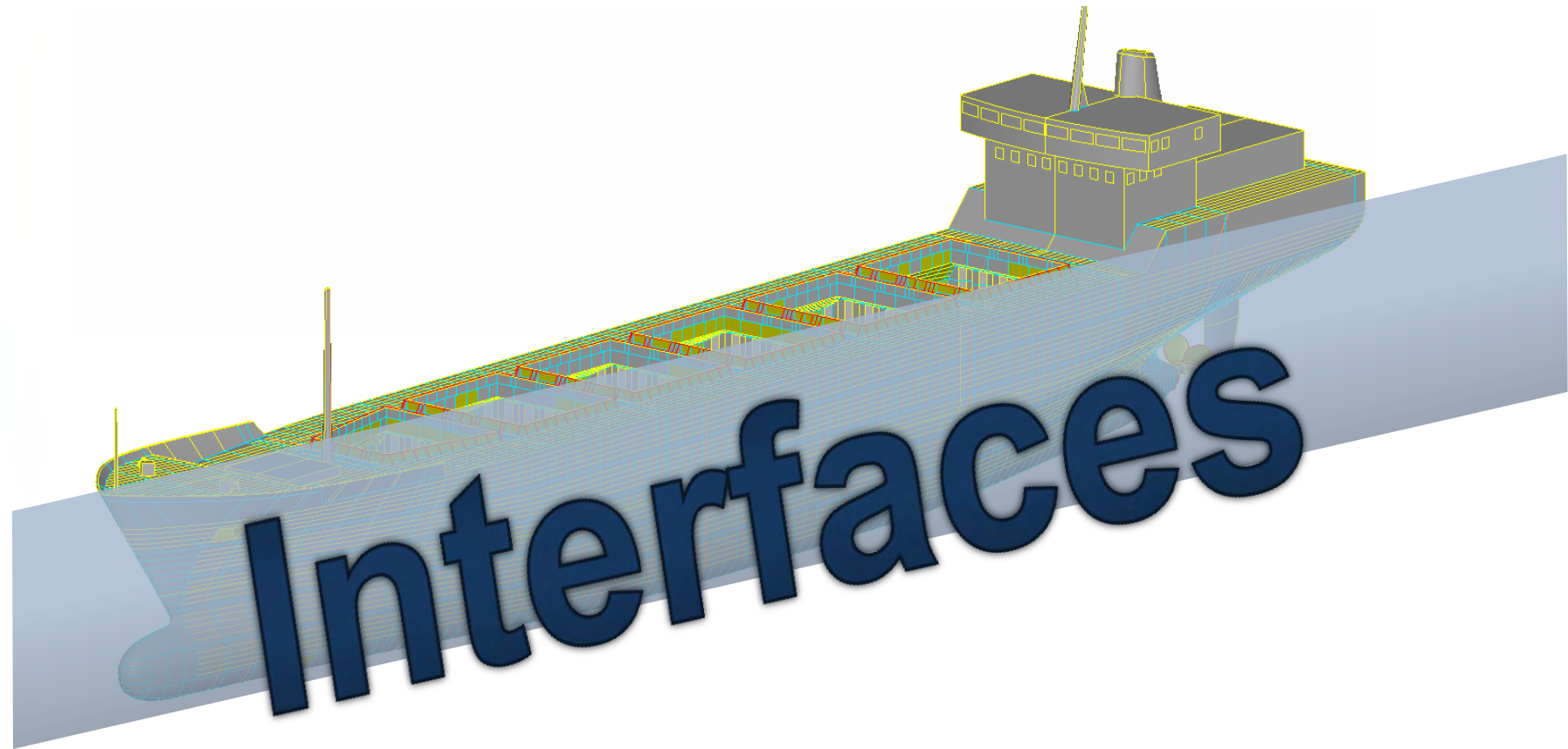
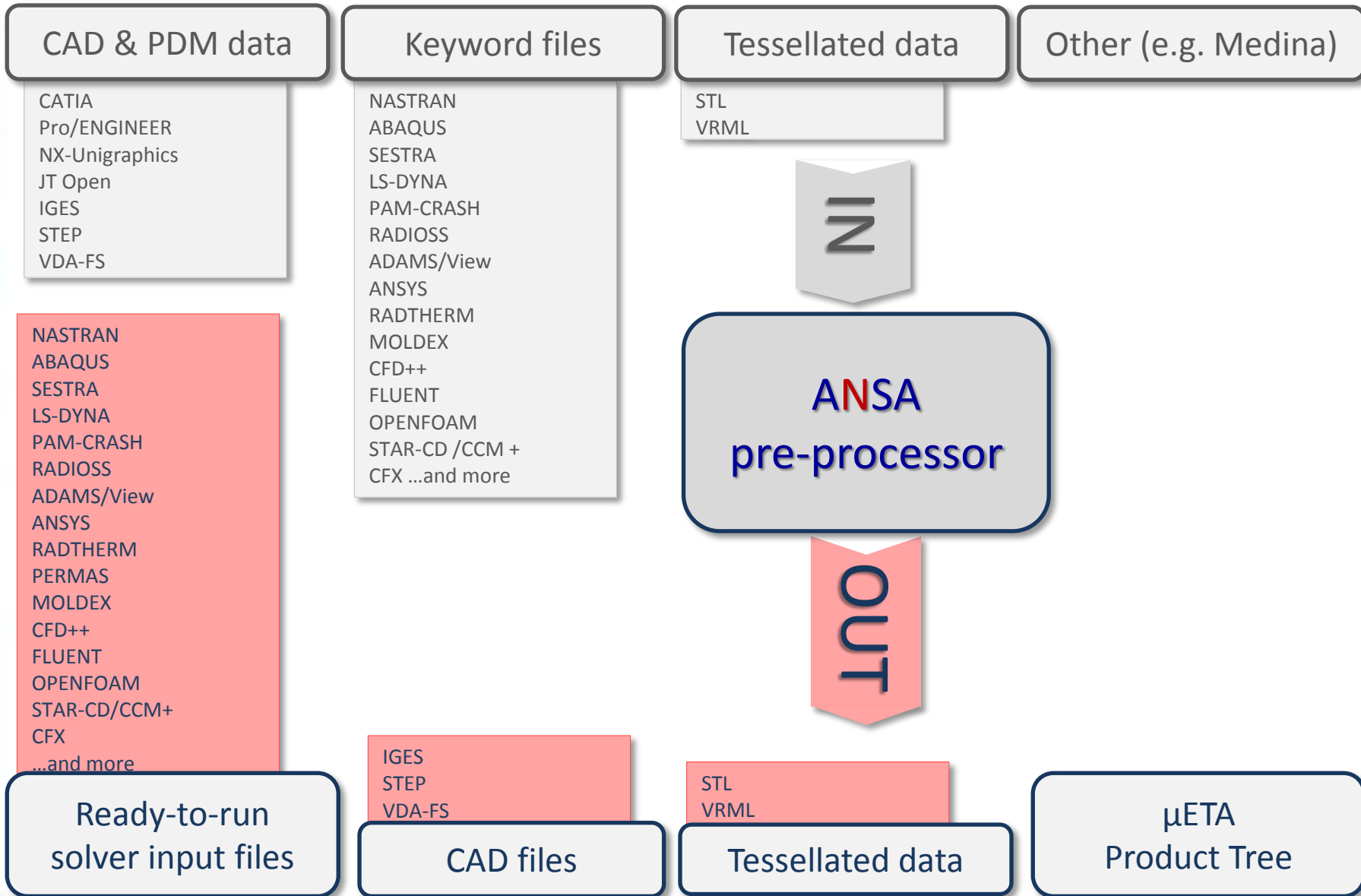


# **ANSA and $\mu$ ETA for the Maritime and Offshore structures analysis**

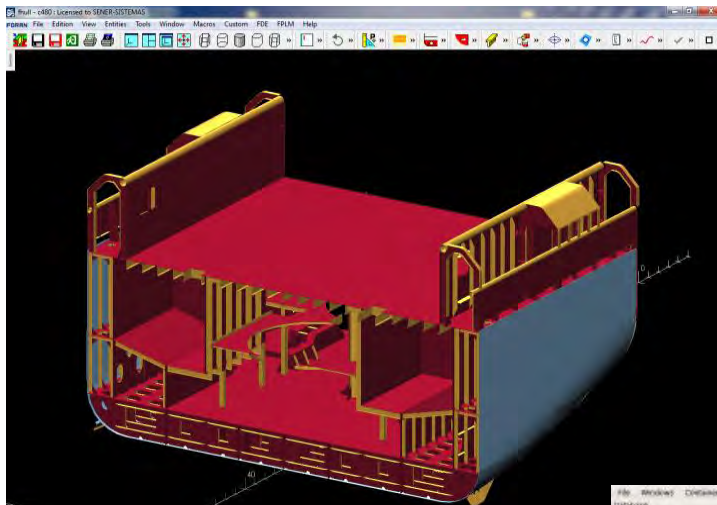


# Interfaces



# 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



FORAN 3D model, courtesy of SENER

```

import anttree ElementTree as ET
import os
from ANSA import base
from ANSA import constants
from collections import defaultdict

class cog:
    def __init__(self, x, y, z):
        self.x = x
        self.y = y
        self.z = z

    def print(self):
        print('cog: %s, %s, %s' % (self.x, self.y, self.z))

class wall:
    def __init__(self, red, green, blue):
        self.red = red
        self.green = green
        self.blue = blue

    def print(self):
        print('wall: %s, %s, %s' % (self.red, self.green, self.blue))

class strip:
    def __init__(self, material):
        self.material = material

    def print(self):
        print('strip: %s' % self.material)

class plate:
    def __init__(self, material, thickness):
        self.material = material
        self.thickness = thickness

    def print(self):
        print('plate: %s, %s' % (self.material, self.thickness))

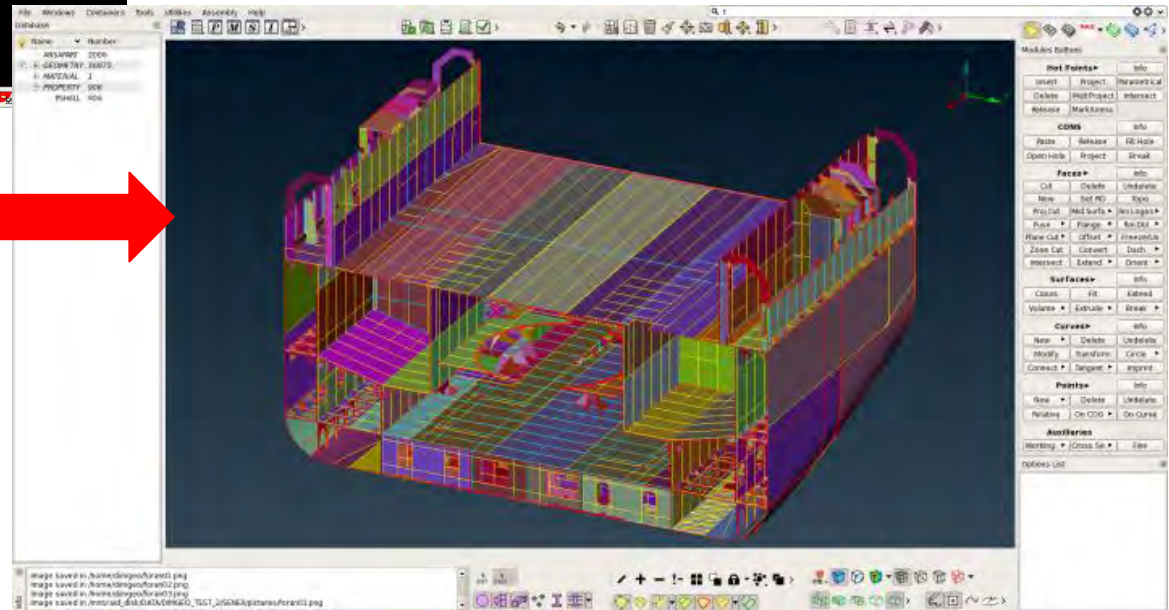
class profile:
    def __init__(self, material, scanning, inertia_x, inertia_y, inertia_z, section_area):
        self.material = material
        self.scanning = scanning
        self.inertia_x = inertia_x
        self.inertia_y = inertia_y
        self.inertia_z = inertia_z
        self.section_area = section_area

    def print(self):
        print('profile: %s, %s, %s, %s, %s, %s' % (self.material, self.scanning, self.inertia_x, self.inertia_y, self.inertia_z, self.section_area))

class part:
    def __init__(self, name, material, scanning, inertia_x, inertia_y, inertia_z, section_area):
        self.name = name
        self.material = material
        self.scanning = scanning
        self.inertia_x = inertia_x
        self.inertia_y = inertia_y
        self.inertia_z = inertia_z
        self.section_area = section_area

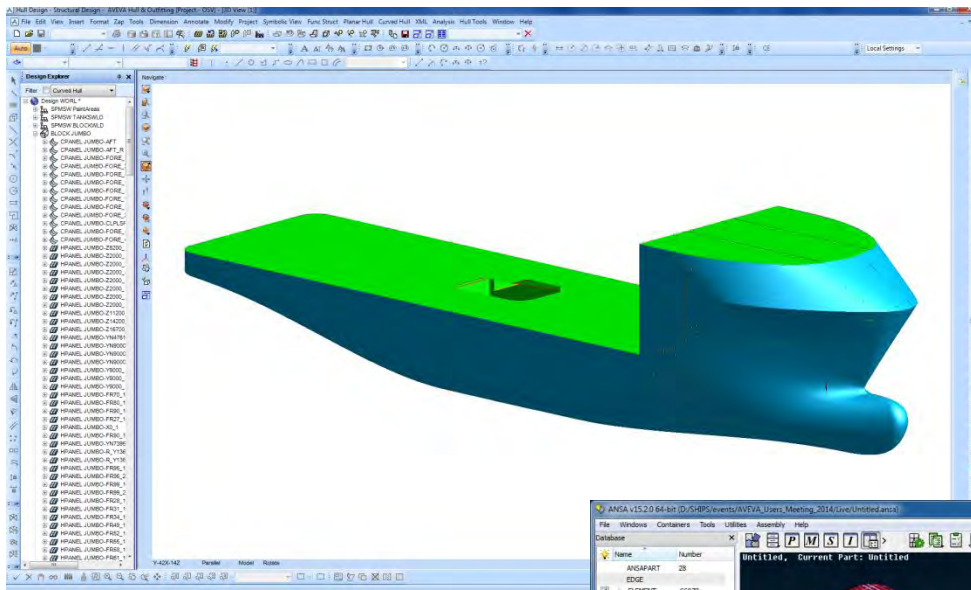
    def print(self):
        print('part: %s, %s, %s, %s, %s, %s' % (self.name, self.material, self.scanning, self.inertia_x, self.inertia_y, self.inertia_z))

```



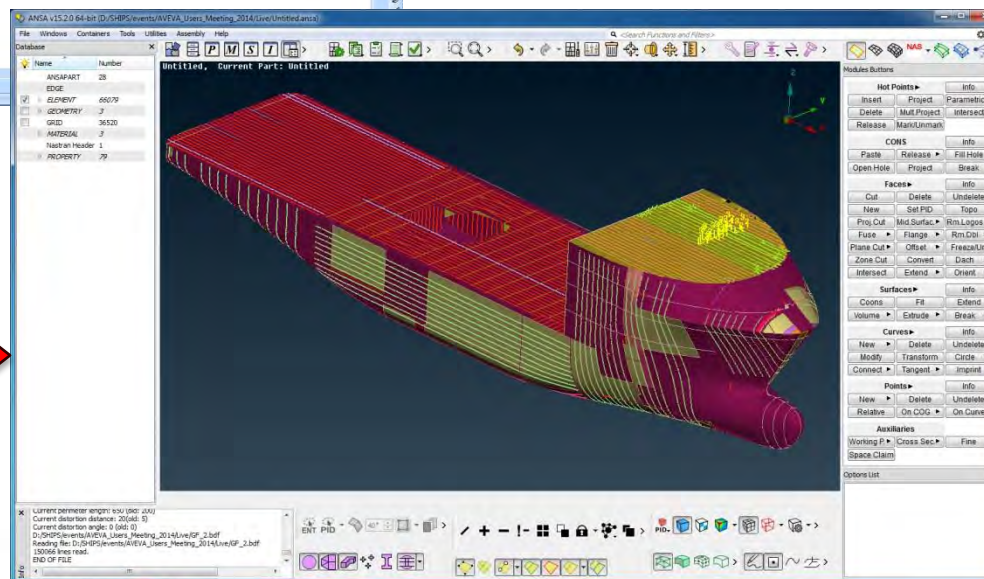
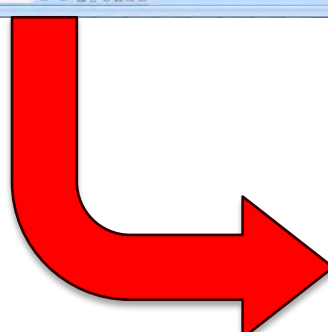
# From AVEVA Marine to ANSA

Additional information is applied on the FE model by reading an XML file produced by AVEVA



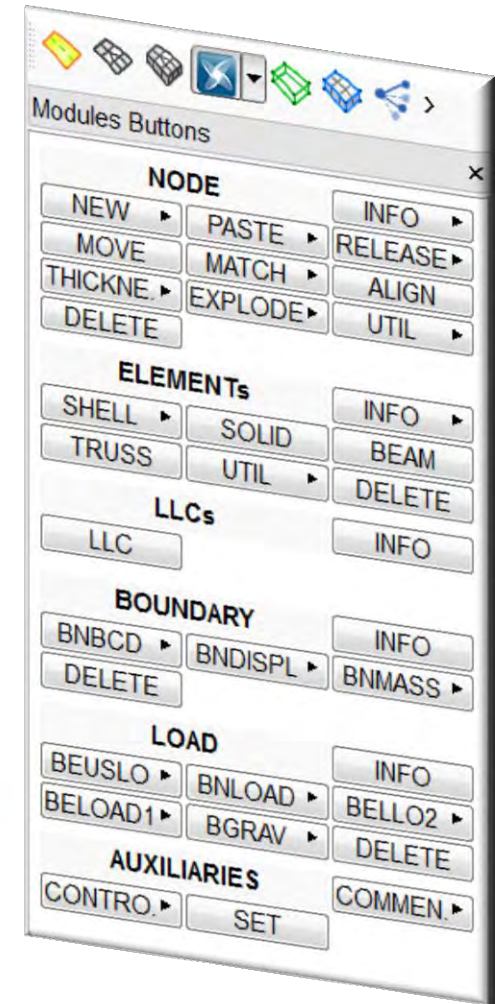
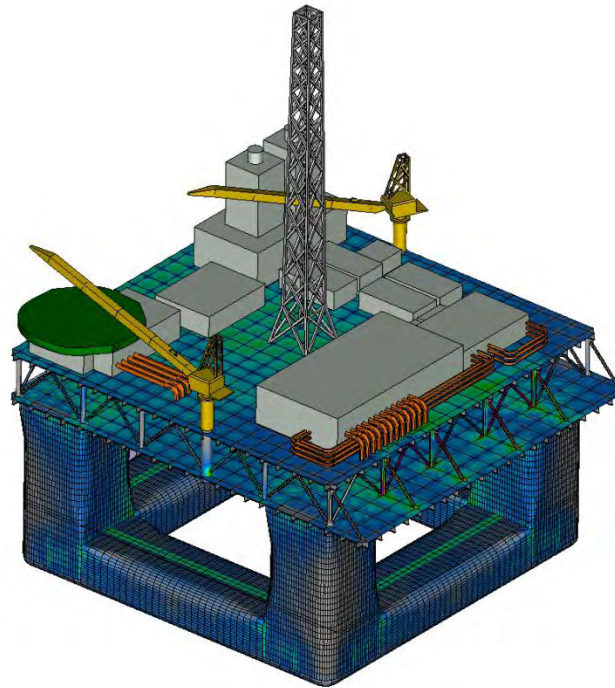
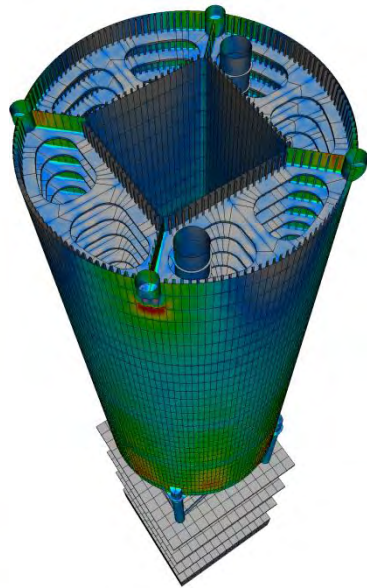
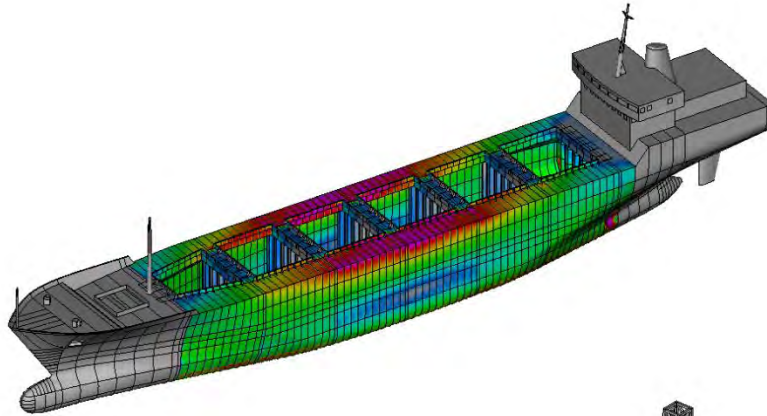
```

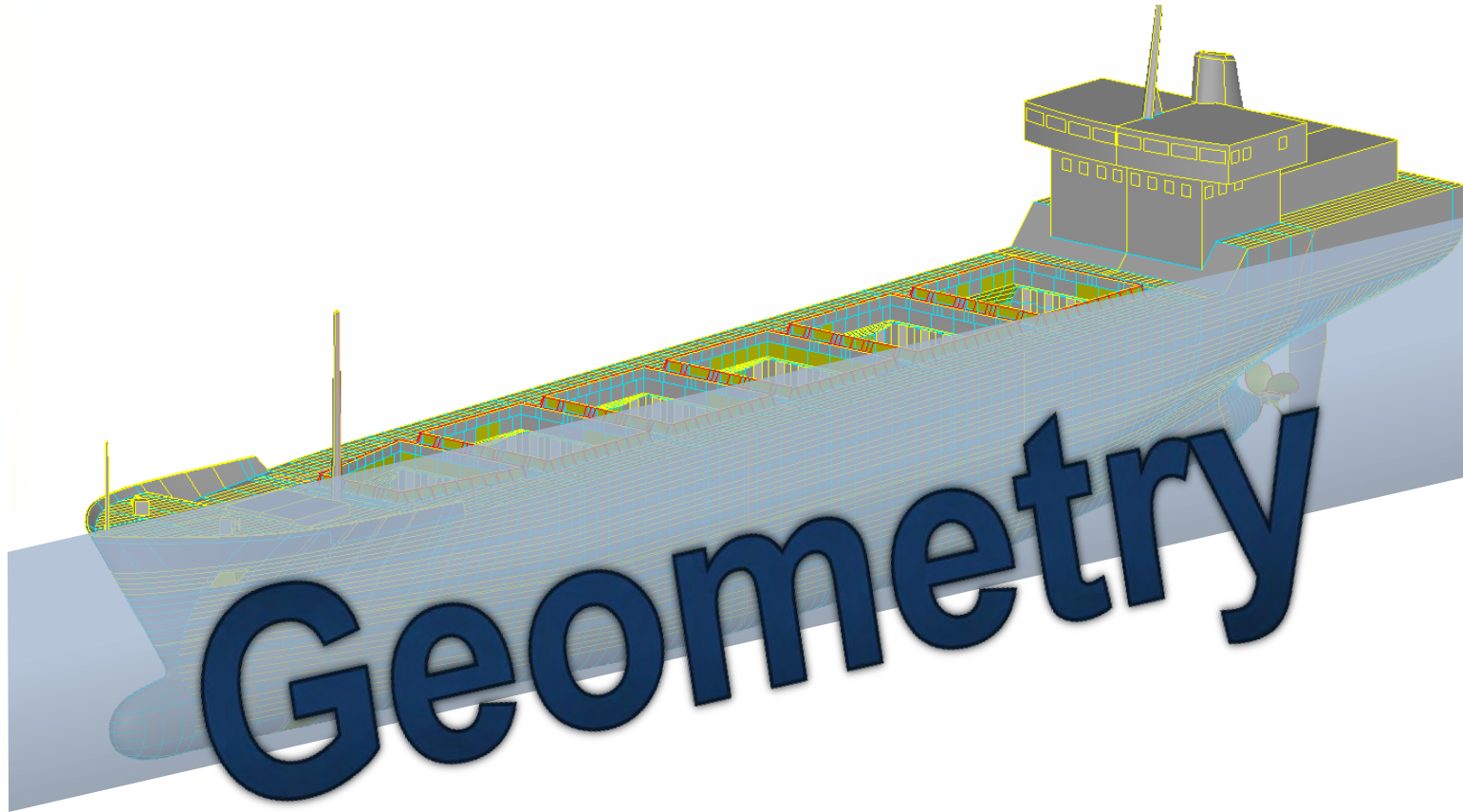
<Ship Version="pre 2.0">
  <Rendering Type="Idealised" />
  <Material Grade="Default" Density="7840" YoungsModulus="206000"
  PoissonRatio="0.3" YieldStress="220" UltimateStress="360" />
  <BarSection BarSectionId="BulbFlat120*23*5*6">
    <BulbFlat Height="120" Width="23" BulbRadius="5" WebThickness="6" />
  </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">
    <FlatBar Height="200" Width="12" />
  </BarSection>
  
```



# Interface for SESTRA

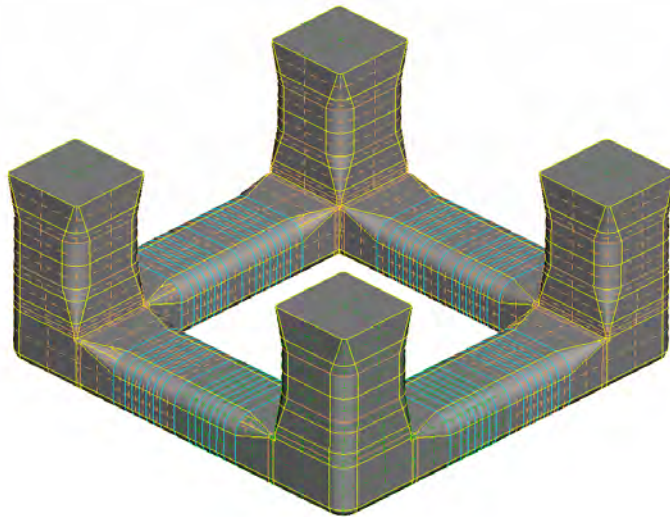
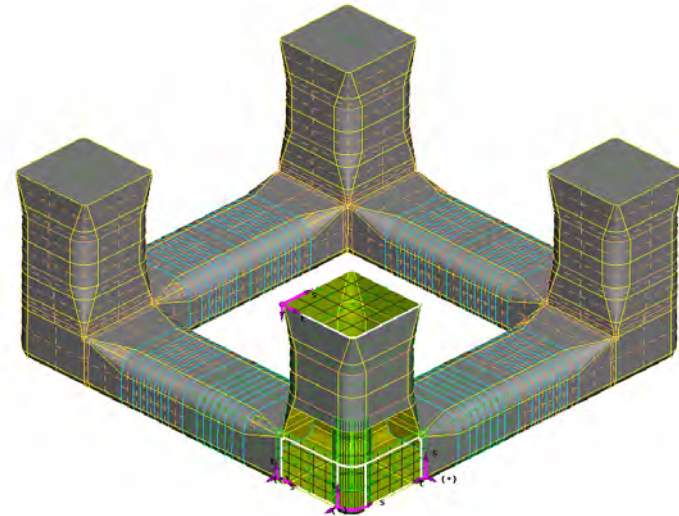
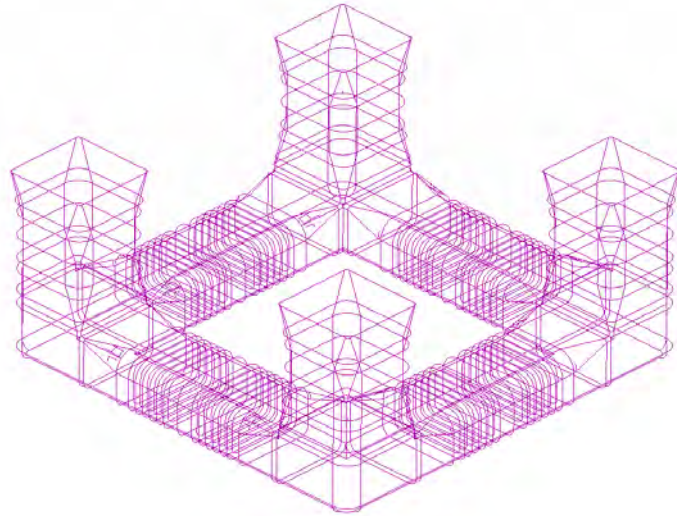
Input, output and new DECK for SESTRA





# Geometry creation

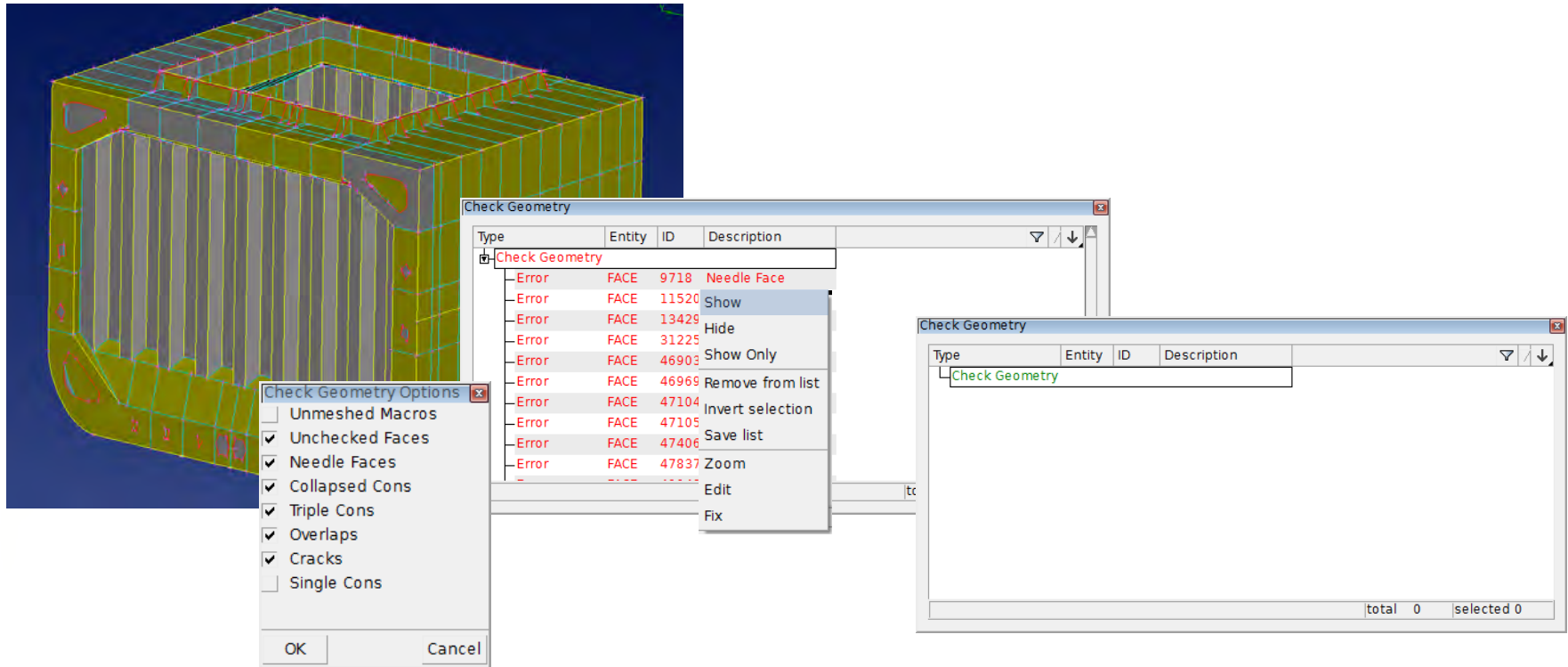
Easy generation of 3d geometrical entities





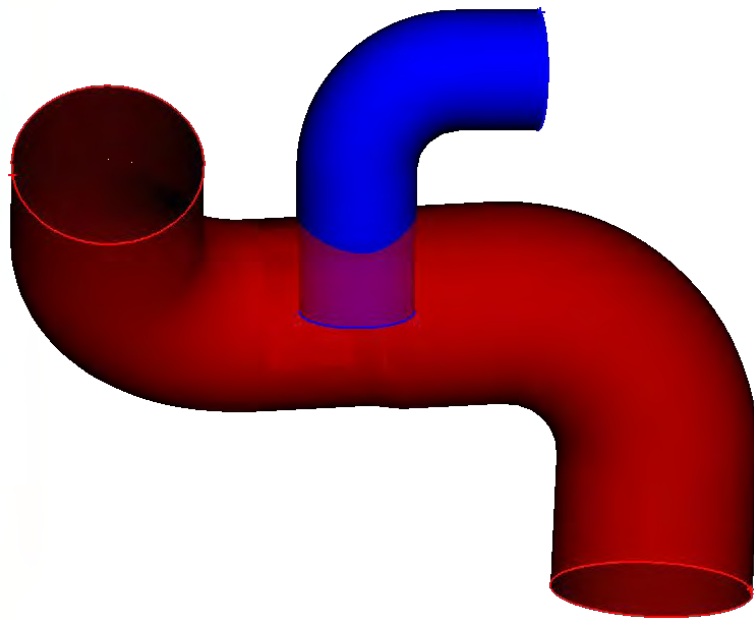
# Geometry checks

## Topological errors identification and fix

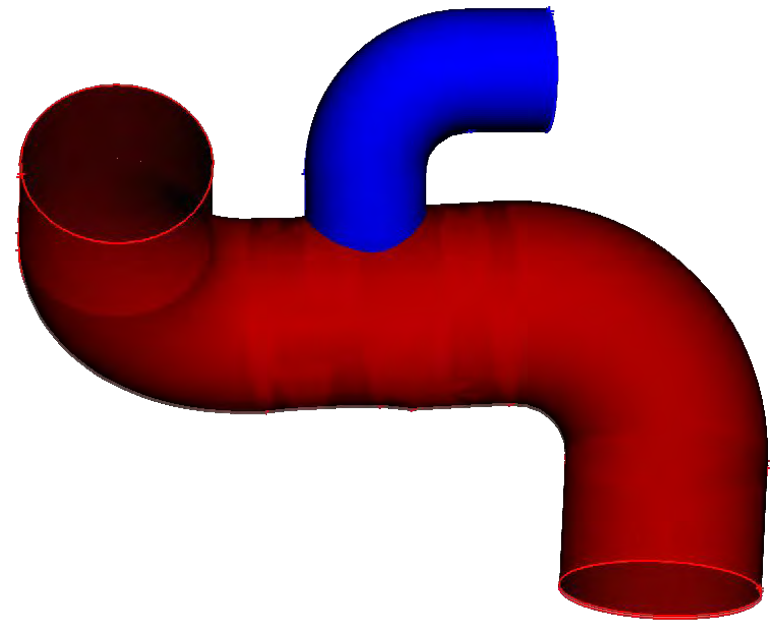


# Intersections

## Geometry trimming



**Before**

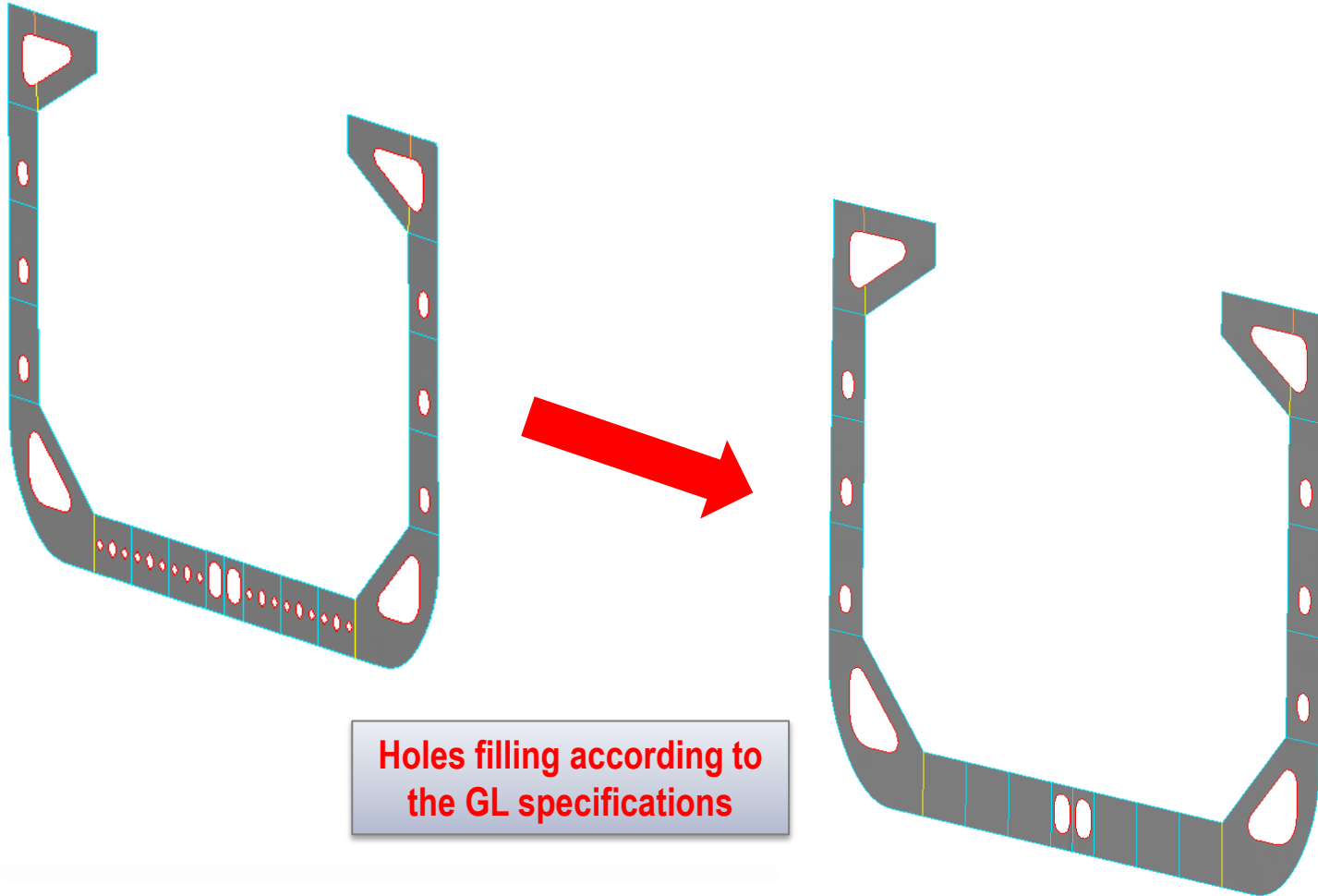


**After**

# Geometry check & model simplification

Identifying unchecked faces, needle faces, collapsed CONS, triple bounds, overlaps, cracks

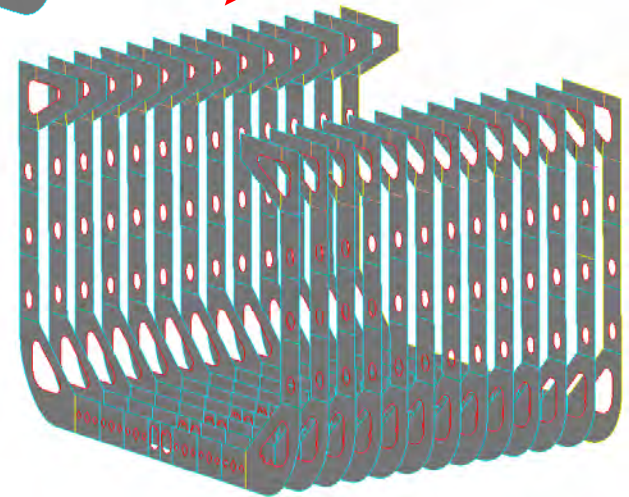
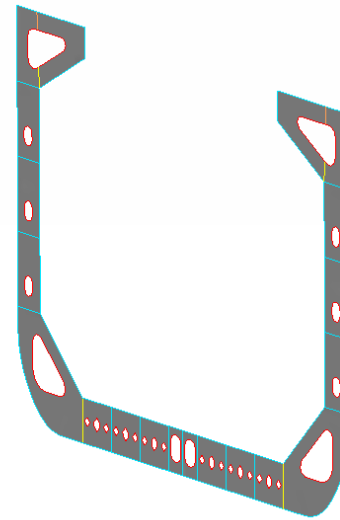
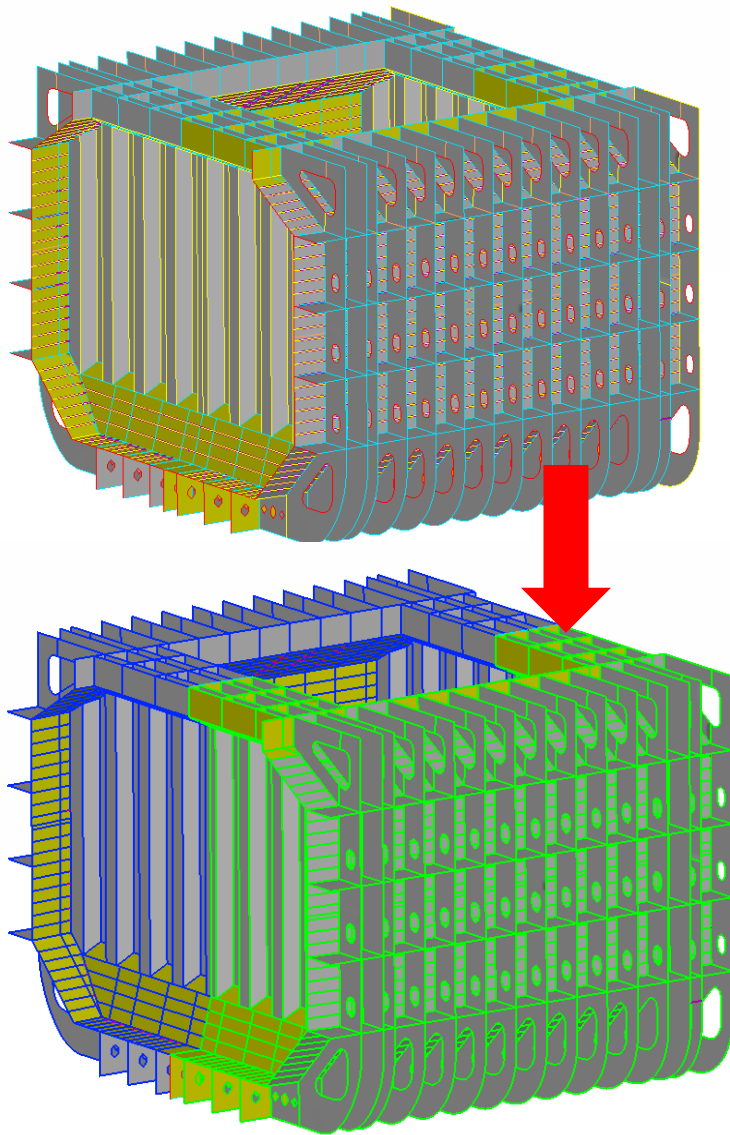
Treatment of holes, fillets, chamfers, features



Holes filling according to the GL specifications

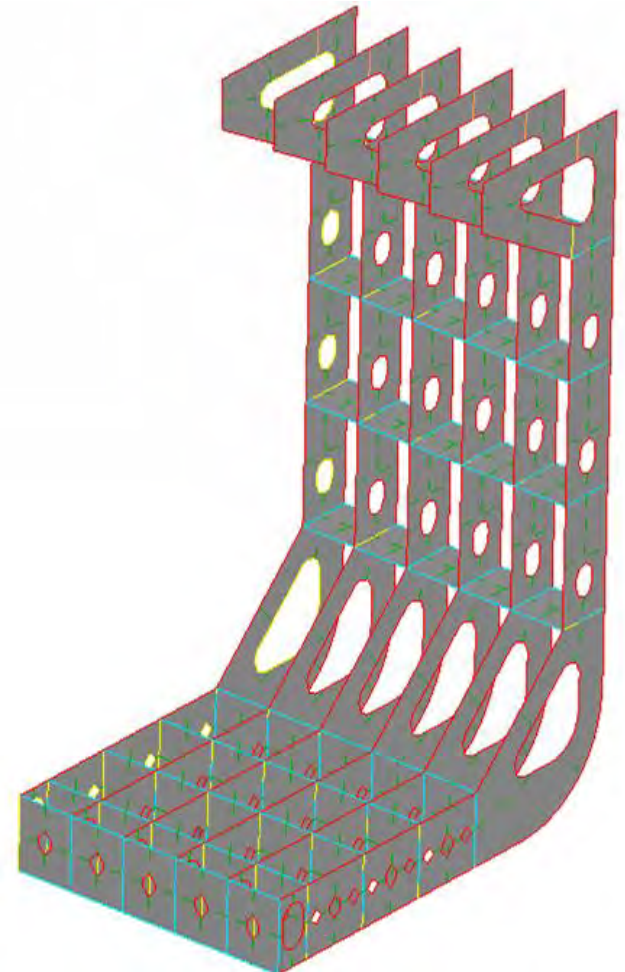
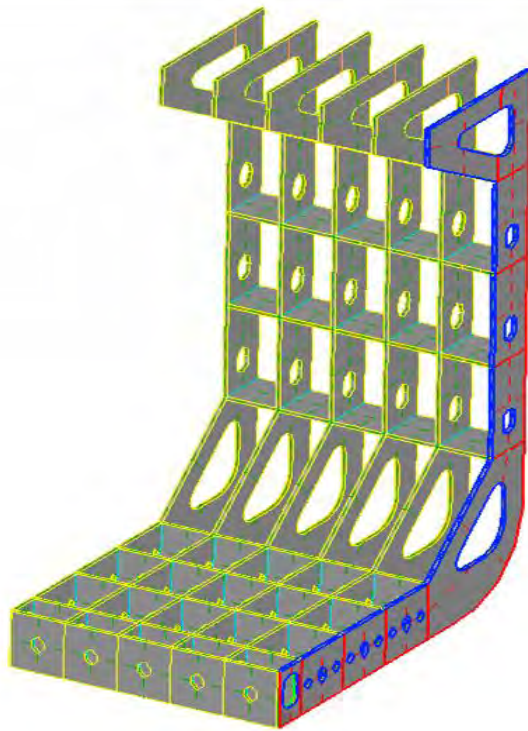
# Handling pattern-wise faces & parts

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



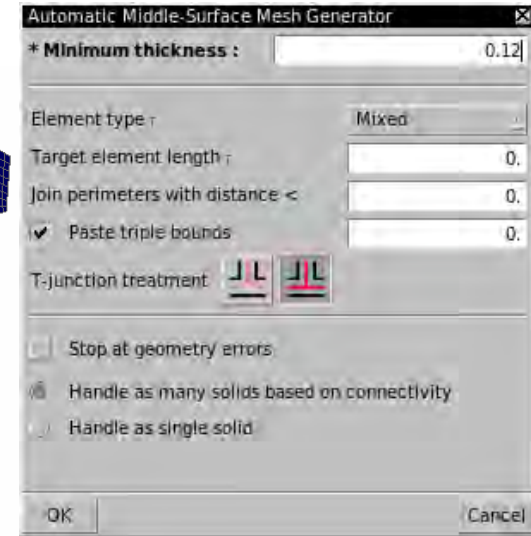
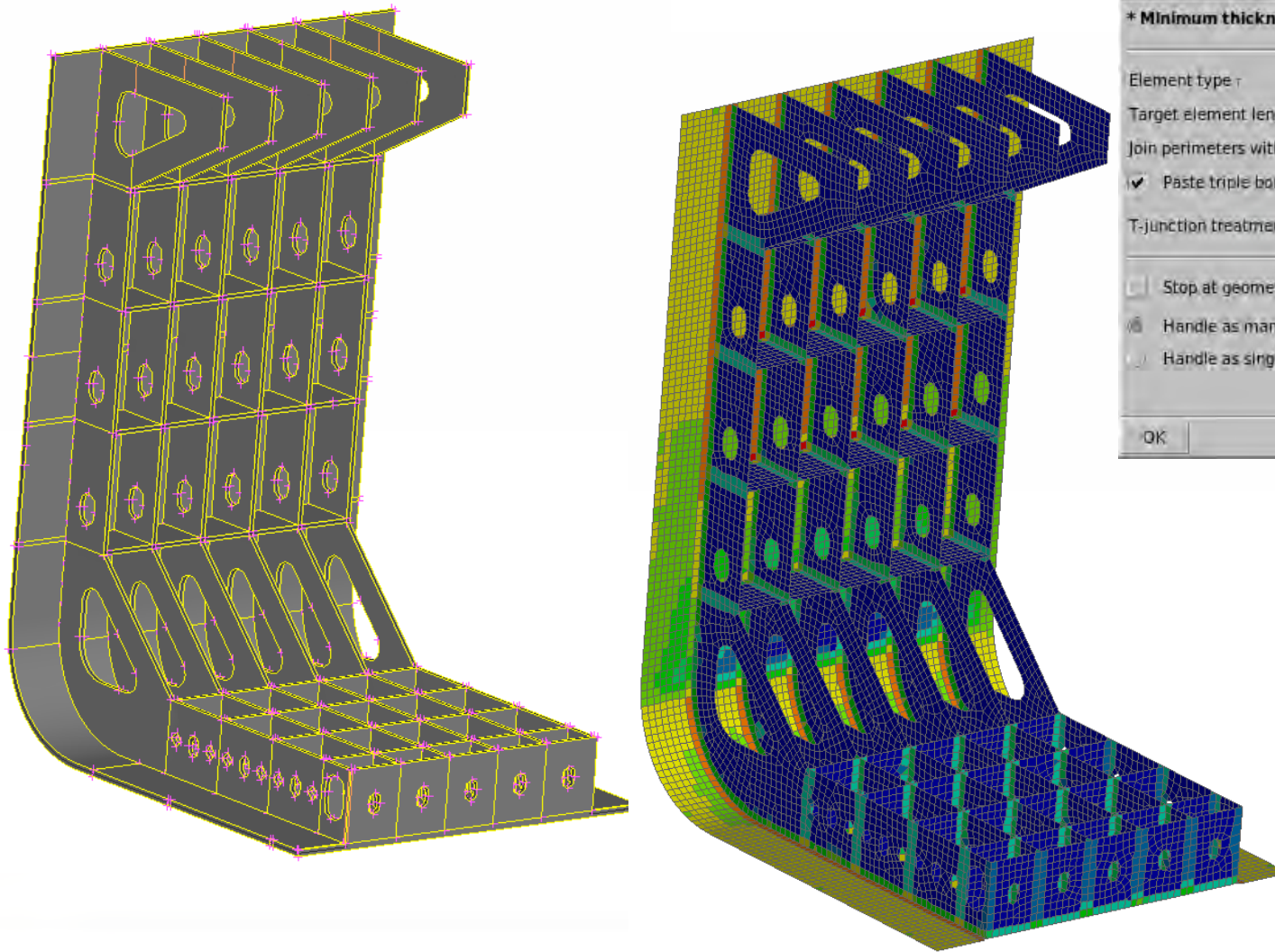
# Middle skin extraction

Special tool for middle skin extraction creates new geometry



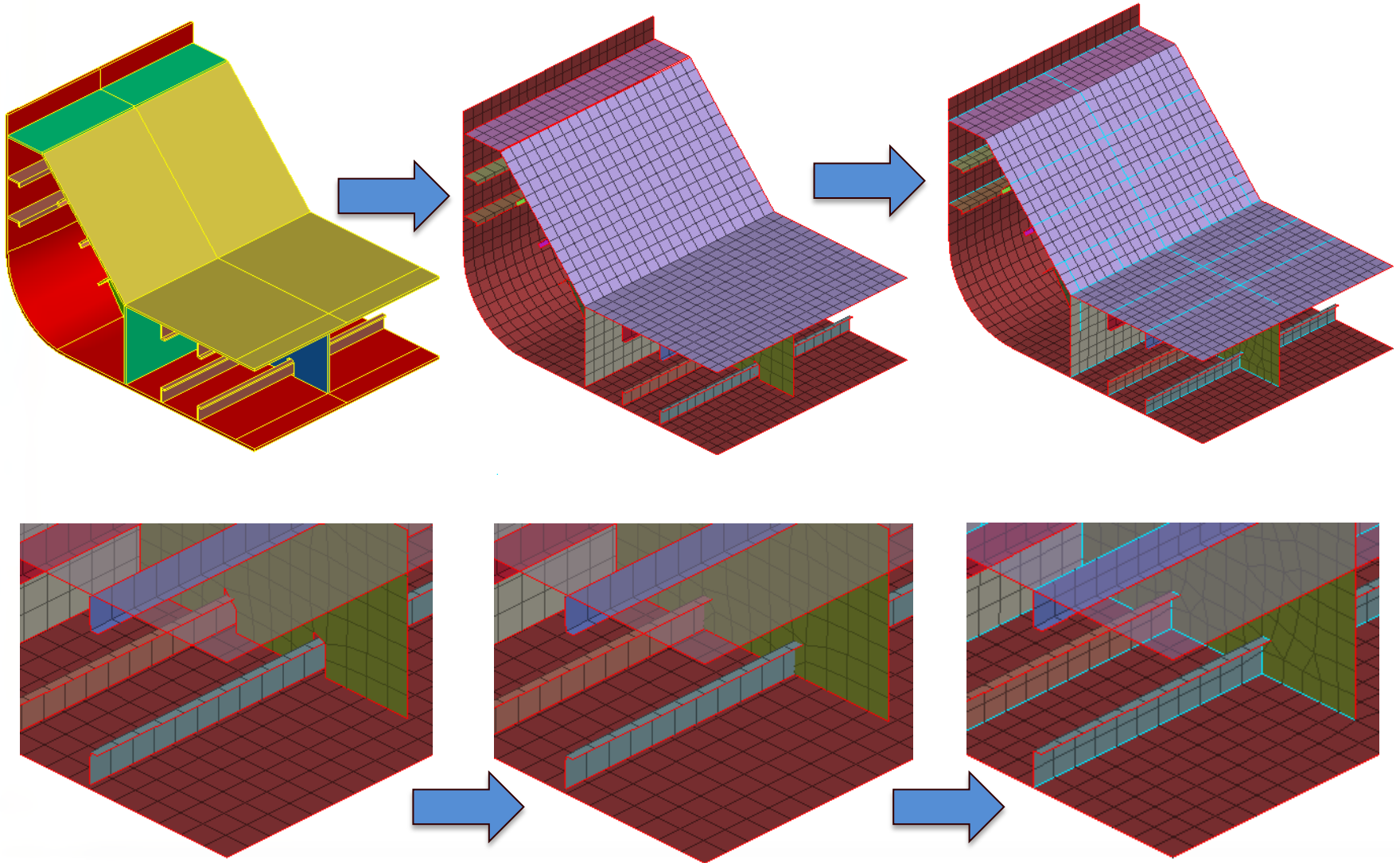
# Middle skin extraction

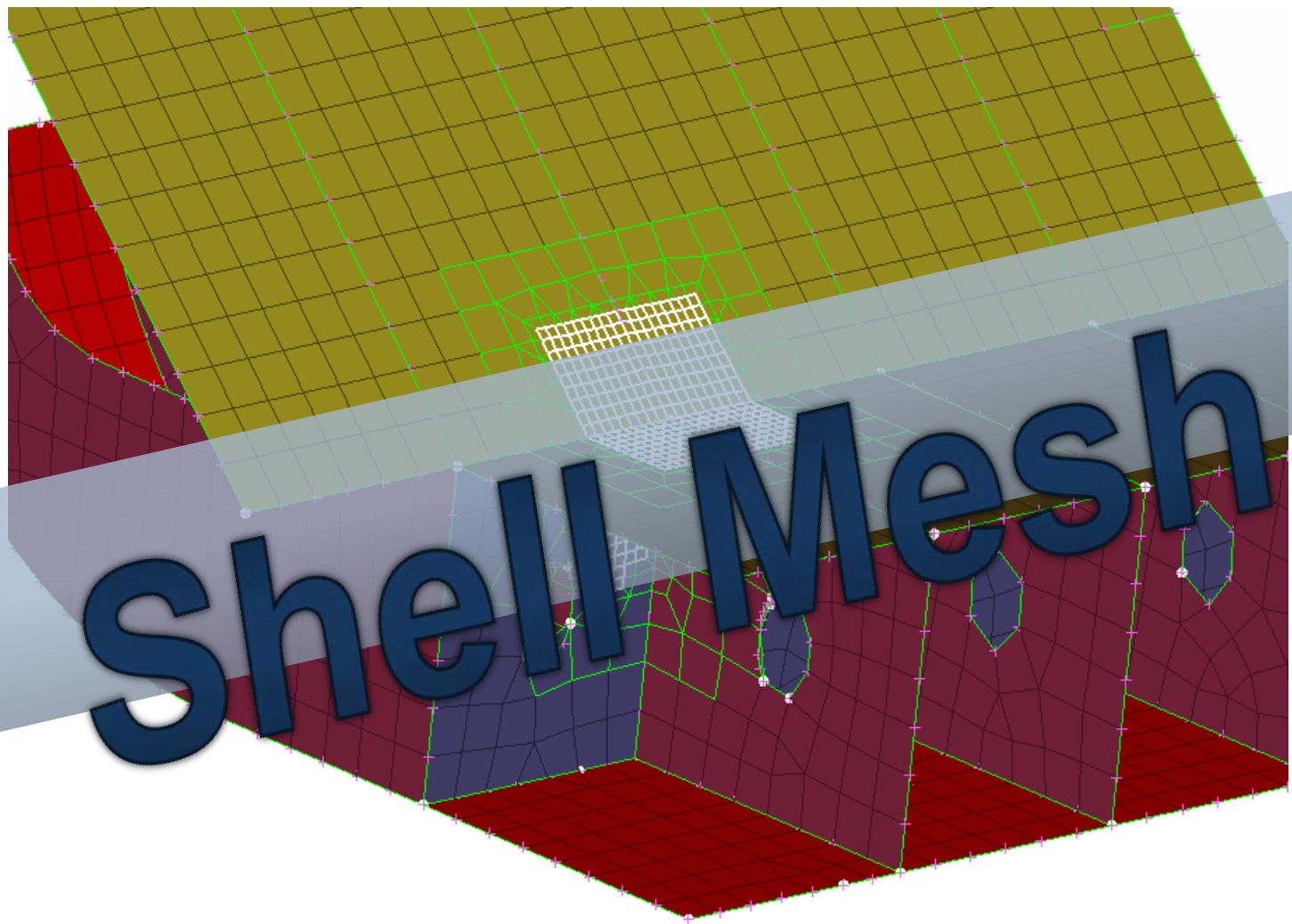
Fully automatic middle surface extraction creates FE Model and assigns nodal thickness



# Automatic middle skin extraction and assembly

Creates the middle skin, connects the parts and fills unwanted openings



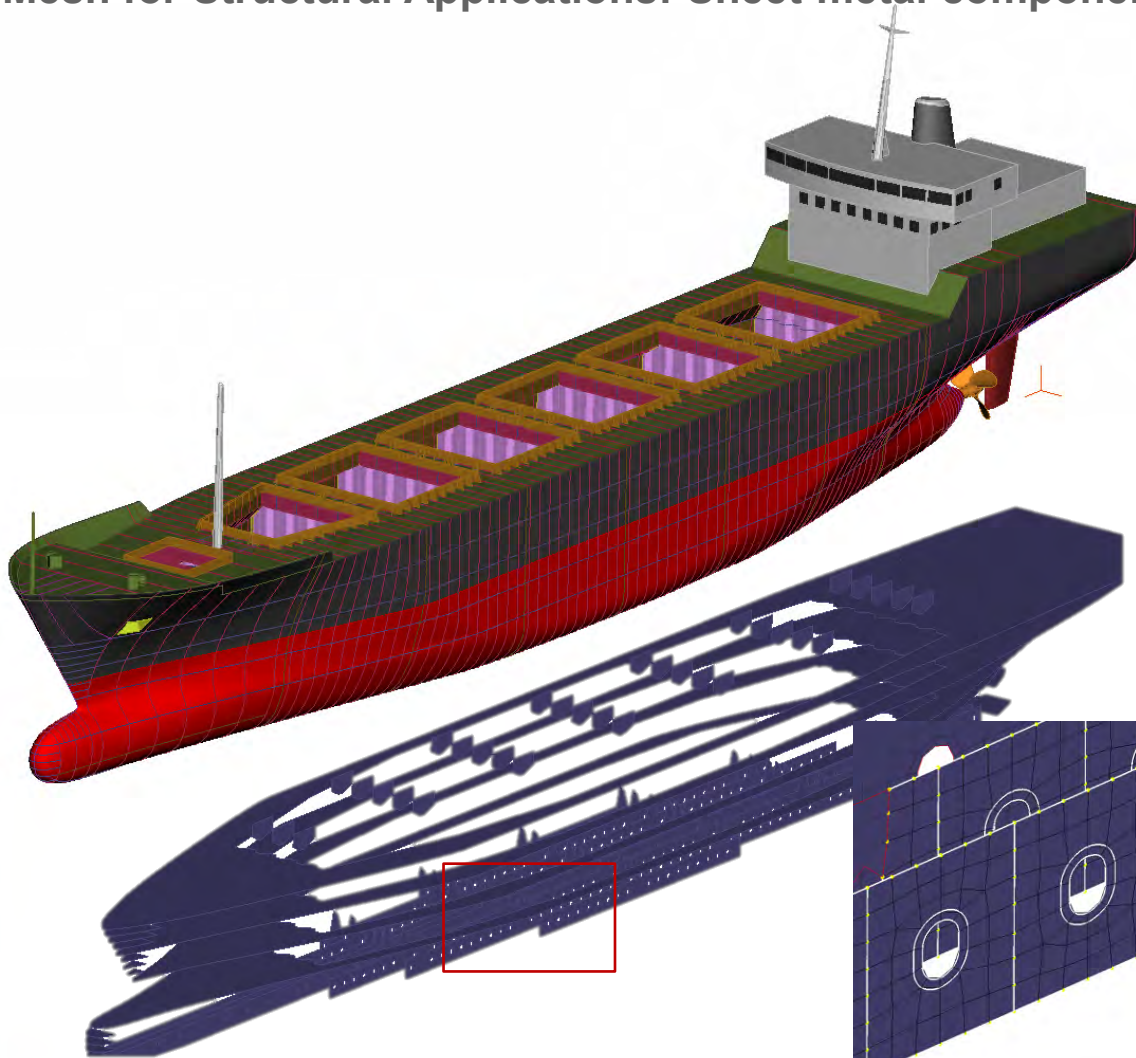


# Shell Mesh



# Shell Meshing

Mesh for Structural Applications: Sheet-metal components



Automatic defeaturing

Mesh quality control

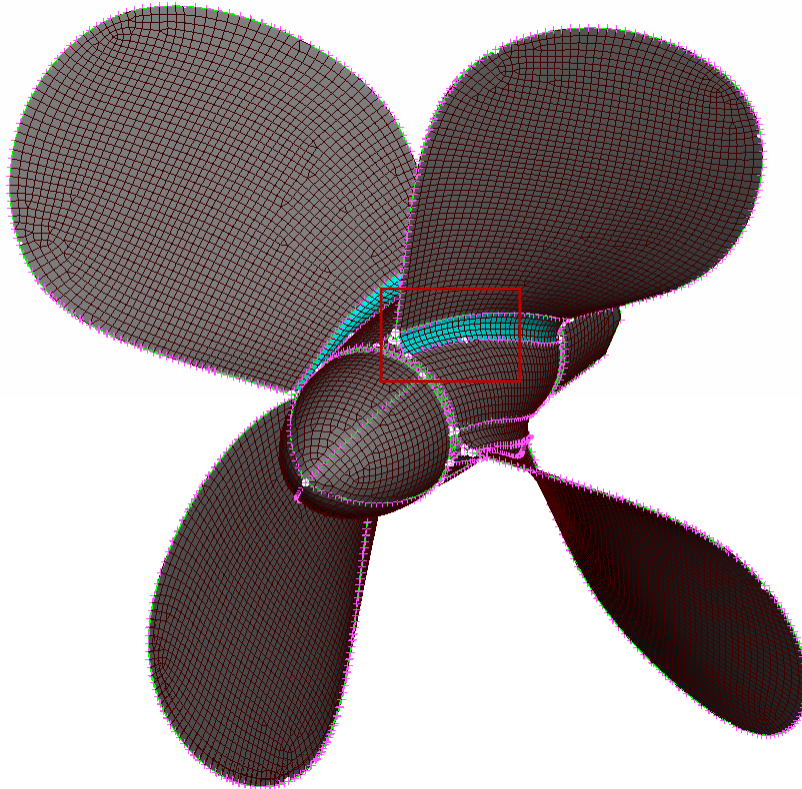
Feature treatment

- Holes
- Fillets
- Flanges

Fully automated through Batch Mesh

# Shell Meshing

Mesh for Structural Applications: Sheet-metal components

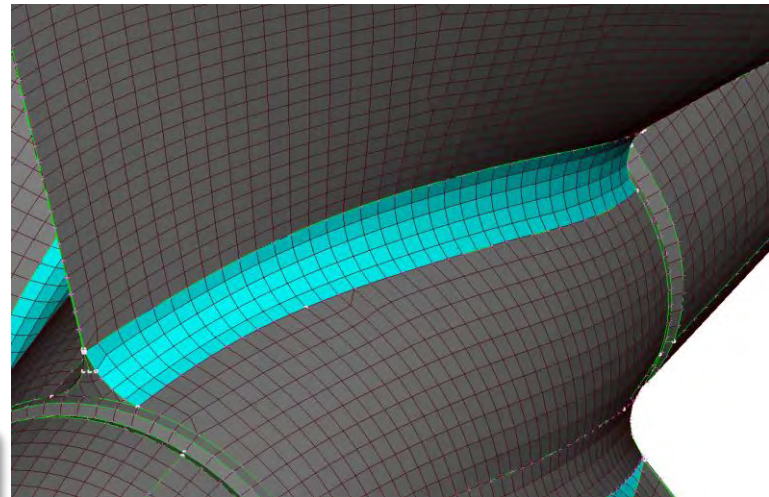


Automatic defeaturing

Mesh quality control

Feature treatment

- Holes
- Fillets
- Flanges



Fully automated through Batch Mesh

# Shell Meshing

## Numerous quality criteria

Quality Criteria

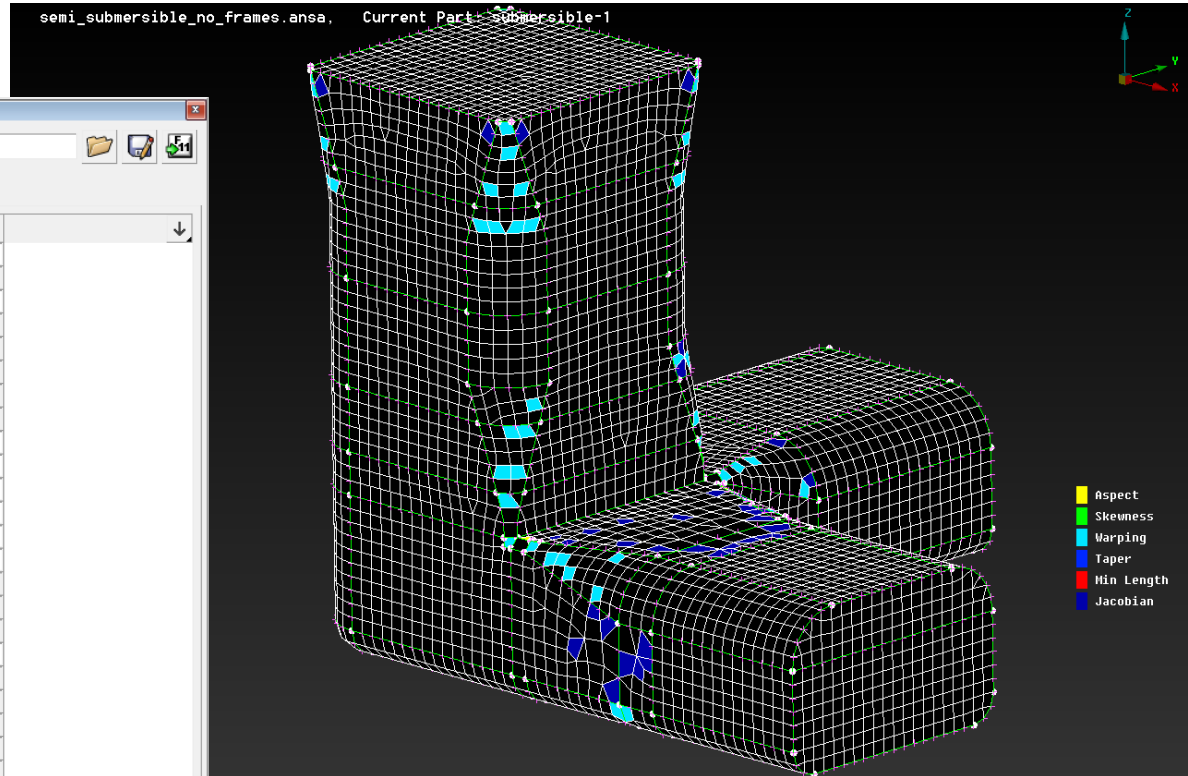
Name:

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 %		Grey	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		Purple	4.
<input type="checkbox"/> distance from geometry		Purple	2.
<input type="checkbox"/> distance from origin		Light Green	2.
<input type="checkbox"/> multi violation		Yellow	2.
<input type="checkbox"/> growth ratio		Light Blue	1.2
<input type="checkbox"/> incomplete element		Cyan	

Enable Ranges Edit Criteria Visibility

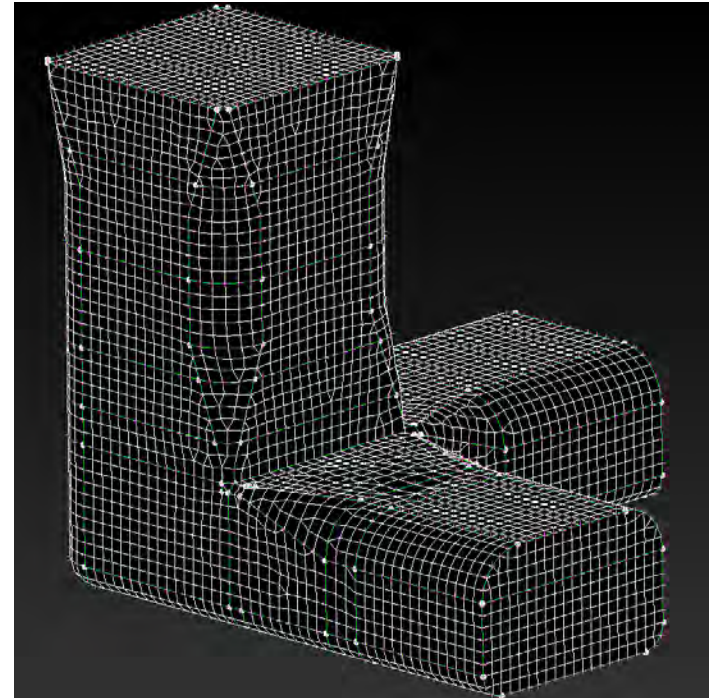
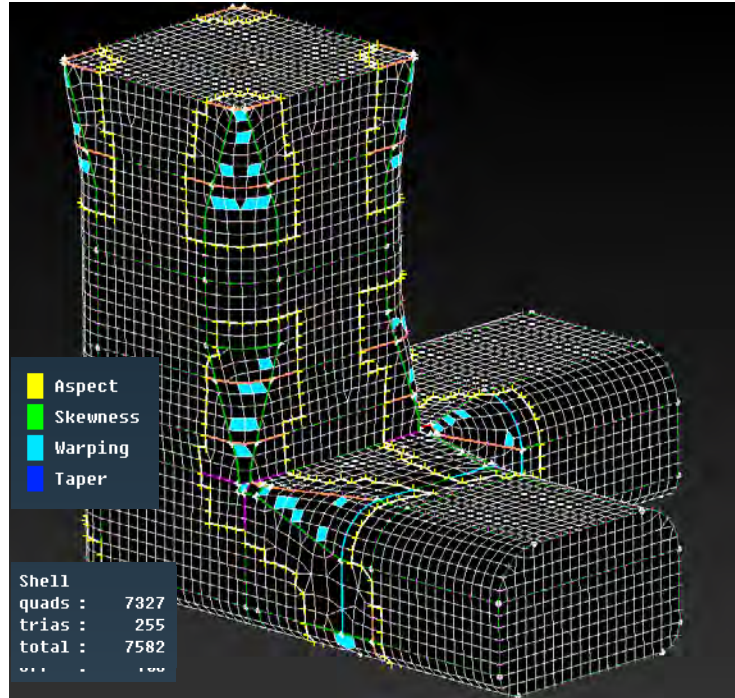
OK Cancel



Color-coding of violations

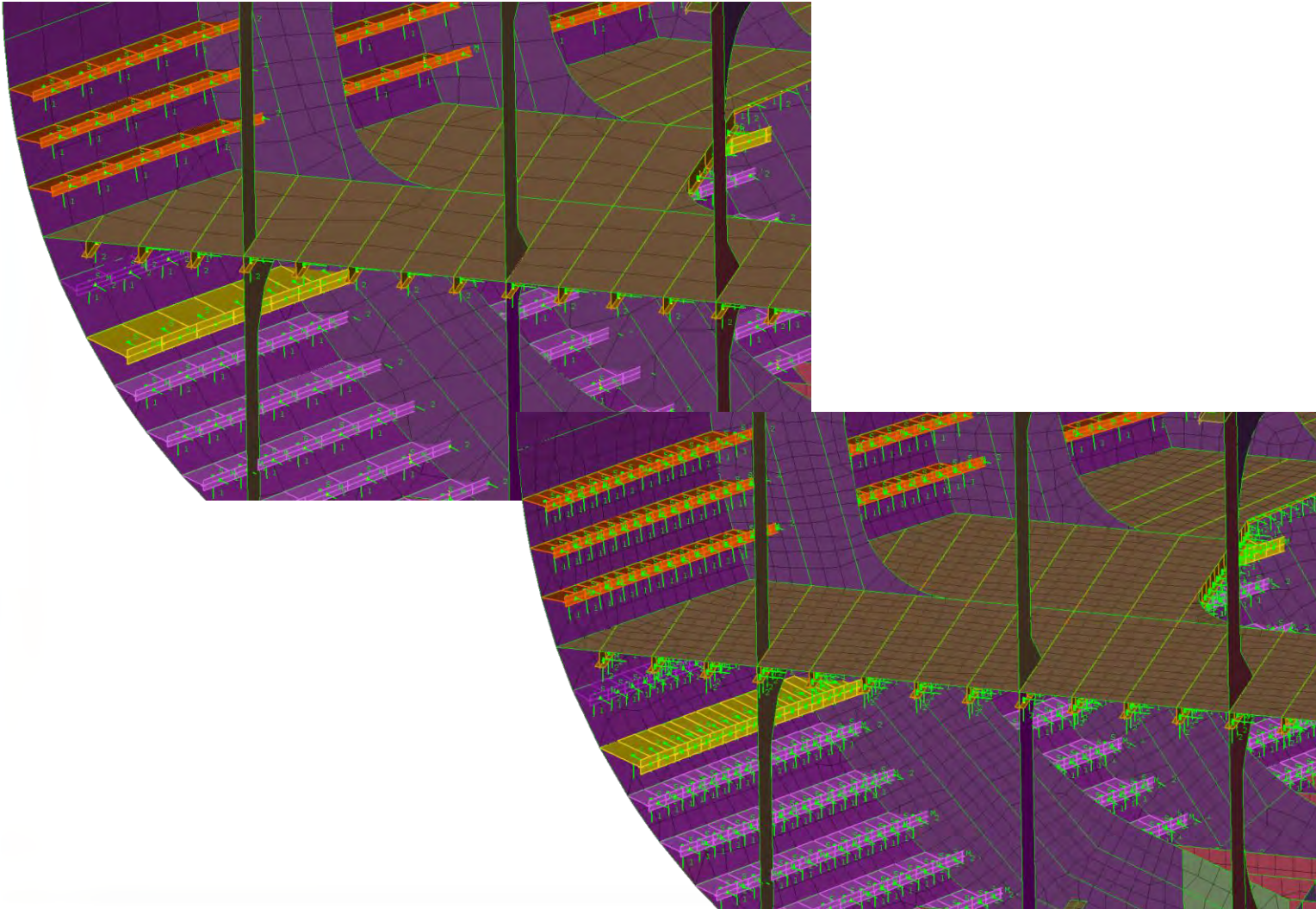
# Shell Meshing

Automatic fix of quality violations



# Reconstruct shells and beams

Reconstruct shells and the attached beams at the same time

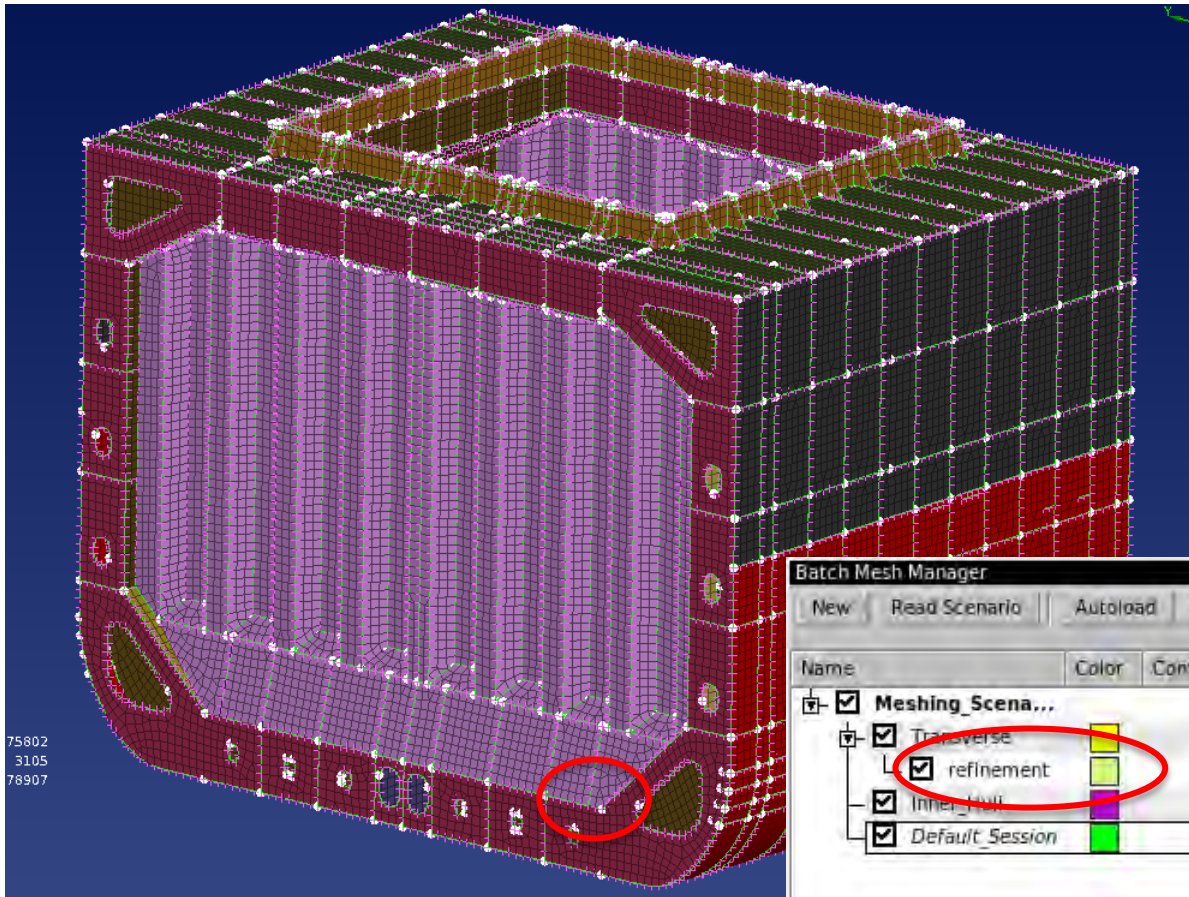


# Batch meshing

Definition of meshing parameters and quality criteria

Features treatment and model simplification

Automatic meshing and quality improvement



Batch Mesh Manager

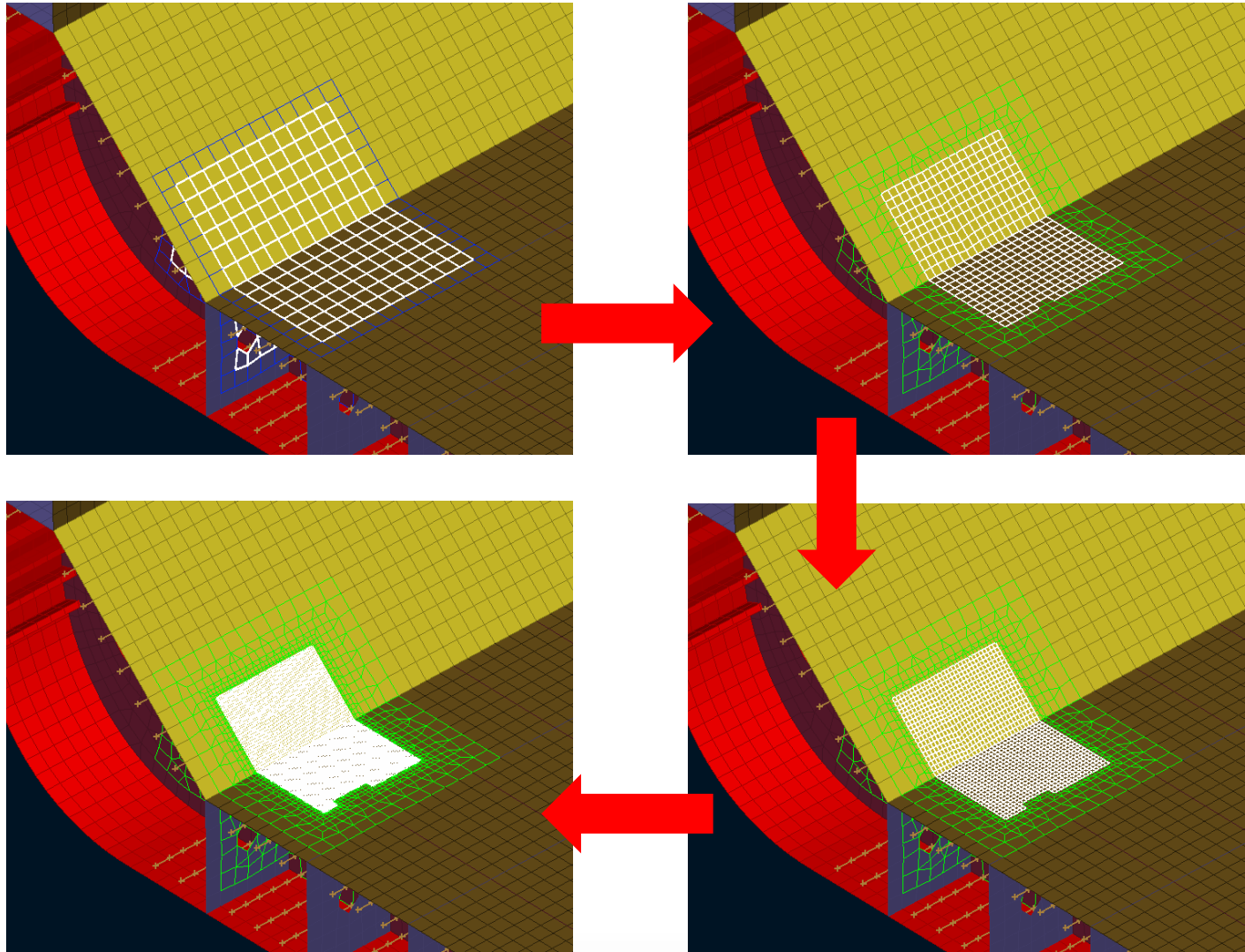
New Read Scenario Autoload Run

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

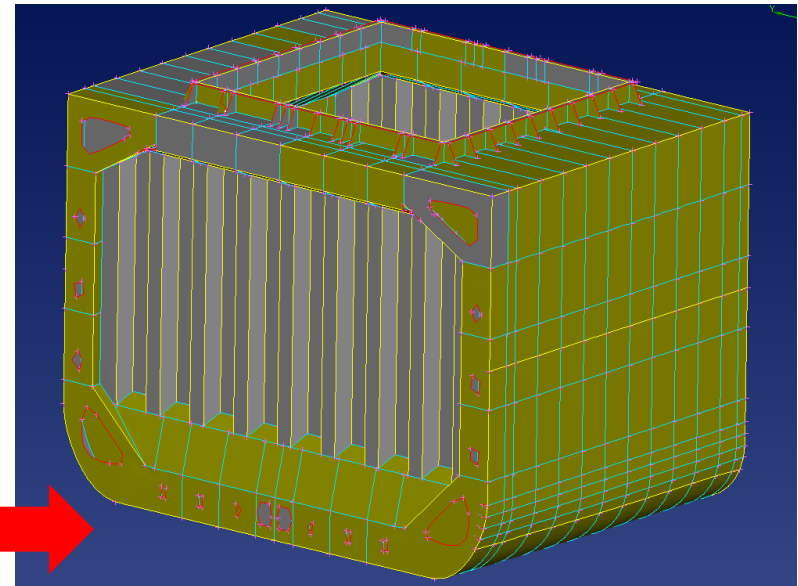
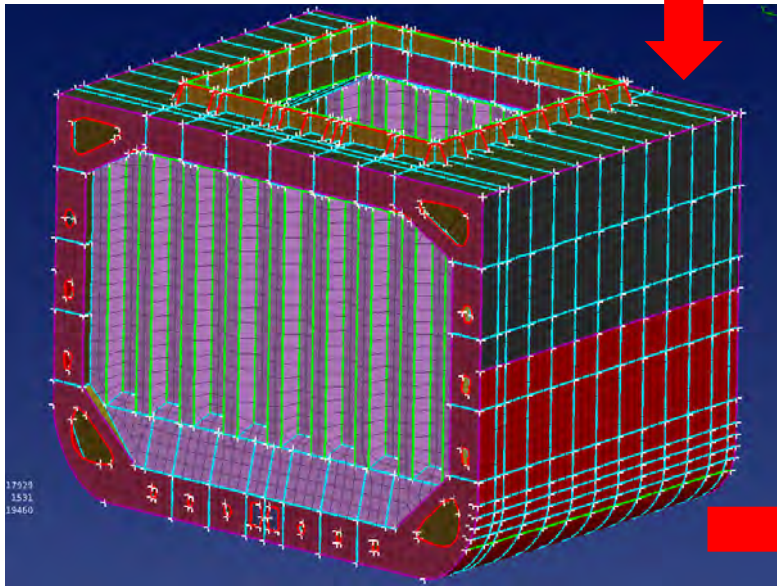
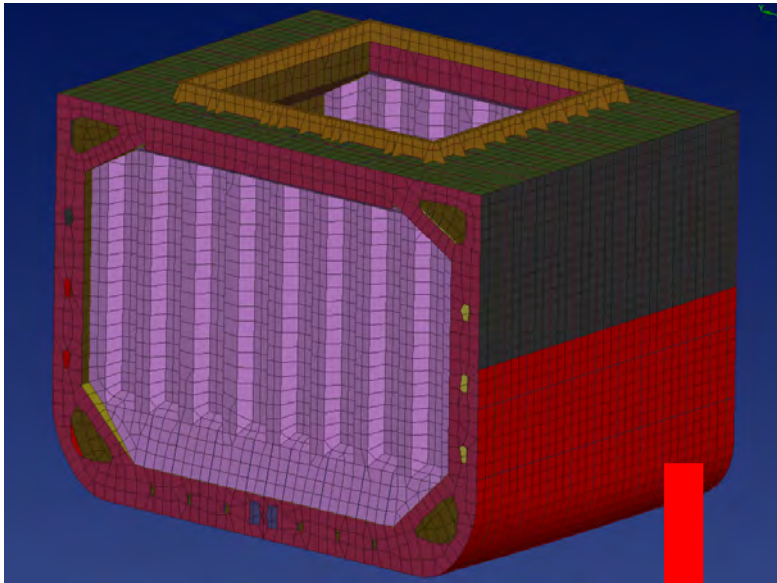
75802  
3105  
78907

# Local refinement

Local mesh refinement of geometry mesh and FE



# Automatic definition of geometry from FE model



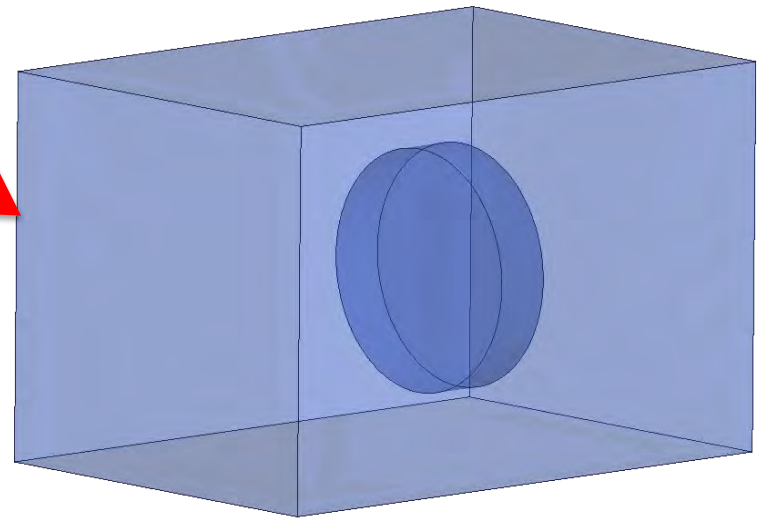
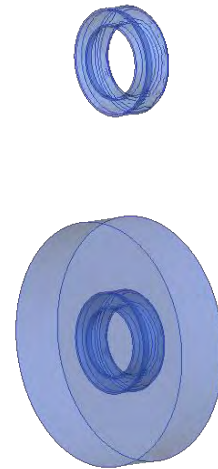
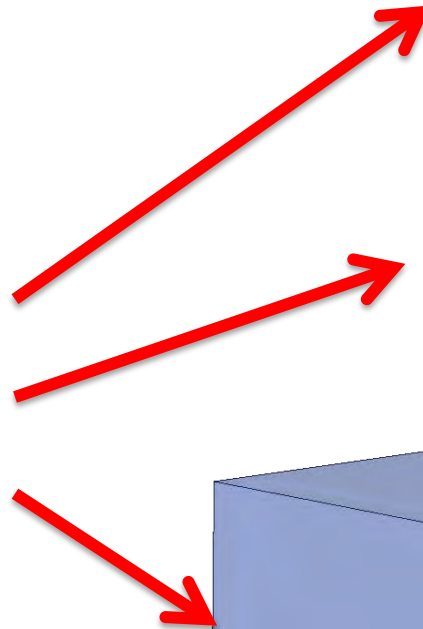
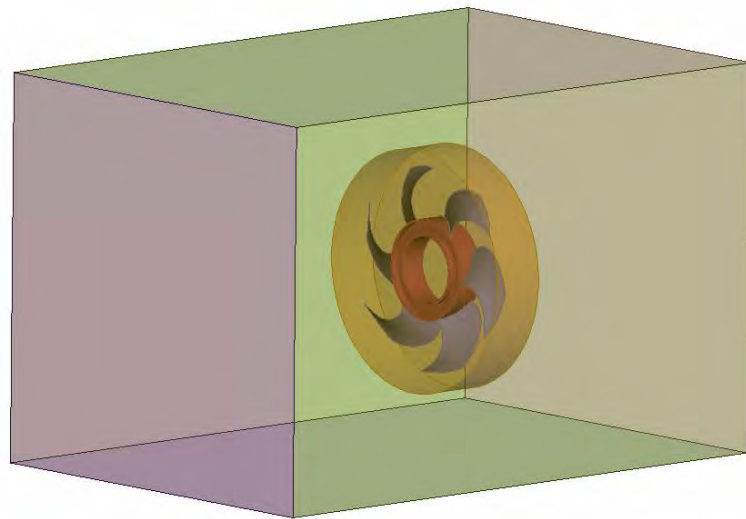




# Volume Mesh

# Volume Meshing

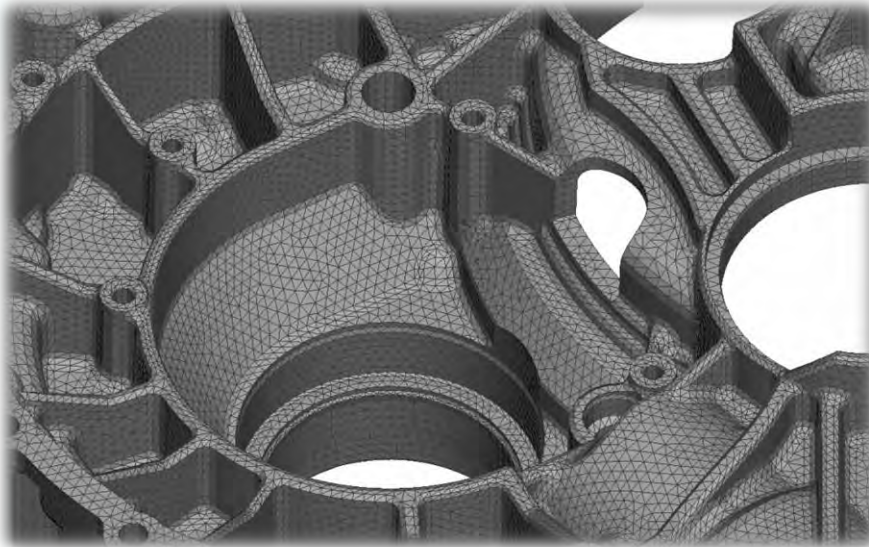
Automatic volume detection



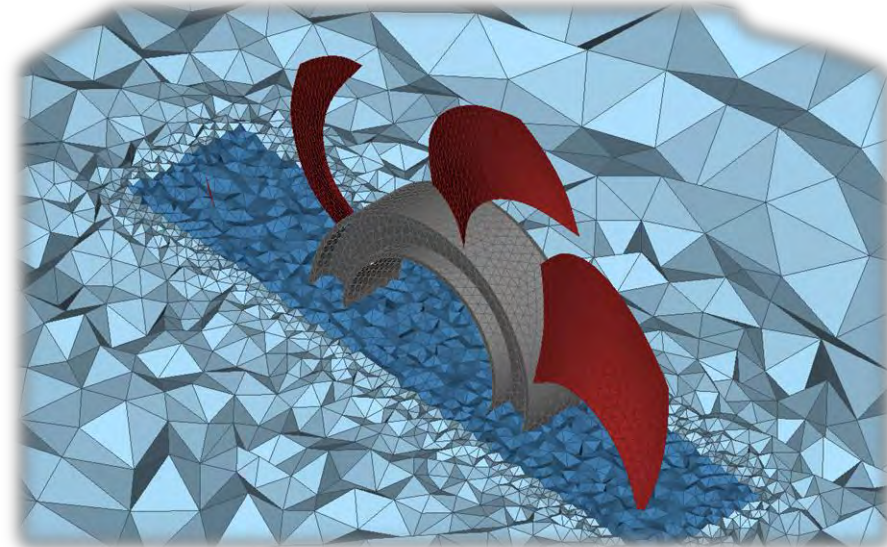
Detection of all valid volumes and sub-volumes

# Volume Meshing

## Tetrahedral Mesh



**Structural**

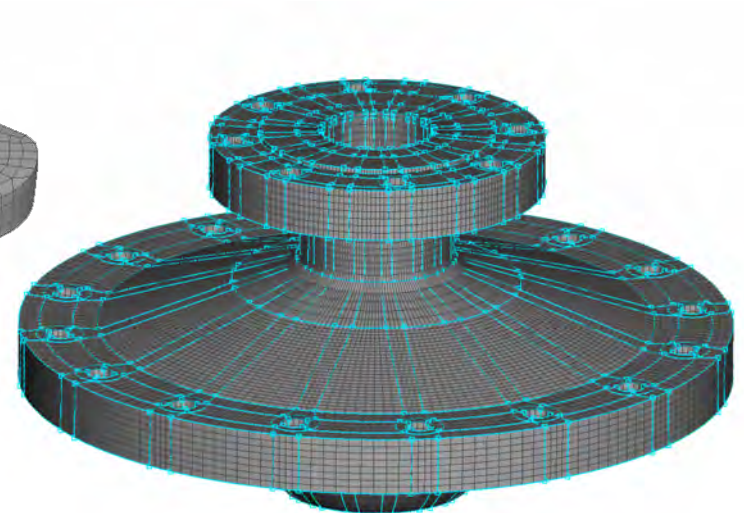
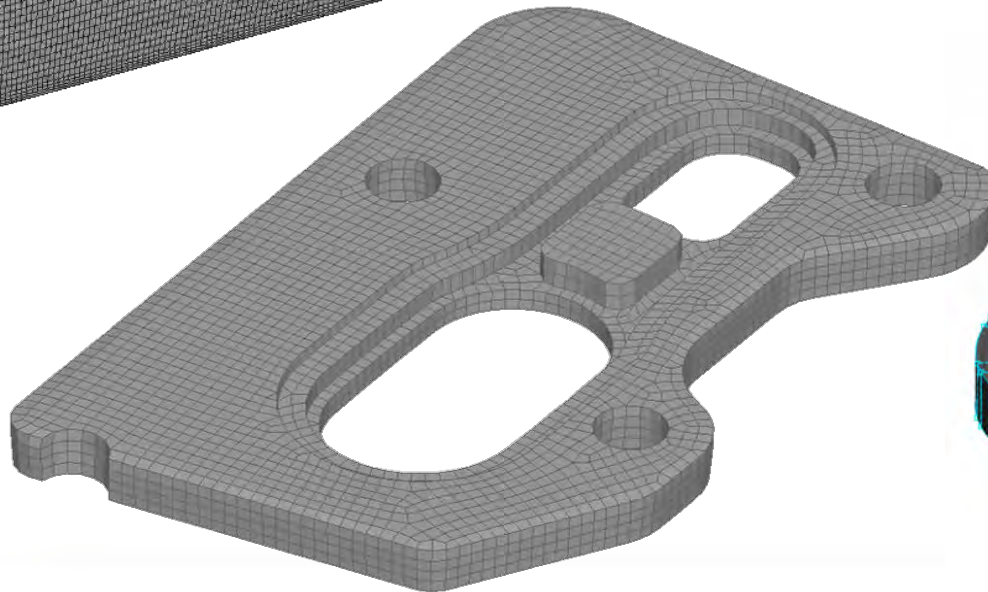
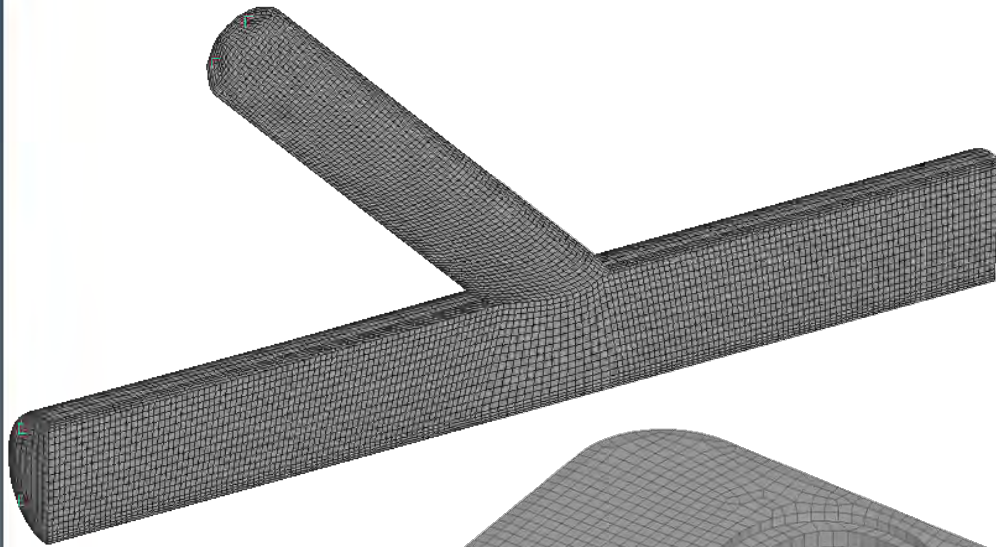


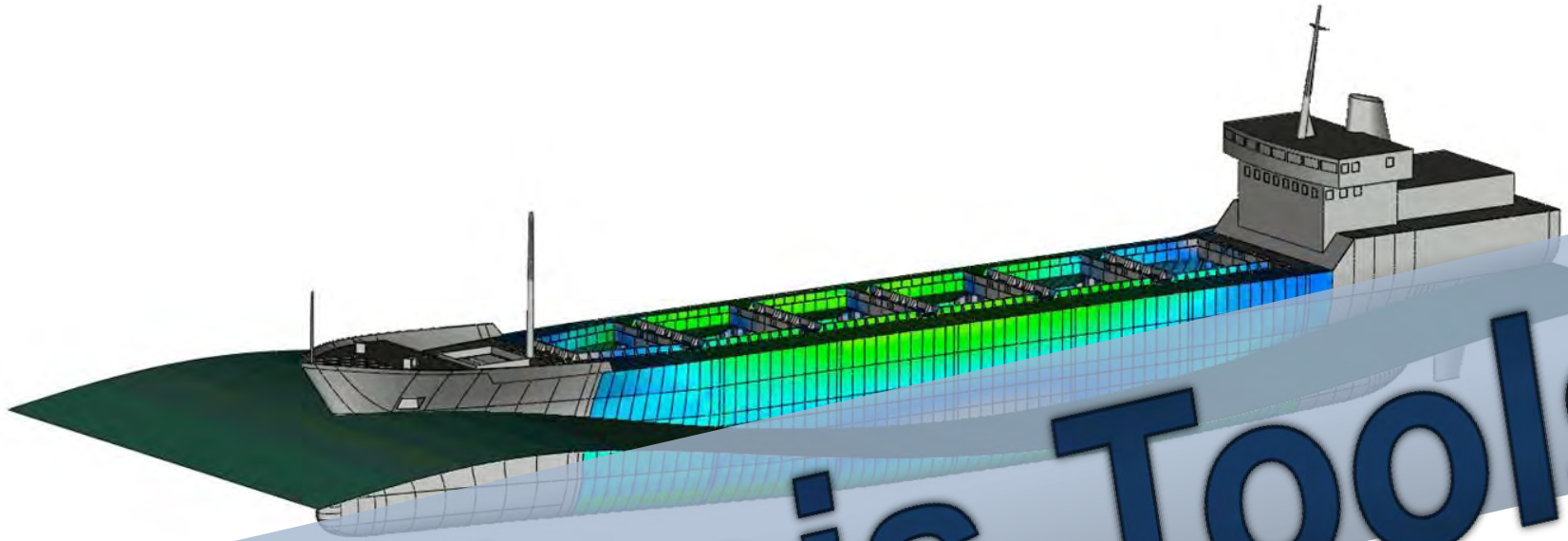
**CFD**

Fully automated through Batch Mesh

# Volume Meshing

## Hexa Meshing





# Analysis Tools

# Stiffeners creation using beams

Creating database of cross sections of any shape

The screenshot displays a CAE software interface with the following components:

- Database:** A list of model entities including ANSPART (12), CROSS (75), CROSS\_CURVE (56), CROSS\_SECTION (19), EDGE, ELEMENT (10698), GEOMETRY (18370), GRID (6240), MATERIAL (23), MORPH (8), PROPERTY (56), and SET (1).
- Cross Section 1:** A detailed view of a cross-section with a vertical axis 'Y' and a horizontal axis 'X'. It shows a complex profile with a top flange and a bottom flange. Geometrical results are listed below the diagram.
- Cross Section Database:** A table listing 15 different cross-sections with their IDs and names.
- Geometrical Results:** A list of numerical values for various properties such as Area (A), Centroids (xs, ys), Moments of Inertia (Ix, Iy, Ixy, I1, I2, I12), Section Moduli (Wx, Wy), and Radii of Gyration (Ax, Ay).
- Modules Buttons:** A panel on the right side containing various tool buttons for Points, Curves, Hot Points, Visib., Properties, Calculation, Sections, Library, Connections, and Results.

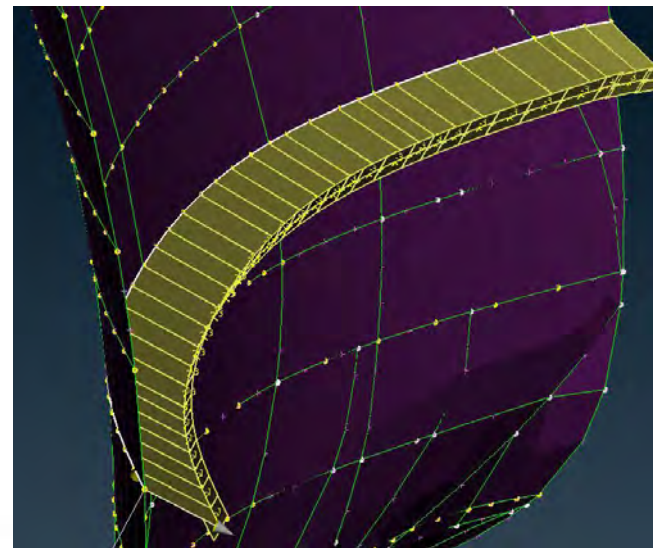
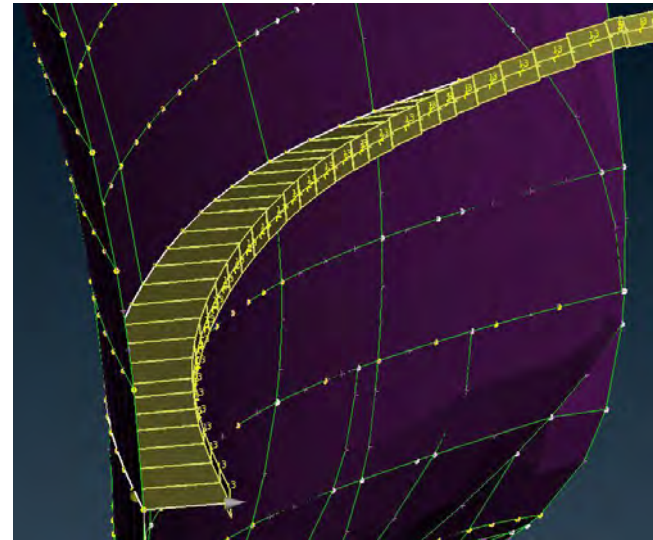
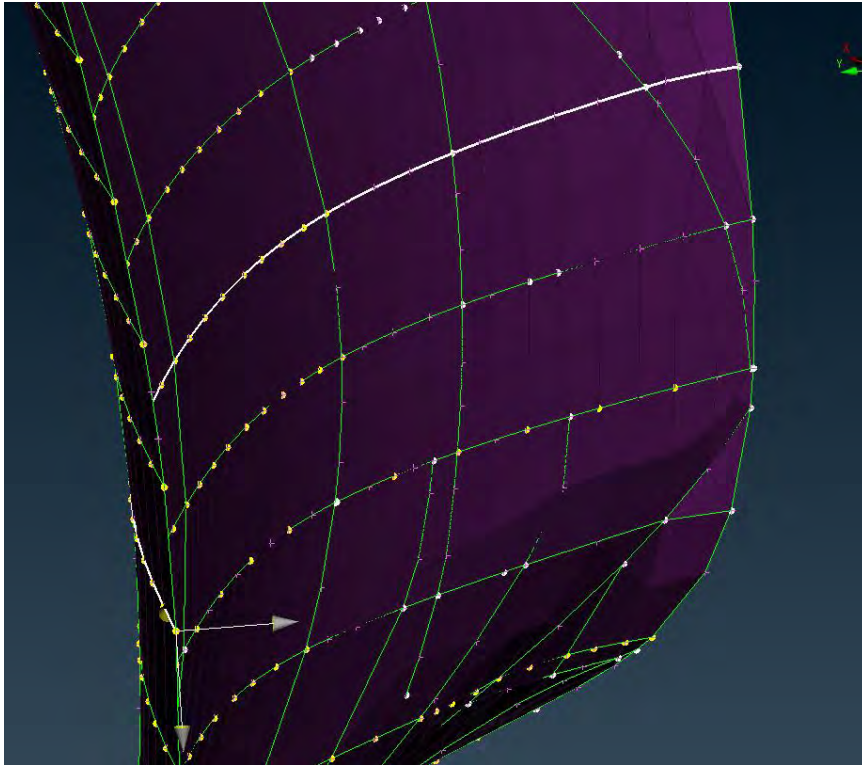
Id	Name
1	FB(T)_650x13+220x30
2	FB(T)_600x12+175x25
3	FB(T)_625x12+200x27
4	FB(T)_1100x17+275x17
5	FB(T)_470x12+200x27
6	FB(T)_500x12+150x20
7	FB(T)_450x12+150x20
8	FB(T)_600x12+150x12
9	FB(T)_425x12+150x18
10	FB(T)_375x12+150x18
11	FB(T)_600x13.5+175x20
12	FB(T)_925x18.5+260x18.5
13	FB(T)_300x12+175x25
14	FB(T)_375x15+150x20
15	FB(T)_575x12+175x20

```

Geometrical Results
A      : 0.01505
xs     : 0
ys     : 0.467525
Ix     : 0.000688919
Iy     : 2.662e-05
Ixy    : 0
I1     : -0
I2     : 0.000688919
I12    : 2.662e-05
It     : 2.45602e-06
xm     : 0
ym     : 0.65
Wx(s) : 8.8637e-07
Wx(m) : 1.05926e-37
Gamma : 0
Wx     : 0.00147355
Wy     : 0.000242
Wpx    : 0.00284153
Wpy    : 0.000363
A11    : 0.00549986
A2     : 0.00727967
A12    : 0
Ax     : 0.00549986
Ay     : 0.00727967
    
```

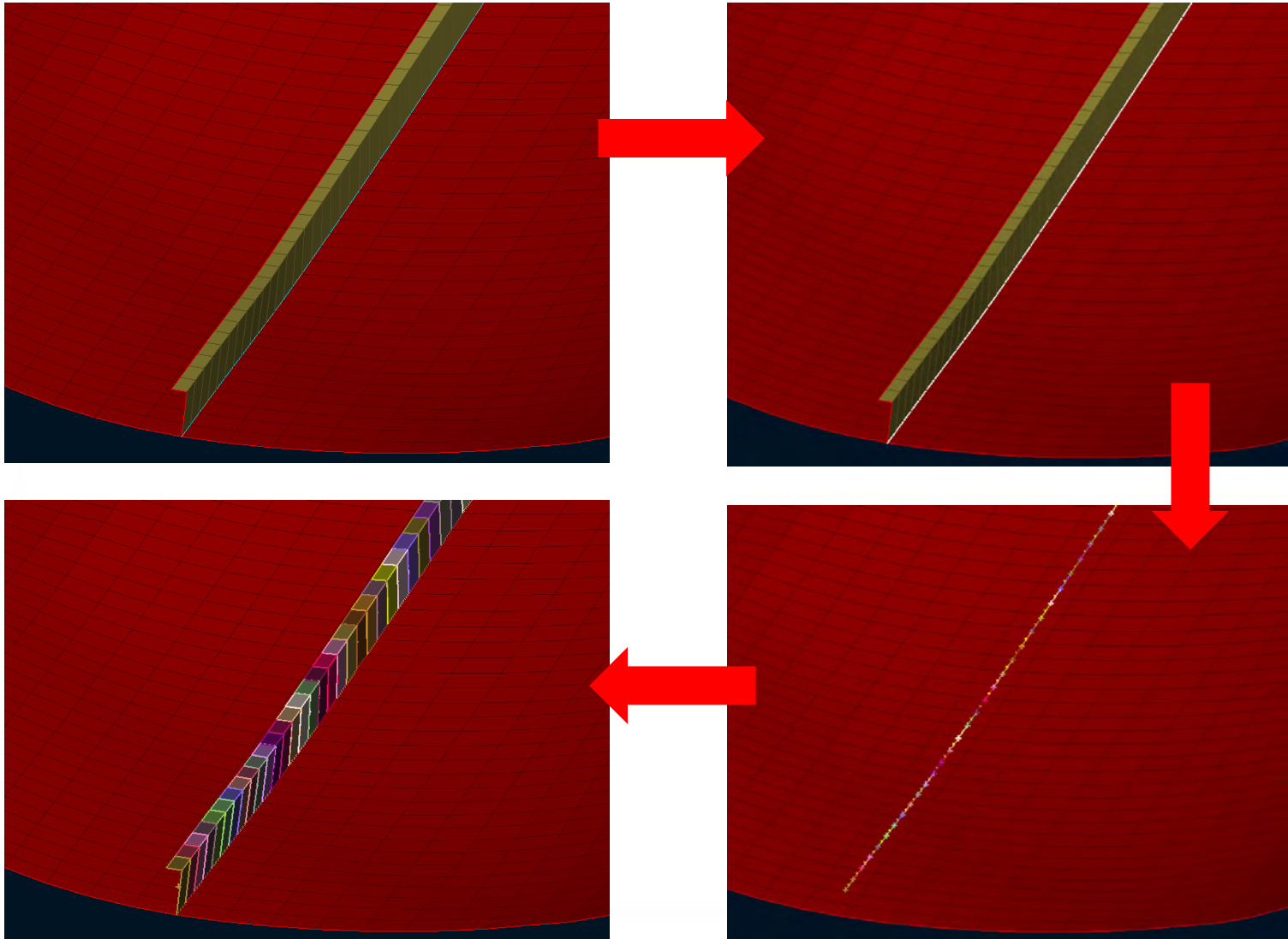
# Stiffeners creation using beams

Creating database of cross sections of any shape



# Stiffeners replacement with beams

Replacement of standard cross section reinforcements “L”, “I”, etc. with beams



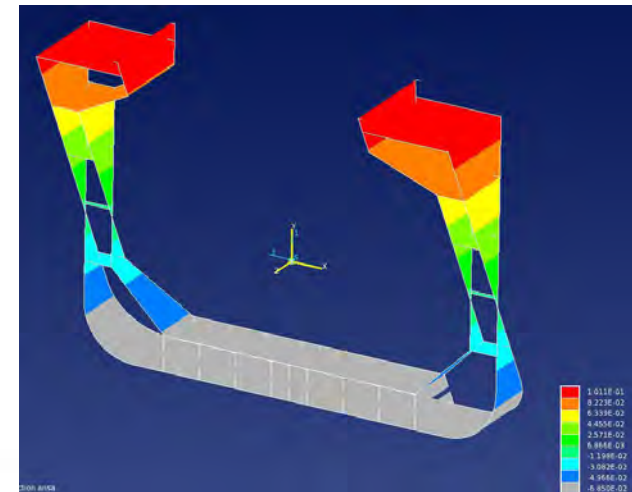
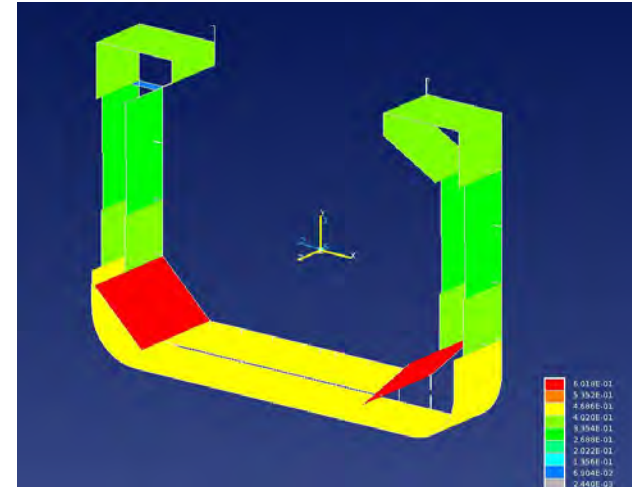
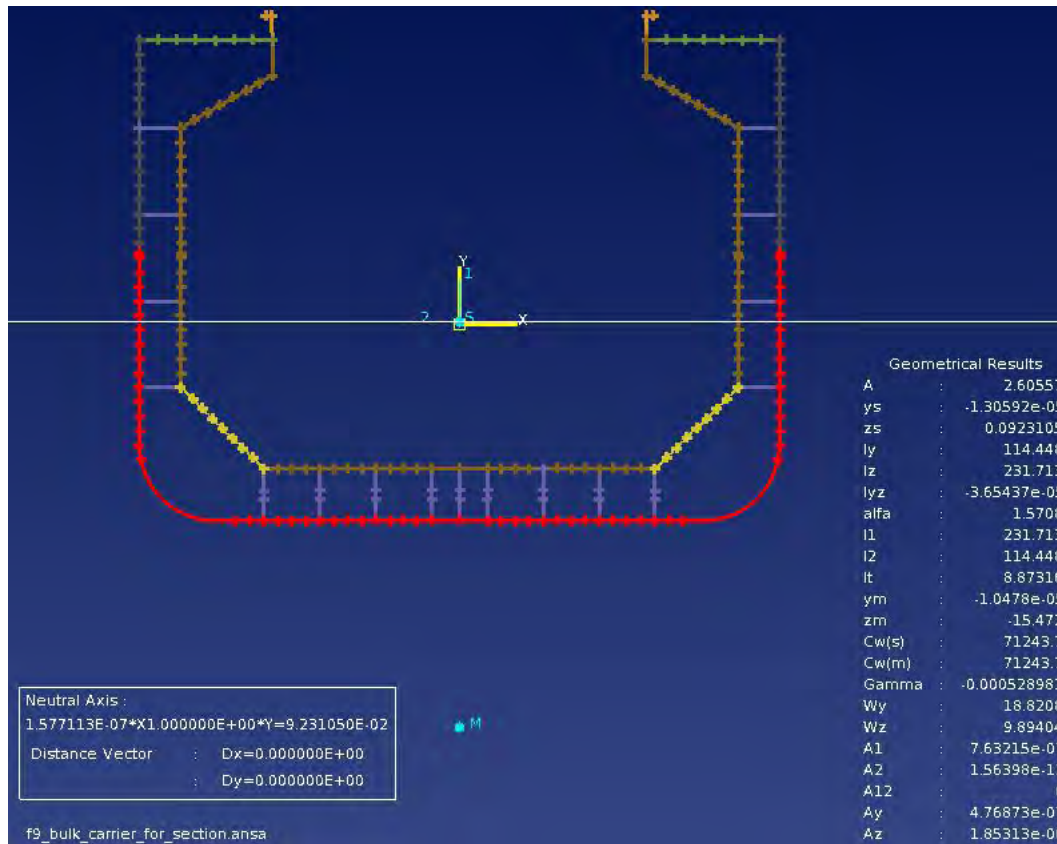


# Calculating Cross Sections

Extraction of Cross Sections from the geometrical model

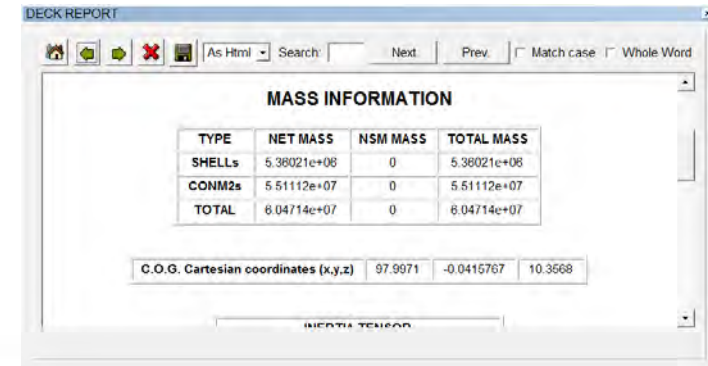
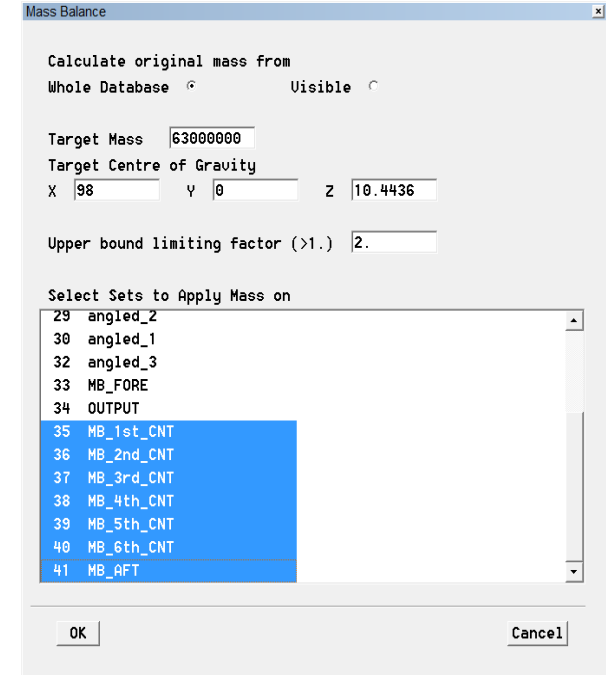
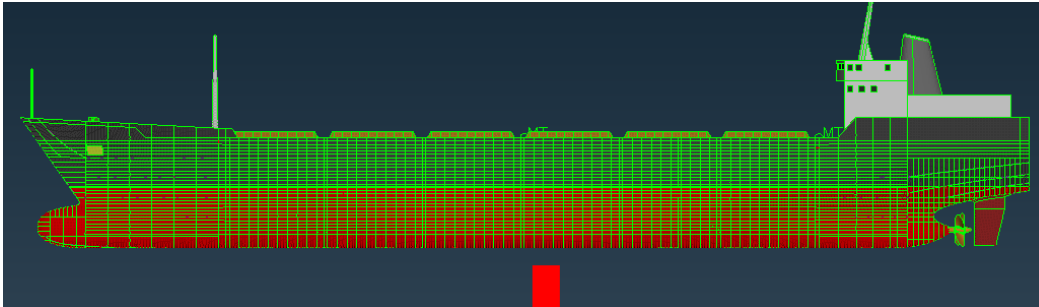
Editing of Cross Sections

Calculating geometrical results (A, Iy, Iz, etc..) neutral axis stresses and moments



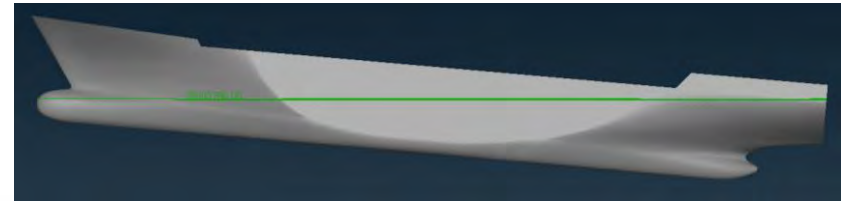
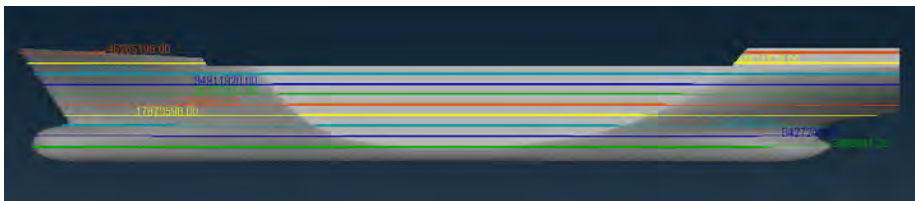
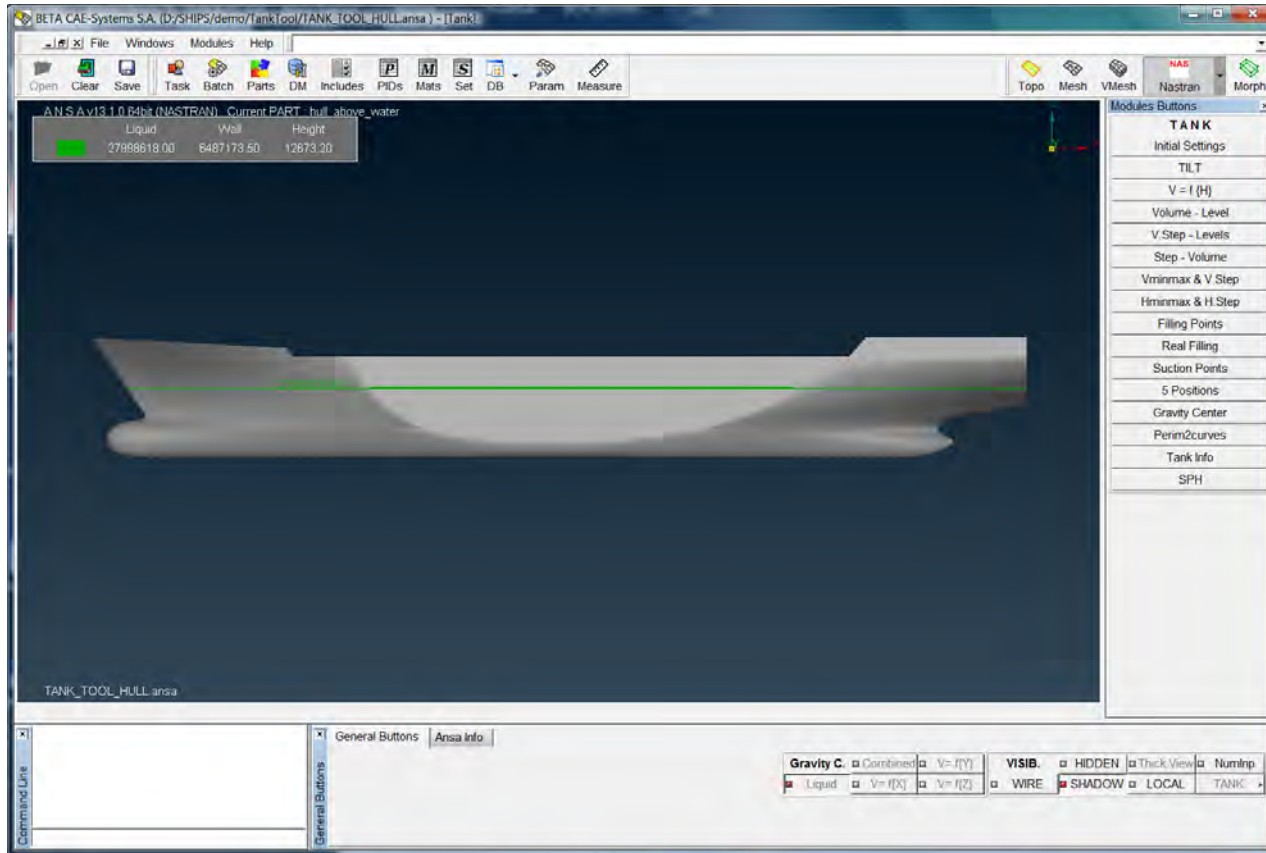
# Mass distribution

Applying additional mass to model by fulfilling balance criteria



# Waterline calculation

Calculating the waterline for variable ship loading



# Wave creation

Sinusoidal or trochoidal wave profile can be defined as a 3D Curve or Face

**BETA CAE-Systems S.A. (/mnt/raid\_disk/titanas/KORBETIS/SHIPS\_Data/f9\_bulk\_carrier.ansa) - [Model View 1]**

File Windows Modules Help mass ba

Open Clear Save Task Batch Parts DM Includes PIDs Mats Set DB Param Measure

Topo Mesh VMesh NAS Nastran Morph

Database Task Manager Filters

Visible Name Number

- ANSAGROUP 4
- ANSAPART 29
- BC 82551
- B.C. SET 12
- COORD 3
- CROSS 3
- EDGE
- ELEMENT 1207073
  - CBEAM 83240
  - CONM2 529970
  - RBE2 2
  - SHELL 593861
  - GEB 3
  - GEB\_MT 3
- GEOMETRY 743007
- GRID 559187
- MATERIAL 4
- MORPH 28
  - Nastran Header 1
- PROPERTY 28
- SET 44

A N S A v13.1.0 64bit (NASTRAN) Current PART : NSM

**Create Wave**

Wave Type

Trochoidal  Sinusoidal

Wave Length 169

Wave Height 8

Number of Periods 2

OK Cancel

f9\_bulk\_carrier.ansa

Command Line: USER>load\_script; USER>trochoidal

General Buttons: FOCUS OR AND ENT GEOME.. D.UTIL ASSE.. REALIZE VISIB... HIDDEN GEOM. PERIMS HOT PNT CURVES MORPH INOT NOT ALL ISOLATE TRANSF CHECK ENT TEMPLATE PID SHADD. FE-Mod SINGLE GRIDs POINTS MEAS CROSH WIRE MACROS DOUBLE SPOTS DWRK. PL. SIZEBOX LOCK UCHECKE. C.PLANE UNMES. DELETE RENUM CONVE.. ERASE CNCTN BOUNDS VOLUM. TRIPLE M.Prt. CRS.SCT C.NODE

Modules Buttons

GRIDs INFO NEW PASTE RELEASE MOVE MATCH ALIGN EXPLODE UTIL DELETE

COORDs INFO CORD1 CORD2 HIERARC..

ELEMENTs INFO CBAR CBEAM CBEND ROD CTUBE CGAP RBE2 RBE3 RBAR CELASI CDAMP1 CBUSH CONMI CMASS CVISC RSSCON RSPLINE CFAST CWELD SHELL SOLID CHACAB CHACBR CAABSF RJOINT UTIL DELETE

P.NODES PLOTEL

B.C. SETs INFO SPC MPC LOAD DLOAD IC NOLIN SUPORT DMIG NSM TEMP HEADER

BCs INFO SPC SUPPORT MPC DMIG PANEL ACMODL PLOADI FORCES DAREA NOLIN GRAV TIC TEMP TEMPP1 TEMPRB NSM RFORCE DOFSetS DELETE

AUXILIARIES COMME SET TABLE BOX CONTACT BCBODY BCPARA SOL200 TOSCA TRANSFORM HARD P C.PLANE BOLT GEB CONNECT. LAMINA. MODALM DISP. M. D.PATCH. APARAM K JOINT K RBODY K CONF AUTOMECH. RES.MAP

# Static Equilibrium

The static position of the ship on the wave is calculated through ANSA scripting

The screenshot displays the ANSA software interface for a static equilibrium analysis of a ship hull. The main window shows a 3D model of the hull above a wave, with a mesh of elements. A red circle highlights the 'BC' (Boundary Conditions) set in the left-hand 'Sets' panel. A 'Balance On Wave' dialog box is open, showing parameters for the analysis: Weight to be balanced (24000000), X coordinate of CoG (125), Weight/Buoyancy difference - % Weight (0.05), CoG/CoB difference - distance (0.3), and Ships Height (21). A 'User Script Buttons' dialog box is also visible, showing a button labeled 'OFFSHORE BalanceOnWaveAssignBuoyancy'. The bottom command line shows the results of the calculation: 'PRECISION ACHIEVED - SUBMERGENCE FOUND | PRECISION ACHIEVED - TRIM ANGLE FOUND | TRIM ANGLE = 2.25 degrees'. Red arrows and a circle are overlaid on the image to highlight key elements: a red circle around the 'BC' set, a red circle around the command line output, and red arrows pointing to the 'Balance On Wave' dialog box and the ship model.

**Balance On Wave**

Weight to be balanced | 24000000  
 X coordinate of CoG | 125  
 Weight/Buoyancy difference - % Weight | 0.05  
 CoG/CoB difference - distance | 0.3  
 Ships Height | 21

**User Script Buttons**

OFFSHORE BalanceOnWaveAssignBuoyancy  
 CreateWave

**Command Line**

```

Move ANGLE has been divided by 2, new ANGLE = 0.375
.....
| PRECISION ACHIEVED - SUBMERGENCE FOUND |
| PRECISION ACHIEVED - TRIM ANGLE FOUND | TRIM ANGLE = 2.25 degrees
  
```

# Tanks Loading

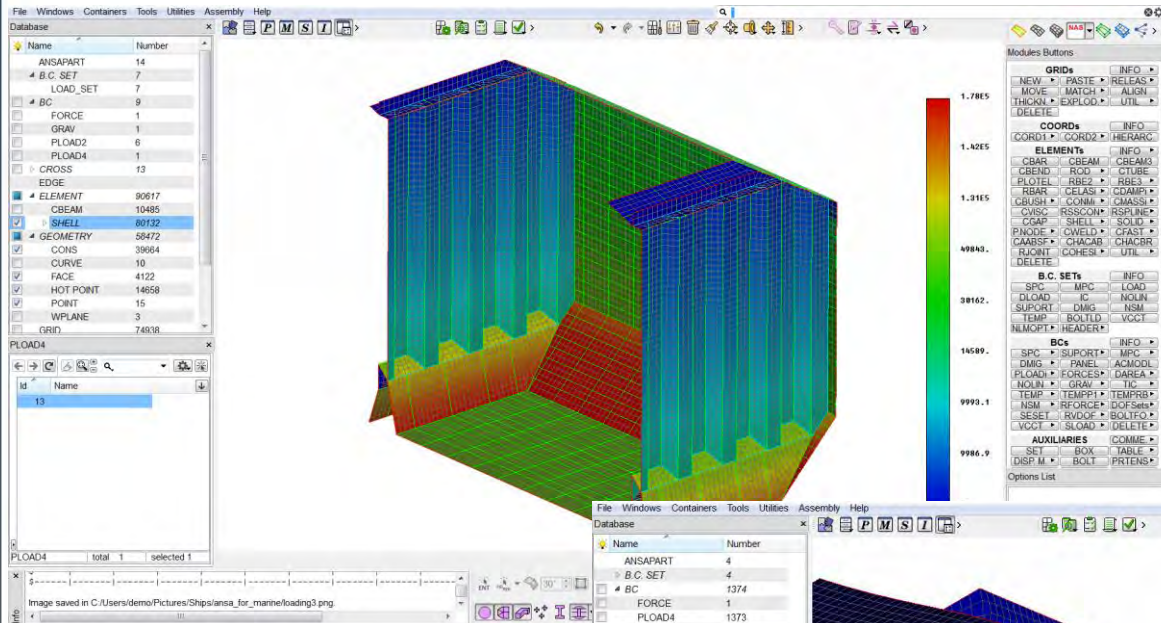
Tanks recognition and cargo load definition using Pressure Loads

The screenshot displays the BETA CAE Systems S.A. software interface for a ship's hull model. The main window shows a 3D view of the hull with a mesh overlay. A color scale on the right indicates pressure values, ranging from 0 (blue) to 1.55E8 (red). The interface includes several panels and dialog boxes:

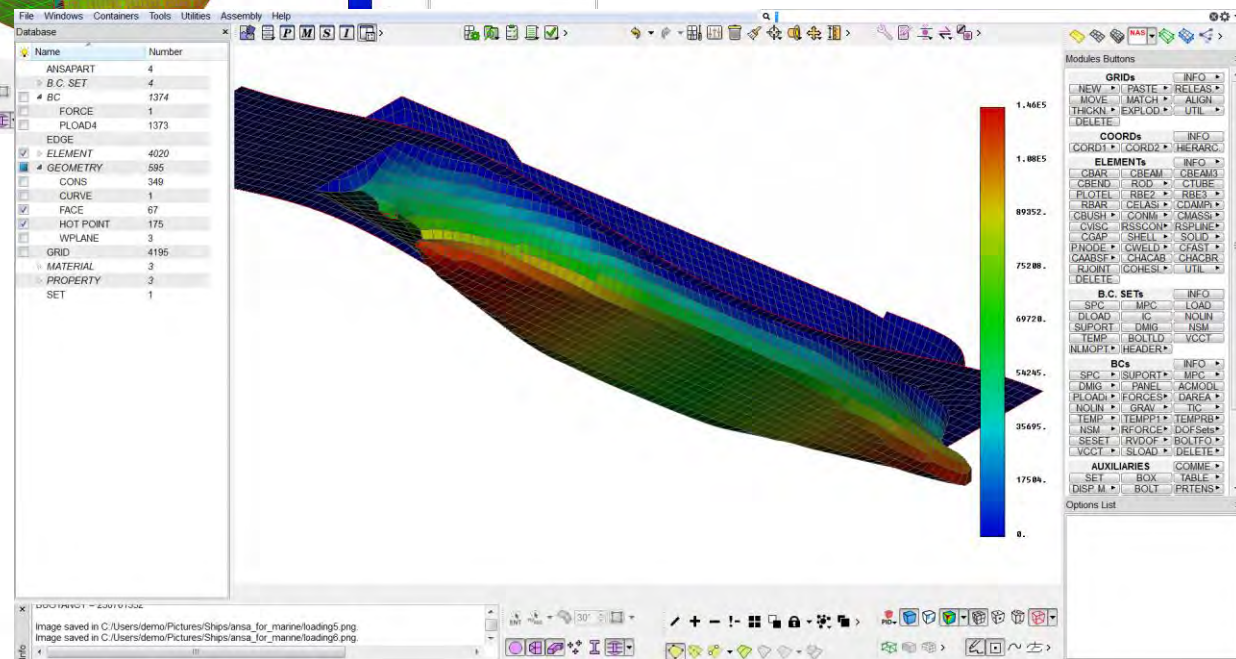
- Database:** Lists various entities such as ANSPART (39), B.C. SET (1), BC (9596), EDGE, ELEMENT (32256), GEOMETRY (7517), GRID (31277), LOCK\_VIEW (1), MATERIAL (34), PROPERTY (36), SET (42), and VOLUME (12).
- PROPERTY:** Shows the current part as 'Auto Detected Volume04'.
- TopWindow:** A dialog box for defining tanks. It contains a table with columns 'tank', '%level', and 'density'.
 

tank	%level	density
5	70	1024
6	70	1024
7	70	1024
8	70	1024
9	70	1024
10	70	1024
11	70	1024
total	0	selected 0
- Modules Buttons:** A panel with various function buttons categorized into HOT POINTS, PERIMETERs, MACROs, GRIDs, MESH GEN., SHELL MESH, and ELEMENTs.

# Pressure loads application

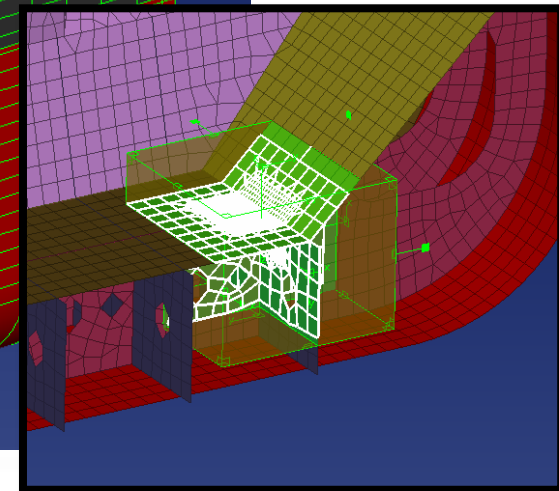
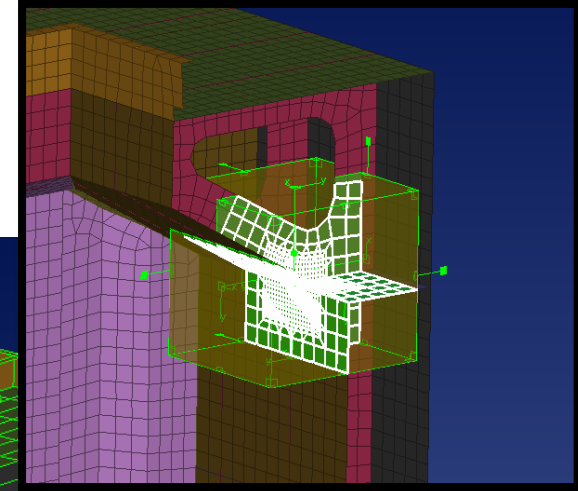


- Pressure loads definition in SETs using expressions
- Buoyancy application for any water profile



# 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



**SUBSTRUCTURE ANALYSIS CREATOR**

Id: 1  
Name: Substructure\_1  
Select Eigenmodes:   
Frequency Step:  Edit  
Substructure Step: 2 Edit  
ELEMENT, TYPE = Z 1  
Position tolerance: 0. Substructure Property: 0

Define 'User Area volume' with planes:

- 1|Anonymous Cutting Plane 1
- 2|Anonymous Cutting Plane 2
- 3|Anonymous Cutting Plane 3
- 4|Anonymous Cutting Plane 4
- 5|Anonymous Cutting Plane 5
- 6|Anonymous Cutting Plane 6

Add Remove New

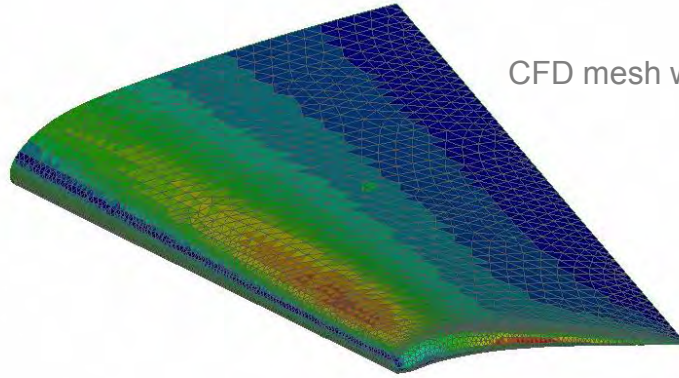
Additional Retained Nodal DOFS set: 0  
Put unconnected parts in 'Substructure Area':   
Preview Check 1D elements

OK Cancel

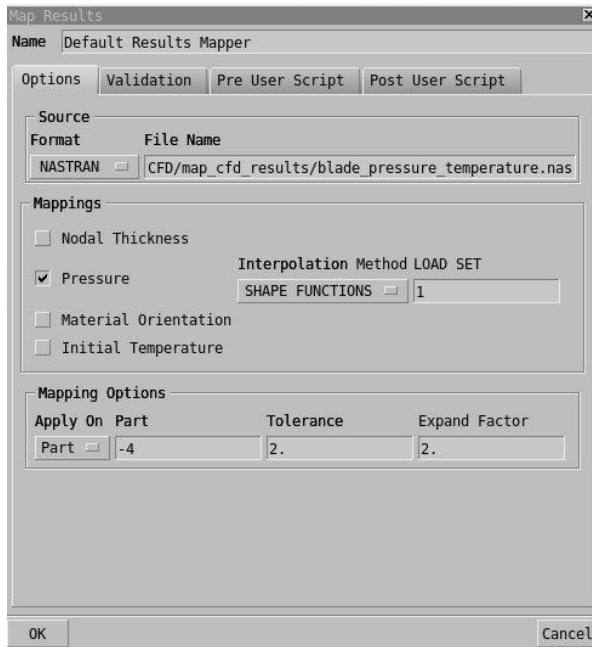
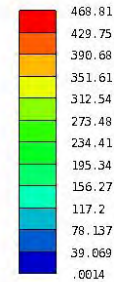


# Mapping CFD results to FEA models

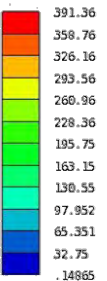
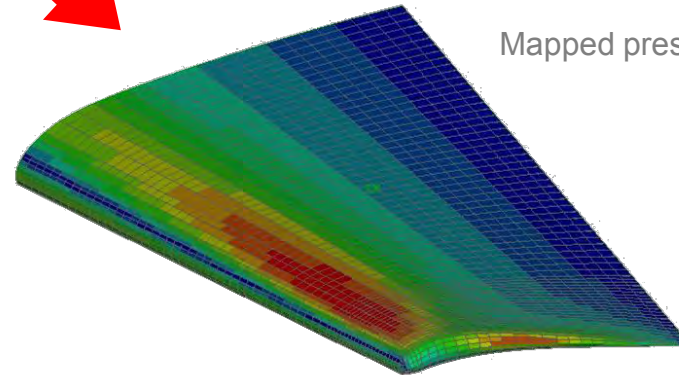
Map pressure or temperature results from a CFD simulation to an FEA model



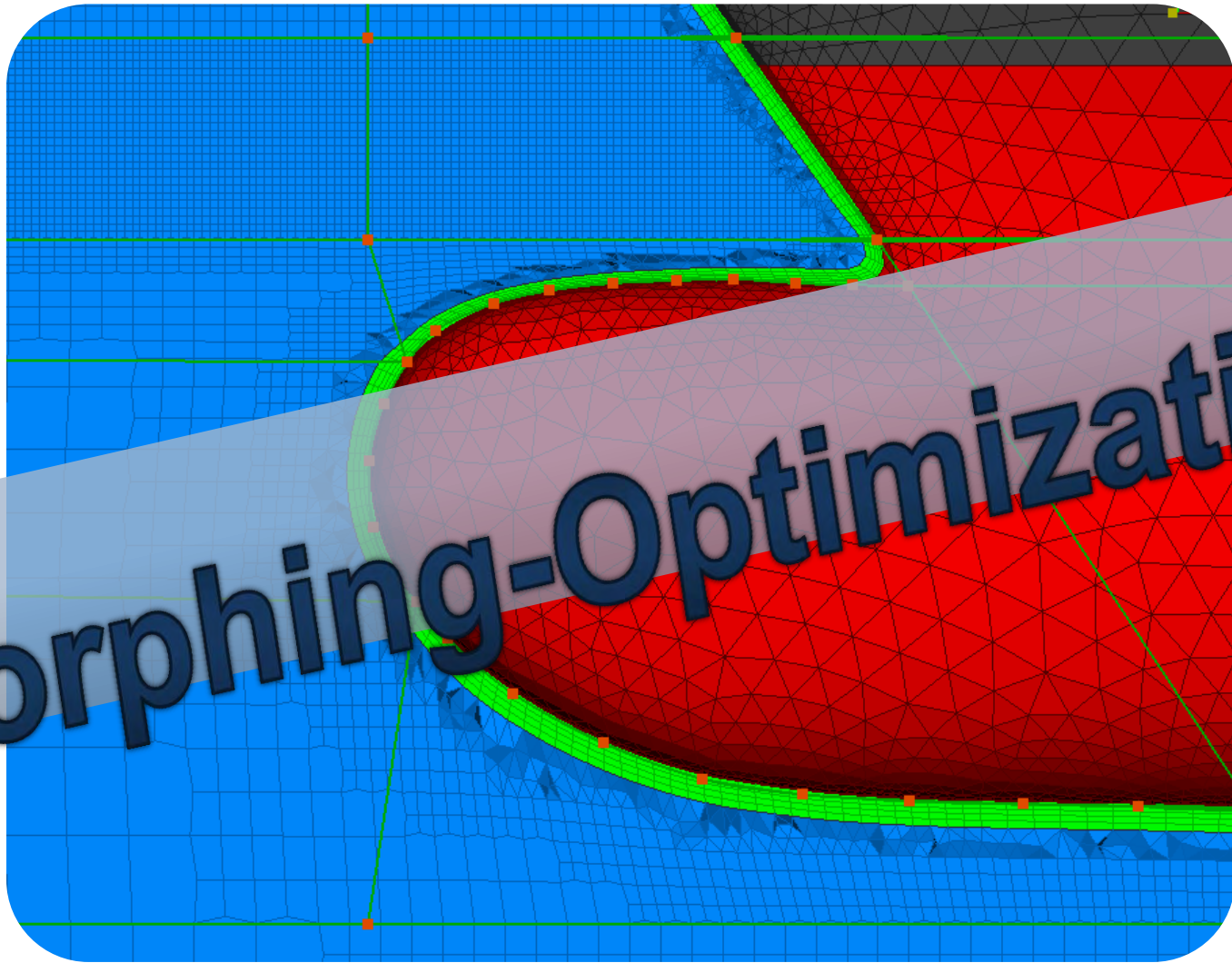
CFD mesh with pressure results



Mapped pressures on FEA mesh

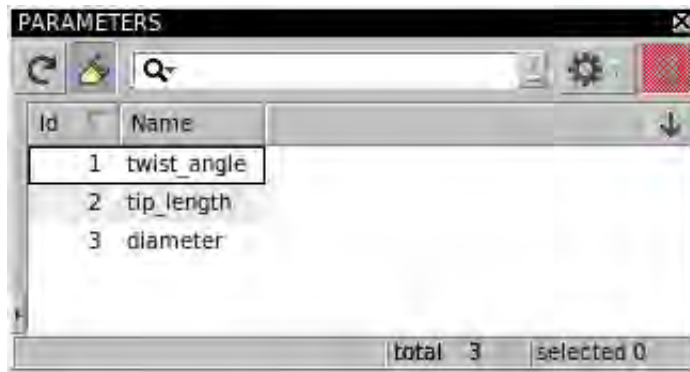
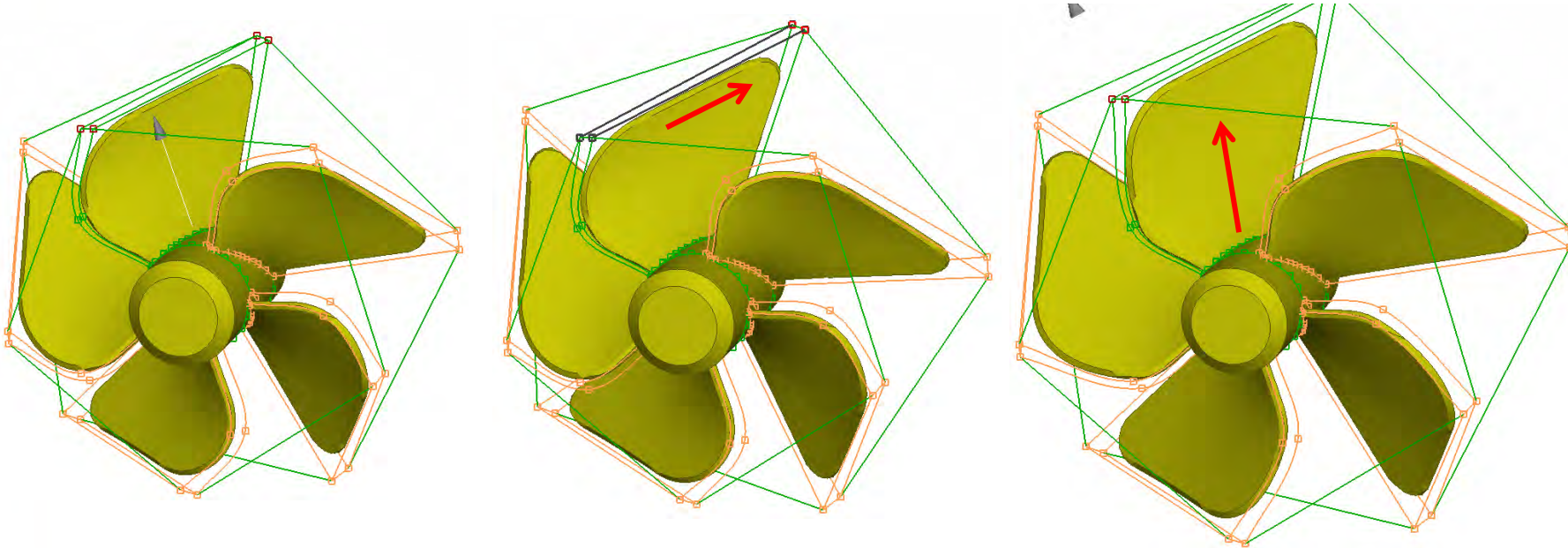


# Morphing-Optimization



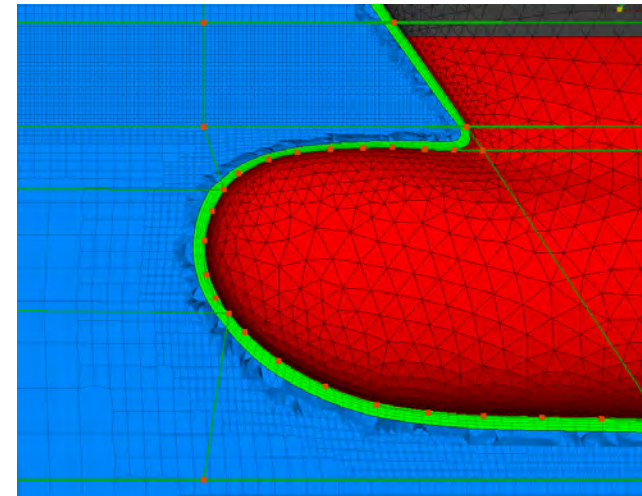
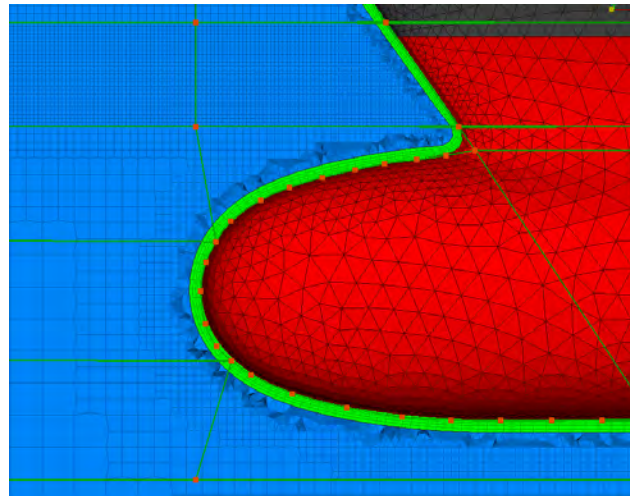
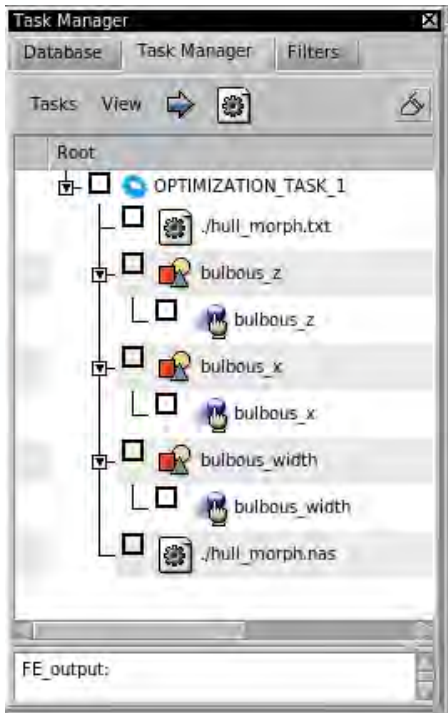
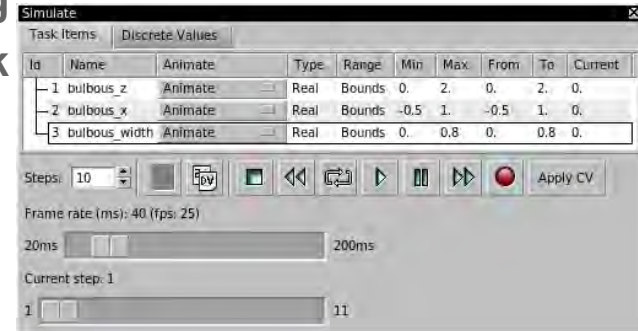
# Shape optimization using the Morphing Tool

Parametric morphing applied on FE or geometry



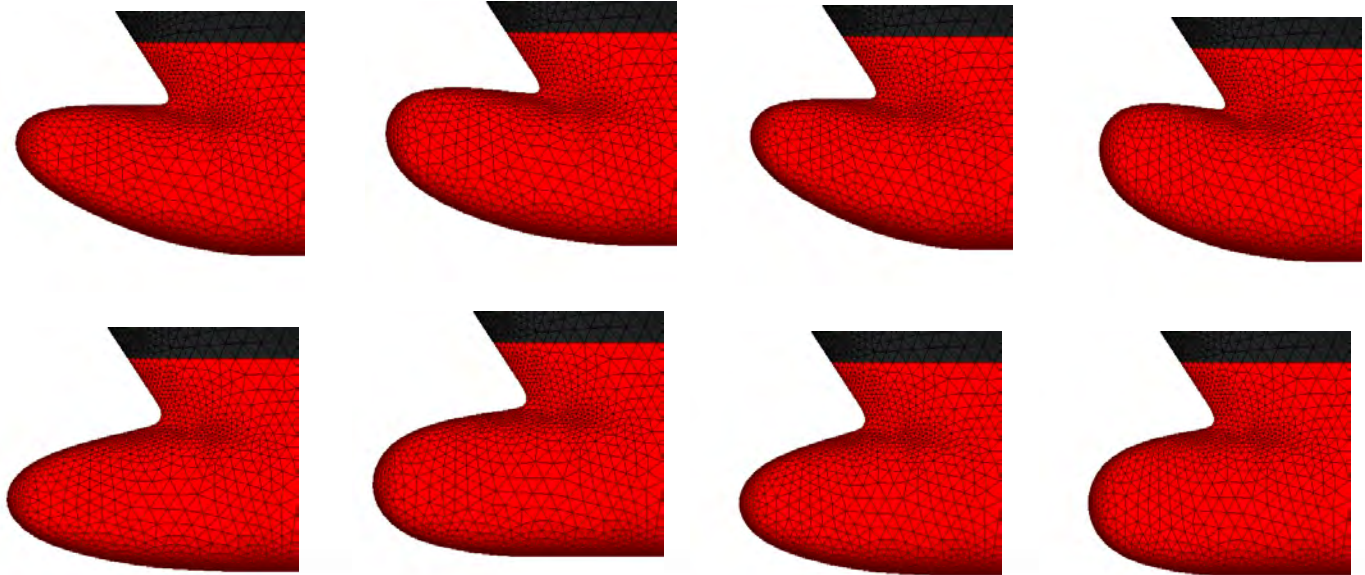
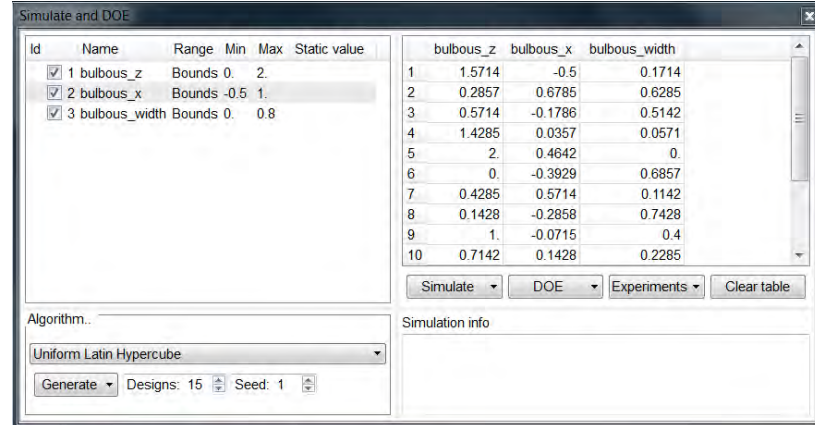
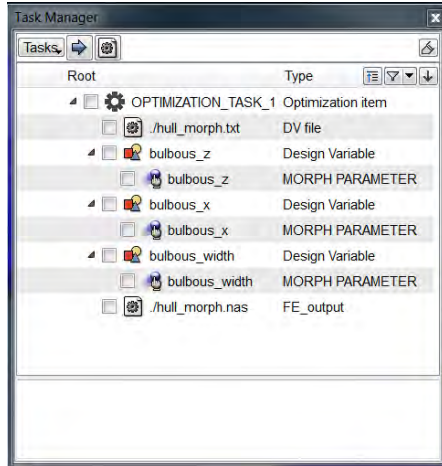
# Shape optimization of the bulbous bow

- 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



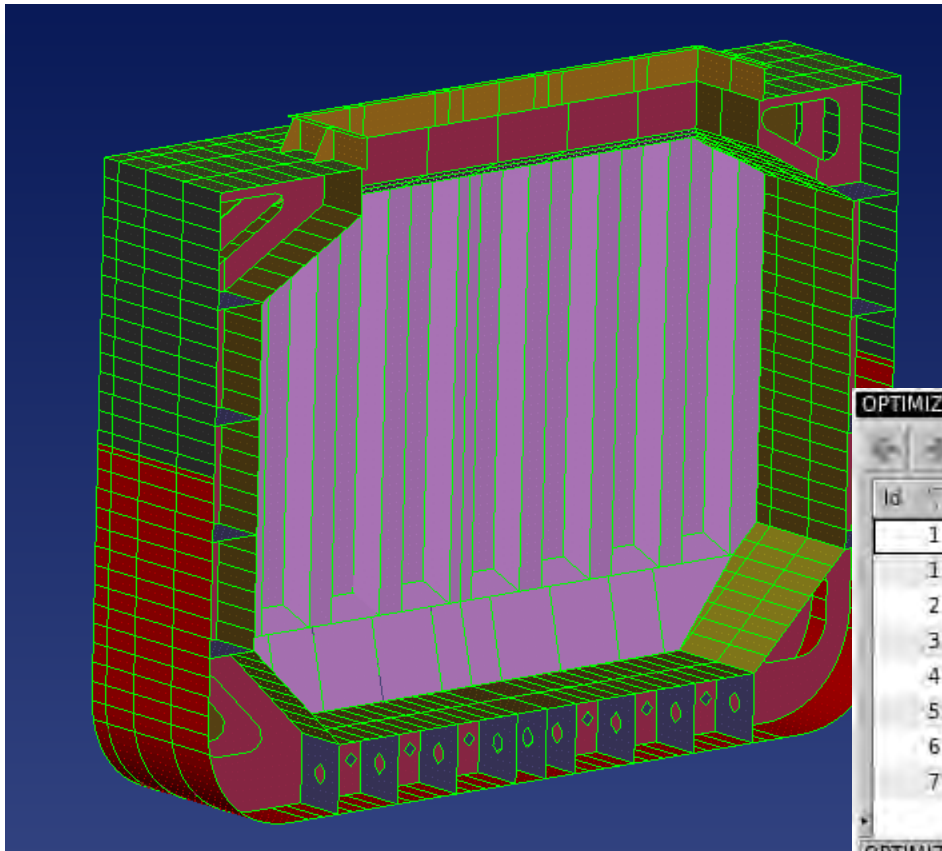
# Shape optimization of the bulbous bow

## Definition of Design of Experiments



# Shell thickness optimization

Automatic definition of design variables for shell thickness and creating 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	Selected
NUMBERING_RULE_NAME	<input type="checkbox"/>
PID	<input type="checkbox"/>
STAR_ID	<input type="checkbox"/>
T	<input checked="" type="checkbox"/>
TRANSPARENCY	<input type="checkbox"/>
TS/T	<input type="checkbox"/>

OPTIMIZATION TASK

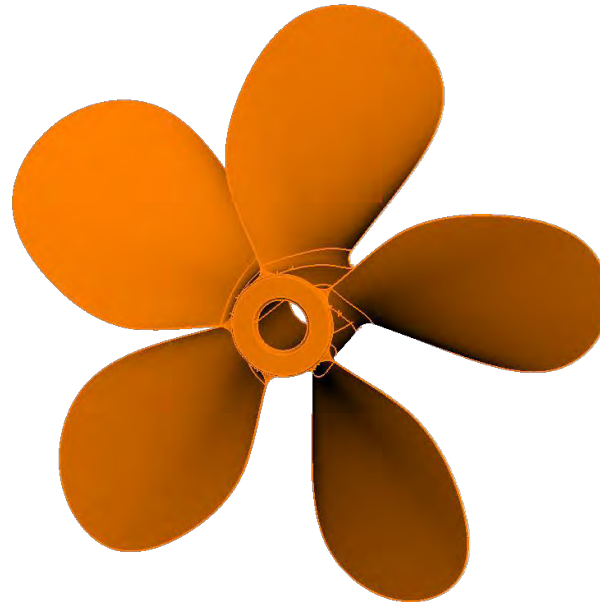
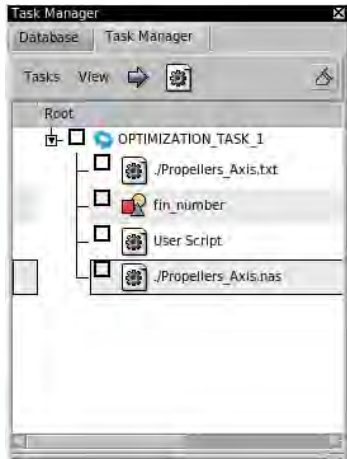
Id	Name	Min Value	Current Value	Max Value
1	OPTIMIZATION_TASK_1			
1	PSHELL_20_T	0.01125	0.0125	0.01375
2	PSHELL_28_T	0.01125	0.0125	0.01375
3	PSHELL_21_T	0.016199...	0.018	0.0198
4	PSHELL_29_T	0.016199...	0.018	0.0198
5	PSHELL_18_T	0.01035	0.0115	0.01265
6	PSHELL_6_T	0.016199...	0.018	0.0198
7	PSHELL_15_T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

OPTIMIZATION\_TASK total 15 selected 0

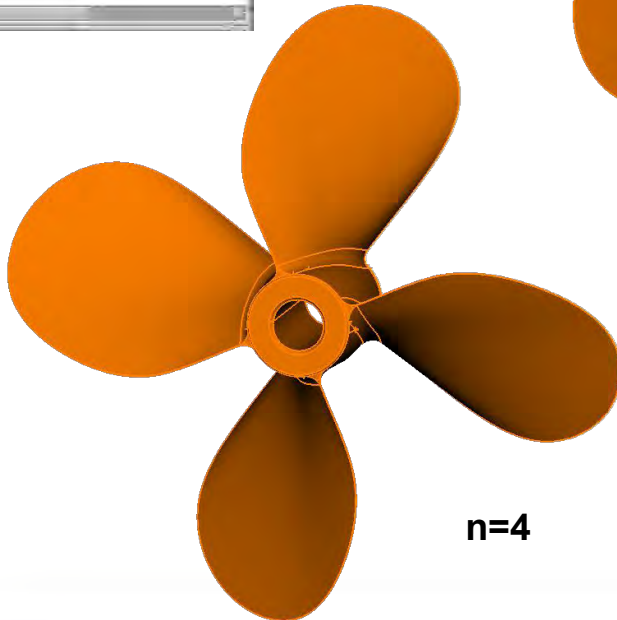
# Creating / importing features parametrically

Defining the fin number of a propeller as design variable

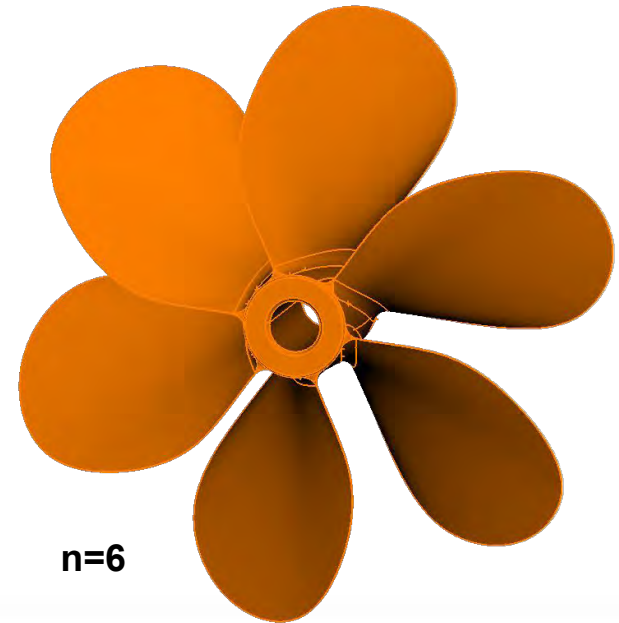
A new shape is created just by changing the design variable value



**n=5**

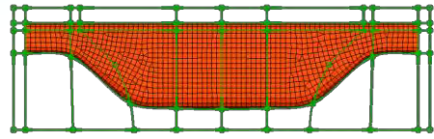


**n=4**



**n=6**

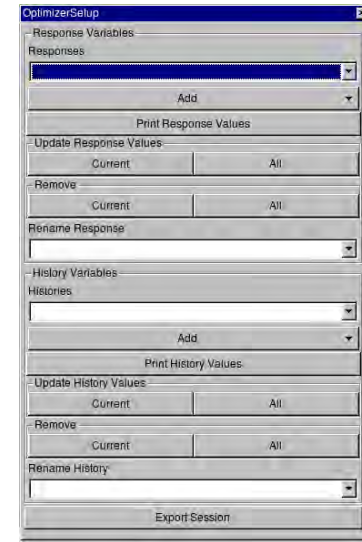
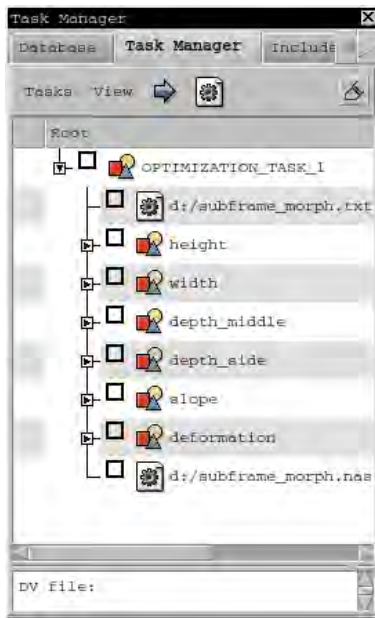
# Coupling ANSA and $\mu$ ETA to parametric optimizers



**OPTIMIZER**

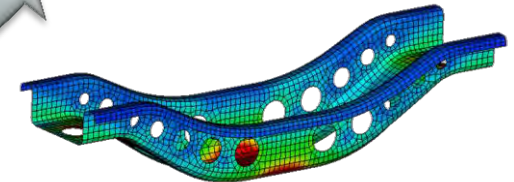
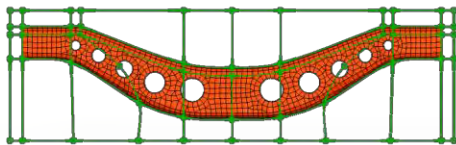
LS-OPT  
modeFRONTIER  
Optimus  
Isight  
HEEDS

**ANSA Optimization Task**

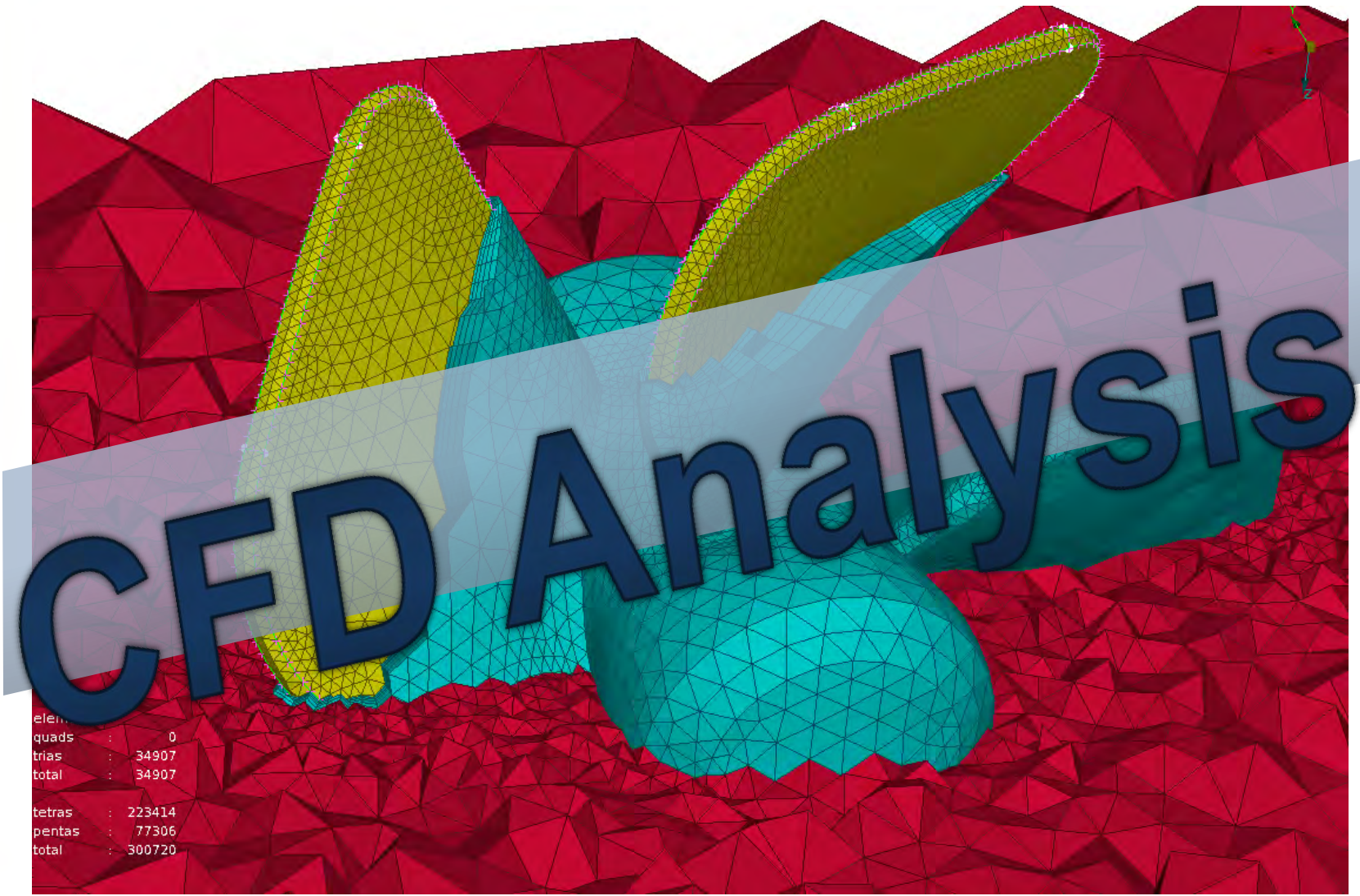


**$\mu$ ETA OptimizerSetup**

**SOLVER**





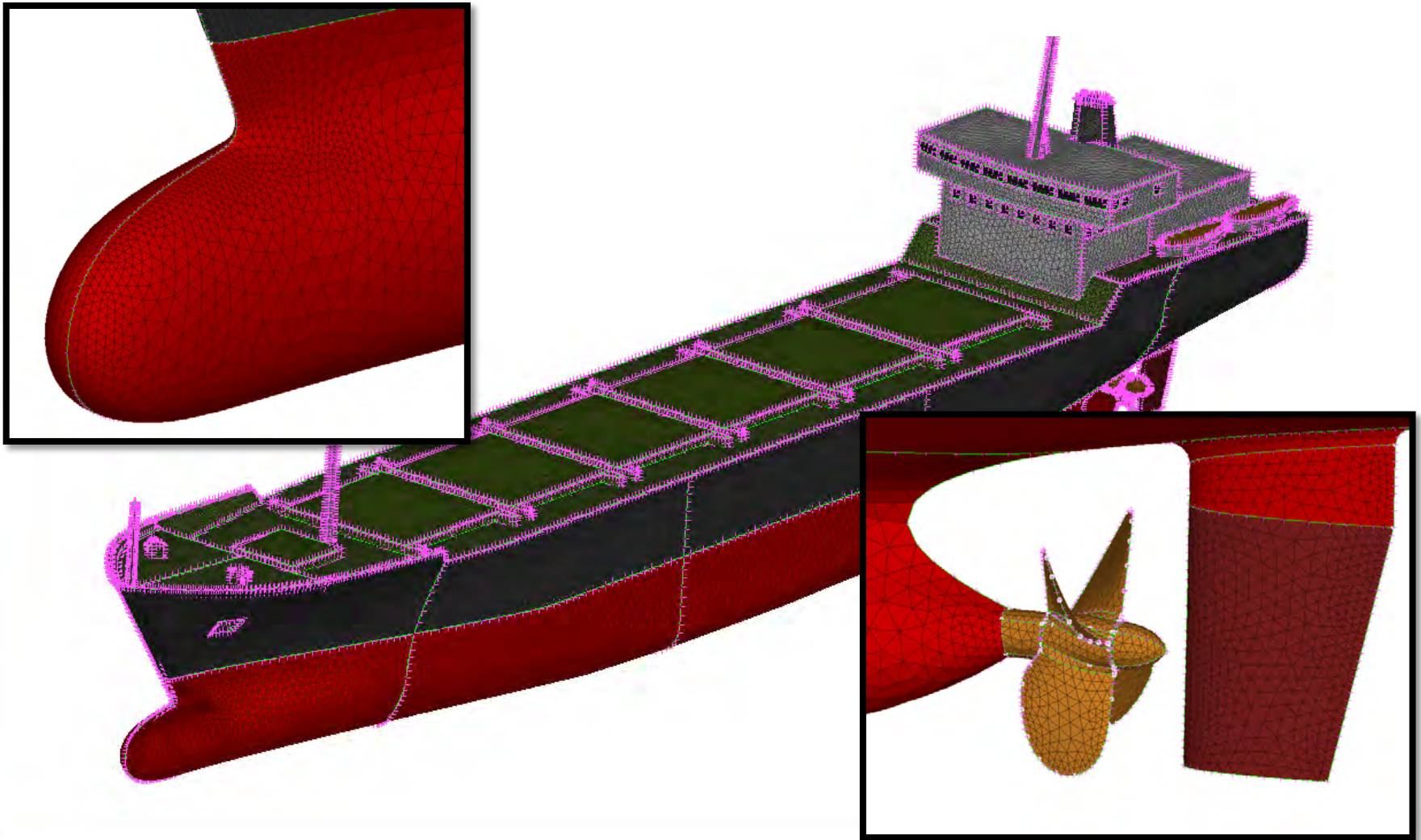


# CFD Analysis

```
elem  
quads : 0  
trias : 34907  
total : 34907  
  
tetras : 223414  
pentas : 77306  
total : 300720
```

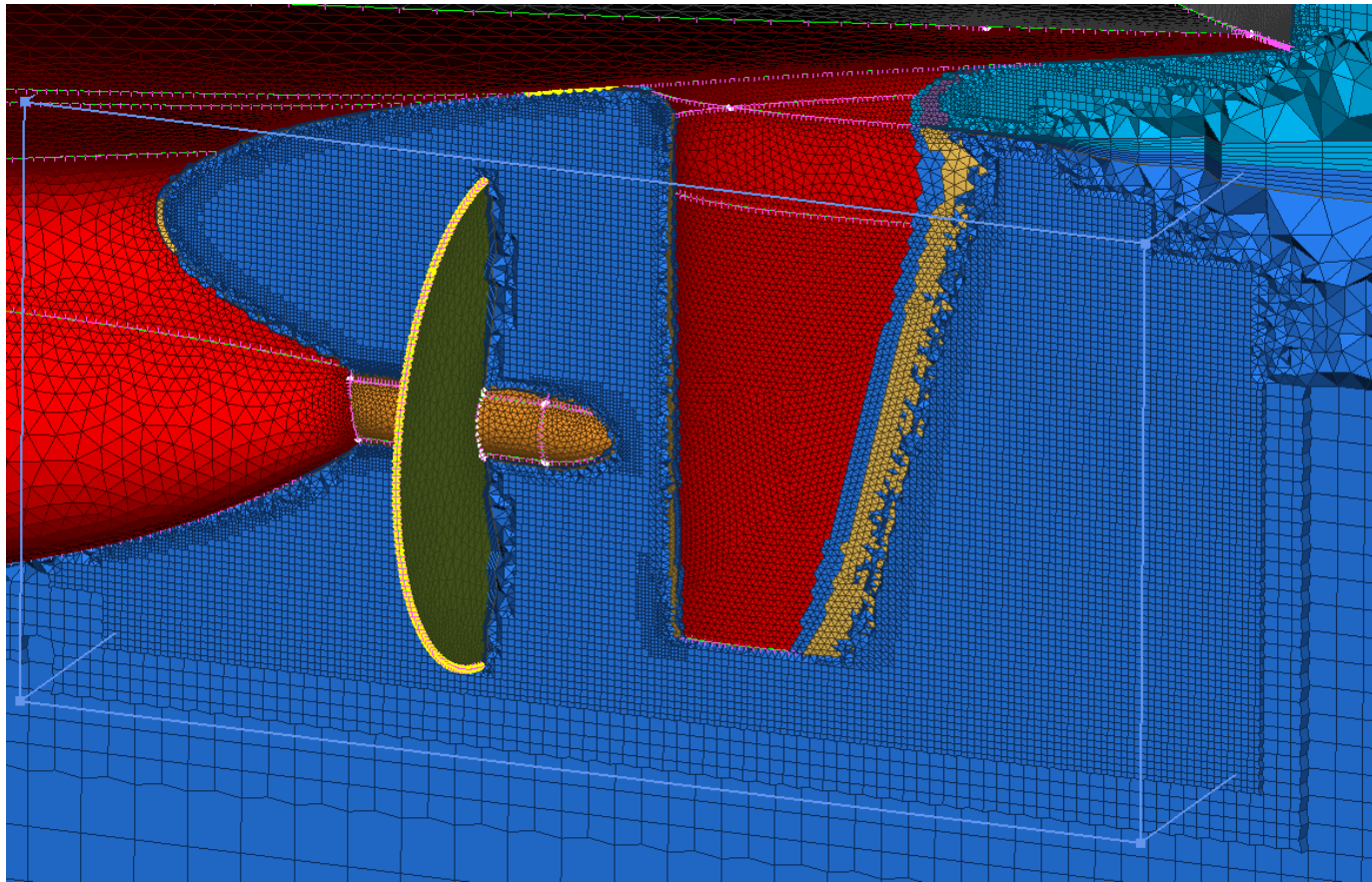
# Model definition for CFD analysis

Fully automatic Curvature Dependant surface meshing with user controlled growth rate, min & max element size and mesh feature angle



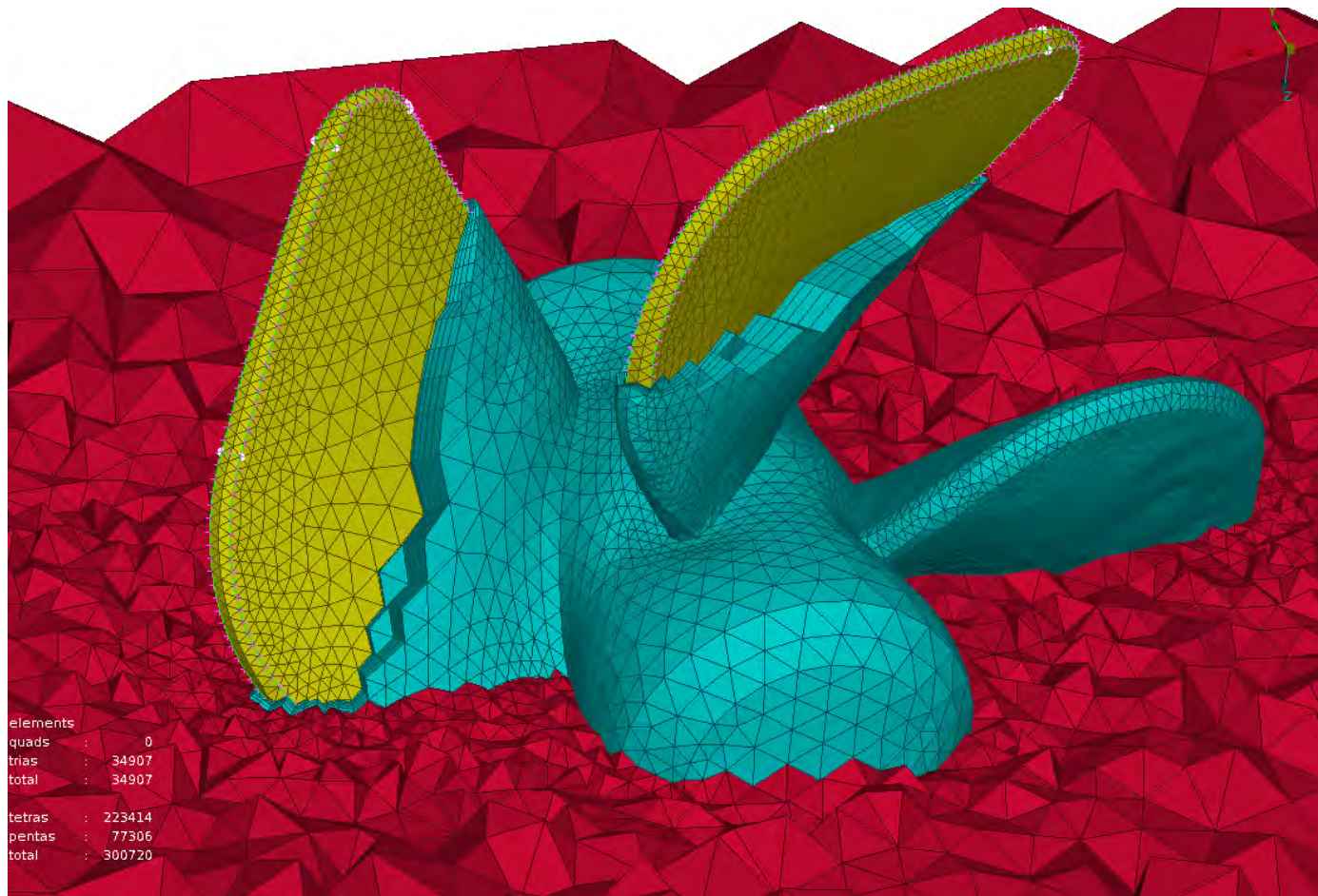
# Model definition for CFD analysis

## Local refinement using SIZE BOXES



# Boundary layers generation

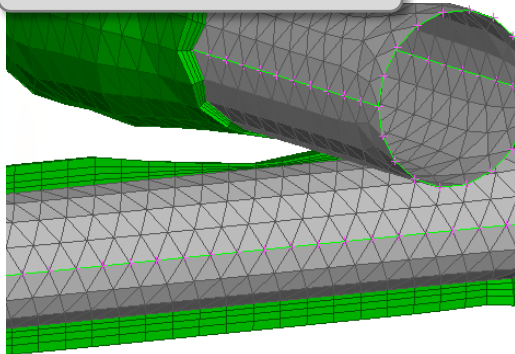
- Auto exclude or collapse areas
- Controlled Layer Squeezing to avoid intersections
- Layers from selected areas with different settings
- Layers from zero-thickness walls



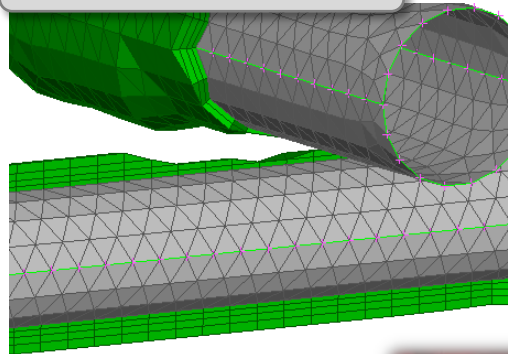
# CFD Analysis

## Advanced boundary layers generation

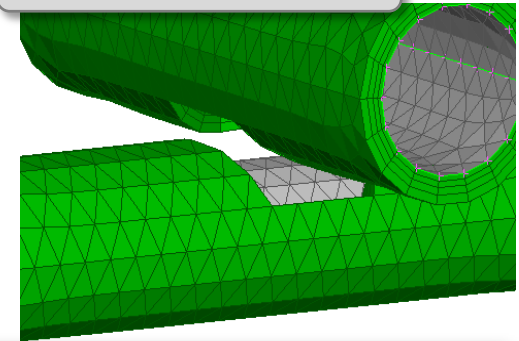
**Squeeze**



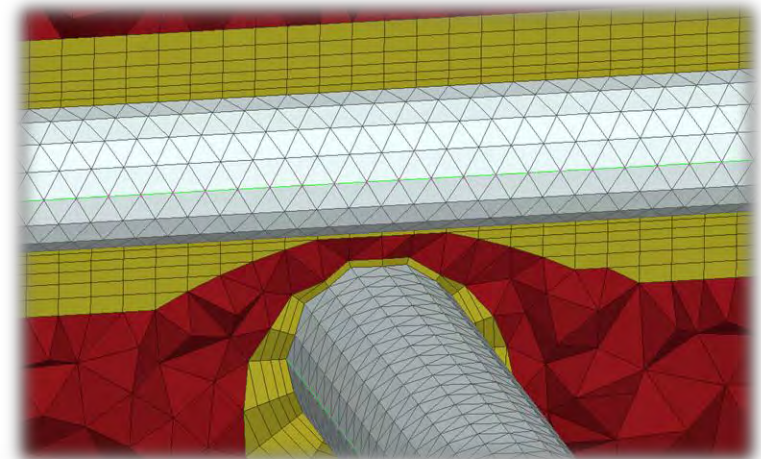
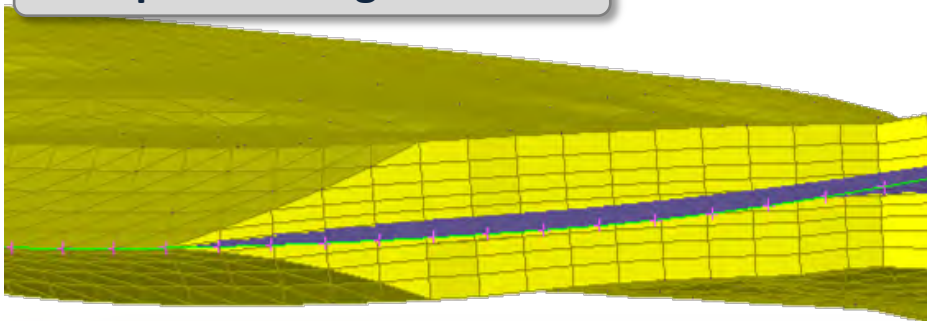
**Collapse**



**Exclude**



**Collapse free edges**



# Boundary Condition type specification for various CFD solvers

The screenshot displays the ANSYS 13.2.0 interface with a meshed propeller model. A dialog box titled 'FACE\_ZONE [SHELL\_PROPERTY]' is open, showing the following configuration:

ZONE_ID	ZONE_TYPE
4	wall
propeller_surface	axis

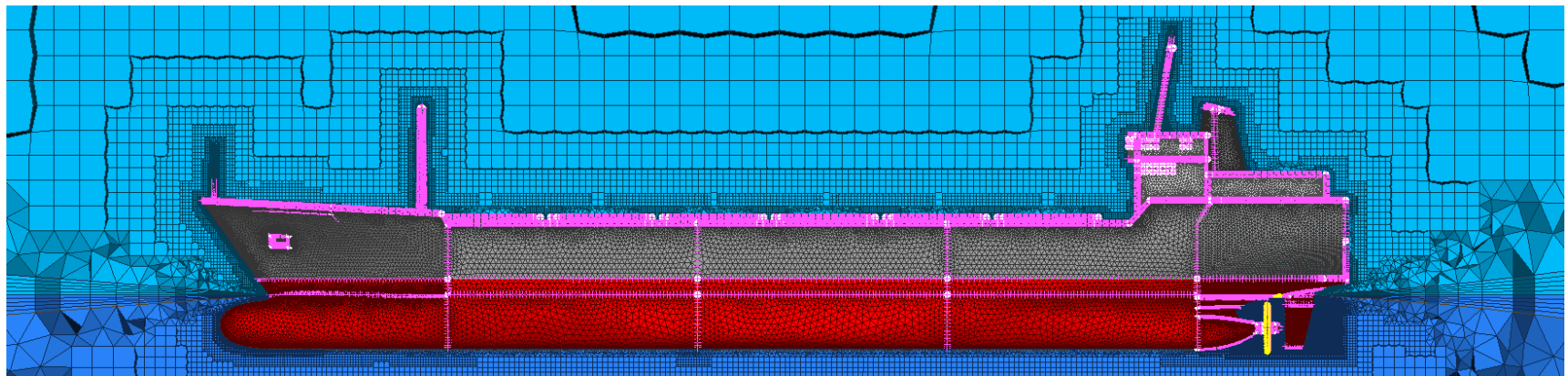
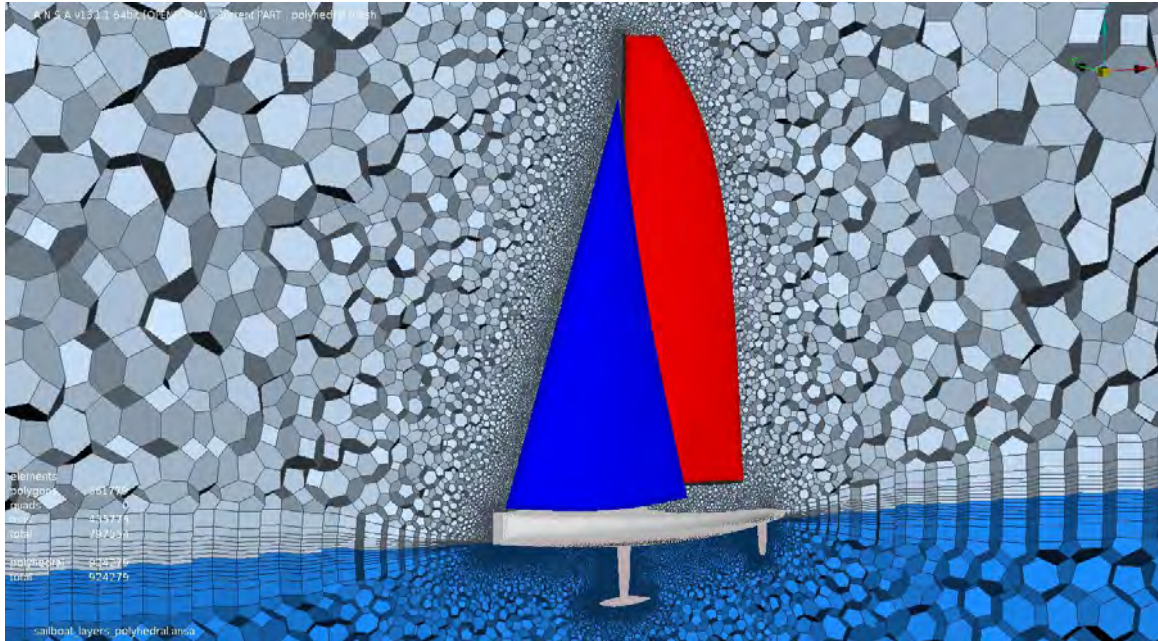
The 'PROPERTIES' table in the foreground shows the following data:

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

The 'FACE\_ZONE' dialog also includes fields for Name (propeller\_surface), FROZEN\_ID (NO), FROZEN\_DELETE (NO), DEFINED (YES), TRIM (YES), and USE\_IN\_MODEL (YES). A dropdown menu is open, listing various boundary condition types such as axis, exhaust-fan, fan, inlet-vent, intake-fan, interface, interior, mass-flow-inlet, outflow, outlet-vent, periodic, periodic-shadow, porous-jump, pressure-far-field, pressure-inlet, pressure-outlet, radiator, symmetry, velocity-inlet, and wall.

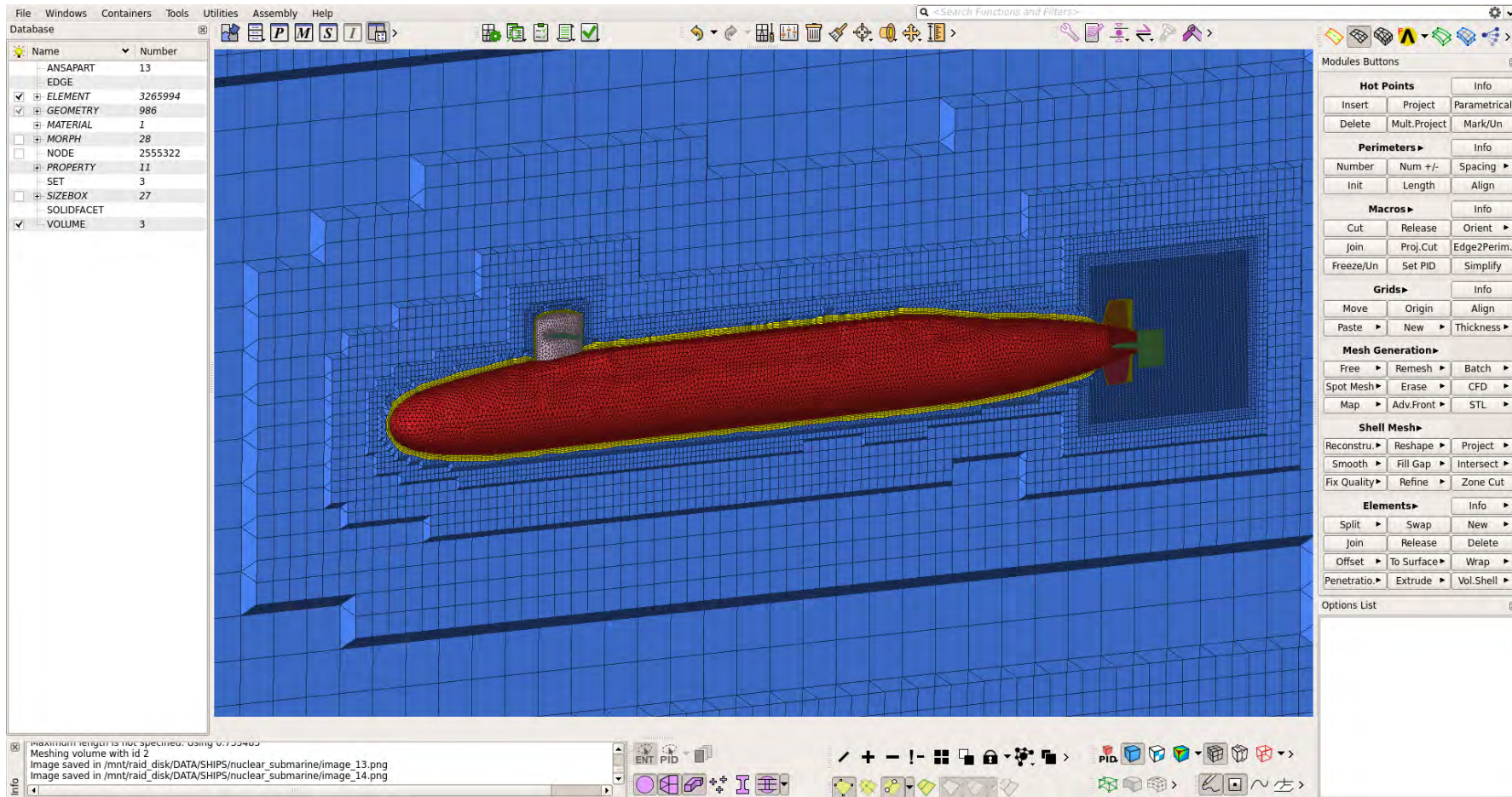
# Model definition for CFD analysis

Fast and robust volume meshing for all types of elements (tetra, pyramid, prism, hexa and polyhedron)



# Model definition for CFD analysis

Fast and robust volume meshing for all types of elements (tetra, pyramid, prism, hexa and polyhedron)





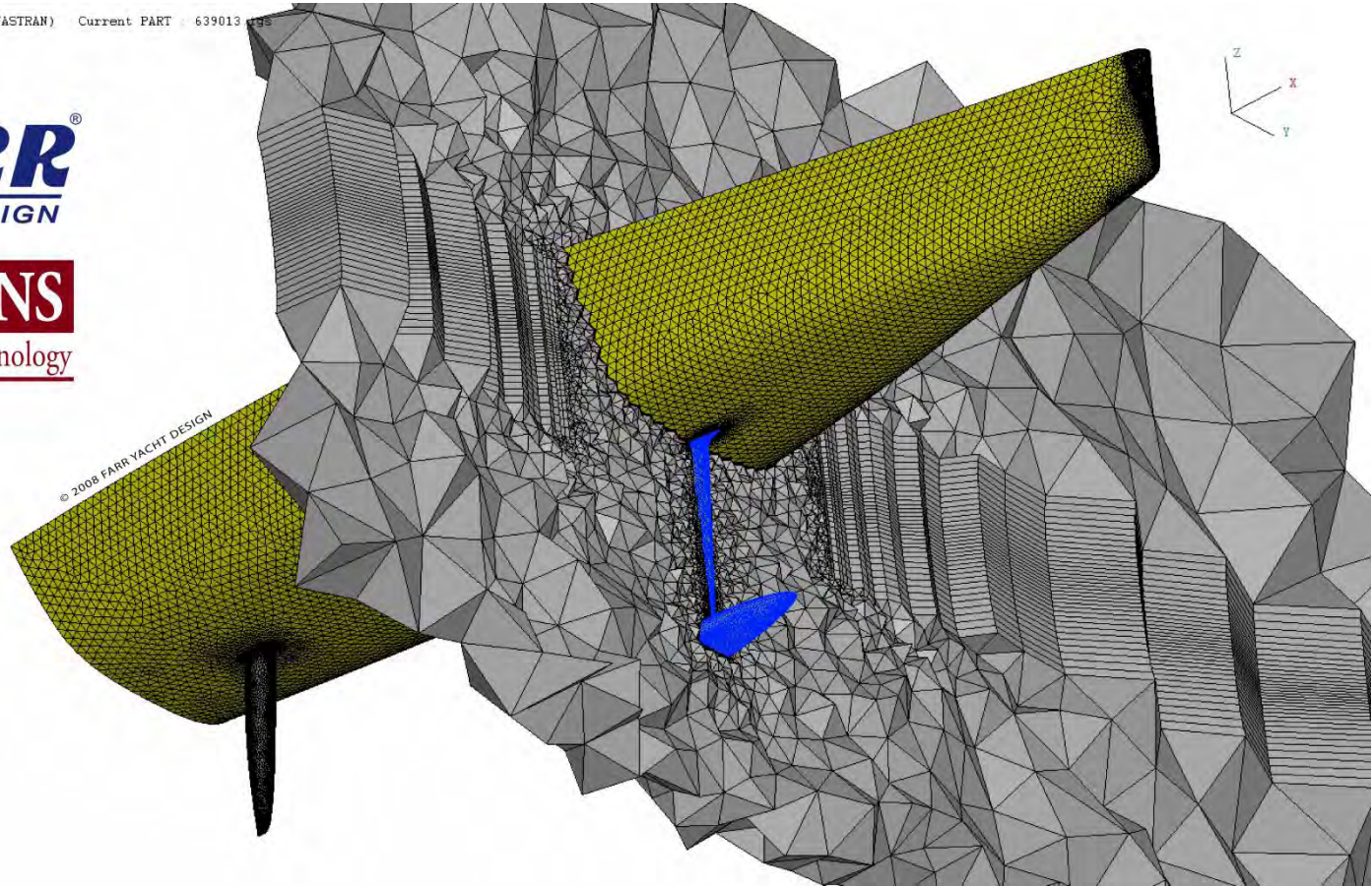
# Model definition for CFD analysis

Fast and robust volume meshing for all types of elements (tetra, pyramid, prism, hexa and polyhedron)

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

**FARR**<sup>®</sup>  
YACHT DESIGN

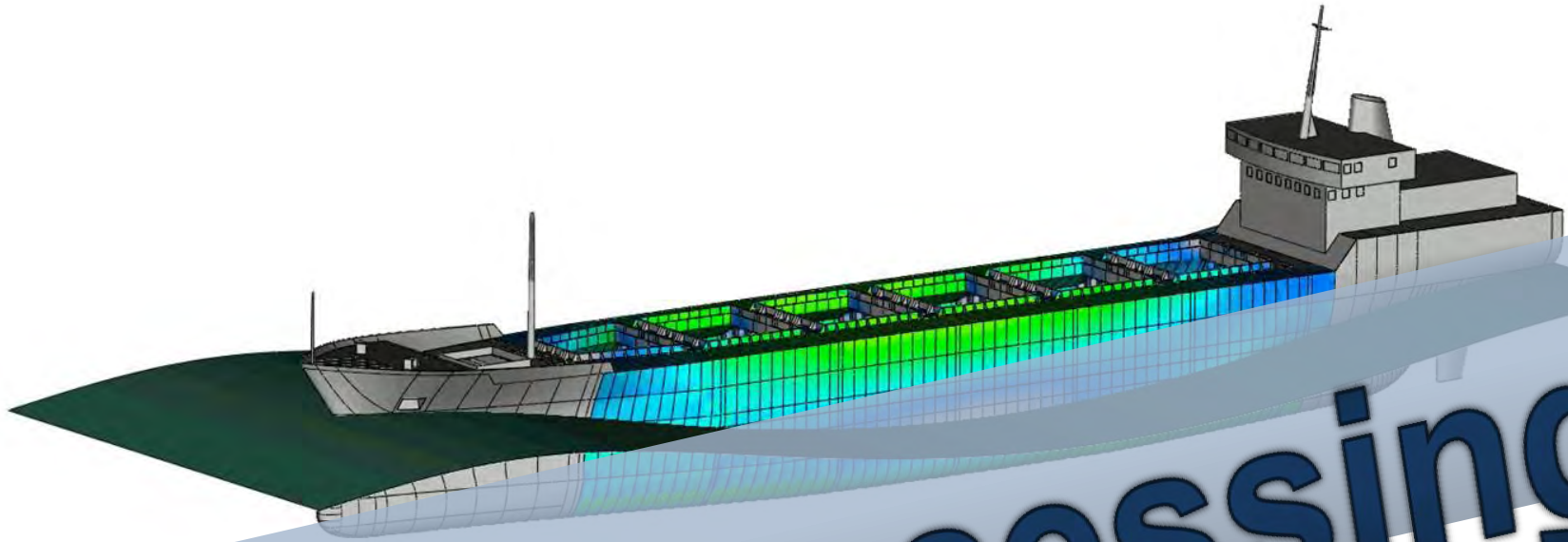
**STEVENS**  
Institute of Technology



elements  
quads : 0  
trias : 62456  
total : 62456  
  
tetras : 235424  
pentas : 253385  
pyramid : 1107  
total : 489916

639013\_geom.21.ansa

*Courtesy of Farr Yacht Design*

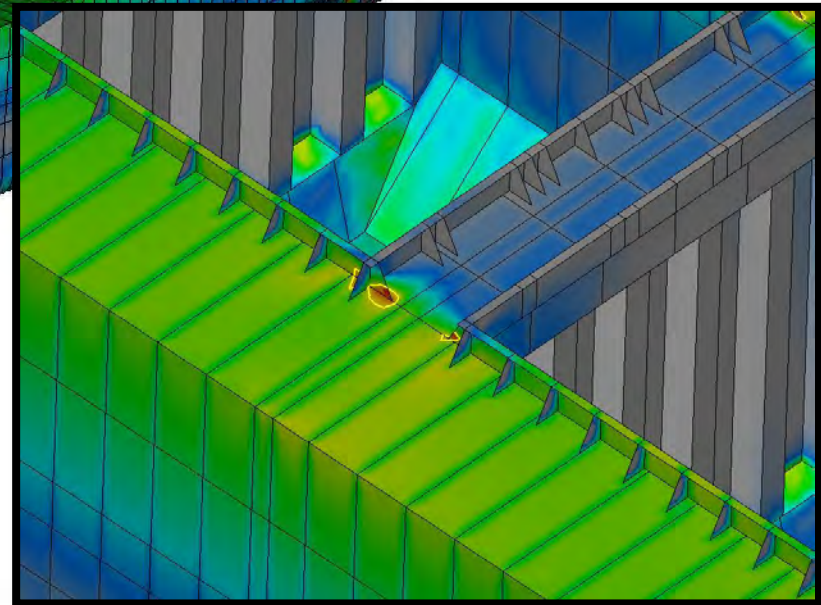
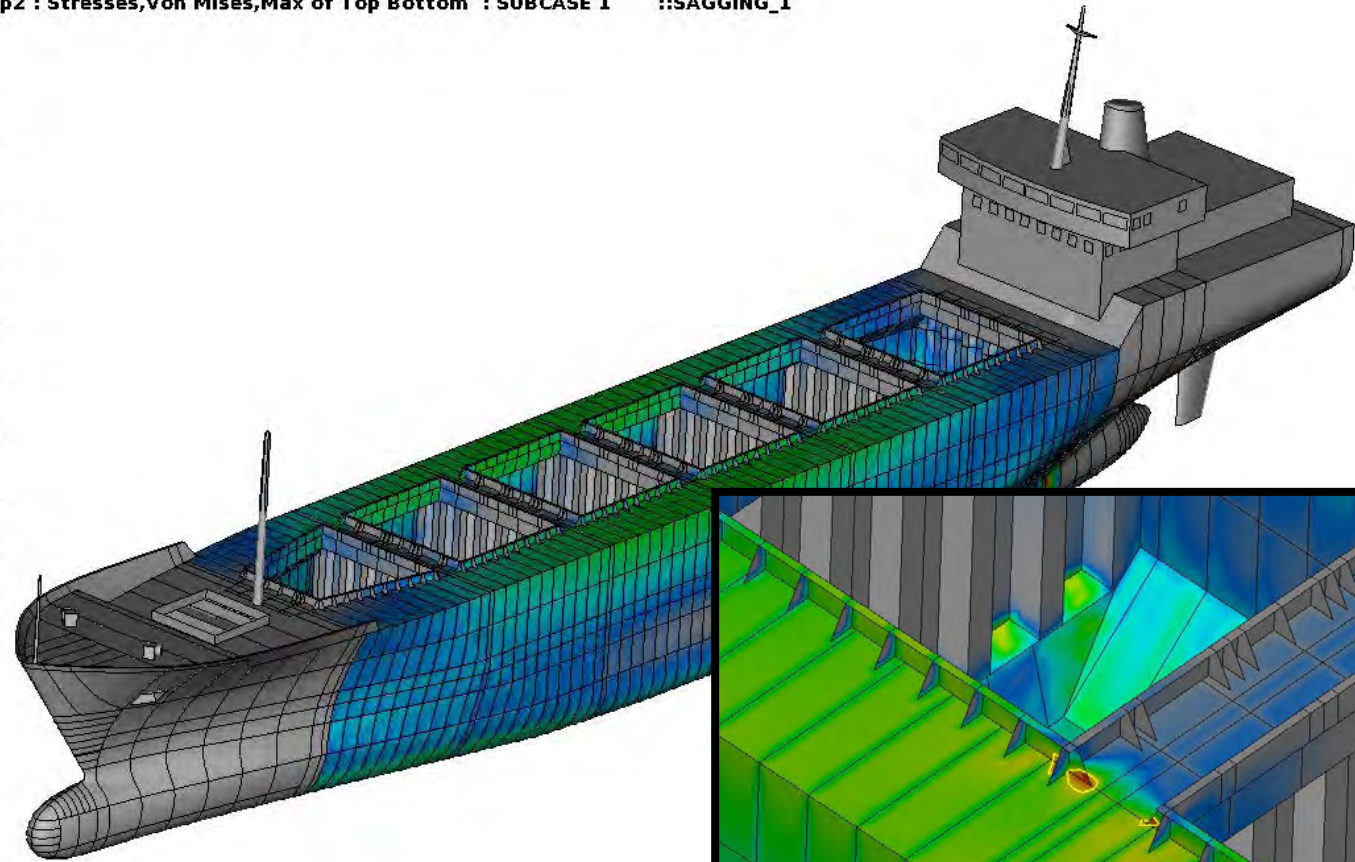
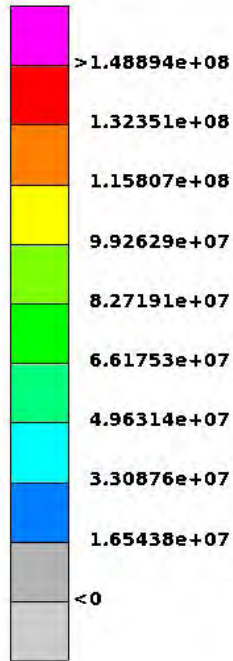


# Post Processing

# Viewing results in $\mu$ ETA

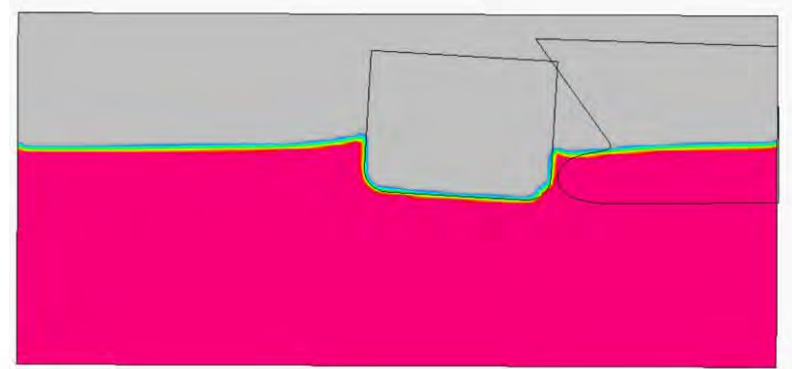
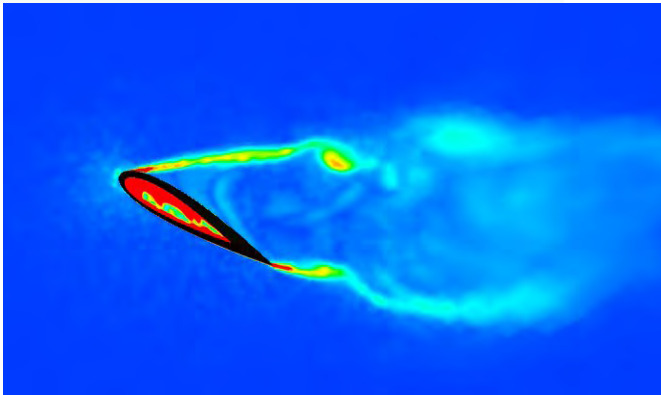
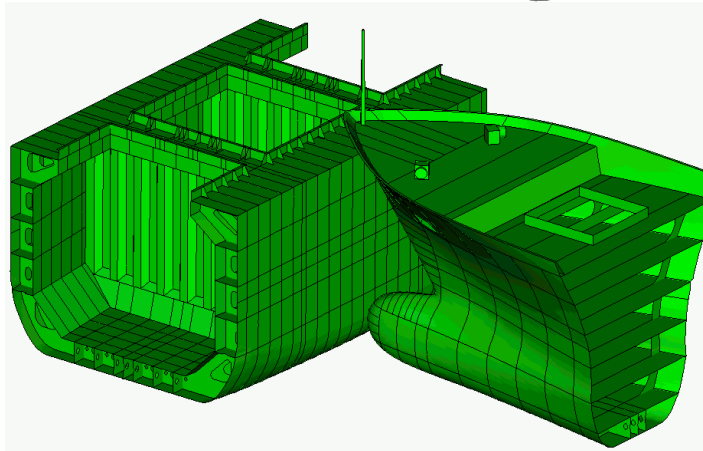
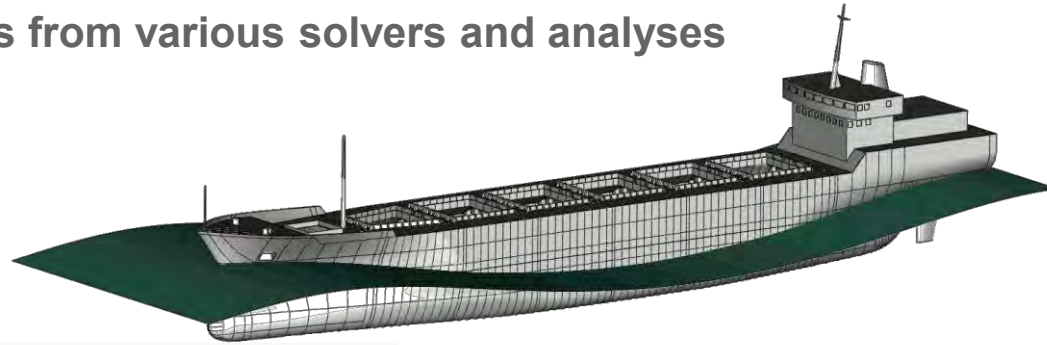
## Global and local stress inspection

f7\_L\_STOOL\_INREL.op2 : Stresses,Von Mises,Max of Top Bottom : SUBCASE 1 ::SAGGING\_1



# Viewing results in $\mu$ ETA

Supports results from various solvers and analyses



Results for:

Static

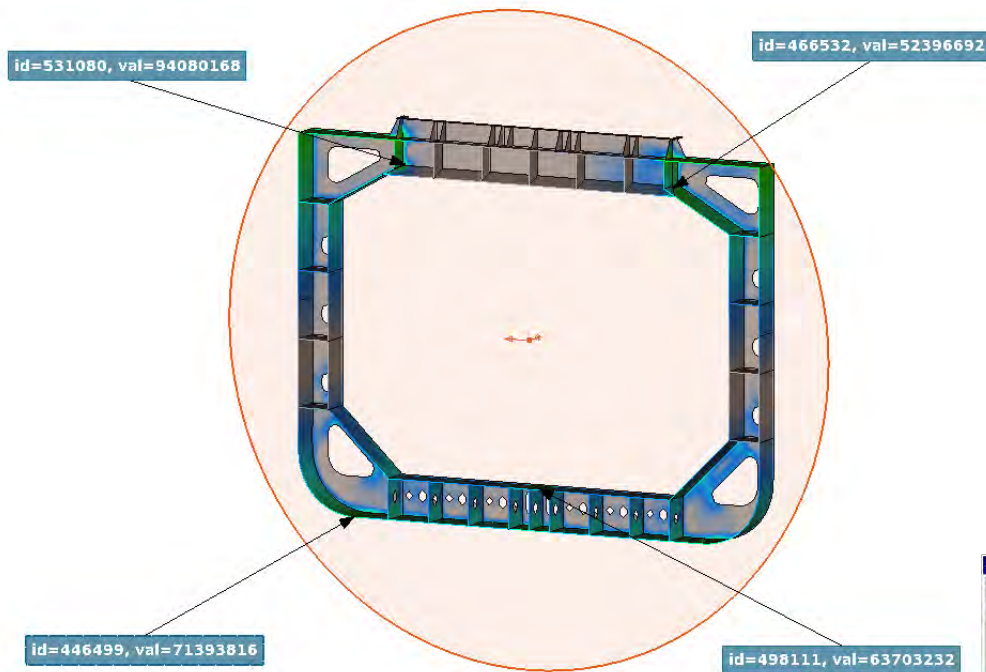
Dynamic

CFD

ALE

# μETA Reporting capabilities

Creating sections, annotations, statistics, reports...



Statistics

Current State: Stresses, Von Mises Max of Top Bottom : SUBCASE 1 : SAGGING\_1

id	CD	Max	C1	At	C2	Min	C3	At	C4	Range	C5	Range%	C6	Elms	C7	Nodes	C8	Mid	C9	Type
1	0	15	0	15	0	0.0	1596	1669	1	PShell										
4	1.17E+08	20285	0	30	0	0.0	44736	45860	1	PShell										
5	8.36E+07	77879	0	255	0	0.0	49428	50462	1	PShell										
6	1.65E+08	457721	0	45	0	0.0	75308	75655	1	PShell										
10	0	724	0	724	0	0.0	1282	1270	1	PShell										
12	1.5E+08	204557	0	356578	0	0.0	9160	10758	1	PShell										
13	0	10	0	10	0	0.0	19258	19669	1	PShell										
14	0	27406	0	27406	0	0.0	680	1000	1	PShell										
15	1.4E+08	4491	0	219	0	0.0	73052	76132	1	PShell										
16	0	1142	0	1142	0	0.0	2382	2639	1	PShell										
18	1.64E+08	170199	0	2191	0	0.0	41870	41542	1	PShell										
19	0	10234	0	10234	0	0.0	1402	1420	1	PShell										
20	7.72E+07	260032	0	303	0	0.0	94166	102708	1	PShell										
21	7.68E+07	536753	0	22	0	0.0	155324	178978	1	PShell										
22	0	10518	0	10518	0	0.0	8291	8886	1	PShell										
26	1.33E+08	170411	0	59	0	0.0	11352	12242	1	PShell										
29	7.42E+07	169229	0	1	0	0.0	4574	5680	1	PShell										

All | Invert | Visible | Pick | Filtering:

Model 0 | Function | All | All Entities

# μETA Reporting capabilities

Report Composer

Report: HTML, PPTX

Slide: [Icons]

Insert: [Icons]

Zoom: 90%

Align: [Icons]

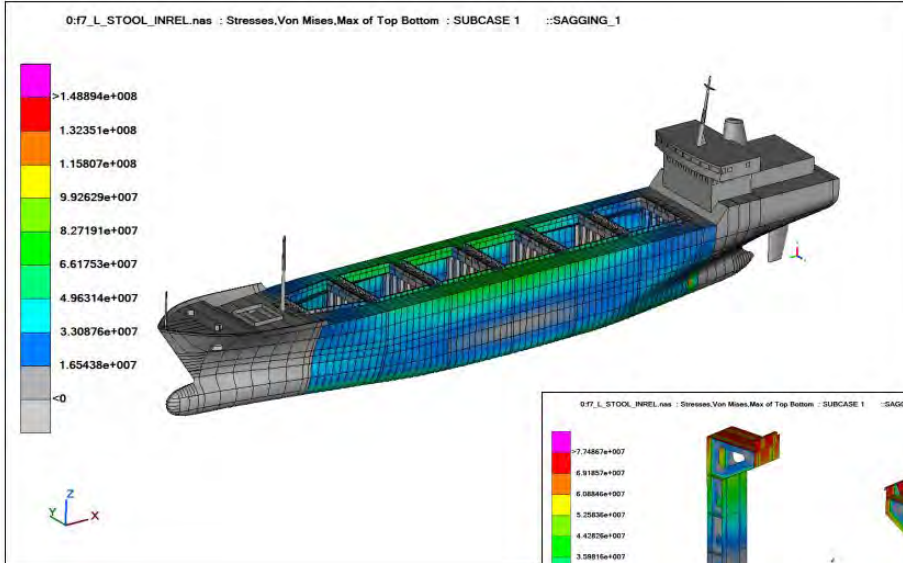
Distribute: [Icons]

Report

- Slide 1
  - Title
  - Image 1
  - Textbox 1
  - Image 2

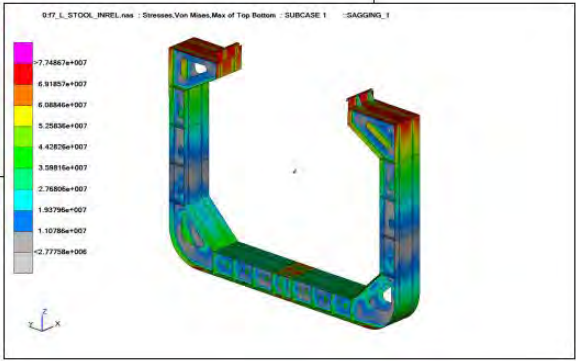
### Sagging Case Study

0:17\_L\_STOOL\_INREL.nas : Stresses,Von Mises,Max of Top Bottom : SUBCASE 1 ::SAGGING\_1



Legend for 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



Legend for 0:17\_L\_STOOL\_INREL.nas : Stresses,Von Mises,Max of Top Bottom : SUBCASE 1 ::SAGGING\_1:

- 7.34867e+007
- 6.91857e+007
- 6.08566e+007
- 5.25836e+007
- 4.42826e+007
- 3.58916e+007
- 2.76806e+007
- 1.93796e+007
- 1.10786e+007
- 2.77756e+006

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

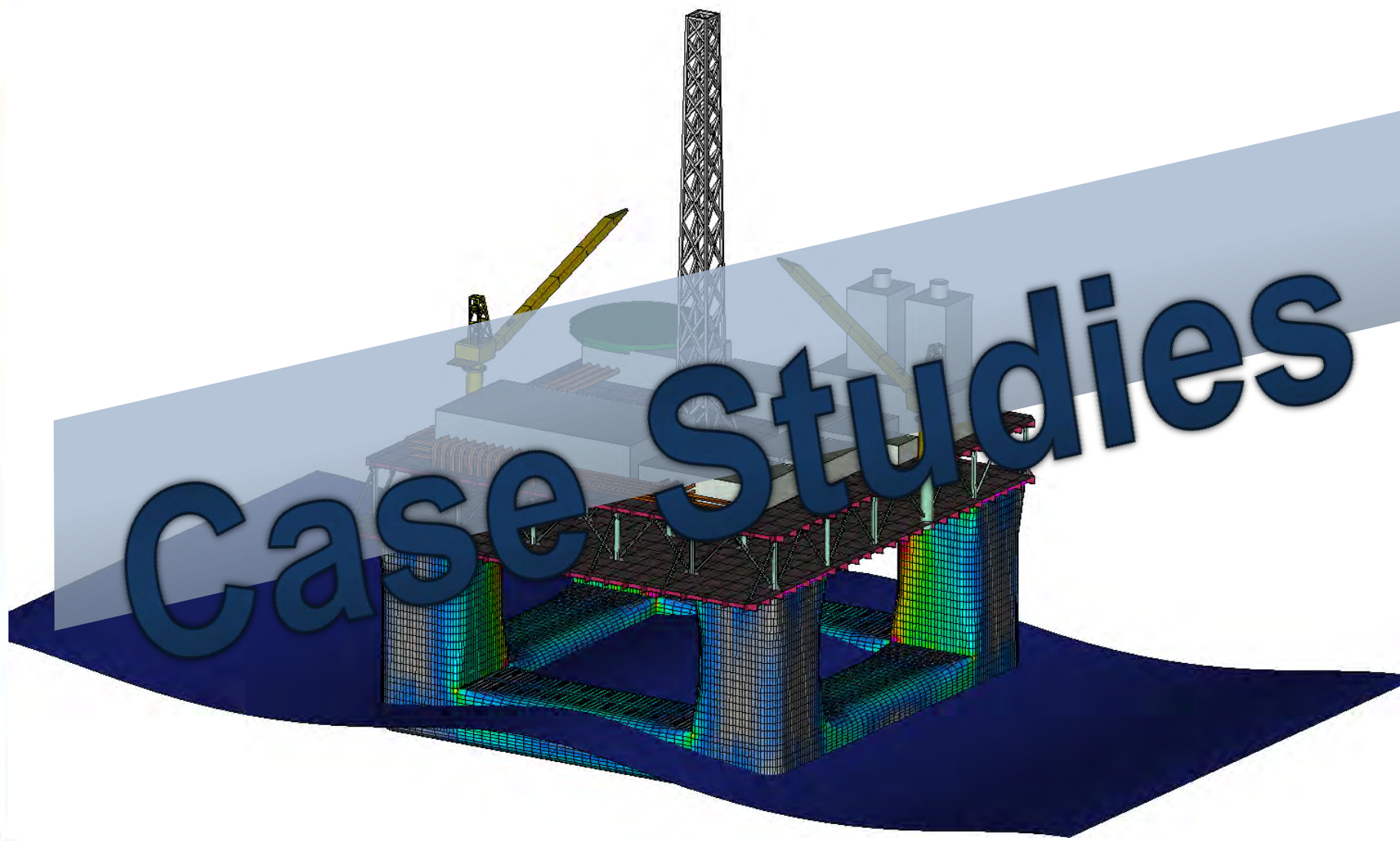
Create Slide Master Save Cancel

HTML / PPTX  
Reports

Drag and Drop  
functionality

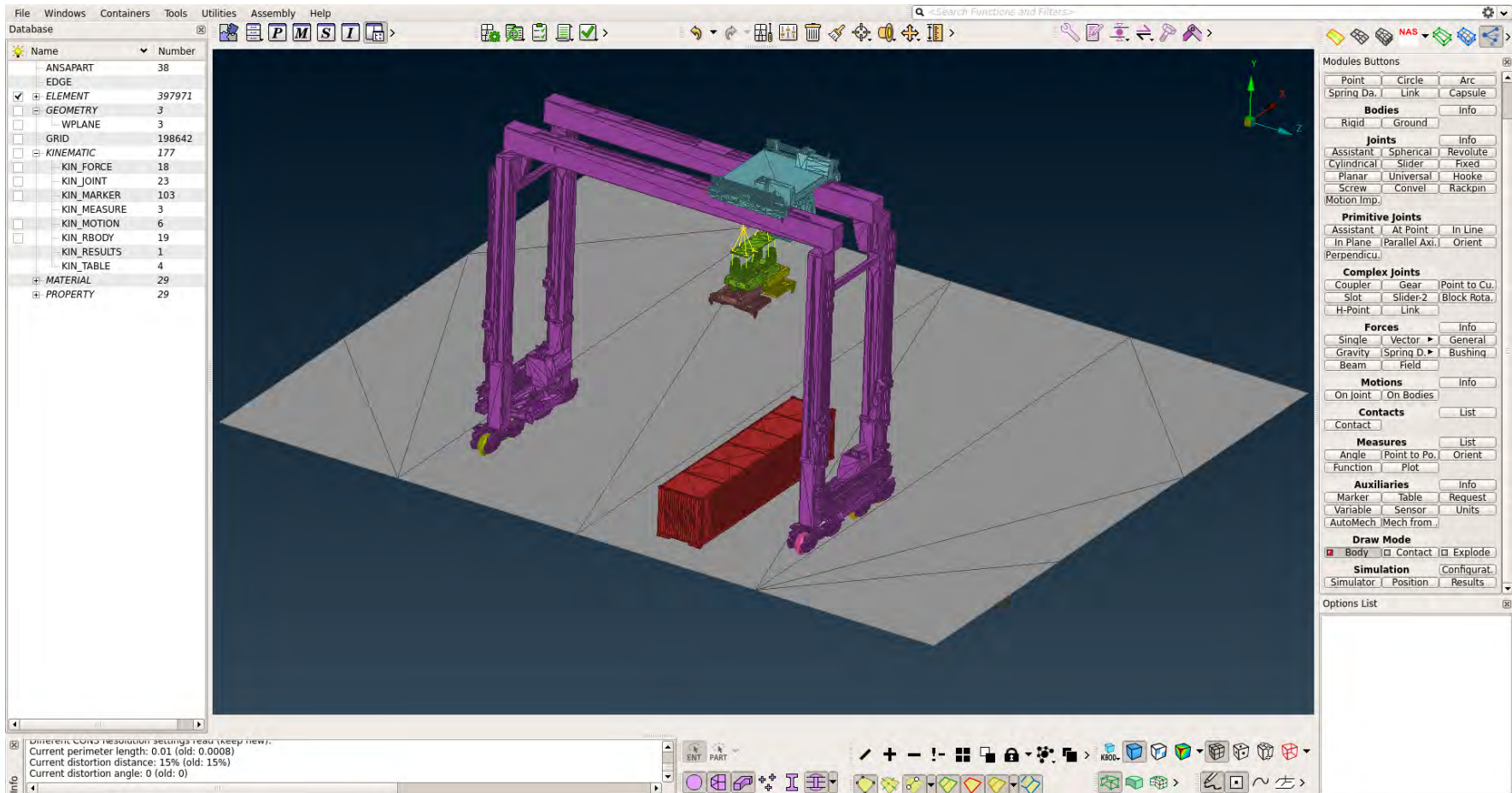
Standard Model  
Reports

# Case Studies



# Crane Lifting simulations with the Kinetics Module

## Kinetic mechanism definition



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

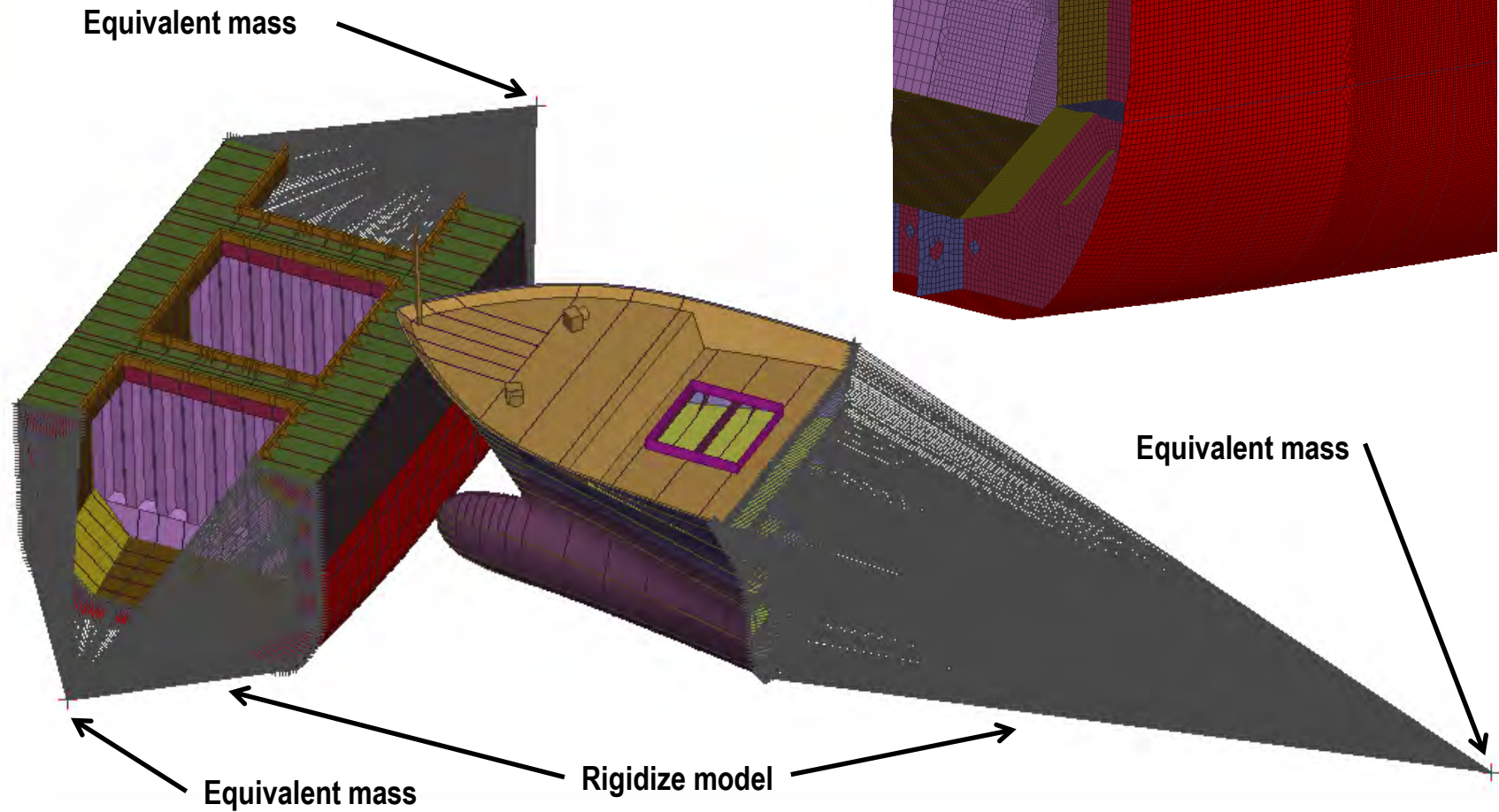


# Ship collision analysis

Replacing part of the model with rigid body and equivalent mass

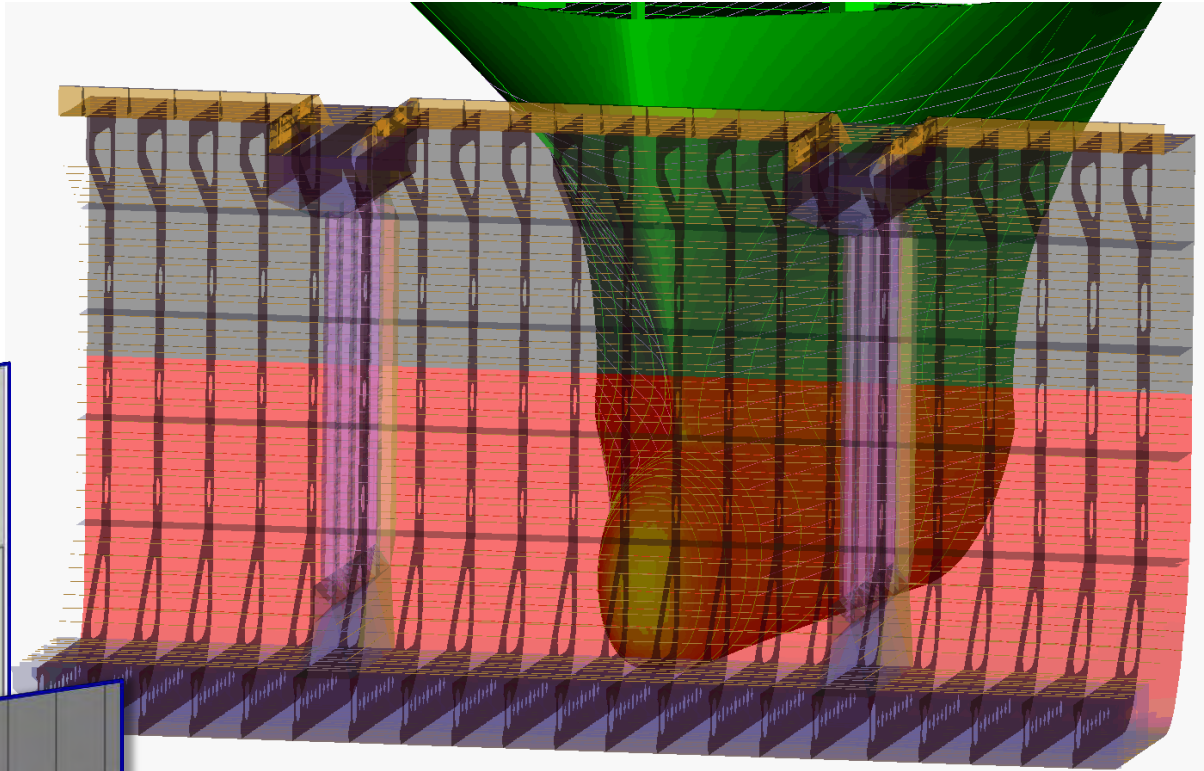
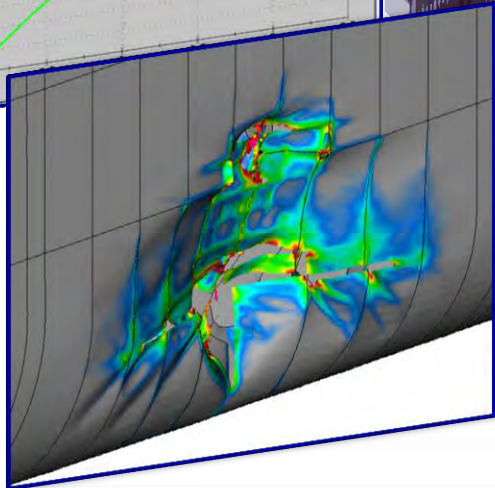
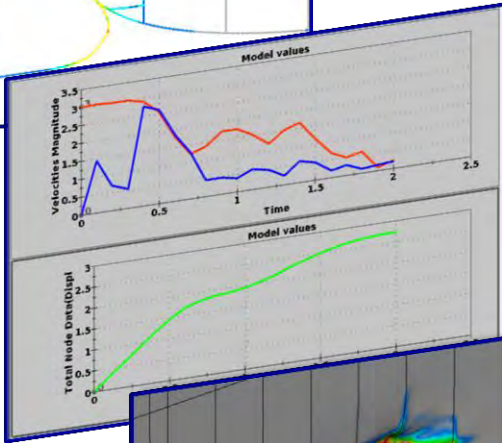
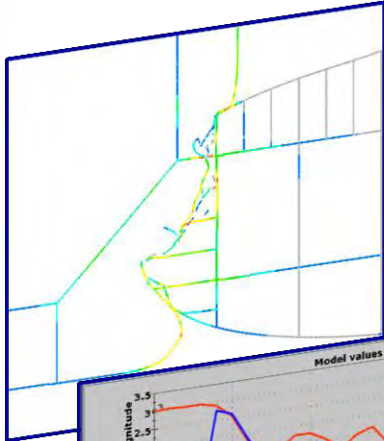
Defining boundary conditions and contacts

Local refinement at the collision area

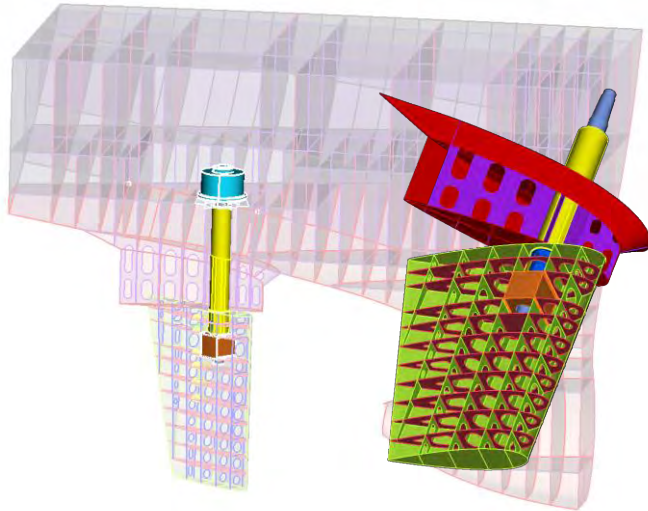


# Ship collision analysis

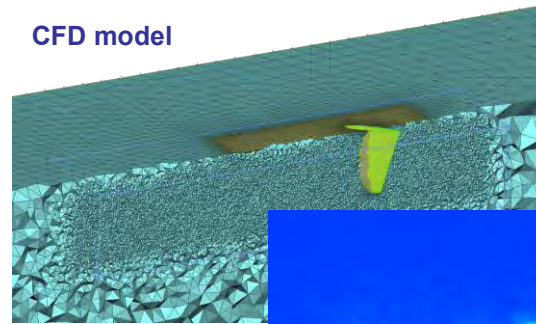
Viewing results with  $\mu$ ETA



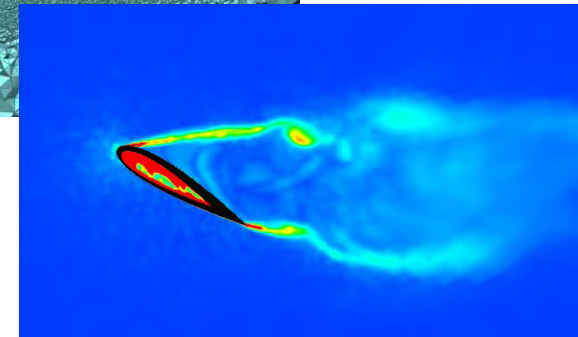
# Rudder Optimization Study



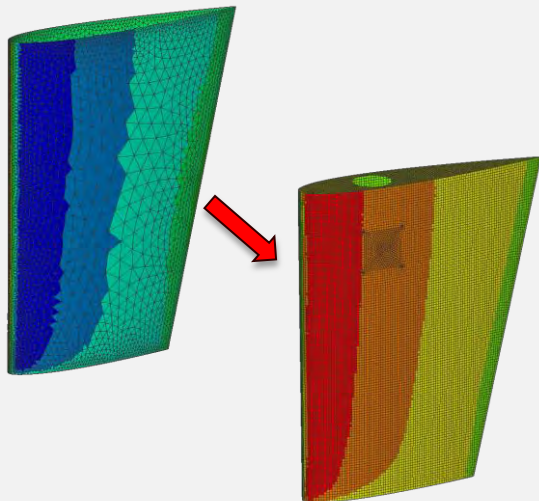
CFD model



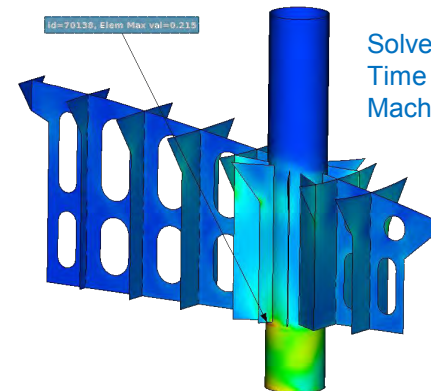
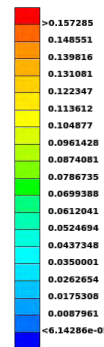
Solver: **FLUENT v13.0**  
Iterations to converge: **~1000**  
Time elapsed: **24 hours**  
Machine: **Linux, Core i7 8 CPUs**



## Results Mapping Tool



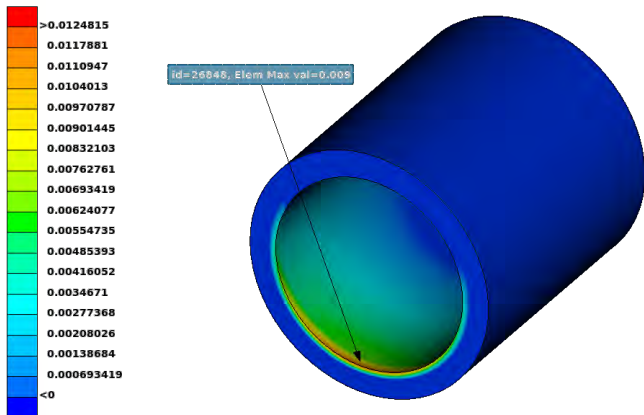
0:rudder\_from\_cfd.odb : propeller asping : Stress components,Von Mises,Max of In Out,Centroid : : STEP 1 (AnonymousSTEP1),TIME 4.00000006E-01,



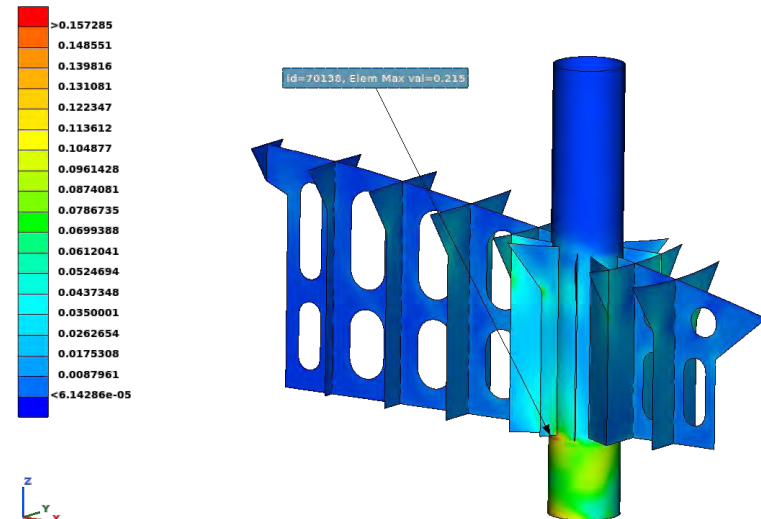
Solver: **ABAQUS v 6.10**  
Time elapsed: **30 min**  
Machine: **Linux, Core i7 8 CPUs**

# Rudder Optimization Study

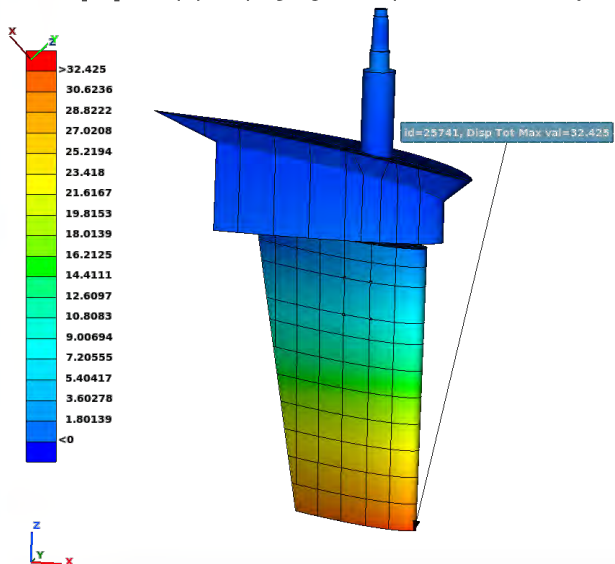
O:rudder\_from\_cfd.odb : propeller asping : Contact pressure,All Surfaces : : STEP 1 (AnonymousSTEP1),TIME 4.00000006E-01,



O:rudder\_from\_cfd.odb : propeller asping : Stress components,Von Mises,Max of In Out,Centroid : : STEP 1 (AnonymousSTEP1),TIME 4.00000006E-01,



O:rudder\_from\_cfd.odb : propeller asping : Magnitude of Displacements : STEP 1 (AnonymousSTEP1),TIME 4.00000006E-01,



## Objectives

✓ Minimize maximum contact pressure

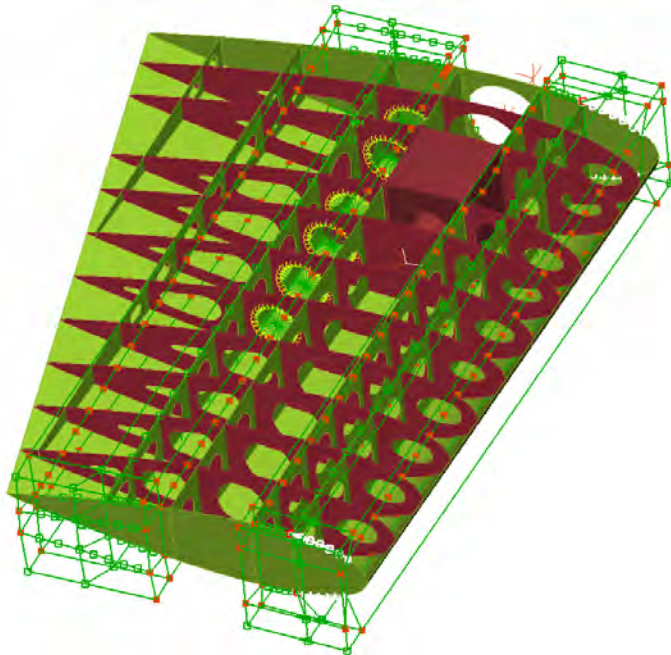
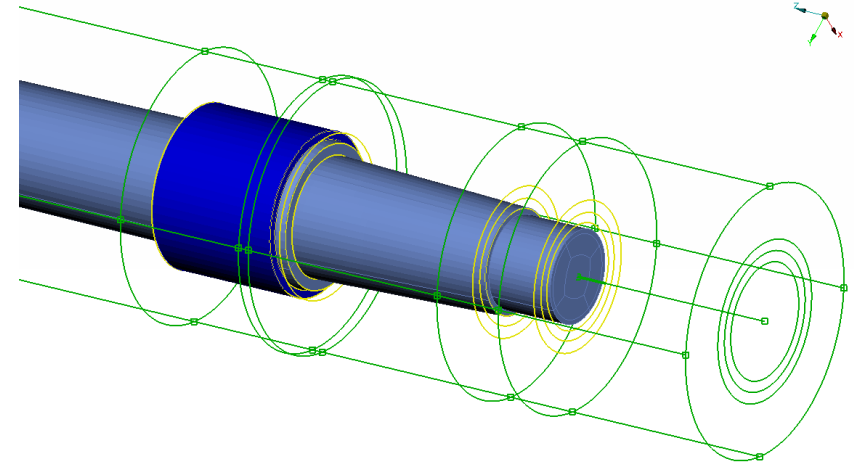
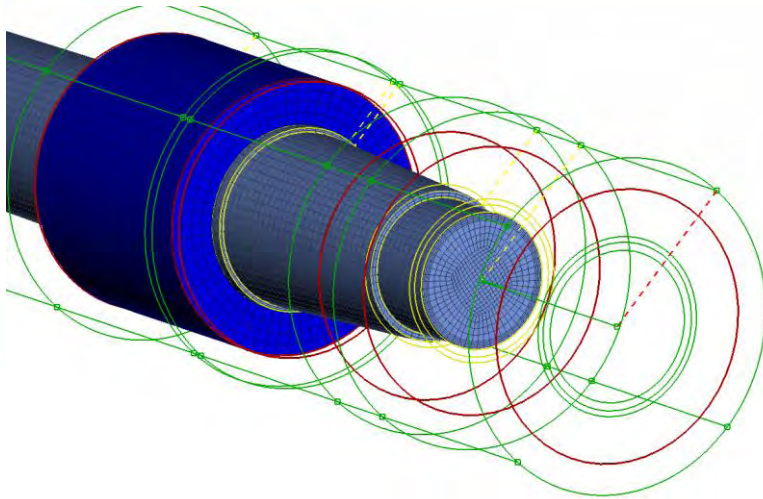
✓ Minimize Model mass

## Constraints

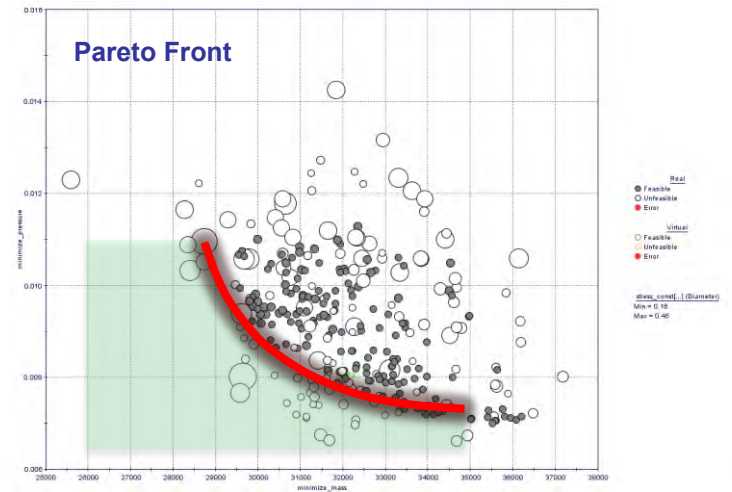
✓ Maximum stresses

✓ Maximum deflection

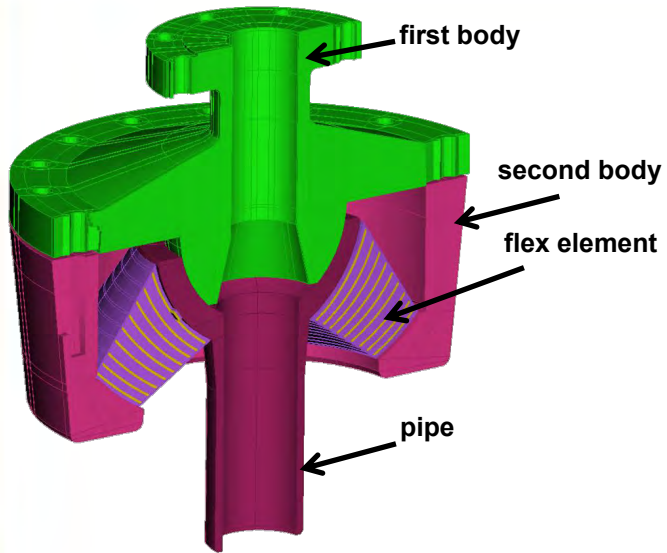
# Rudder Optimization Study



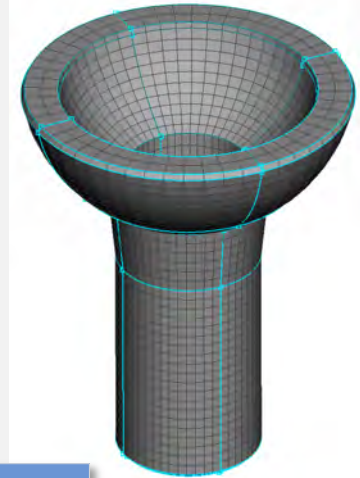
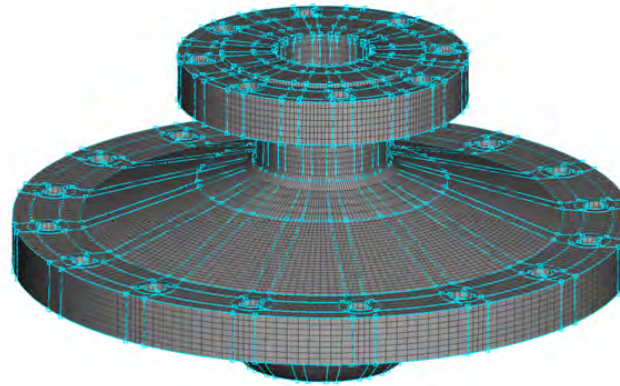
- ✓ **12.5% Maximum pressure reduction**
- ✓ **7.98% Mass reduction**



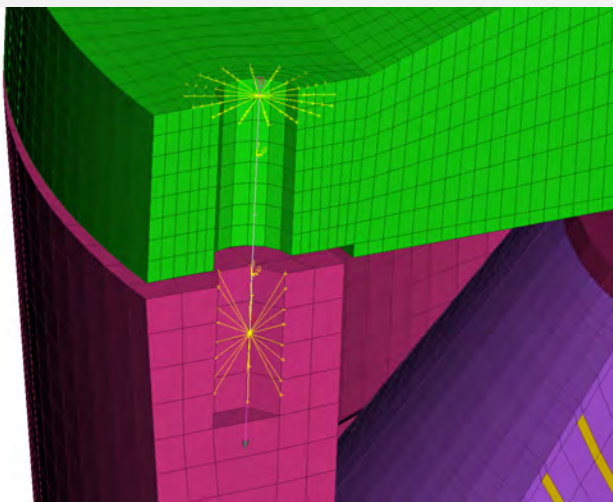
# Flex Joint Contact analysis



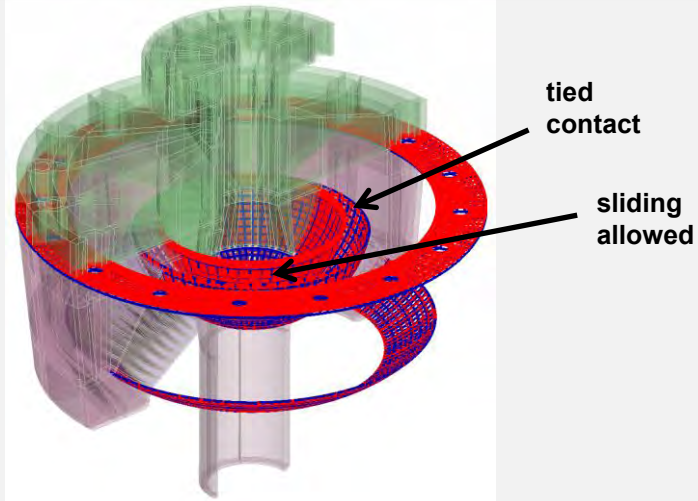
## Pure Hexahedral Mesh Creation



## Automated Bolts recognition



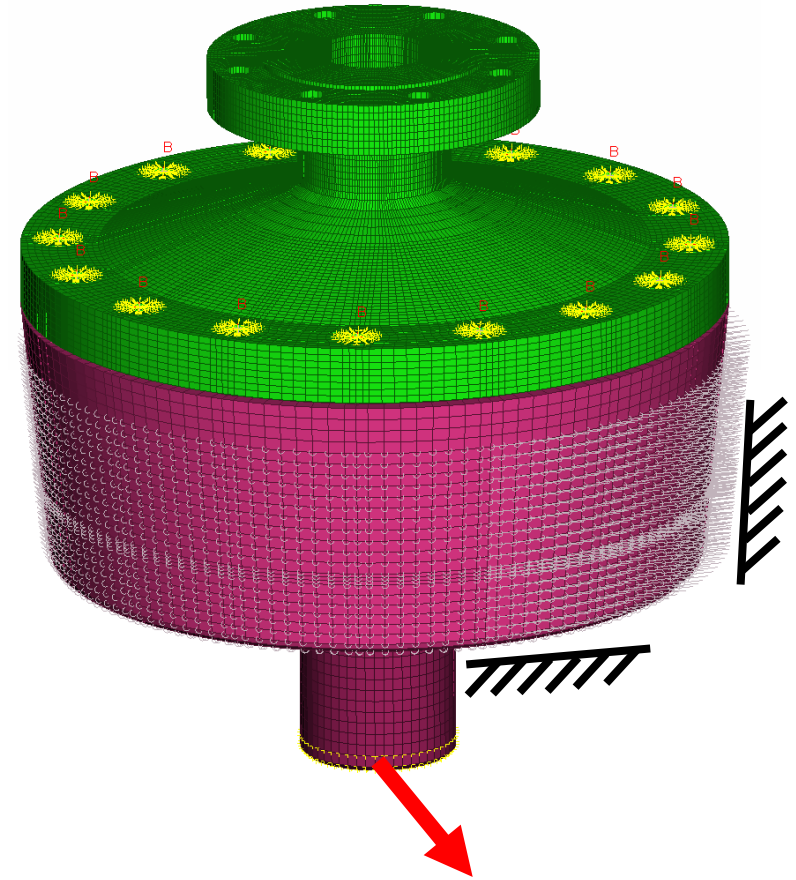
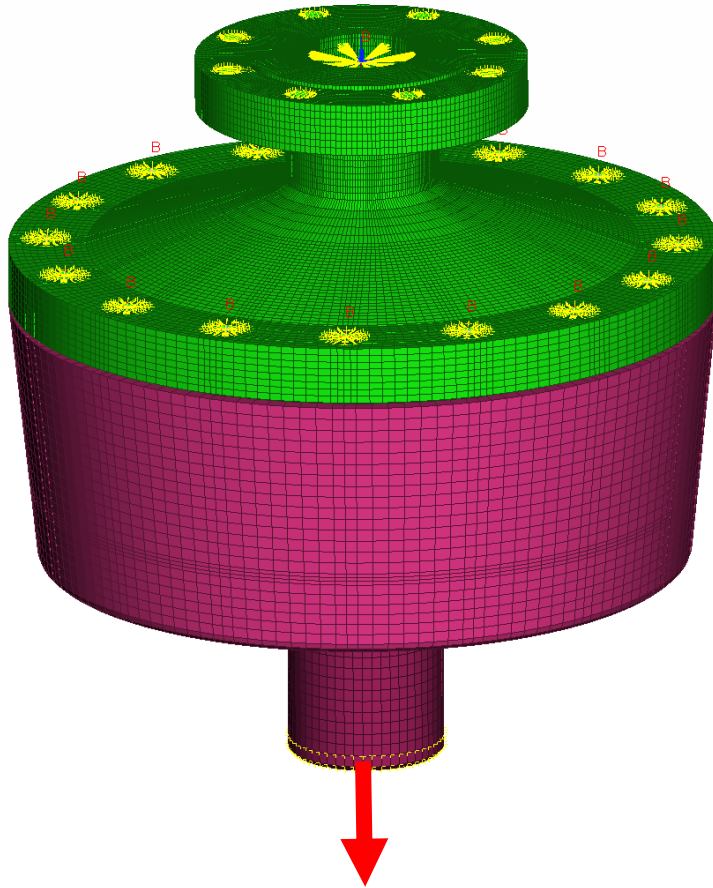
## Automated Contact Detection



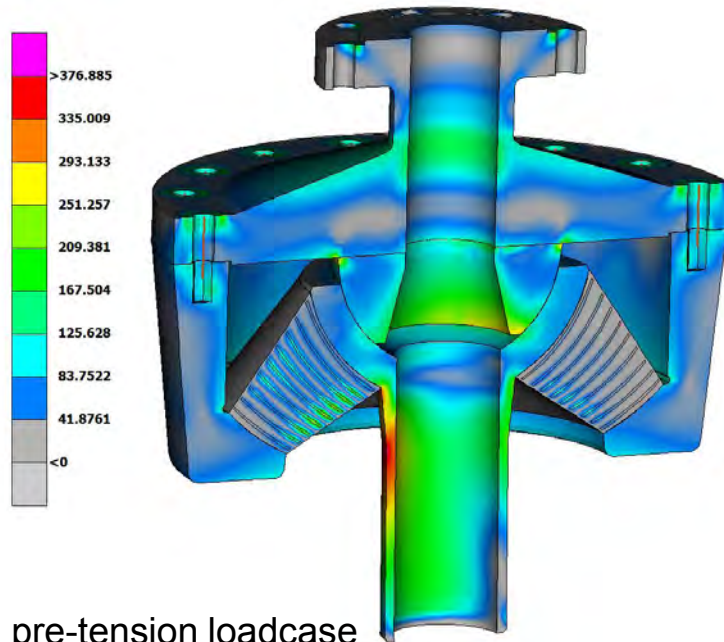
# Boundary conditions

1<sup>st</sup> load case: axial riser force

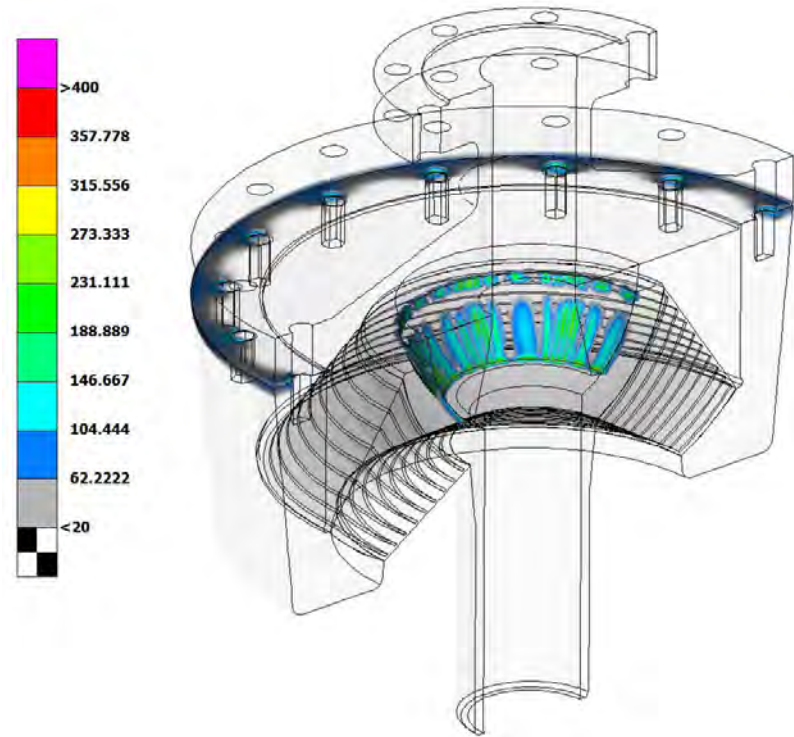
2<sup>nd</sup> load case: bending riser force



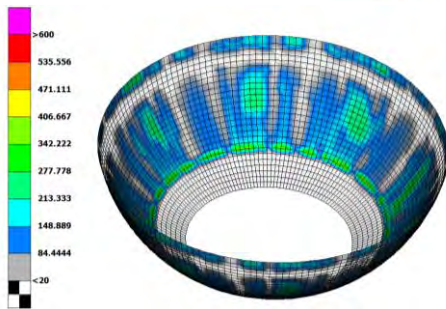
# Flex Joint Contact analysis



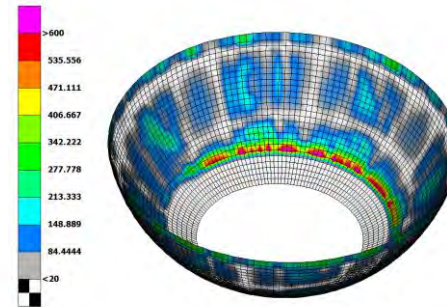
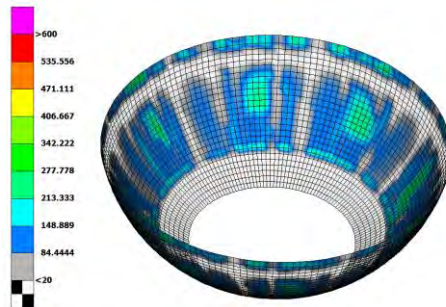
pre-tension loadcase



axial force loadcase



bending force loadcase

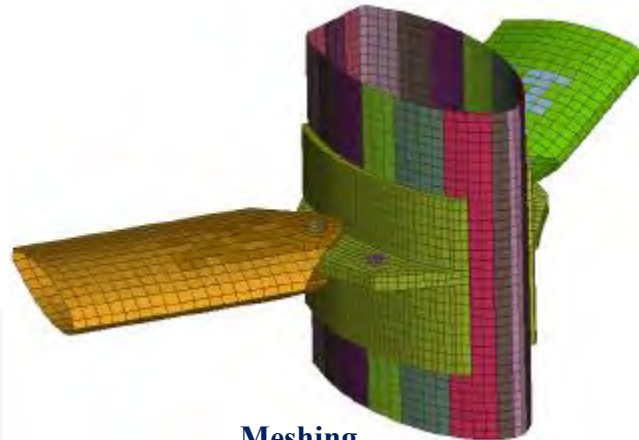




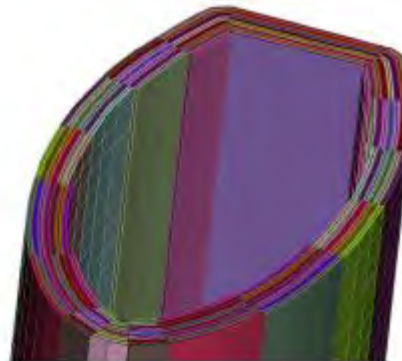
# Safran Open 60' race yacht composite mast modelling for crashworthiness analysis



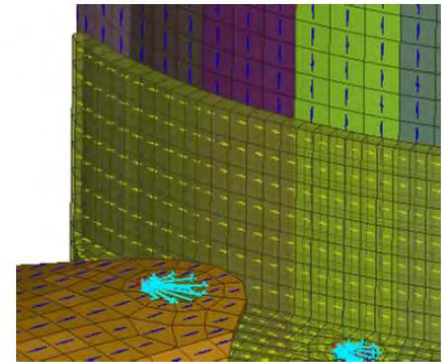
**Mast modelling**



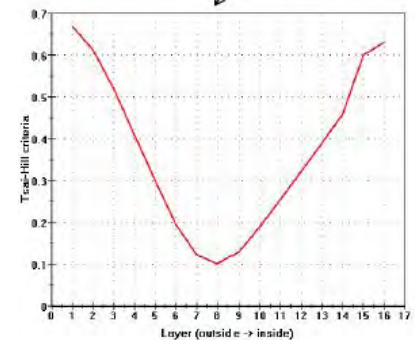
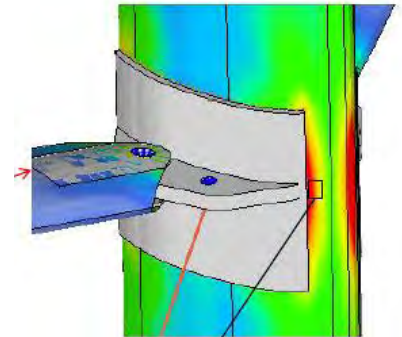
**Meshing**



**Composite materials**

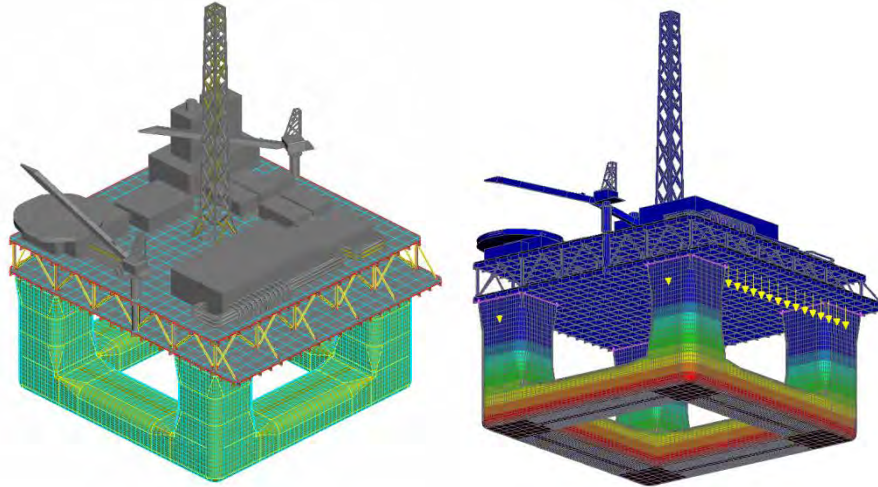


**Elements orientation**

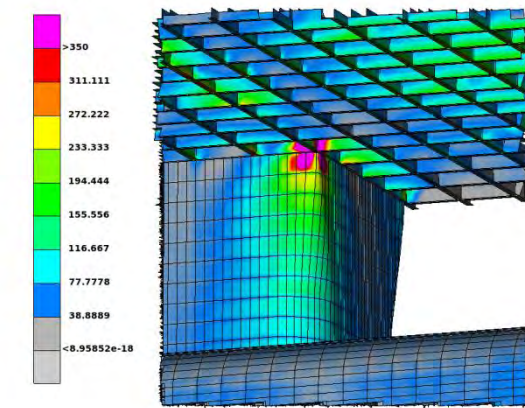


# Static Analysis for Offshore Models

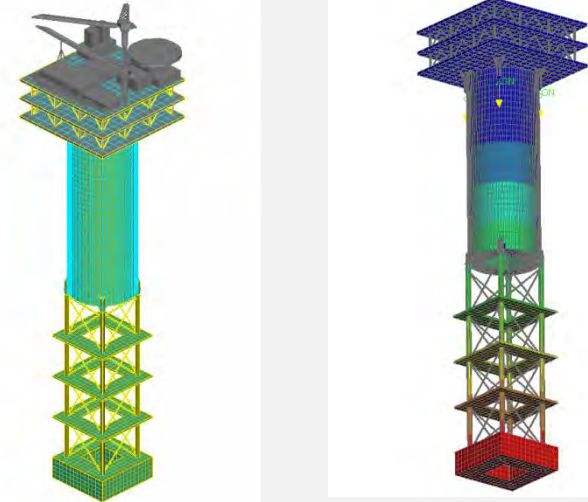
## Semi Submersible Platform



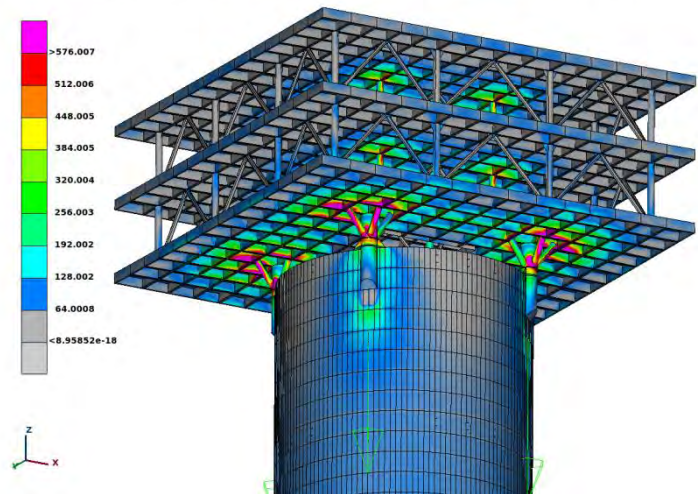
0:oilplatform-sol6.nas : Stresses,Von Mises,Max of Top Bottom : SUBCASE 1  
 1:superstructure.nas : ORIGINAL STATE  
 2:wave\_low\_5meter.nas : ORIGINAL STATE



## Spar Platform

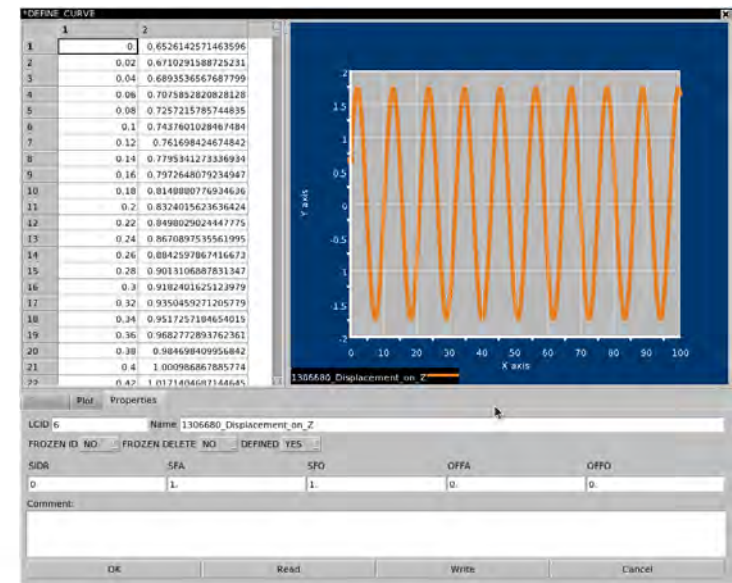
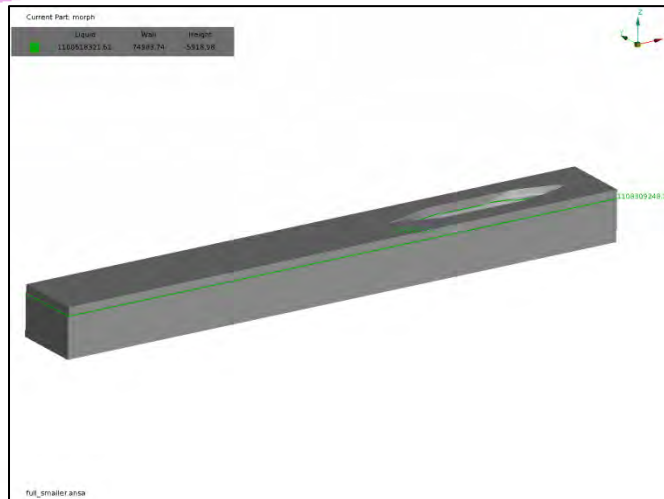
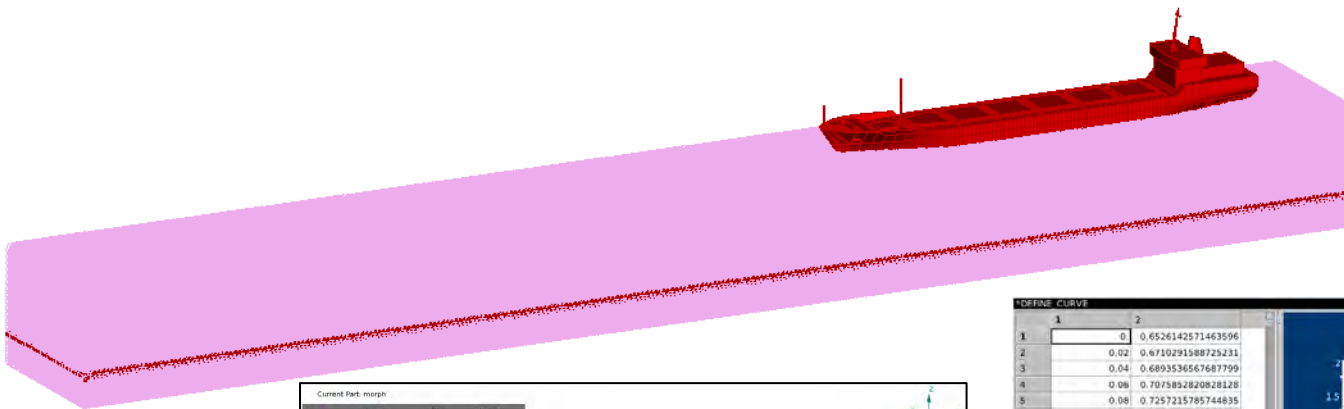


0:SPAR\_sol3.nas : (fo1 375643) : Stresses,Von Mises,Max of Top Bottom : SUBCASE 1  
 ::SPAR-PLATFORM:SOL1: SUBCASE 1



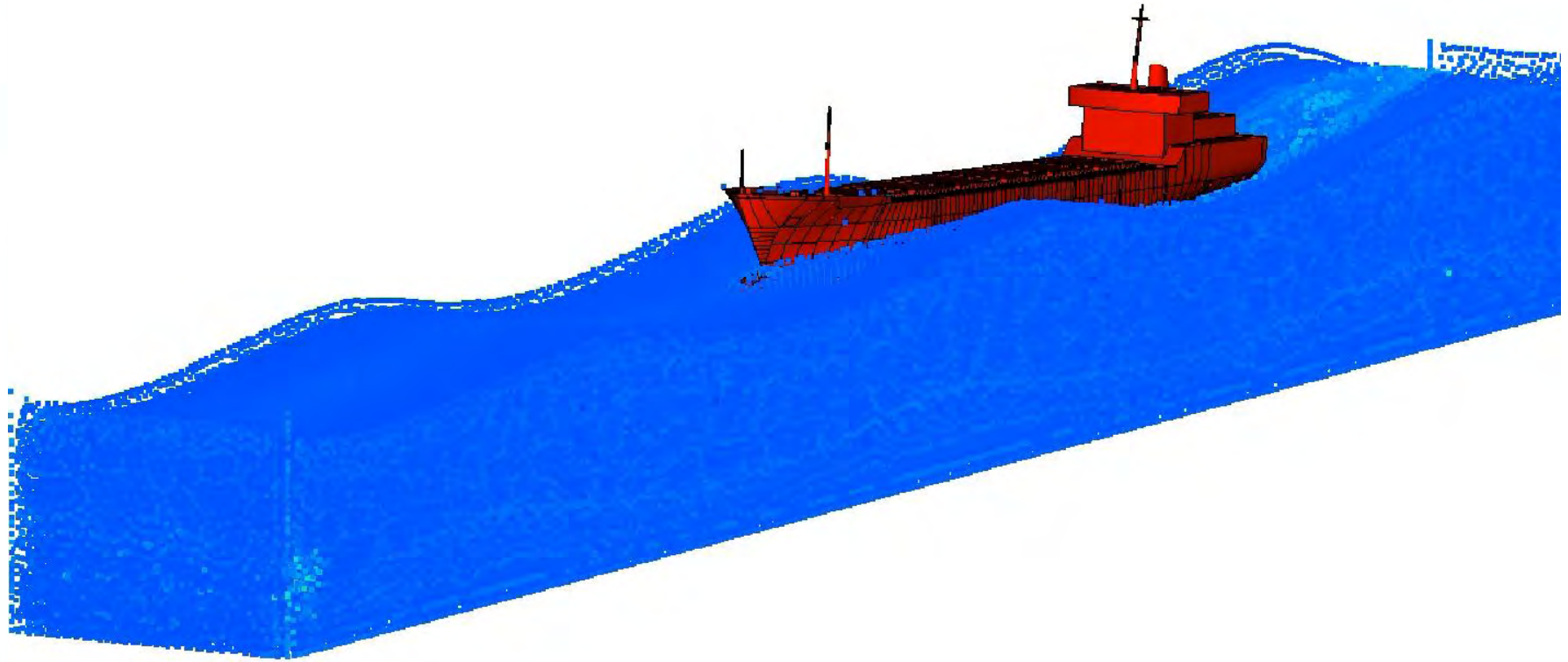
# Wave on SPH case study

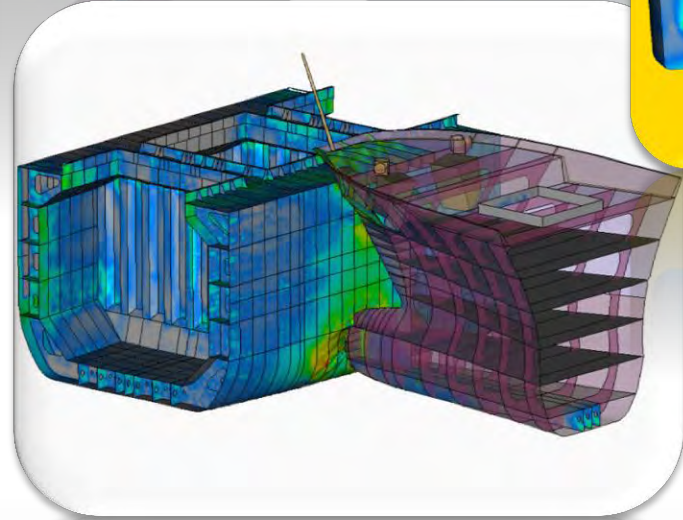
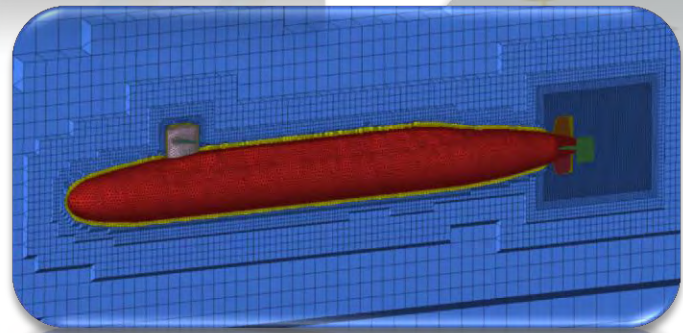
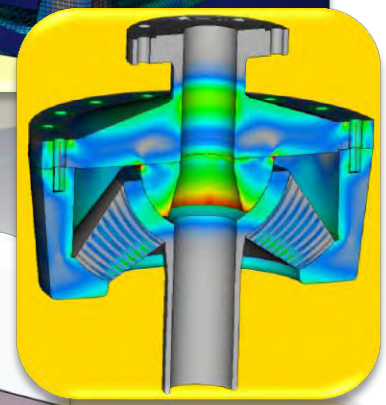
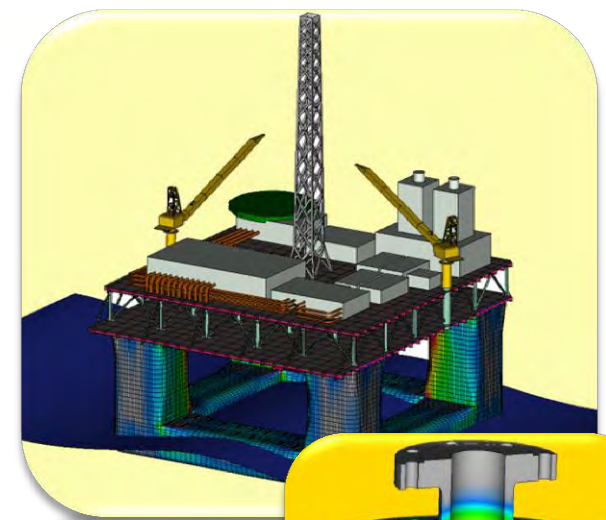
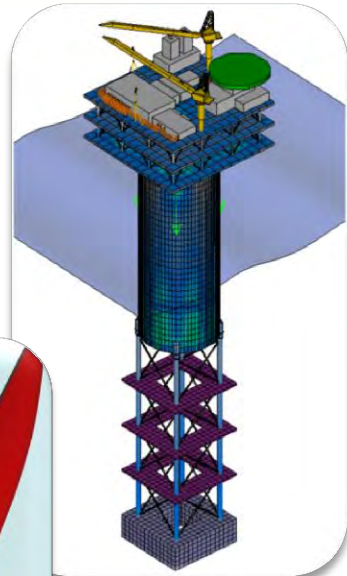
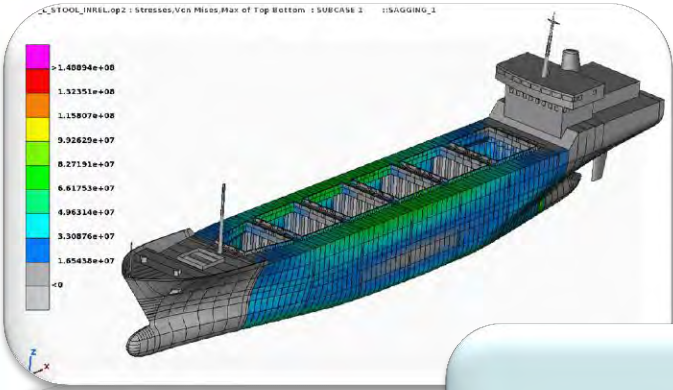
- Closed volume creation
- Automated SPH creation at user defined level using the Tank Tool
- Merging the rigid ship model
- Automated application of wave initial conditions using linear Airy wave theory



# Wave on SPH case study

- Results in  $\mu$ ETA





**Thank you!**