Automated Interoperability from Concept Design to Multidisciplinary FE Analysis

George Korbetis, BETA CAE Systems, Thessaloniki/Greece, korbetis@beta-cae.gr
Serafim Chatzimoisiadis, BETA CAE Systems, Thessaloniki/Greece, chatzimos@beta-cae.gr
Dimitrios Drougkas, BETA CAE Systems, Thessaloniki/Greece, d.drougkas@beta-cae.gr

Abstract

This paper, presents a method through ANSA pre-processor to define multiple models which can serve different simulation analyses. The process exploits the ability of the ship design software to output CAD models of varying detail level, to create a model database which contains the whole history of design. The task manager tool of ANSA defines automatically, different representations of the ship model according to the FE analysis that will be performed. Additionally, this method allows the automatic update of the FE model when a new version of a geometrical part or assembly arises.

1. Introduction

The use of CAE in the maritime industry, today, has become a necessity since workload has increased and the quest for efficiency drives the need for interoperability and shorter process times. A suggestion of a process starting from the CAD model and producing multiple FE (finite element) models for different loading conditions is a step forward to the more effective use of CAE tools and the increase of productivity. A great effort has also been made in the ability to fully use all the valuable information and details that come with maritime structure CAD models, in the development of more accurate and usable (configurable and able to be solved) FE models.

2. Process management

During the design process of a new model it is very important to perform analyses during the early design stage to guide the design before the final decision making. As soon as more detailed CAD models arise, more detail should be included to the CAE models and the analyses that taking place. The recommended process, from BETA CAE Systems S.A., uses the Data Management and ANSA Task Manager tools to define such processes by setting up the data workflow and the sequence of actions from CAD to CAE.

Initially, all existing information such as geometrical data, model hierarchy, materials, performed analyses, meshing quality criteria, etc. are collected in a data pool and can be accessed from the Data Management tool of ANSA. When newer or more detailed versions of any part or assembly arise, the Data Management system is informed automatically. Now, the engineer can select the version and consequently the level of detail for each part or assembly that comprise the global model in order to set up the relative FE analysis, Fig. 1.

A second step of the process management is to set up the FE analyses that will be performed. The steps of each analysis are prescribed in a step-wise process through the Task Manager tool of ANSA and are applied automatically on the model when it’s needed in order to define a ready to run FE model. According to each analysis different level of detail and different meshing representation is needed for each part. While the version of the part to conform the requested detail can be selected from the data pool, the different meshing representations are created within ANSA by meshing the part with prescribed meshing parameters and quality criteria. In the following chapters the whole process is described for a crude oil carrier.

Fig. 1: Data work flow

2.1. Geometry Import

CAD data are imported in the data pool as neutral files (.stp, igs) or through a translation process from CAD software native files. The geometry of the concept and the detail models are checked, and existing topological errors are automatically identified and fixed. Using special scripts additional information is extracted from the CAD models such materials, thickness, model hierarchy, stiffeners position, etc. and is saved in the data pool. The concept and the detailed models are shown at Figs. 2 and 3.

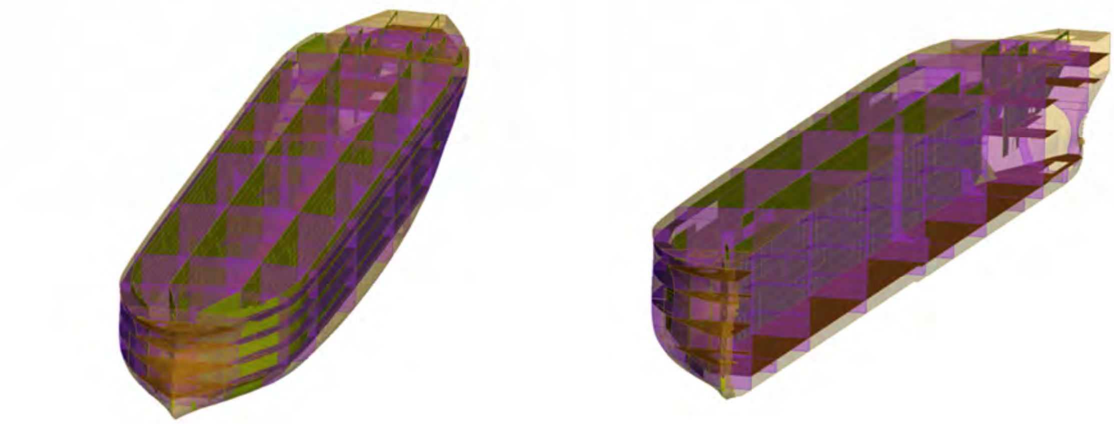

Fig.2: The ship geometry concept model

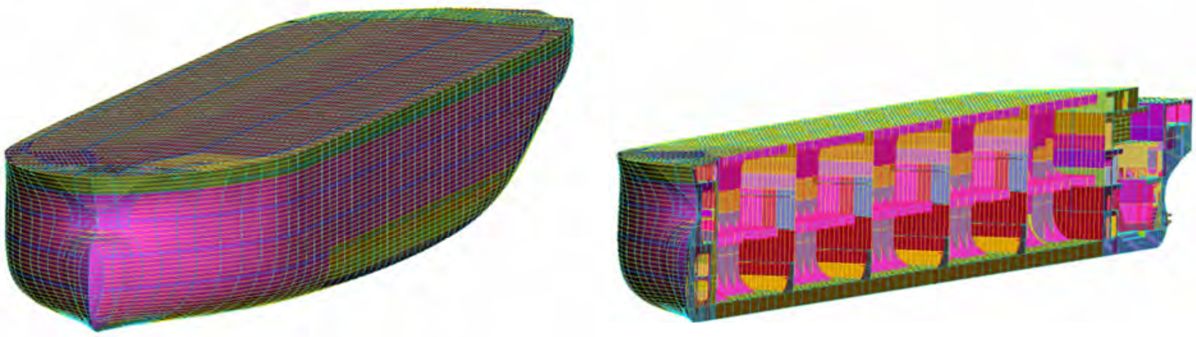

Fig.3: The ship geometry detailed model

2.2. Mesh representations generation

As the next step the FE mesh is generated on both models. The process is automatic and performed by a special tool, the Batch Meshing Tool of ANSA. The meshing characteristics and quality criteria are predefined for each part of each model, depending on the type of the solution, the position of the part on the model and their significance on the global model. So, one part can have numerous mesh representations which are stored at the data pool. Details such as holes, fillets and flanges handling are specifically defined according to the specifications of the analysis. Limit values regarding the quality of the generated mesh are also defined in order to prevent the generation of violating elements. Also, specific mesh type is predefined for selected parts of the model.

2.2.1 The concept model

Prior to the definition of a full scale analysis of the vessel and during the design process of the ship, a simplified CAD model can be used for a draft FE structural analysis. This early stage analysis can give significant feedback to the design process of the product. In that case the CAD model is meshed with a coarse element length. Holes with diameter under 3 [m] are filled. Stiffeners are excluded from this model but their stiffness is enhanced in the panels defining an orthotropic material. The information of such materials is transferred along with the CAD data to the FE model. Meshing parameters and quality criteria of the concept FE model, Fig. 4, are listed at Table I.

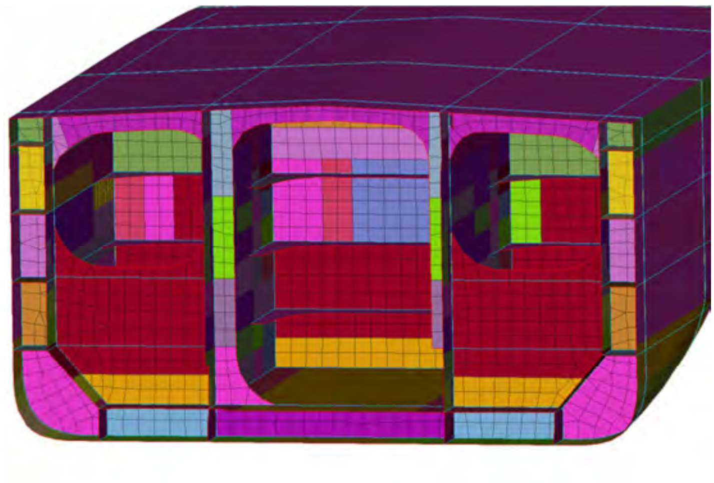

Fig.4: The concept model

Table I: Meshing parameters and quality criteria for the concept model.

Meshing Parameters	
Elements type	First order quads
Global Element Length	1.5 [m]
Features treatment	
Holes with diam. <3 [m]	fill
Quality Criteria	
Skewness (Nastran)	30
Aspect ratio (Nastran)	3
Warping (IDEAS)	10

2.2.2. The detailed model

The detailed model can be created after all geometrical details are available in the CAD model. It will give more accurate analyses and it would be a reference FE model for the generation of various analyses and loading cases. Any holes with diameter under 0.5 [m] are filled and zones of elements are generated around larger holes. Table II shows the meshing parameters and quality criteria, Fig. 5 the FE model.

All settings (meshing parameters and quality criteria) within the Batch Meshing Tool define a meshing scenario that can be saved and be reused for different models. Each meshing scenario can apply different meshing parameters on different parts of the model, Fig. 6. Thus, parts of great importance can have finer mesh while parts that do not contribute to the model strength can have coarser mesh. By running the meshing scenario the mesh is automatically created and quality improvement takes place to fulfil the pre-defined quality criteria. Three meshing scenarios will be defined which will mesh the detailed model for sagging, hogging and fatigue analysis.

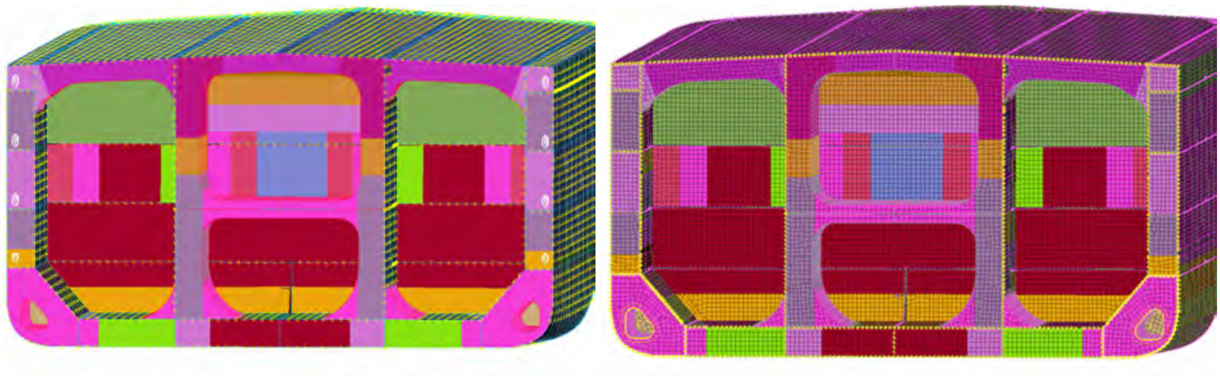

Fig.5: Meshing-Holes treatment of detailed model

Table II: Meshing parameters and quality criteria

Meshing Parameters	
Elements type	First order quads
Global Element Length	0.6 [m]
Features treatment	
Holes with diam. <0.5 [m]	fill
Holes with diam. >0.5 [m]	one zone of elements
Quality Criteria	
Skewness (Nastran)	30
Aspect ratio (Nastran)	3
Warping (IDEAS)	10

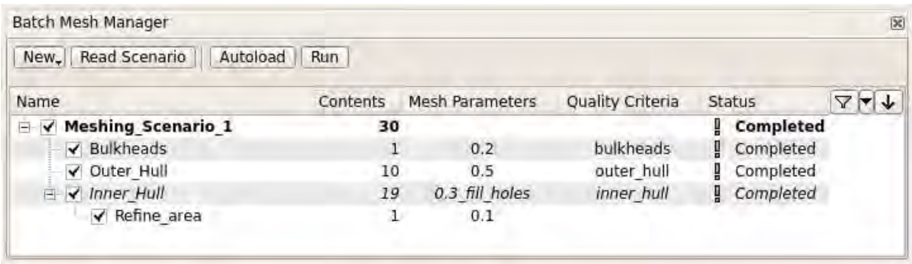

Fig. 6: Batch mesh definition

Following the surface meshing, all the stiffeners are massively replaced by 1-D beam elements in an automated way. The beam elements have new beam properties that contain the cross section's characteristics. Cross sections are recognized and created from the shape of the existing meshed stiffeners, Fig. 7. During mesh improvement actions or mesh representation change, beam elements also follow the mesh changes (length change), stay connected with the model and keep their properties, Fig. 8.

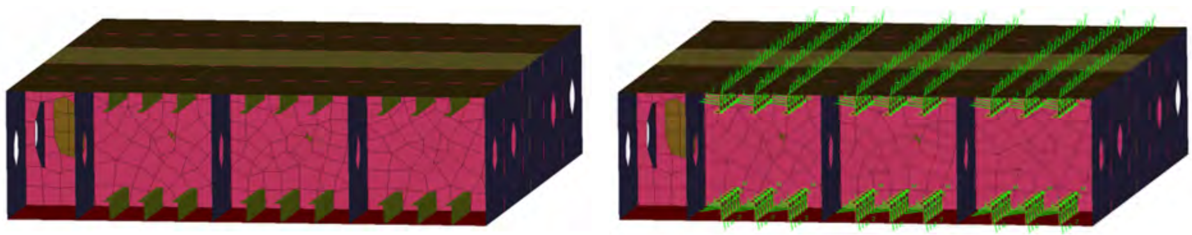

Fig. 7: Replacement of stiffeners with beams

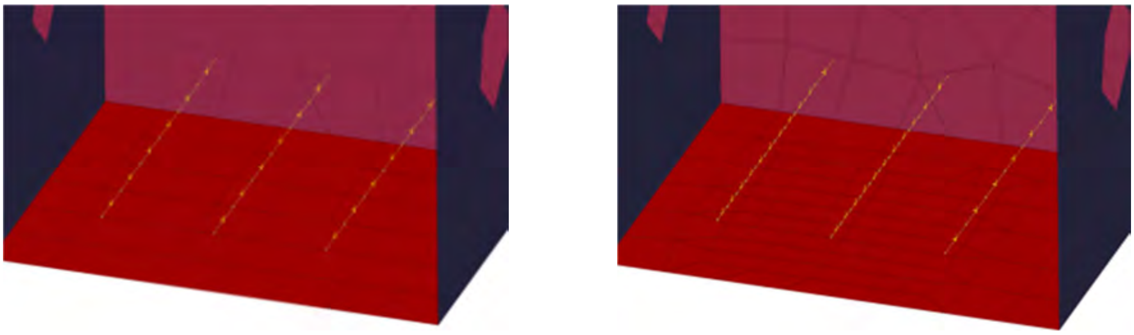

Fig. 8: Re-meshing of stiffeners

The meshing scenario for the first loading condition composes of three different mesh definitions, one coarse for the upper outer hull, a second finer definition for the lower outer hull and inner hull and one even finer for the bulkheads, Fig. 9, Table III.

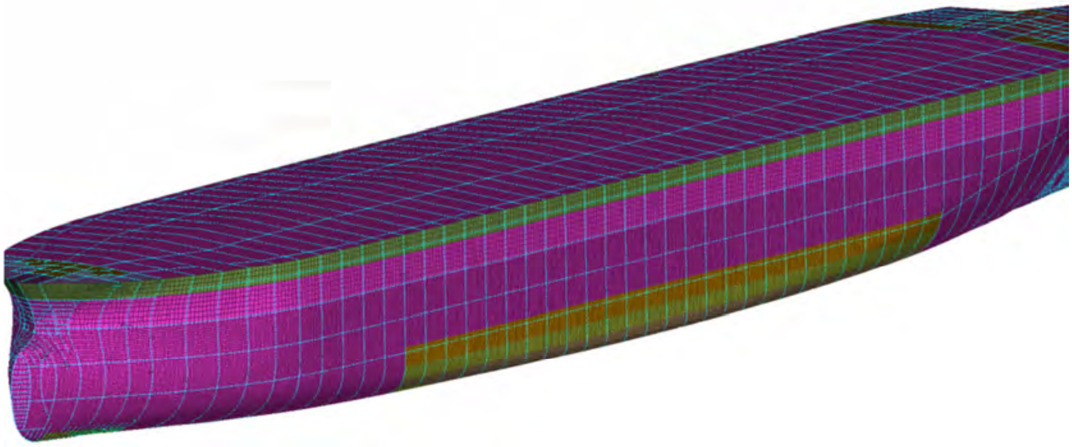

Fig. 9: Meshing Scenario 1- Sagging

Table III: Meshing Scenario 1 representation details

Shells	709755
Quads	676935
Trias	32820
Mean Element Length	0.48 [m]

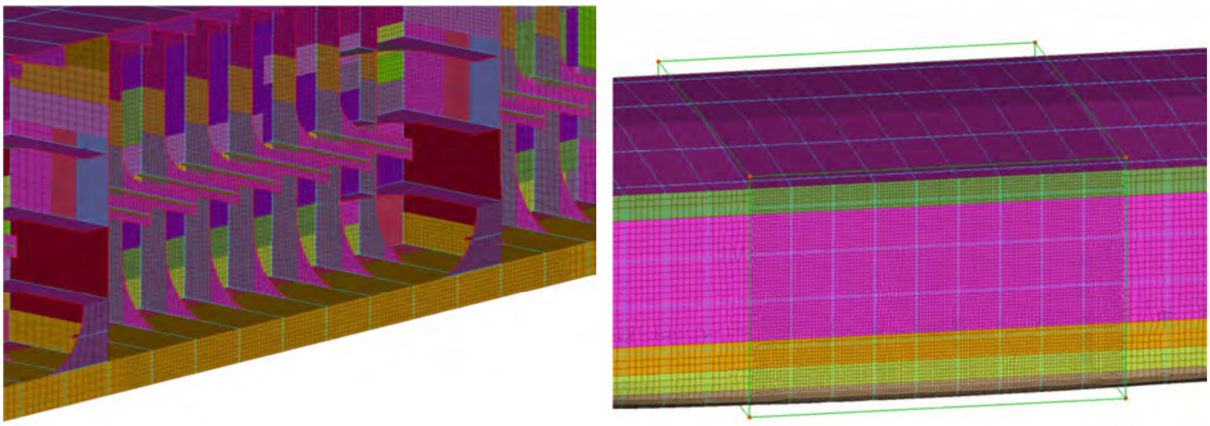

Fig. 10: Meshing Scenario 2- Hogging

The meshing scenario for the second load case composes of three different mesh definitions, a coarse mesh type for the stern and bow of the ship, a finer mesh type for the inner hull and bulkheads and a box with a finer mesh definition at the middle length of the ship, Fig. 10, Table IV. This box encompasses the critical area of the vessel for the hogging load case. This way more accurate results will arise in the area that is expected to be mostly affected by this specific load case.

Table IV: Meshing Scenario 2 representation details

Shells	387973
Quads	361036
Trias	26937
Mean Element Length	0.65 [m]

The meshing scenario for the fatigue loading condition is composed by three different mesh definitions, one coarse for the outer hull, a second finer definition for the inner hull and one even finer for the bulkheads, with an addition of one item. A box was added locally at a critical area of the ship, with mesh refinement for more detailed results for the fatigue analysis, Fig. 11.

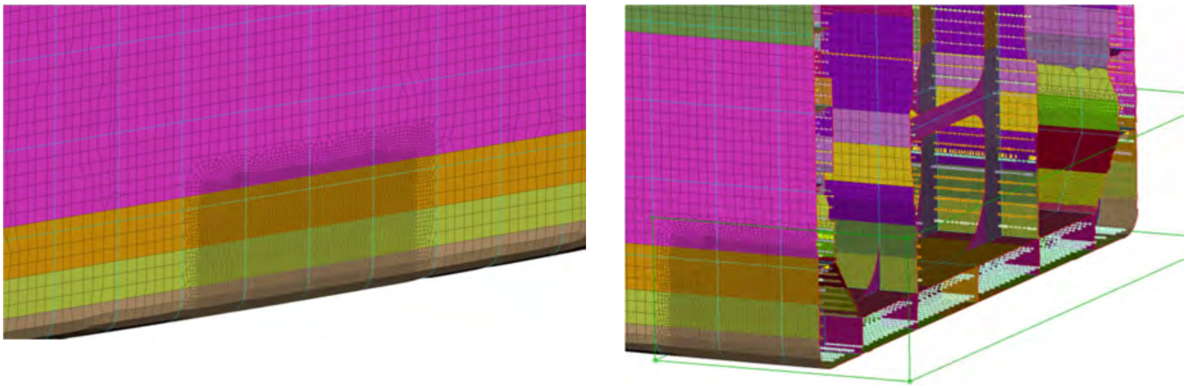

Fig. 11: Meshing Scenario 3- Fatigue

Table V: Meshing Scenario Fatigue representation details

Shells	553003
Quads	523816
Trias	29187
Mean Element Length	0.45 [m]

3. ANSA Data Management

Mesh representations of the model's parts are "stored" in a specified storage space "pool". The saved meshing representations are used on demand according to the analysis type. For this work, after running the first Meshing scenario, every part of the model is meshed with the initial mesh definition. This mesh representation is saved in the Data Repository for each part separately. The first representation is saved under the name common, Fig. 12.

Continuing, the model is re-meshed when the second (Hogging) Meshing scenario is applied and all the mesh representations of the re-meshed parts with the new mesh definition are saved in the Data Repository. The third (Fatigue) Meshing scenario is applied in a similar manner creating the third available mesh representation. Through the Data Management functionality it is possible for each part to change between the existing representations, creating different model for a different load case. With this functionality it is also possible to create more representations for a specific part thus creating multiple models.

In the likely event that a part of the model is re-designed and needs to be replaced in the original model, an automated check for updated versions of each saved part recognises the new version of the part (that is saved in the Data repository) and can replace it in the model, Fig. 13. The new part is meshed and connected to the rest of the updated version of the FE model

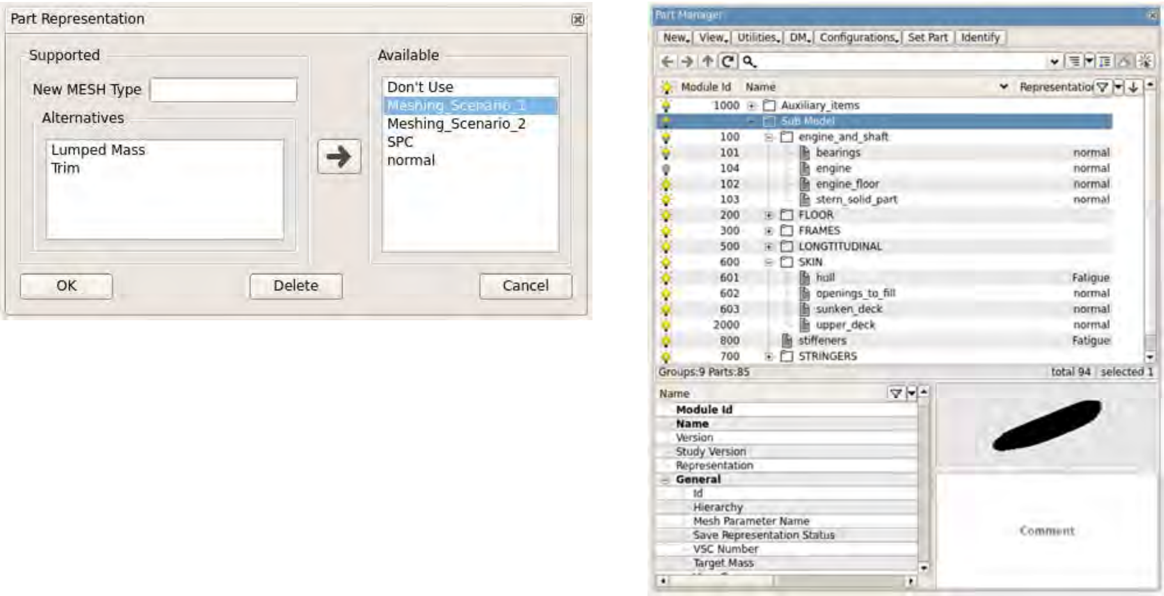

Fig. 12: Part Representation Change - Selection

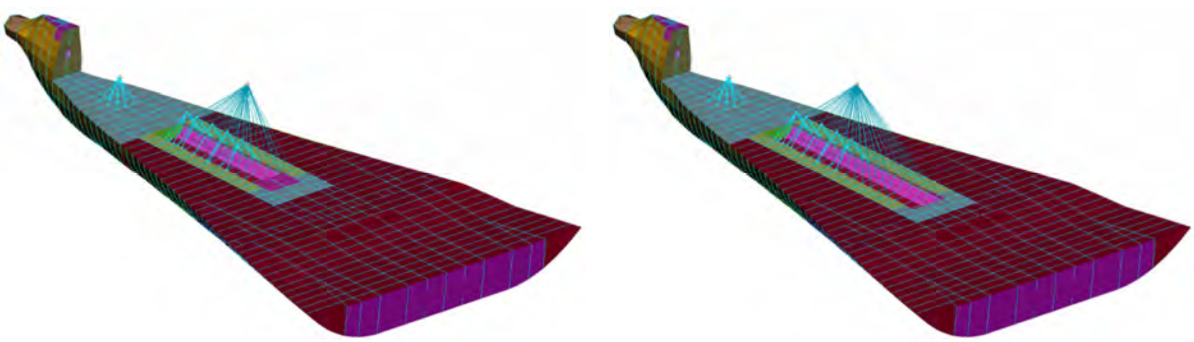
Fig. 13: Part Replacement

4. Loading – Boundary Conditions

In this step the loading conditions Sagging, Hogging and Fatigue are defined. All keywords and actions needed for these three analyses are pre-defined in a special tool of ANSA, the Task Manager. The Task Manager is able to apply the pre-defined actions in a stepwise manner on the meshed model. The above actions are not dependent on the model mesh so, they can be applied on any of the meshing representations creating a variation of ready-to-run FE models, Fig. 14. The actions taking place are described at the following paragraphs.

Auxiliary structures and machinery that do not contribute to vessel's strength are modeled as non-structural mass. This mass is appropriately distributed over the FE model, so as to reach the prescribed lightship weight and the corresponding center of gravity. This procedure is performed automatically through the Mass Balance tool of ANSA which adds mass to specified areas of the model in order to achieve a target total mass and a target center of gravity, Fig. 15.

For the definition of the Sagging and Hogging analyses the ship is considered to be fully loaded. A typical loading condition is selected where the all holds and auxiliary tanks are full while the water ballast tanks are empty. The loading is performed in an automated way, with the definition of each tank's area and the centre of gravity of the load. A concentrated mass which represents the tanks load is distributed on the tank area by an RBE3 element, Fig. 16.

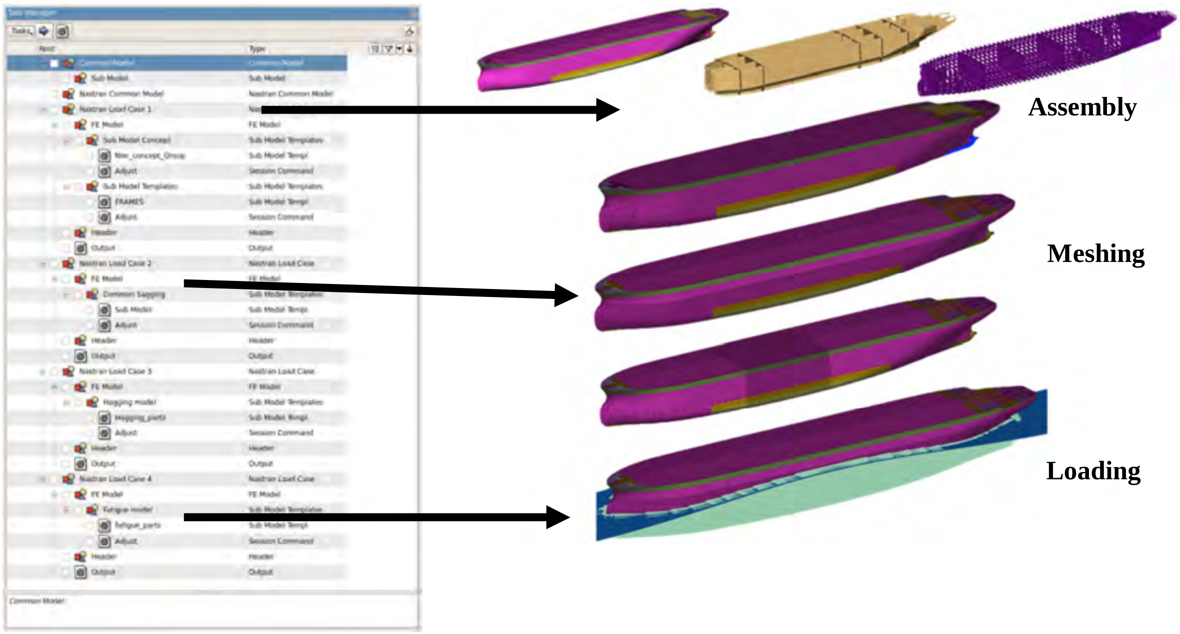

Fig.14: Task Manager

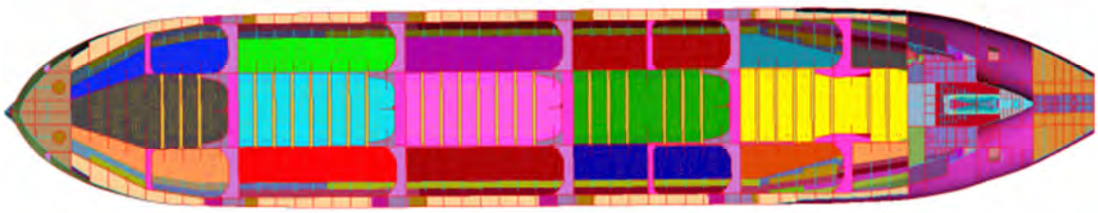

Fig.15: Adding non-structural mass

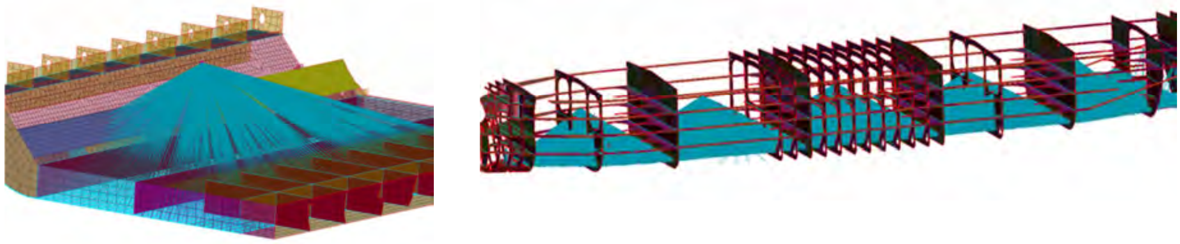

Fig.16: Tank loading

Finally the vessel is positioned and trimmed on a trochoidal wave where static equilibrium should be obtained. A special tool developed using the ANSA Scripting Language is used to iteratively adjust the vessel until the equilibrium is achieved between weight and buoyancy and buoyancy is applied as PLOAD4 on hull elements underneath the water, Fig. 17.

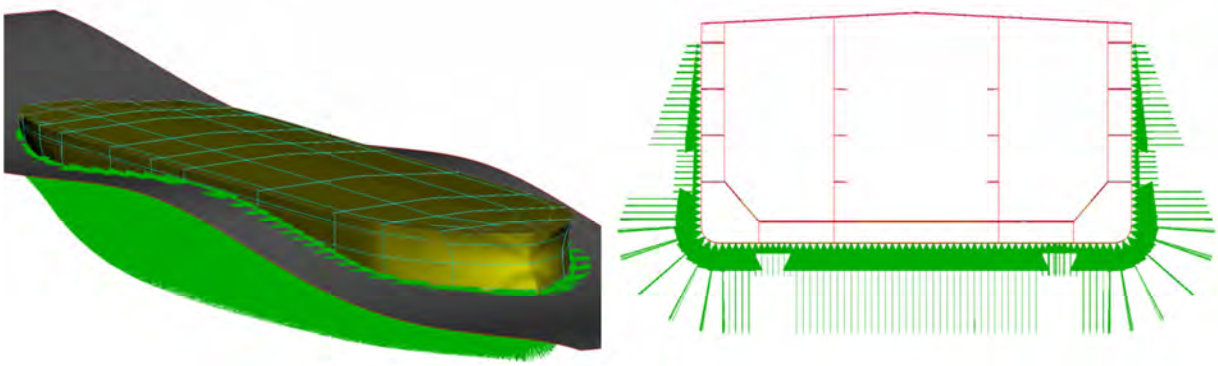

Fig.17: Loading Condition 2

5. Automation

All the above actions that have been used to define a single analysis have been recorded in the Task Manager process. These actions can be executed again in another meshing representation in order to define a new analysis. Additionally the parameters of these actions can be altered in order to define a new loading condition. Specifically for the ship loading, the masses values and CoG can be extracted from a data sheet which contains all loading conditions that the ship may be subjected. So numerous FE models, one for each loading condition can arise in a single step, Fig. 18.

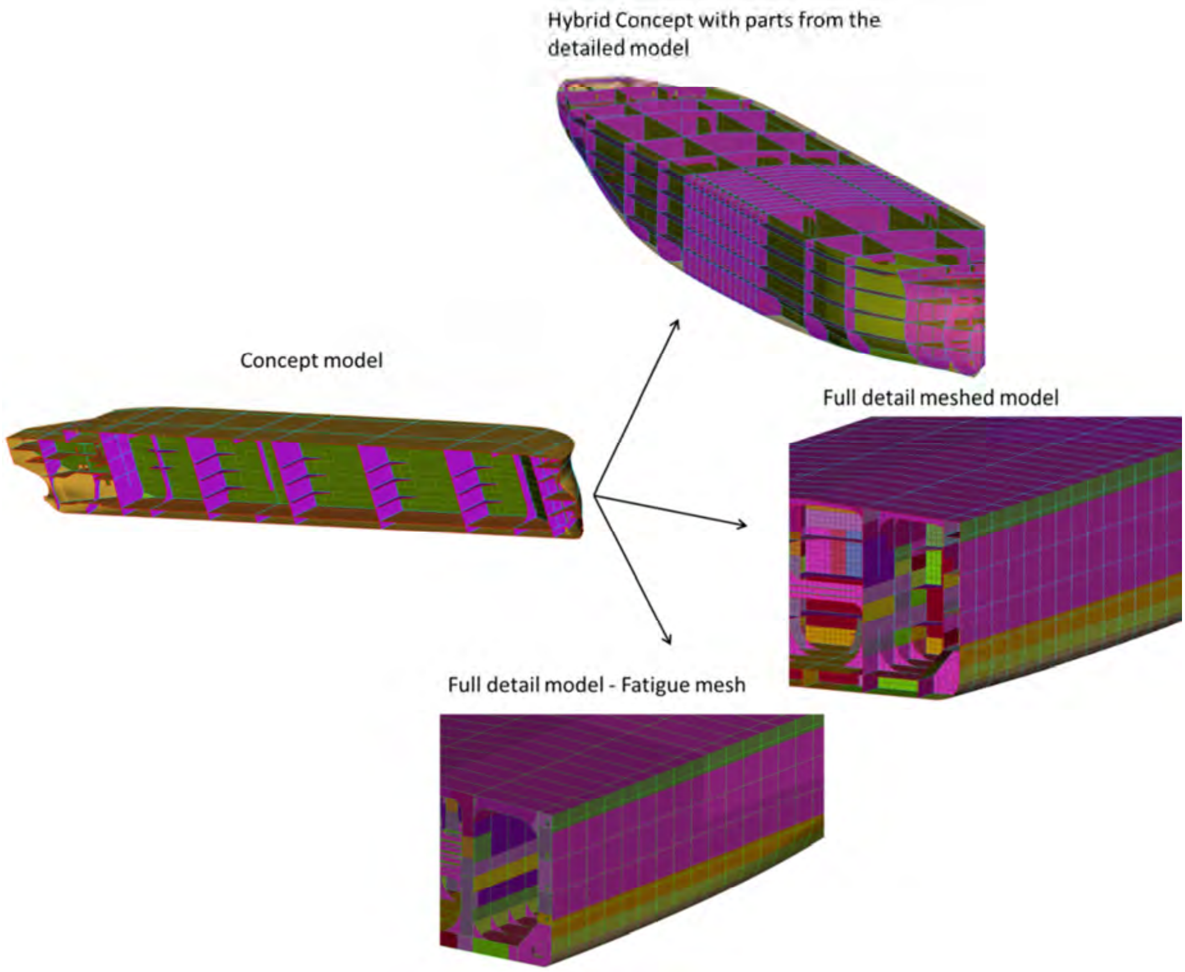

Fig. 18: From Geometry model to multiple FE models

6. Conclusions

This paper presents an effective process starting from a CAD concept model of a ship and resulting to multiple FE-models for several loading conditions. Design information and details concerning material properties, stiffeners, cross sections, tank loading and equipment masses are kept through the process and are used as input data in the produced FE models. This process ensures that all valuable information is passed from the CAD design to the final analysis, resulting in a more accurate FE model.

Process organization and standardization is possible using the ANSA Task Manager tool. Finally any special tools needed for the definition of loading conditions in the marine design such as, mass balance, vessel balance on a wave profile, buoyancy calculation, and cargo loading are provided by the ANSA software.

References

CHE, X.L.; LEE, F.K.; LIBBY, D.H. (1999), *Simplified stress analysis of ship structures*, ABS, pp.4-7

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

PHELPS, B.P. (1997), *Determination of wave loads for ship structural analysis*, DSTO Aeronautical and Maritime Research Laboratory, Melbourne, pp.19-24

RAWSON, K.J.; TUPPER, E.C. (1997), *Basic Ship Theory*, Elsevier, pp.555-563