Introducing Highly-Efficient CAE Pre- and Post-Processing Solutions in Maritime Design

George Korbetis*, BETA CAE Systems S.A., Thessaloniki/Greece, ansa@beta-cae.gr
Dimitris Georgoulas, BETA CAE Systems S.A., Thessaloniki/Greece, ansa@beta-cae.gr

Abstract

In our days, the CAD and CAE models in maritime products development become increasingly complex while more analyses are necessary before a new design is ready to be constructed. Additionally to the standard assessments, extensive calculations are often needed to ensure the product's performance characteristics and minimize the failure risk through its lifetime. For the fast employment of such analyses with CAE simulation tools, the use of high efficient pre- and post-processing software becomes essential. This work showcases how ANSA pre-processor and μETA post-processor fulfill this requirement of the Maritime Industry by offering sophisticated tools for advanced simulation techniques, automation capabilities and robust performance.

1. Introduction

The scope of this paper is to present an efficient way to set up CAE analyses, for several disciplines and load cases, starting from a common CAD model. Three case studies are demonstrated where a handysize class double skin bulk carrier is subjected to static, full scale collision and CFD analysis. The recommended process includes, among others, model simplification and idealization, meshing, element quality improvement, loads and boundary condition definition. The behavior of the bulk carrier model can be improved according to the above analyses using optimization techniques. The use of the capabilities of ANSA pre-processor in model shaping and parameterization for the set up of such optimization processes is also demonstrated.

2. Process management

During the design process of a product, the CAD model is distributed to the CAE departments to perform simulations. However, continuous updates of several model parts can cause problems to the analysis flow since the CAE models should be updated with the new version. Furthermore, the complex analysis process should be standardized in order to eliminate any dependency of the model quality to the engineer's expertise. The standardized process should be shared among the engineers and thus to exchange the analysis know-how. A specialized tool for the process organization is provided within the ANSA pre-processor, the Task Manager, Fig. 1.

All analyses that the bulk carrier will be subjected can be defined in the ANSA Task Manager. All the actions needed to define the FE models for the different load cases or disciplines are set in this tool, in a step-wise sequence. Running the Task Manager sequence, ANSA realizes every Task Item and performs the corresponding action on the model. When needed the user is prompted to interact. Furthermore, the Task Manager checks if every Task is defined correctly.

The process starts from the collection of the CAD data that is common for all analyses. Starting from the geometric model, ANSA is able to create different representations for each part of the assembly, which suit the requirements of the different analysis types. Thus, a part can have representations with different geometrical detail level, meshing parameters, element quality criteria, etc. according to the analysis needs. When the Task Manager runs to create the FE model for a specific analysis, it composes the assembly by collecting the appropriate part representations.

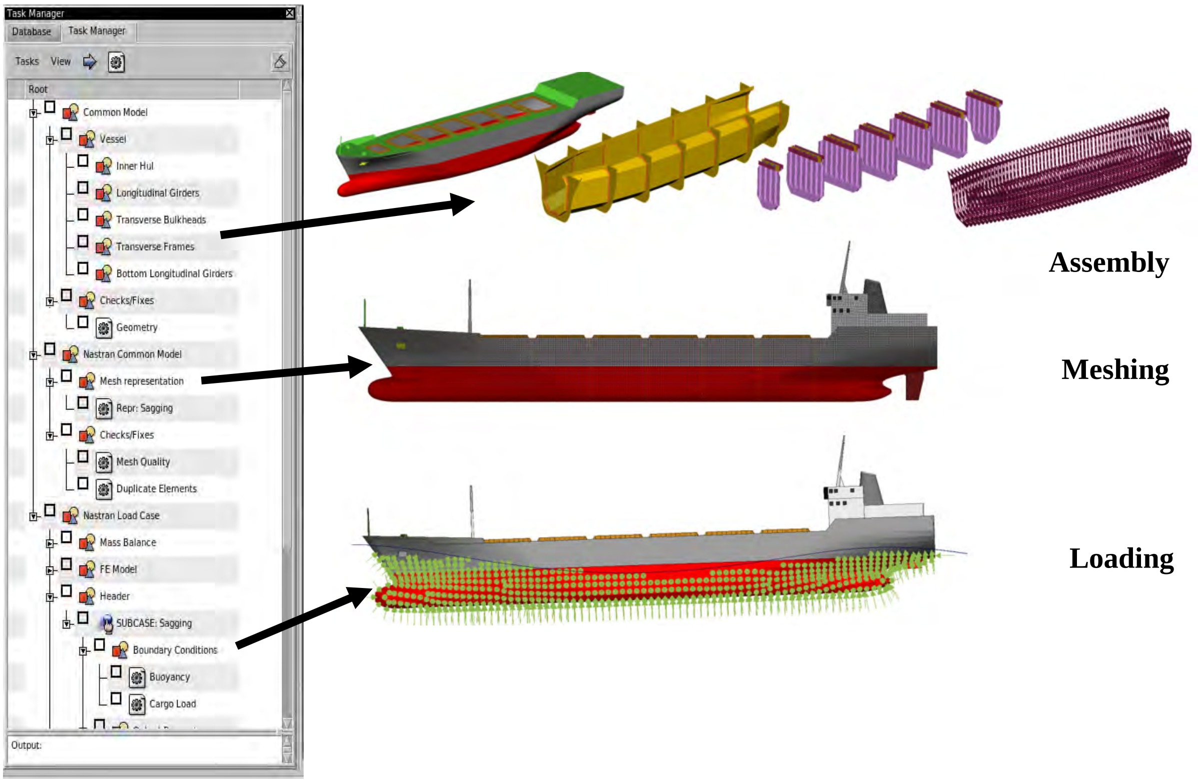


Fig.1: The Task Manager for the sagging load case

The meshing parameters and quality criteria are prescribed for each part, assembly or region within the ANSA Batch Meshing tool, creating meshing scenarios. These scenarios can be related to each analysis so a part can be meshed with different parameters providing a part representation for each analysis as shown at Fig. 2. A meshing scenario is defined for the sagging case study which applies mesh with different element length to various parts of the assembly, Fig. 3.

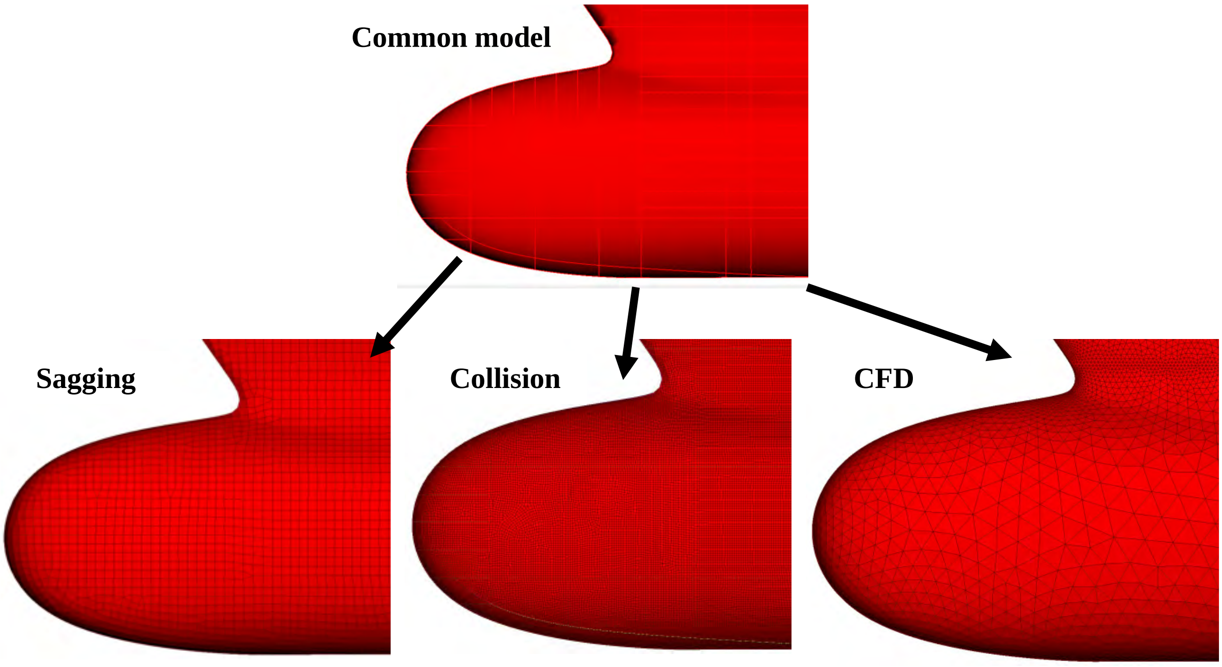
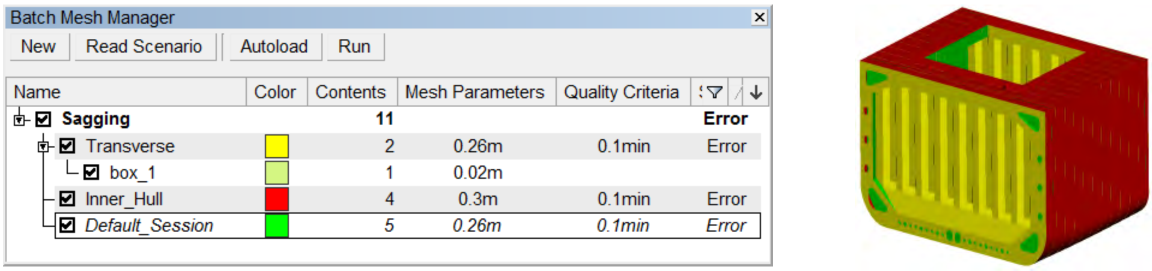


Fig.2: Sagging, collision and CFD representations of the same model



Name	Color	Contents	Mesh Parameters	Quality Criteria	
☑ **Sagging**		**11**			**Error**
☑ Transverse	🟨	2	0.26m	0.1min	Error
☑ box_1	🟨	1	0.02m		
☑ Inner_Hull	🟥	4	0.3m	0.1min	Error
☑ *Default_Session*	🟩	*5*	*0.26m*	*0.1min*	*Error*

Fig.3: The Batch Meshing tool

3. Sagging case study

The first case study is a static analysis of a full structural model of a bulk carrier. The model is subjected to sagging loading conditions when the ship's holds are fully loaded. The target of this analysis is the determination of the maximum stresses on critical areas. Geometrical model and specifications are presented on Fig. 4 and Table I.

Table I: Ship model specifications

Type of vessel	Handysize class double skin bulk carrier
Length	169 m
Breadth	25 m
Depth	18 m
Lightweight tonnage	9500 t
Deadweight tonnage	26000 t
Number of holds	6

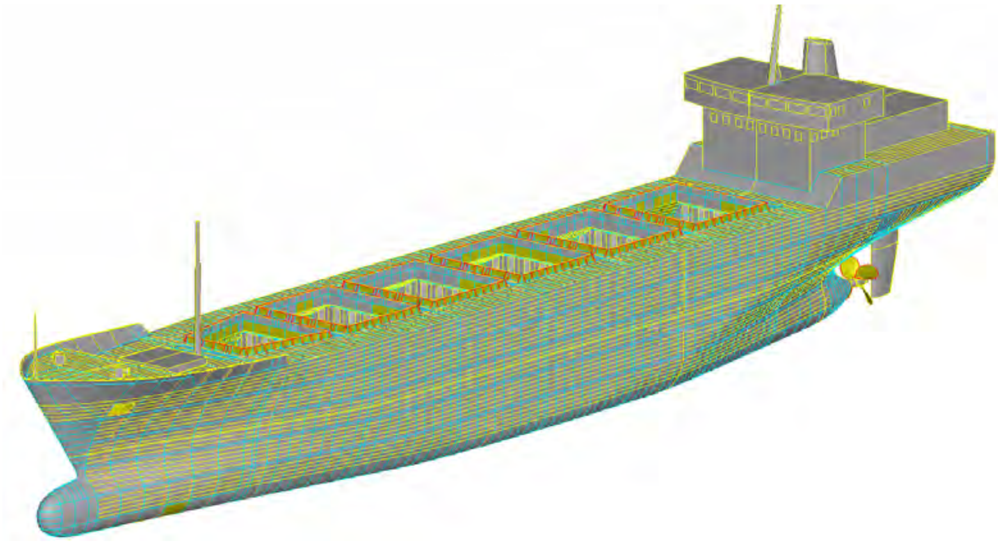


Fig.4: The geometrical model

3.1. Model set-up

The cargo ship model is relatively large so, coarse mesh should be applied to avoid very long simulation time. In addition, geometrical simplifications should be applied on the model. The first action to simplify the model is to fill small holes (diameter < 0.4 m) that are not significant for the model behavior. Such holes are automatically identified by their diameter and filled. This action improves the elements quality while reduces the number of elements, Fig. 5. The process of filling holes is prescribed at the meshing parameters of a Batch Meshing Scenario that is applied to the respective parts by the Task Manager. Thus, the whole process runs in batch mode without the need of user interaction.

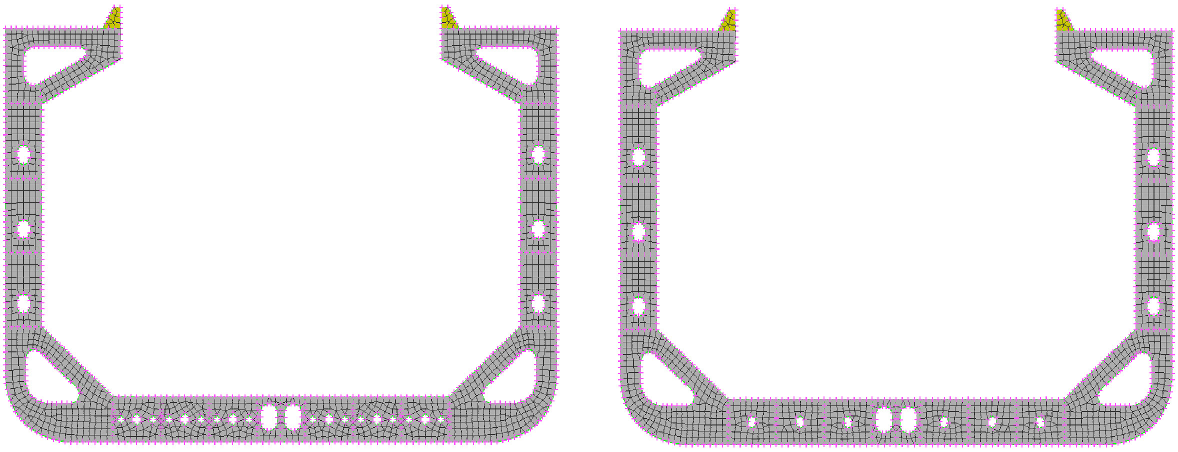


Fig.5: Holes filling at transverse section

The second simplification action is the replacement of longitudinal stiffeners by beam elements. This action reduces significantly the number of the small elements that represent the stiffeners. The beams that are applied have the same characteristics and behavior with the replaced stiffeners. Beams replacement is an automatic process in ANSA that is able to replace the whole model's stiffeners with little interaction. Beams are offset and oriented to fit the stiffener position while they are connected to the shells of ship model, Fig. 6.

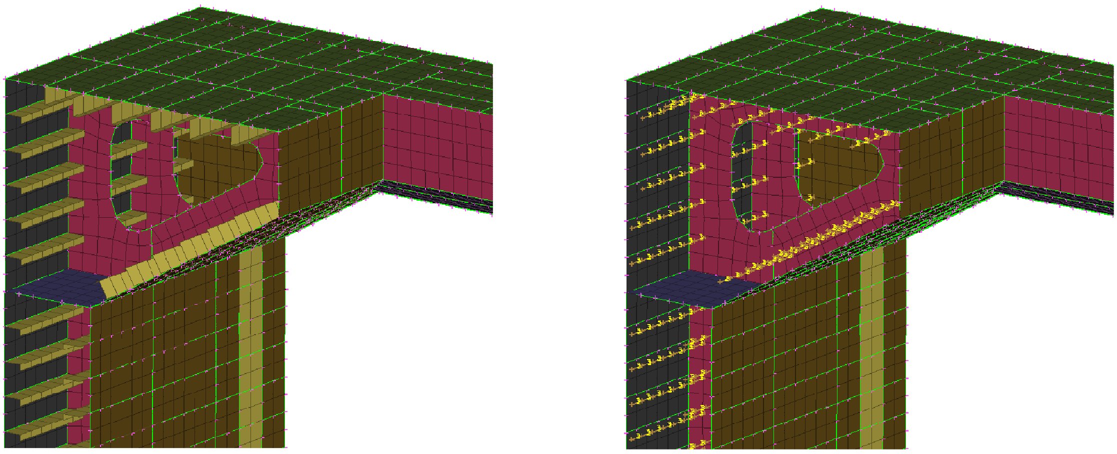


Fig.6: Stiffeners replacement with beams

The meshing parameters and quality criteria are defined in the ANSA Batch Meshing tool. When the Task Manager runs the process the mesh is applied to the whole model. The re-meshing algorithms ensure that the generated mesh fulfill the prescribed quality criteria. The user can identify critical areas on the model where accurate results should be extracted. A different meshing scenario that creates a local refinement is applied on these areas, Fig. 7. Meshing information and quality criteria are shown at Table II.

Table II: Ship model specifications

Global element length	0.26 m
Local element length	0.02 m
Number of shell elements	595977
Number of beam elements	81352
Quality Criteria	
Skewness (Nastran)	30
Aspect ratio (Nastran)	3

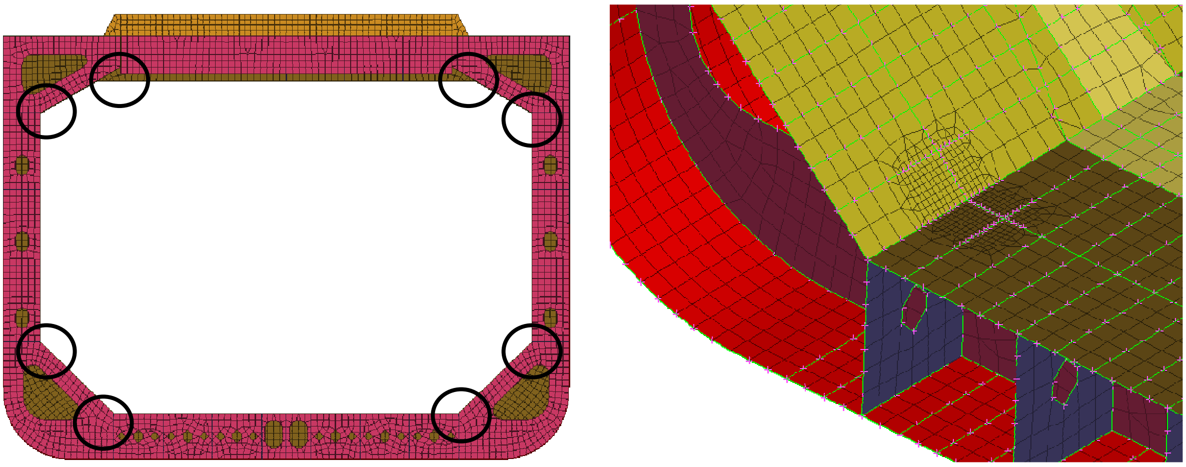


Fig.7: Refinement areas

The first step of the analysis is to apply still water loading conditions, in order to obtain the proper weight distribution for the analysis that will follow. The ship is at rest in a state of equilibrium between its own weight and cargo payload and the resultant buoyancy. The weight is calculated in ANSA from the mesh net, the properties of the shell and material characteristics, while the payload is applied as pressure inside the holds, using cargo's density. In this case all holds are considered full. The buoyancy is applied as hydrostatic pressure in the elements bellow waterline and varies linearly with water depth, Fig. 8.

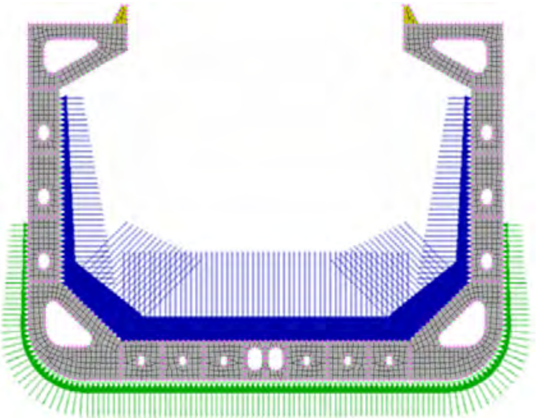


Fig.8: Cargo loads and buoyancy

The mass of auxiliary structures that doesn’t contribute to ships strength are added by three sets of non-structural mass at the bow, stern and middle respectively. This amount of mass is distributed among the sets in such portion to achieve balance without having trim angle in the still water load case. This is achieved by moving the center of gravity in such a position in relation to the center of buoyancy that the resultant force produces zero moments along the ships length and width. This procedure can be performed automatically with a special tool of ANSA that adds mass to specified areas of the model in order to achieve a target total mass and a target center of gravity. The areas where the added mass is distributed are shown at Fig. 9.

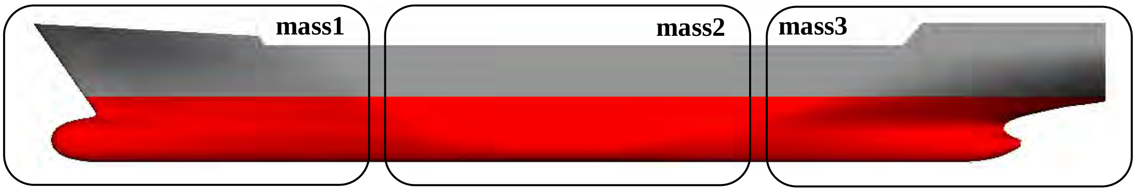


Fig.9: Added mass distribution

The next step of the analysis definition is the application of sagging loading condition. An 8 m height trochoidal wave is used and the balanced position is calculated by iteratively adjusting the draught and trim until the resultant net force and moment of the ship is ideally zero. The definition of the wave, the balance and buoyancy are again calculated by a special tool developed using the ANSA Scripting Language. Buoyancy is applied as PLOAD4 on hull elements, Fig. 10.

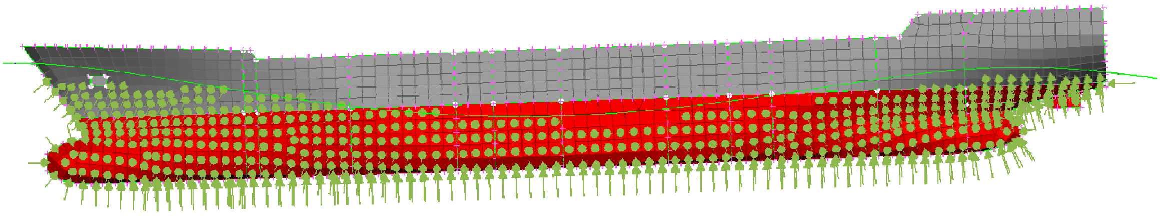


Fig.10: Balance on wave and buoyancy application

3.2. Analysis results

The model is solved with Nastran and the run lasted about 1 hour and 30 minutes in a dual core processor while the results are presented in μETA post-processor. The maximum developed Von Misses stresses in the cargo hold area are about 140 MPa, lower than yield stress of steel. High stress concentrations occur at the third and forth hatch coaming end brackets but the scantlings of the ship can be considered adequate since there appear no critical stresses. The standardize statistics tool can give an overview of the hull behavior while the areas of interest can be easily identified and displayed using annotations and iso-functions. Results from μETA are shown on Figs. 11 and 12.

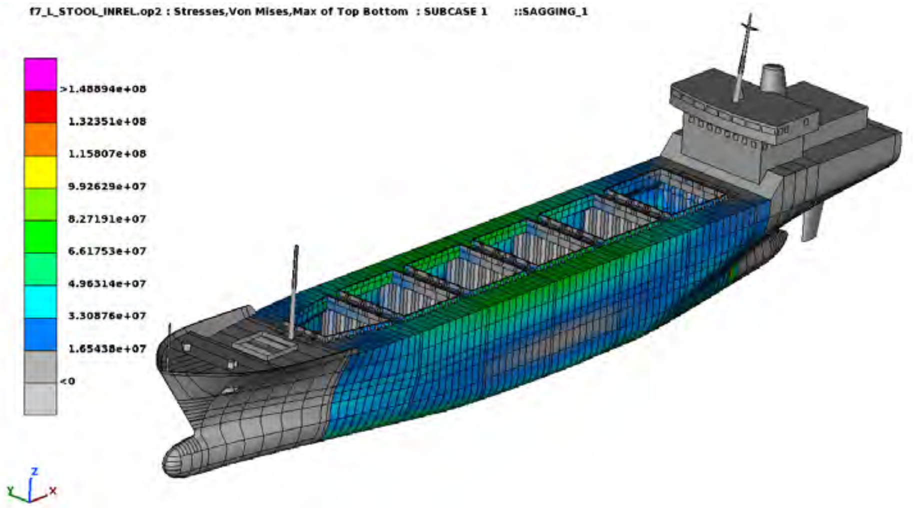
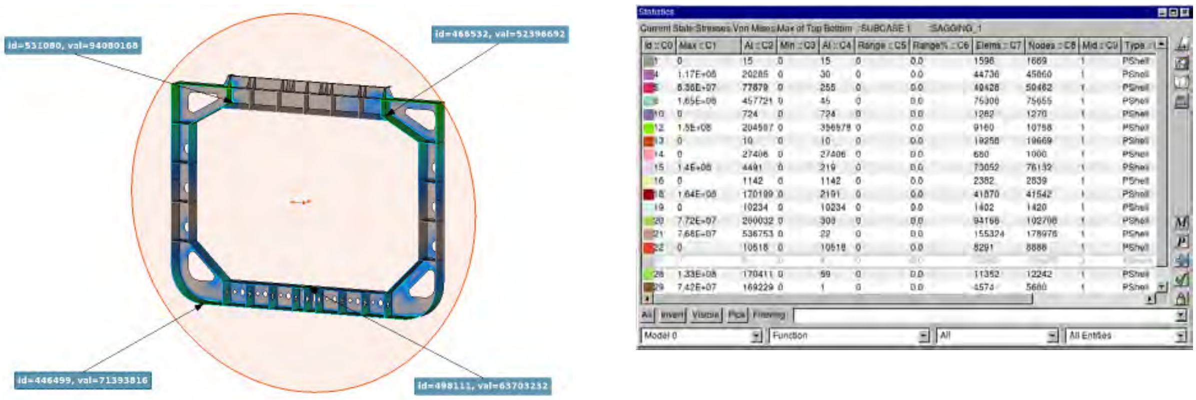


Fig.11: Von Mises stresses on the global model



Current State Stresses Von Mises Max of Top Bottom :SUBCASE 1 :SAGGING_1

Id : C0	Max : C1	Al : C2	Min : C3	Al : C4	Range : C5	Range% : C6	Elems : C7	Nodes : C8	Mid : C9	Type
1	0	15	0	15	0	0.0	1598	1669	1	PShell
4	1.17E+08	20285	0	30	0	0.0	44736	45860	1	PShell
5	8.38E+07	77879	0	255	0	0.0	49426	50462	1	PShell
8	1.65E+08	457721	0	45	0	0.0	75300	75655	1	PShell
10	0	724	0	724	0	0.0	1262	1270	1	PShell
12	1.5E+08	204587	0	356578	0	0.0	9160	10758	1	PShell
13	0	10	0	10	0	0.0	19256	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	2839	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	308	0	0.0	94196	102708	1	PShell
21	7.66E+07	536753	0	22	0	0.0	155324	178976	1	PShell
22	0	10518	0	10518	0	0.0	8291	8886	1	PShell
28	1.33E+08	170411	0	59	0	0.0	11352	12242	1	PShell
29	7.42E+07	169229	0	1	0	0.0	4574	5600	1	PShell

Fig.12: Annotations and statistics in μETA

4. Ships collision

The second case study tested in this paper is the collision between two identical bulk carriers. Collision mechanics are usually separated into external dynamics and internal mechanics. External dynamics deals with the rigid body global motion of the vessels and the effect of the surrounding water, while the internal mechanics is concerned with the structural failure response. In this case study only the internal mechanics is taken into account. To observe the behavior of both the holds and the bow, both ships are modeled as deformable bodies. Collision angle is chosen to be 90° and strike location at amidships. The initial velocity of the striking ship is 6 knots while the struck ship is standstill. Both vessels are loaded.

4.1. Model set-up

A part of both models that are not significant for the analysis are substituted by rigid bodies entities (CONSTRAINED_NODAL_RIGID_BODY) which contain the mass and inertia of the substituted model parts. The “Rigidize” treatment that is automatically applied within ANSA, minimizes significantly the calculation time and simplifies the model. At the striking model the middle and aft parts are substituted from rigid bodies while at the struck both fore and aft parts are substituted by such bodies, Fig. 13. For this case study the LS-DYNA explicit solver will be used.

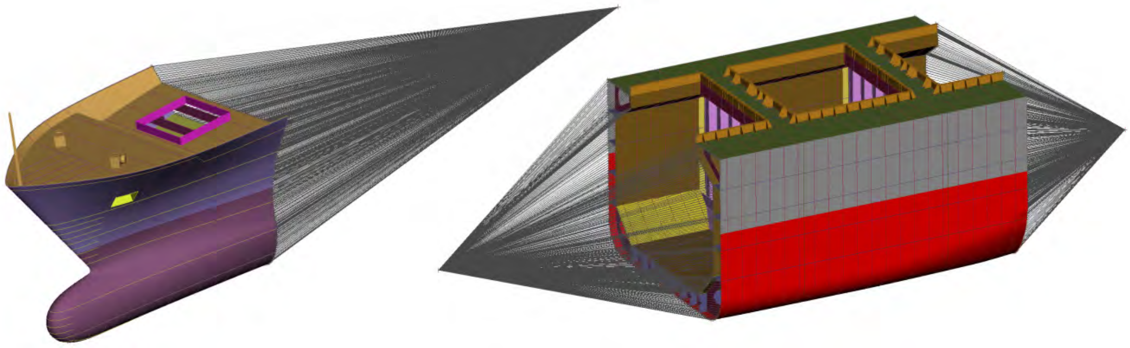


Fig.13: “Rigidized” models

The meshing parameters and quality criteria for the collision analysis are prescribed at the ANSA Batch Meshing Tool. Both models are meshed with mean element length of 0.13 m. However at the collision area of both models fine mesh of 0.06 m is applied to ensure accurate results. Transition areas are also provided to connect coarse and fine mesh. Since the beams that represent the stiffeners are pasted on the shell elements, a re-meshing action on the shells updates the beams definition. This is an automatic process in ANSA which redefines any entity is attached on shells after their re-meshing. At the area of local refinement new beams are created. This technique eliminates the need of redefining the beams in every change of the model mesh, Fig. 14.

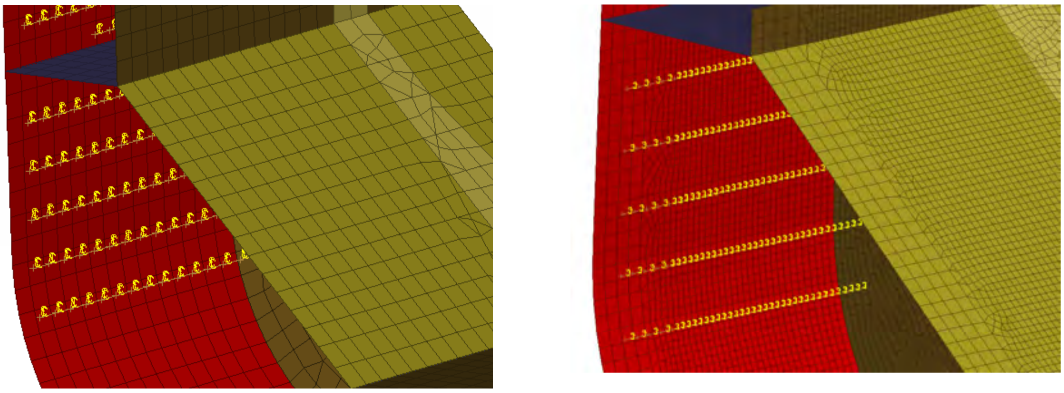


Fig.14: Local refinement and beam re-definition

Velocity and gravity are applied to the striking model. The balance of the striking model is defined by the applied gravity and a rigid wall at the model bottom. Thus, the model slides on the rigid wall until hits the target. The struck ship is restrained in all degrees of freedom by constraining the nodes movement of the outer hull on the side that is not going to be hit. Contact entities of type AUTOMATIC_SURFACE_TO_SURFACE are defined between the two models and AUTOMATIC_SINGLE_SURFACE for each of the models. The last ones are defined to eliminate any penetration between the shells of the model itself during the collision. All the above entities are prescribed as sequential steps at the Task Manager. These actions are realized when the Task Manager is invoked. Meshing information and quality criteria are shown at table III. The two models are assembled in one FE model as shown at Fig. 15.

Table III: Ship model specifications

Global element length	0.13 m
Local element length	0.06 m
Striking ship	
Number of shell elements	473066
Number of beam elements	25355
Struck ship	
Number of shell elements	853114
Number of beam elements	46620
Quality Criteria	
Skewness (Nastran)	30
Aspect ratio (Nastran)	3
Crash time step	7.0E-6 s

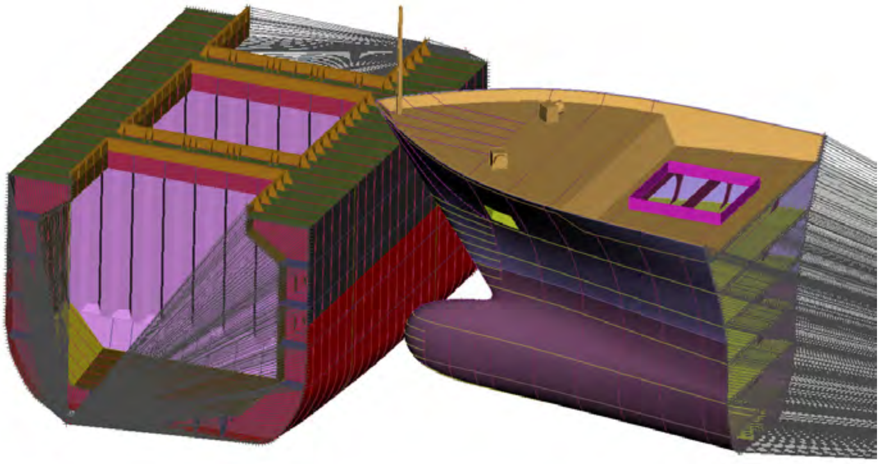


Fig.15: The assembly model

4.2. Analysis results

If the external dynamics had been taken into consideration, then the impact energy would have been consumed in both the structural deformation and water resistance. In this study, in which only the internal mechanics are taken into account, the whole impact energy is absorbed by the deformable vessels. As a result, this load case can be considered as the worst case scenario. The results though showed that only the outer hull is penetrated by the bow's bulb while bow's upper edge is severely damaged. The spring back effect started 2 s after the impact moment. Striking force, velocity and damage are shown at Figs. 16 and 17. The time needed to complete a 4 s simulation, was approximate 24 h using a cluster of 16 processors.

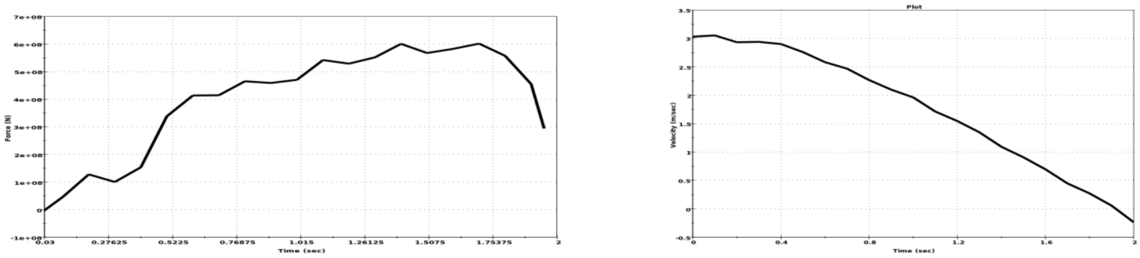


Fig.16: Striking force and velocity

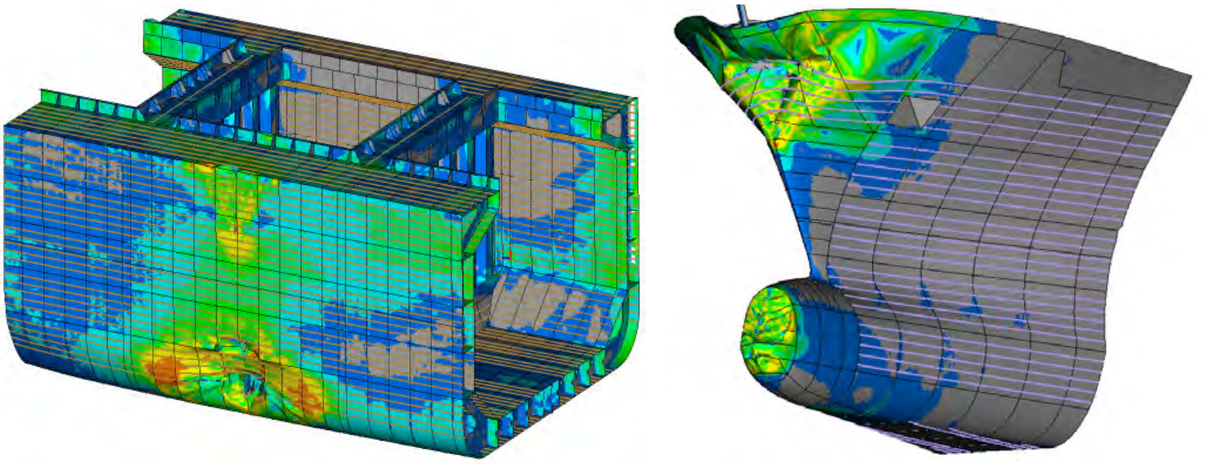


Fig.17: Collision results in μETA

5. CFD analysis

The third case study is a CFD analysis of the same cargo ship. The computational domain of this simulation consisted of the cargo vessel geometry and an open ocean area. The vessel geometry has been meshed with a variable size surface mesh with additional refinement on the hull region, Fig. 18.

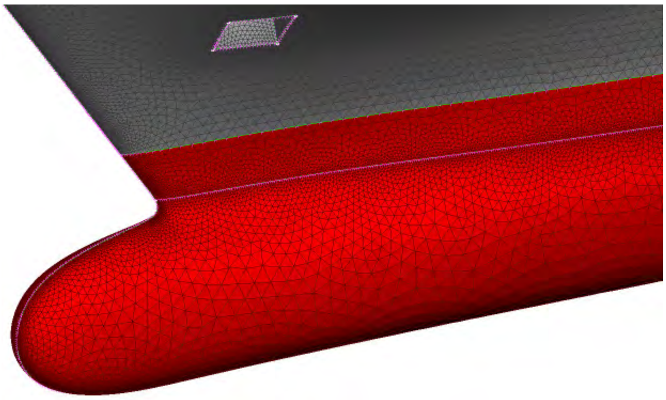


Fig.18: The CFD surface mesh

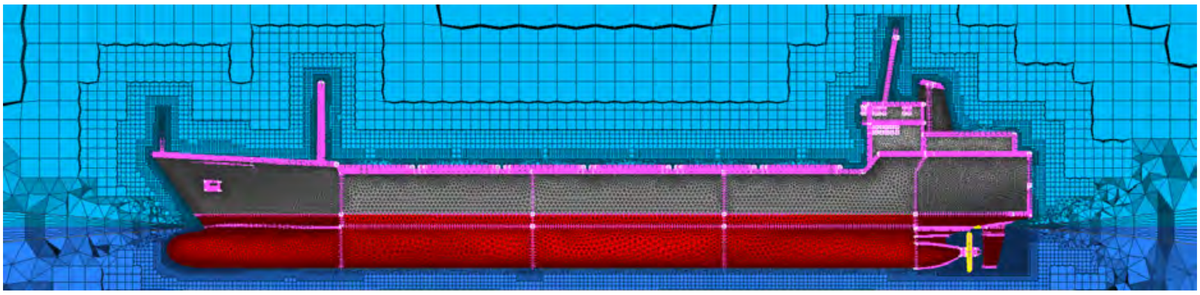


Fig.19: The hexa hybrid mesh

The water level has been modeled with a separation surface between the two fluid regions, running across the hull at a mid-distance. The boundary layer, generated on the vessel and the fluid surface separation, consist of five layers of prisms, generated in aspect mode with a growth factor of 1.05 and first height of 10 mm. A hexa hybrid mesh has been generated on both fluid domains with a total size of nearly 11 million hexas and polyhedrals, Fig. 19. The whole meshing process of the vessel and the fluid domains is elaborated through the ANSA Batch Meshing Tool.

Local refinement is applied at areas of interest such as the bulbous bow and the propeller and rudder. Special entities of ANSA, the SIZE BOXES are defined in these areas. Fluid domains that reside inside the SIZE BOXES volume are meshed with a prescribed element length. SIZE BOXES can have any shape in order to simulate any refinement area, Fig. 20.

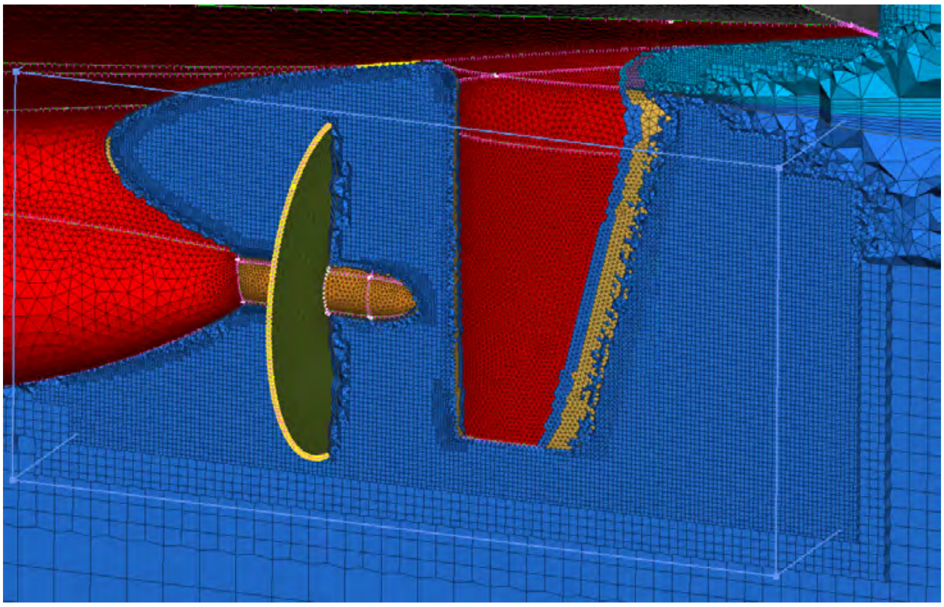


Fig.20: Local refinement using SIZE BOXES

6. Optimization

The model behavior according to the above defined analyses can be improved by defining an optimization process which can alter model shape and properties. Such properties can be parts thickness, stiffeners cross sections, material properties, etc. The shaping of the model is achieved through the ANSA Morphing Tool. This tool provides several ways of morphing FE or geometrical models while ensures smooth, predictable and controllable results. Since the ANSA Morphing Tool is able to control a ready to run FE Model, there is no need of re-defining any entities like mesh, loads, boundary conditions, etc. The model is ready to run just after shaping. Thus, the defined shaping or parameterization process can be easily coupled with an optimizer and run in batch mode within the optimization loop.

Morphing in ANSA is performed by special entities, the Morphing Boxes. These entities are created around the area of the model to be modified. As the shape of the Morphing Boxes can be modified in several ways, the model surrounded by them follows the modification and the shaping takes place. Design variables can drive parametrically the morphing process. This enables the connection of any shaping action to an optimization process through the design variables. In this example morphing is performed on the bulbous bow of the ship. A design variable drives the position of the bow along the Z axis. In this CFD model the Morphing Boxes control the ship geometry, the surrounding fluid and layers. After morphing there is no need to redefine the mesh so the model is ready to run to the solver, Fig. 21.

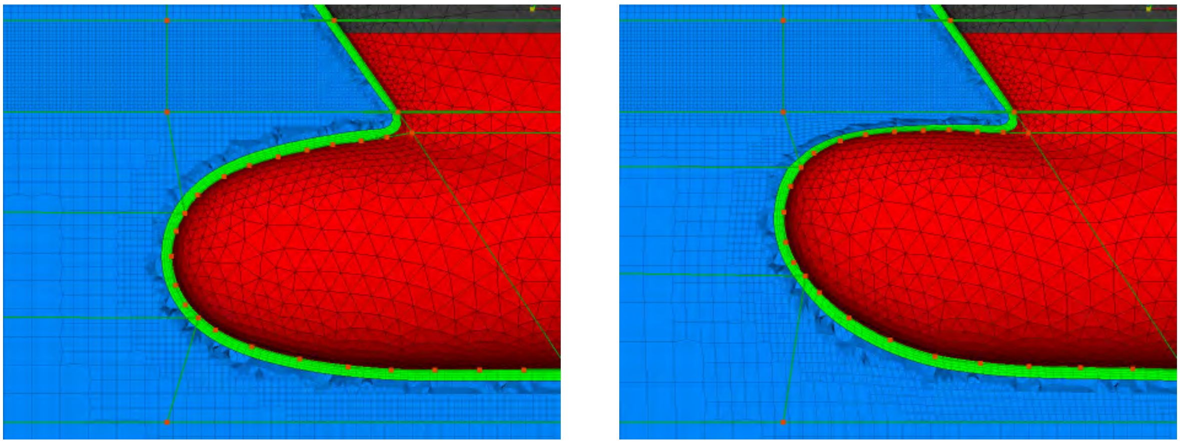
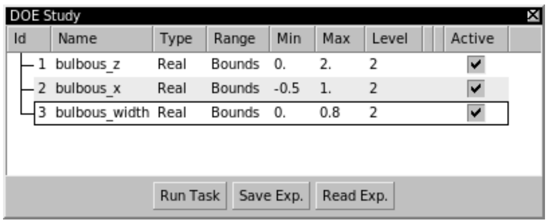


Fig.21: Local refinement using SIZE BOXES

Model behavior and validity check for different shapes can be performed by the DOE functionality provided in ANSA. A Full Factorial algorithm creates several experiments upon user's request. In this example three design variables are defined to alter the bulbous bow shape. A DOE study runs to define several experiments as shown at Figs. 22 and 23.



DOE Study

Id	Name	Type	Range	Min	Max	Level	Active
1	bulbous_z	Real	Bounds	0.	2.	2	☑
2	bulbous_x	Real	Bounds	-0.5	1.	2	☑
3	bulbous_width	Real	Bounds	0.	0.8	2	☑

Fig.22: The DOE tool

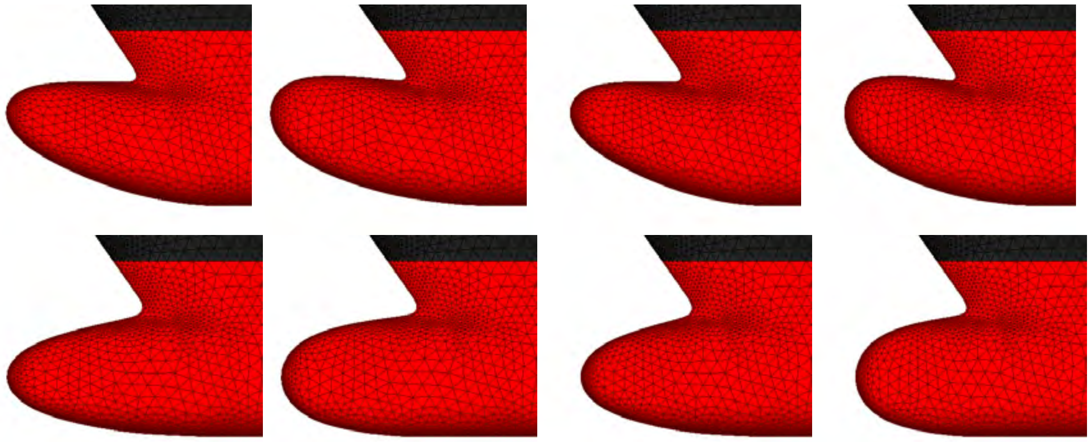


Fig.23: Design experiments of the bulbous bow

7. Conclusions

ANSA is able to set up efficiently several CAE analyses for different disciplines and load cases. Process organization and standardization is possible using the Task Manager tool. Great scatter can be achieved in simulation results caused by slight changes in model parameters. Special tools that needed in marine design have been developed using ANSA Scripting Language. Applications like wave creation, mass balance, vessel balance on a wave profile, buoyancy calculation and cargo loading can be automated using these tools. μETA is a versatile post-processor which provides sophisticated tools for results reporting and evaluating. ANSA also provides powerful model shaping and a versatile optimization set-up tool which is able to automate the whole definition of the CAE model. The Optimization Task also provides easy coupling with most of the commercial optimizers.

References

PHELPS, B.P. (1997), *Determination of Wave Loads for Ship Structural Analysis*, pp.19-24

CHE, X.L.; FAITH, L.K.; DARON, L.H. (1999), *Simplified stress analysis of ship structures*, pp.4-7

SERVIS, D.; SAMUELIDES, M. (1999), *Ship collision analysis using finite elements*, Safer Euroro Spring Meeting, Nantes, pp.3-16

SERVIS, D.; VOUDOURIS, G.; SAMUELIDES, M.; PAPANIKOLAOY, A. (2001), *Finite element modeling and strength analysis of hold No. 1 of bulk carriers*, pp.3-10

IACS, (2006), *Common Structural Rules for Bulk Carriers*, Chapter 4 pp.14-43

OZGUC, O.; DAS, P.K.; BALTROP, N.; SHAHID, M.; SAMUELIDES, M. (2006), *Numerical modeling of ship collision based on finite element codes*, pp.1-9

QUINTON, W.T. (2007), *Ship Collision Mitigation: Redesign of an Oil Tanker Side-shell*, 8th Canadian Marine Hydromechanics and Structures Conference, St. John's, pp.1-8