Increasing Efficiency in the Ship Structural Design Process

Tomi Holmberg, Napa Ltd, Helsinki/Finland, tomi.holmberg@napa.fi
Shaun D. Hunter, DRS Defense Solutions LLC, Stevensville/USA, SHunter@drs-ds.com

Abstract

This paper presents new possibilities and practices that can enable shorter design cycle times by utilizing currently available commercial-off-the-shelf structural modeling technology combined with full-ship FEA and limit state analysis technology. Essentially, the use of a single 3D structural design model in both design and analysis has the potential to reduce the overall time needed in the design process by decreasing the time needed to run the design cycle anew for another variation, resulting in better designs

1. Introduction

Traditionally the early structural design of ships is done in 2D, i.e. the design information is stored in 2D drawings. All the information needed to validate design integrity is read from the drawings and input to several different systems where the actual feedback for the structural integrity is coming. The same information is re-entered multiple times making the process very inefficient and exposed to human errors. Often, the information for a structural design calculation is not required to be as a 3D representation of the structures because cross-sectional information is adequate enough (e.g., information for longitudinal scantling checks).

The benefits of having 3D information right from the beginning of the design process has been discussed for years; however, few organizations have changed their way of working to support this notion. The main obstacle is the resistance to change the way of performing design work. When 3D information is the main source of the design input, it requires a change in the existing working methods in order to realize any benefits of this 3D approach. It is necessary to implement a new way of working for the ship engineering process in which the project team collaborated for utilization of the product model. The previous big change in the design process was done when moving from hand-drawing to computer aided drafting tools. This change was in some ways much easier to do, because the design process could remain the same as the design information was stored the same way, in 2D drawings, but now in electronic format.

One major statement against 3D models is it takes time to create one. Everything is relative. It depends on the type and the quantity of the design information to justify if 3D models bring any benefit. If the requirement is to create just a few structural drawings then it is hard to speak on behalf of 3D models. The drafting is not the only task needed to be carried out in order to get the structural drawings ready. For example, the scantling information needs to be initialized and created, which means additional information to support particular calculations needs to be derived. The 3D model has this type of information; therefore, the 3D model can help to reduce the time for carrying out these types of additional activities.

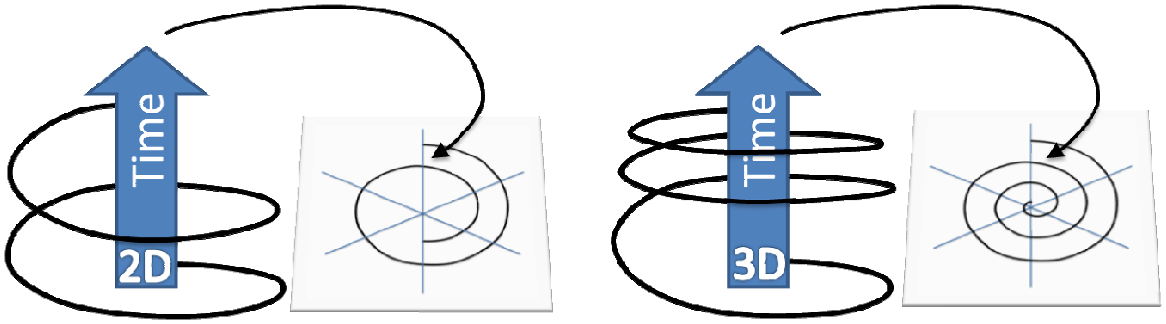


Fig. 1: Design spiral in two different approaches for storing design data

Fig. 1 illustrates how two different design approaches tend to behave in the design spiral relative to the time. In the early adaptation of the 3D design approach, the 2D approach tends to be faster to get the first round activities completed. However, the ship design is usually very iterative and the numbers of modifications increase when an optimum design is explored. Updating the design information in a centralized 3D model compared to multiple sets of 2D drawings make more sense. Also, the probability to have consistent data is much higher in a 3D model because the information is not duplicated. The benefits of a working product model concept are undeniable.

2. FEA as a Part of Structural Design Process

Not having the design information in a 3D format is one of the reasons why finite element analysis (FEA) is not typically introduced in the very early stages of design though it acts as a validation tool in later design stages. FEA is many times the first and also the only reason for the creation of a 3D model in the early structural design phases. This means that the finite element (FE) model is possibly the only place where the design information is in 3D. Usually, the geometry and property information of FE models are too idealized to be fully utilized by other disciplines; therefore, the FE model cannot be considered the optimum archive for design information.

FEA is a very time consuming task, normally carried out as few times as possible, and serves more as a validation tool of the design. Therefore the first FEA is carried out rather late in the design process, Fig. 2. However, if 3D design information was available, it could dramatically reduce the time in creating the FE model and makes it possible to carry out the analysis earlier in the design process.

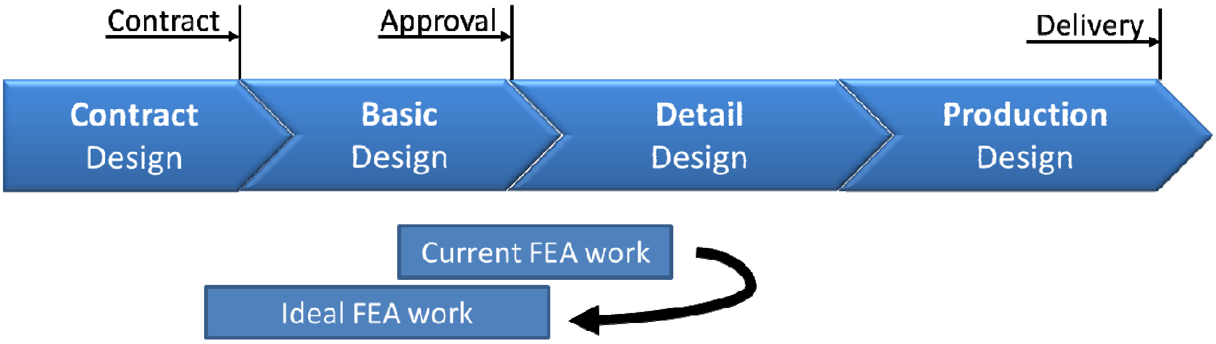


Fig. 2: FEA in the design process

During the design process different types of FEA analyses are carried out. The main purpose of FEA in the early design stages is to make sure the global responses of the ship are within tolerable limits and to ensure the general arrangement of ship structures are reasonable. As stated before, the FE models needed for the global response often require a long time to create with general purpose FEA systems. Additionally, local models may have to be created to support the design and validation of critical structural details as well as areas of high stress concentration.

In many ship types (e.g. tanker and bulkers), only the cargo area is subjected to FEA to satisfy the requirements of Classification authorities. The remaining structural members are validated through prescriptive rules issued by Classification authorities. Due to the above mentioned reasons, new and innovative designs are not pursued as they tend to be labor intensive (i.e., very expensive). This situation could be changed if the FEA was carried out in a much shorter time; therefore, making it a more attractive undertaking even though Classification authorities are not requiring it.

It would lead to better design if FEA could be introduced earlier in the design process. The process would be even more efficient if the FE model was extracted from the most recent design information for every analysis. Often, the FE model mesh for the new analysis is based on the previous FE model mesh and it is updated according to the new design information. With a 2D approach this would be the only reasonable way of working, but if the 3D design information is available, it opens a new opportunity. Fig. 3 illustrates the ideal process for different FEA. Whenever there is a need for a new analysis, the creation should start from the most recent design information than updating the previously defined FEA information. It might be feasible to update the FE model mesh manually in later design stages where changes for design are not that frequent, but this is hardly ever the case in the early design stages. If the extraction of the FE model mesh for the FEA is well supported, it reduces the time and possibility for human errors. It is always better if the design information is maintained in as few systems as possible and FEA should not increase the number of systems.

Fig. 3: The location of FEA activities in ship design process typically and ideally

3. Steps in the FEA Process in Detail

The overall FEA process can be decomposed into key activities and individually examined. These activities are introduced in Fig. 4 and discussed in detail in the following sections.

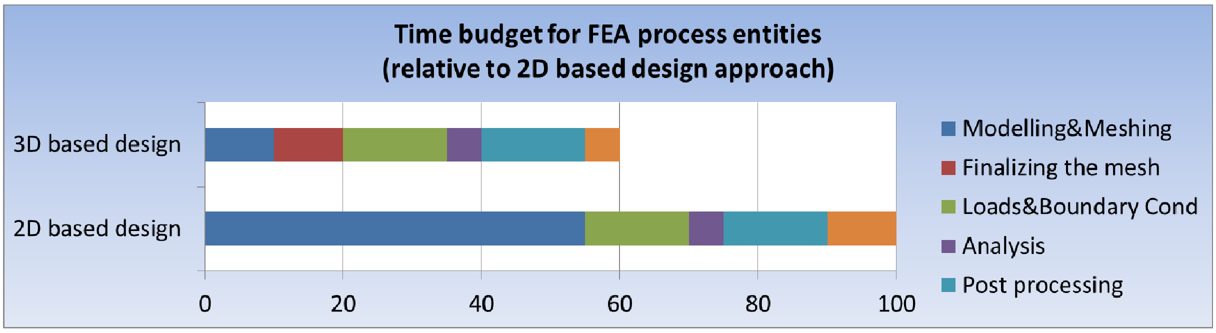


Fig. 4: FEA time budget

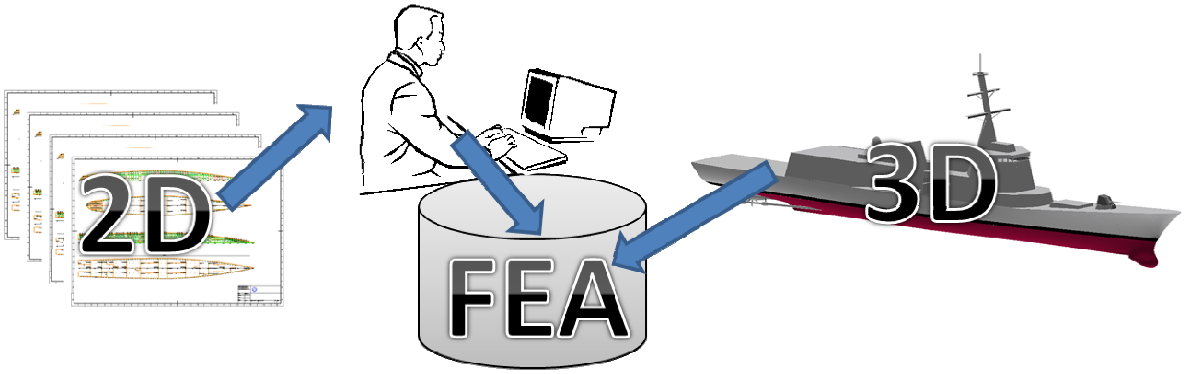


Fig. 5: The creation of FEA in different design approaches

3.1 Modelling and Meshing

The ultimate goal of the modelling and meshing process is to generate an adequate and valid calculation mesh (i.e., FE mesh) for submittal to an FEA solver. A significant part of this process is to define the appropriate structural properties of the design at hand (e.g., material properties, plate thickness, and cross-sectional quantities). The differences in the selected design approach (i.e., 2D vs. 3D) have a significant effect on the performance of the modelling and meshing work. These differences are introduced in the detail below, but simply stated, it means there is always a greater degree of human interaction when the FE model is created on the basis of 2D design information while a true 3D design approach has a less degree of human interaction, Fig. 5.

3.1.1 The 2D Design Approach

If the design information is stored in 2D drawings it is tedious to extract the information necessary to create the FE mesh. The work usually starts with the manual idealization process wherein structural details are left out, or their effect taken into account in the scantling property definition. This work is done “on paper”, typically by drawing grid lines on top of the structural drawings that serve as guidelines to the modelling process and the creation of a valid FE mesh. Therefore, the geometry in the FEA system is started as an idealized 3D model in order to reduce the time in creating the FE mesh and to get as few degrees of freedom for the calculation. When the design is changed (a common event during any design process), it has to be updated manually within the FEA system. Therefore the modelling is often the most time consuming task in the FEA, *Doig et al. (2009).*

3.1.2 The 3D Design Approach

If the design information is stored in a 3D format, the potential to save time in the process of creating an FE mesh is enormous. The amount of time that can be saved depends mainly on the capabilities of the 3D design system rather than the FEA system. There are bigger variations on the capability support to generate FE meshes among the different 3D design systems than in the geometry modelling tools of FEA systems.

The idealization process can be automated when the design information is in a 3D format. Different methods for idealizing structural details are introduced in *Doig et al. (2009)* and *Kurki (2010).* The main idea is to keep the 3D design information in an as-built state and not to introduce any idealization in the modelling of the geometry there. Otherwise, the design information is not adequate for the other disciplines if simplifications are carried out.

There is an extra step, finalizing the mesh, in the FEA process when the mesh is generated from the 3D model however. This activity requires the designer to clear the errors produced by the automatic mesh generation process (e.g., eliminating bad nodes, elements, and connections), which is required to successfully run an analysis. Although this task is present when an FE mesh is created straight in a FE pre-processor, it typically occurs throughout the creation of geometry and the FE mesh. The time required to perform this work depends on the system used in this process. Often, the mesh is only created once on the base of the design information, because it is easier to update the FE model in the FEA system than to manually correct the mesh incorrectness produced by the poor output from the 3D design system. For instance, there are many tools to create FE models based on a general CAD files, but the quality of the mesh is not as good as in well-integrated systems. Also, the transfer of scantling and material property information for the finite elements is insufficient.

3.2 Applying Loads and Boundary Conditions

The correct application of loads is a critical factor to sound structural design assessment. In some ways, the correct application of loads is most important. In ship design, there are several common load “patterns” that need to be considered: e.g. lightship distributions, tank loading, dead loads, hydrostatic loading, and in some ship types, hydrodynamic loading. The effort required to complete this activity within the FEA process can also be a large and tedious task. Similar to the modelling and meshing activities described in the previous sections, the process of applying loads can benefit from the data captured in the development of a 3D product model. The following sections describe what typical data is required to compose the complete loading scenario and how the 3D design approach can make this a more efficient process.

3.2.1 The 2D Design Approach

There are a number of data sources the designer must first locate, check its relevance, and extract for purposes of loading the FE model properly. Common input data include: weight reports, tank capacity plans, existing stability analysis reports (or run files), and perhaps even vendor data for significantly large weight items. Once the designer has this data, it is their task to model these loads appropriately using the available capability of the chosen FEA system. Most general purpose FEA systems have a base level of common loading patterns to accommodate the varying load experienced by ships.

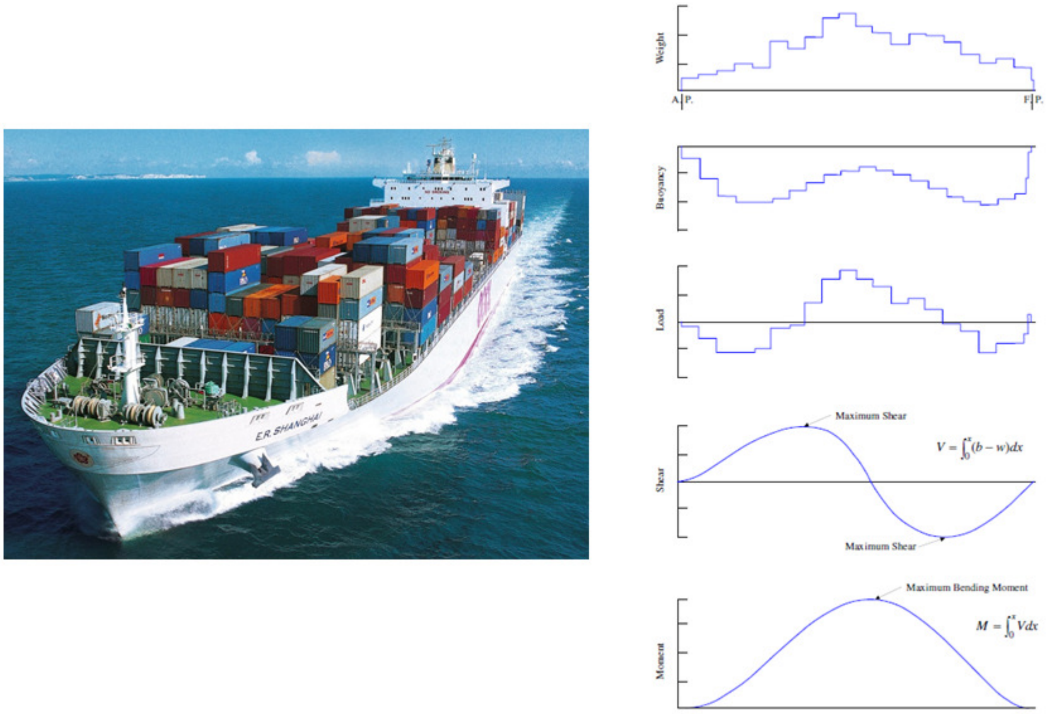


Fig. 6: Ship-specific loading

There are fewer ship-specific FEA systems that facilitate the modelling of these common loading patterns found in ship design, Fig. 6. For example, Fig. 7 shows an FE model that has tank loading defined as well as localized deck loading. Hydrostatic loads are another common ship load that ship-specific FEA systems can easily define, Fig. 8. The task of defining the loads must continue until all aspects of loading are accounted for and the loads are in equilibrium resulting in sound distributions such as those shown in Fig. 6.

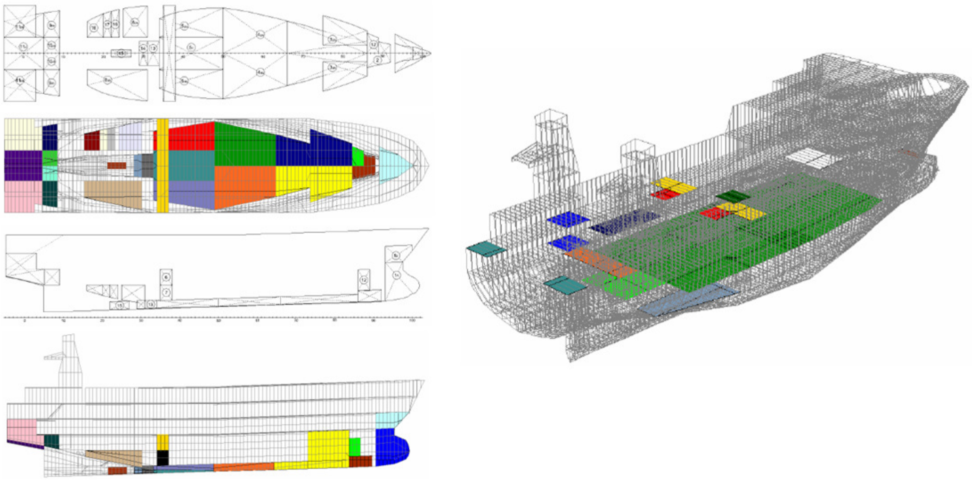


Fig. 7: Manual loading pattern creation

The final step for the designer prior to processing is to define the proper boundary conditions such that the possibility of rigid body motion is avoided while minimally supporting the structure so as not to influence the natural response of the ship. This is a well-known strategy to FEA analyst.

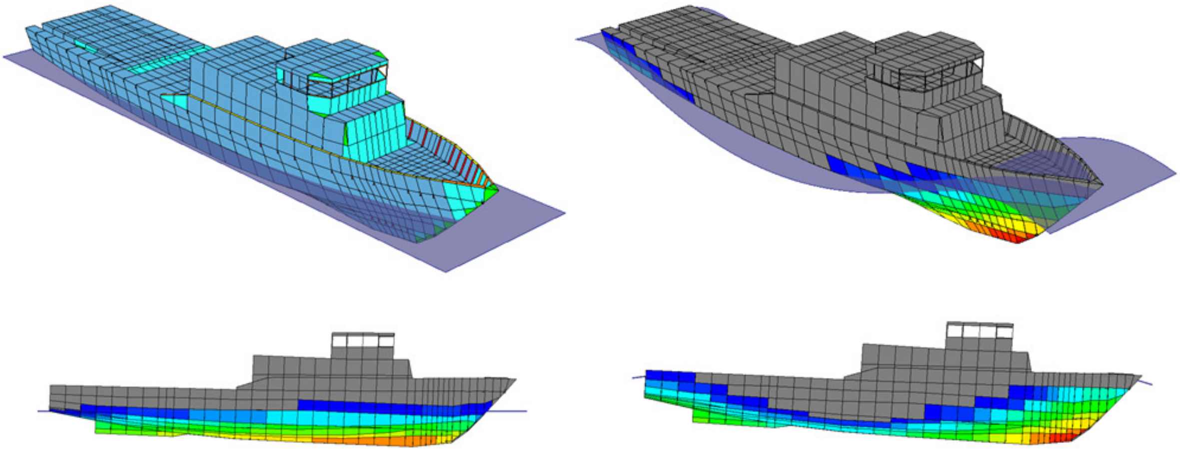


Fig. 8: Hydrostatic loading

3.2.2 The 3D Design Approach

Similar to what was described in the Modelling and Meshing section, information stored in the 3D model also has the potential to save significant time in the process of creating loading scenarios. The amount of time that can be saved depends on the extent of the loading definition in the 3D design system and the ability of the FEA system to consume the loading data. Fig. 9 illustrates the data flow for loading information where the 3D definition can automatically be consumed by the FEA system. As the loading definition changes with the changing design, the loading information can be re-exported for FEA consumption. Next, the task of performing load equilibrium checks and defining boundary conditions must still be accomplished by the designer as does the process of creating the appropriate boundary conditions. Any time savings that can be realized for these tasks are a function of the FEA system's capability.

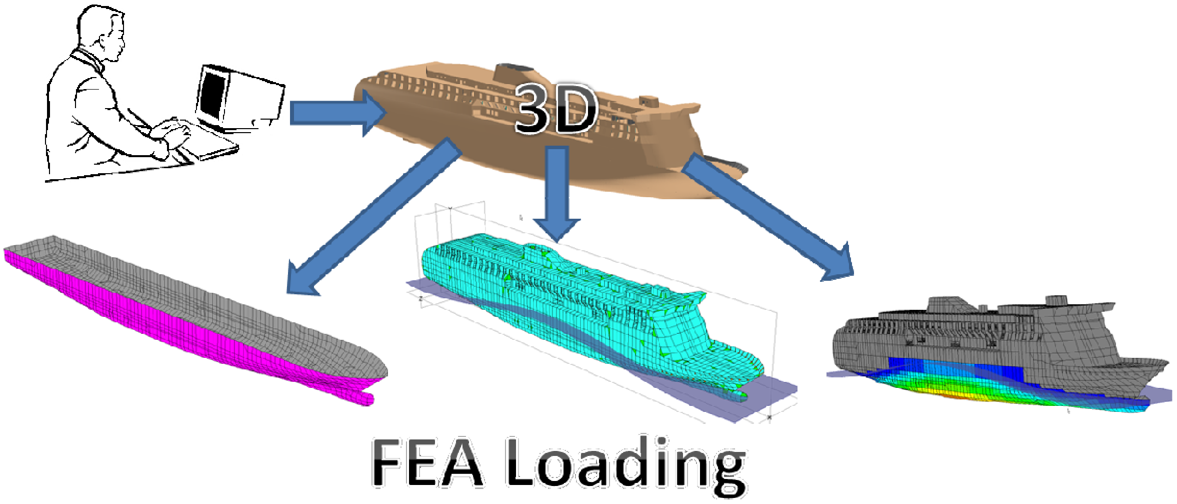


Fig. 9: 3D product model loads to FEA

3.3 Analysis and Post Processing

To perform structural assessment, it is first required to find the structural response of the design based on the defined loading scenarios. In this step, the FEA system (using finite element methodologies) must perform calculations to determine the ships deformations and stresses. FEA systems are designed to present to the designer these computed deformations and stress. This usually entails the recovery of results from the FE model. Fig. 10 shows some example stresses that would be expected to be recovered from a ship FE model. Further, stress results are graphically plotted, which allows the designer to effectively post-process a given structural response.

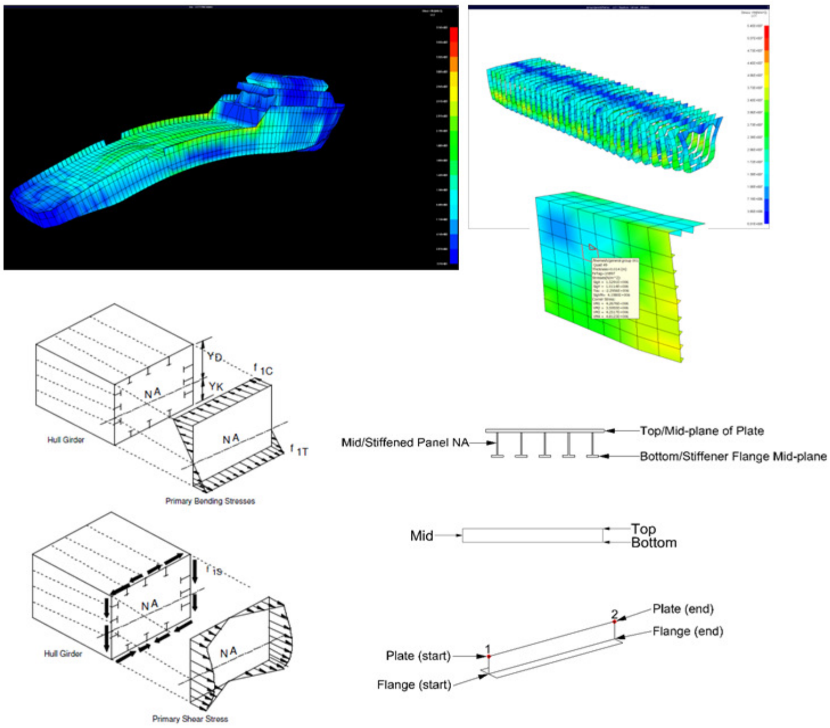


Fig. 10: Structural response analysis

3.4 Limit State Analysis

Structural design assessment does not end with deformation and stress assessment. Comprehensive structural assessment should include evaluating structural stability and load-carrying capacity. This includes the assessment of different types of structural failure: stiffened panel collapse failure modes, local member failure modes, and hull girder ultimate strength.

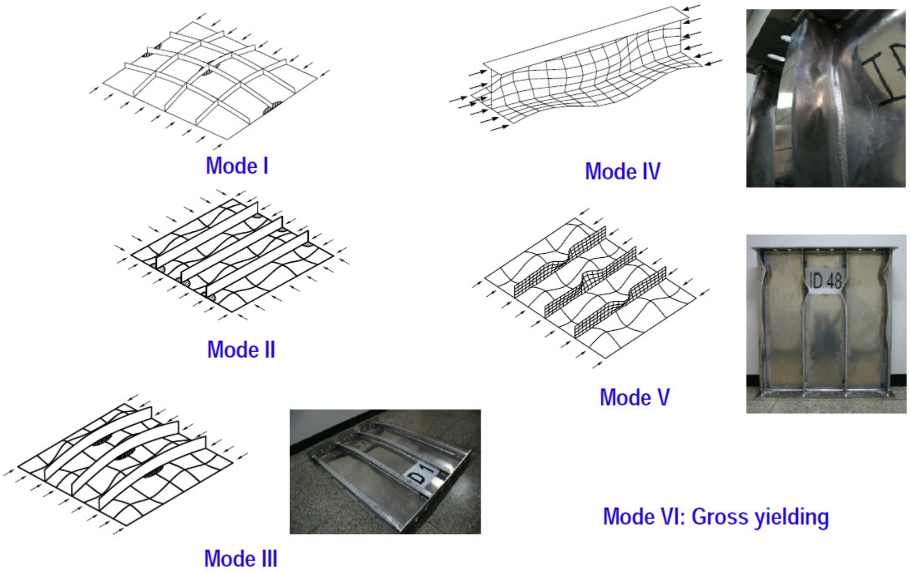


Fig. 11: Ultimate strength of stiffened panels, collapse modes

The following are specific examples of six failure modes, Fig. 11:

- Mode I: Overall collapse after overall buckling
- Mode II: Collapse of the plating between stiffeners without their failure
- Mode III: Beam-column type collapse of a stiffener with attached plating
- Mode IV: Local buckling of stiffener web
- Mode V: Flexural-torsional buckling of a stiffener
- Mode VI: Gross yielding

3.5 Providing Results to Product Model

After conducting the finite element analysis, limit state analysis, and post-processing the results, the designer can revise the scantlings in the 3D product model. These changes in the structural arrangement and scantling definition can then be rerun through the FEA process as described in the previous sections. This feedback loop, within the context of a 3D design approach is how the FEA process becomes more active in the earlier phases of the structural design. When the structural design is adequate and sufficiently optimized to meet the objectives of the owner, the next step is to produce a complete set of structural drawings (i.e., the scantling plans) suitable for submittal to a Classification authority. At this juncture in the design process, the updated 3D product model serves as the source for creating these 2D drawings. This leads to a remarkable savings in developing Class drawings.

4. FEA Process Supported by NAPA and MAESTRO Software

NAPA, Naval Architectural Package, is a design tool specializing mainly in the early design stages of ship design process. NAPA contains a wide range of design solutions with the topological 3D product model as the core. The structural design tools, NAPA Steel, has been developed solely for the initial and basic design phases offering functionalities for multiple disciplines of which FEA is of main interest in this paper. Similar to NAPA, MAESTRO is used during early stage ship structural design. MAESTRO is a design, analysis, and evaluation tool specifically tailored for floating structures and has been fielded as a commercial product for over 20 years and has a world-wide user base. MAESTRO's history is rooted in rationally-based structural design, which is defined as a design directly and entirely based on structural theory and computer-based methods of structural analysis (e.g., finite element analysis). MAESTRO core components are: rapid coarse-mesh finite element modeling, ship-based loading, finite element analysis, limit state analysis (e.g., at the hull girder level, stiffened panel level, and local member level), and design evaluation.

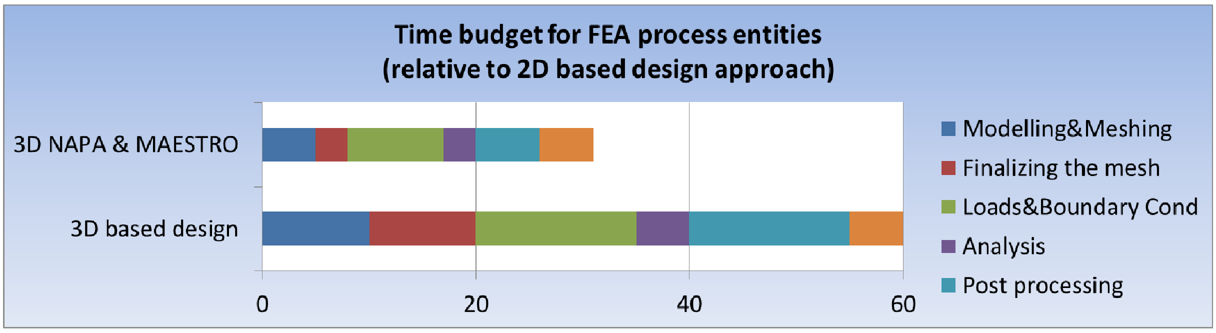


Fig. 12: The summation of time spent in different FEA process entities

Another fact that makes NAPA-MAESTRO combination interesting is the cooperation between the companies, Napa Ltd and DRS Defense Solutions LLC, who are developing these ship design specific tools. This cooperation has resulted in an interface between NAPA and MAESTRO that will significantly shorten the overall time in the FEA process compared to many other current market solutions. The efficiency gained through this interface is described in the detailed in the following chapter for each individual FEA process entities. Fig. 12 shows a summary of the spent time.

4.1 Modelling and Meshing

The creation of FE model in NAPA is based on a process where the start point is the real, as-built, representation of ship structures, Fig. 13. The first task is naturally to create the 3D model of ship structures. Usually, this work is done for other purposes therefore actual modelling work is minimal for the FEA. However, if the 3D model does not exist NAPA Steel could be used to create the model from scratch solely for the FEA purposes. The modelling tools have been proven to be very efficient in NAPA and the model can be created in a matter of days to accurate enough for the global FEA.

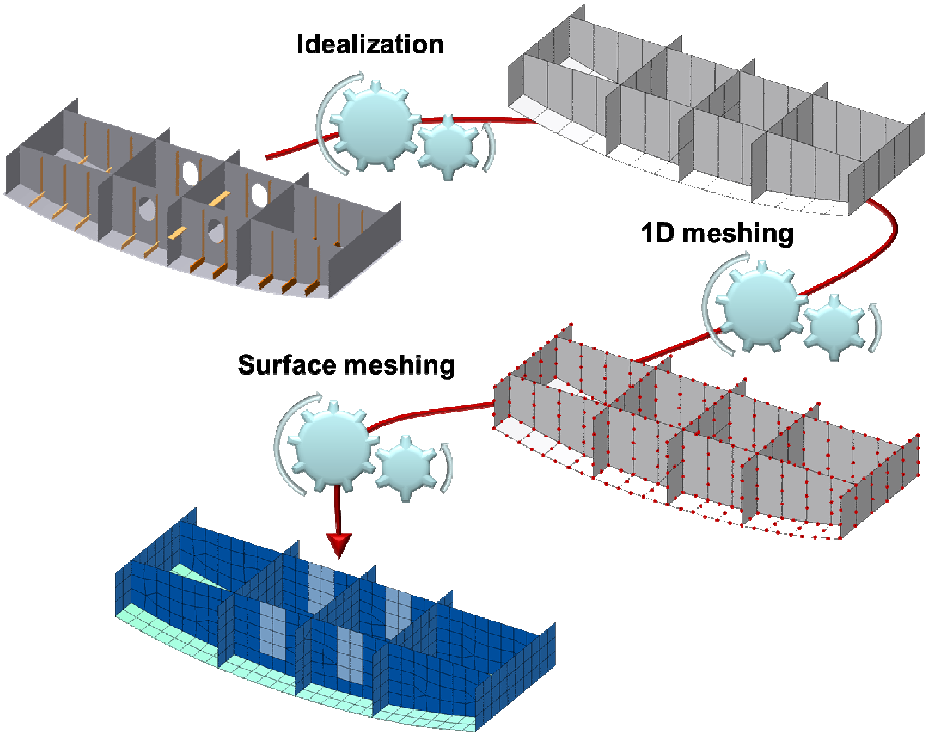


Fig. 13: FE model creation process and its separable entities in NAPA

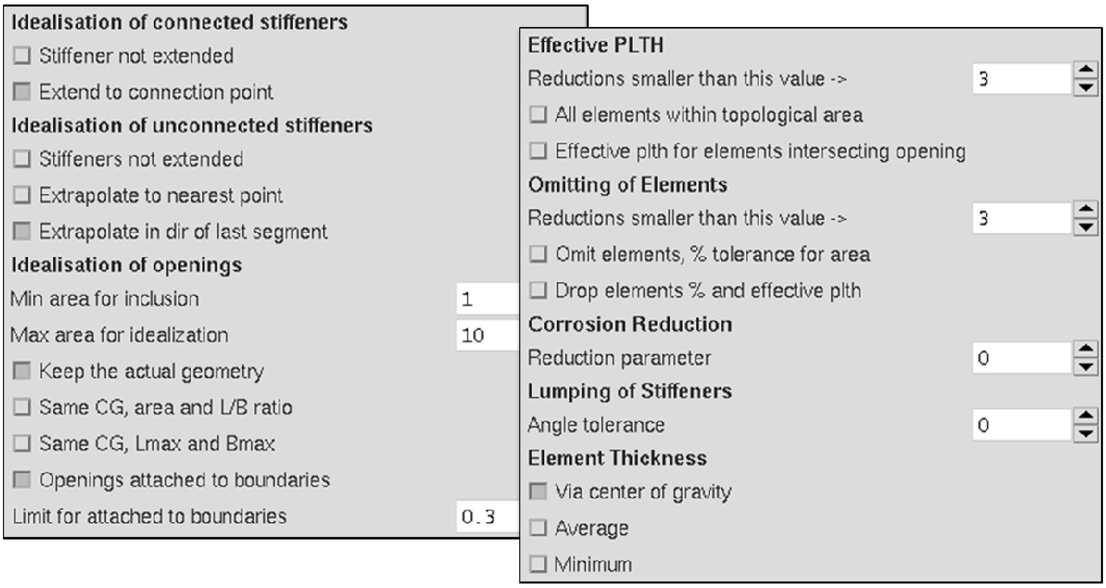
4.1.1 Idealization

The idealization process is done on the base predefined set of parameters in NAPA. The user is able to modify the values and store them as individual sets of rules. By applying different values to the rules various kinds of FE models can be generated from the same 3D structural model. The user is simply applying rules to get different detail level of FE model and not conducting any modelling work.

The parameters and rules define two main components; which structural details are considered in the FE model and how they are considered. An example of the rules is illustrated in Fig. 14. Different idealization methods are well introduced in *Doig et al. (2009)* and the idealization capabilities of NAPA in *Kurki (2010).*

One of the main advantages of having the mesh created inside the same tool as the 3D design information is to be able to have full control on the topology of the geometry. This will make the idealization and mesh generation more robust and offers better possibility make simplifications correctly compared to finding the connections between geometry on the base of a general purpose CAD output.

It is very efficient to create different kind of FE models when the generation is done by applying a set of rules for a product model. There is no need to create new, more detailed geometry appropriate for the target analysis though different representation of structures can be extracted on the fly by following the user defined rules in NAPA system, Fig. 15.



Idealisation of connected stiffeners
☐ Stiffener not extended
☑ Extend to connection point
Idealisation of unconnected stiffeners
☐ Stiffeners not extended
☐ Extrapolate to nearest point
☑ Extrapolate in dir of last segment
Idealisation of openings
Min area for inclusion: 1
Max area for idealization: 10
☑ Keep the actual geometry
☐ Same CG, area and L/B ratio
☐ Same CG, Lmax and Bmax
☑ Openings attached to boundaries
Limit for attached to boundaries: 0.3

Effective PLTH
Reductions smaller than this value ->: 3
☐ All elements within topological area
☐ Effective plth for elements intersecting opening
Omitting of Elements
Reductions smaller than this value ->: 3
☐ Omit elements, % tolerance for area
☐ Drop elements % and effective plth
Corrosion Reduction
Reduction parameter: 0
Lumping of Stiffeners
Angle tolerance: 0
Element Thickness
☑ Via center of gravity
☐ Average
☐ Minimum

Fig. 14: Parameters and rules controlling idealization and deriving the properties

For instance, the stiffeners can be described the following ways depending on the target analysis:

1. Taken into account in the properties as lumped stiffeners where the influence of neglected stiffeners are merge to beam elements on the element boundaries
2. As beam elements, line segments with cross sectional properties
3. Web as surface elements and flange as beam elements
4. Web and flange as surface elements

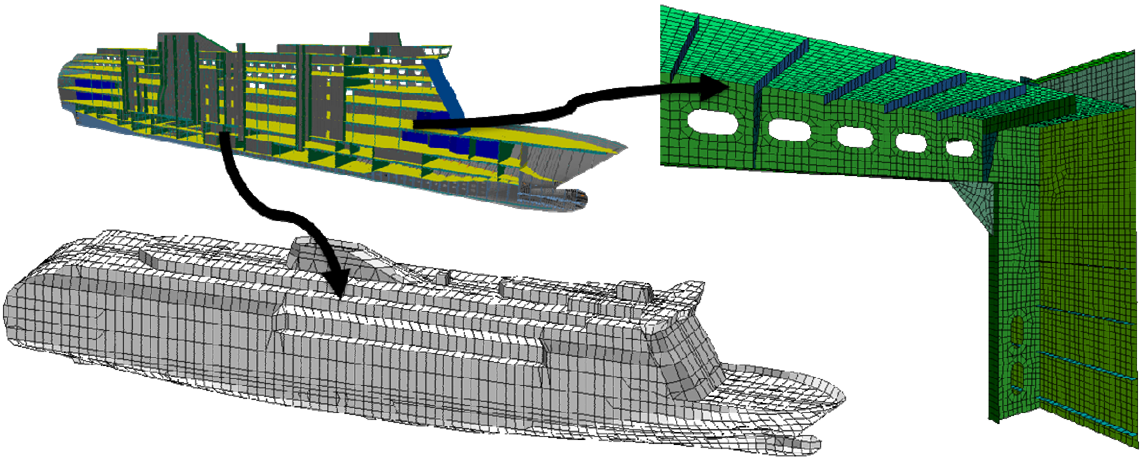


Fig. 15: Different kind of models created with separate parameter set

In a local analysis one typical approach in the mesh generation is to define the area of interest with small elements and the surroundings with more coarse mesh to reduce the size of the model and to get better representation of the global behavior in the analysis. To reduce the time in creating such models NAPA has capabilities to define a different set of rules to limited area where the idealization and the mesh size differ from the surroundings. This will enable to create refined areas to any selected place in the 3d model without additional modeling worked to be carried out. The examples of refined model are illustrated in the Fig. 16.

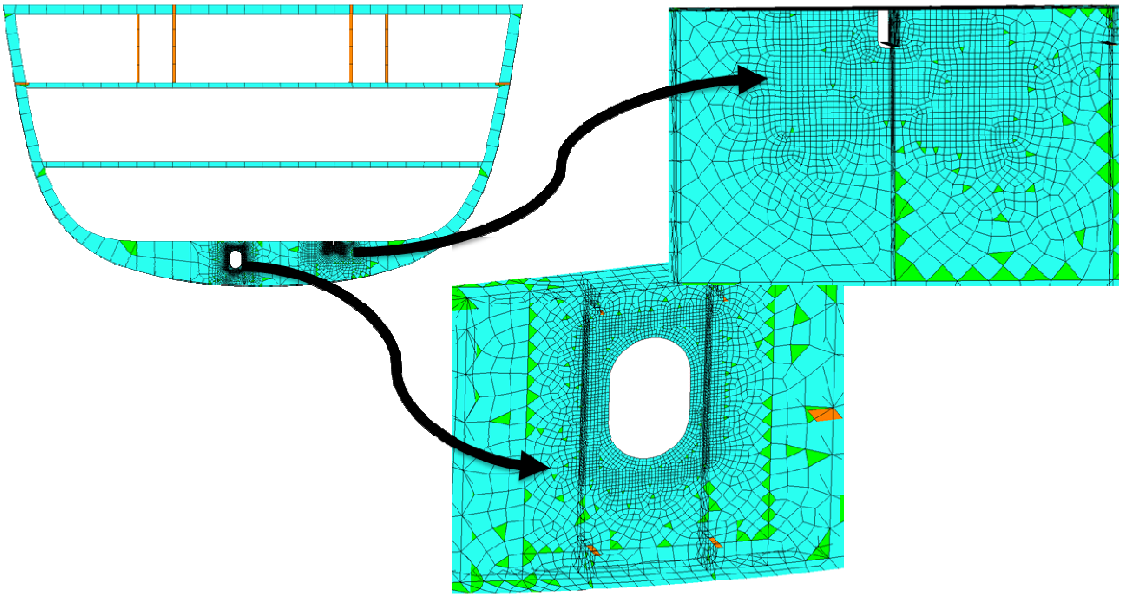


Fig. 16: Local refinements

4.1.2 Properties for the FE Model

If not as tedious job as creating geometry of the mesh at least equally important task is to define the properties for the FE mesh. In case of a global FEA, the idealization plays an important role i.e. a lot of structures are neglected as such, but their effects are taken into account. The options and parameters for deriving properties are illustrated in Fig. 14. Naturally, the actual properties of the 3D design information can be automatically inherited to the FE model reducing the time for creation of the properties significantly.

The connection to the structural design information is very important in order to have the latest property information available. The compartment information in the ship is also very important to have for reducing the compartment information can be utilized especially in the loading, which is described in more detail on the following chapter. The information on compartments is also important when applying the properties for the mesh. Typically, the design information is presented as gross scantlings whereas the FEA is carried out often with net scantlings i.e. gross scantlings deducted by the corrosion addition. The calculation of corrosion addition is heavily based on the information on compartments and especially on their contents. NAPA model has the information on compartments and their contents making it possible to derive the net scantling information for the mesh automatically.

4.2 Finalizing the Mesh

In the early design stages the 3D design information accuracy and the correctness of the geometry is not always sufficient to generate flawless meshes automatically. The more the mesh generation is based on a topological 3D model these imperfections can be corrected in the idealization process. In case of incorrect geometrical information it is better to try to correct the errors in the model as early stage as possible in the FE model creation process, Fig. 13. Here are the different options for finalizing the mesh in a recommendable order:

1. *Modify the 3D design information*. If the geometry is wrong producing bad quality mesh it should be corrected into the 3D NAPA model. Then it is also available for other design disciplines. FEA is recognized as a good tool for validating the design information.
2. *Define additional information to produce better quality meshes*. The user can define additional helping lines to NAPA structural model to guide the automatic meshing to produce better or more desired results. For instance, new traces can be modelled to topology to be

used in the in the element generation only. It is good to introduce these in the topology level as they can be utilized again when the 3D design information is changed and new FE models are needed.

3. *Correct the resulting mesh manually*. The error in the mesh can be corrected manually in MAESTRO.

4.3 Loads and Boundary conditions

MAESTRO's loading capability addresses both general loading patterns as well as ship-specific loading patterns. The following are some specific loading capability found within MAESTRO's existing system:

1. *Tank Loading*. Using the existing FE mesh definition, elements are collected to form the tank boundary. With the tank boundary defined, the designer can specify the tank contents and the amount of content found within the tank. The tank loading can be different for different loading scenarios.
2. *Hydrostatic Loading*. The hull definition is deemed wetted in MAESTRO terminology (see figure) and has the ability to be automatically loaded with hydrostatic loading. The definition of wetted elements, within the FE model, greatly facilitates the application of different still-water and wave conditions experienced by ships. This automatic application of hull pressure also plays an important role in properly finding force equilibrium for a given loading scenario.
3. *Longitudinal Distribution*. Achieving the correct lightship distribution can be accomplished by defining a known weight density or weight at defined longitudinal locations. Further, this definition allows the designer to define the transverse and vertical center of gravity for the total weight distribution to achieve proper nodal distribution.

Napa and DRS AMTC have collaborated to extract the pertinent loading information from the 3D product model and translate it to the corresponding MAESTRO loading capability described above. Currently, the loading data includes: longitudinal weight distributions, longitudinal bending moment distributions, hull definition for hydrostatic loading, Fig. 17, tank boundary definitions, Fig. 18, tank content and fill definitions, and hydrostatic equilibrium definition (i.e., trim and heel).

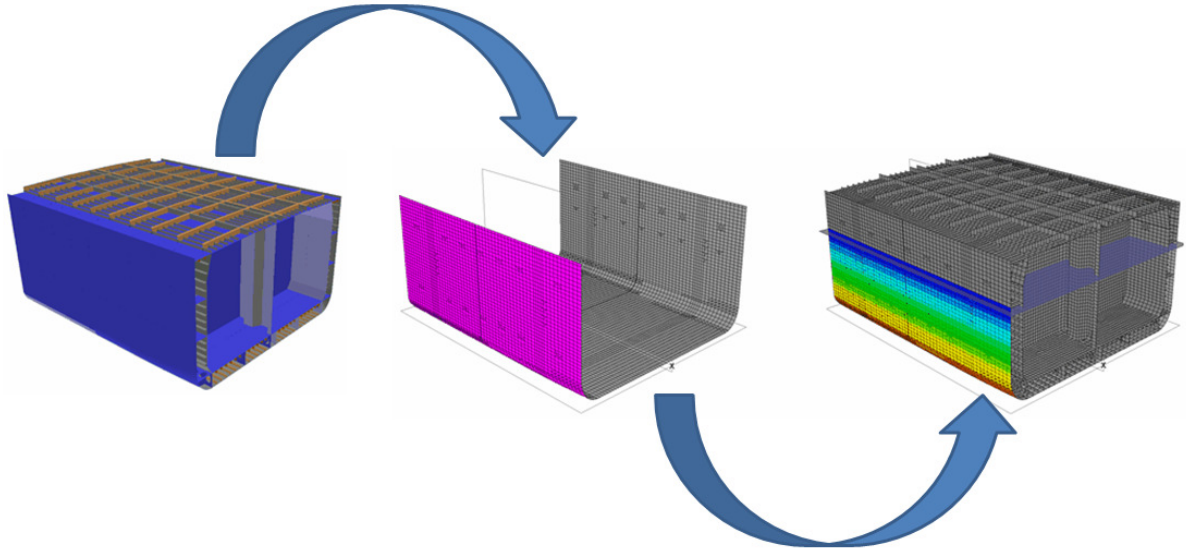


Fig. 17: NAPA hydrostatic loading data to MAESTRO

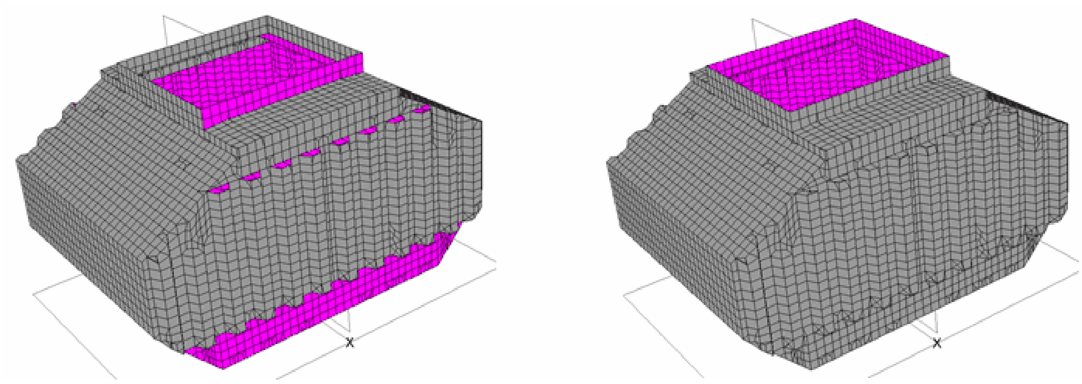


Fig. 18: Tank boundary definition and creating consistent normal definition

4.4 Response Analysis, Limit State Analysis, and Post Processing

MAESTRO has the ability to perform comprehensive structural assessment for floating structures. This includes performing response analysis (i.e., deformation and stress analysis) and limit state analysis. The limit state analysis includes hull girder collapse analysis, stiffened panel buckling analysis, and local member buckling analysis.

The first step to structural assessment is conducting response analysis. This encompasses the computation of deformations and stresses. MAESTRO's response analysis has been verified against theoretical and other industry standard FEA software results. MAESTRO's FEA solver uses the Intel Pardiso Sparse solver, which is a high-performance, robust, memory efficient, and easy to use solver for solving large sparse symmetric and non-symmetric linear systems of equations on shared memory multiprocessors. Deformation and stress can be recovered from individual elements as well as stiffened panels, Fig. 10.

The next step in structural design assessment, limit state analysis, has been a core component to MAESTRO from its inception. MAESTRO has a comprehensive structural assessment capability and includes the evaluation of structural stability and load-carrying capacity. The formulation of MAESTRO's limit state analysis is covered in *Hughes and Paik (2010)* and *Paik and Thayamballi (2003)*. These textbooks constitute the theoretical manual for MAESTRO's limit state analysis. MAESTRO's limit state analysis capability computes a number of different stiffened panel collapse failure modes, local member failure modes, and hull girder ultimate strength, including the six modes of failure previously described and illustrate in Fig. 11. MAESTRO's limit state analysis is done automatically and comprehensively for the entire FE model and for all loading conditions. To properly perform this strength assessment, the true stiffened panel must be found and assessed in the FEM. This is done by automatically searching the entire model and collecting multiple finite elements (plates or beams) so the true boundary conditions and true spans are represented, Fig. 19.

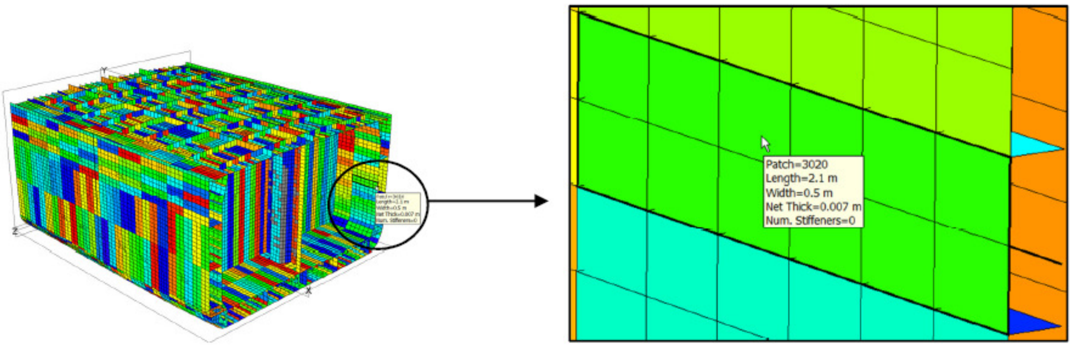


Fig. 19: Limit state analysis evaluation panels

4.5 NAPA-MAESTRO Interface Summary

Using a 2D approach is certainly one way to build a 3D FE model; however, this interface provides a more efficient method by leveraging a 3D approach. Although tools like FEMAP, PATRAN, etc., all offer the capability to build a 3D FE mesh using a 3D surface model, what makes the NAPA 3D model unique is its 3D surface model's tight coupling to the 3D product model, its capability of structural idealization, its ability to generate different FE mesh models from the same 3D product model to support different analyses, and the linking of the NAPA hydrostatic model.

Combining this technology with a tool like MAESTRO has a great potential to improve the efficiency of the structural design process and brings FEA more so to the early stage ship structural design, analysis, and evaluation process. It does so by allowing the designer to leverage one 3D model from start to finish within the scope of structural design and direct analysis activities. This will eliminate the very common practice of recreating 3D structural models to serve different activities (e.g., one 3D model for Classification drawings and one 3D model for structural analysis). Further, by interfacing these two products, the designer does not have to recreate key loading scenarios in different products

At the core of the interface is the MAESTRO Neutral File, which contains the NAPA generated data that is pertinent for creating and analyzing the MAESTRO finite element model. Currently, Napa and DRS AMTC have successfully translated all of the finite element mesh and scantling information (e.g., unit system, FE nodes, material properties, and finite elements). Added to this, Napa and DRS AMTC have also been able to translate the pertinent loading information, which makes this interface unique. The loading data will include: longitudinal weight distributions, longitudinal bending moment distributions, hull definition for hydrostatic loading (i.e., the wetted elements in MAESTRO terminology), tank boundary definitions, tank content and fill definitions, and hydrostatic equilibrium definition (i.e., trim and heel).

4.6 Future Development Topics in NAPA-MAESTRO Interface

The current version of the interface supports well the FE model creation process in one direction i.e. NAPA pushes a lot of information to MAESTRO where the actual response of the structure is evaluated. Currently, NAPA also pushes pertinent loading information for consumption by MAESTRO. In order to support the design process better the information generated in the FEA should be fed back to the design information. This should now be done manually.

All the tasks that are carried out in the FEA consisting manual work is under investigation. For instance, it helps the handling of a large FE model if the elements are grouped. Certain groups are already now created, but new groups and other similar supporting information is under considerations to be included in the interface.

5. Conclusions

There are many advantages maintaining design information in a 3D model throughout the whole design process. This is even more emphasized when FEA is closely present in the design activities. Many times the FEA is carried out only in the mandatory cases because it is considered as tedious job. This will lead to designs reliant on the previous knowledge. With combination of NAPA and MAESTRO the FEA can be carried out in much shorter time enabling it to be used in the earlier design stages giving confidence that new innovative designs are functional.

Acknowledgement

This paper is cleared for open publication: Ref. DRS Defense Solutions Case # DRSAMTC-11002.

References

DOIG, R.; BOHM, M.; STAMMER, J.; HERNANDEZ, P.; GRIESCH, S.; KOHN, D.; BRÅNHULT, J.; BITTERLING, B. (2009), *Integrating Structural Design and Assessment,* 8st Int. Conf. Computer and IT Appl. Maritime Ind. (COMPIT), Budapest, pp.374-389

HUGHES, O.F.; PAIK, J.K. (2010), *Ship Structural Analysis and Design,* SNAME

KURKI, T. (2010), *Utilization of Integrated Design and Mesh Generationin Ship Design Process,* 9th Int. Conf. Computer and IT Appl. Maritime Ind. (COMPIT), Gubbio, pp.311-318

PAIK, J.K.; THAYAMBALLI, A.K. (2003), *Ultimate Limit State Design of Steel-Plated Structures*, Wiley