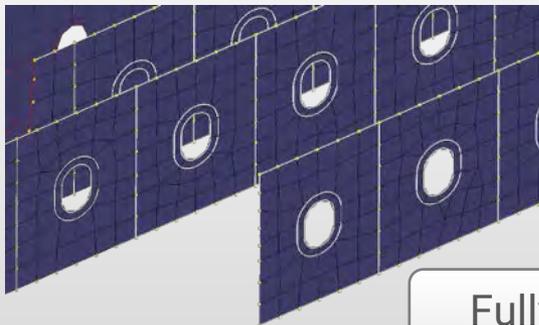
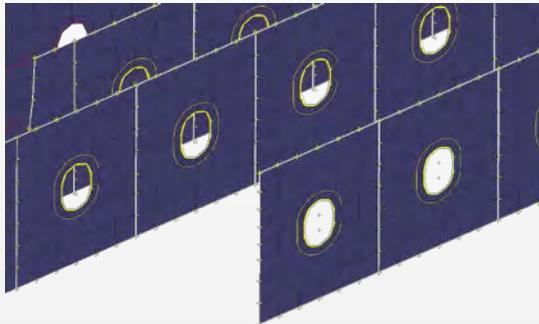


Automatic defeaturing

Mesh quality control

Feature treatment

- Holes
- Fillets
- Flanges



Fully automated through Batch Mesh

Shell Meshing

Numerous quality criteria

Quality Criteria

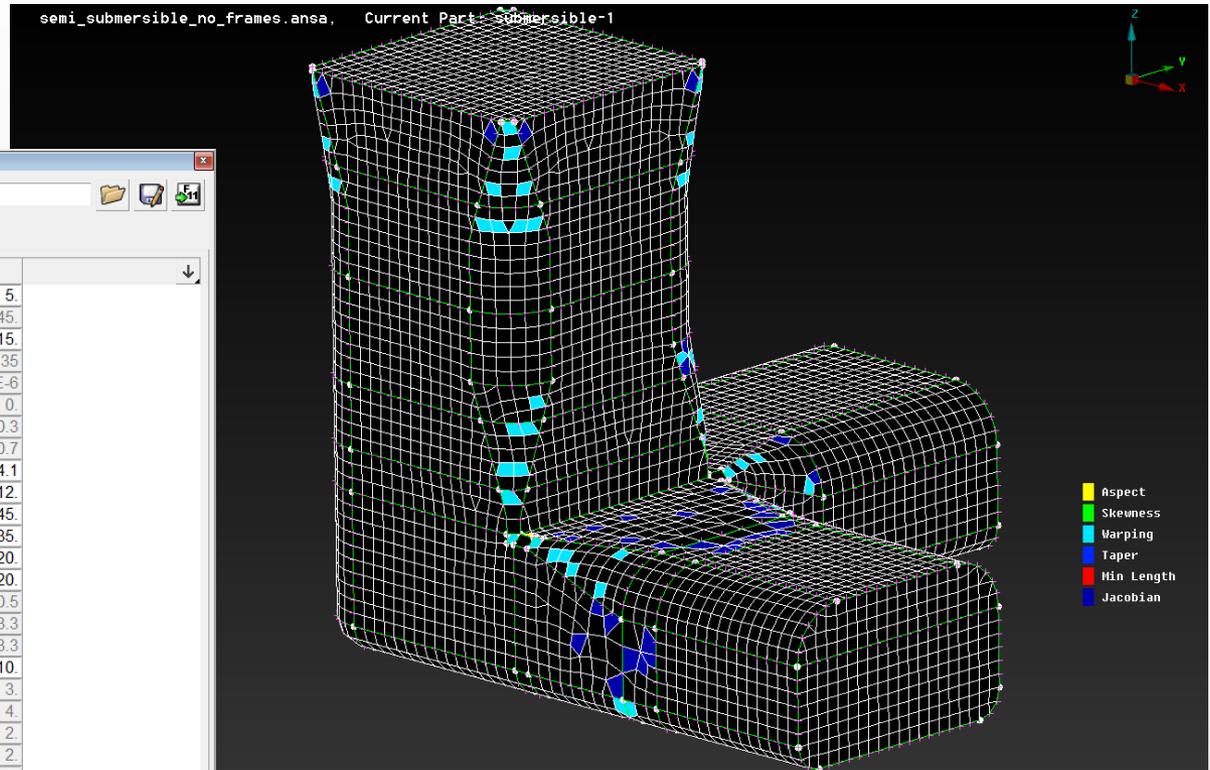
Name: |

Shells Solids

Criteria	Calculation	Color	Failed
<input checked="" type="checkbox"/> aspect ratio	NASTRAN	Yellow	5.
<input type="checkbox"/> skewness	PATRAN	Green	45.
<input checked="" type="checkbox"/> warping	IDEAS	Cyan	15.
<input type="checkbox"/> taper	PATRAN	Blue	0.35
<input type="checkbox"/> crash time step	LS-DYNA	Purple	1.E-6
<input type="checkbox"/> min height	QUADS & T	Teal	0.
<input type="checkbox"/> squish		Green	0.3
<input type="checkbox"/> jacobian	ANSA	Blue	0.7
<input checked="" type="checkbox"/> min length		Red	4.1
<input checked="" type="checkbox"/> max length		Red	12.
<input checked="" type="checkbox"/> min angle quads	IDEAS	Orange	45.
<input checked="" type="checkbox"/> max angle quads	IDEAS	Orange	135.
<input checked="" type="checkbox"/> min angle trias	IDEAS	Orange	20.
<input checked="" type="checkbox"/> max angle trias	IDEAS	Orange	120.
<input type="checkbox"/> stretch		Grey	0.5
<input type="checkbox"/> mid point deviation %		Black	33.3
<input type="checkbox"/> mid point alignment %		Orange	33.3
<input checked="" type="checkbox"/> triangles %		Green	10.
<input type="checkbox"/> triangles per node		Olive	3.
<input type="checkbox"/> mesh distortion		Pink	4.
<input type="checkbox"/> distance from geometry		Purple	2.
<input type="checkbox"/> distance from origin		Light Green	2.
<input type="checkbox"/> multi violation		Olive	2.
<input type="checkbox"/> growth ratio		Light Blue	1.2
<input type="checkbox"/> incomplete element		Cyan	

Enable Ranges Edit Criteria Visibility

OK Cancel

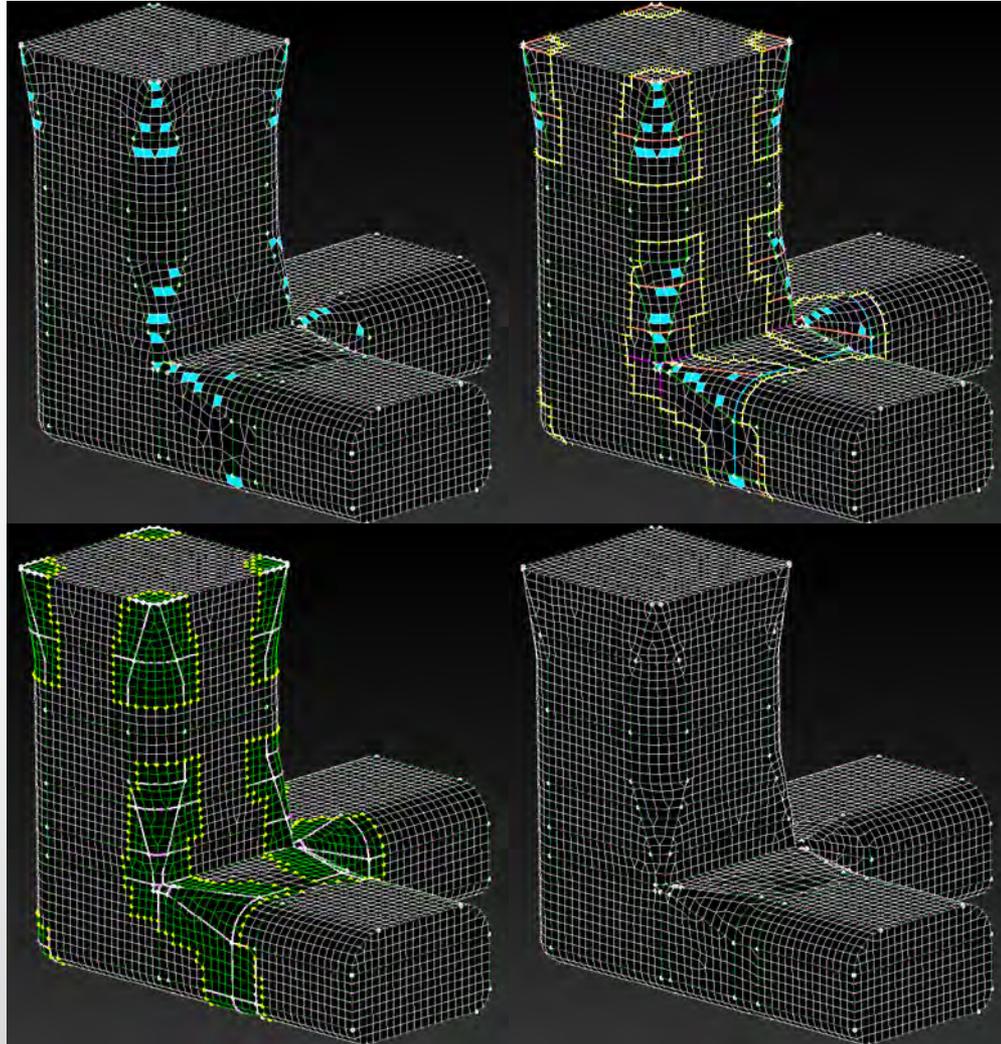


Color-coding of violations

Shell Meshing

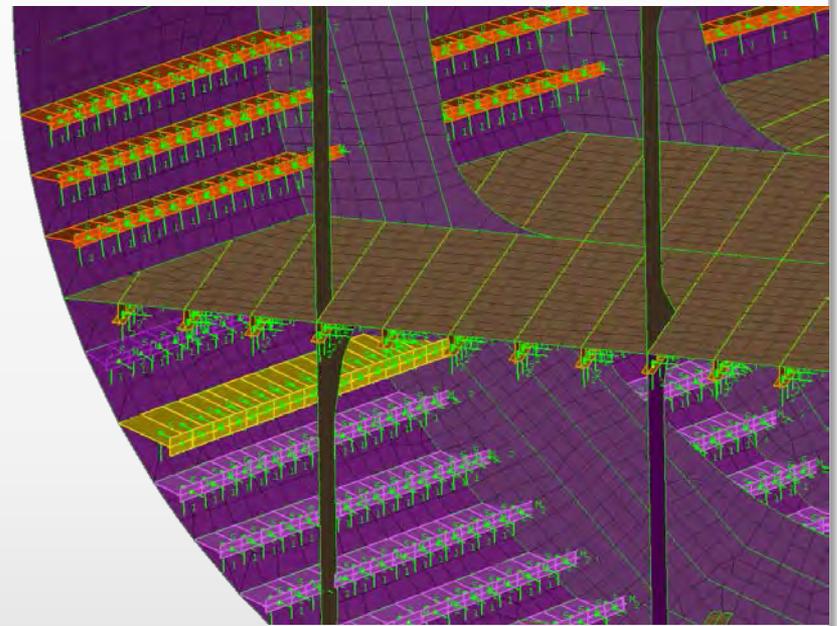
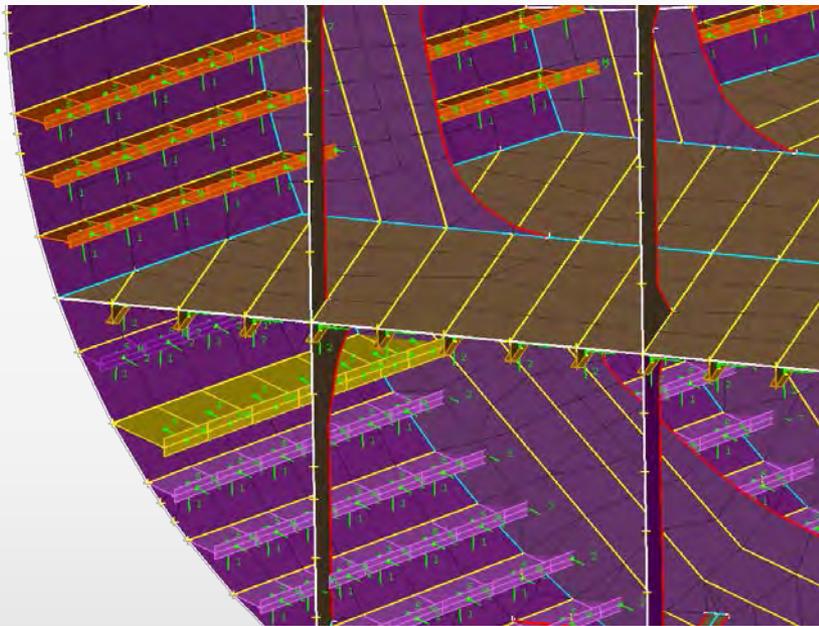
Automatic fix of quality violations

- Aspect
- Skewness
- Warping
- Taper



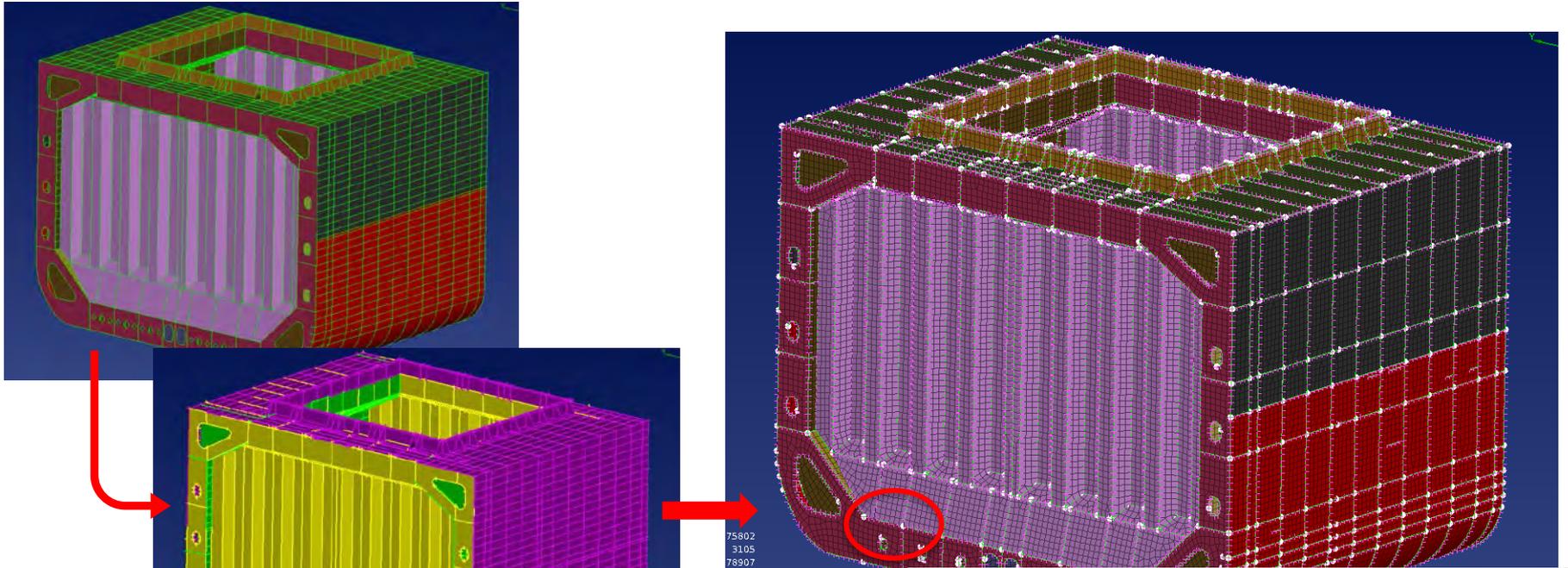
Reconstruct shells and beams

Reconstruct shells and the attached beams at the same time



Batch meshing

- Definition of meshing parameters and quality criteria
- Features treatment and model simplification
- Automatic meshing and quality improvement



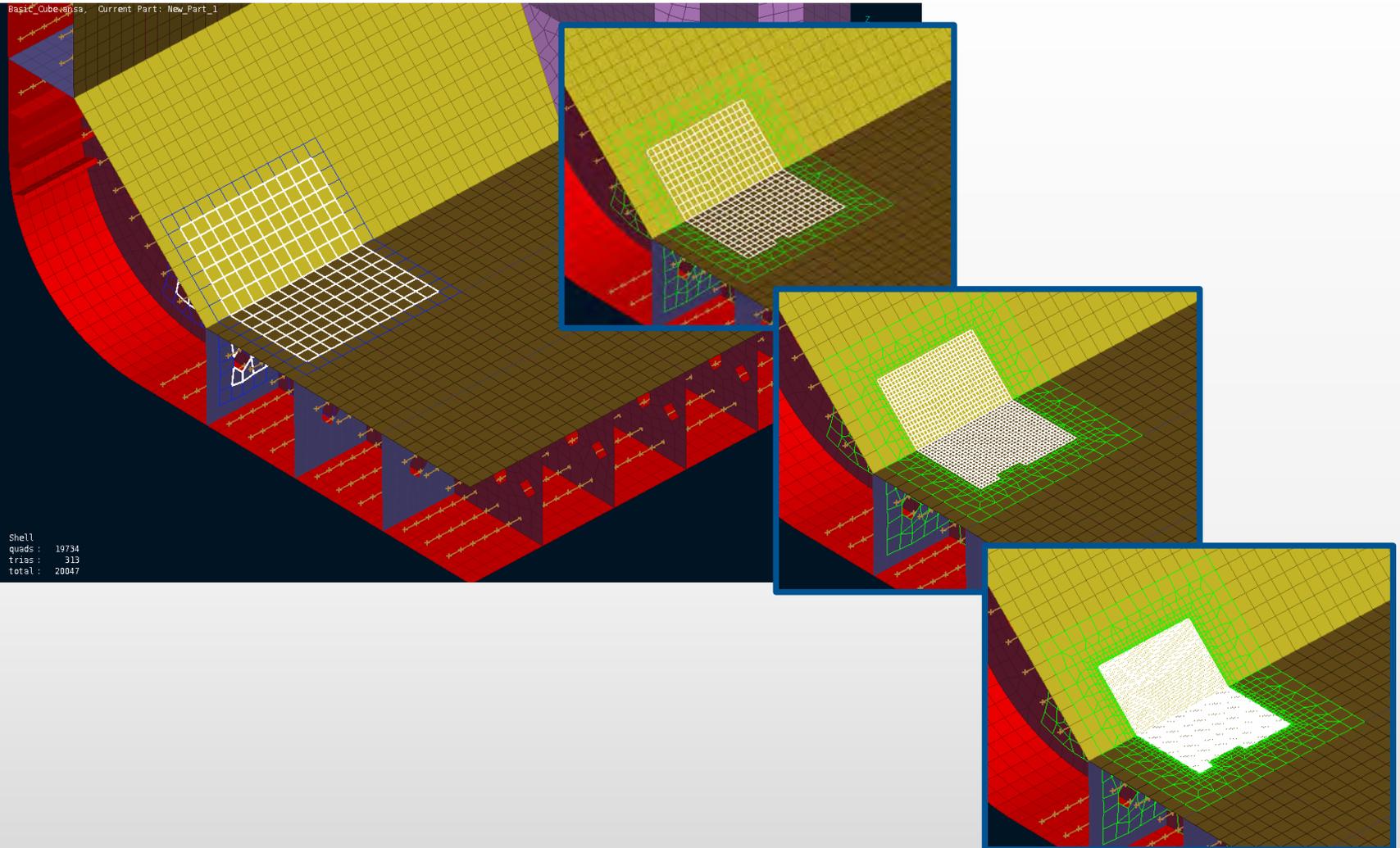
75802
3105
78907

Batch Mesh Manager

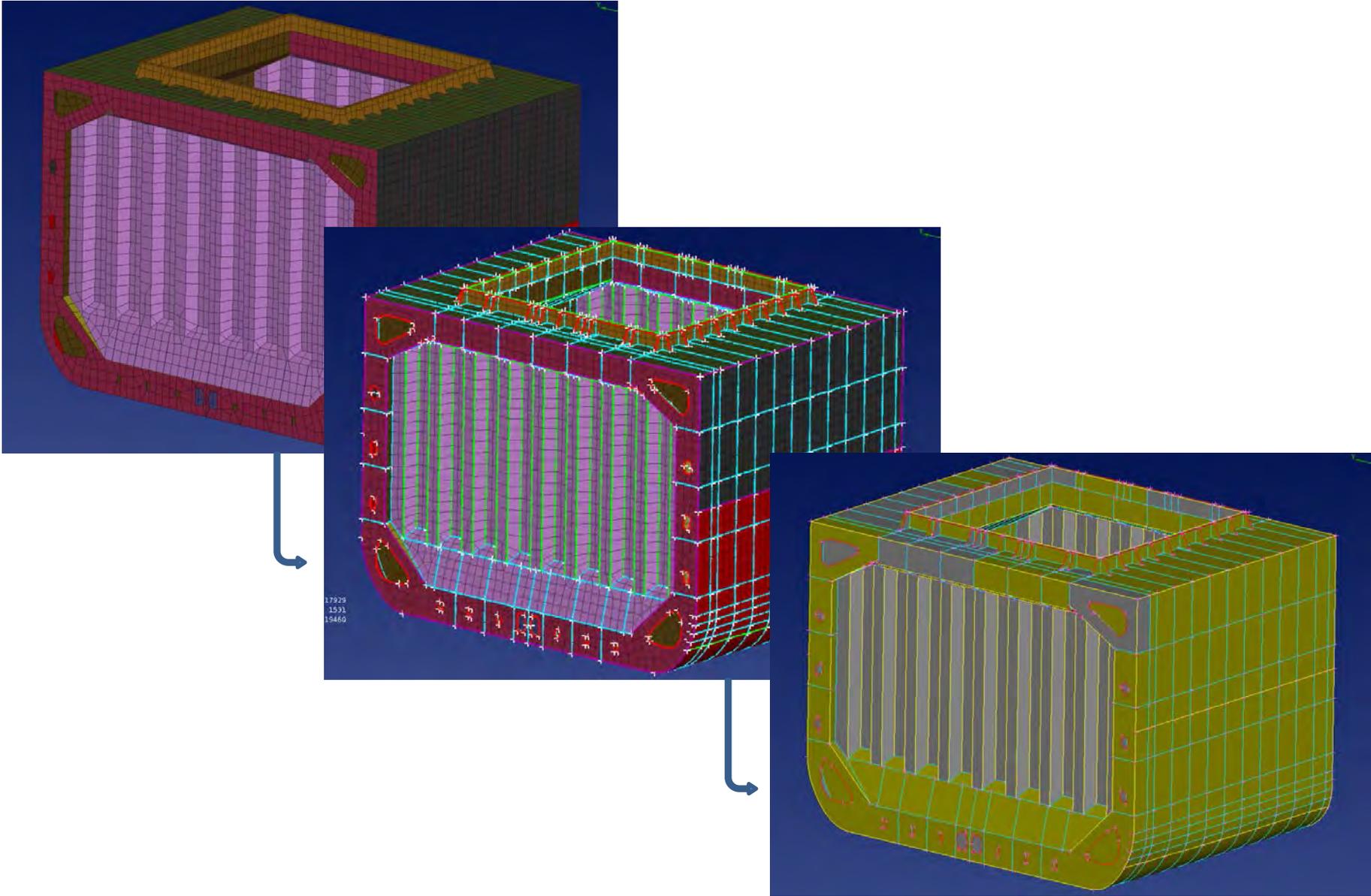
Name	Color	Contents	Mesh Parameters	Quality Criteria	Sta
<input checked="" type="checkbox"/> Meshing_Session		11			Error
<input checked="" type="checkbox"/> Transverse		2	0.25m	0.1min	Error
<input checked="" type="checkbox"/> refinement		0	0.02		
<input checked="" type="checkbox"/> Inverse		4	0.3m	0.1min	Error
<input checked="" type="checkbox"/> Default_Session		5	0.26m	0.1min	Error

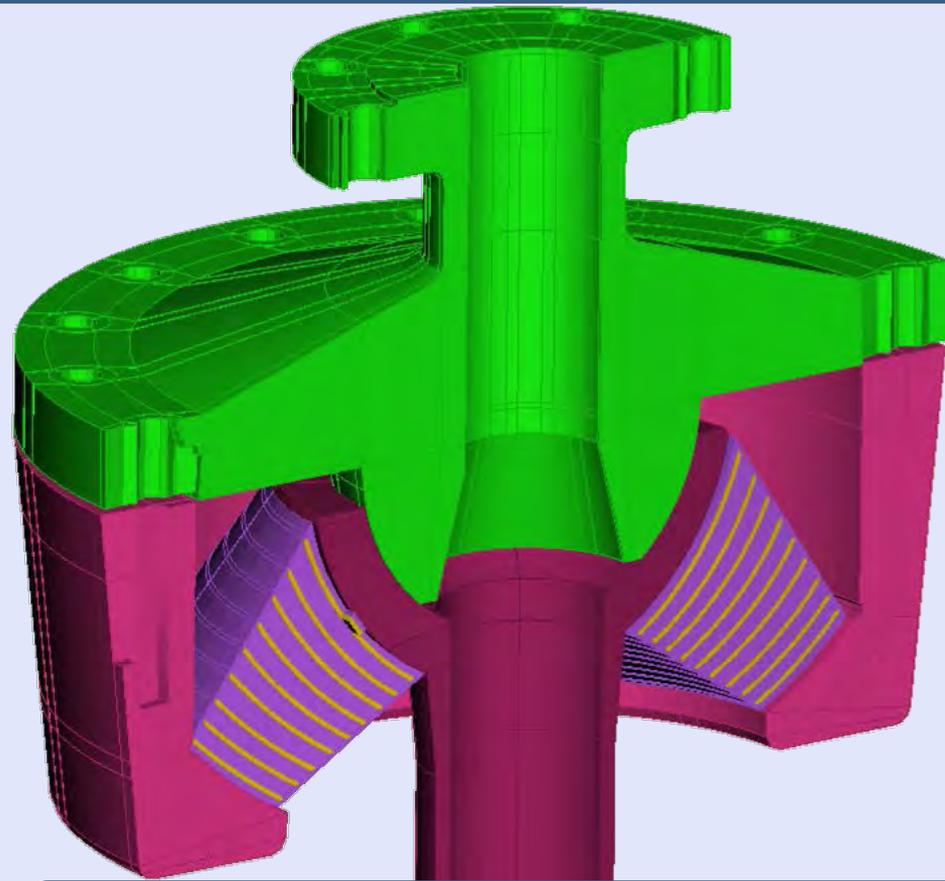
Local refinement

Local mesh refinement of geometry mesh and FE



Automatic definition of geometry from FE model

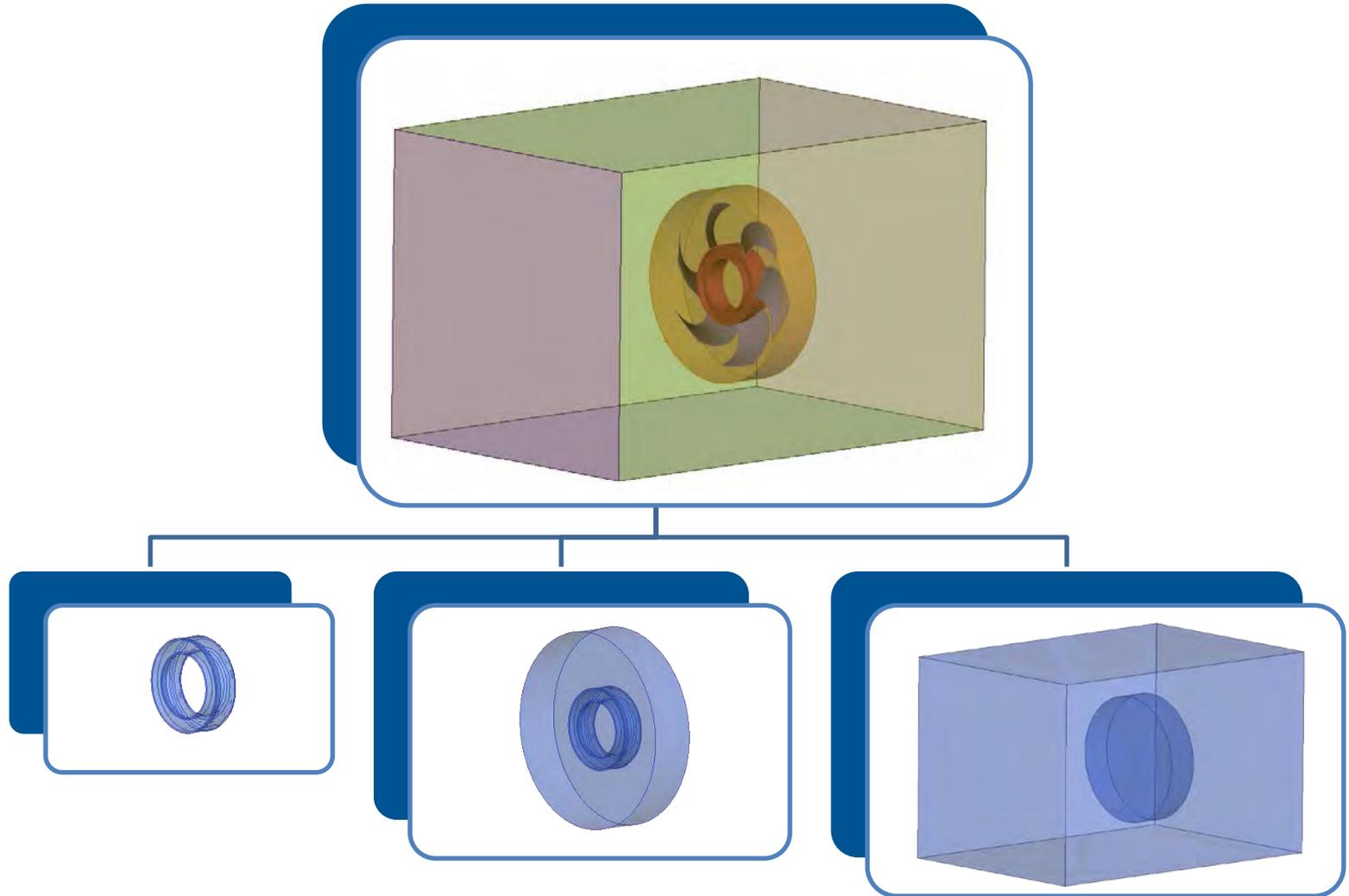




VOLUME MESH

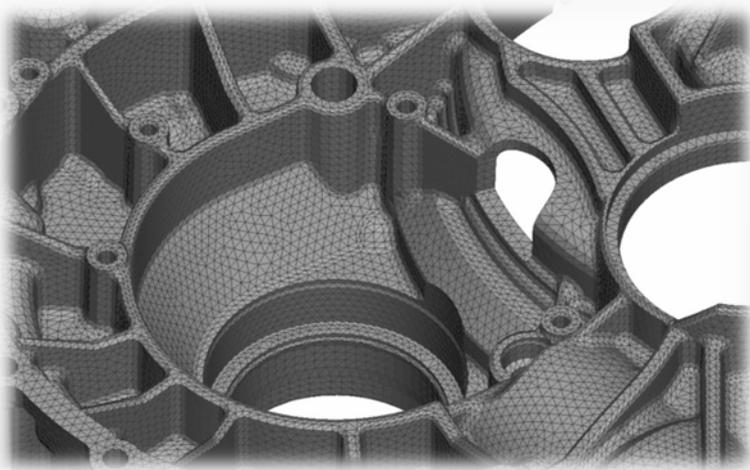
Volume Meshing

Automatic detection of all valid volumes and sub volumes

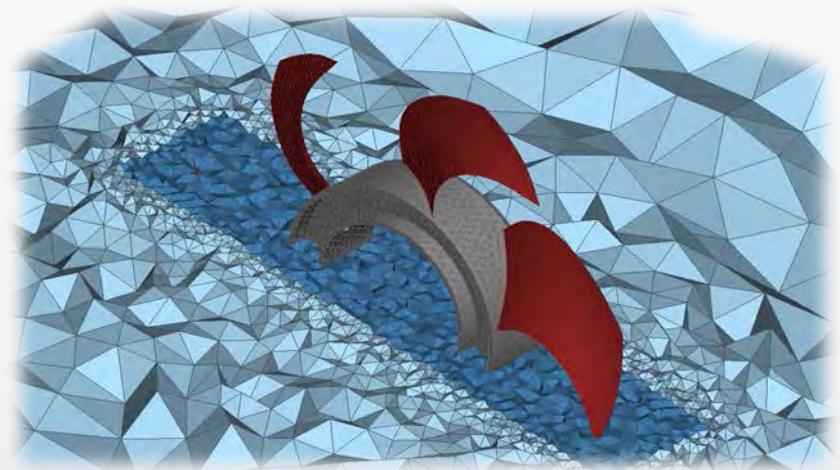


Volume Meshing

Tetrahedral Mesh



..for structural

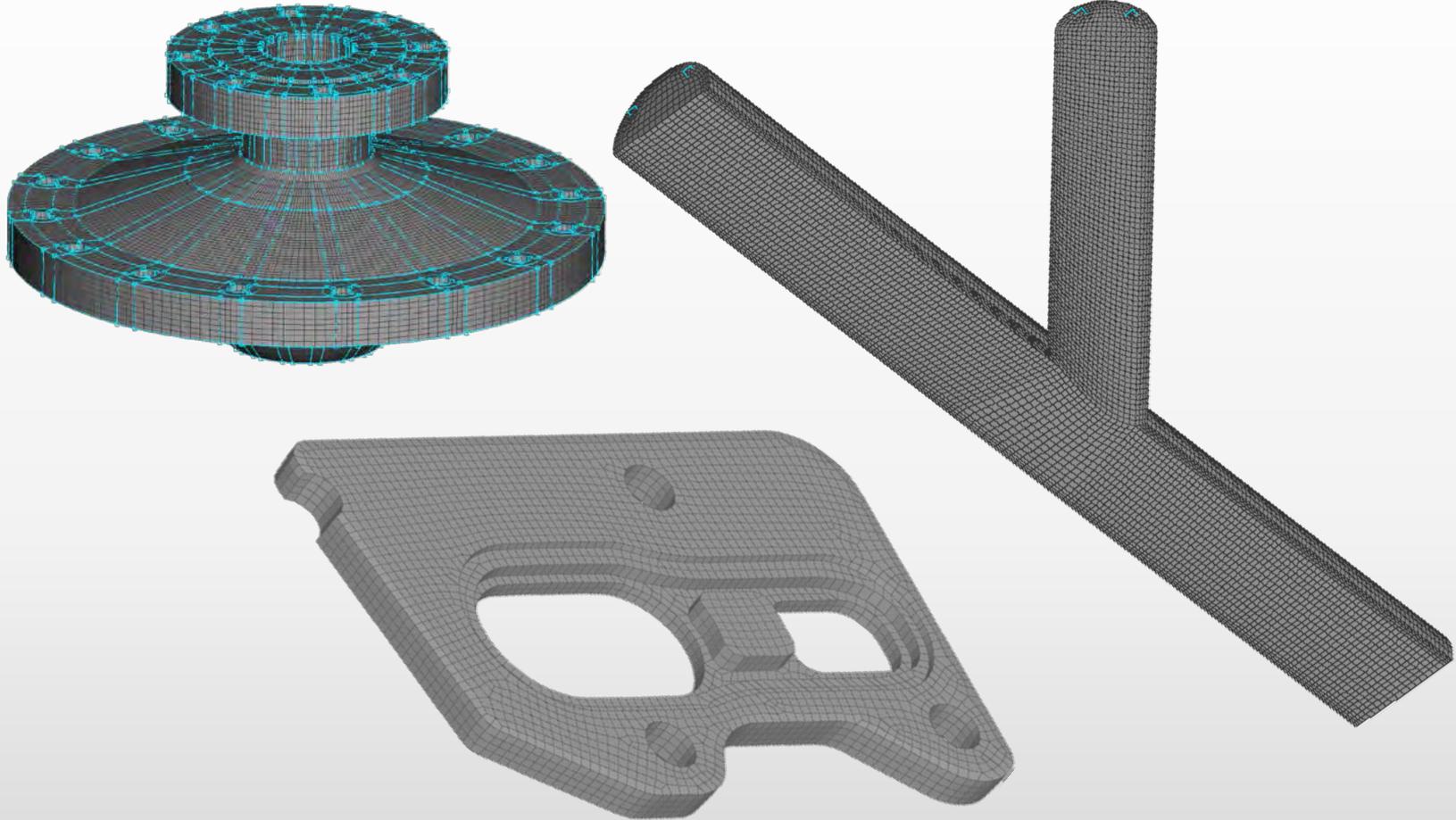


..for CFD

Fully automated through Batch Mesh

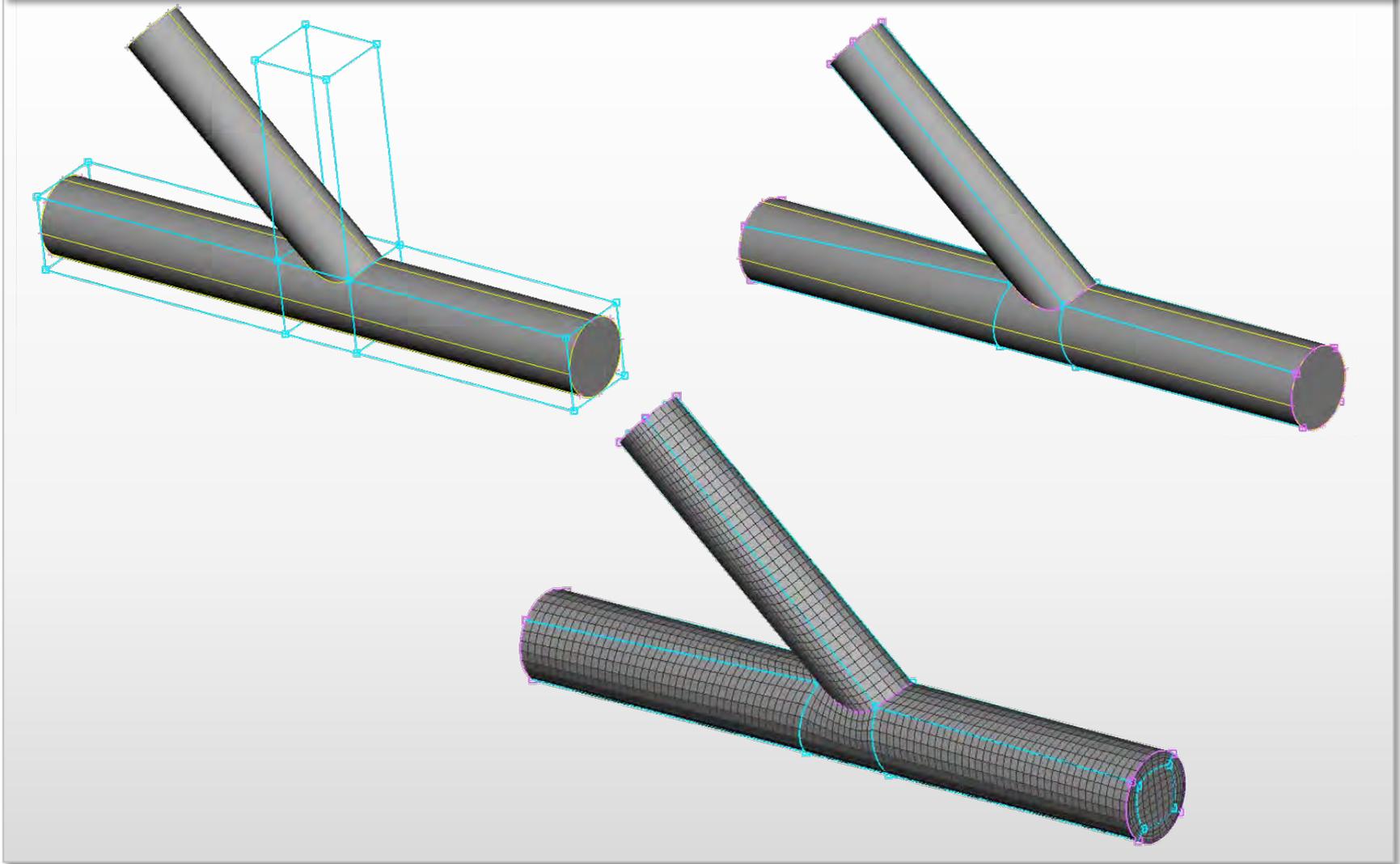
Volume Meshing

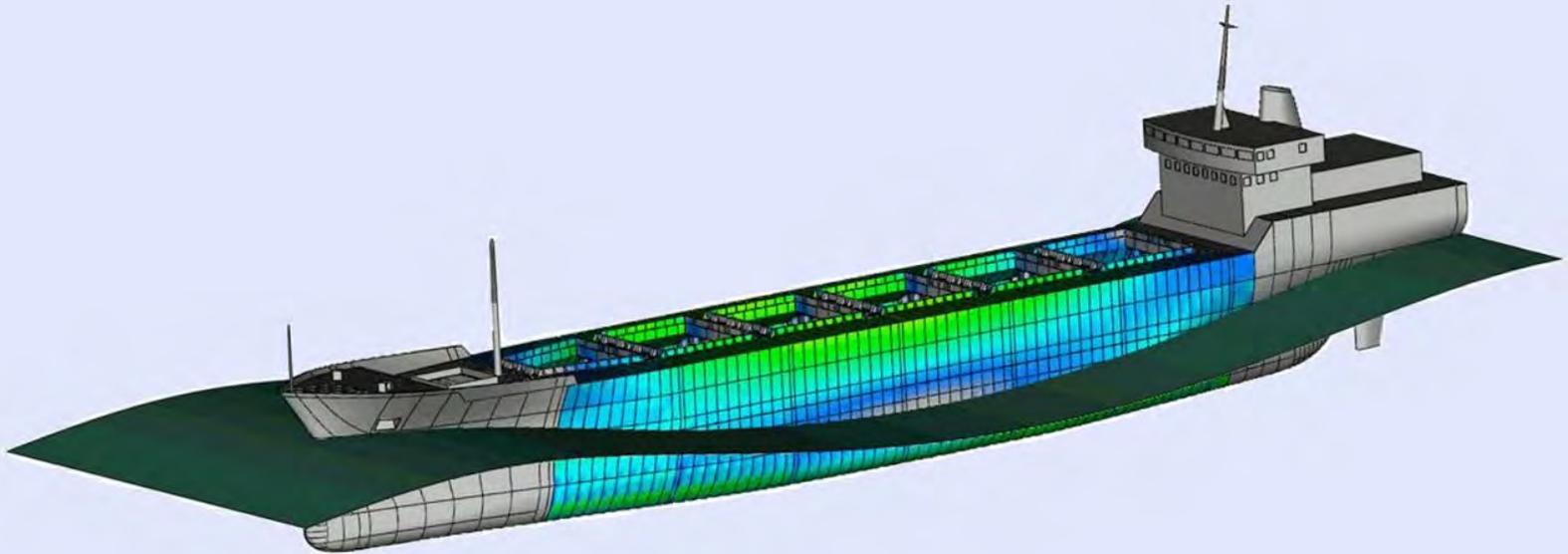
Hexa Meshing



HexaBlock meshing

Hexa meshing based on block structures associated to the model





ANALYSIS TOOLS

Stiffeners creation using beams

Creating database of cross sections of any shape

The screenshot displays a software interface for creating a database of cross sections. The main window shows a 3D model of a ship's hull with stiffeners. A cross-section view is shown on the right, labeled "Cross Section 19". The cross-section is a vertical rectangle with a coordinate system (X, Y, Z) and a local coordinate system (1, 2, 3). The cross-section is defined by the beam type "FB(T)_650x13+220x30".

The "Database" window on the left lists the following items:

Name	Number
ANSAPART	12
CROSS	75
CROSS_CURVE	56
CROSS_SECTION	19
EDGE	
ELEMENT	10698
GEOMETRY	18370
GRID	6240
MATERIAL	23
MORPH	8
PROPERTY	56
SET	1

The "Cross Section 19" window shows the following Geometrical Results:

Parameter	Value
A	0.01175
xs	0.0154468
ys	0.416277
Ix	0.00548179
Iy	1.05064e-05
Ixy	4.24298e-05
a1/a	-0.0792319
I1	0.00551805
I2	7.1801e-06
I1	1.46602e-06
x#	-1.09288e-16
y#	0.65
Cw(s)	1.01103e-06
Cw(m)	1.35303e-35
Gamma	-4.95842e-21
Kx	0.00132806
Ky	9.44835e-05
Wpx	0.00225272
Wpy	0.00268209
A1	0.00101334
A2	0.00739319
A12	0
Ax	0.00101316
Ay	0.0074029

The "Cross Section" window on the left shows a list of cross sections:

id	Name
5	FB(T)_470x12+200x27
6	FB(T)_500x12+150x20
7	FB(T)_450x12+150x20
8	FB(T)_600x12+150x12
9	FB(T)_425x12+150x18
10	FB(T)_375x12+150x18
11	FB(T)_600x13.5+175x20
12	FB(T)_925x18.5+260x18.5
13	FB(T)_300x12+175x25
14	FB(T)_375x15+150x20
15	FB(T)_575x12+175x20
16	FB(T)_625x12+175x20
17	FB(T)_650x12+175x22
18	FB(T)_575x12+200x25
19	FB(T)_650x13+220x30

The "Info" window at the bottom left shows the following data:

Shell	quads	total
quads	4948	
trias	1726	
total	6674	

The "Info" window at the bottom right shows the following data:

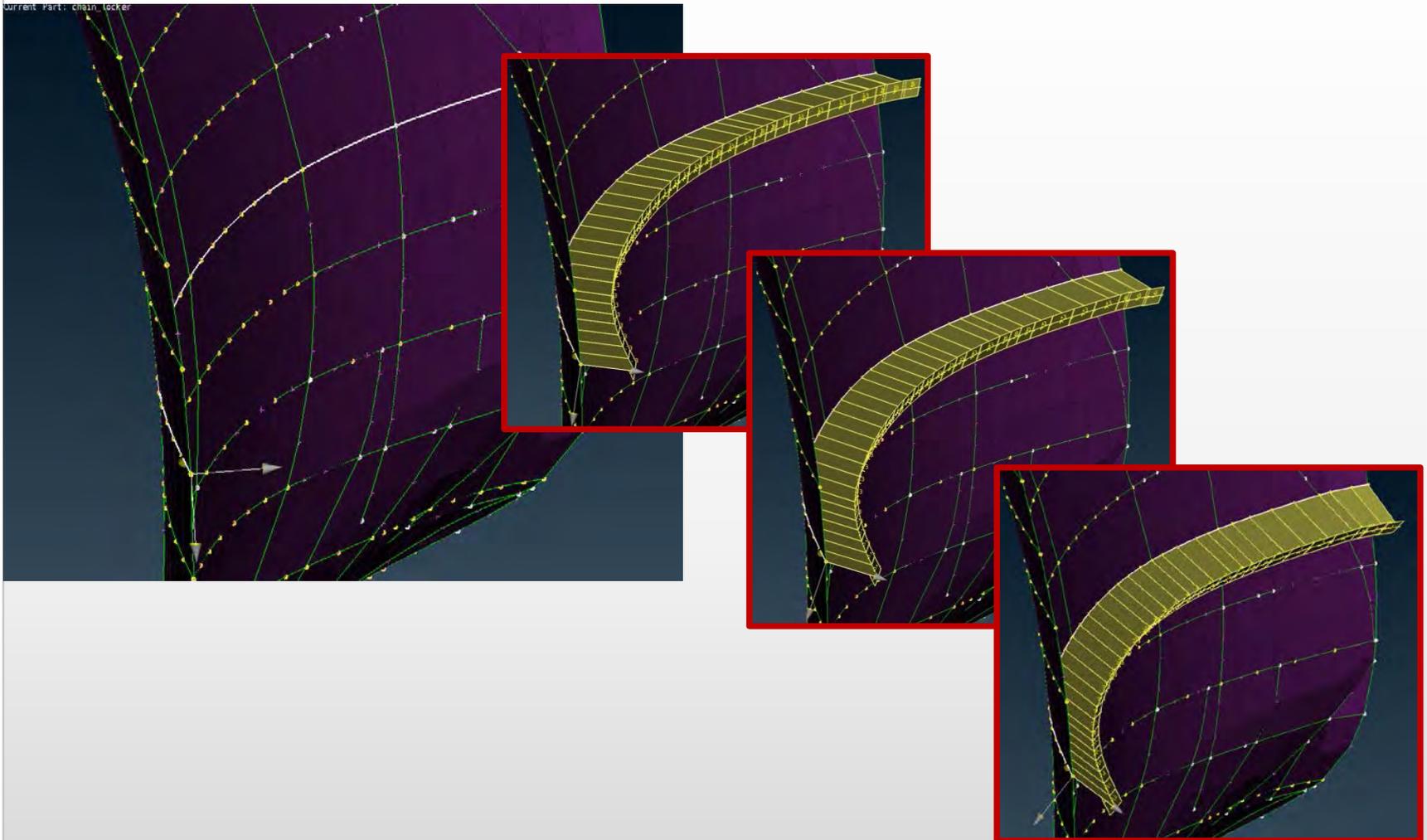
CROSS_SECTION	total	selected
CROSS_SECTION	19	selected 1

The "Info" window at the bottom left also shows the following data:

Image saved in	data
/home/dimgeo/CROSS_TOOL02.png	
/home/dimgeo/CROSS_TOOL03.png	
/mnt/raid_disk/DATA	

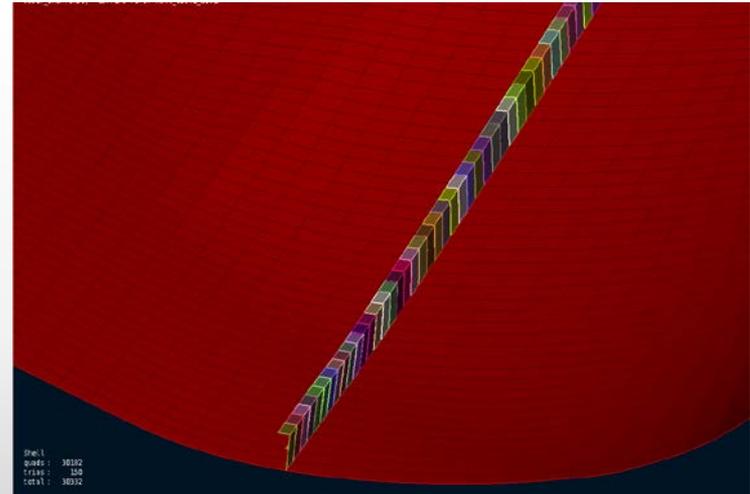
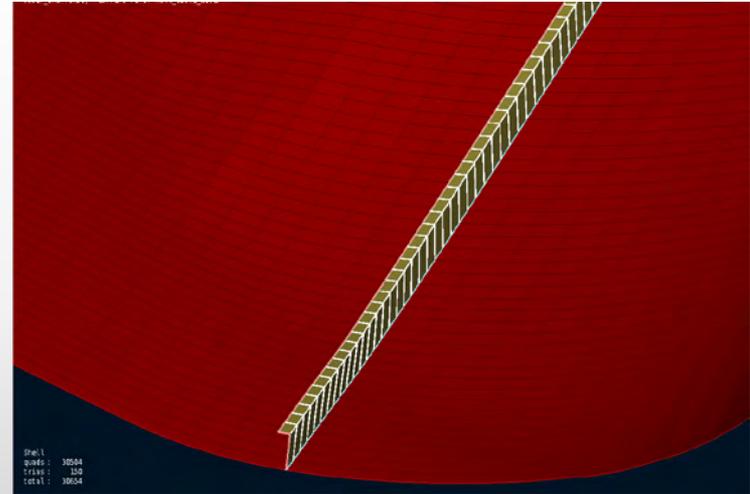
Stiffeners creation using beams

Creating stiffeners using beams with a selected cross section



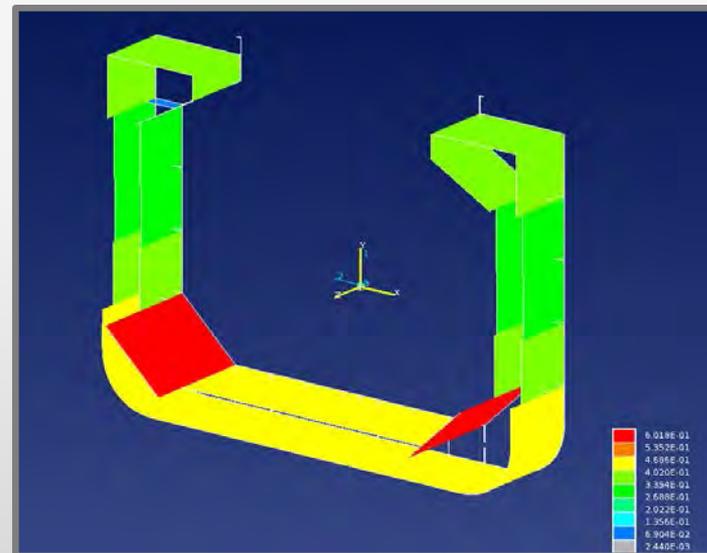
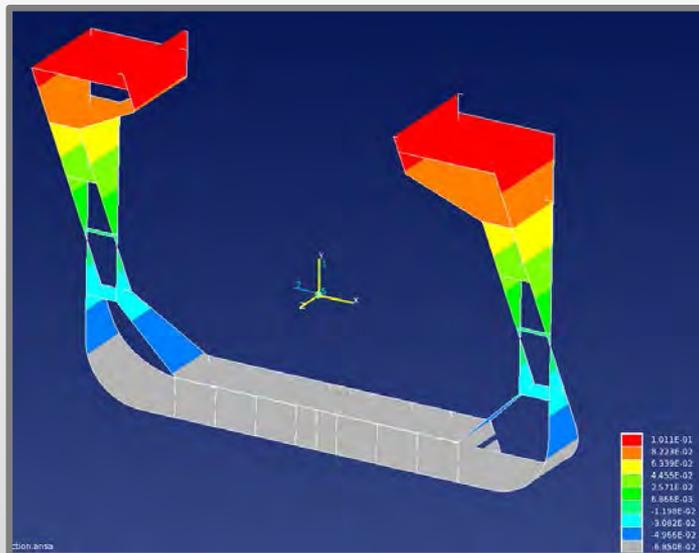
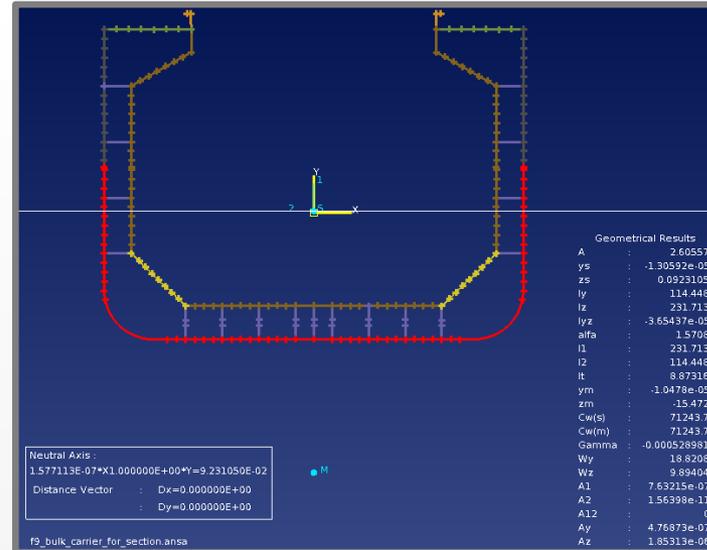
Stiffeners replacement with beams

Replacement of standard cross section reinforcements with beams



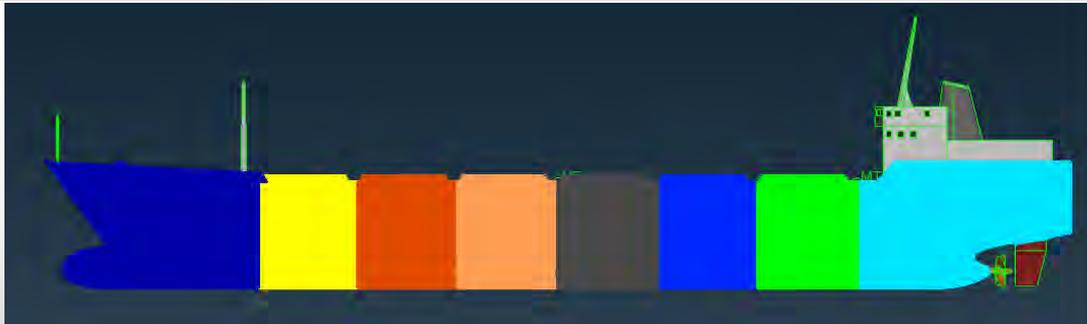
Calculating Cross Sections

- Extraction of Cross Sections from the geometrical model
- Editing of Cross Sections
- Calculating geometrical results (A, ly, lz, etc..) neutral axis stresses and moments



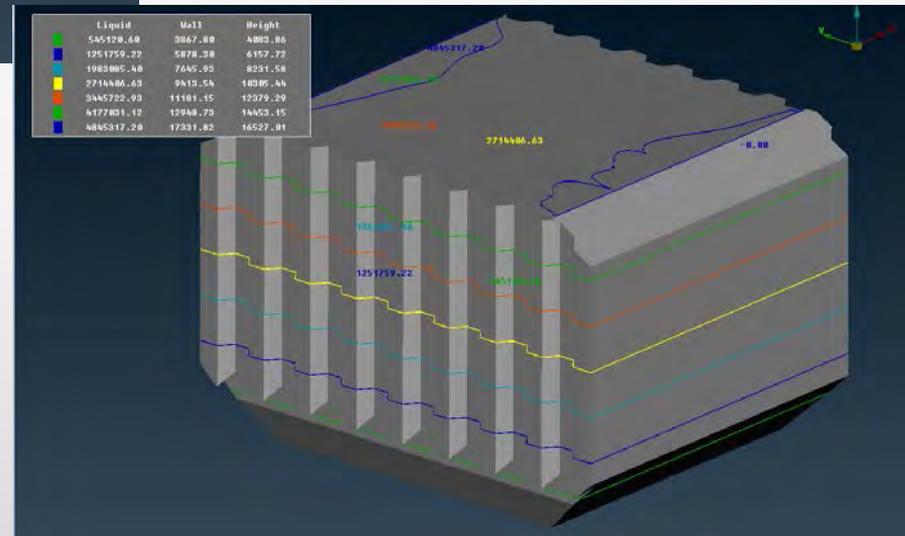
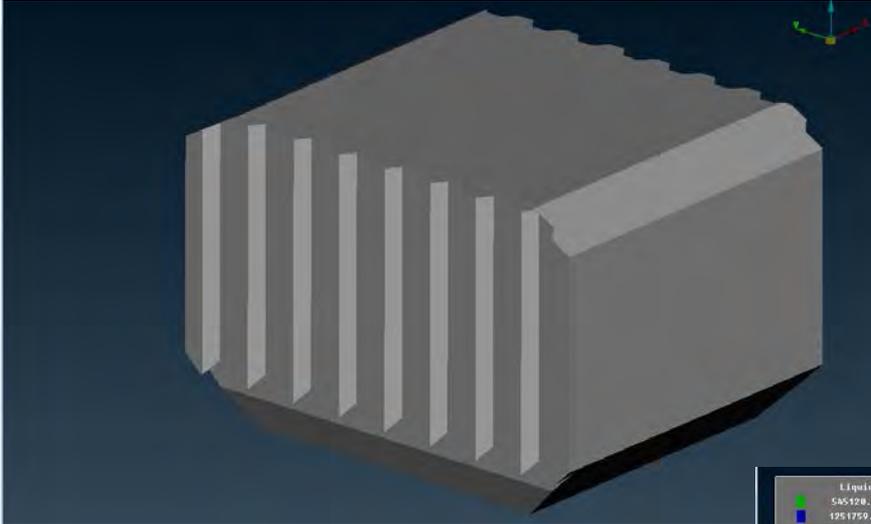
Mass distribution

Applying additional mass to model by fulfilling balance criteria



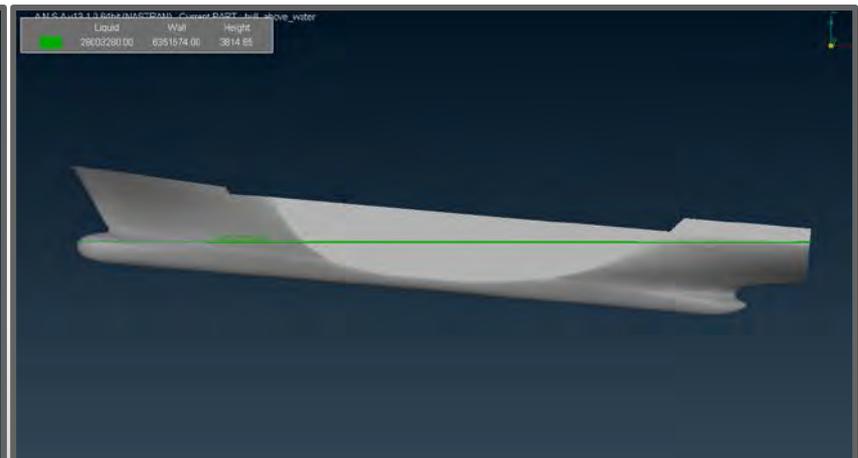
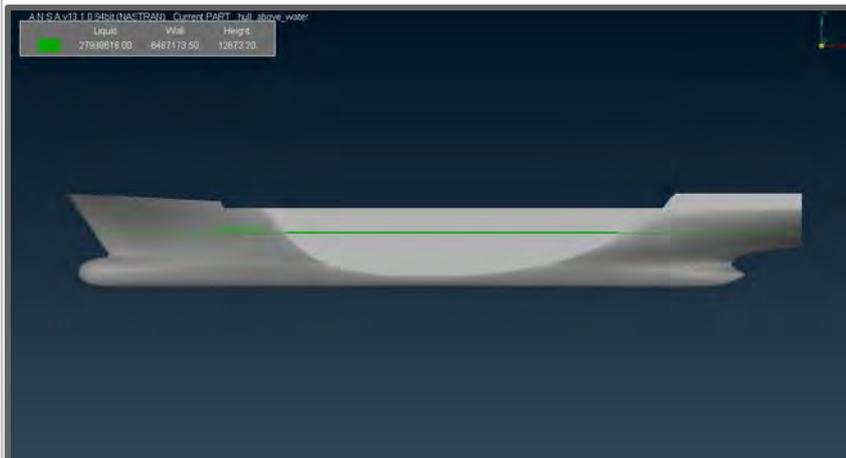
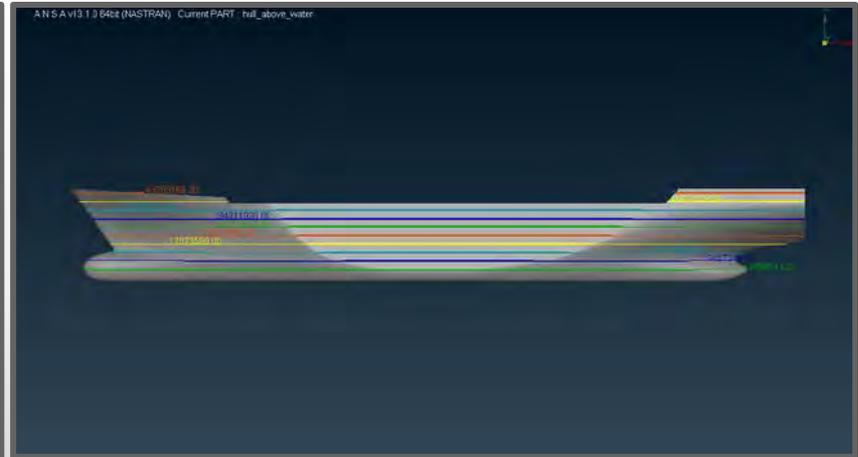
Tank Tool

Calculating the level of cargo in a tank



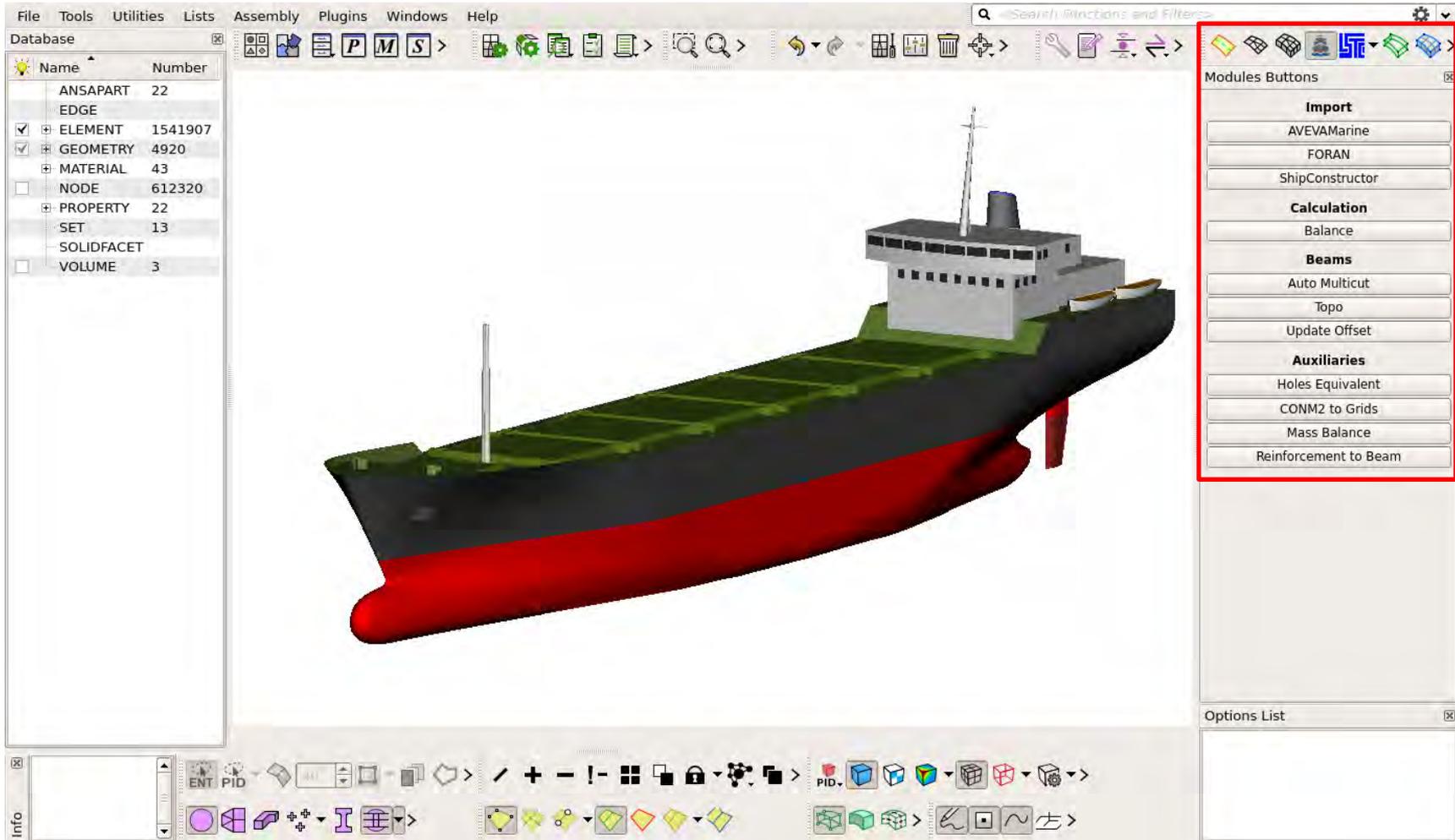
Waterline calculation

Calculating the waterline for variable ship loading



Marine & Offshore Tool

Dedicated tools for Marine & Offshore applications



Marine & Offshore Tool

Wave profile creation

The screenshot displays a software interface for creating wave profiles. The main workspace shows a 3D visualization of a blue wave profile. A dialog box titled 'Balance' is open, allowing the user to configure the wave profile parameters:

- Wave type: Trochoidal
- Wave's length: 350
- Wave's height: 7
- Wave's width: 70
- Rotation angle: 0
- Offset in X-axis: 0

The 'Balance' dialog also includes a 'Create' button and a 'Loading' section with options for 'Mass | Balance | Buoyancy' and 'Shear Force | Bending Moment'. The interface includes a database tree on the left, a toolbar at the top, and a 'Modules Buttons' panel on the right.

✓ Sinusoidal

✓ Trochoidal

Marine & Offshore Tool

Ship balance calculation

Balance

Wave Profile

Loading

Mass | Balance | Buoyancy

Physical Properties

Gravity: 9.81

Density: 1024

Mass and Loads Information

Current Mass: 1741558.8981

Loads: 289000000.0000

Current C.O.G (X, Y, Z): (82.2193, 0.0002, 8.1695)

Total Target Mass: 1741558.8981

Target C.O.G

X: 82.2193

Y: 0.0002

Z: 8.1695

Update

Model Setup

SET containing the hull: 100

SET containing the sea profile: 50

Calculate balance based on a coarse model to speed up the process

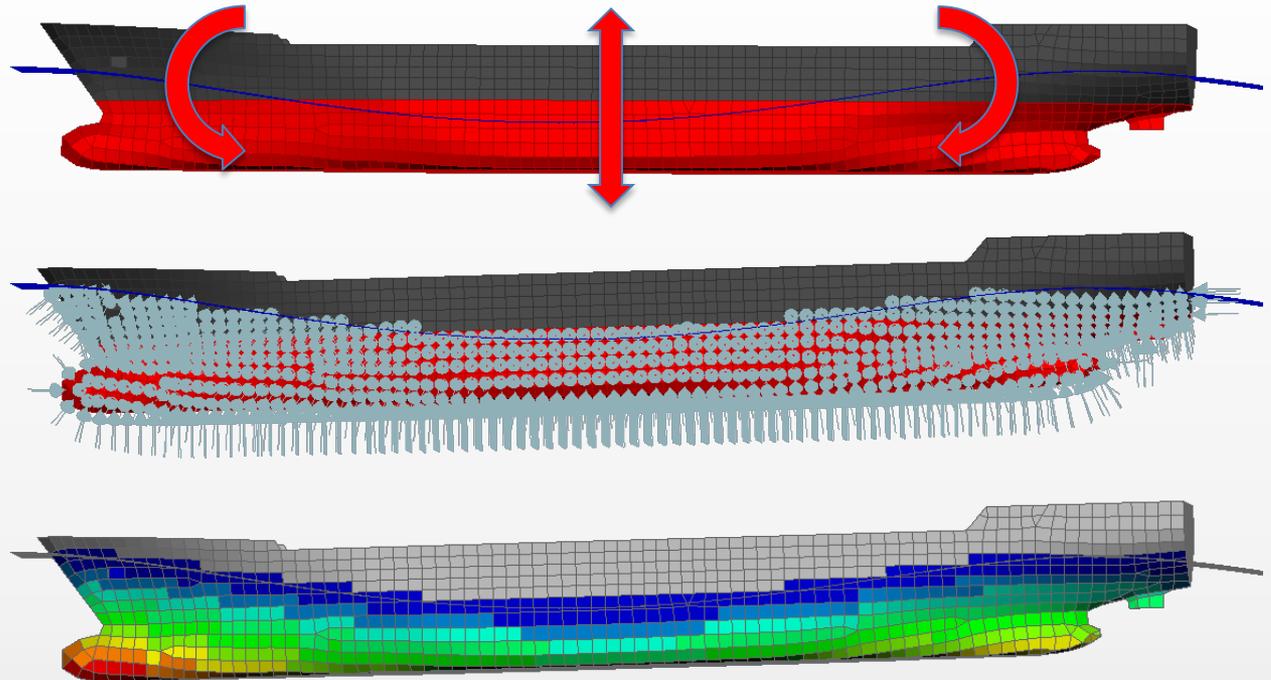
Coarse model target element length: 12

Weight-Buoyancy accepted difference(%): 0.5

C.O.G - C.O.B accepted distance: 0.15

Balance

Shear Force | Bending Moment

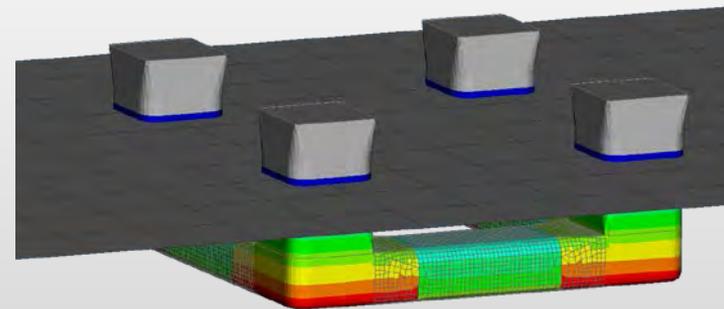
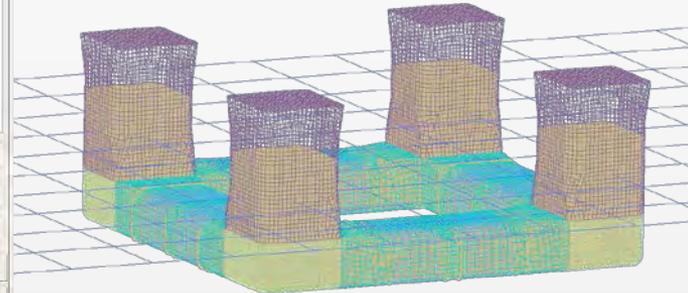
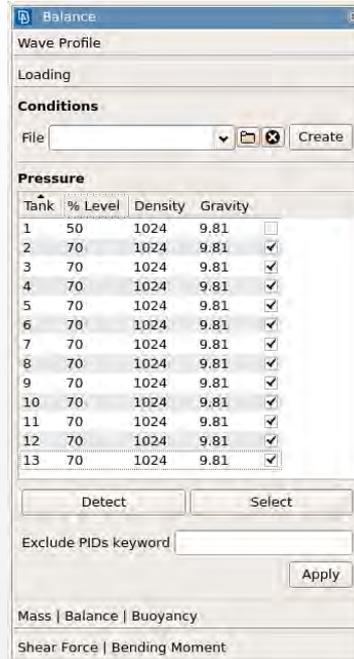


- Balance through
 - Heave
 - Pitch
 - Roll
- Convergence criteria
 - Weight – Buoyancy difference
 - C.O.G – C.O.B distance
- Buoyancy assignment

Marine & Offshore Tool

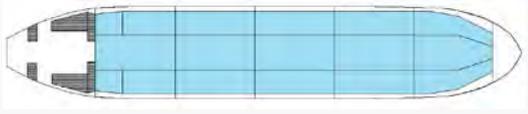
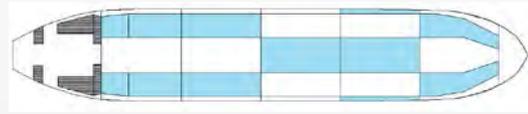
Pressure loads application

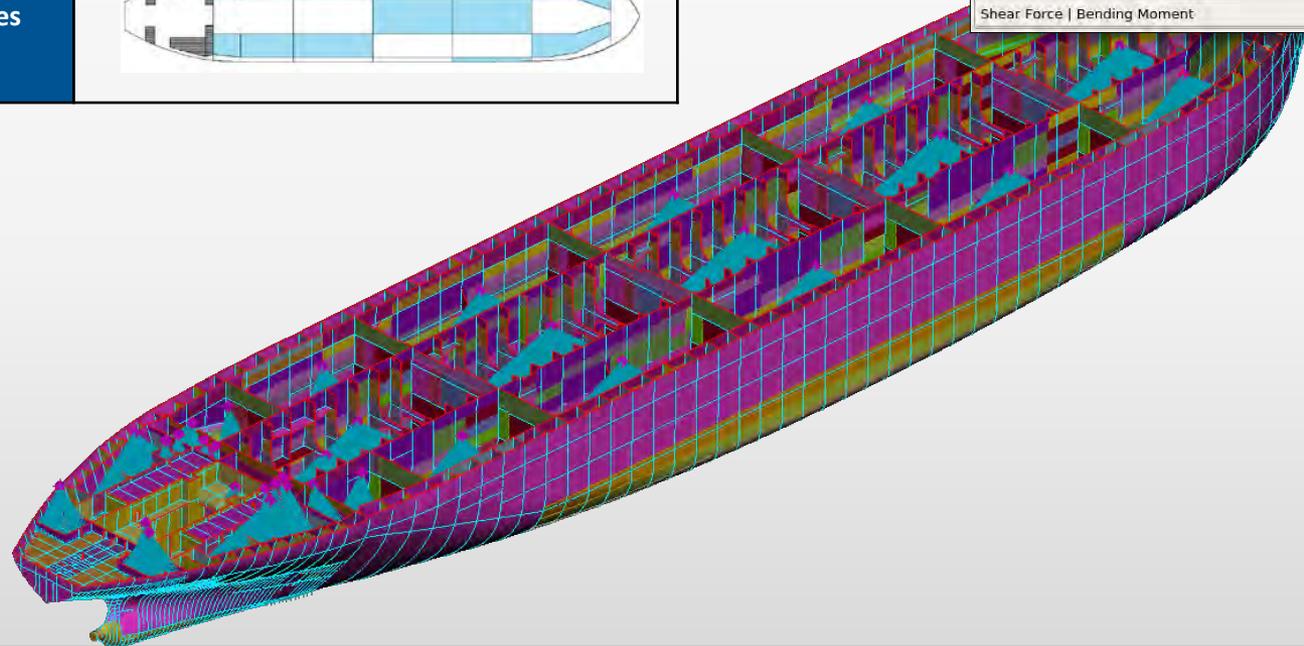
- Automatic recognition of tanks
- Definition of load properties
- Exclusion of areas from pressure application
- Pressure application on selected tanks



Marine & Offshore Tool

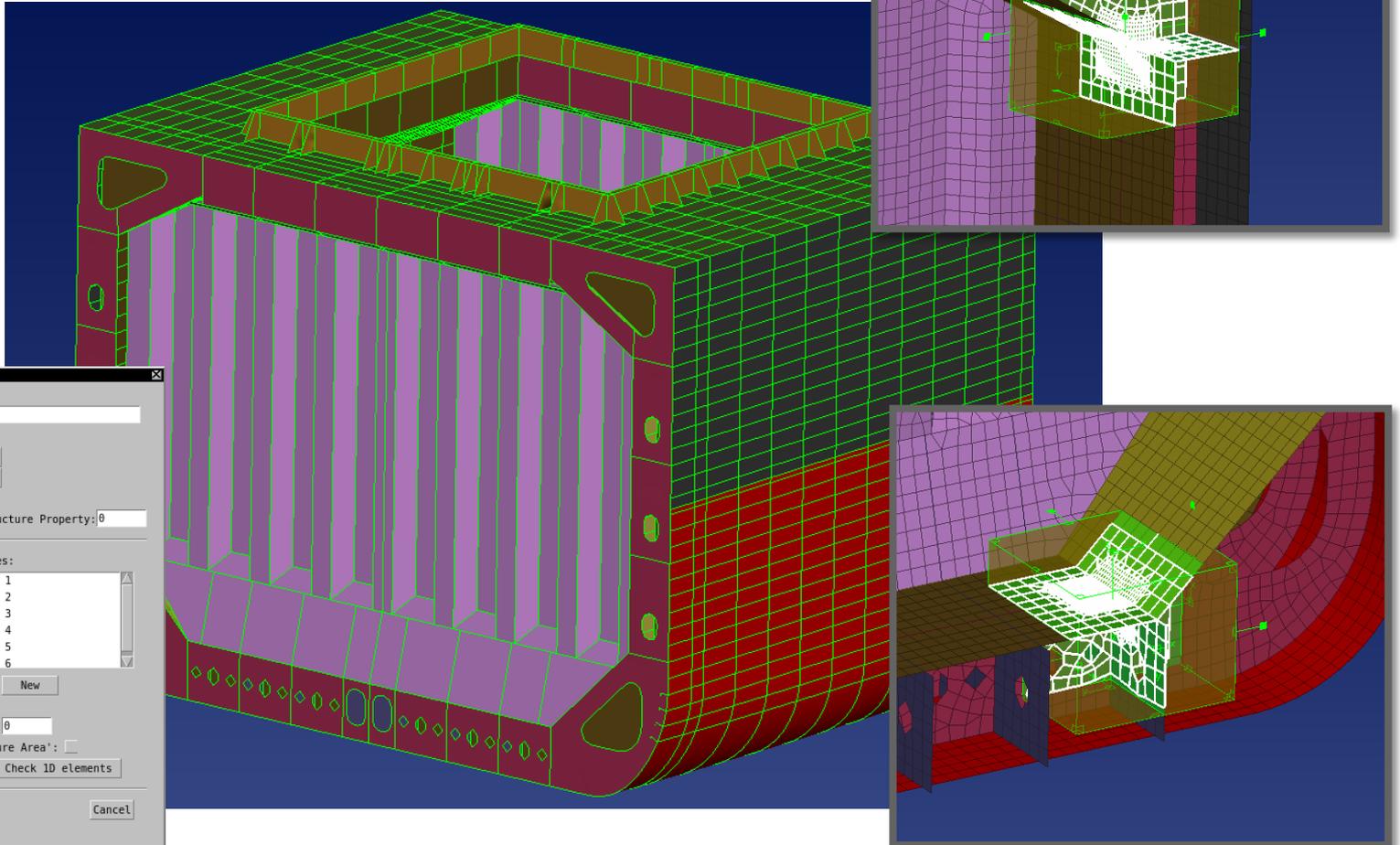
Loads application through mass

Ballast arrival condition (L.C. 1) Displacement: 145647 tones Draft: 9.69 m Trim: 2.12 m	
Full-load departure condition (L.C. 2) Displacement: 364074 tones Draft: 22.52 m Trim: 0.11 m	
Departure with partial load (L.C. 3) Displacement: 229276 tones Draft: 14.78 m Trim: 3.05 m	



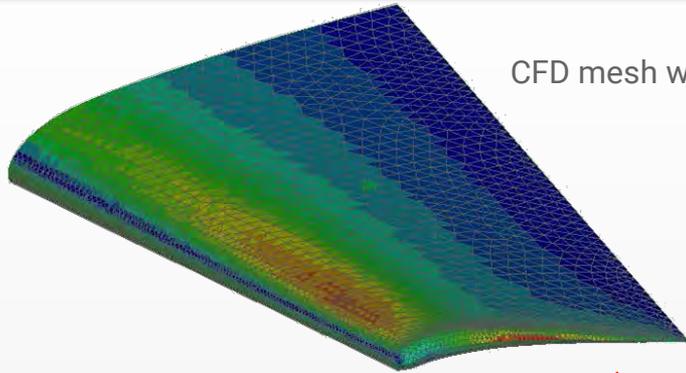
Sub-structuring and Sub-modeling

- Local refinement at the areas of interest
- Output local and global models separately
- Merging local and global back to one model

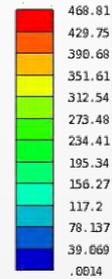


Mapping CFD results to FEA models

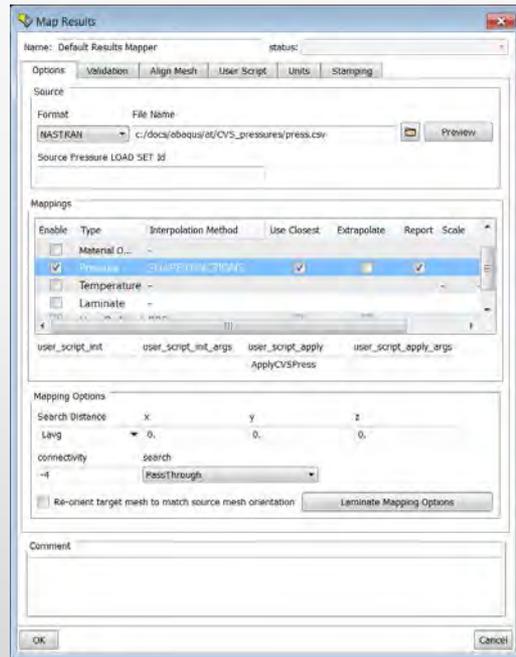
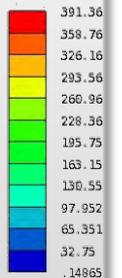
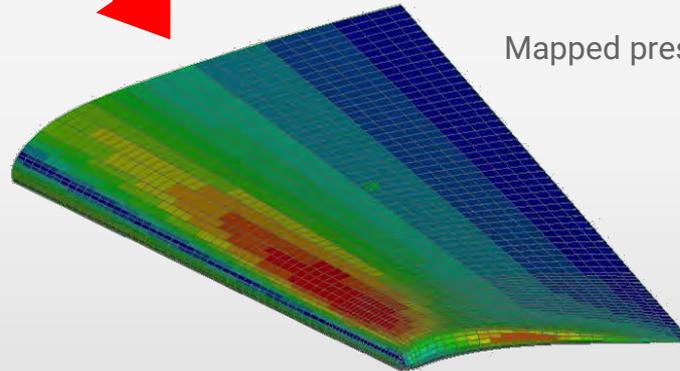
Map pressure or temperature results from a CFD simulation to an FEA model

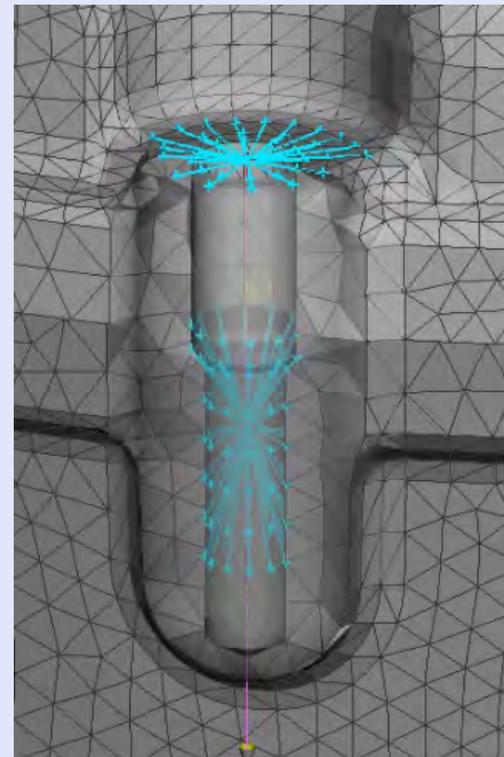
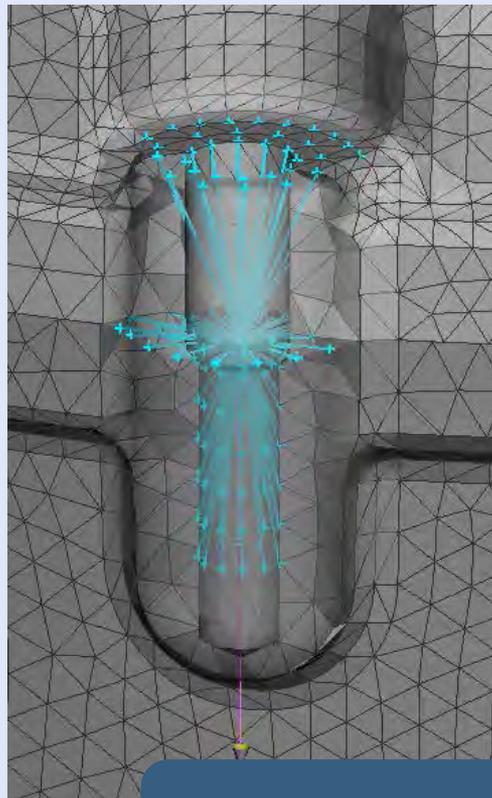
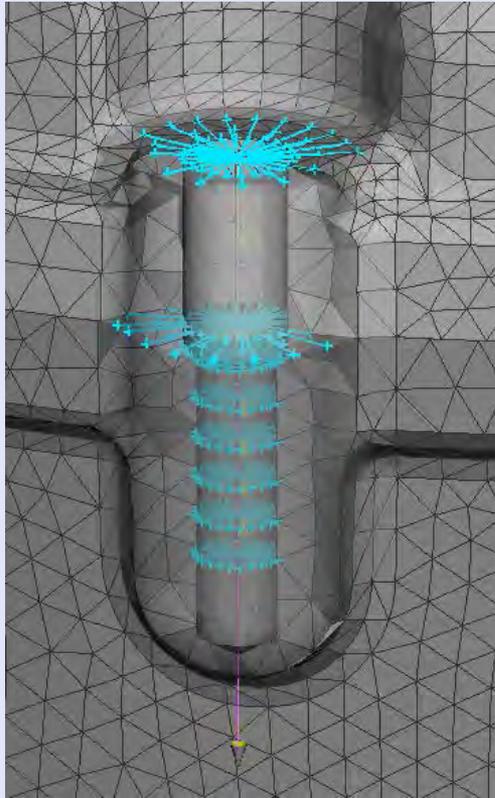


CFD mesh with pressure results



Mapped pressures on FEA mesh

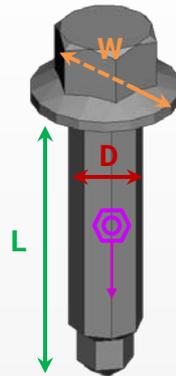




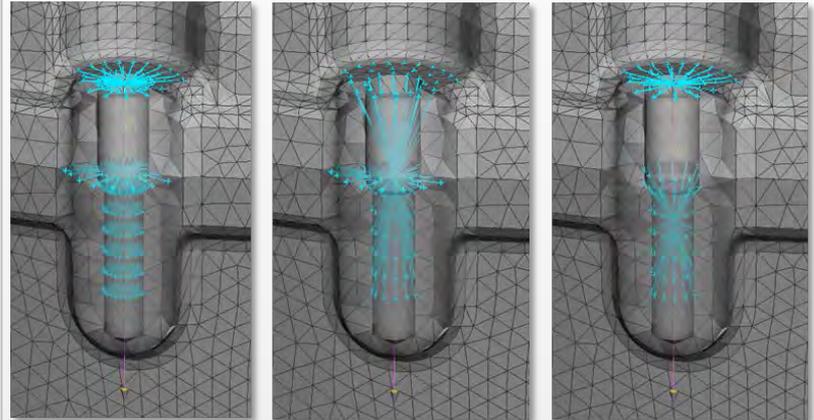
ASSEMBLY

Connection Manager: Bolts

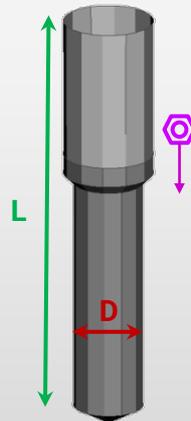
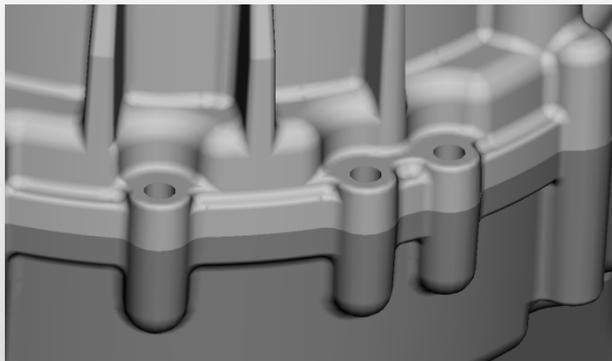
From bolt geometry



Numerous realization patterns

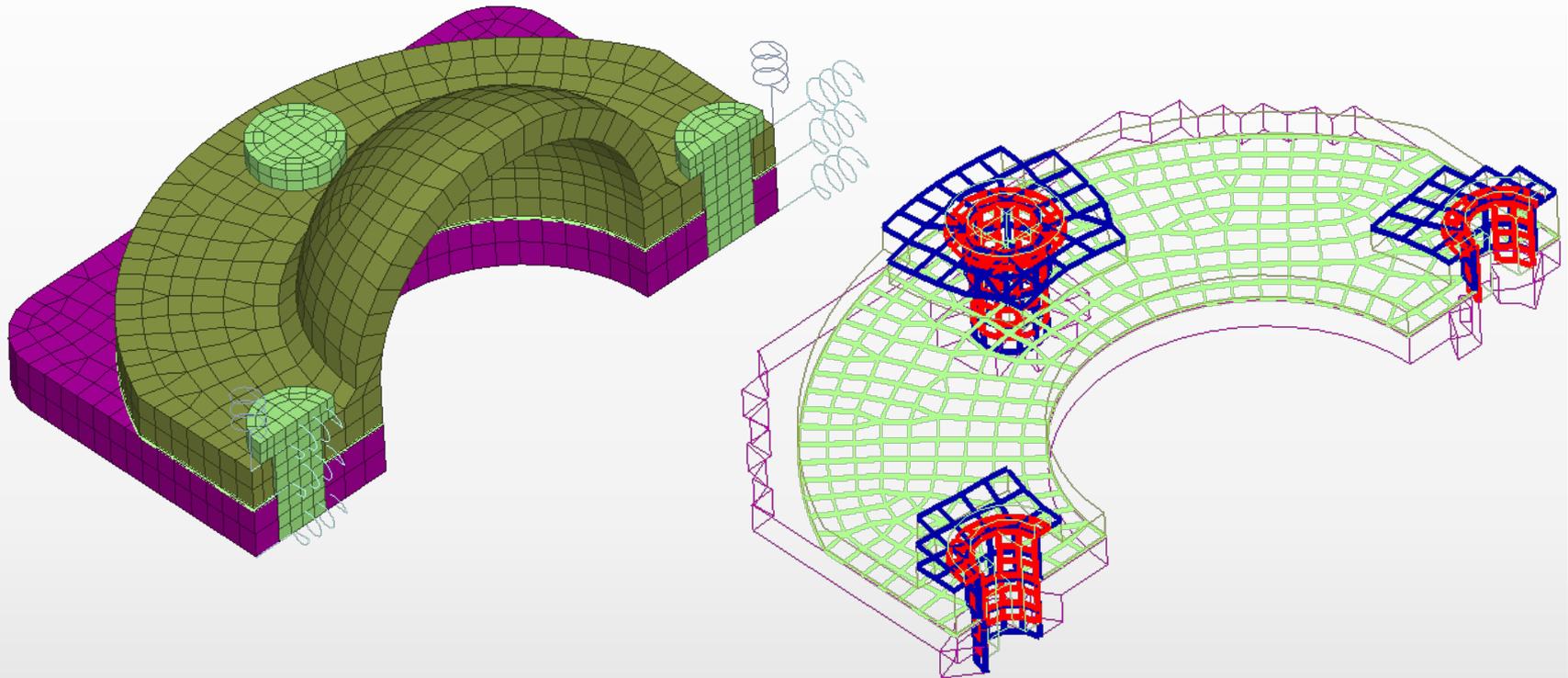


From tubes



Automated Contact definition

Contact interfaces



Automatic detection of contact pairs based on proximity

Analysis Set-up

Material database

- Loaded automatically during start-up
- Can load one MatDB per deck
- Materials update from MatDB:
 - By material name
 - By material id

Material mapping



Automatic transfer of curves and tables between frequently used materials

Analysis Set-up

Model validation through build in Checks

Hundreds of model checks

General and solver checks

Automatic fix functions

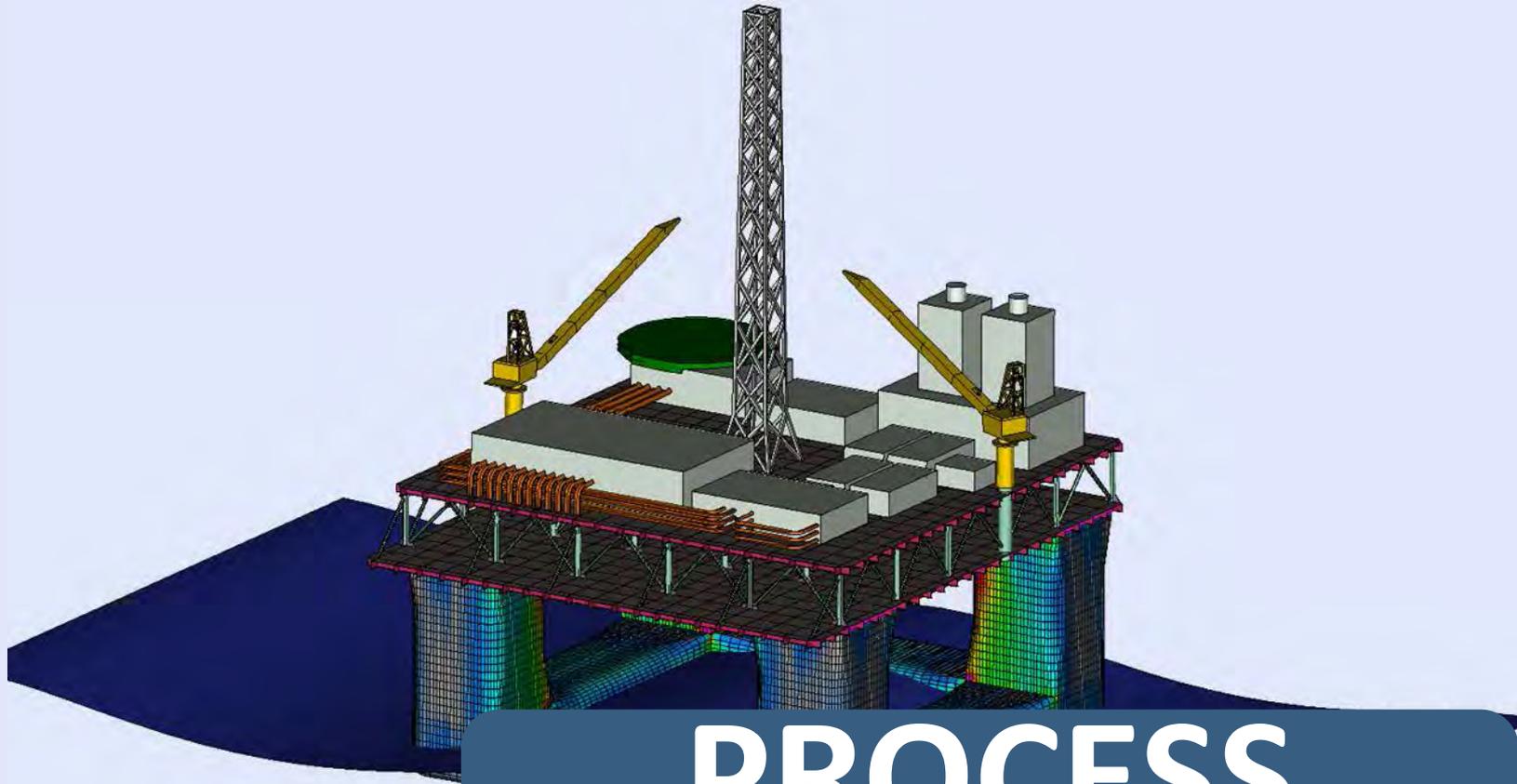
Custom checks through script

Check templates

Color coding of results

Type	Entity	ID	Description
[-] Duplicate			
[-] PLINKS			
[-] Mass Scale Ratio			
[-] Initial Velocity			
[-] Incomplete			
[-] Funct:End Time			
[-] Curves:Axes			
[-] Dependency Pamcrash			
[+] Check Contact 409999			
[-] Undefined Properties			
[+] Undefined Materials			
[-] Range			
[-] Multiple Boundary Conditions			
[+] Massless			
[+] Free:Whole DB			
[-] Warning	NODE	313	Node is Free and doesn't belong to any element
[-] Warning	NODE	312	Node is Free and doesn't belong to any element





PROCESS AUTOMATION

Process Automation

Python Scripting Language

Task Manager



Scripting Capabilities using Python

Entities Handling

Create Delete
Modify Acquire Info

Custom GUI

Line edits Buttons
List boxes Check buttons
Radio groups Tables

ASCII files I/O

Read lists Write XML files
Create lists Read XML files

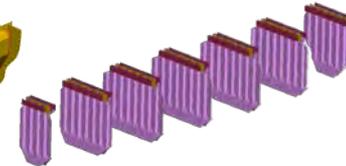
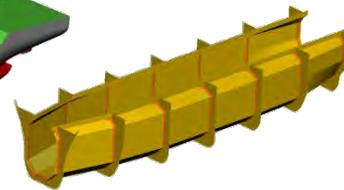
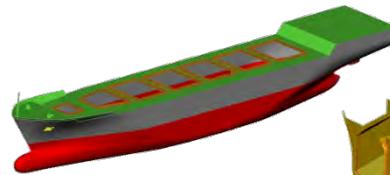
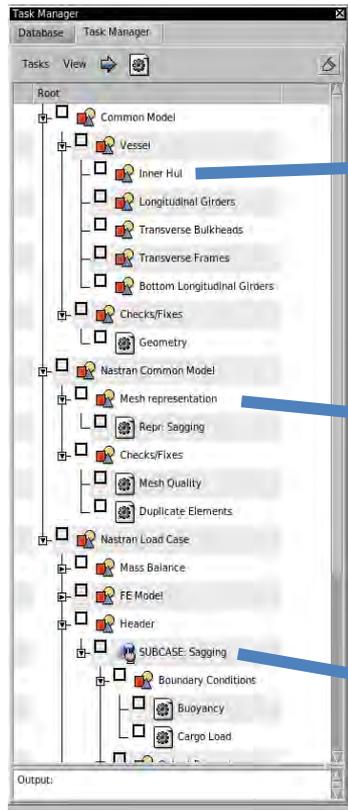
ANSA core functionality in batch

CAD files translation
Geometry checks
Batch Mesh
Middle skin extraction

Process Automation

Python Scripting Language

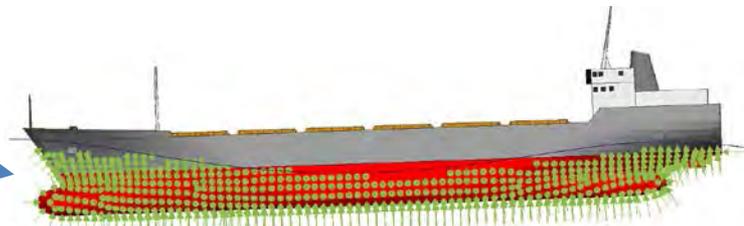
Task Manager



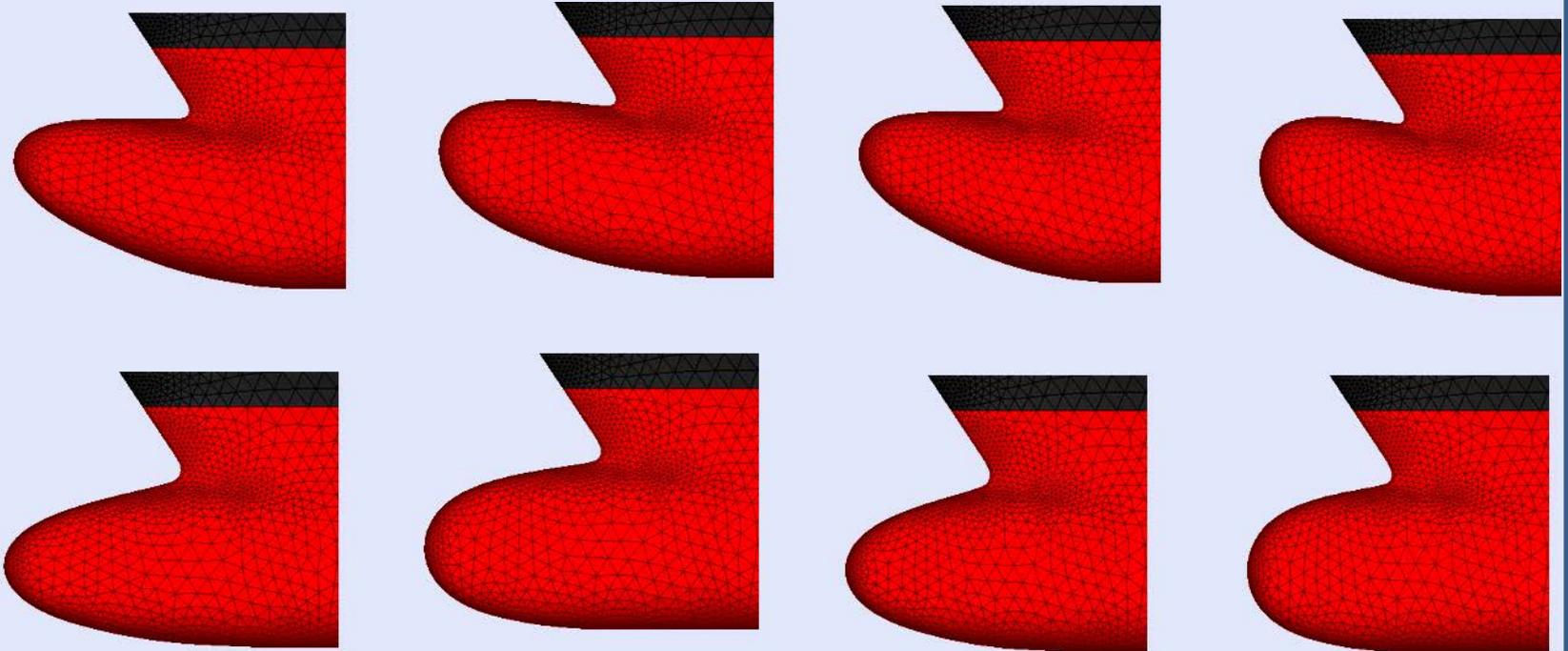
Assembly



Meshing



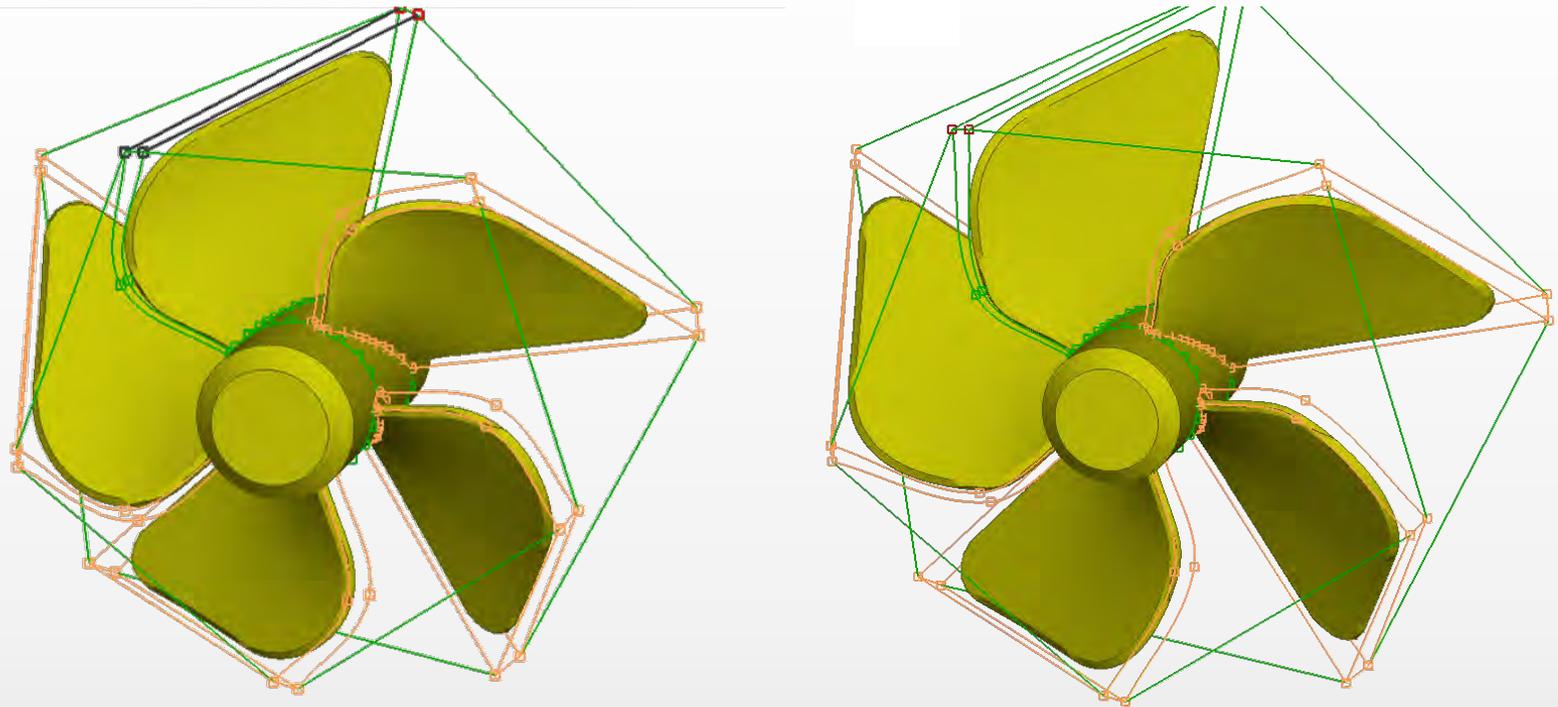
Loading



MORPHING - OPTIMIZATION

Shape optimization using the Morphing Tool

Parametric morphing applied on FE or geometry



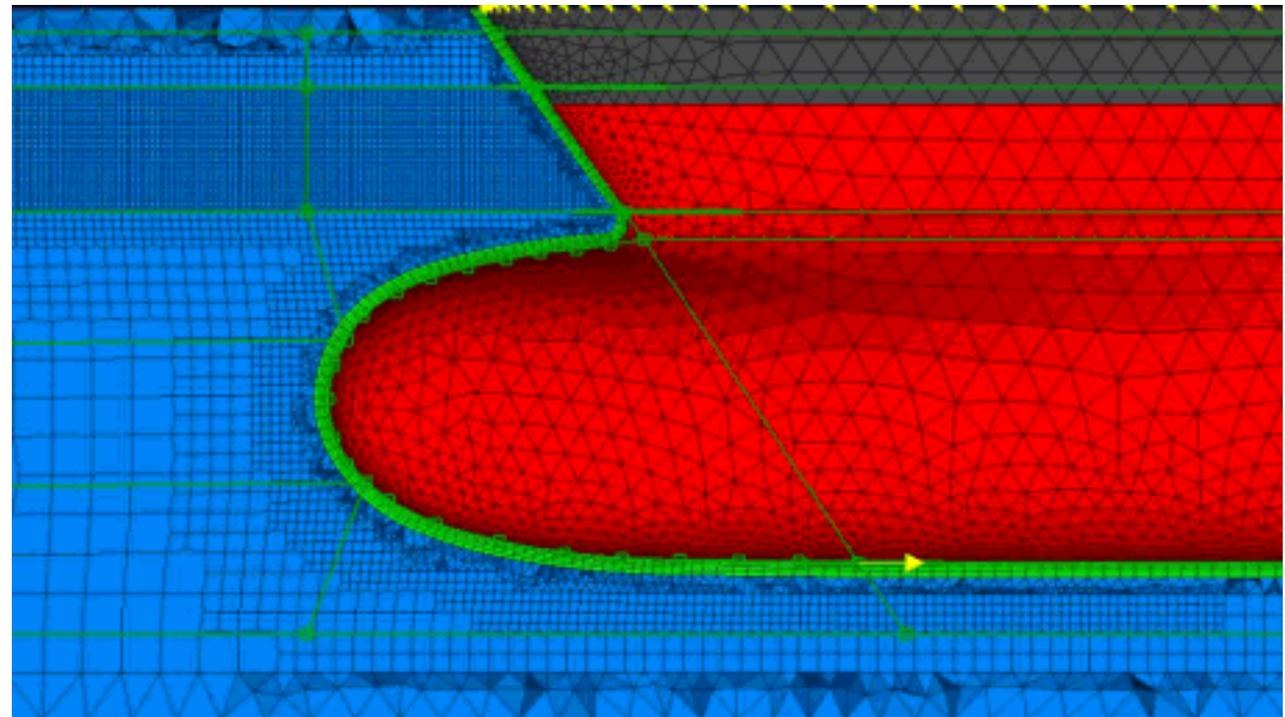
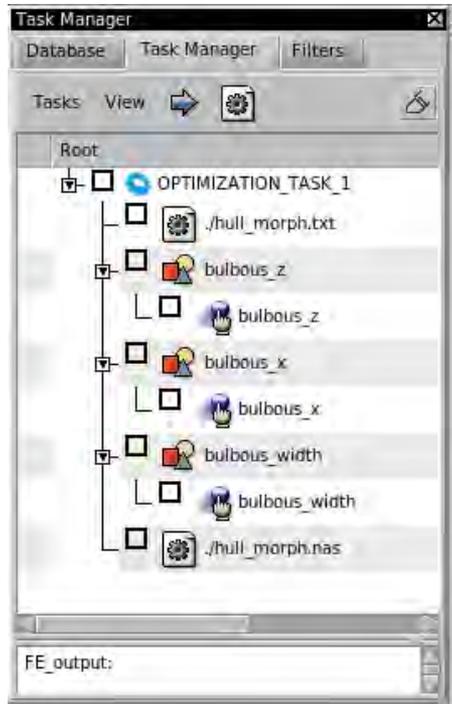
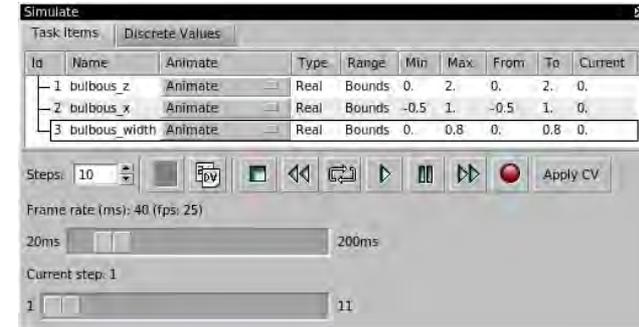
PARAMETERS

Id	Name
1	twist_angle
2	tip_length
3	diameter

total 3 selected 0

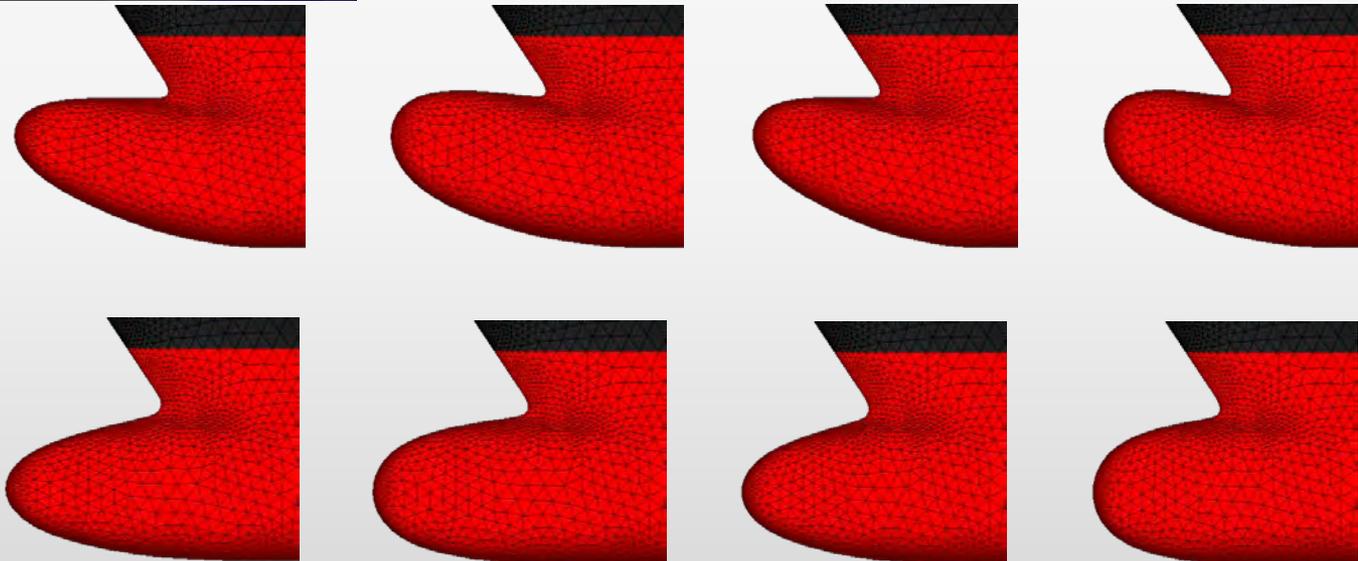
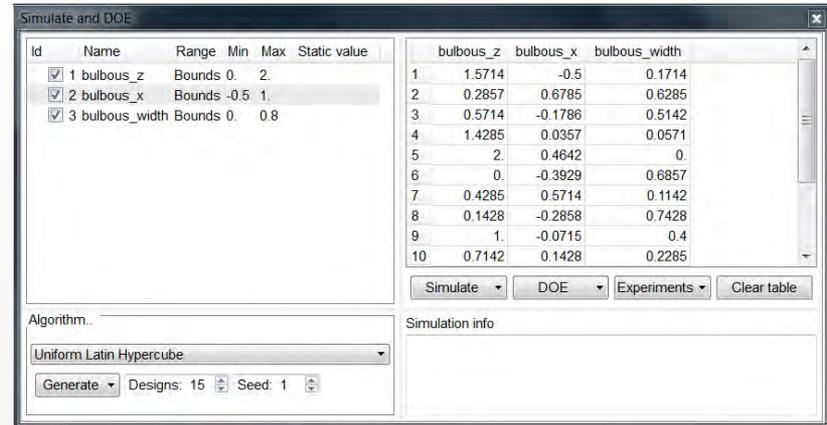
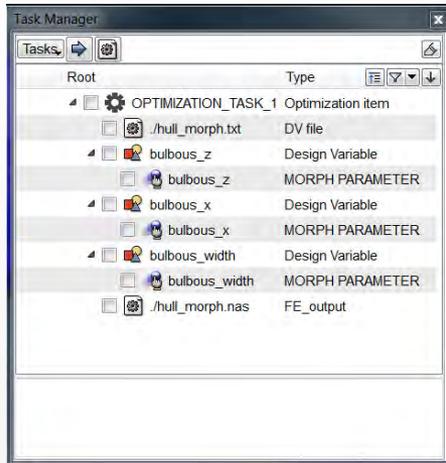
Shape optimization of the bulbous bow

- Morphing Tool controls model and fluid mesh simultaneously
- Shaping is performed without the need of re-meshing
- Design Variables are defined at the Optimization Task
- Morphing results are simulated



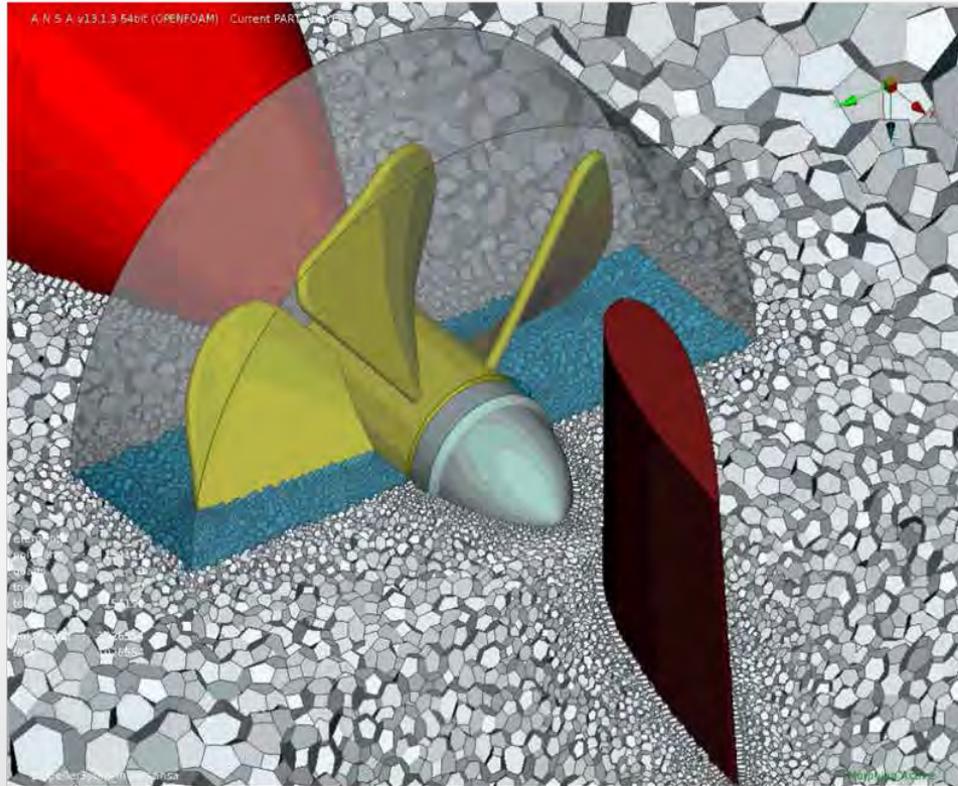
Shape optimization of the bulbous bow

Definition of Design of Experiments



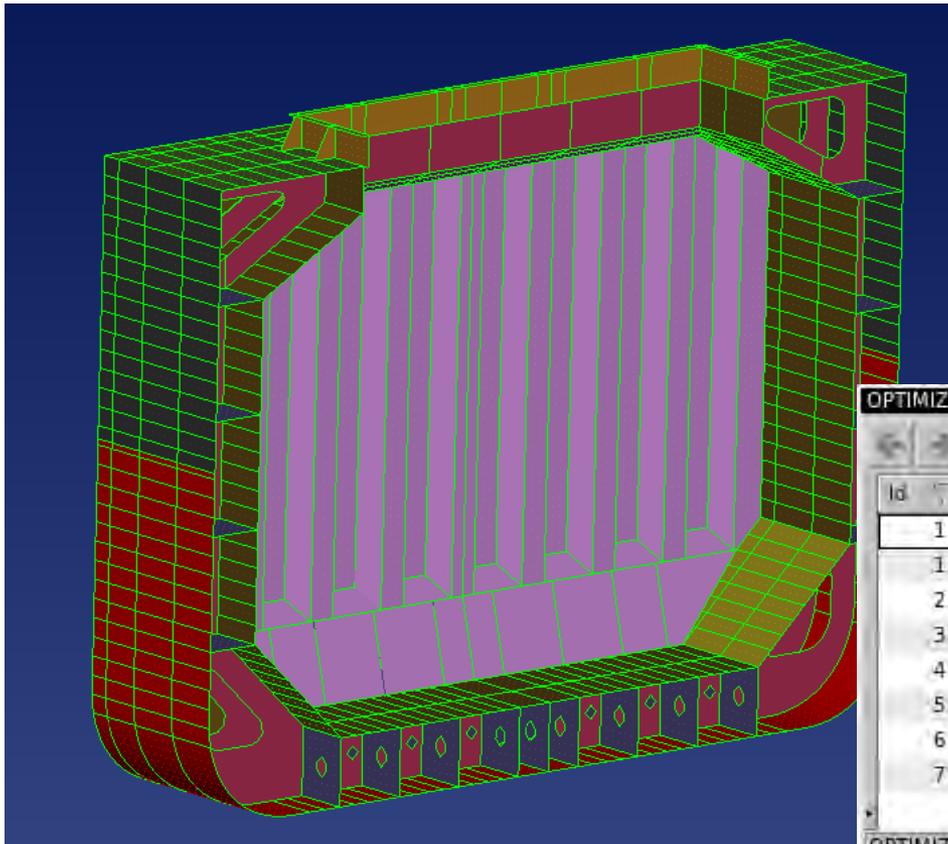
Shape optimization using the Morphing Tool

Volume mesh morphing



Shell thickness optimization

Automatic definition of design variables for shell thickness and creation of the Optimization Task



PROPERTY

Id	Name	T	MID1	MID	type
5	hull_above_water	0.018	1		PSHELL
6	hull_below_water	0.018	1		PSHELL
11	sea_level	1.	1		PSHELL
12	storage_hatches	0.016	1		PSHELL
15	Inner	0.018	1		PSHELL
18	Walls	0.0115	1		PSHELL
20	Horizontal_Plates	0.0125	1		PSHELL
21	Sections	0.018	1		PSHELL
25	L_0.1*0.3_old			3	PBEAM
28	Inner_Hopper_Side	0.0125	1		PSHELL
29	Inner_Wall_Endings	0.018	1		PSHELL
31	L_0.1*0.3				

Available fields for list: Entities
List fields for keyword: PSHELL

Field
<input type="checkbox"/> NUMBERING_RULE_NAME
<input type="checkbox"/> PID
<input type="checkbox"/> START_ID
<input checked="" type="checkbox"/> T
<input type="checkbox"/> TRANSPARENCY
<input type="checkbox"/> TS/T

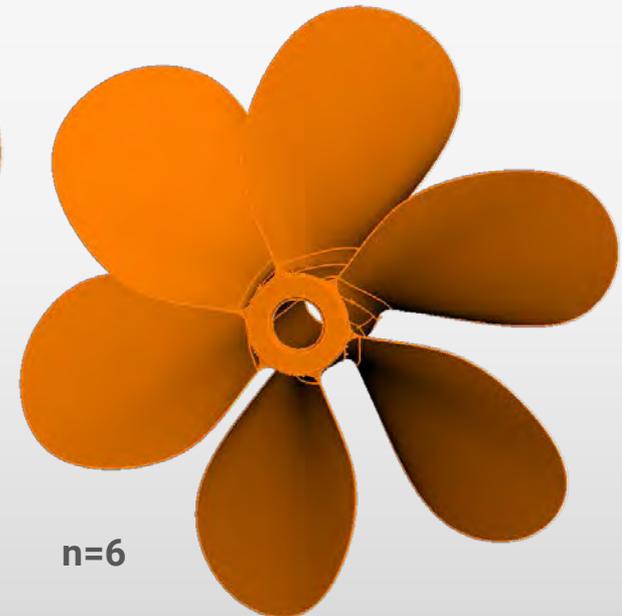
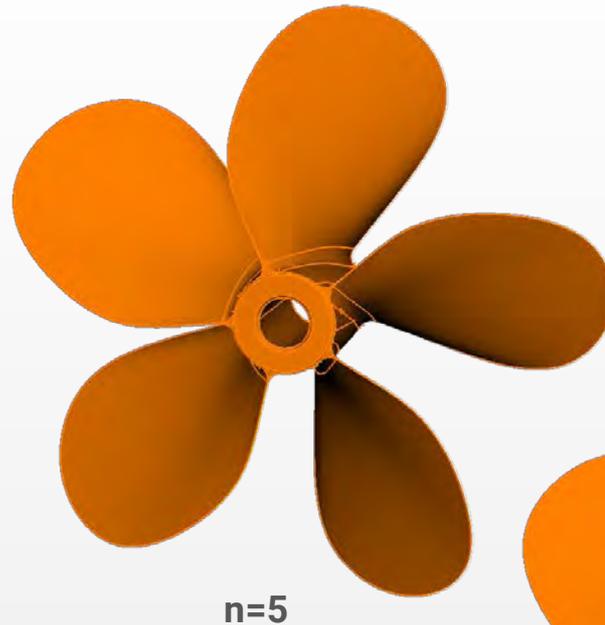
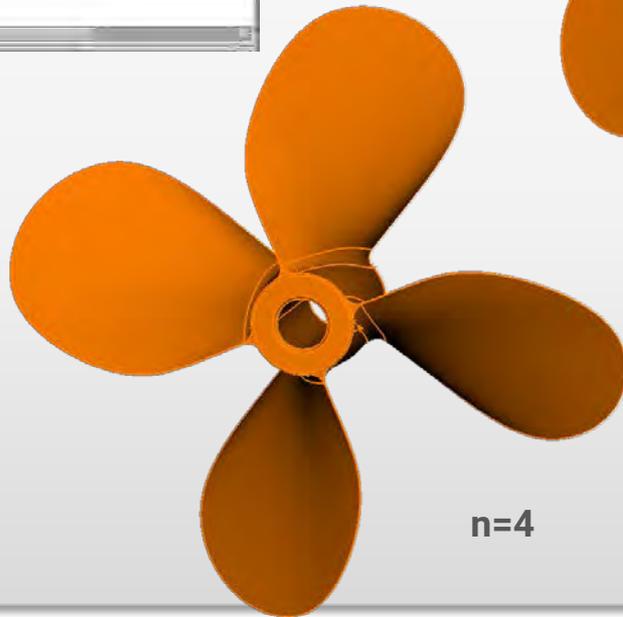
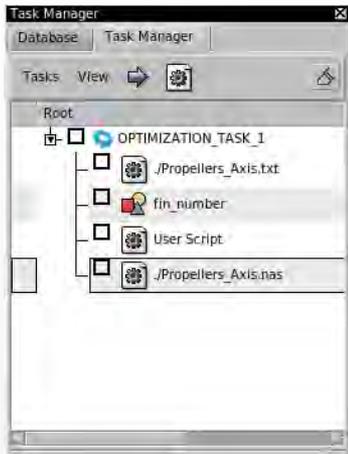
OPTIMIZATION TASK

Id	Name	Min Value	Current Value	Max Value
1	OPTIMIZATION_TASK_1			
1	PSHELL_20_T	0.01125	0.0125	0.01375
2	PSHELL_28_T	0.01125	0.0125	0.01375
3	PSHELL_21_T	0.016199...	0.018	0.0198
4	PSHELL_29_T	0.016199...	0.018	0.0198
5	PSHELL_18_T	0.01035	0.0115	0.01265
6	PSHELL_6_T	0.016199...	0.018	0.0198
7	PSHELL_15_T	0.016199...	0.018	0.0198

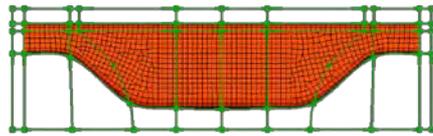
OPTIMIZATION TASK total 15 selected 0

Creating / importing features parametrically

Defining the fin number of a propeller as design variable

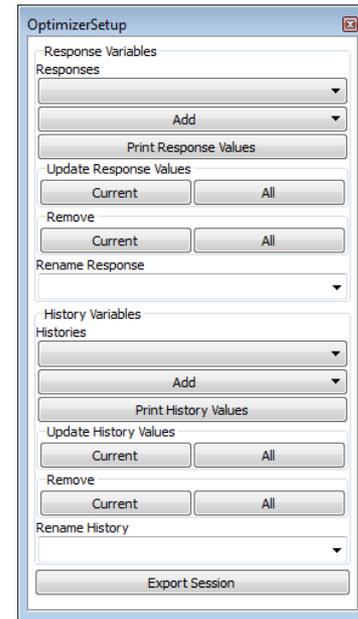
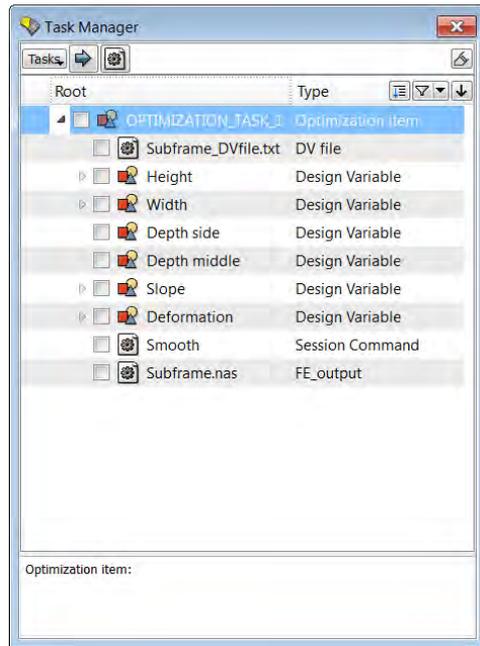


Coupling ANSA and META to parametric optimizers



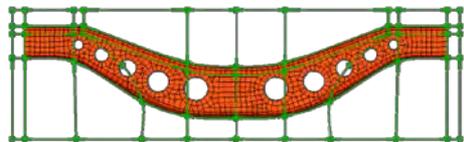
OPTIMIZER

ANSA Optimization Task



META OptimizerSetup

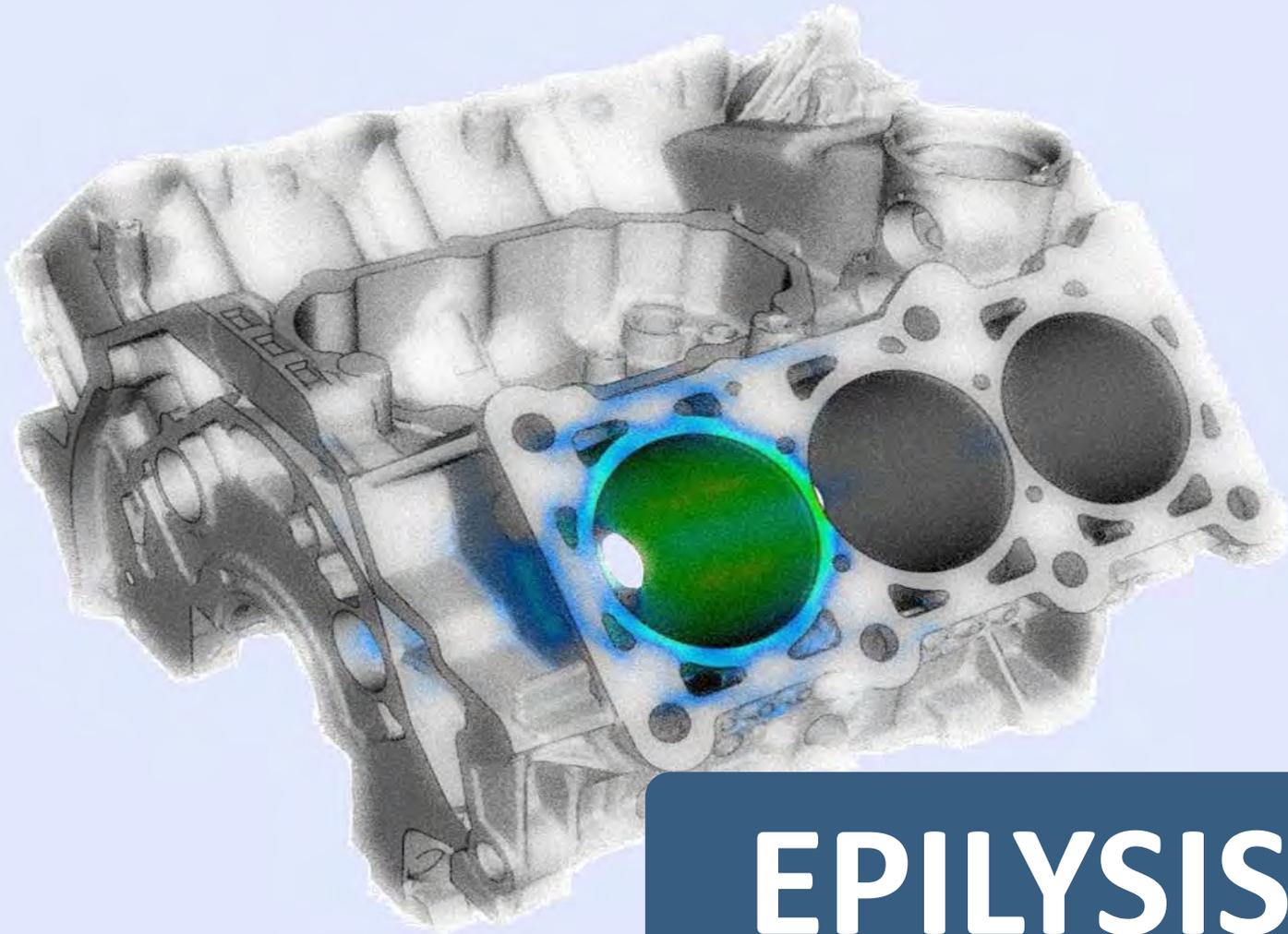
SOLVER



Coupling ANSA and META to parametric optimizers

- LS-OPT of LSTC
- modeFRONTIER of ESTECO
- OPTIMUS of NOESIS
- Isight of SIMULIA
- Heeds of Red Cedar

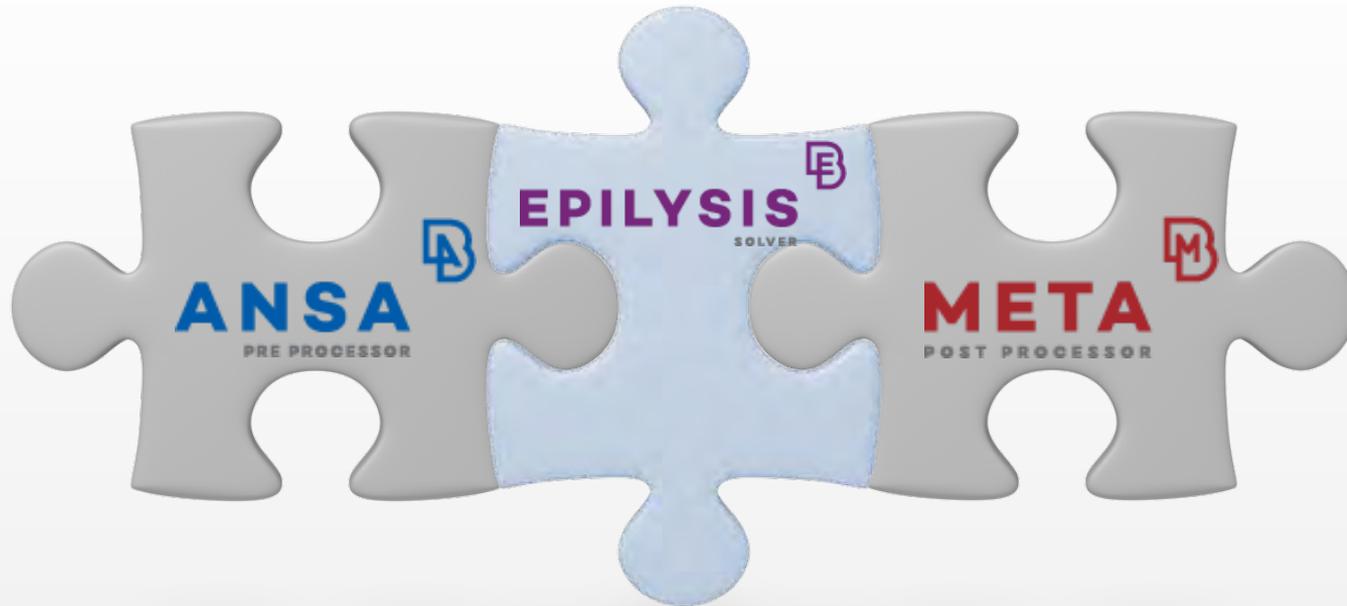




EPILYSIS

EPILYSIS

Solver for FE analysis



- General-purpose FE Analysis Software
- One Solver for many engineering problems
- Standalone & integrated in ANSA

Επίλυσις

[epi-li-sis]

noun 1. solution

EPILYSIS

EPILYSIS Characteristics

Programming Language

- C++
- Double Precision

Quality

- Benchmarks accurate (NAFEMS, SFM and other – More than 3500 tests)
- Robustness
- Repeatability

Performance

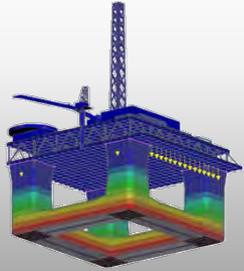
- Shared Memory Parallel Processing (SMP)
- Automatic utilization of system resources without time consuming tuning
- Multiple high-performance direct sparse linear solvers

Easy to Use

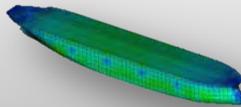
EPILYSIS

Linear analyses

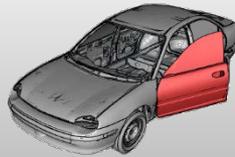
Static



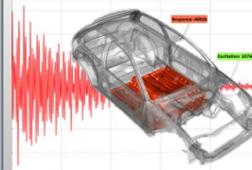
Real Eigenvalue



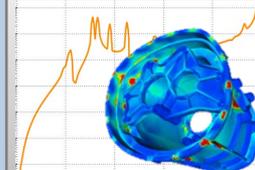
Direct Transient Response



Direct Frequency Response



Modal Transient Response

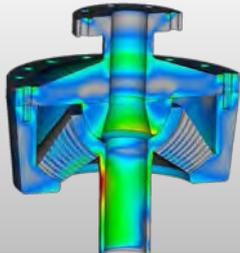


Modal Frequency Response



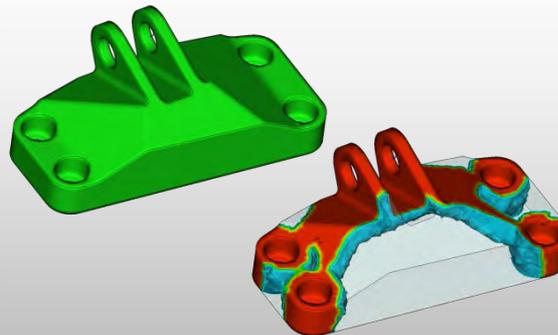
Non - linear analyses

Static: linear elements / materials -non linear contacts



Optimization

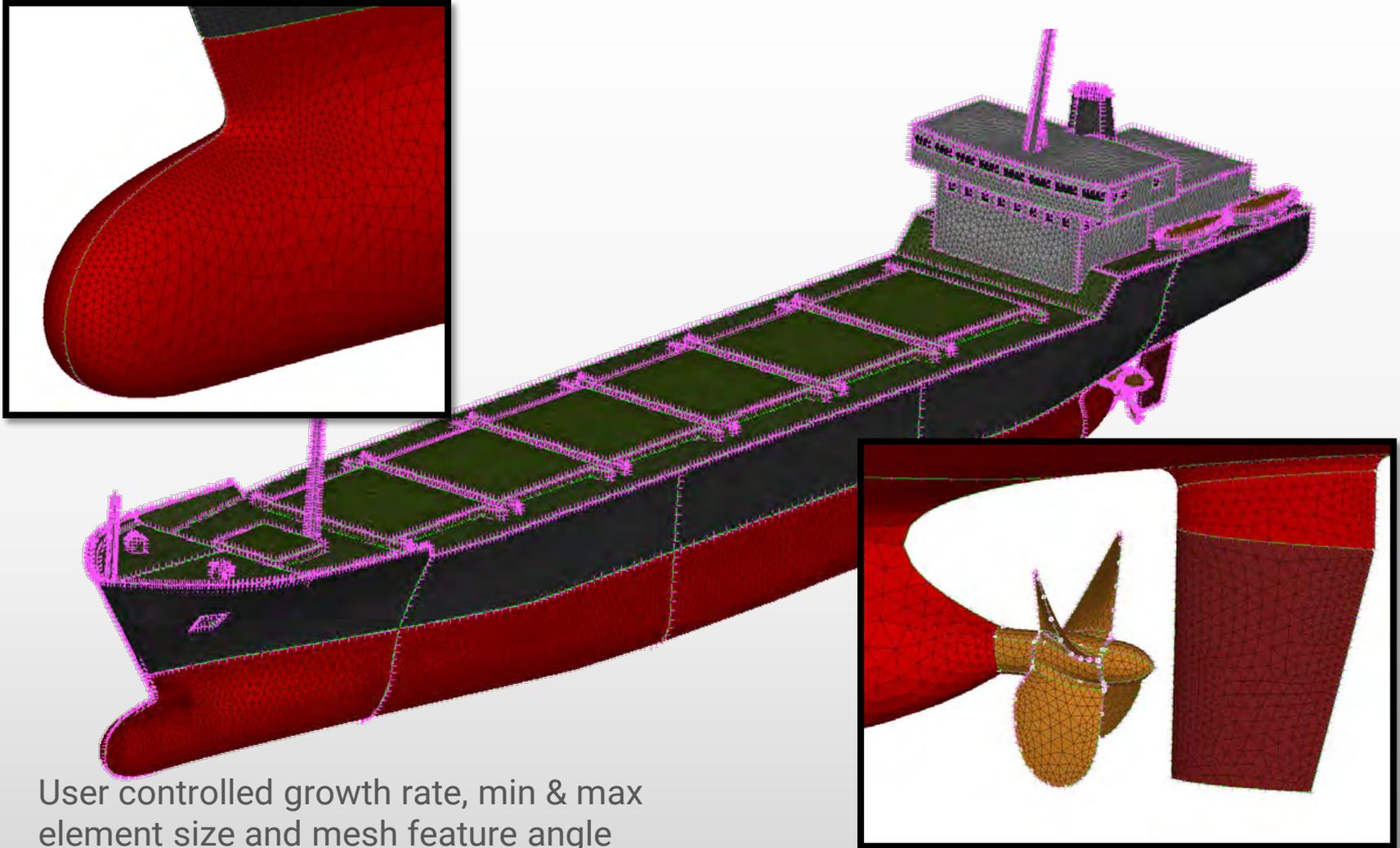
Topology optimization





Model definition for CFD analysis

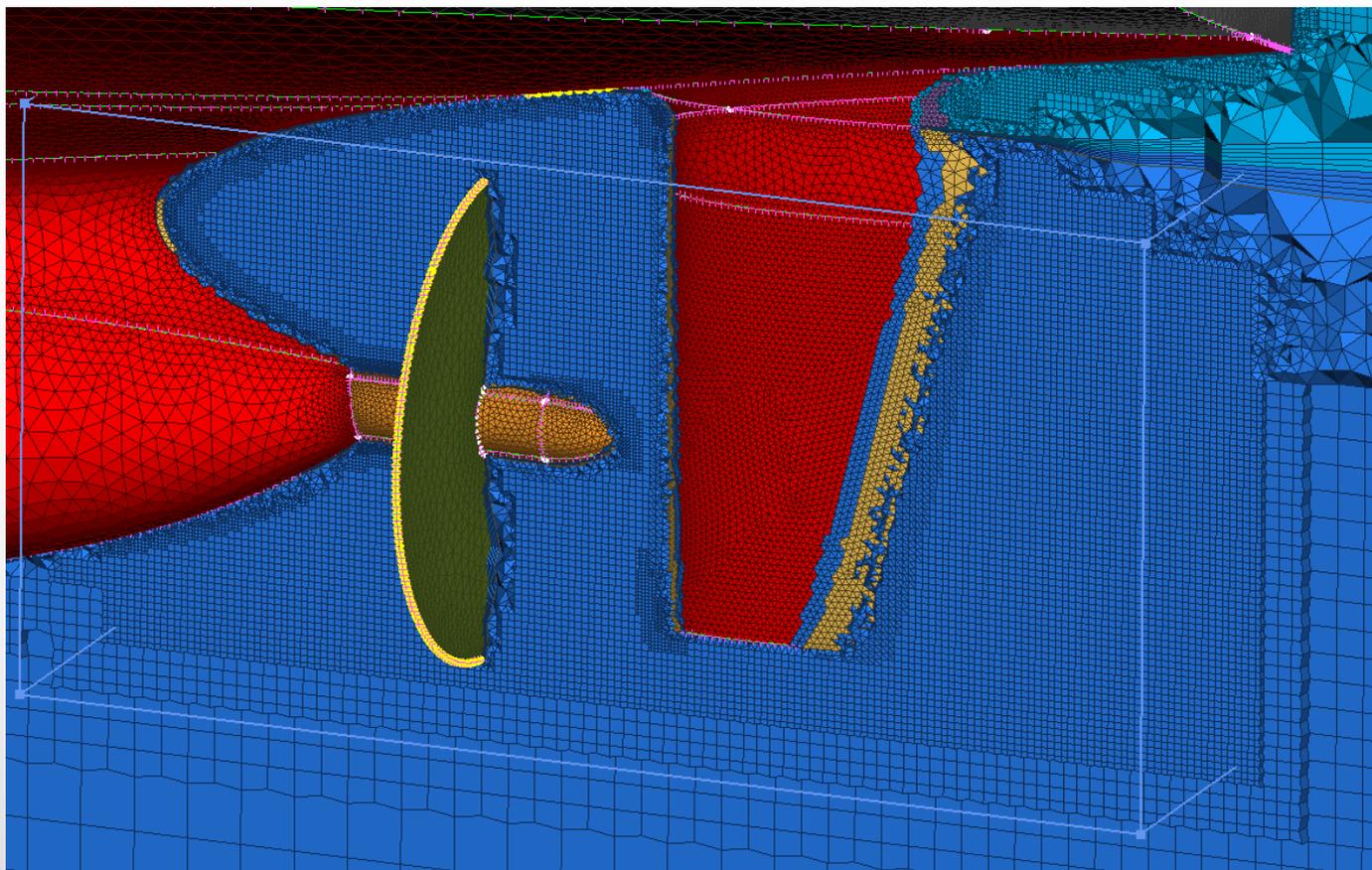
Fully automatic Curvature Dependant surface meshing



User controlled growth rate, min & max element size and mesh feature angle

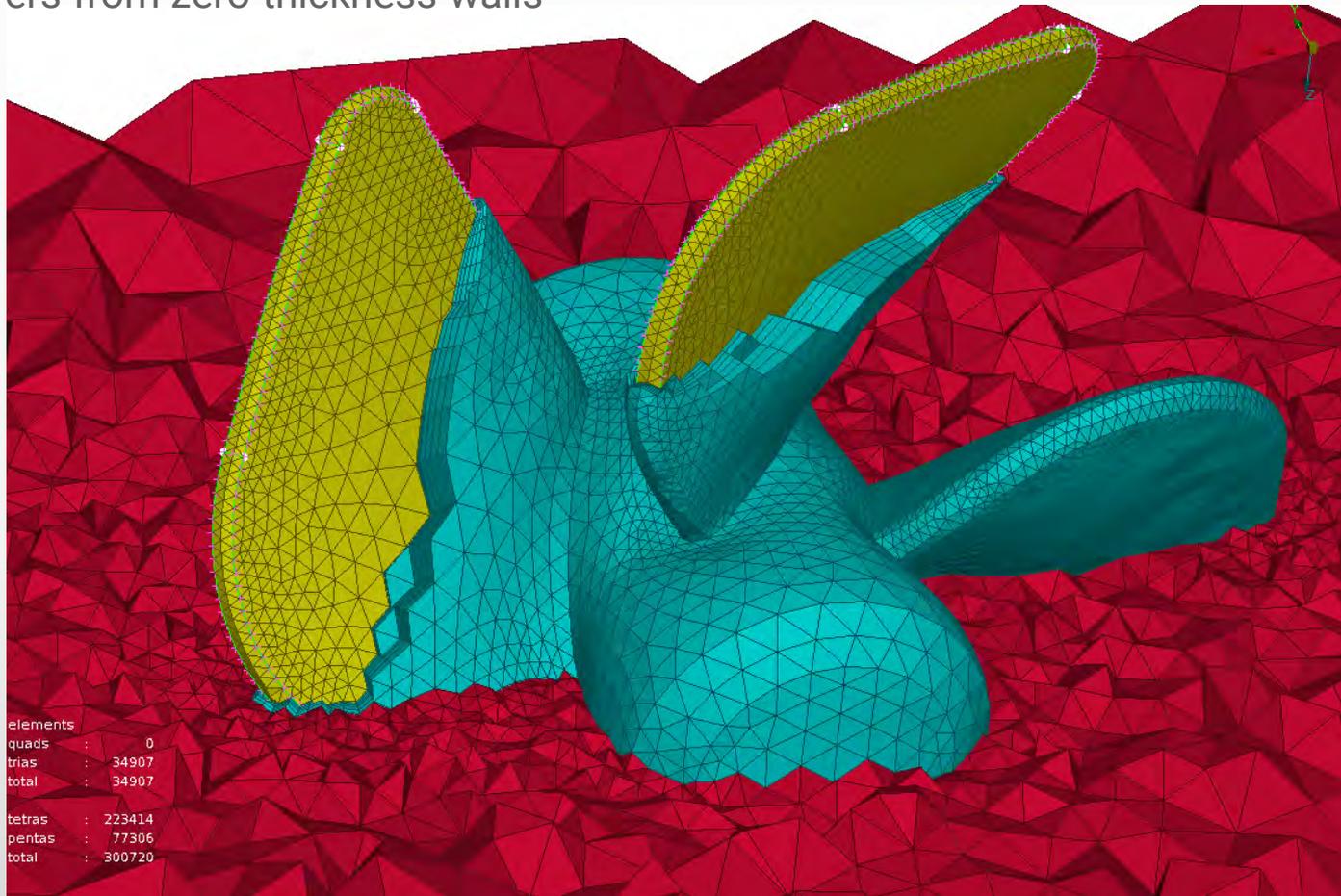
Model definition for CFD analysis

Local refinement using SIZE BOXES



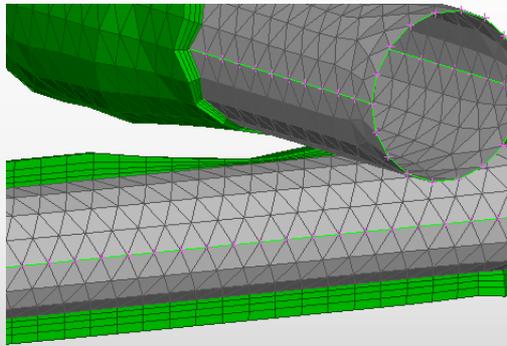
Boundary layers generation

- Auto exclude or collapse areas
- Controlled Layer Squeezing to avoid intersections
- Layers from selected areas with different settings
- Layers from zero-thickness walls

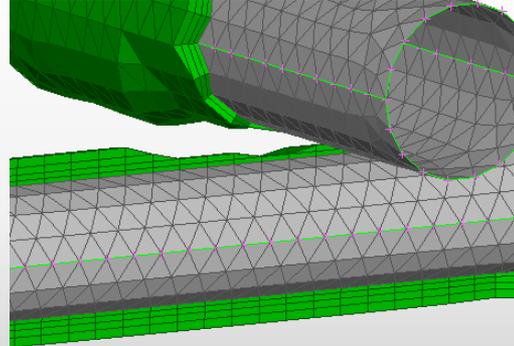


Advanced boundary layers generation

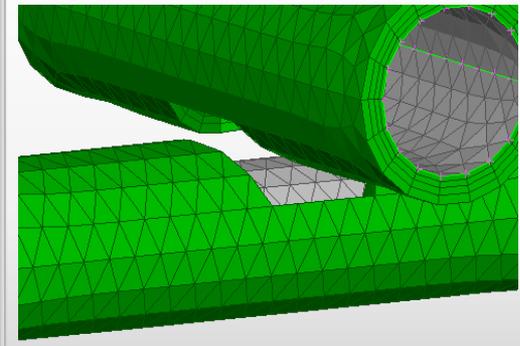
Squeeze



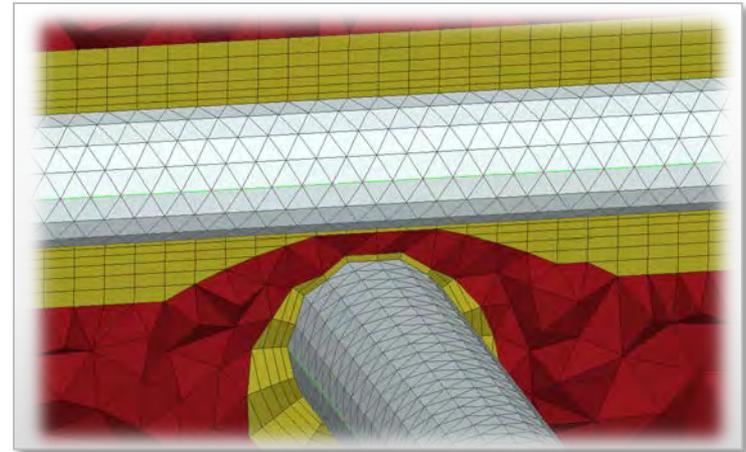
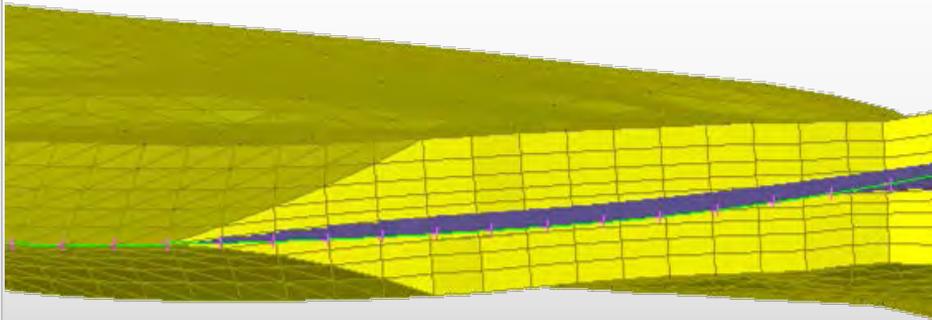
Collapse



Extrude



Collapse free edges



Boundary Condition type specification for various CFD solvers

Current Part: Untitled

Shell
quads : 482, 83.54%
trias : 95, 16.46%
total : 577

FACE ZONE [SHELL PROPERTY]

Name: propeller_surface

FROZEN_ID: NO, FROZEN_DELETE: NO, DEFINED: YES, TRIM: YES, USE_IN_MODEL: YES

ZONE_ID	ZONE_TYPE
4	wall
propeller_surface	axis

OK

PROPERTIES

Id	Name	T	MID1	MID	_type_
1	outer_geometry	1.	16		PSHELL
2	top_cap_Fluid_layers	1.	15		PSHELL
3	Fluid_layers			14	PSOLID
4	propeller_surface	1.	4		PSHELL
5	Auto Detected Volume 5			17	PSOLID

total: 5, selected: 0

SOLVER INFO

script programs end
Creating index for tooltips from file /home/titanas/ANSA_2001-exe/ansa_v13.2.0/config/ANSA_CARDS
Done.
Reading file: house.tdf
Reading file: cellphone.tdf
Reading file: demoship.tdf

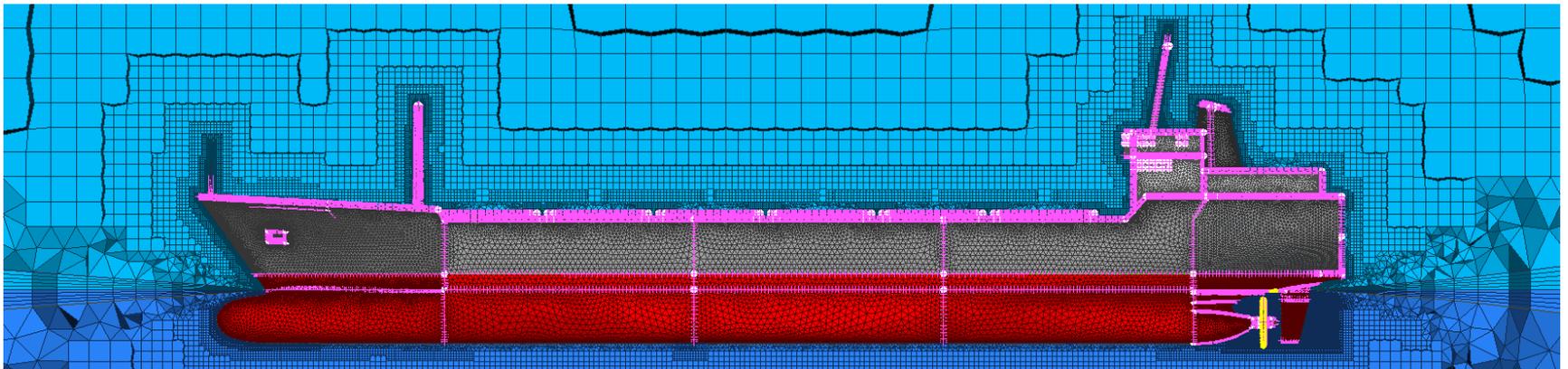
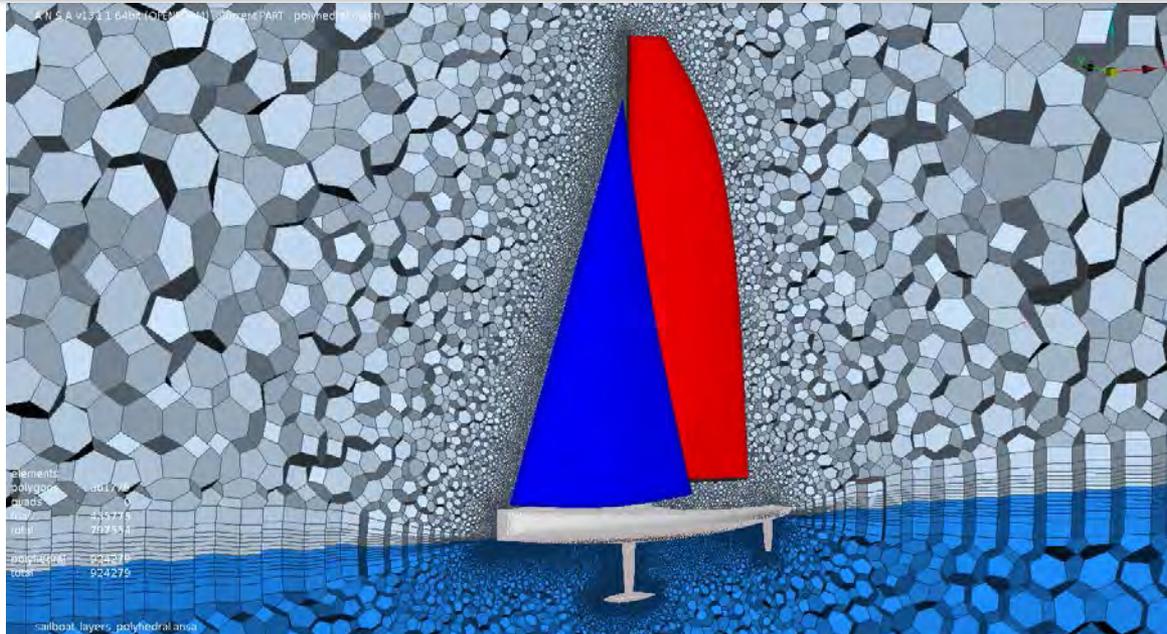
General Butt...

FOCUS OR AND !NOT
ENT NOT INVERT PEEL
C.PLANE BEST ALL NEIGHB NEAR
LOCK ISOLATE EXTREME UCHECKE UNMESH

GEOME.. D.UTIL VISIB GEOM. MACROS PERIMS HOT PNT MORPH SIZEBOX
TRANSF CHECK PID VOLUM FE-Mod SINGLE CURVES CROSH HEXAB...
FINE D.INFO WIRE BOUNDS SPOTS DOUBLE POINTS M.Pnt WRK.PLN
DELETE SHADD. HIDDEN GRIDS TRIPLE CENTERs MEAS.

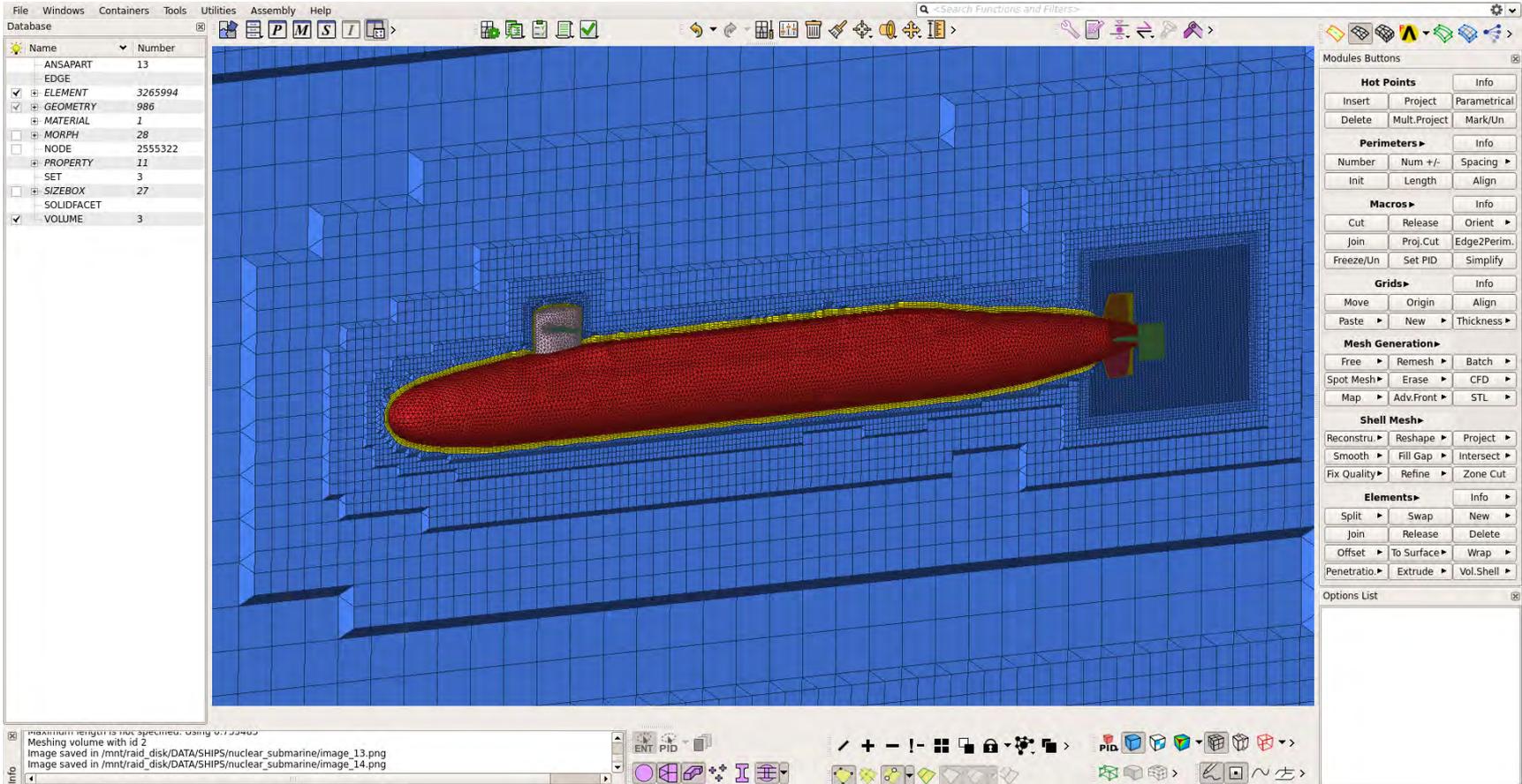
Model definition for CFD analysis

Fast and robust volume meshing for all types of elements



Model definition for CFD analysis

Fast and robust volume meshing for all types of elements



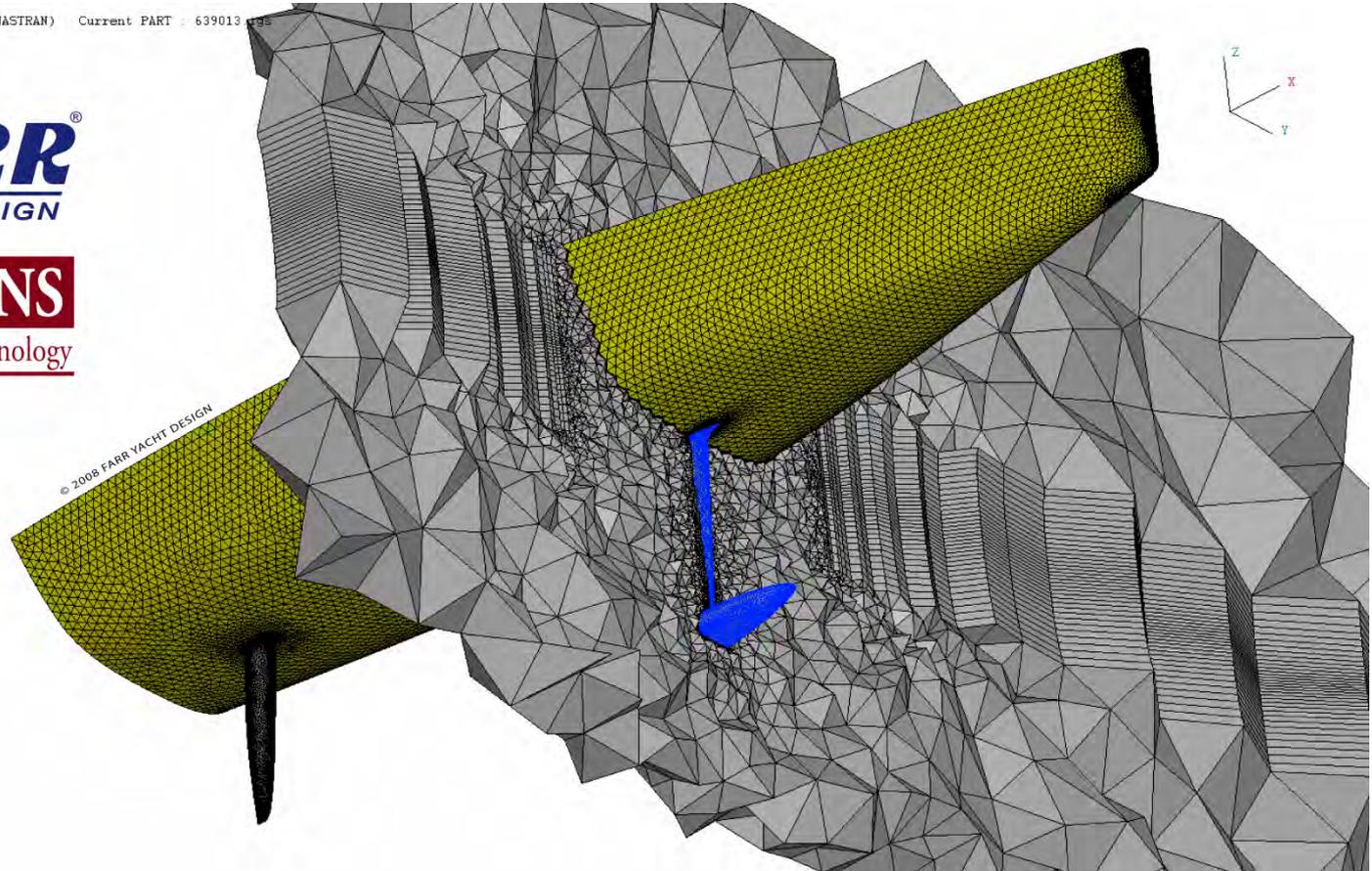
Model definition for CFD analysis

Fast and robust volume meshing for all types of elements

A N S A v12.1.3 64bit (NASTRAN) Current PART : 639013

FARR[®]
YACHT DESIGN

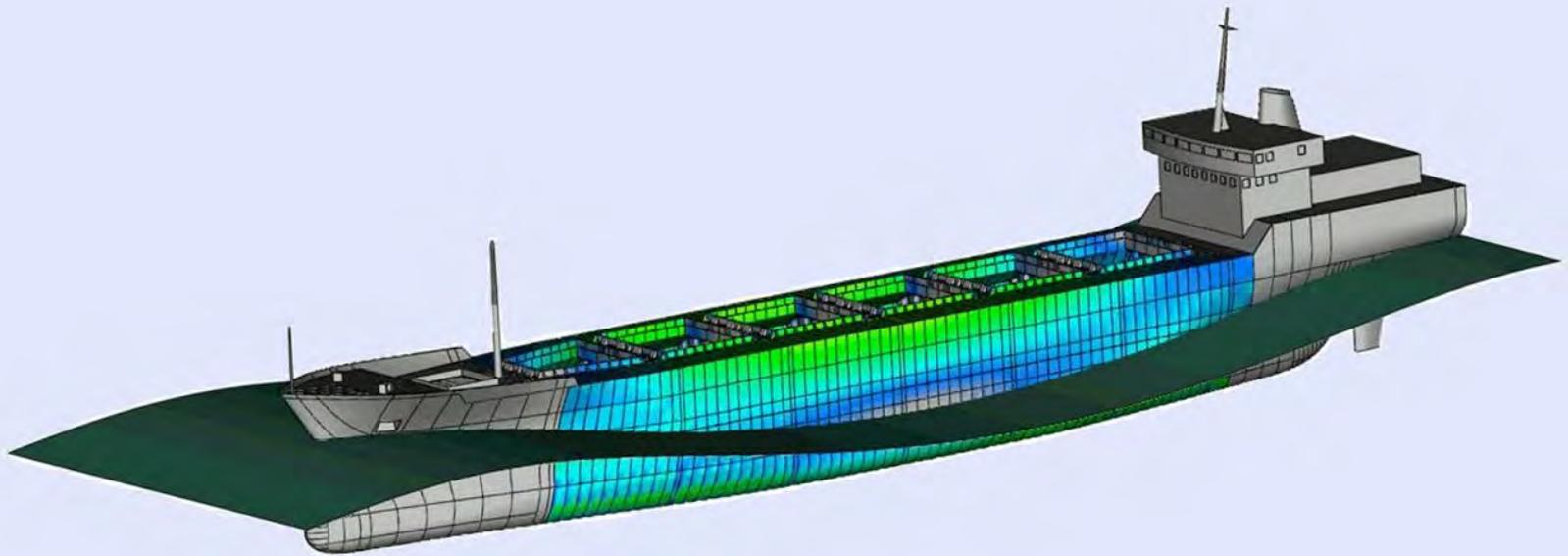
STEVENS
Institute of Technology



```
elements
quads : 0
trias : 62456
total : 62456

tetras : 235424
pentas : 253385
pyramid : 1107
total : 489916
```

639013_geom.21.ansa

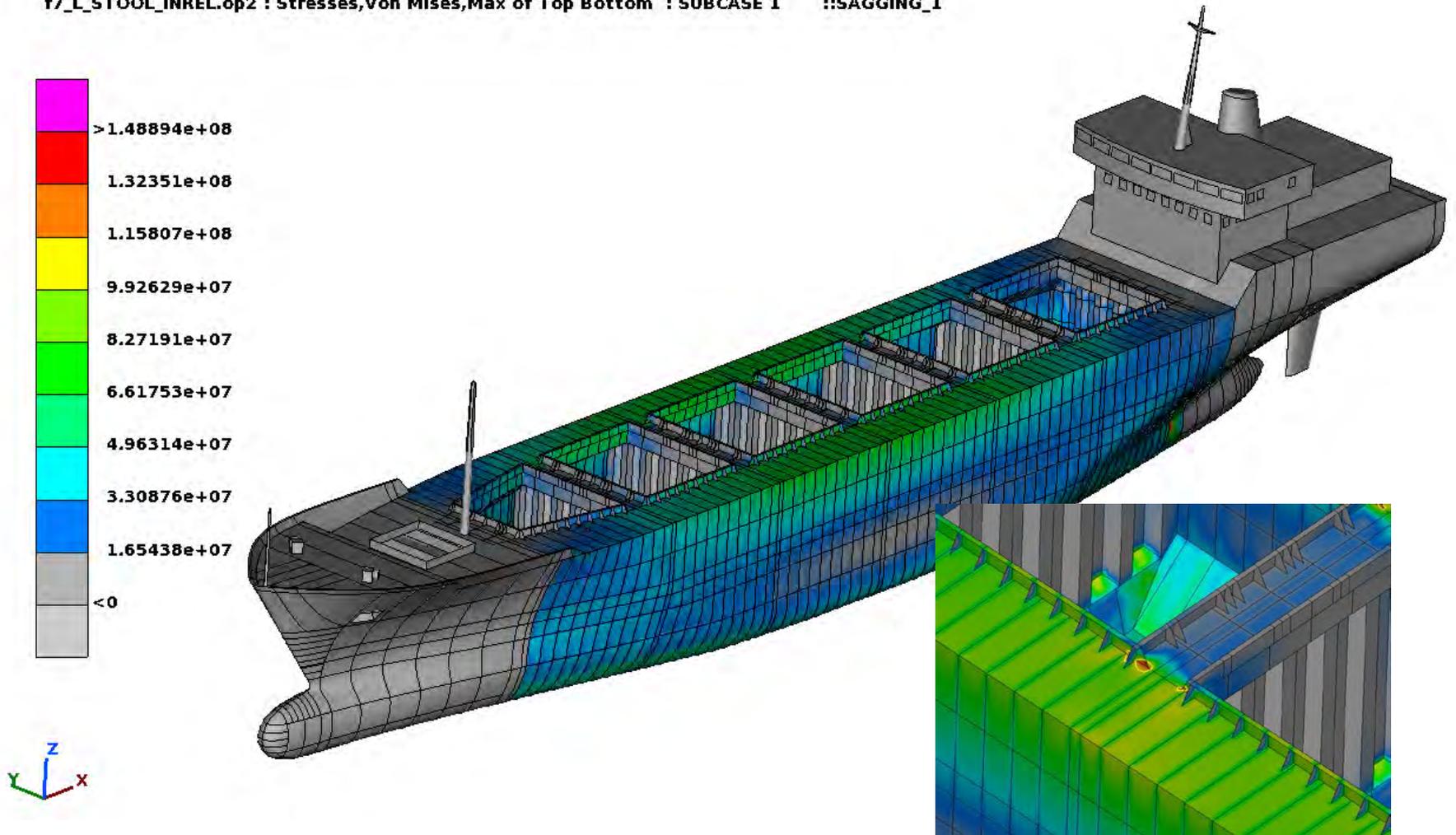


Post Processing

Viewing results in META

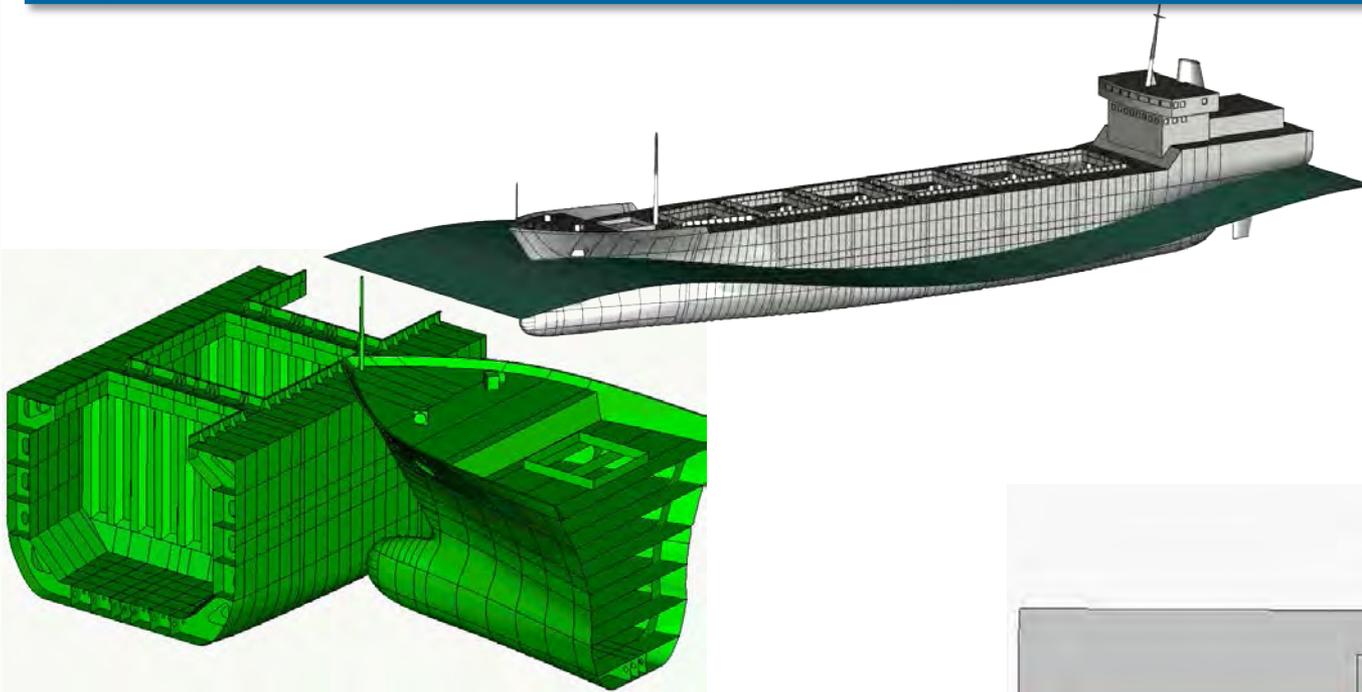
Global and local stress inspection

f7_L_STOOL_INREL.op2 : Stresses,Von Mises,Max of Top Bottom : SUBCASE 1 ::SAGGING_1



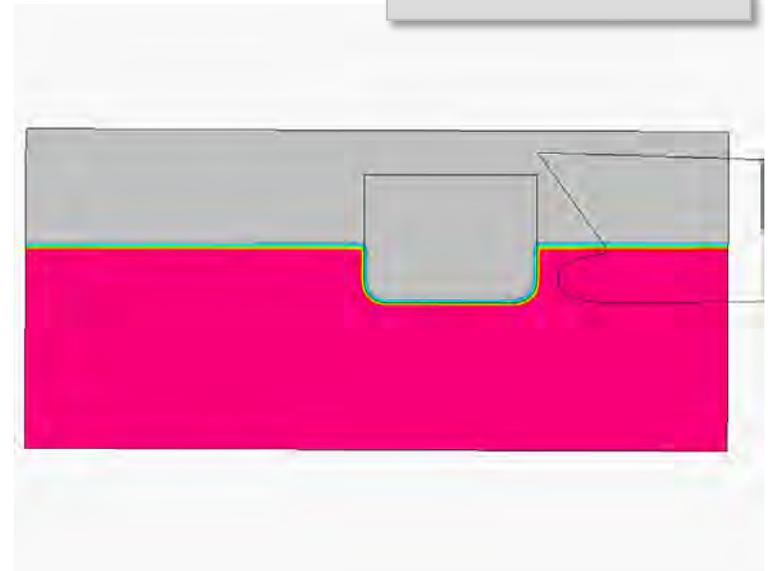
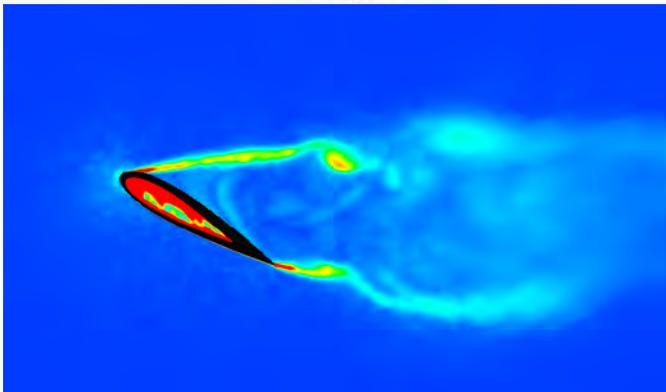
Viewing results in META

Supports results from various solvers and analyses



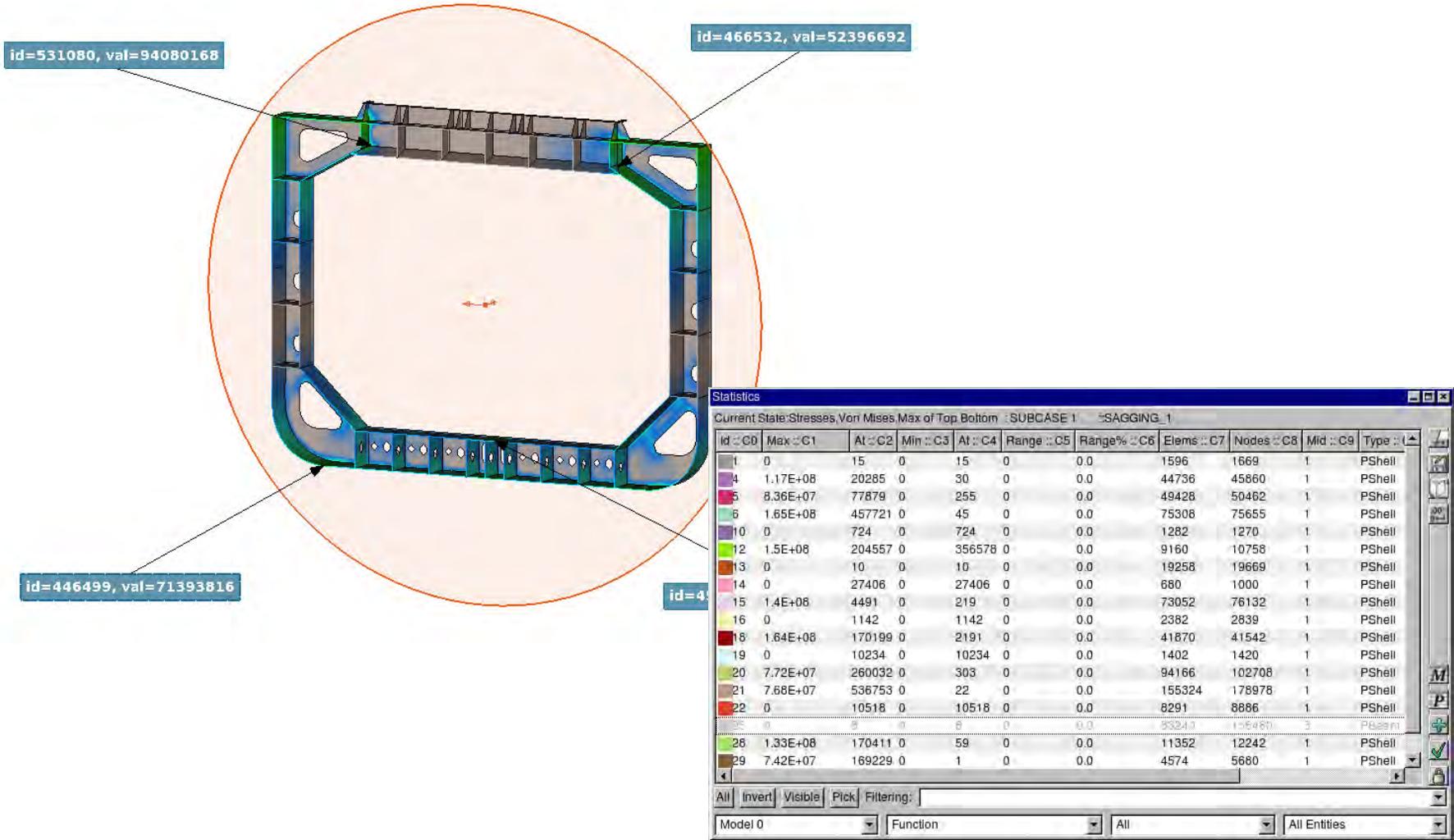
Results

- Static
- Dynamic
- CFD
- ALE



META Reporting capabilities

Creating sections, annotations, statistics, reports...



META Reporting capabilities

Report Composer

Report
HTML
PPTX

Slide
Insert

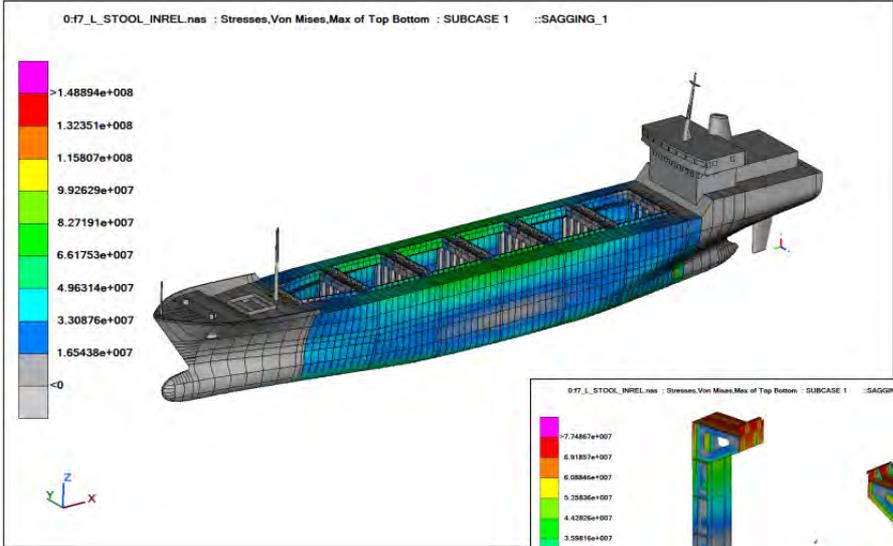
Zoom
90%

Align
Distribute

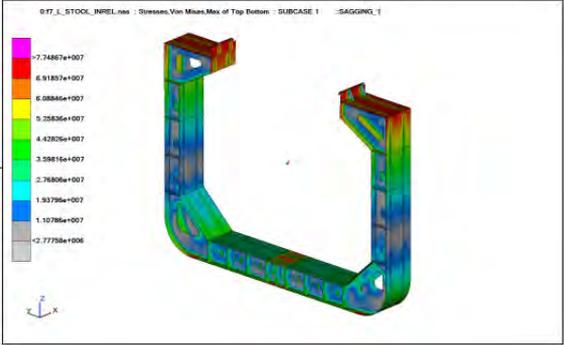
Report
Slide 1
Title
Image 1
Textbox 1
Image 2

Sagging Case Study

0:17_L_STOOL_INREL.nas : Stresses,Von Mises,Max of Top Bottom : SUBCASE 1 ::SAGGING_1



>1.48894e+008
1.32351e+008
1.15807e+008
9.92629e+007
8.27191e+007
6.61753e+007
4.96314e+007
3.30876e+007
1.65438e+007
0



7.74887e+007
6.91837e+007
6.08846e+007
5.25836e+007
4.42826e+007
3.59816e+007
2.76806e+007
1.93796e+007
1.10786e+007
-2.77798e+006

- Wave Induced Sagging Loading Condition
- Solution lasted 1 hour and 30 minutes
- Maximum stresses at hatch openings
- Ships scantlings can be considered adequate

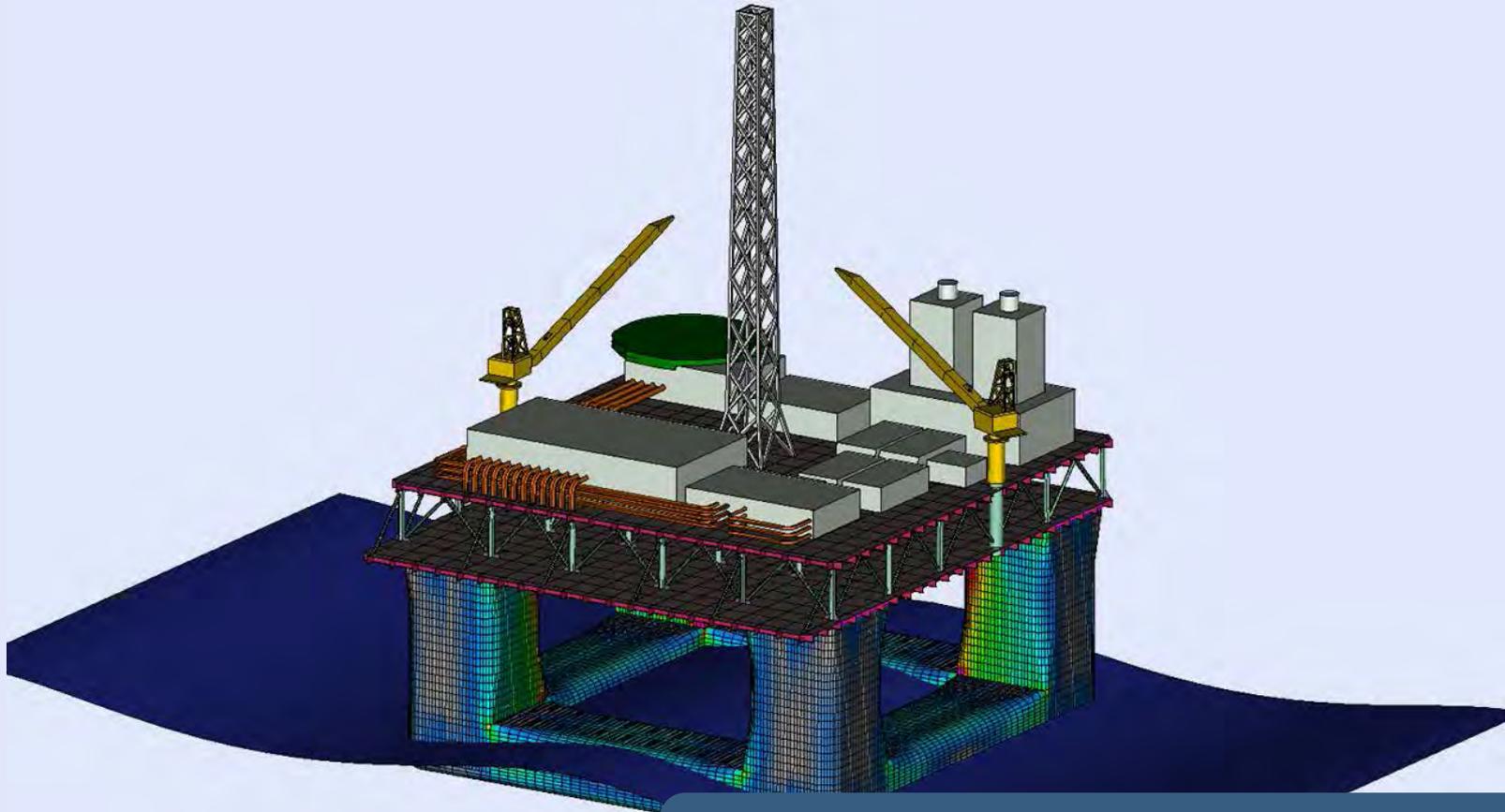
Create Slide Master

Save Cancel

- HTML / PPTX Reports

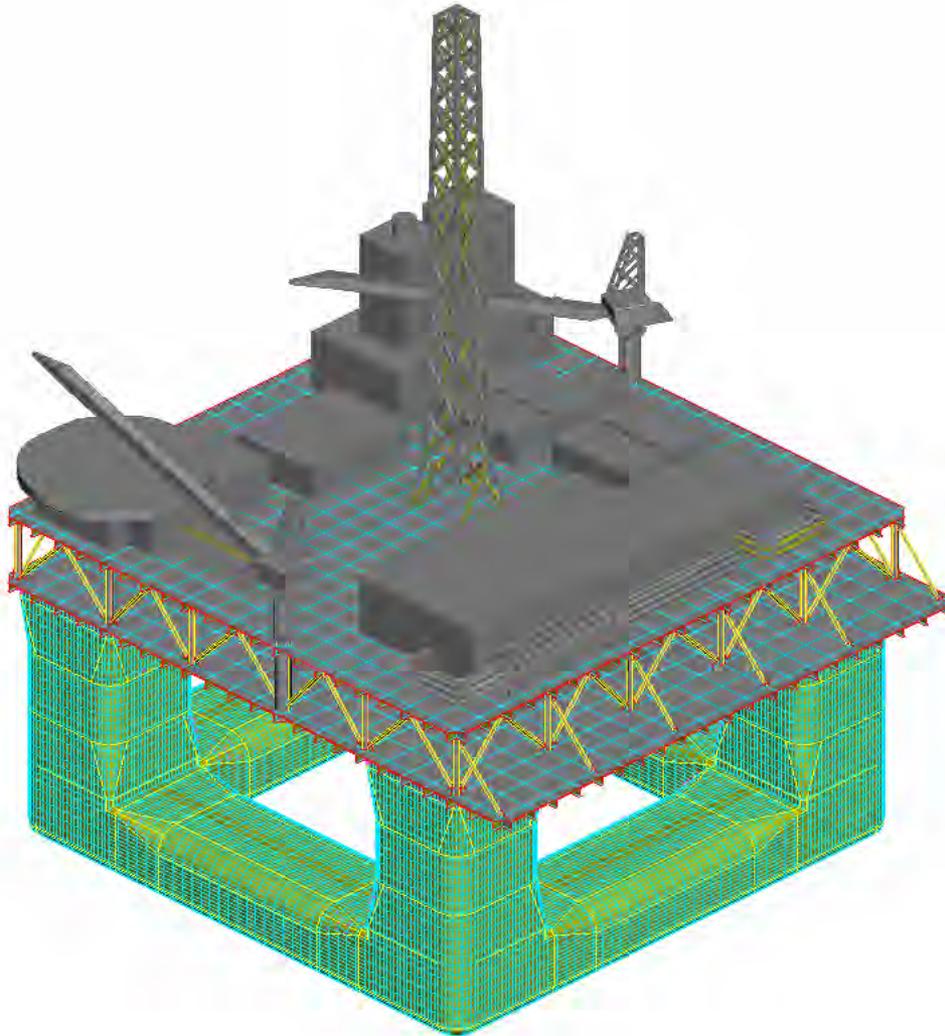
- Drag and Drop functionality

- Standard Model Reports

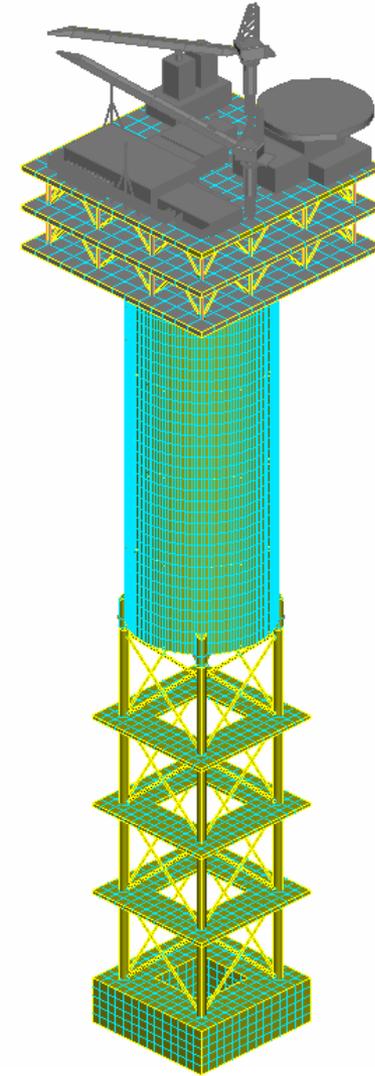


OFFSHORE

Offshore models

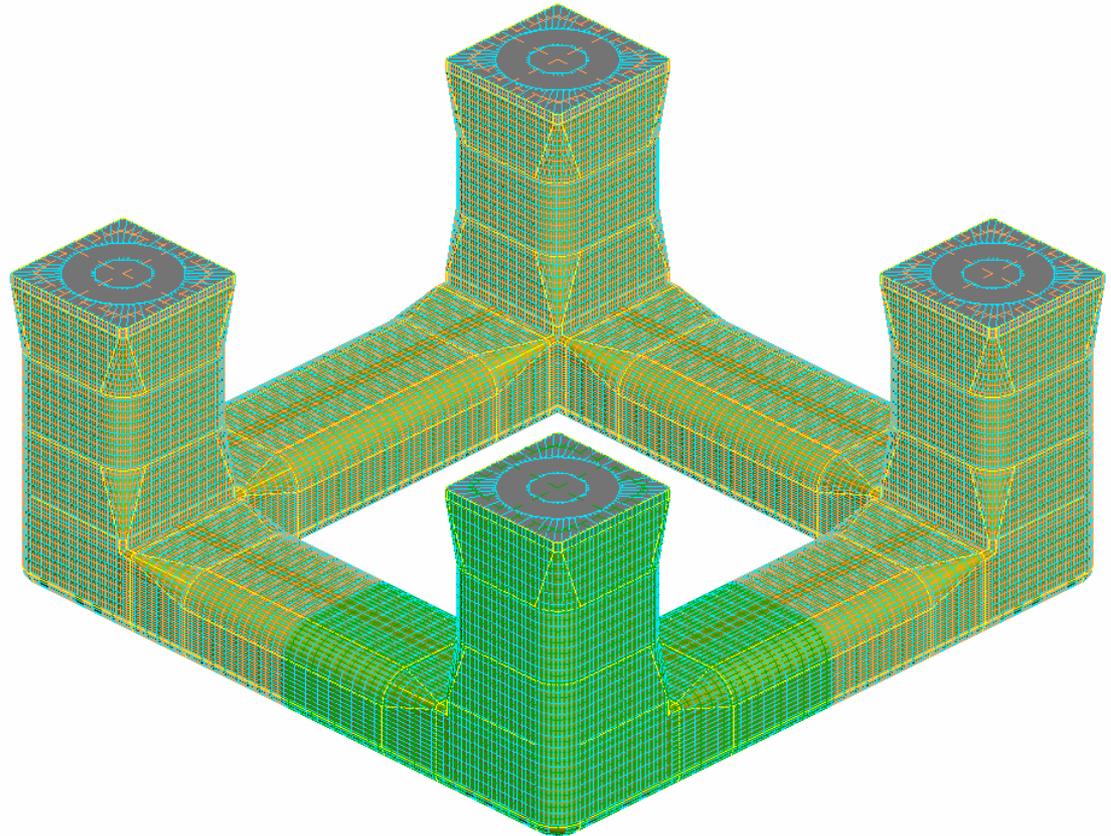
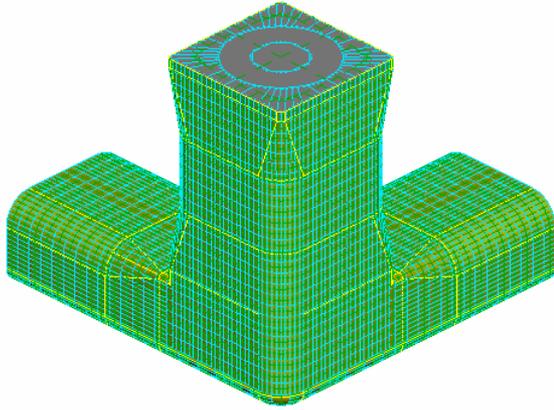


Semi-Submersible offshore platform



SPAR offshore platform

Geometry Handling

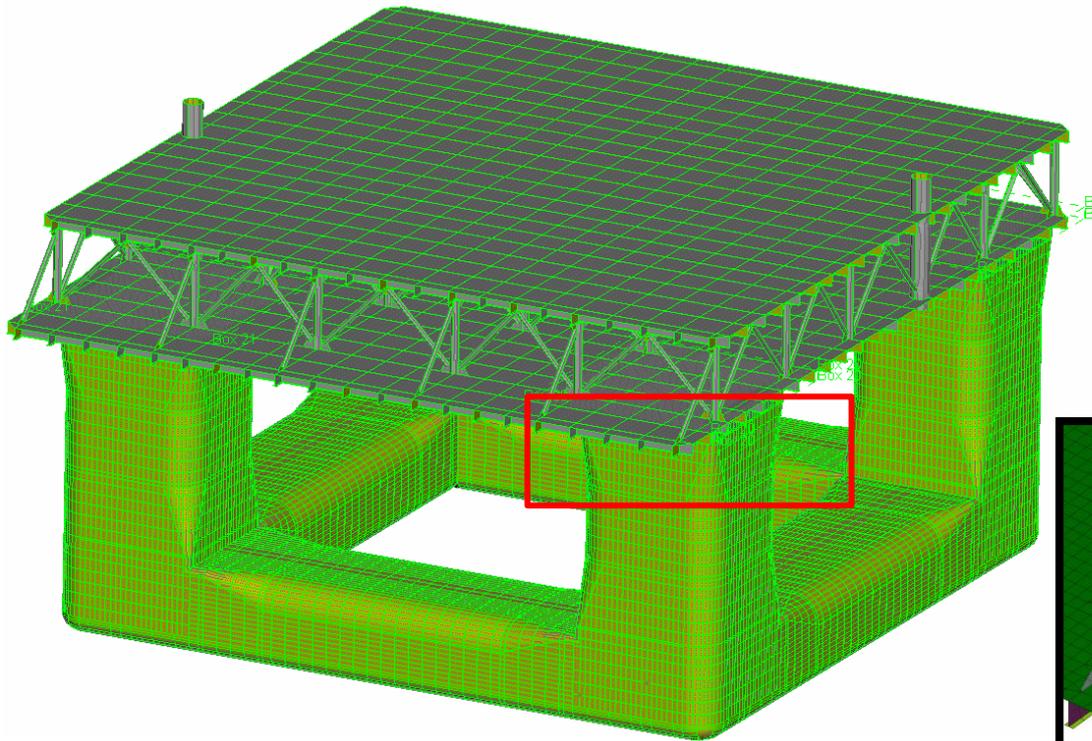


➤ **75% TIME & EFFORT
REDUCTION ON :**

- Geometry Creation
- Geometry Clean Up
- Meshing Generation
- Meshing Refinement

- Automatic creation of symmetrical entities
- Linked with the original entities by symmetry, translation, rotation & transformation
- Actions applied to the parent entities are applied automatically to the linked ones

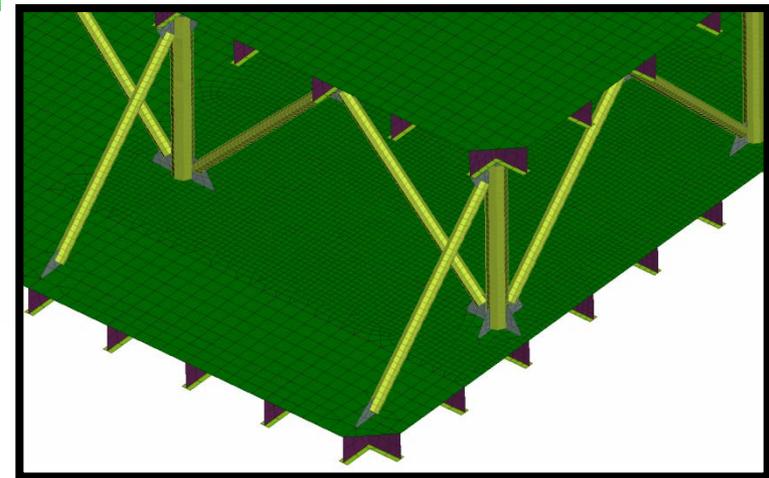
Batch Meshing Tool



Batch Mesh Manager

New Read Scenario Autoload Run

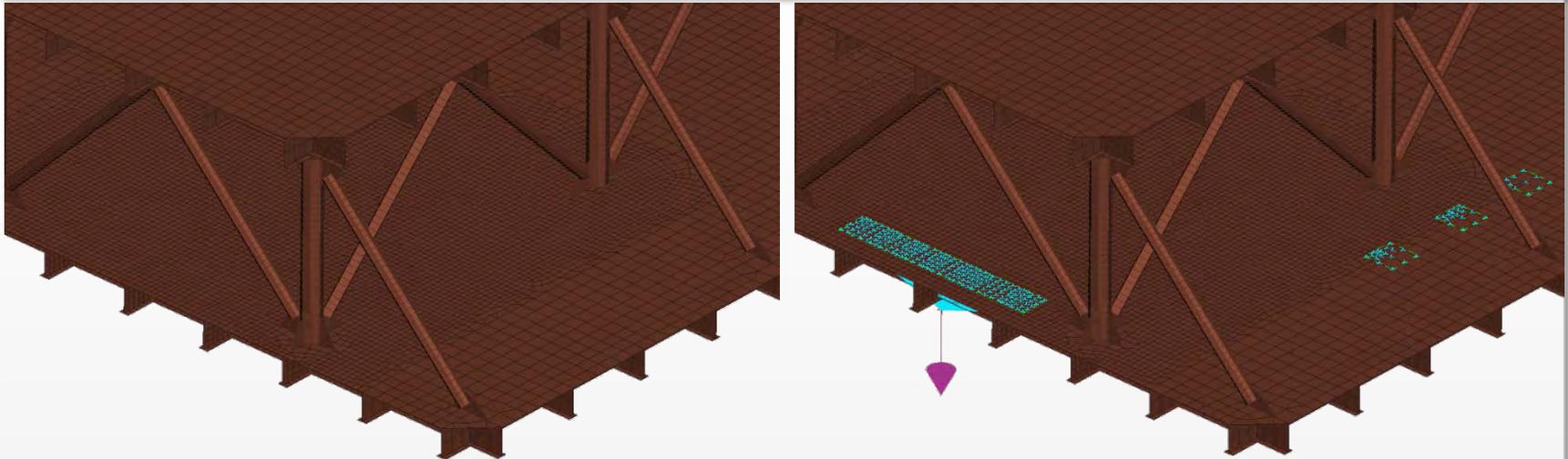
Name	Color	Contents	Mesh Parameters	Quality Criteria	Status
<input checked="" type="checkbox"/> Meshing_Scenario_1		28			Error
<input checked="" type="checkbox"/> Session_2	Orange	7	500mm	500mm	Error
<input checked="" type="checkbox"/> Default_Session	Green	21	620mm	620mm	Error
<input checked="" type="checkbox"/> Box_1	Yellow	0	250mm		



- Automated de-featuring
- User defined mesh parameters and quality criteria
- Automated part assignment in different scenarios
- Special treatment for specific areas of the model defined with boxes

Generic Entity Builders GEBs

Riser and Mooring Forces

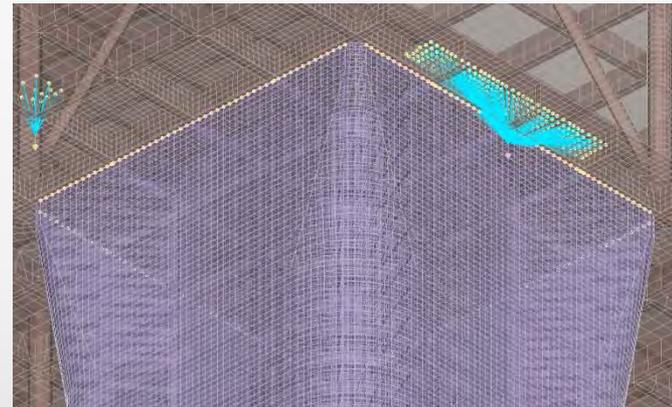
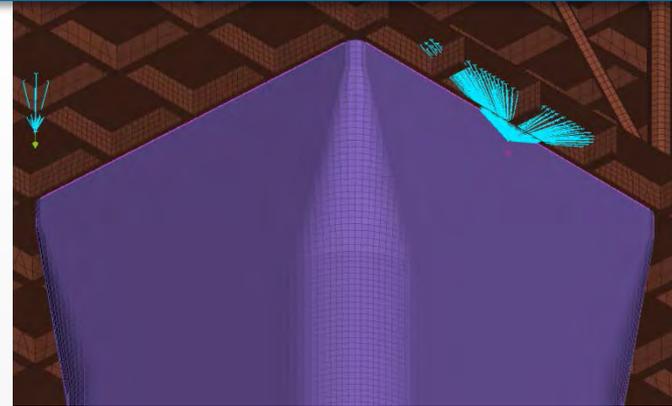


➤ **SEMI-AUTOMATED
BOUNDARY
CONDITIONS
DEFINITION**

- Performing actions determined by predefined rules
- Automated application on different mesh representations
- Use of library items

Connections Handling

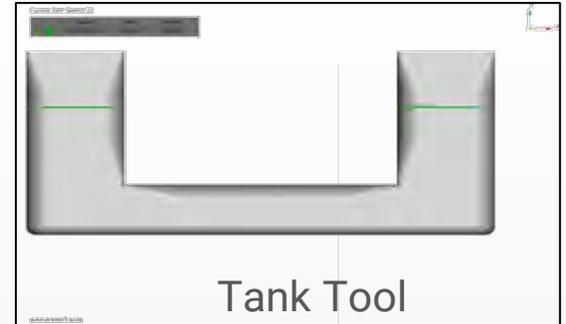
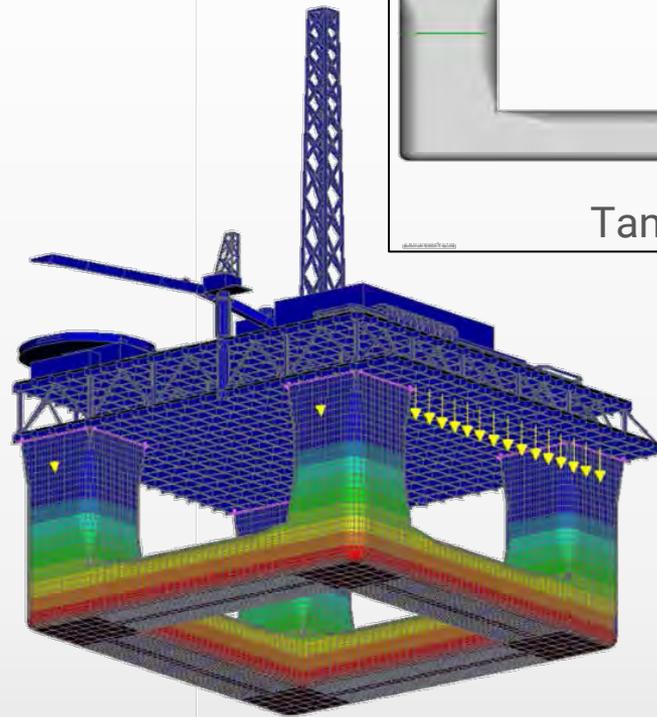
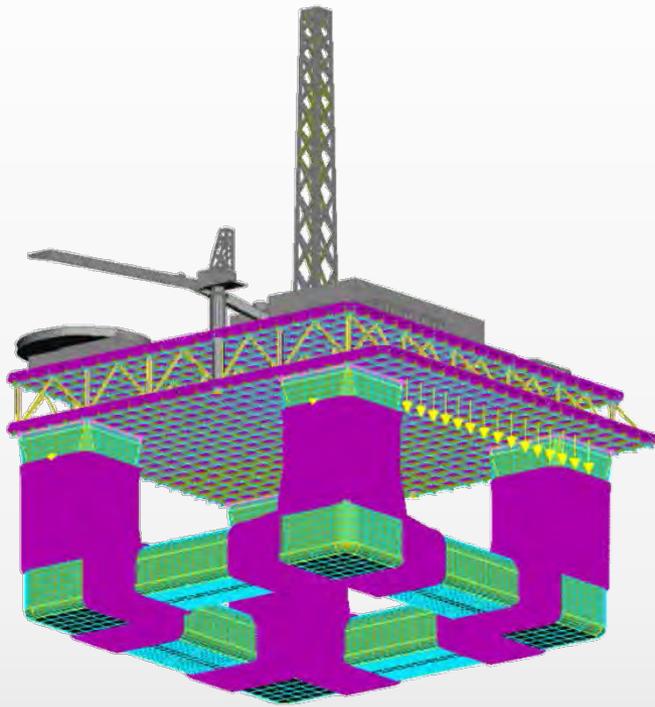
Hull and Deck connection



- Connection lines definition for application of various FE representations
- User specified connection parameters
- Massive connection generation

State of Equilibrium

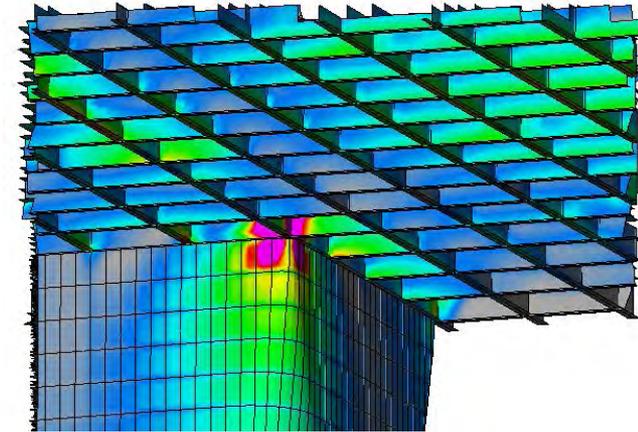
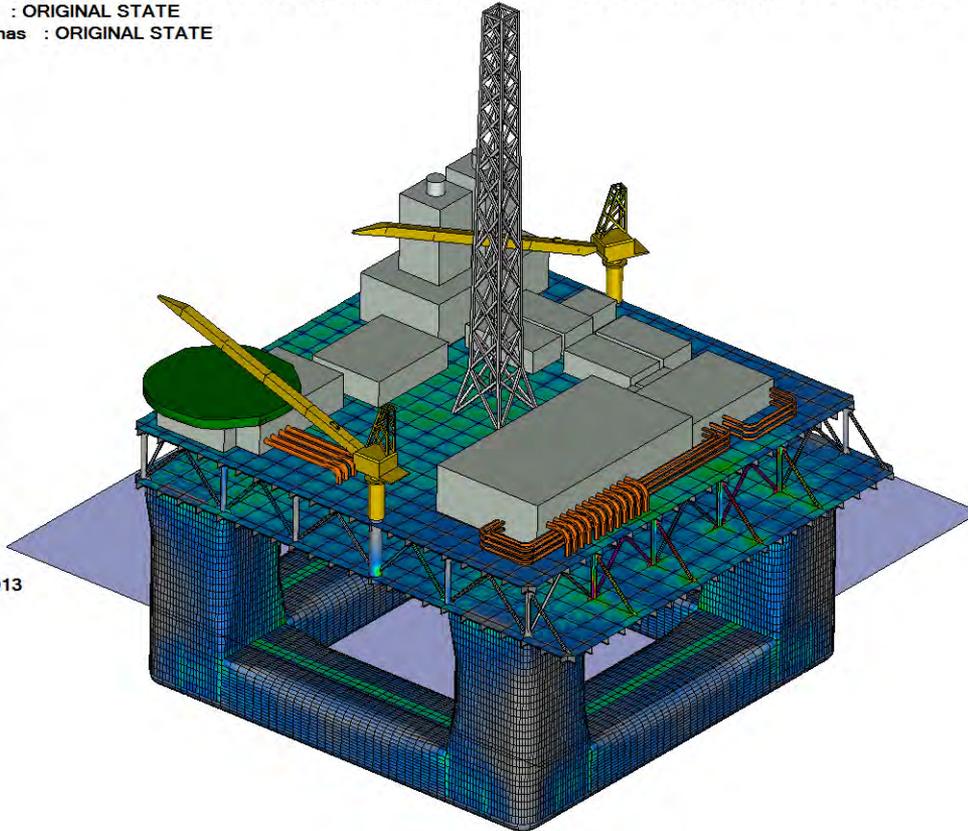
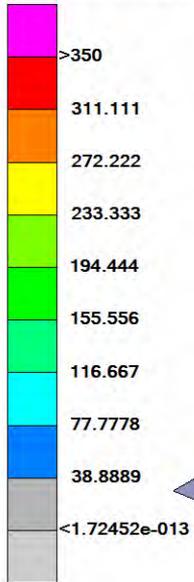
Fully loaded SEMI platform



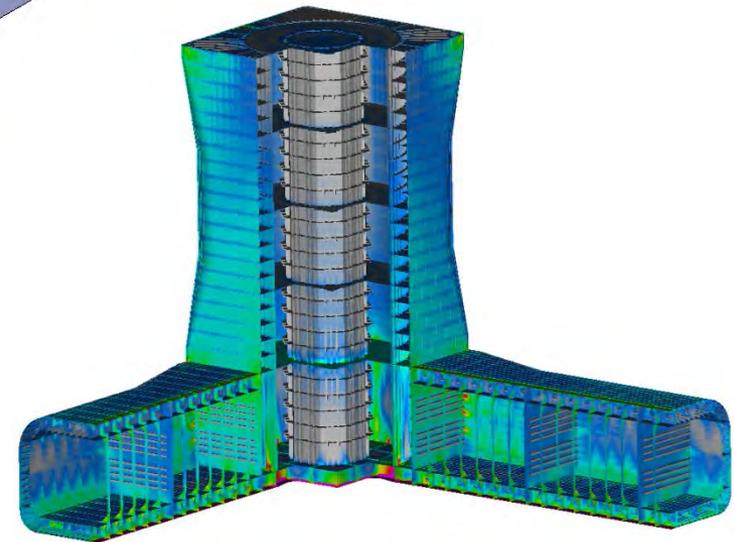
- Auxiliary mass distribution
- Waterline calculation in Tank Tool
- Buoyancy application with Marine & Offshore tool

Results in META

0:oilplatform-sol6.op2 : Stresses,Von Mises,Max of Top Bottom : SUBCASE 1 ::OILPLATFORM-SOL6:CASE2-HIGH_LEVEL: SU
1:wave_high.nas : ORIGINAL STATE
2:superstructure.nas : ORIGINAL STATE

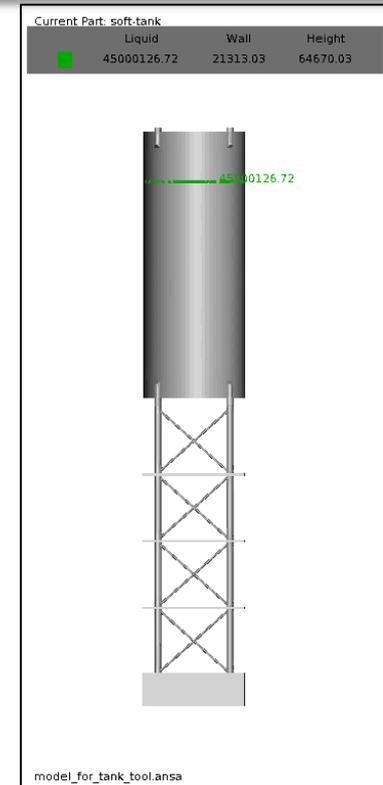
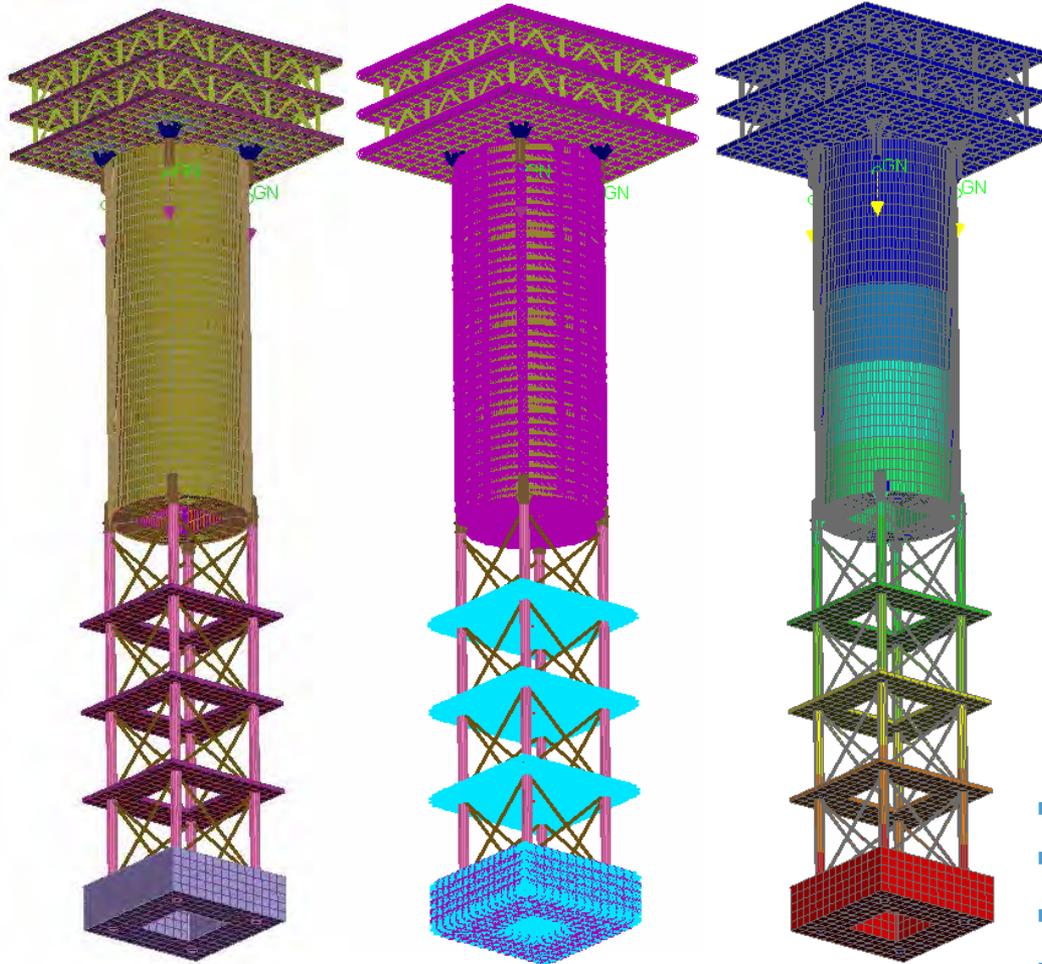


- Von Misses stresses
- The most Critical area
- Displacements



State of Equilibrium

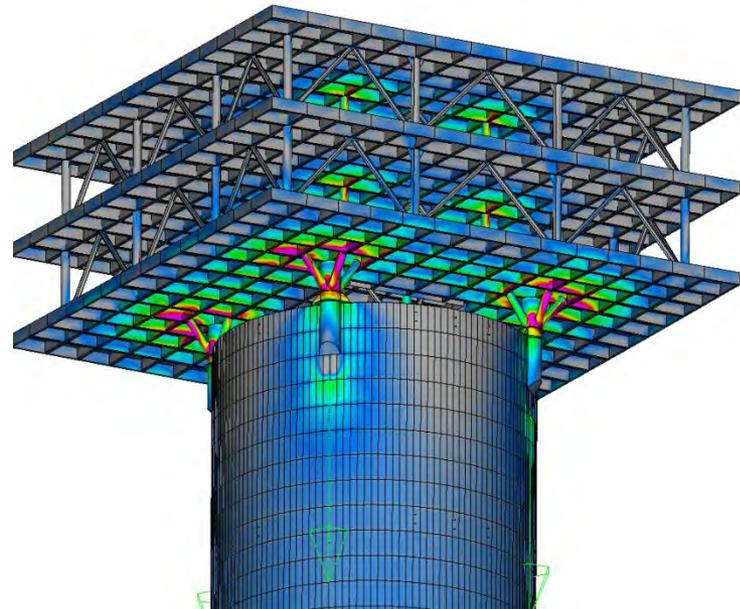
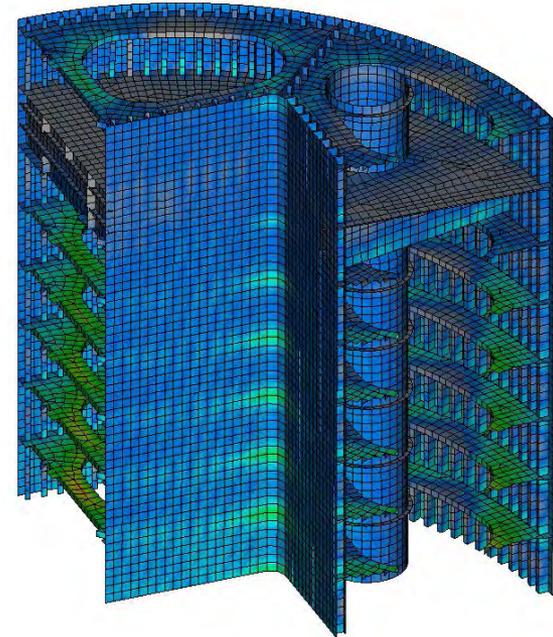
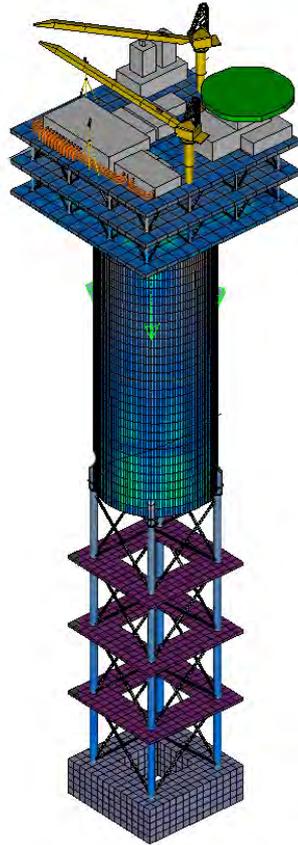
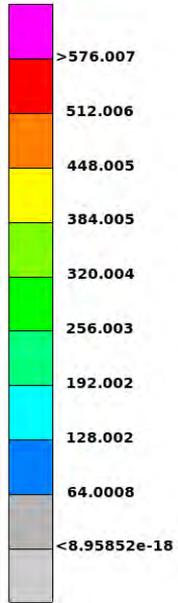
Fully loaded SPAR platform



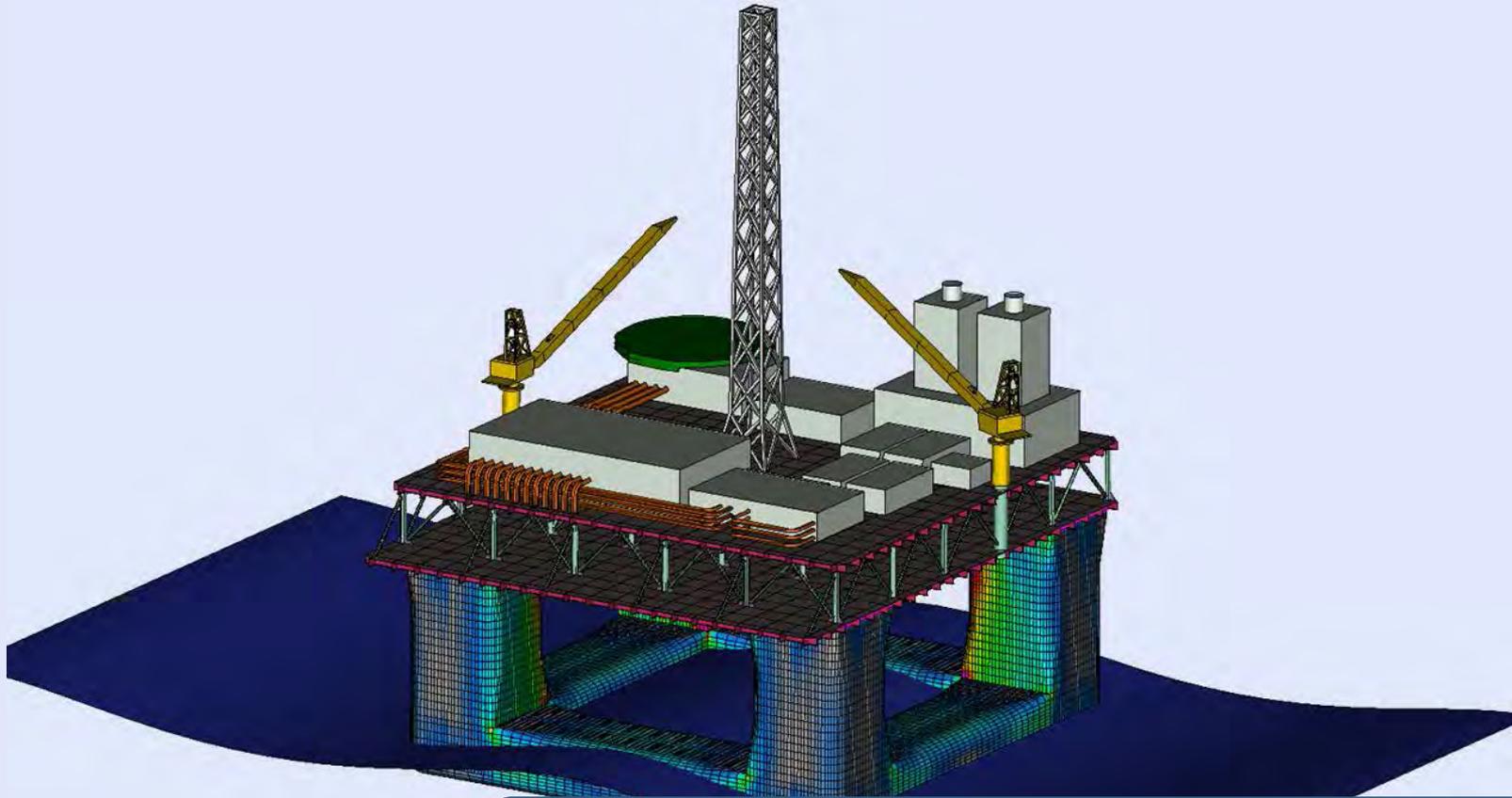
- Mooring Forces defined with GEBs
- Rigidization of the Soft Tank
- Auxiliary mass distribution
- Waterline calculation in Tank Tool
- Buoyancy application

Results in META

0:SPAR_sol3.nas : (fo1 375643) : Stresses,Von Mises,Max of Top Bottom : SUBCASE 1
2:wave_h64670.nas : ORIGINAL STATE
3:accessories.nas : ORIGINAL STATE

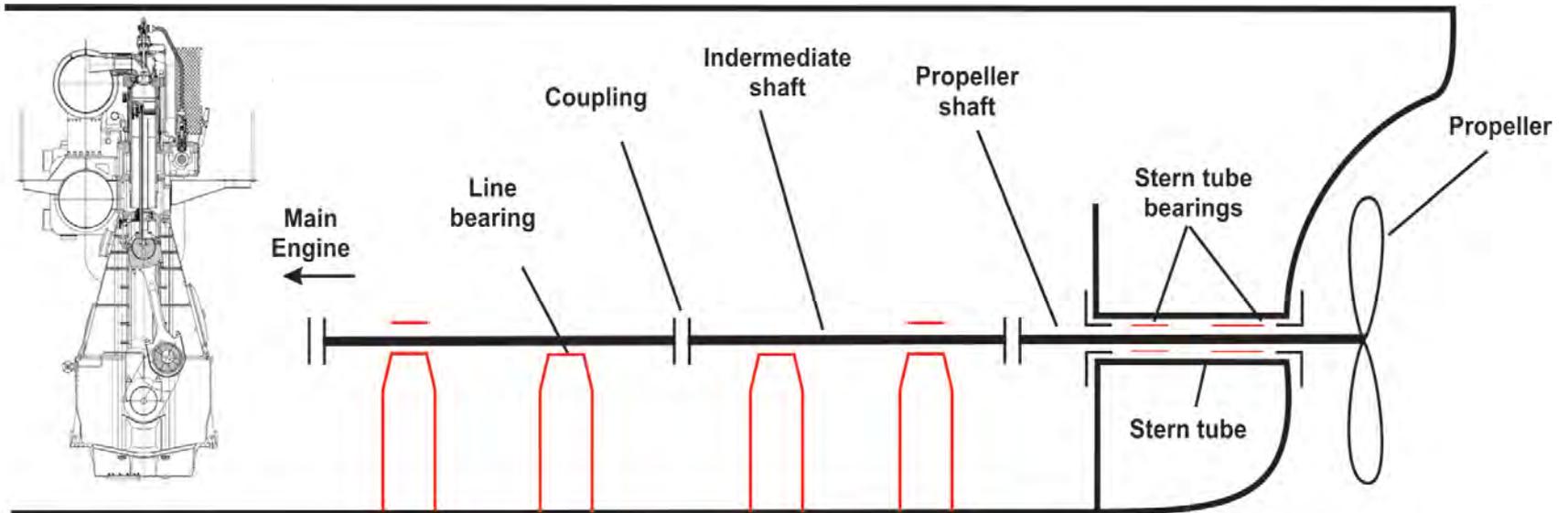


- Von Misses stresses
- The most Critical area
- Displacements



CASE STUDIES

Shaft alignment calculation



*Presented at:
COMPIT 2014 12-14 May, Redworth, UK*

Shaft alignment calculation

Process workflow

Matrix analysis

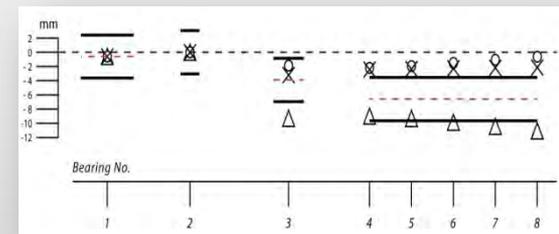
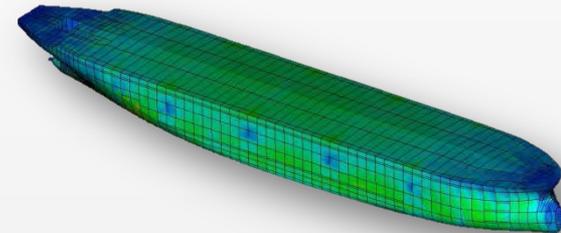
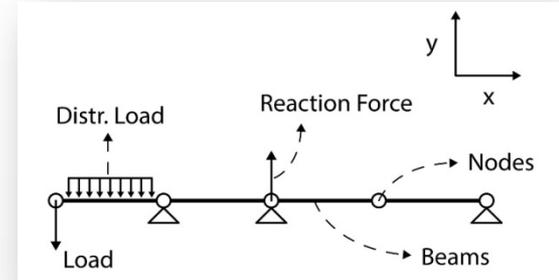
- Statically indeterminate multi-supported beams
- Bearing stiffness and clearance
- Dry dock conditions

FE analysis

- Hydrostatic equilibrium
- Hull deformations calculation
- Determination of vertical displacements at the bearing locations

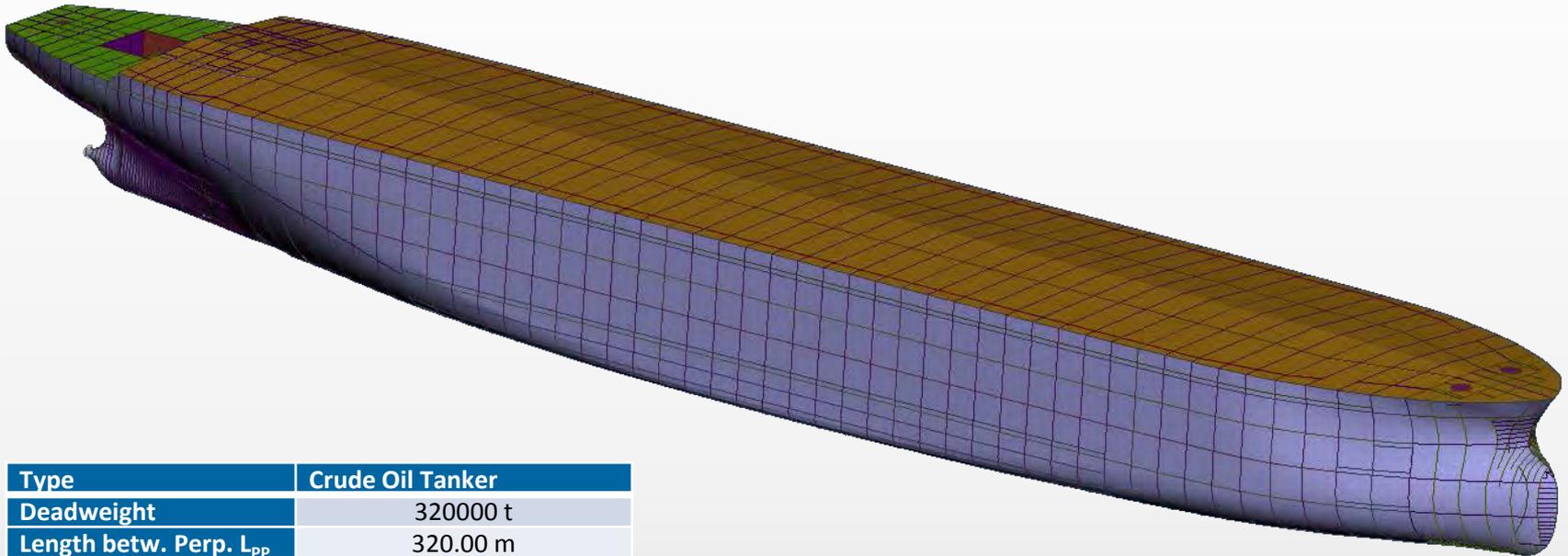
RE-evaluation

- Re-evaluation of static shaft equilibrium



Shaft alignment calculation

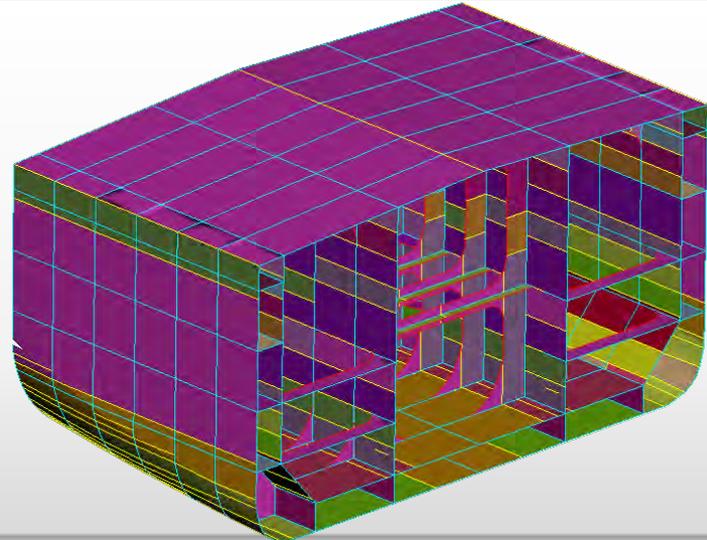
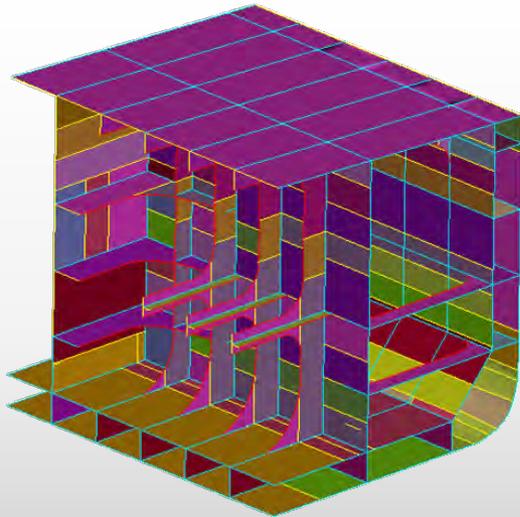
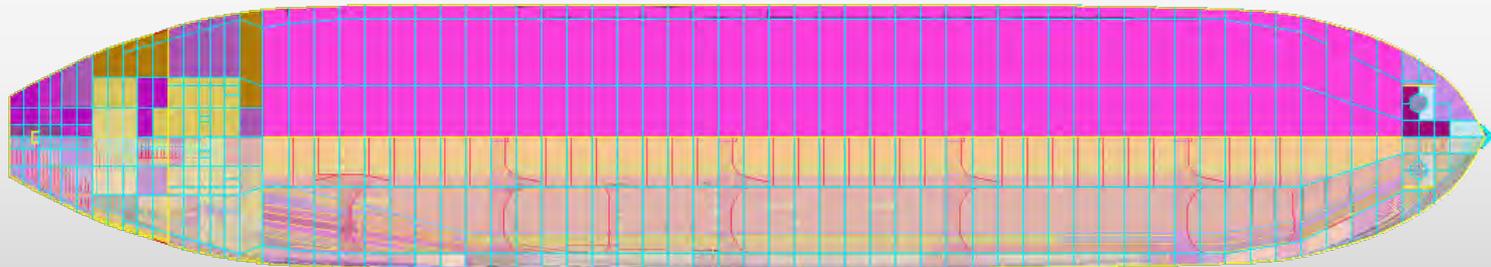
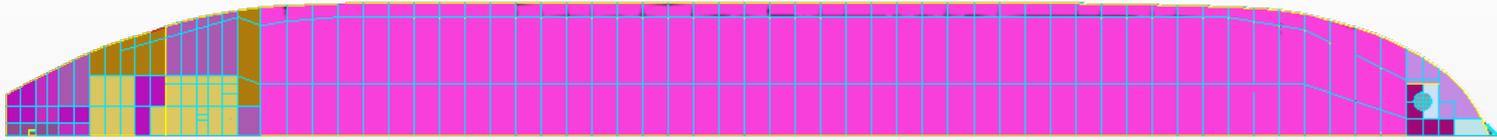
The geometrical model



Type	Crude Oil Tanker
Deadweight	320000 t
Length betw. Perp. L_{pp}	320.00 m
Breadth B	60.00 m
Depth D	30.50 m
Scantling draft T	22.50 m
Service speed V_s	15.9 kn
Main engine	Wärtsilä 7RT-FLEX84T-D
Keel laid	April 2010

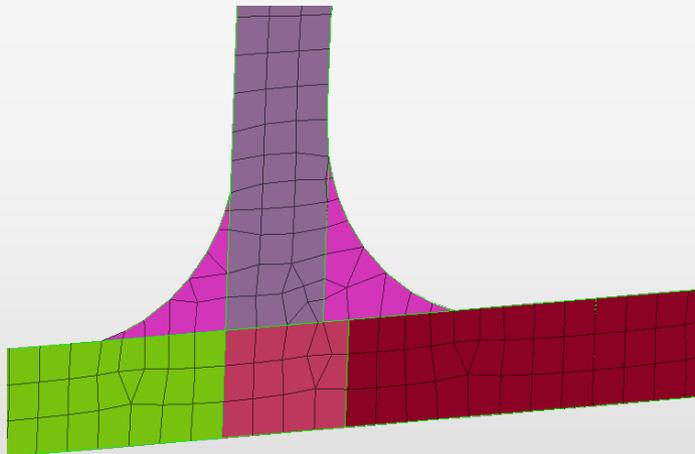
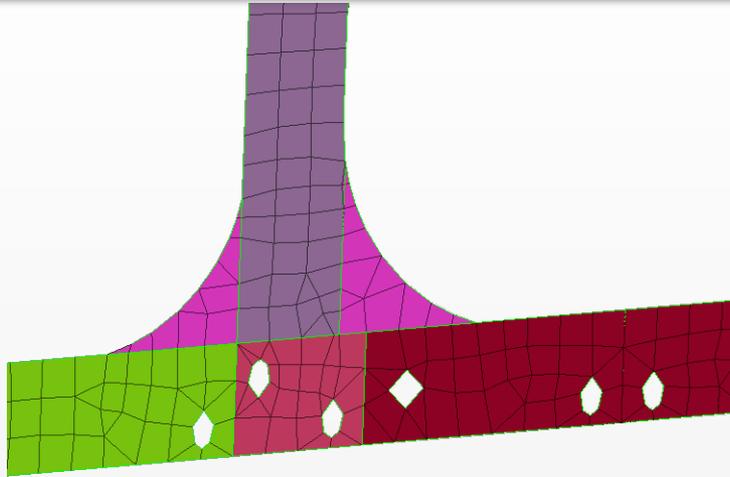
Shaft alignment calculation

Definition of the symmetrical model using geometry representation

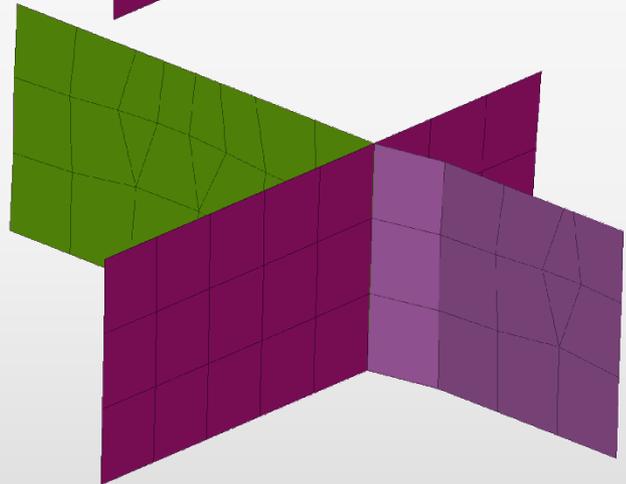
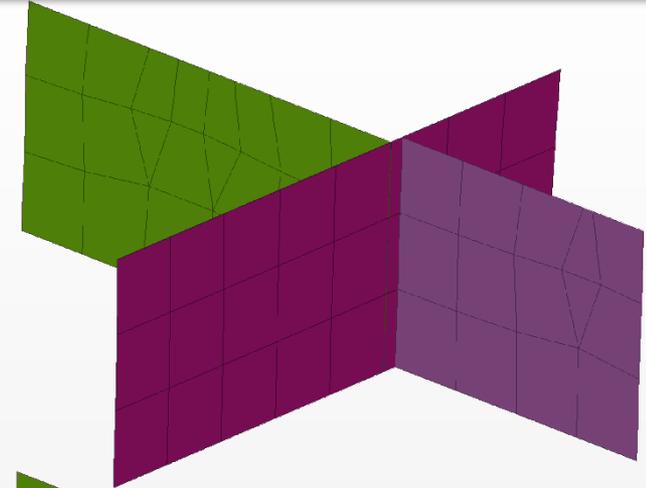


Shaft alignment calculation

FE model simplification



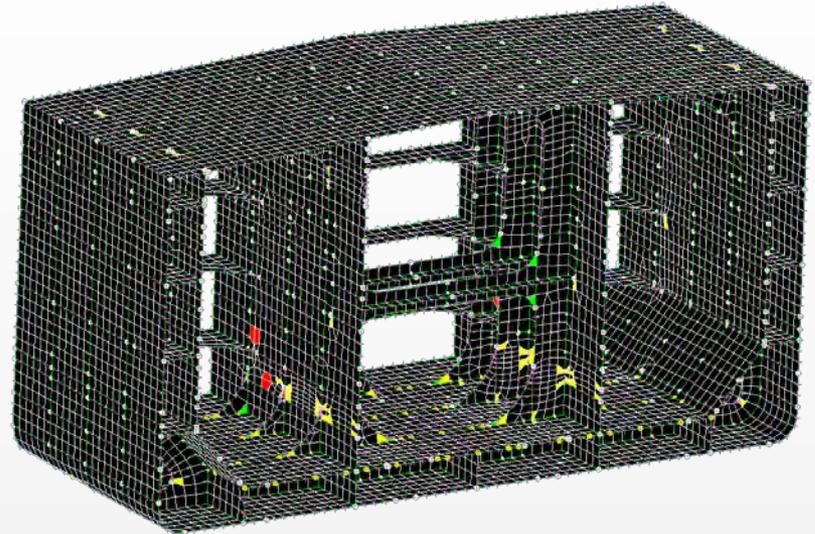
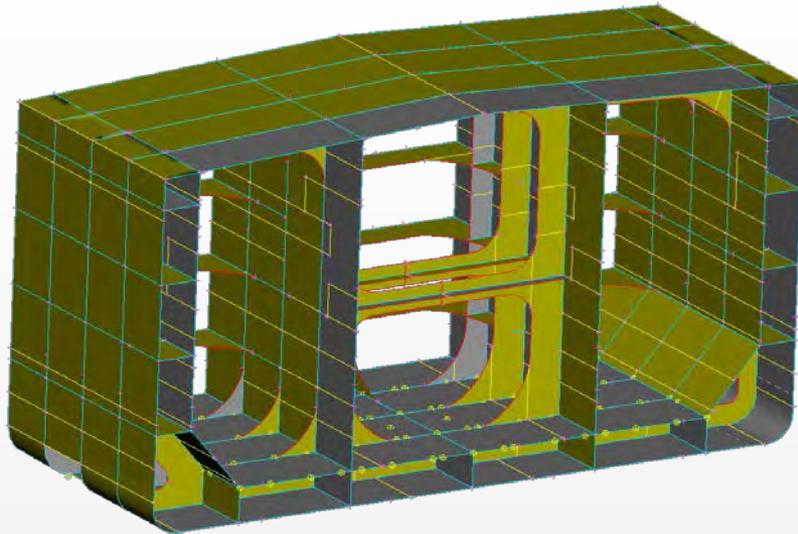
- Removing holes with diameter smaller than 1m



- Moving T-Junction ribs to avoid narrow elements

Shaft alignment calculation

Automatic meshing using the Batch Meshing tool of ANSA



Name	Contents	Mesh Parameters	Quality Criteria	Status
<input checked="" type="checkbox"/> Meshing_Scenario_1 <input checked="" type="checkbox"/> Default_Session	29	1m	Untitled	Completed
<input checked="" type="checkbox"/> Meshing_Scenario_2 <input checked="" type="checkbox"/> Default_Session	18	0.2m	Untitled	Completed
<input checked="" type="checkbox"/> Volume_Scenario_3 <input checked="" type="checkbox"/> Default_Session	0	Untitled	Untitled	Empty

Global Meshing Parameters (Scenario I)

Element length 0.95 m

Filling openings with diameter < 1m

Engine room floor Meshing Parameters (Scenario II)

Element length 0.2

Filling openings with diameter < 0.5m

Quality Criteria

Skewness (Nastran) 30°

Aspect ratio (Nastran) 3

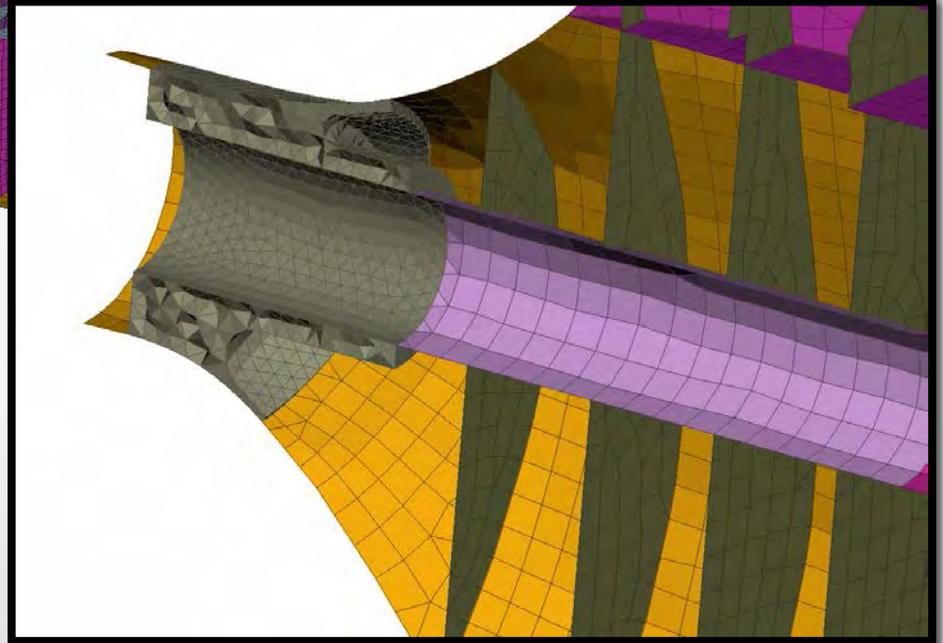
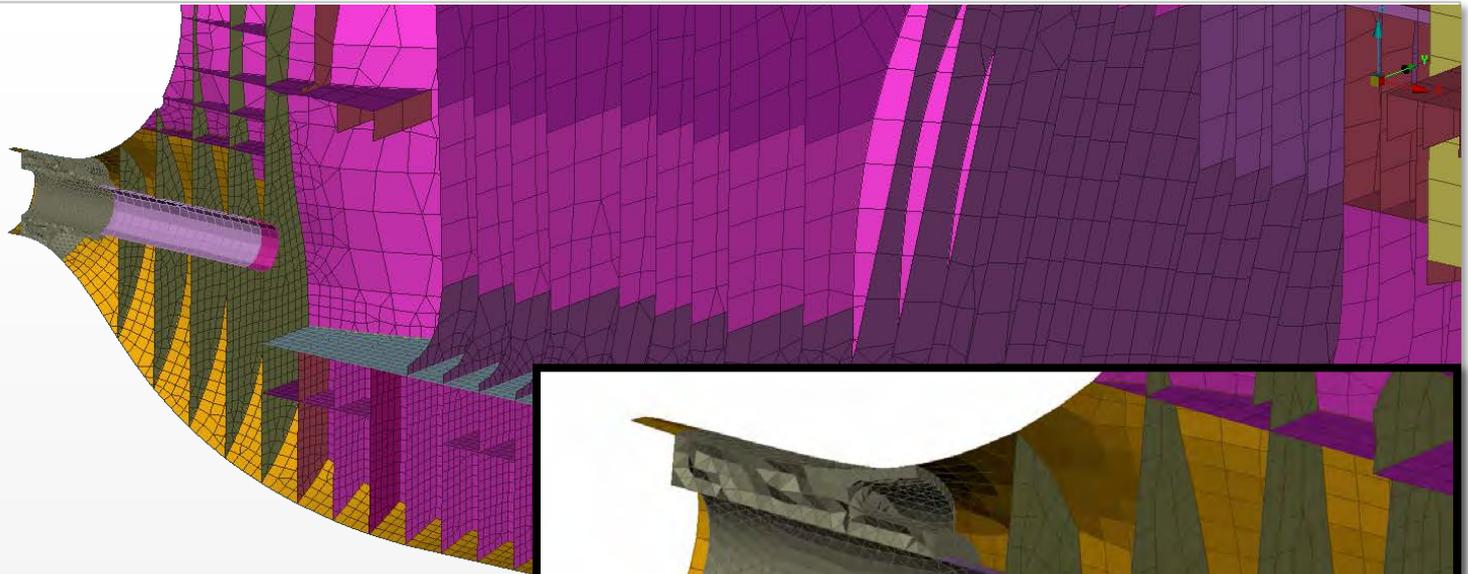
Angle (Quads) 45-135°

Angle (Trias) 30-120°

Minimum Element Length 0.01 m

Maximum Element Length 1.5 m

Shaft alignment calculation

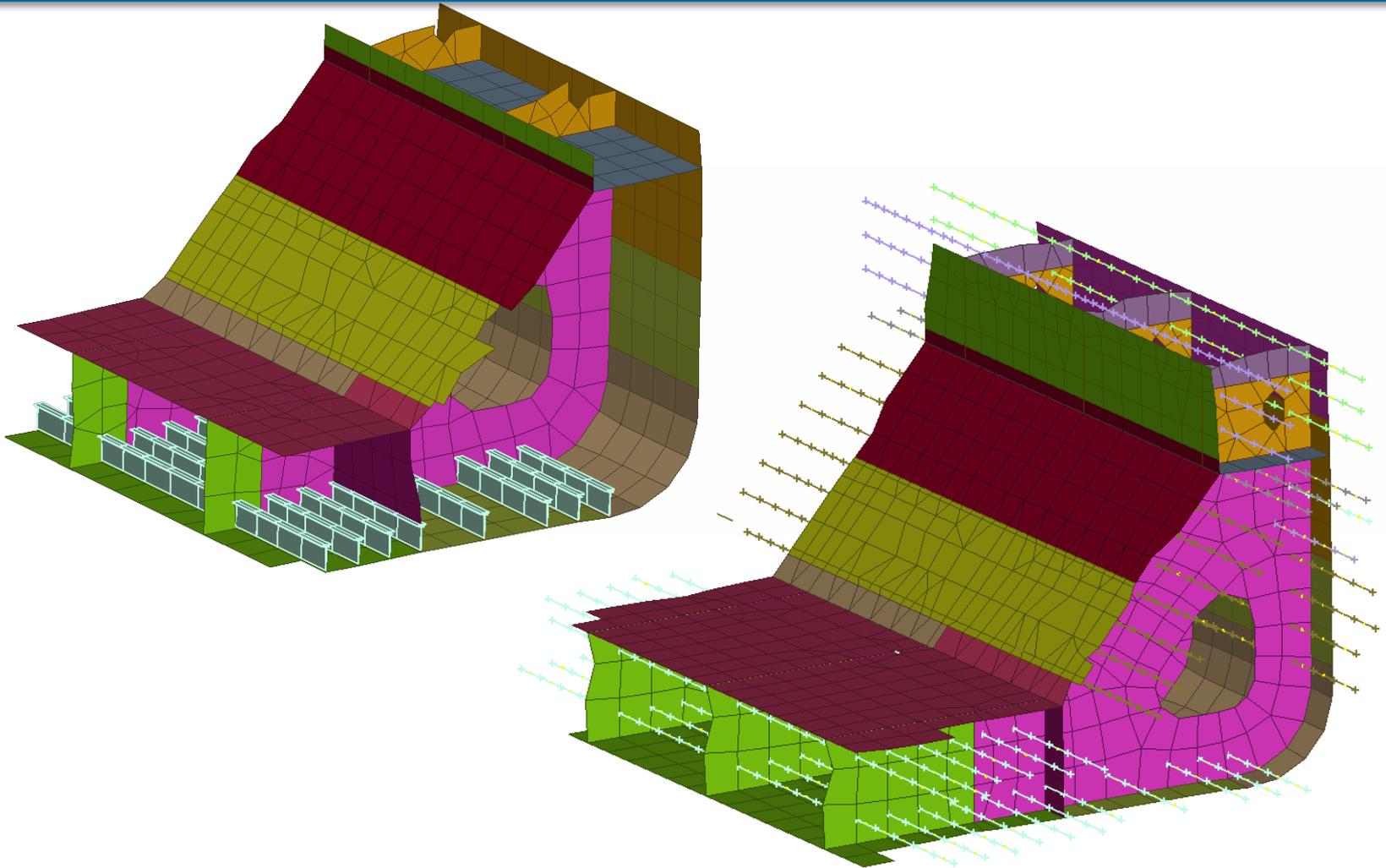


- Fine mesh on engine room
- Solid mesh on stern tube

Total number of elements	
Shell elements	402000
Beam elements	143000
Solid tetrahedrals	17000

Shaft alignment calculation

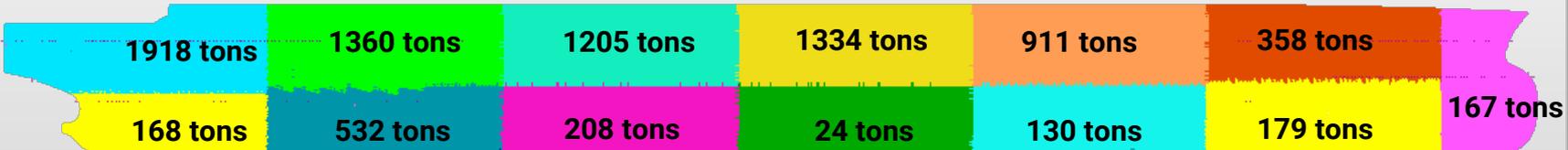
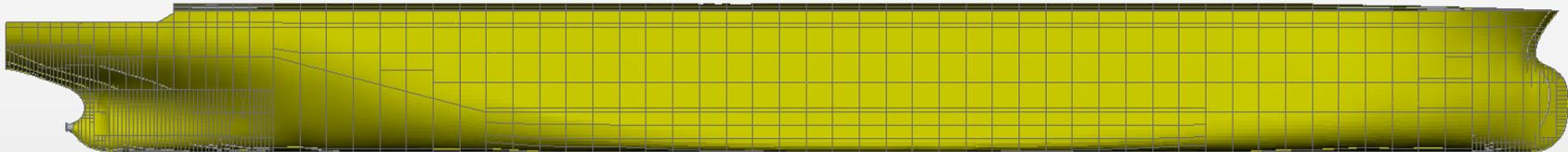
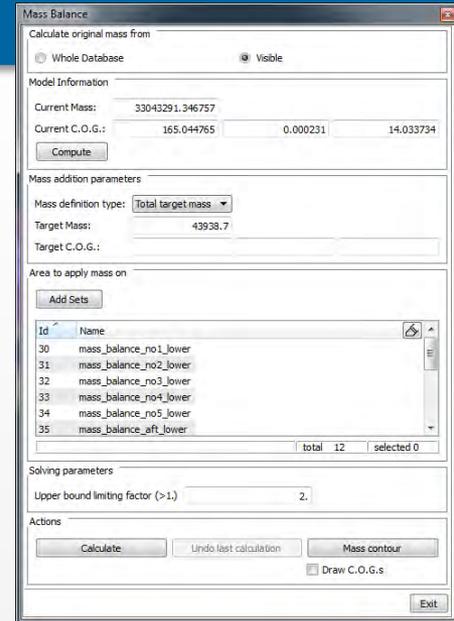
Stiffeners representation with BEAMS and TRUSS



Shaft alignment calculation

Non-structural mass distribution

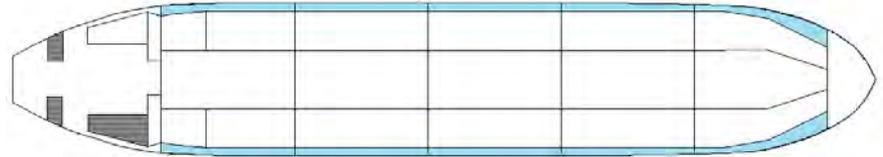
- FE structure: 34442 tons
- Lightship: 43938.7 tons
- Added mass: 9496.7 tons



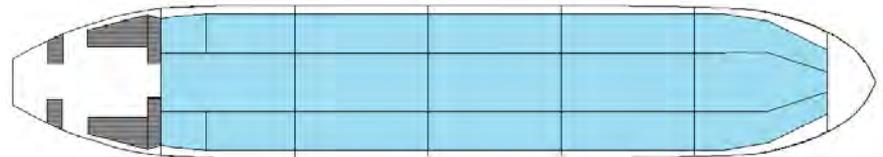
Shaft alignment calculation

Loading conditions

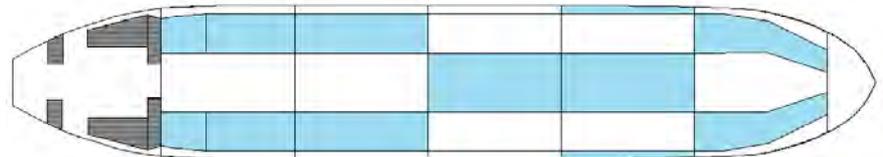
Ballast arrival condition (L.C. 1)
Displacement: 145647 tones
Draft: 9.69 m
Trim: 2.12 m



Full-load departure condition (L.C. 2)
Displacement: 364074 tones
Draft: 22.52 m
Trim: 0.11 m

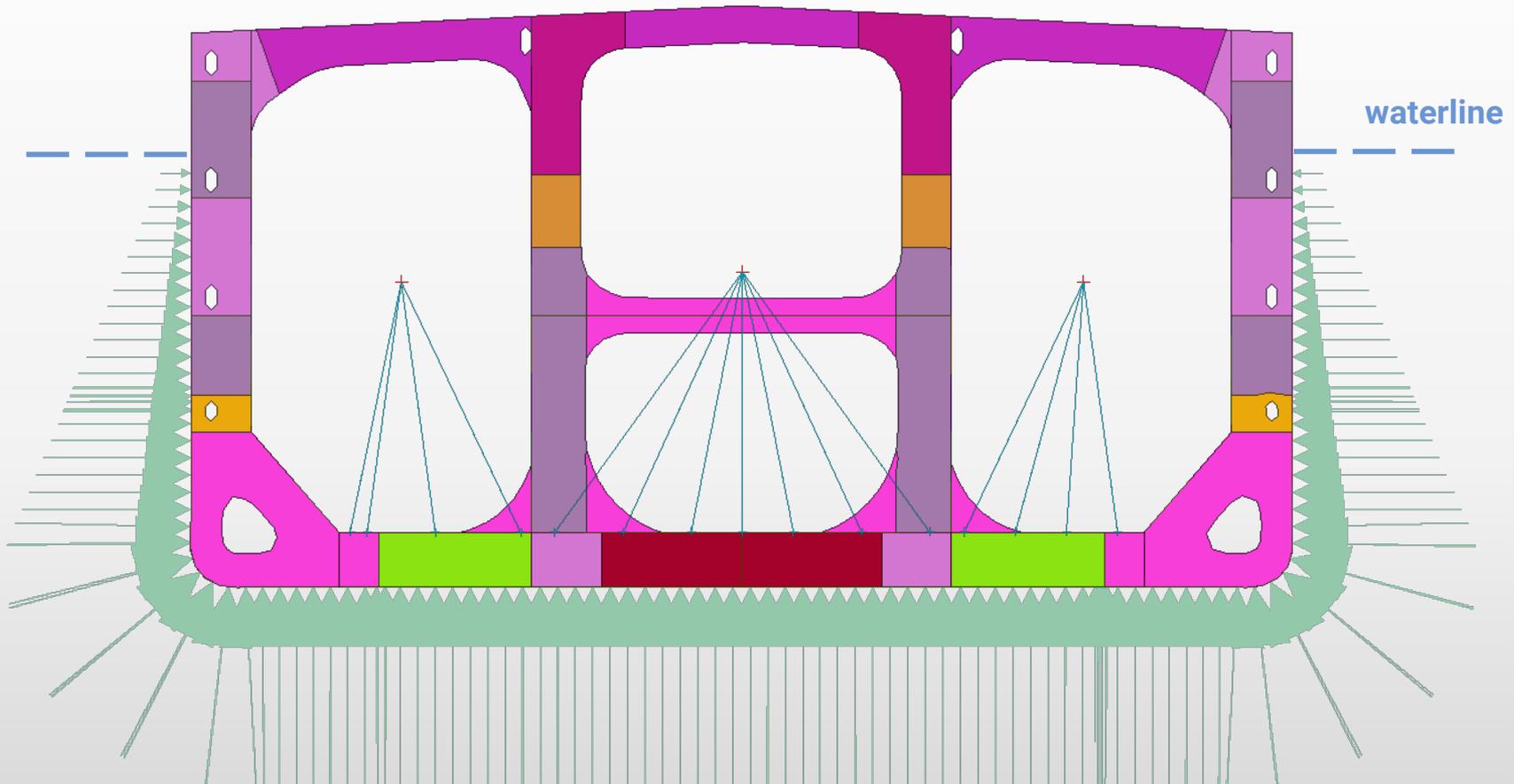


Departure with partial load (L.C. 3)
Displacement: 229276 tones
Draft: 14.78 m
Trim: 3.05 m



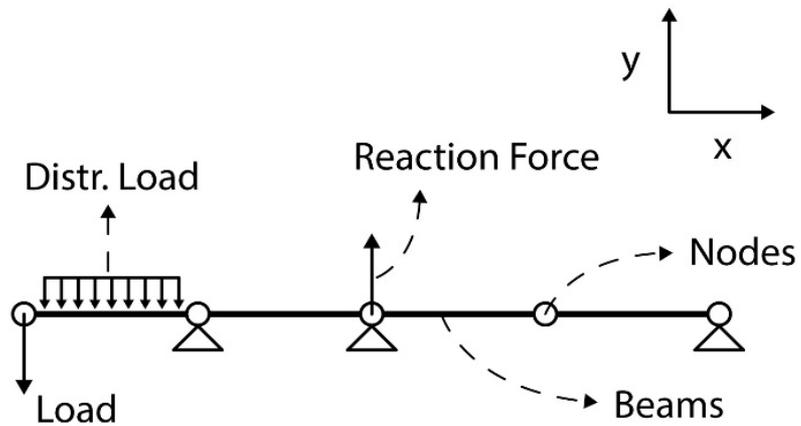
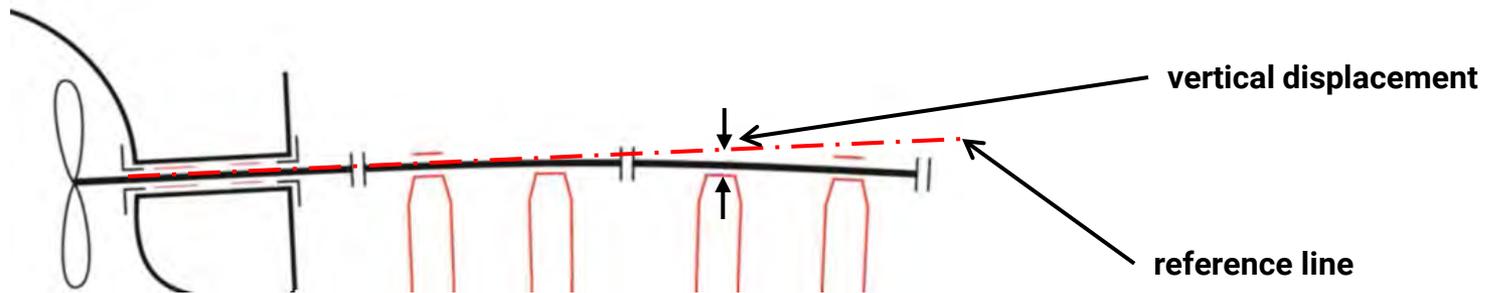
Shaft alignment calculation

- Application of hydrostatic pressure due to buoyancy
- Application of loads as mass connected with RBE3
- Balance of the hull and Inertia Relief application



Shaft alignment calculation

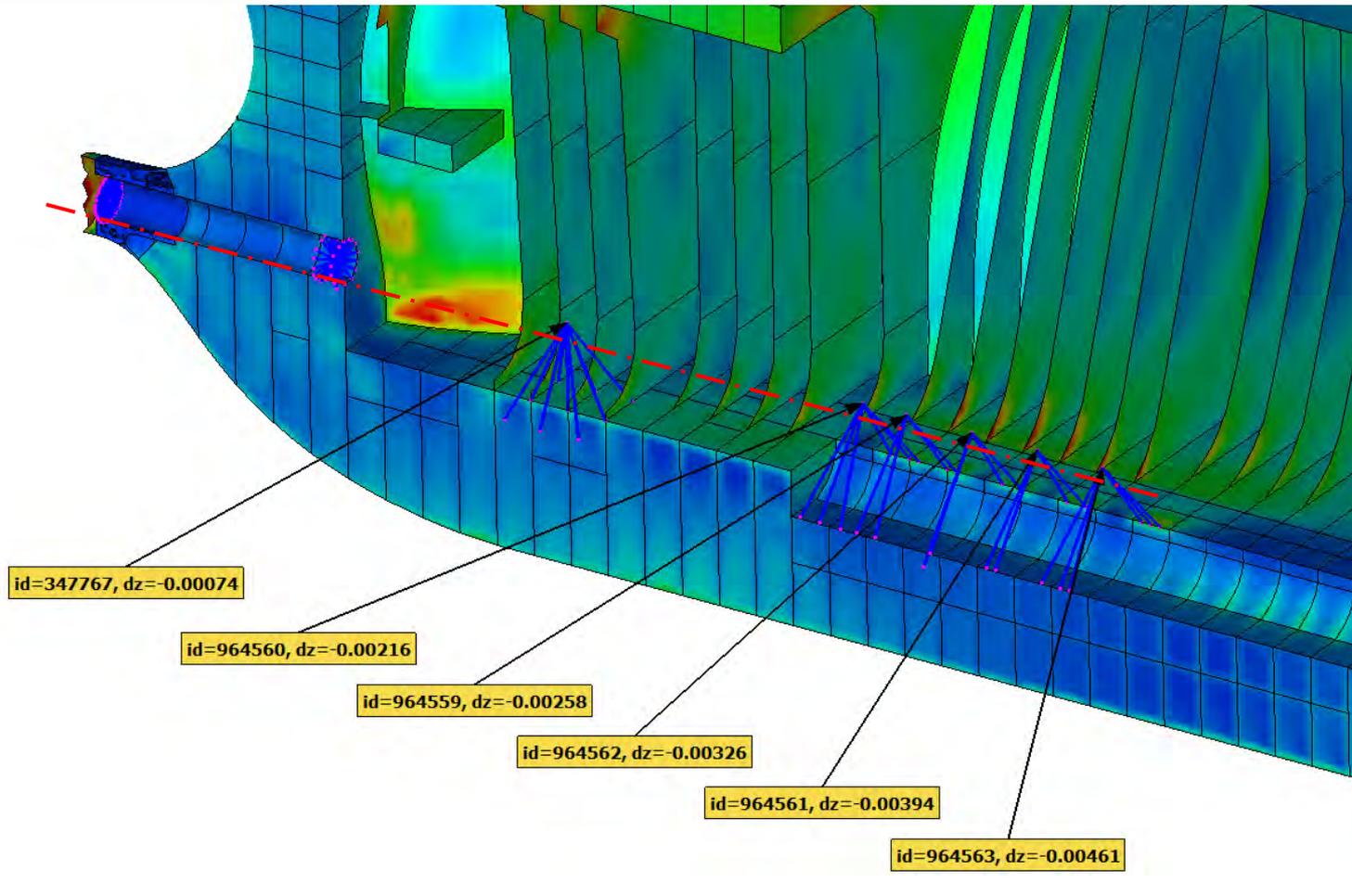
- Set up reference line
- Define the static conditions of the vessel
- Modeling the shafting system



Shaft alignment calculation

Computational Results

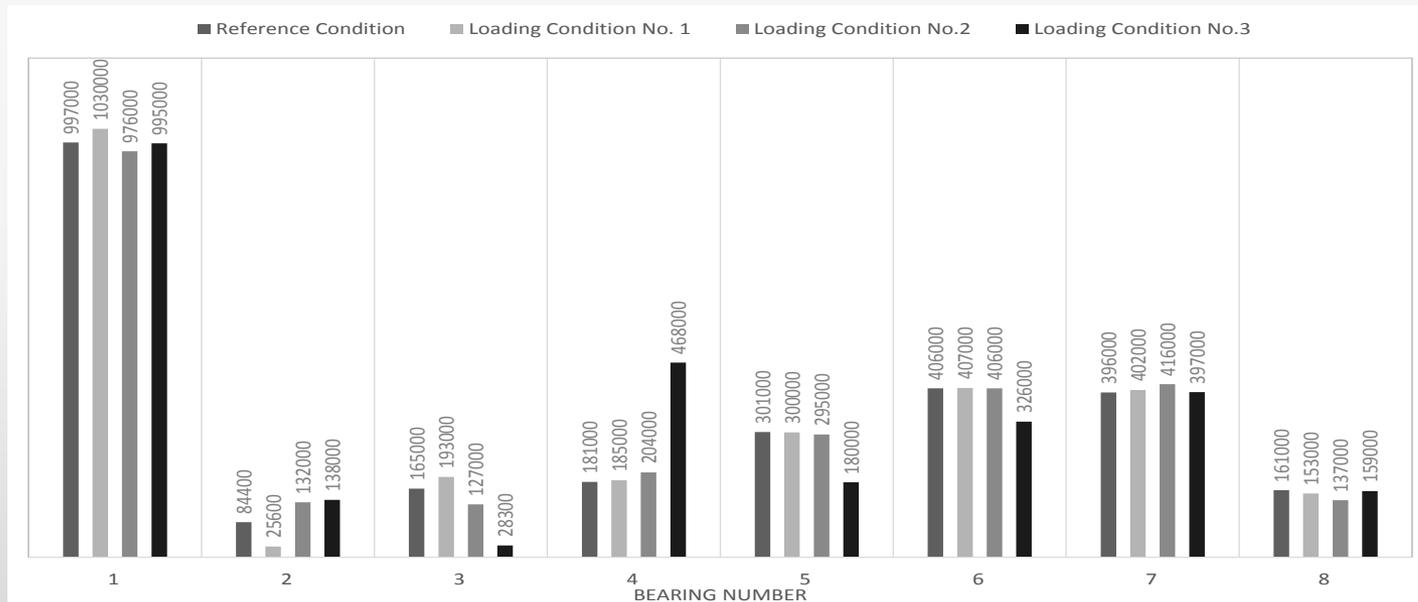
Displacements of bearings according to reference line



Shaft alignment calculation

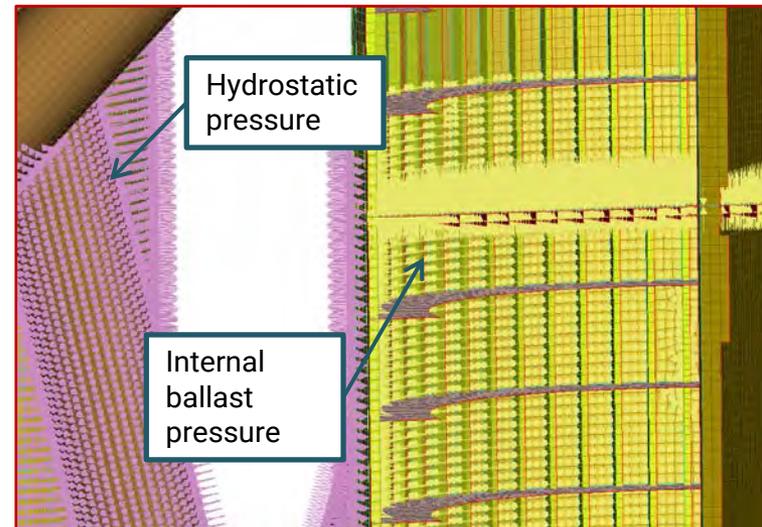
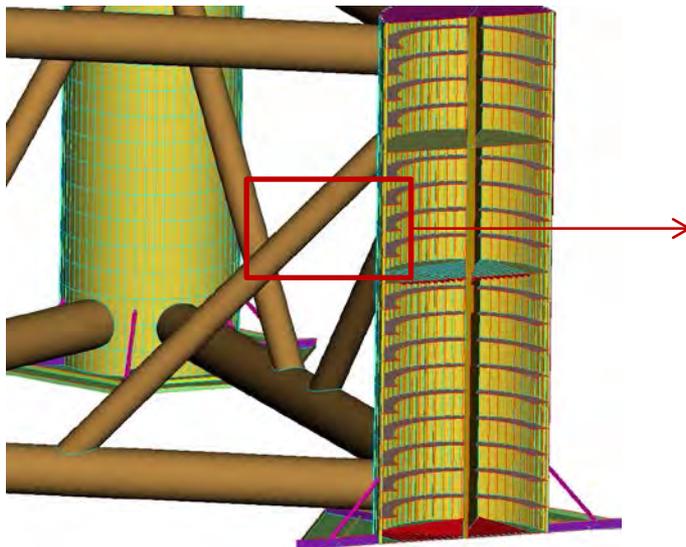
Bearing vertical offsets at different loading conditions

Bearing	Initial case (even keel)	Loading Condition 1	Loading Condition 2	Loading Condition 3
Aft S/T	-0.06	-0.06	-0.06	-0.06
For S/T	0.00	0.00	0.00	0.00
Intermediate	-3.9	-1.89	-4.66	-3.12
M/E 1	-6.60	-2.27	-8.99	-2.44
M/E 2	-6.60	-1.96	-9.32	-2.37
M/E 3	-6.60	-1.50	-9.85	-2.28
M/E 4	-6.60	-1.06	-10.44	-2.20
M/E 5	-6.60	-0.63	-11.07	-2.14



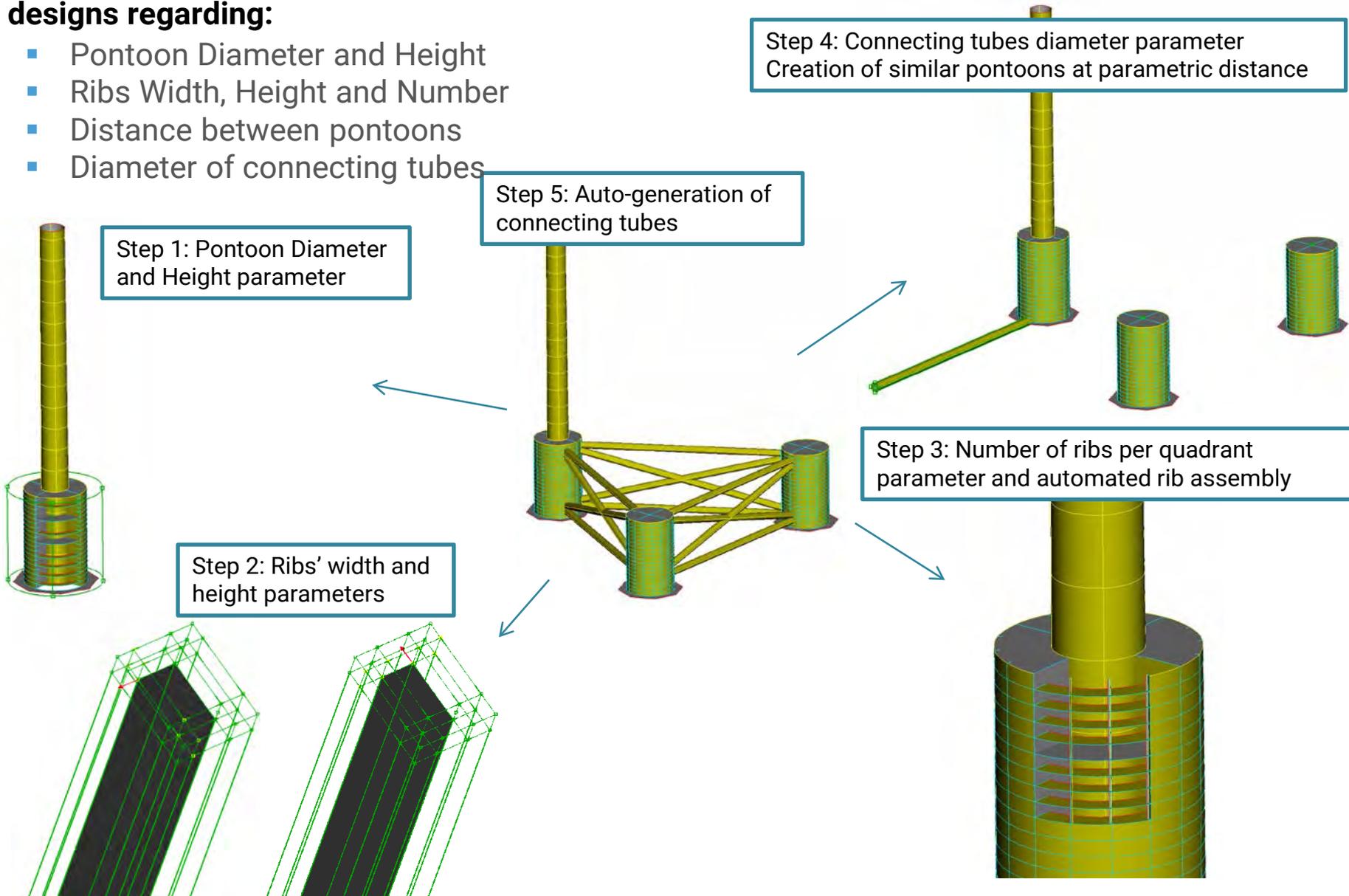
Offshore Wind Turbine Geometry Parameterization and structural analysis

- Structural analysis of offshore wind turbine aiming to see displacements and stresses under load.
- Highly detailed model including ribs and stiffeners
- Pressure loads were auto calculated and applied both for the sea hydrostatic pressure and the Ballast reservoirs.
- Gravity and an equivalent concentrated air pressure at the turbine nacelle



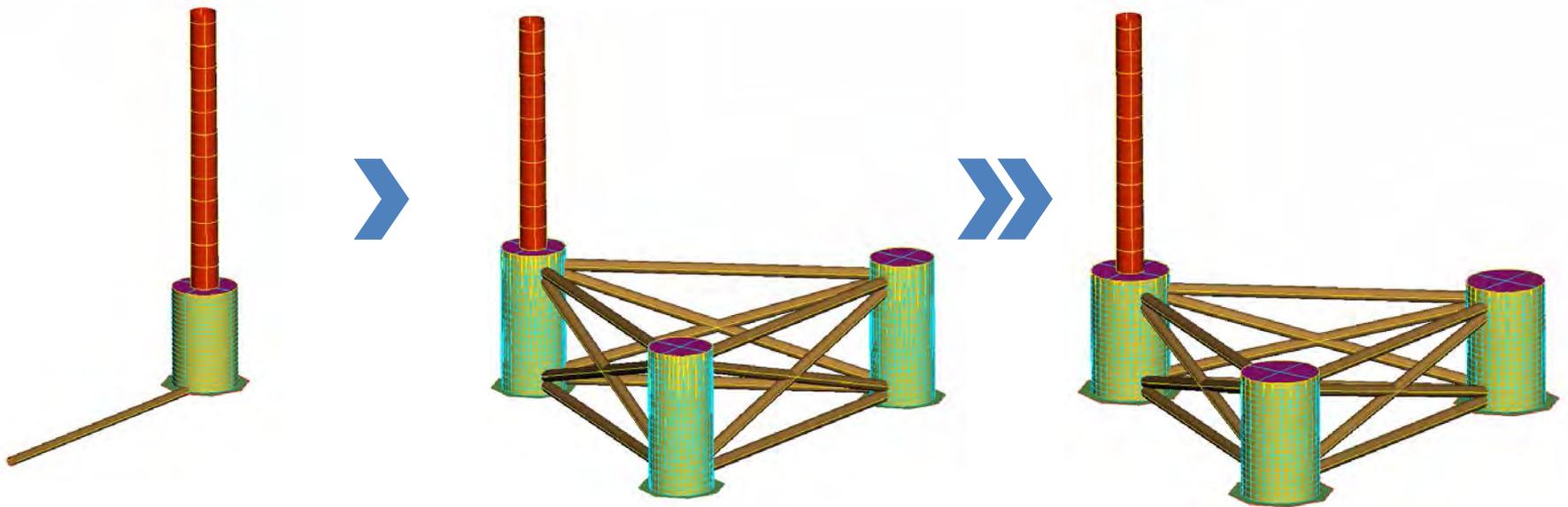
The geometrical model was parameterized in order to automatically generate different designs regarding:

- Pontoon Diameter and Height
- Ribs Width, Height and Number
- Distance between pontoons
- Diameter of connecting tubes



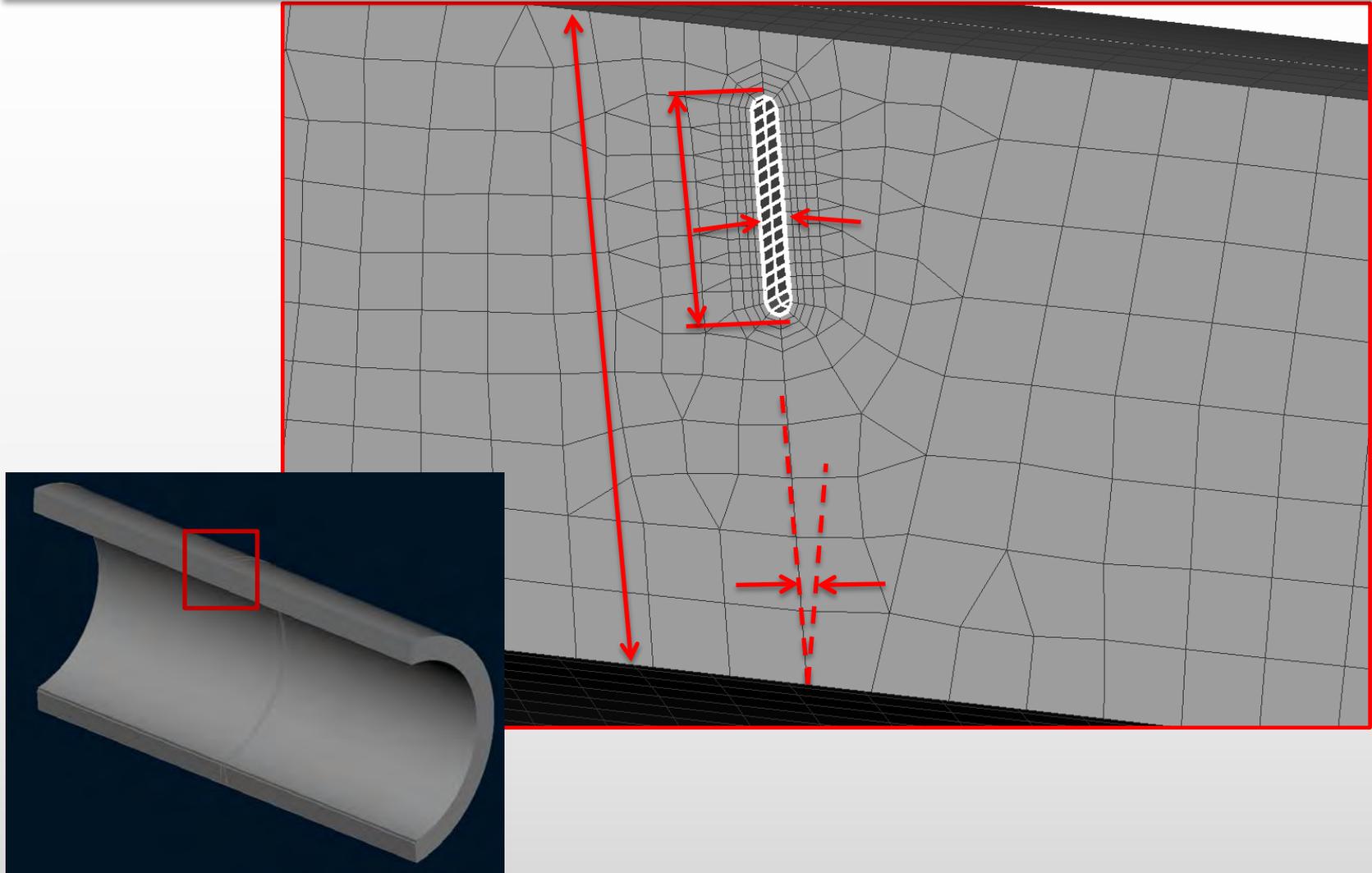
Design Of Experiments

With this automated functionality, Multiple designs can be created with different configurations as a Design Of Experiments study, using several different algorithms (ULH, Random, Full Factorial, Taguchi) to provide the Design Variable values



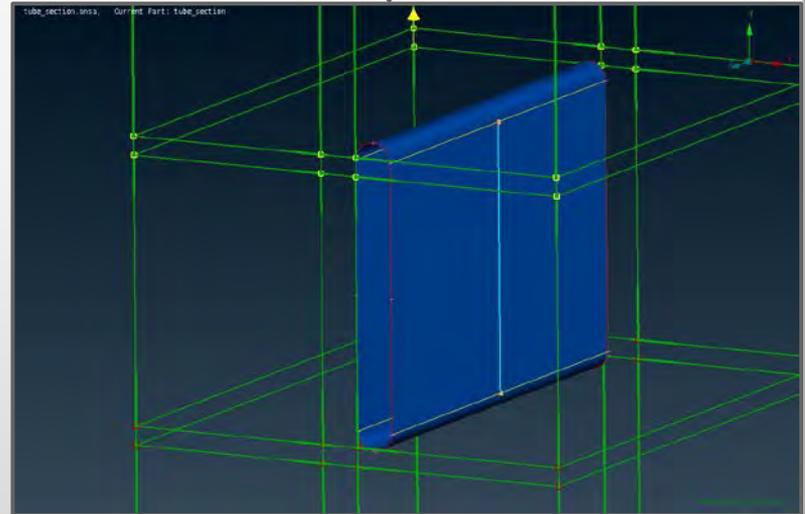
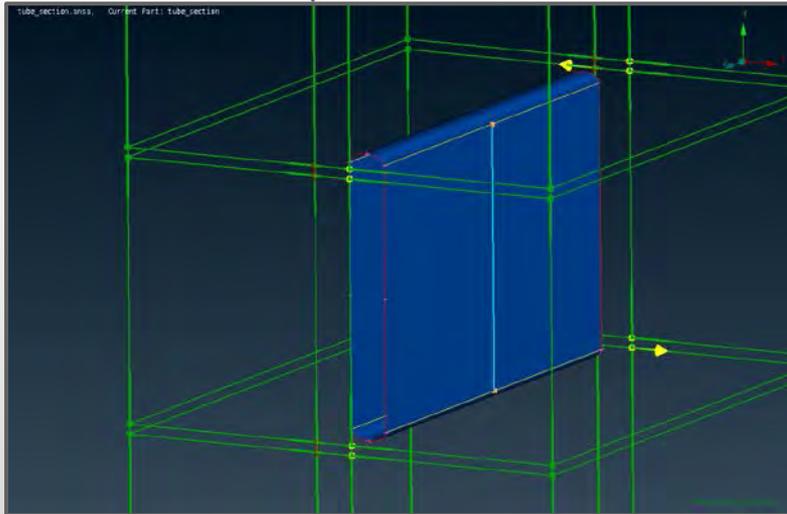
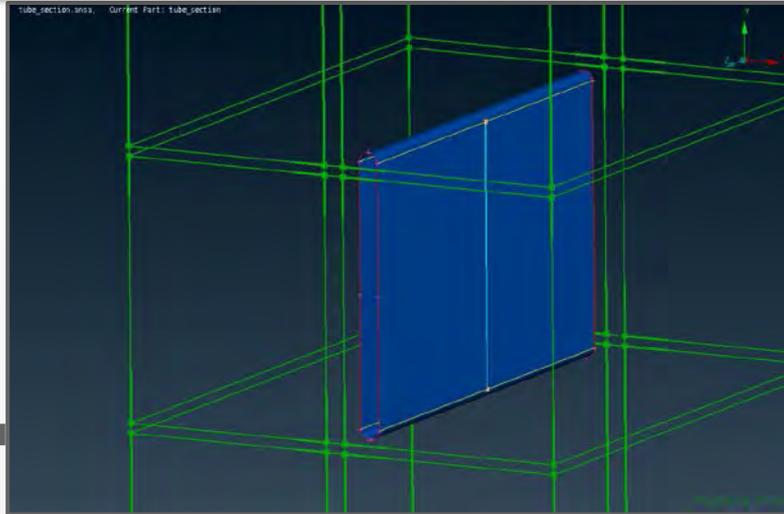
Automated Crack modeling and parameterization

Requirements



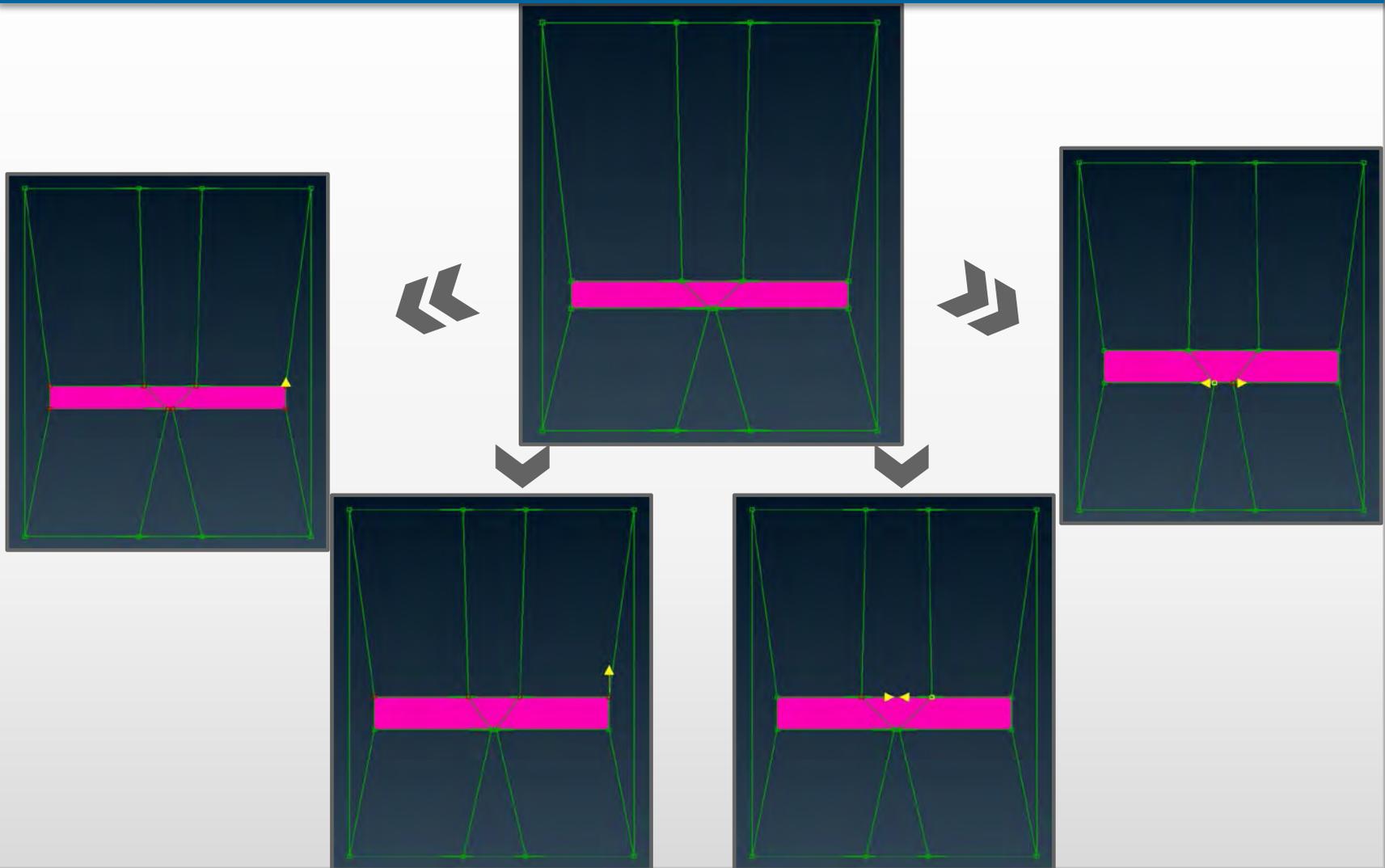
Automated Crack modeling and parameterization

Crack body parameters



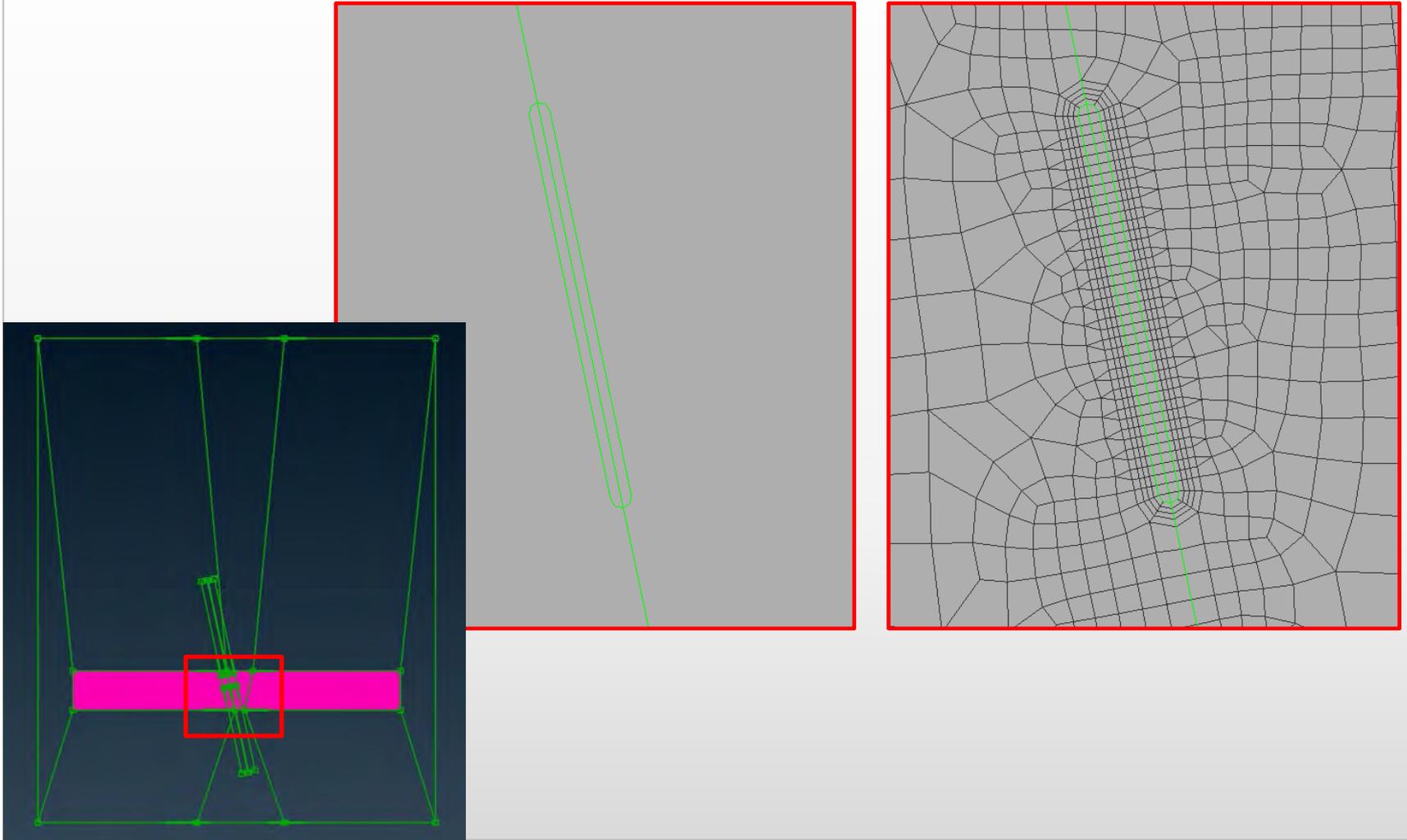
Automated Crack modeling and parameterization

Pipe body parameters



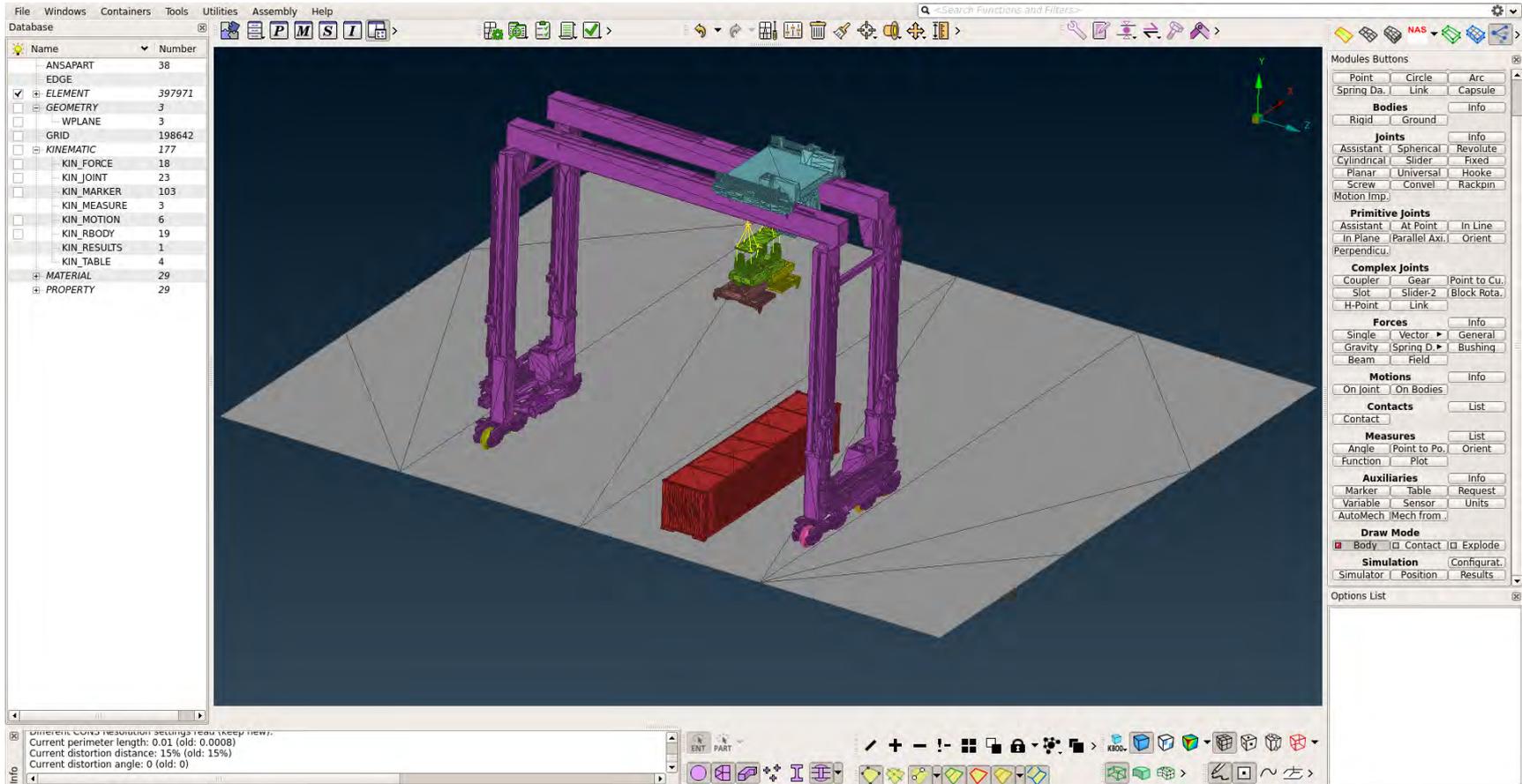
Automated Crack modeling and parameterization

Shell mesh creation



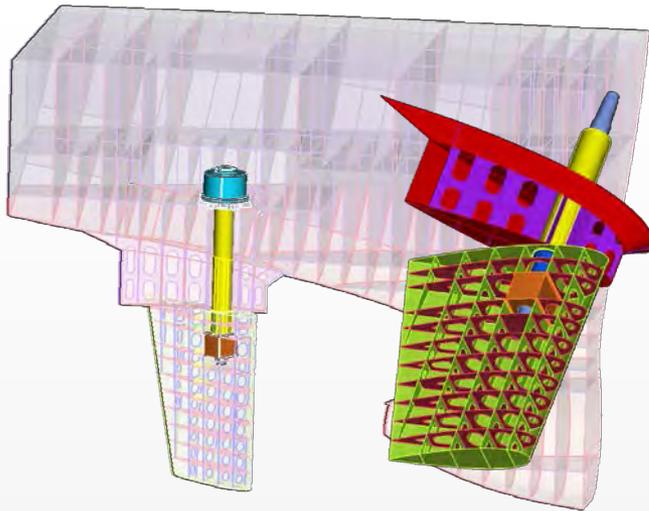
Crane Lifting simulations with the Kinetics Module

Kinetic mechanism definition

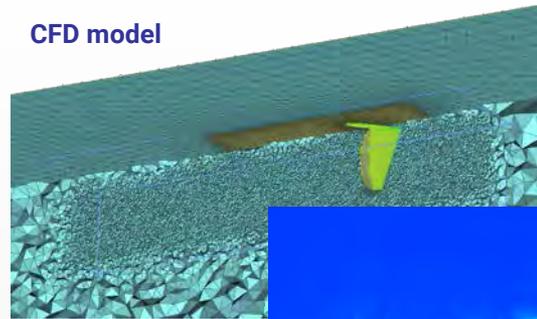


- ✓ 19 bodies
- ✓ 5 sliders
- ✓ 8 revolute
- ✓ 6 imposed motions
- ✓ 18 spring dampers
- ✓ 2 spherical
- ✓ 8 links

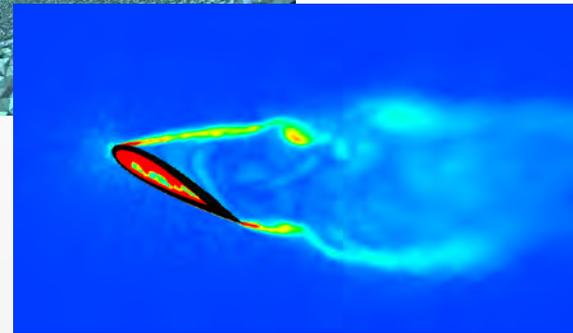
Rudder Optimization Study



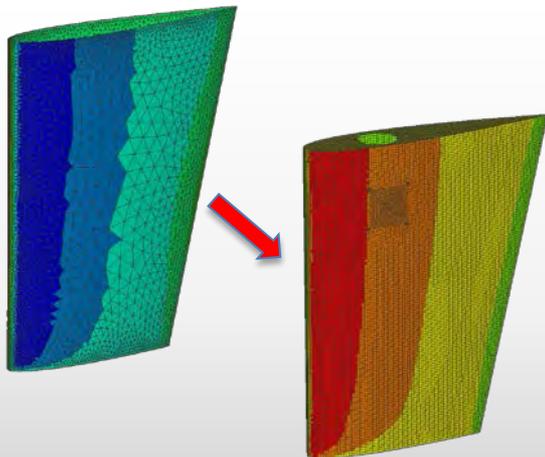
CFD model



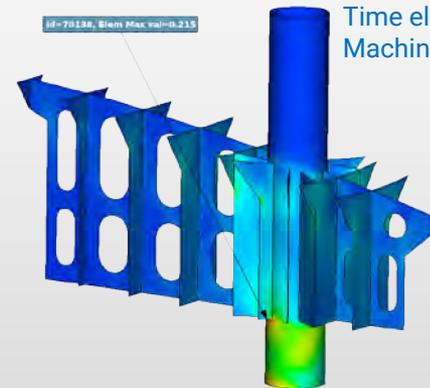
Solver: **FLUENT v13.0**
Iterations to converge: ~1000
Time elapsed: **24 hours**
Machine: **Linux, Core i7 8 CPUs**



Results Mapping Tool



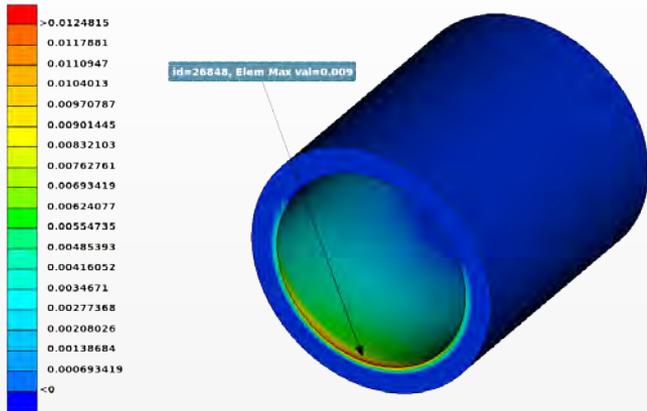
C:\rudder_from_cfd.cad : propeller.aspcing : Stress components,Von Mises,Max of In,Out,Centroid : : STEP 1 [AnonymousSTEP1],TIME 4.00000004E-01.



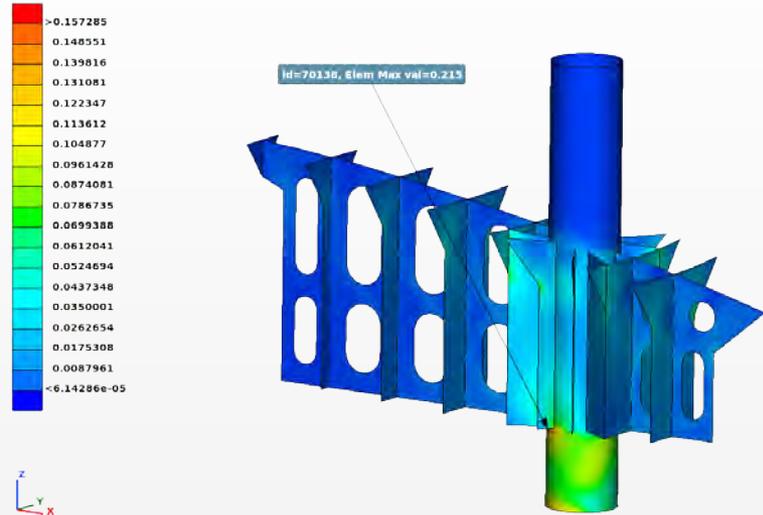
Solver: **ABAQUS v 6.10**
Time elapsed: **30 min**
Machine: **Linux, Core i7 8 CPUs**

Rudder Optimization Study

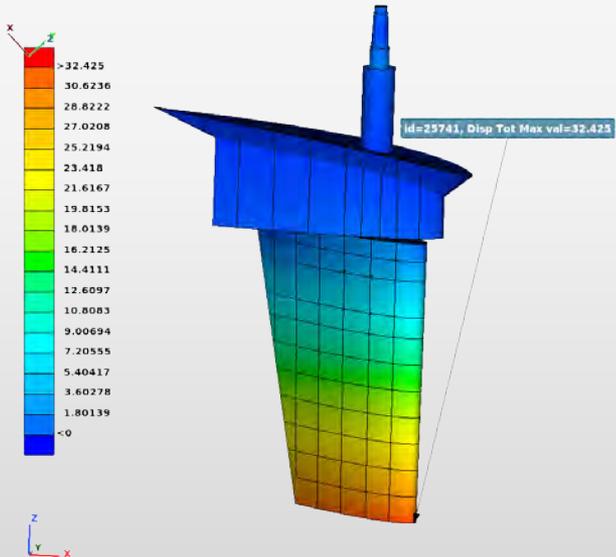
0:rudder_from_cfd.odb : propeller aspicing : Contact pressure,All Surfaces : STEP 1 (AnonymousSTEP1),TIME 4.00000006E-01,



0:rudder_from_cfd.odb : propeller aspicing : Stress components,Von Mises,Max of In Out,Centroid : STEP 1 (AnonymousSTEP1),TIME 4.00000006E-01,



0:rudder_from_cfd.odb : propeller aspicing : Magnitude of Displacements : STEP 1 (AnonymousSTEP1),TIME 4.00000006E-01,



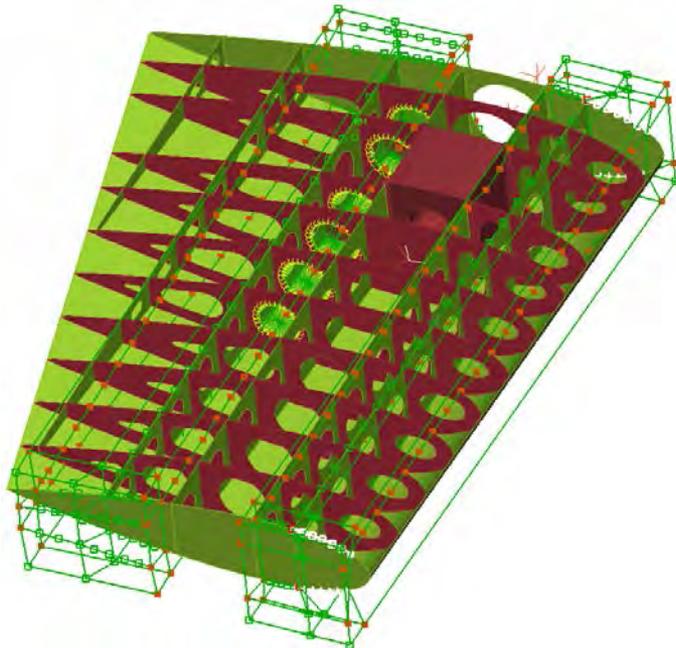
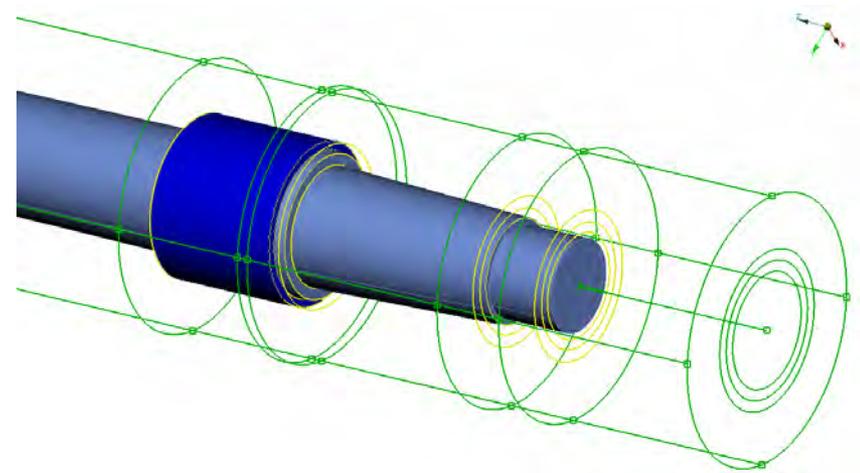
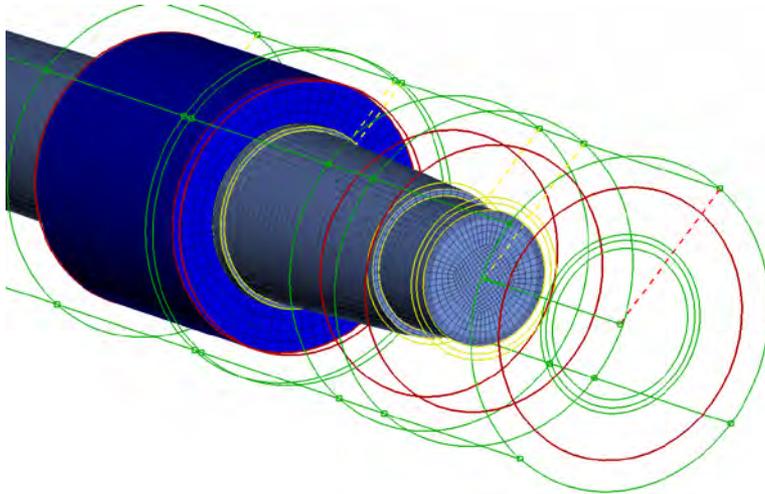
Objectives

- Minimize maximum contact pressure
- Minimize Model mass

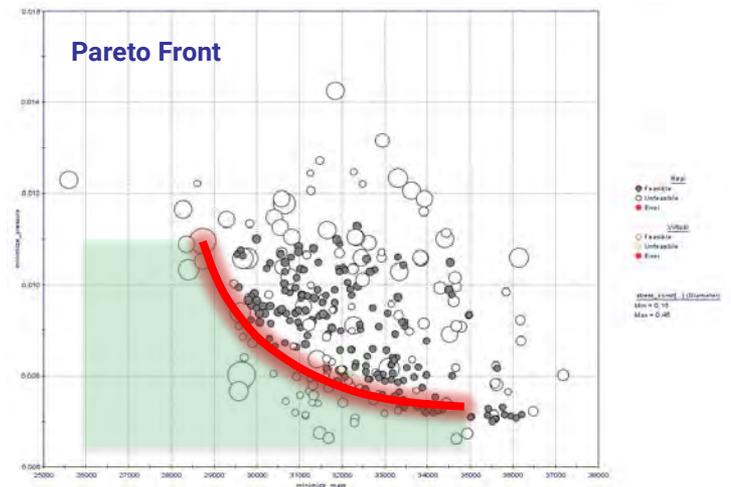
Constraints

- Maximum stresses
- Maximum deflection

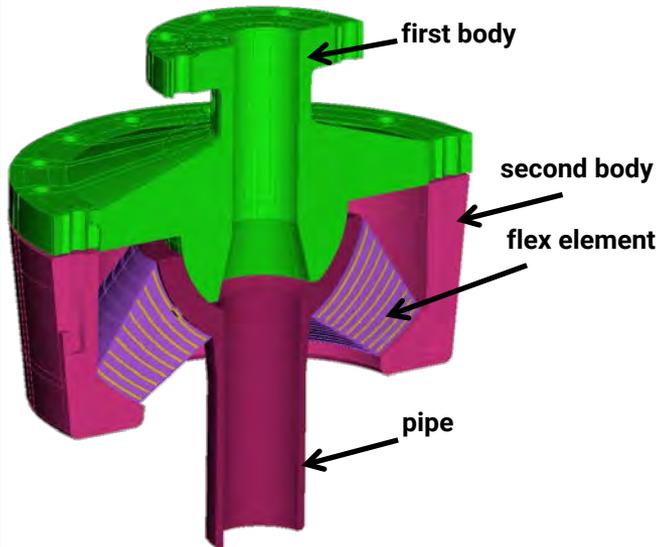
Rudder Optimization Study



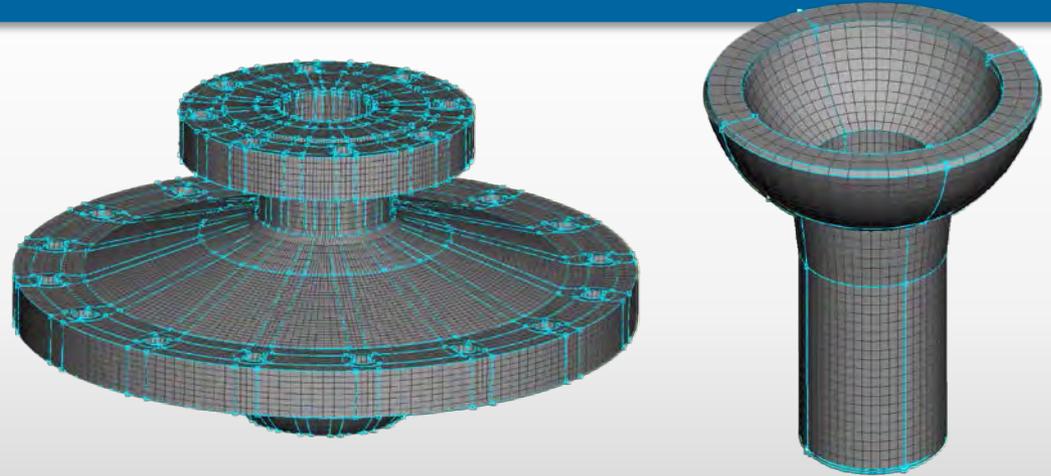
- ✓ **12.5% Maximum pressure reduction**
- ✓ **7.98% Mass reduction**



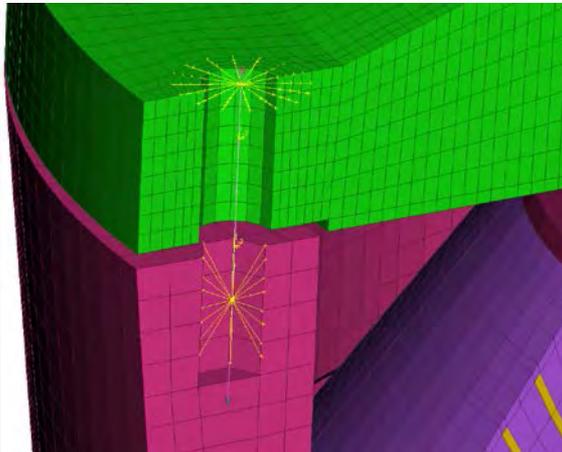
Flex Joint Contact analysis



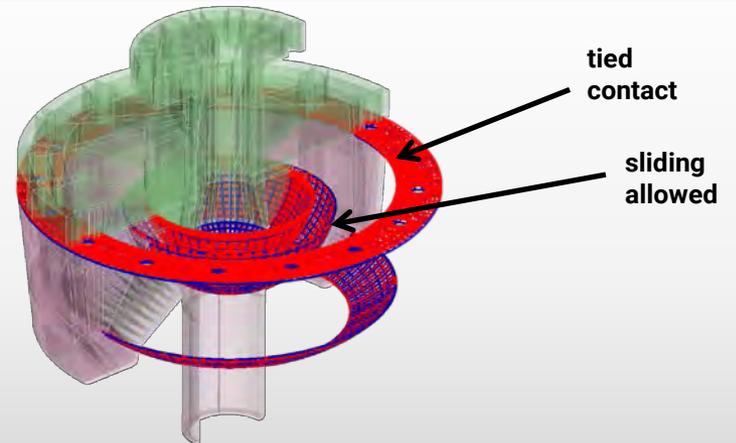
Pure Hexahedral Mesh Creation



Automated Bolts recognition

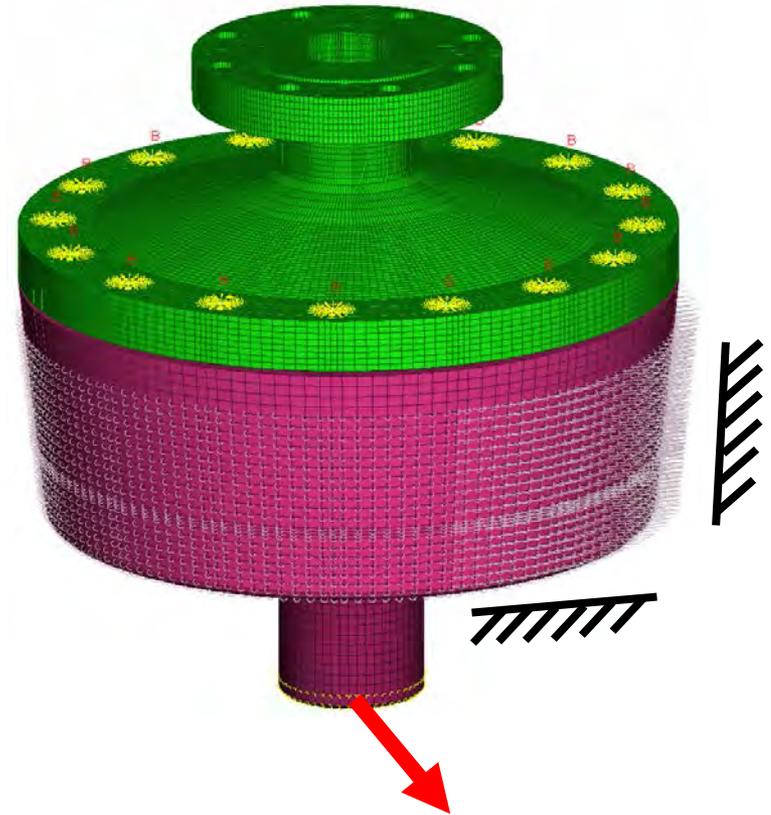
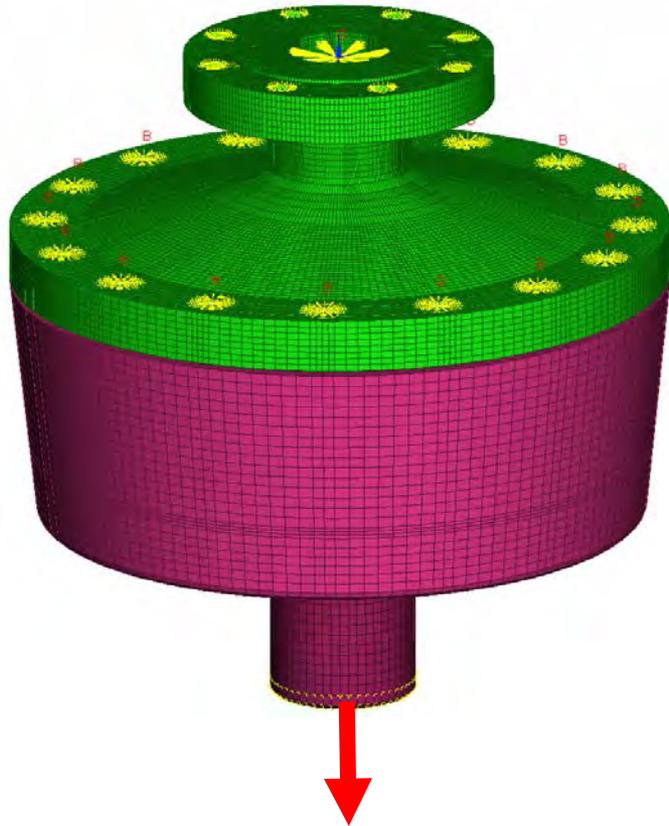


Automated Contact Detection

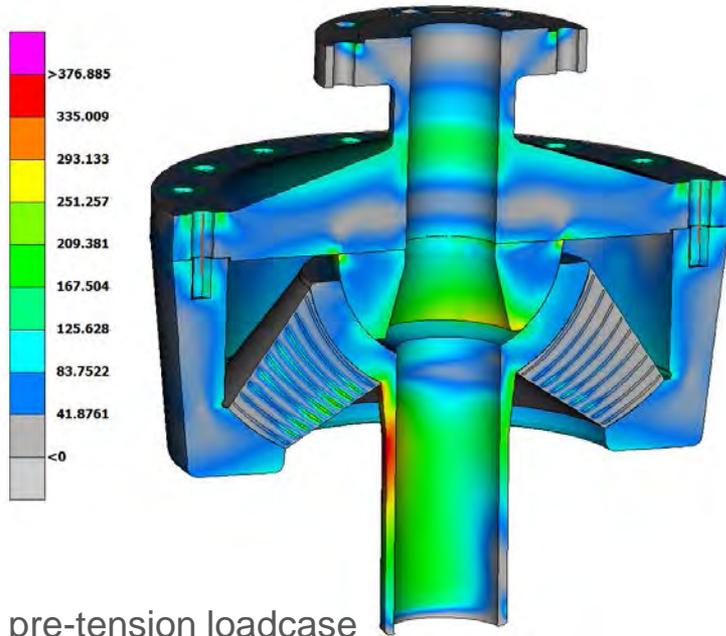


Boundary conditions

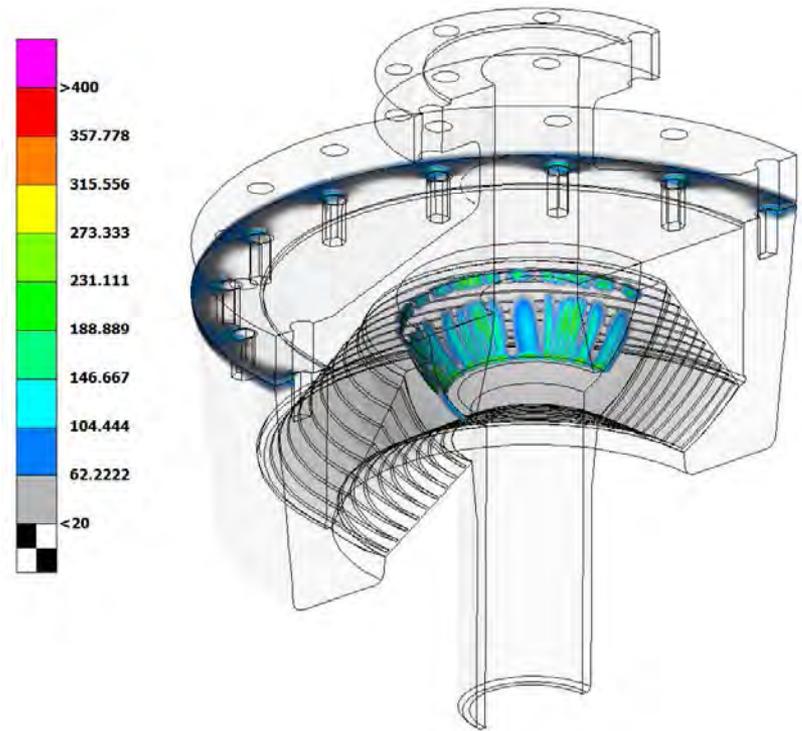
- 1st load case: axial riser force
- 2nd load case: bending riser force



Flex Joint Contact analysis

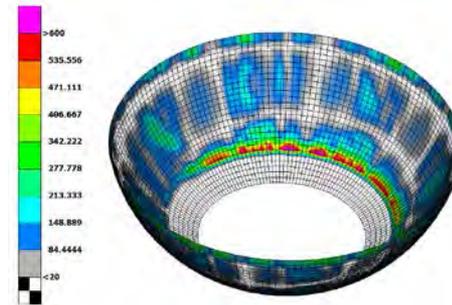
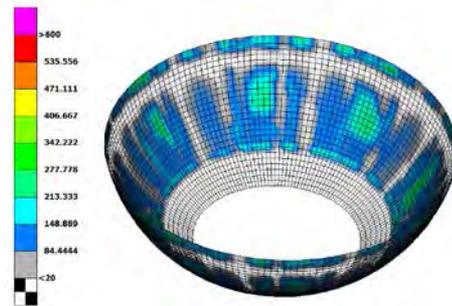


pre-tension loadcase



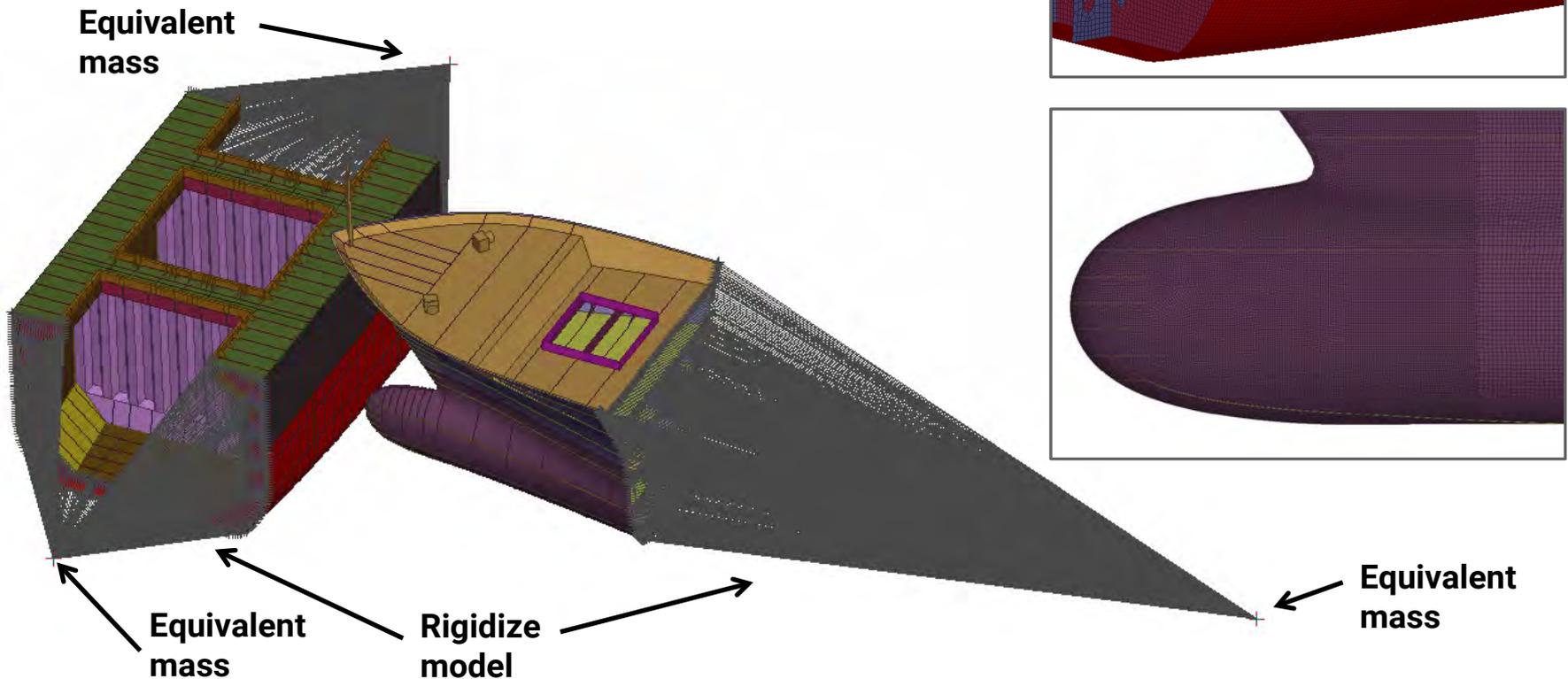
axial force loadcase

bending force loadcase



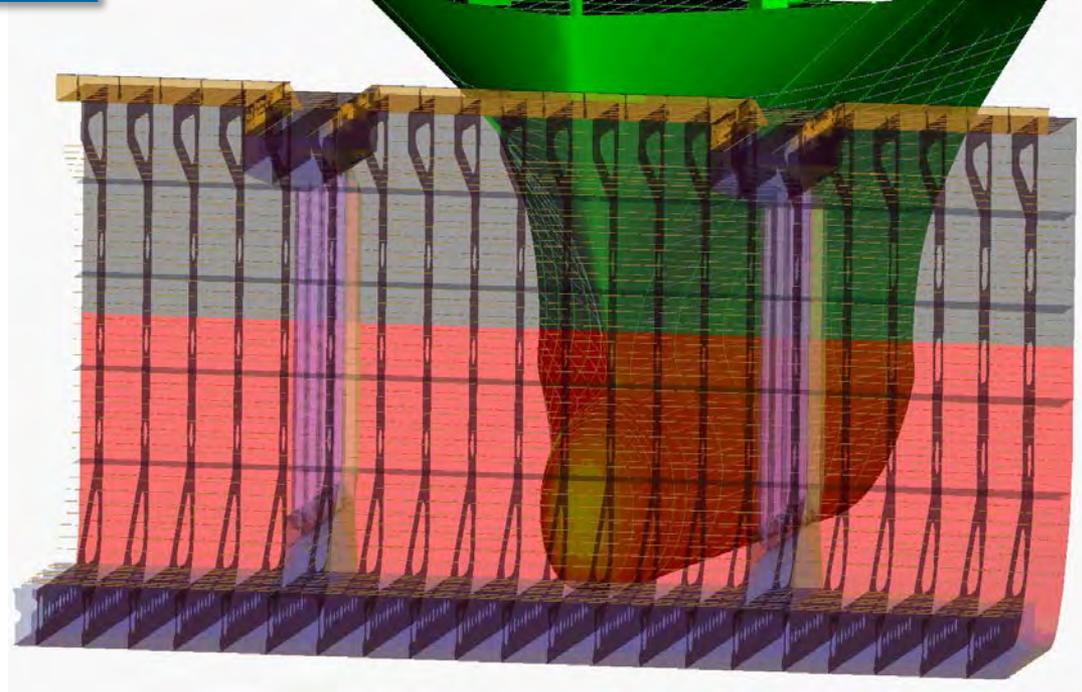
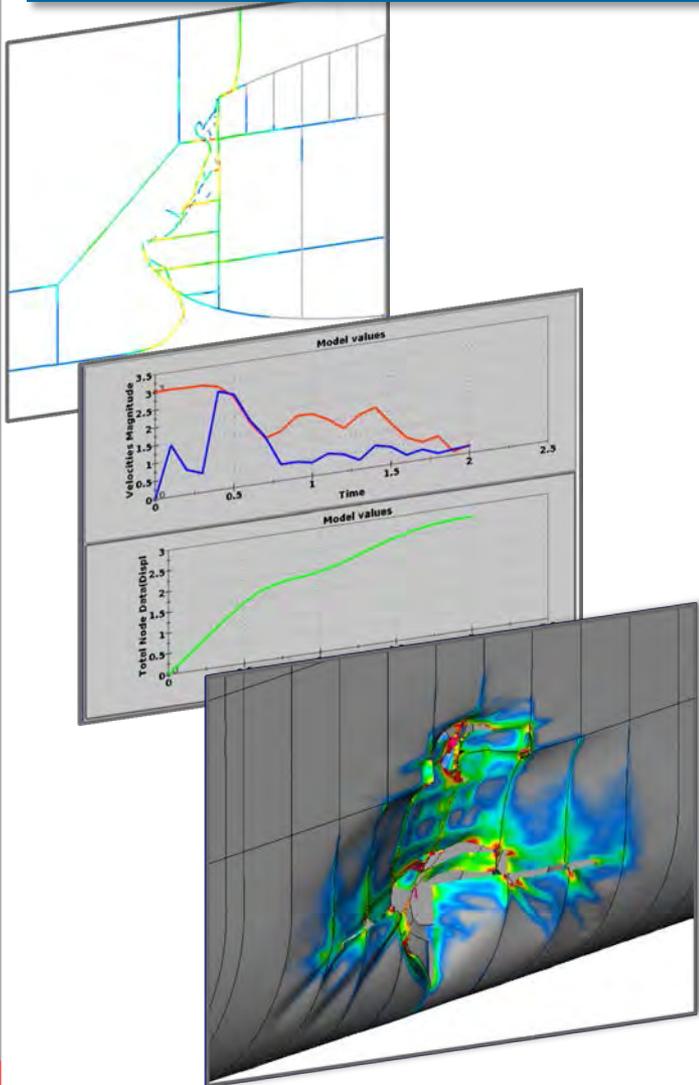
Ship collision analysis

- Replacing part of the model with rigid body and equivalent mass
- Defining boundary conditions and contacts
- Local refinement at the collision area



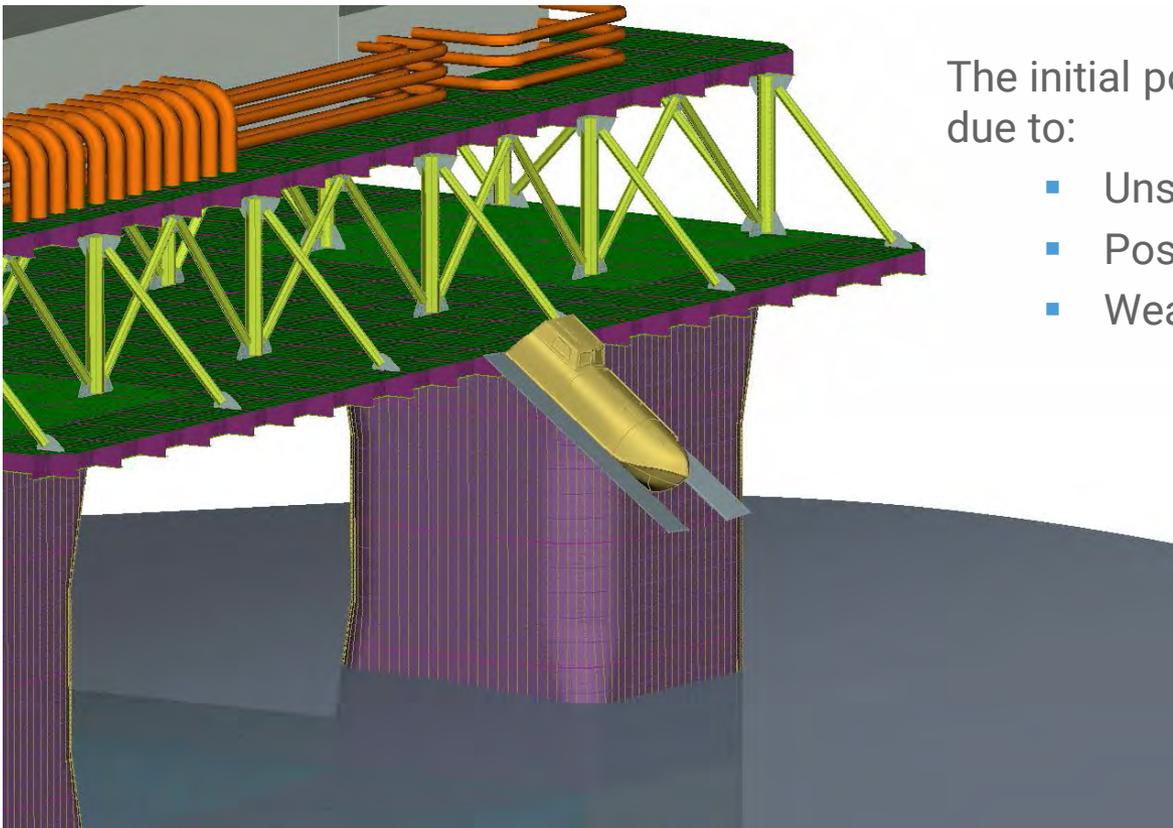
Ship collision analysis

Viewing results with META



Introduction

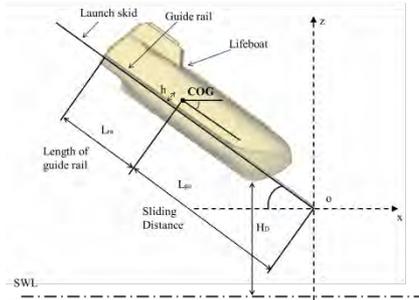
- Free fall lifeboats are found in oil platforms and large transport vessels.
- Carry up to 70 passengers
- Free fall to evacuate as fast as possible
- Safely submerge and surface away from the host structure



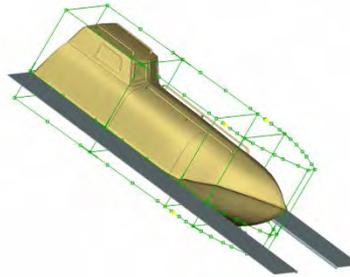
The initial position of the lifeboat is not fixed due to:

- Unstable host structure
- Possible damage in the structure
- Weather conditions variation

Aim Of Optimization



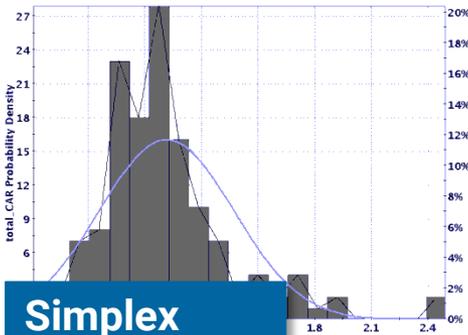
Initial Position



Shape modification

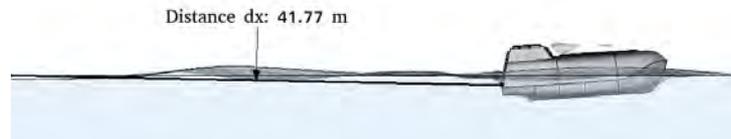
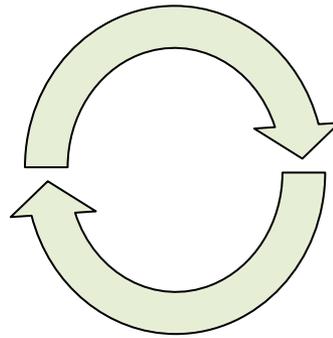


Kinematics Solution Free-fall, Output



Simplex Algorithm

-CAR
-Distance



metaPOST Responses: CAR, Distance

LS-Dyna analysis

*.key

Aim Of Optimization

The aim is to achieve a robust trajectory for the lifeboat that at any circumstances must evacuate the passengers safely

To achieve that, it was needed to:

- ✓ Reduce the accelerations on the passengers
- ✓ Surface as far from the structure as possible (>40m)

Constraints

- Minimum distance from the host structure (40m)
- Maximum CAR* Index value (1)

Objectives

- Minimize the CAR Index value

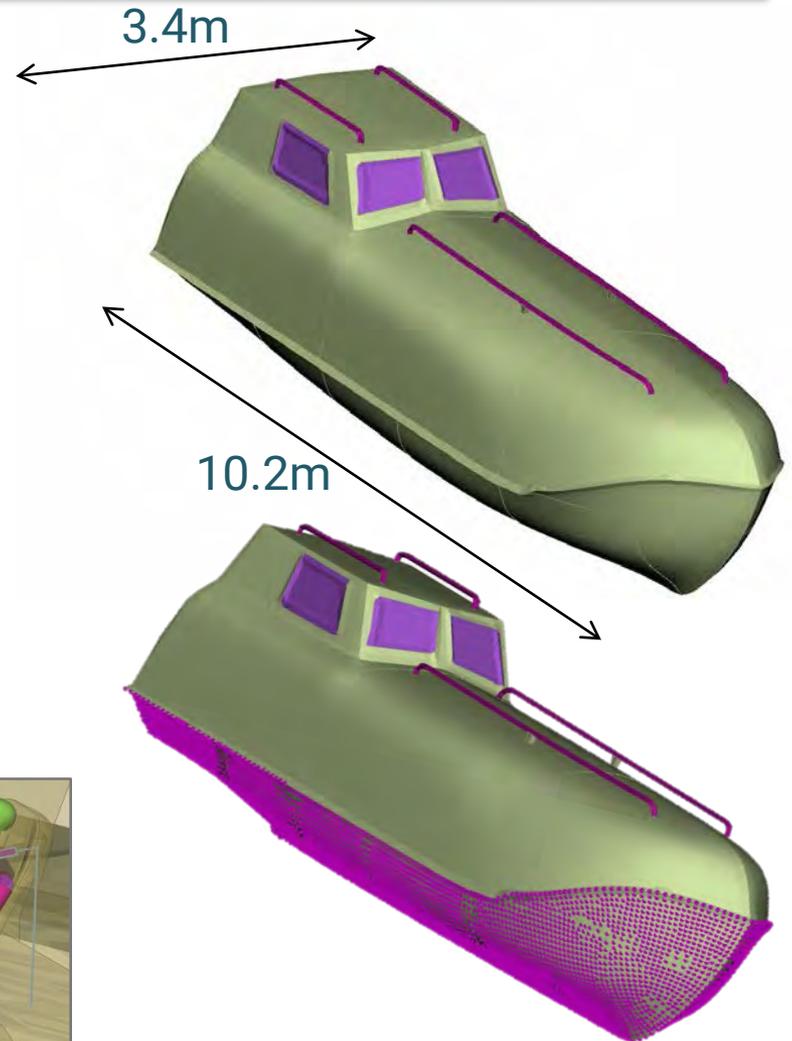
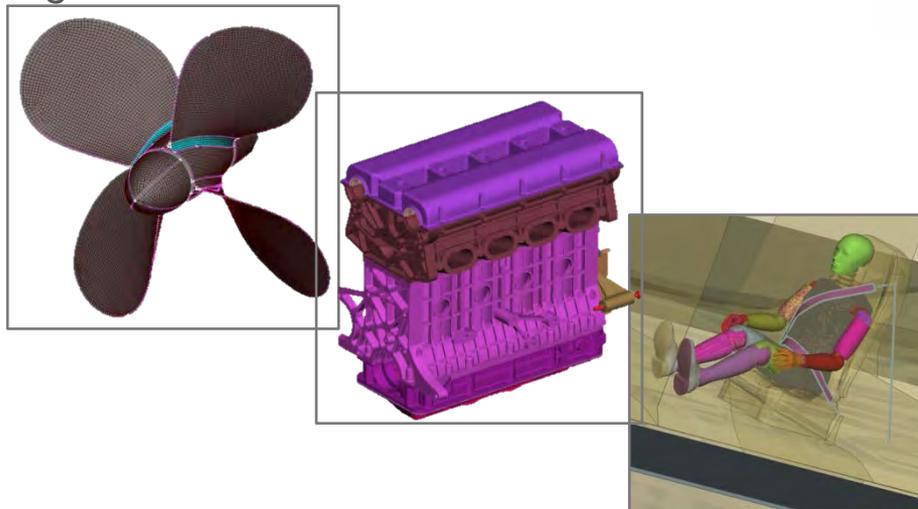
*CAR: Combined Acceleration Ratio

Pre-Processing

FE Model

Lifeboat weight: 9517kg
Capacity: 30 Persons
Material: Glass Fiber Reinforced
Plastic (GFRP)
Length: 10.2m
Width: 3.4m

Added weight of passengers and equipment:
3750kg



Pre-Processing

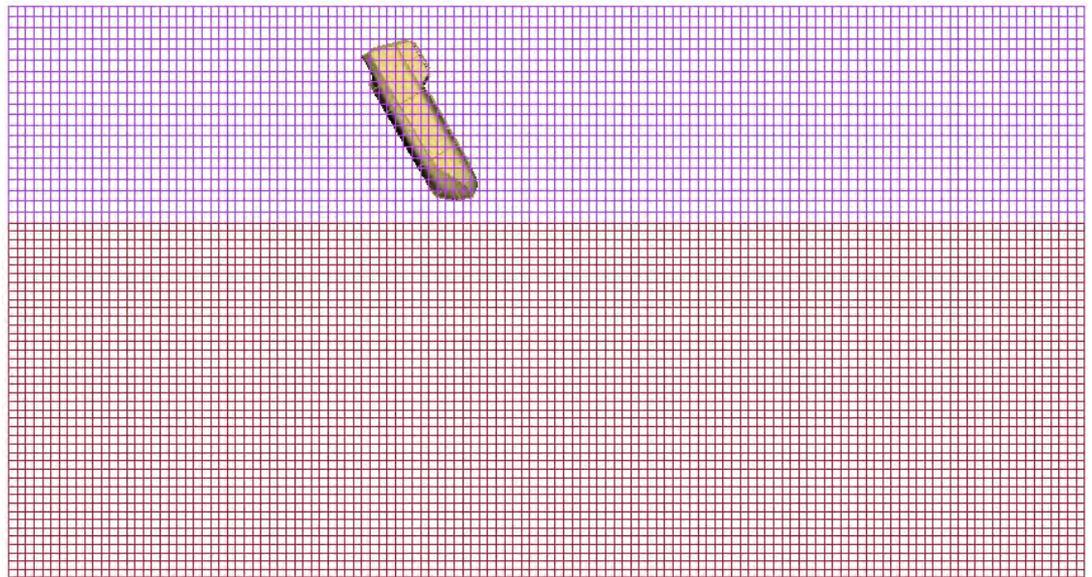
ALE method

The Arbitrary Lagrangian Eulerian method was used to simulate the air and seawater couple using solid HEXA elements

- ✓ The ALE interaction is defined through an LS-DYNA constrained entity that defines the settings of the interaction between the two bodies

Air domain

Water domain

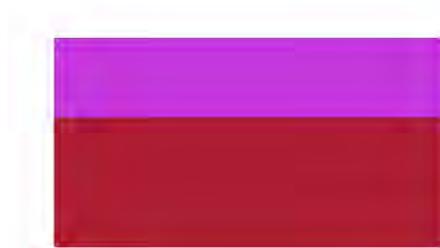


Pre-Processing

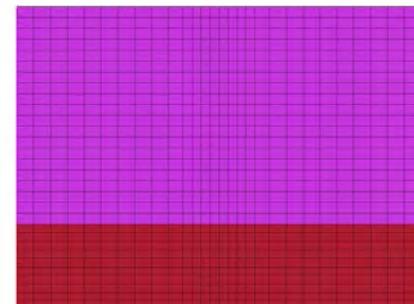
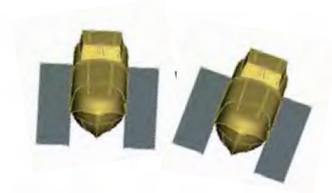
Position Parameters

Taking into account the instability of the host structure, three position parameters were defined using morphing functionality and defined as Stochastic variables for:

✓ Trim rotation



✓ List rotation



✓ Height

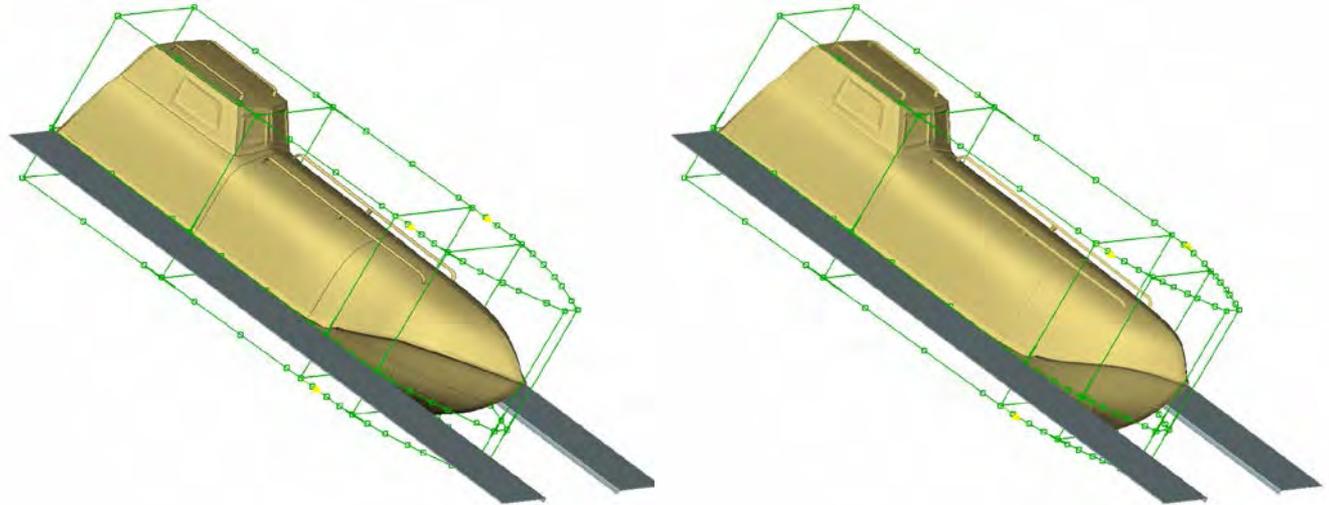


Pre-Processing

Shape Parameters

Two shape parameters were also defined using morphing functionality, modifying the shape of the lifeboat

✓ Nose Shape



✓ Rear Shape

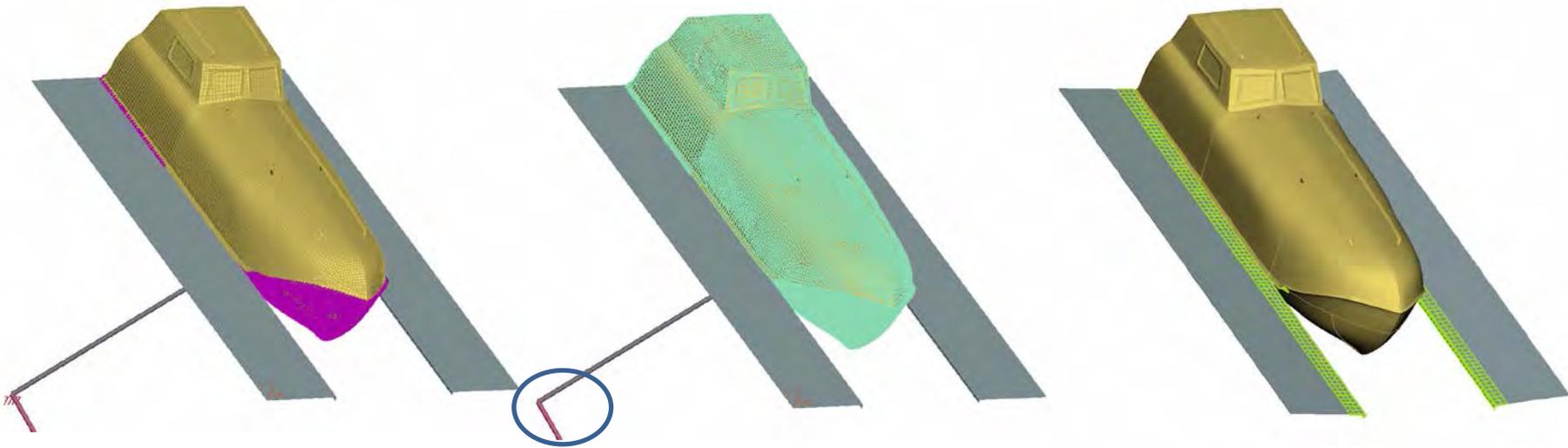


Pre-Processing

Kinematics

A build in Kinematic solver was used for the calculation of the sliding and free fall step of the process

- ✓ Two Kinematic rigid bodies were defined. One for the life boat and one for the launch platform.
- ✓ One Ground point was also necessary to define Kinematic Joints
- ✓ A Kinematic contact pair was defined between the two rigid bodies
- ✓ Friction of 0.15 consistent with nylon blocks use

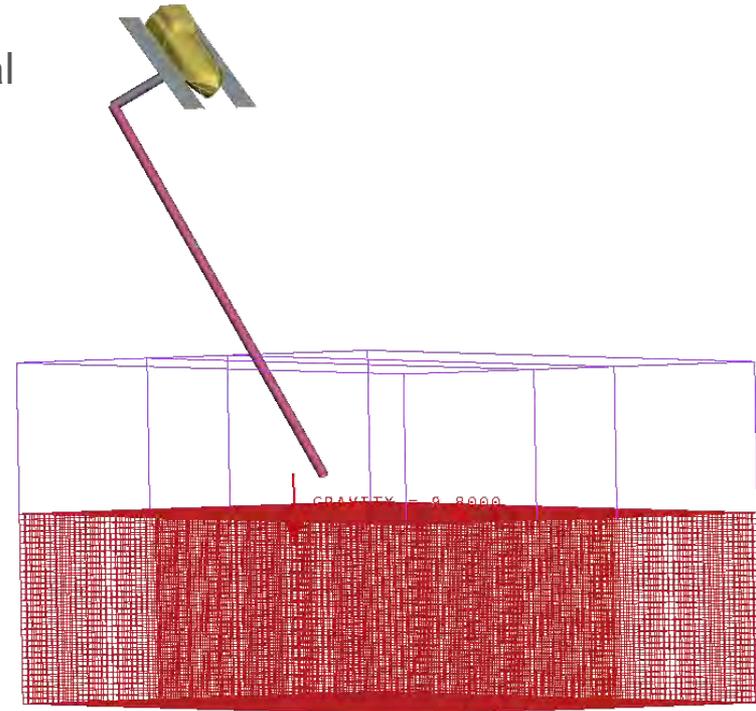


Pre-Processing

Kinematics

A build in Kinematic solver was used for the calculation of the sliding and free fall step of the process

- ✓ With Gravity as the load, the solver calculated the slide and the free fall
- ✓ A Kinematic sensor identifies the distance from the sea level and stops the free fall calculation
- ✓ The calculated velocity vectors are applied as initial conditions



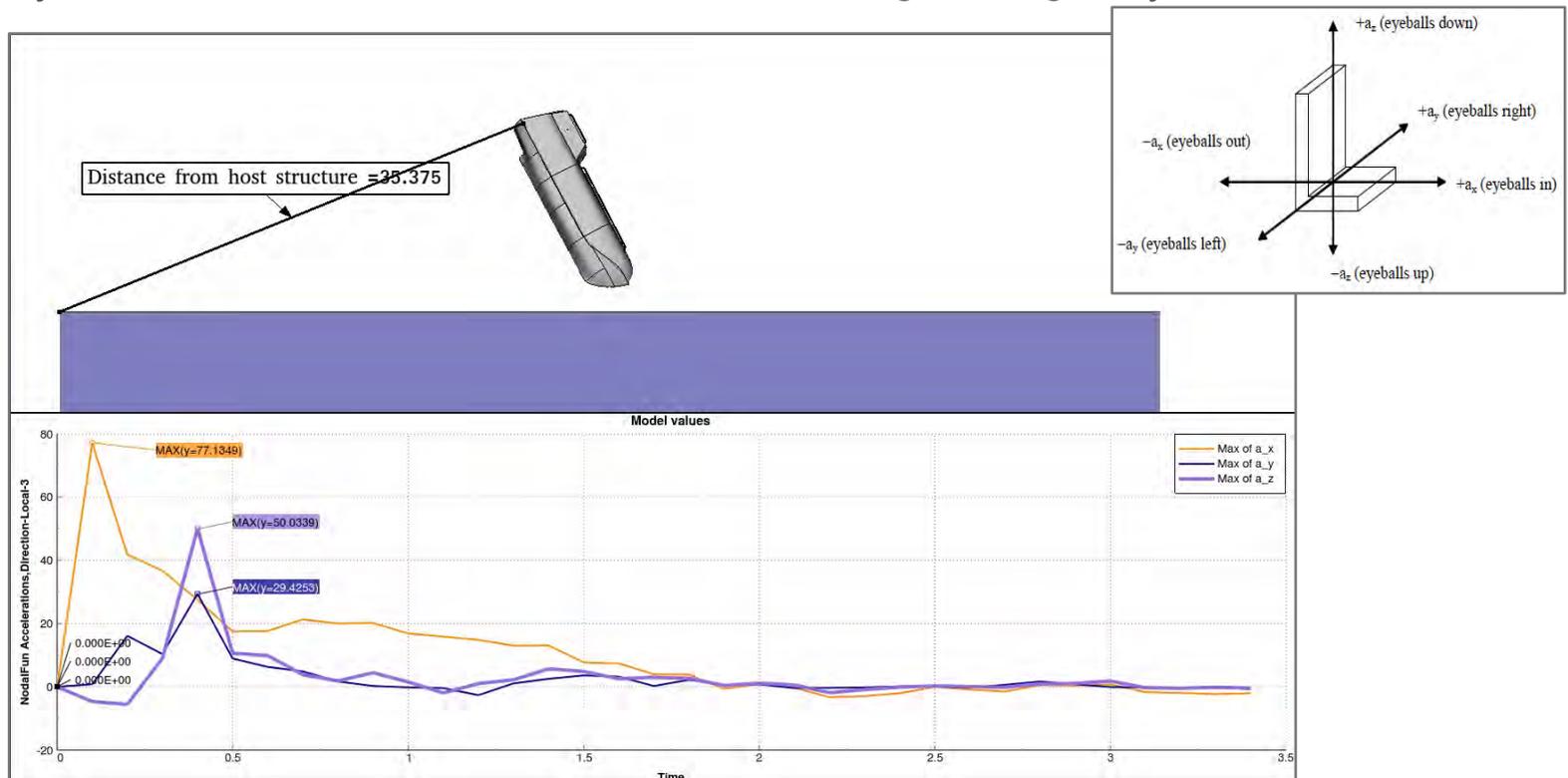
Post Processing

Combined Acceleration Ratio-CAR

The first response acquired at each iteration was the CAR index value, calculated by the nodal accelerations at selected measurement points

$$CAR = \max \sqrt{\left(\frac{a_x}{18g}\right)^2 + \left(\frac{a_y}{7g}\right)^2 + \left(\frac{a_z}{7g}\right)^2}$$

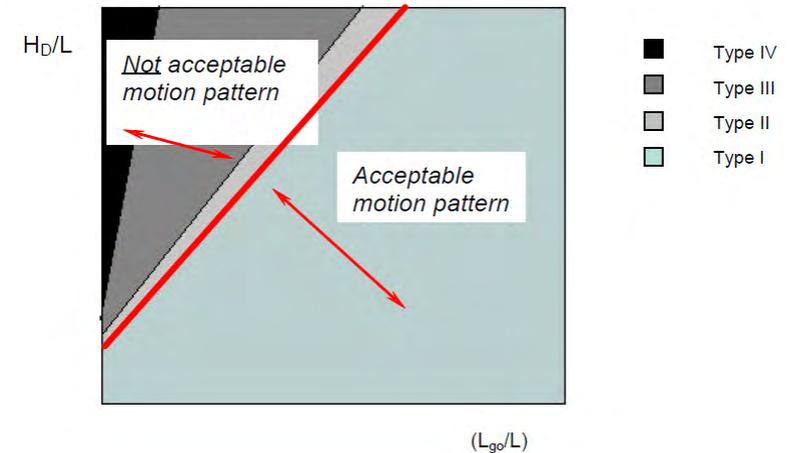
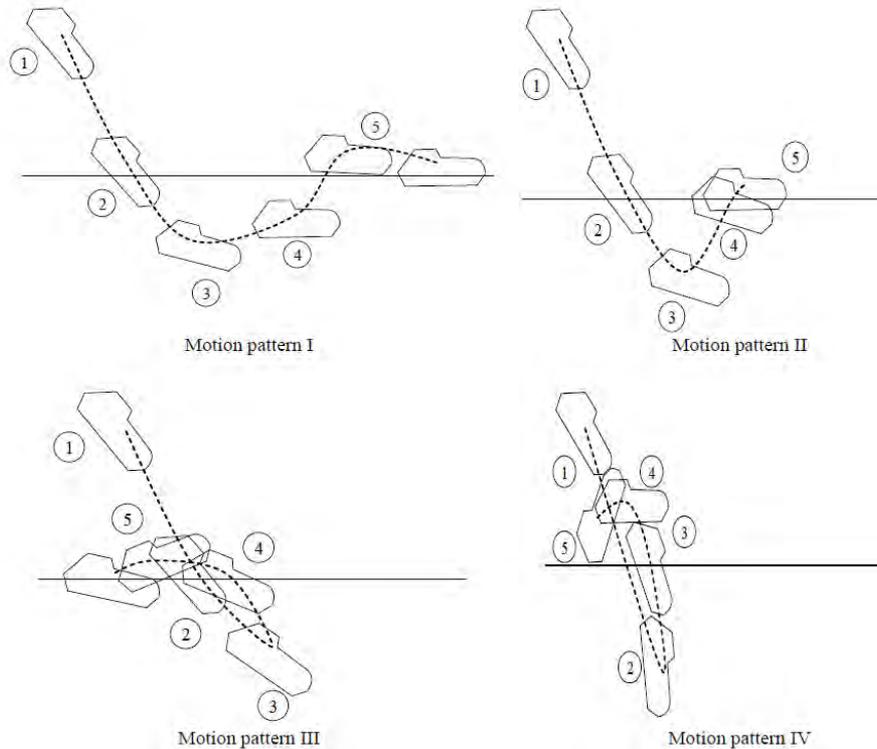
Where a_x , a_y , and a_z are the in-to-seat accelerations and g is the gravity



Post Processing

Distance from host structure- Motion Pattern

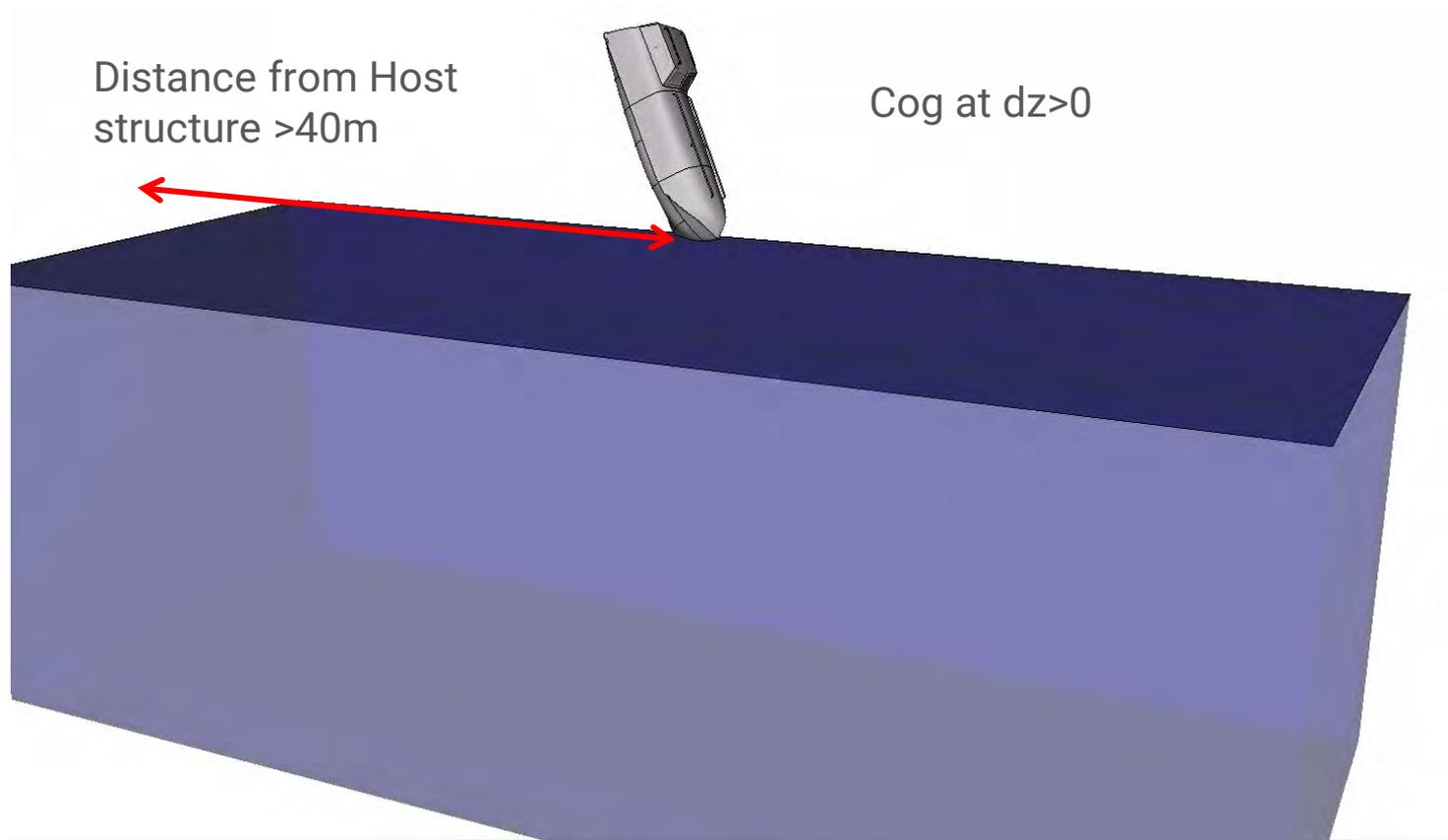
The second response acquired at each iteration was the distance between the host structure and the rear most point of the lifeboat. This measurement was directly dependent on the Motion pattern



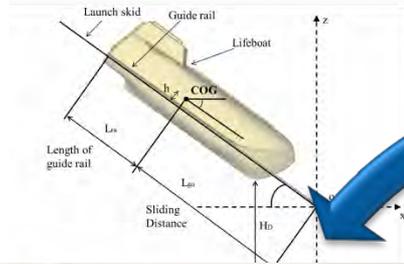
Post Processing

Distance from host structure

The second response acquired at each iteration was the distance between the host structure and the rear most point of the lifeboat



Optimization



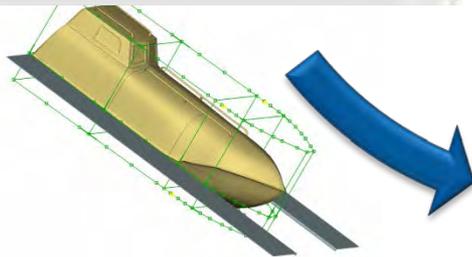
 **modeFRONTIER**



ANSA Optimization Task

Task Manager window showing the optimization task setup:

Item	Type
OPTIMIZATION_TASK_1	Optimization item
./DVfile.txt	DV file
Trim_rotate	Design Variable
Height	Design Variable
list_rotation	Design Variable
sharpen_nose	Design Variable
rear_shape	Design Variable
User Script	User Script
./lifeboat.key	FE_output



LS-DYNA

Results
*d3plot

META OptimizerSetup

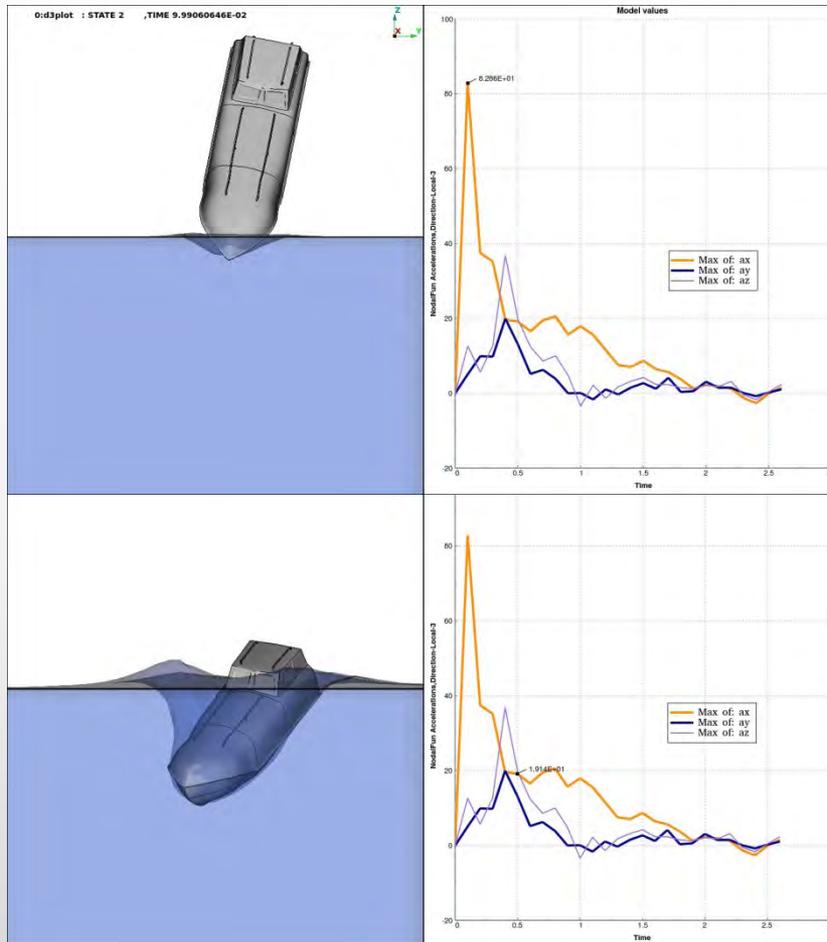
OptimizerSetup dialog box showing configuration options:

- Response Variables
- Responses
- Add
- Print Response Values
- Update Response Values
- Current All
- Remove
- Current All
- Rename Response
- History Variables
- Histories
- Add
- Print History Values
- Update History Values
- Current All
- Remove
- Current All
- Rename History
- Export Session

Response Extraction

Repeated Response acquirement

Using an automated post-processing process, the two responses are acquired at each iteration and the results are fed to the optimizer software



The OptimizerSetup dialog box is shown, displaying the configuration for response variables and history variables. The Response Variables section includes a dropdown menu for Responses, an Add button, and buttons for Print Response Values, Update Response Values (Current, All), Remove (Current, All), and Rename Response (set to CAR = 0.8). The History Variables section includes a dropdown menu for Histories, an Add button, and buttons for Print History Values, Update History Values (Current, All), Remove (Current, All), and Rename History. An Export Session button is located at the bottom.

Response Extraction

Repeated Response acquirement

Using an automated post-processing process, the two responses are acquired at each iteration and the results are fed to the optimizer software

Distance dx: 40.464m

Distance dz: 4.405

OptimizerSetup

Response Variables

Responses

Distance dx

Add

Print Response Values

Update Response Values

Current All

Remove

Current All

Rename Response

History Variables

Histories

Add

Print History Values

Update History Values

Current All

Remove

Current All

Rename History

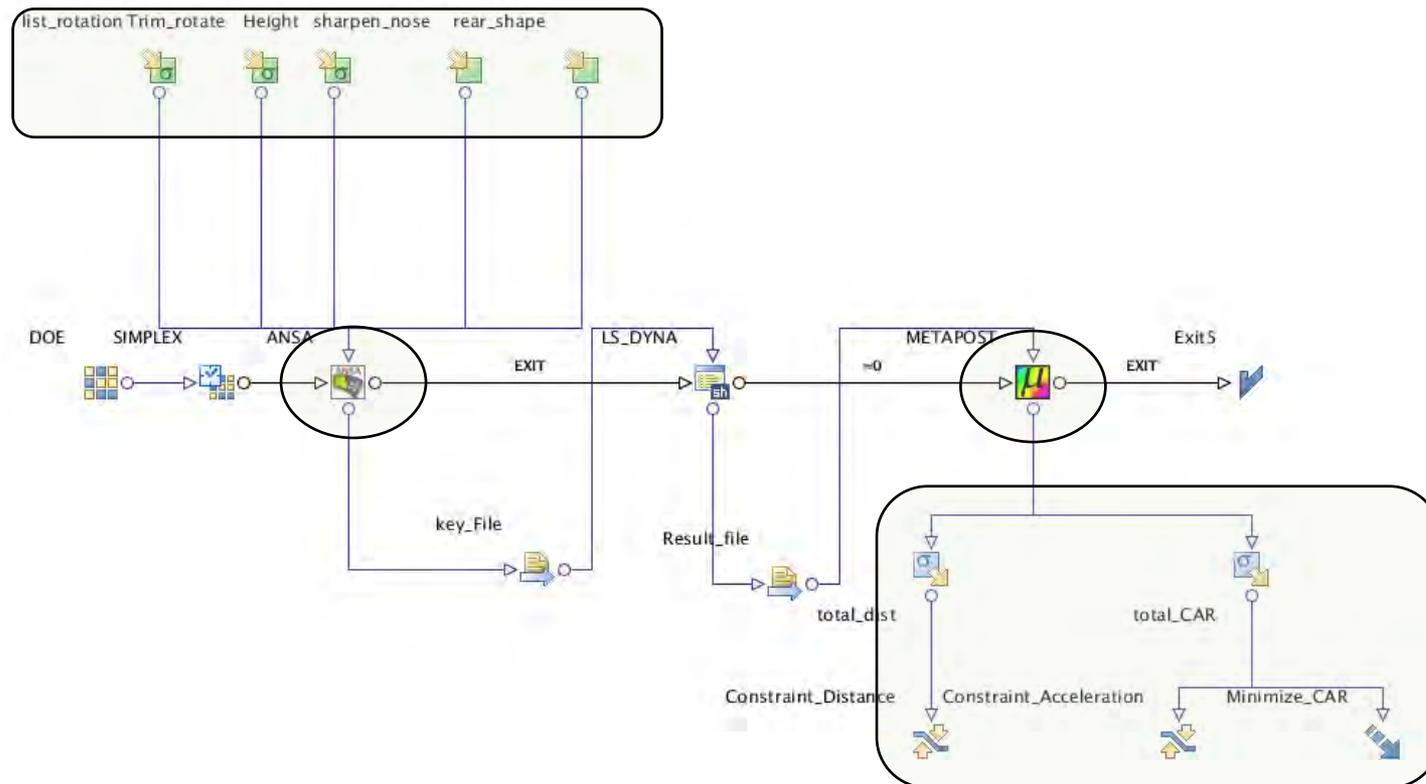
Export Session

Optimization process

Robustness Optimization

modeFRONTIER was used in this study, to couple ANSA, Meta and the FSI solver.

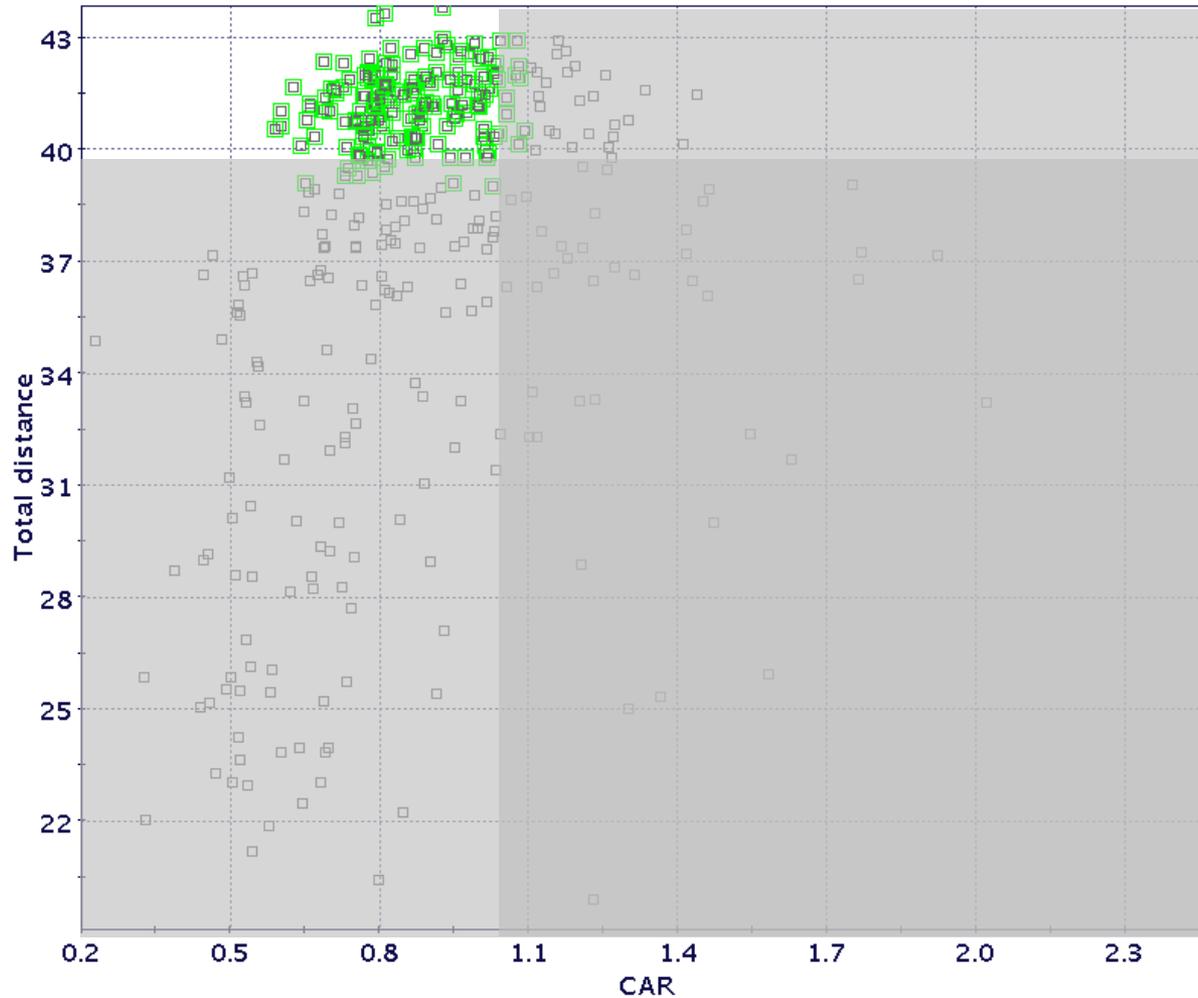
- ✓ Built-in ANSA & mETA Nodes
- ✓ Stochastic values for the three position input variables
- ✓ Constraint and Objective Stochastic responses



Results

Scatter Chart – Feasible designs

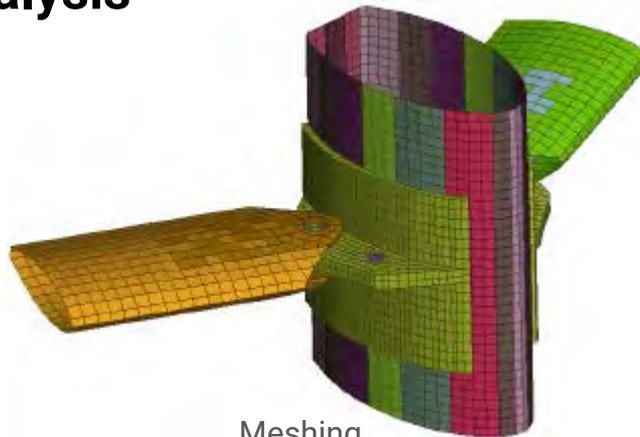
Feasible designs were close to 50% of the total of 400



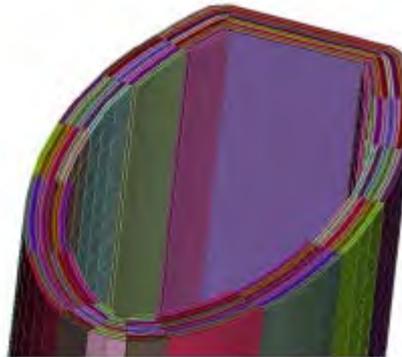
Safran Open 60' race yacht composite mast modelling for crashworthiness analysis



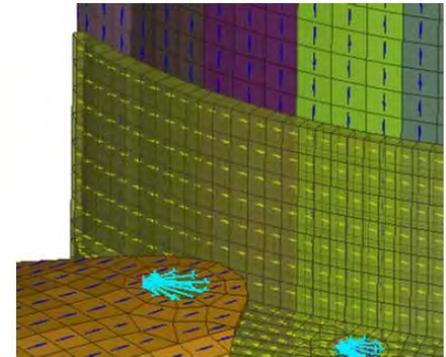
Mast modelling



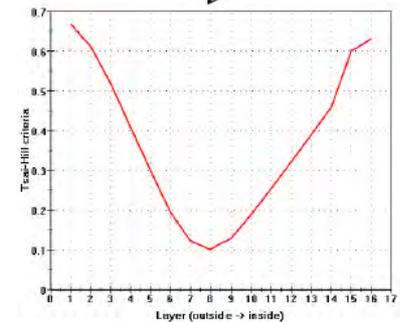
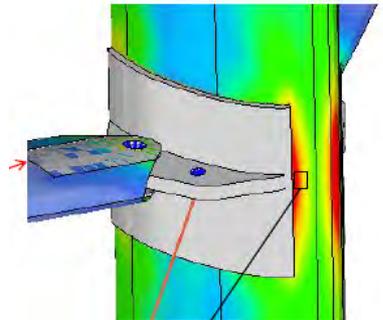
Meshing



Composite materials



Elements orientation

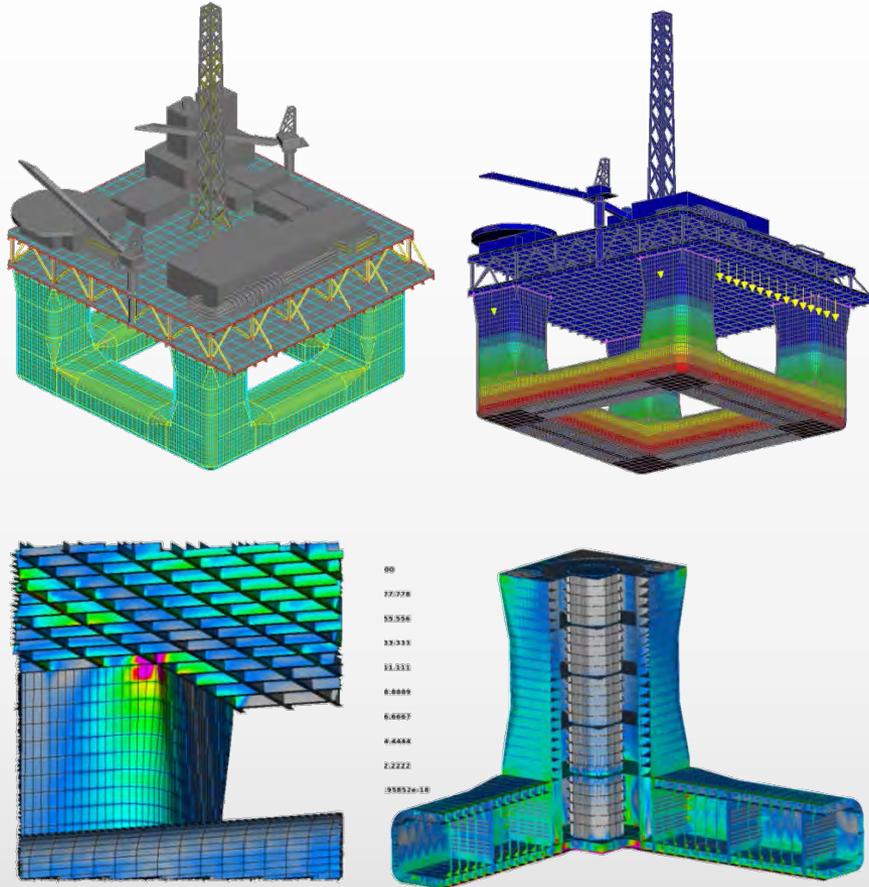


ANSA and META contribution to the study of Safran Open 60' race yacht crashworthiness
Philippe Biagi, Safran Engineering Services, 4th ANSA & META International Conference, 2011

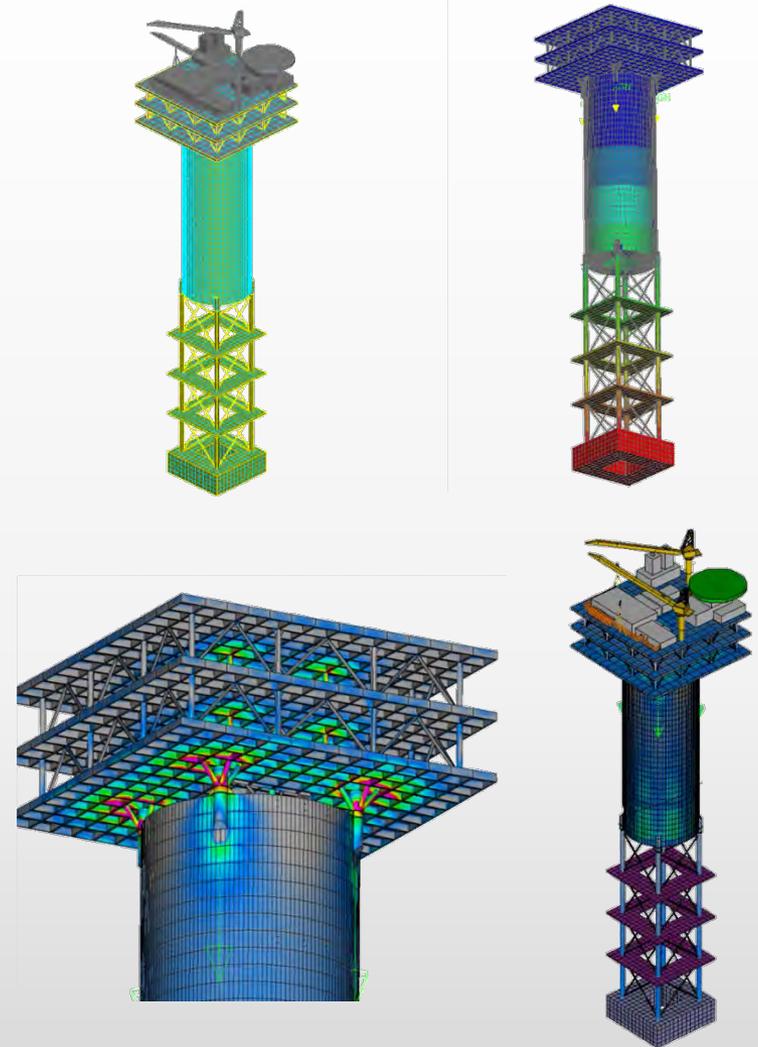


Static Analysis for Offshore Models

Semi Submersible Platform

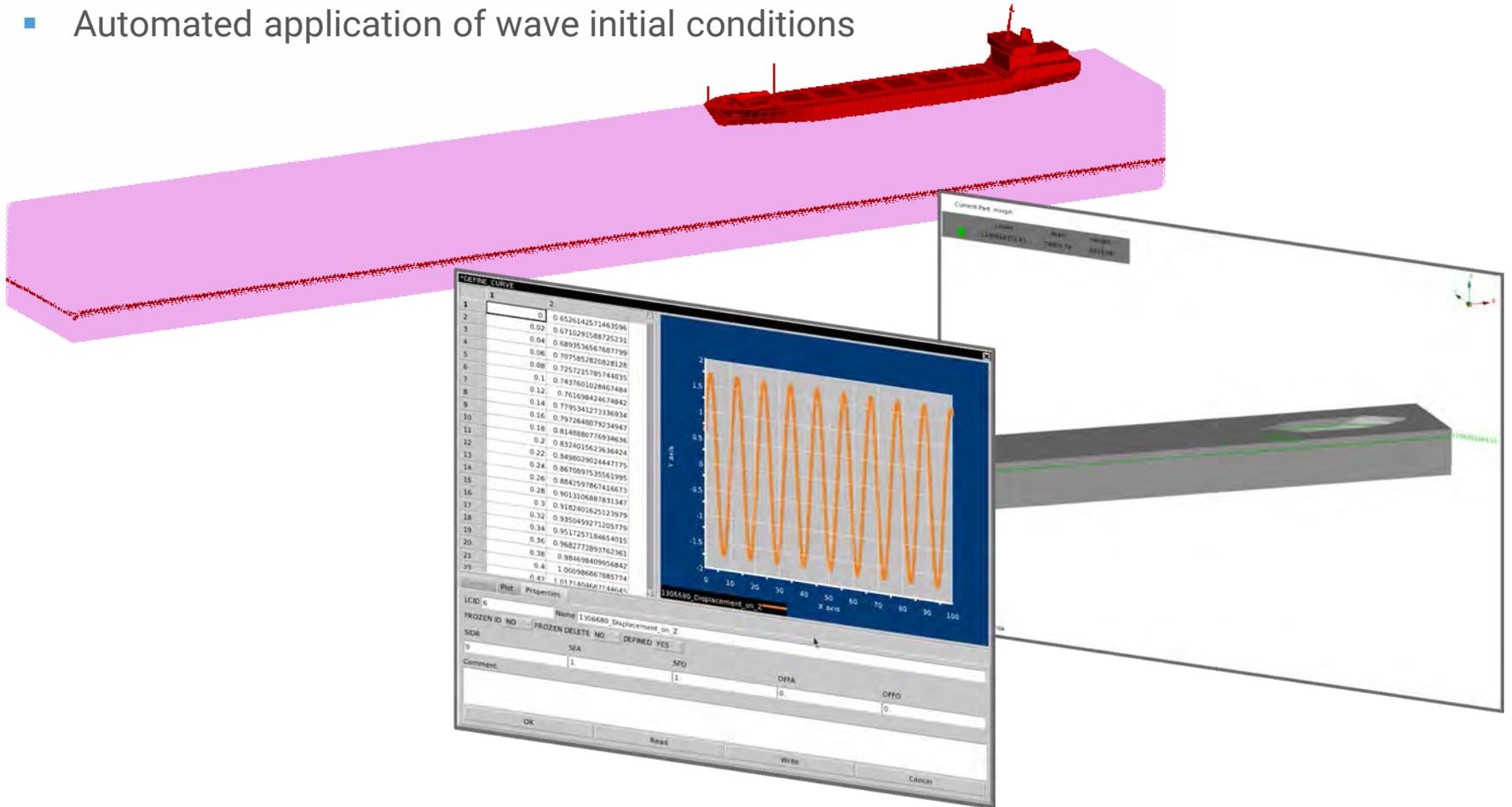


Spar Platform

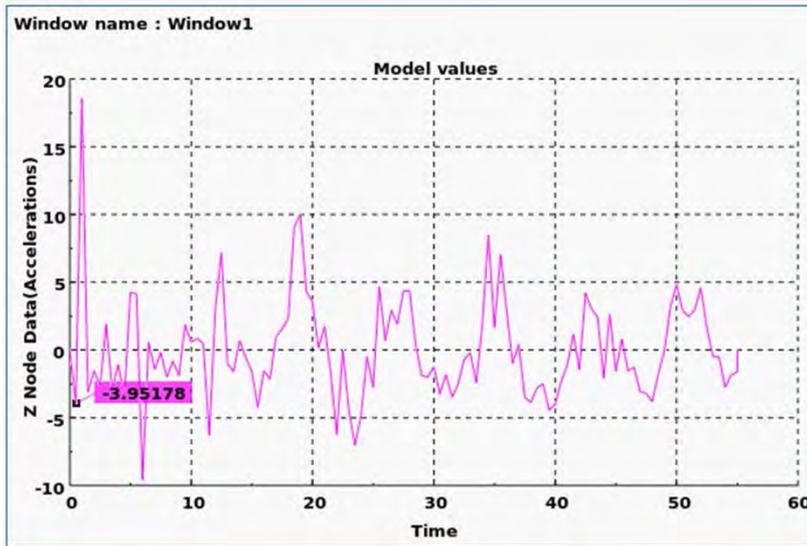


Wave on SPH case study

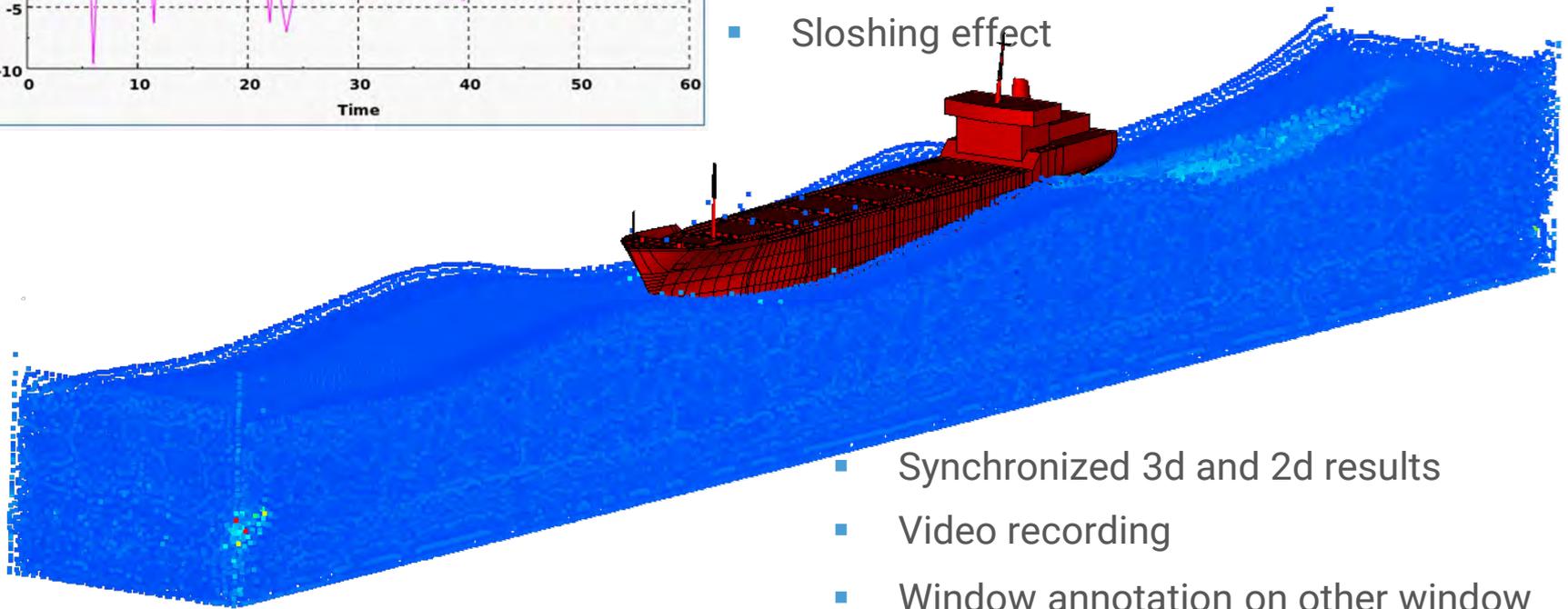
- Volume detection
- Automated SPH creation
- Automated application of wave initial conditions



Results in META



- Accelerations
- Vessels Stability
- Contact pressure and forces
- Hydrodynamic drag
- Stresses
- Sloshing effect



Thank you

Stay connected

website

www.beta-cae.com

support

ansa@beta-cae.com

social media

