Rakenteiden Mekaniikka (Journal of Structural Mechanics)
Vol. 43, No 3, 2010, pp. 129-139

Utilization of integrated design and mesh generation in ship design process

Tommi Kurki

Summary. Advantages within creating FE models from the design model have been confirmed earlier. Application of an integrated 3D product model, as the source of information, helps the FE analysis to follow the pace of the design evolvement. While the general geometry modelling and FE analysis proceed in parallel, the overall design process gains speed. Development over the last few years has progressed towards the ideal where structural engineering more and more utilizes directly the information stored within the design models. This paper presents a process where integration is achieved, and reviews the results that are available for FE analysis. Meshes for global strength, local stress, and fatigue analysis are presented, and the prerequisites for creating these are discussed. Some instances of modifying the design and their implications for updating the FE analysis are shown.

Key words: finite element mesh, local refinement, fatigue, ship design process, integrated design, mesh generation, FEM, FEA

Introduction

It is a time-consuming task to create finite element mesh for ship structures. During the past decades, several attempts and solutions have been made for integrating ship design and mesh generation, *Doig et al. (2009), Dumez, et al. (2008), von Selle et al. (2009).* At the point of concept design (or early design), it is profitable to have a global finite element model as early as possible. If structural concepts can be evaluated effectively with FEA, this yields an optimum where the results of the finite element study can be utilized during the design process, rather than only in the verification of the structural integrity of a ready design.

Currently, the common trend for shipyards and design offices is to develop the working methods away from the separate drafting and 3D design towards unified design process, where 3D product model is created in one system, and that information is used for several purposes, i.e. for analysis, classification drawings and production material, Jang et al. (2008).

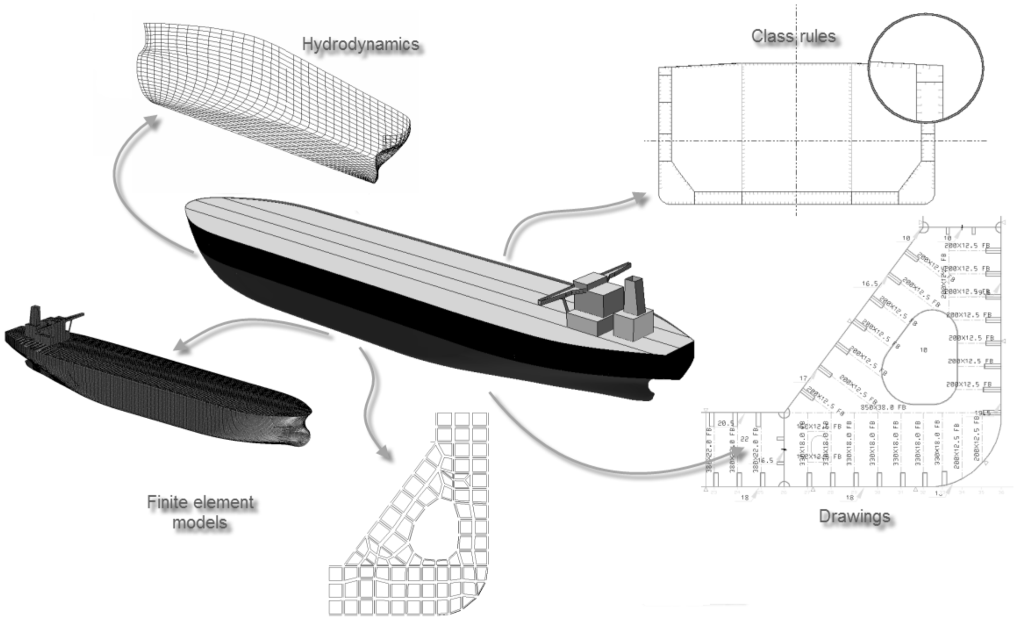

Fig. 1: 3D product model, primary source of information for design and analysis (by Napa Ltd).

In order to effectively manage design changes during the ship design process, it is advantageous to use a system, which allows intelligent modelling of ship structures. Thus, the design changes can be made with the minimum effort and up-to-date mesh is available for FEA at any time during the design process.

This paper shows the methods for parametric modelling and automatic mesh generation based on parametric 3D product model. Fig. 1 illustrates some fields which can utilize the information in a 3D product model. Herein the focus is on the mesh generation based on 3D product model. The following section describes the challenges in creating discrete geometry (mesh) based on as-built ship geometry. Also, it is shown what kind of information from the 3D model can be included in FE models automatically. Chapters 3 and 4 present the variety of finite element models needed.

Idealization of structures

The as-built ship geometry sets a challenge when the mesh is generated from that. The ship is a massive construction with an enormous number of structural details. The FEM analyses should take care of, not only the global behaviour of the whole ship hull, but also the smallest details at the size of fatigue cracks. Not even today do the computer resources allow modelling the whole ship construction at this fine mesh level. The structures must be idealized in a proper way in the FE models.

Not all the structural details are necessary from the global finite element point of view. The big challenge in automatic meshing is not to include realistic details into the model but to remove them by idealization from the model and still get realistic results. In several cases, it is also better to have slightly distorted geometry in the FE model in order to achieve better shaped elements for the analysis, and also to limit the number of created elements.

The following example in Fig. 2 illustrates how the as-built geometry of double bottom and hopper could be idealized automatically for finite element purposes in a tanker model. Idealizations in Fig. 2 are:

1. Connected stiffeners: they are extended to the connection point. In this case, the vertical stiffener is extended to the level of the longitudinal stiffeners in the tank top and the bottom.
2. Openings: in this case, the floor openings are taken into account via effective plate thickness. Alternatively, they can be omitted from the model or idealized as square.
3. Unconnected stiffener: end is moved with a tolerance to adapt to the mesh better. The stiffener in FEM is slightly angled but the meshed structure includes fewer elements with better shape quality than without idealization.
4. Hopper opening: exact curved geometry is discretized.
5. Corner notches: omitted from FEM.
6. Cut-outs and end-cuts: not included in the global level finite element model.
7. Plate thickness: structures are modelled at moulded surfaces with shell elements.

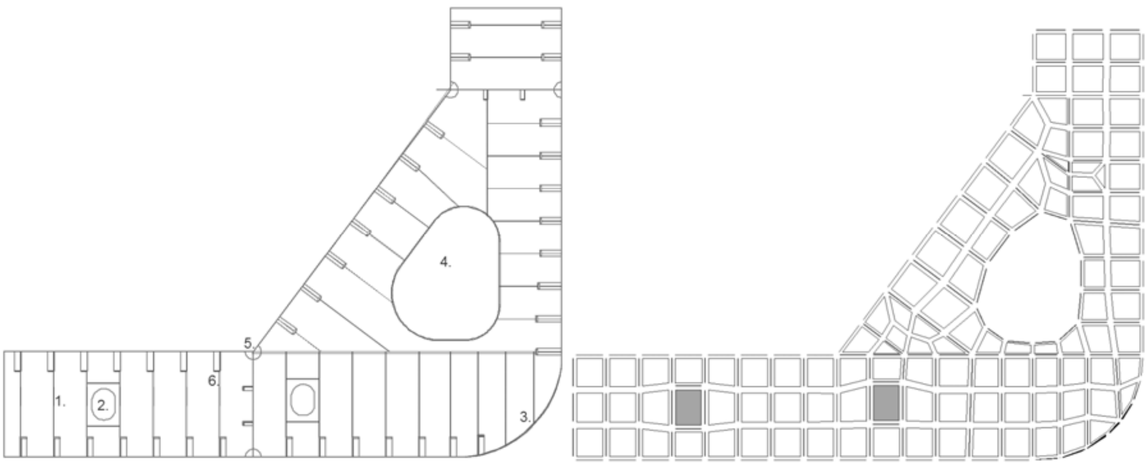

Fig. 2: Left: as-built structures. Right: shrink plot of elements. Lines represent beams. Grey fill colour indicates the elements with effective plate thickness (by Napa Ltd).

The right side in Fig. 2 illustrates the finite element mesh of the double bottom, hopper and side, generated automatically from the as-built 3D model. The target mesh size is approximately 0.85 m, which is the longitudinal stiffener spacing in the example.

Intelligent modelling enables effective handling of design changes

Ship structures change during the design process. As more information is available, the ship geometry varies and also the level of details increases. In practice, a ship designer will find an optimum design in an iterative process. To gain the maximum benefit, the numerical optimising algorithms are used also for ship design, *Kuutti et al. (2005).* In general, the usage of a numerical optimising algorithm with 3D product models needs topological and parameterized product models.

It is massively time-consuming to try to synchronize the finite element model with the classification drawings (or production material). Therefore, if the information is retrieved from one database, it is more likely to succeed in up-to-date geometry and the properties of a finite element model and the drawings, for example. The requirements from a large number of design iterations lead to fully automatic meshing algorithms.

Based on the initial scantling analysis, the strength assessment or perhaps a decision to change the general arrangement of a ship, the geometry and the properties evolve constantly and eventually several analyses are needed. Fig. 3 shows an example and the principles how the structures and compartments could be modelled in a way that the changes are controlled with minimum effort.

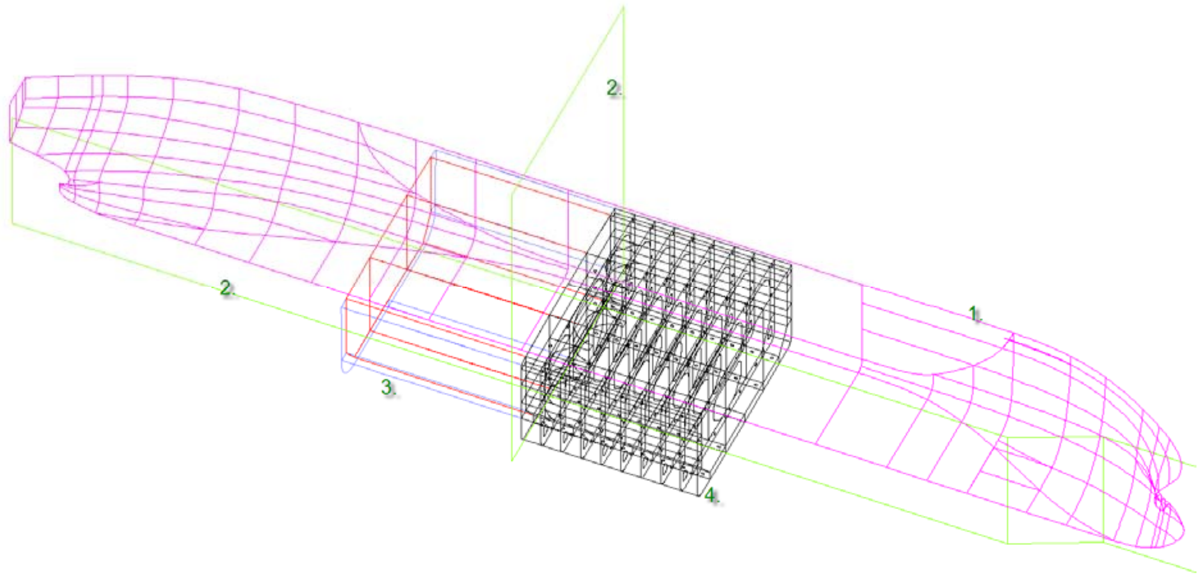

Fig. 3: Hull surface, two reference surfaces, five tanks, and steel structures within one tank are illustrated in a 3D product model (by Napa Ltd). (1) Hull surface; (2) Reference surfaces; (3) Compartment arrangement; (4) Steel structures

The compartment arrangement and the steel structures refer to the same reference surfaces and the hull, yielding unified changes during the entire design process. Also, the model is created with the references to the neighbouring structures and surfaces, see Fig. 4. These principles make the ship geometry fully topological. Topological geometry can be used in the optimization of structures, for instance.

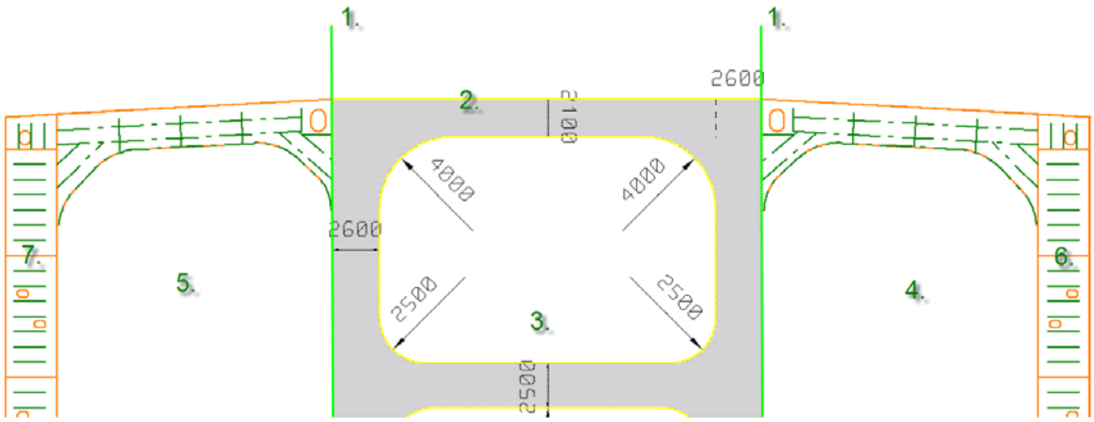

Fig. 4: Intelligent modeling using parametric web frame structures and topological compartment model (by Napa Ltd). (1) Reference surfaces for longitudinal bulkheads; (2) Parametric web frame structures; (3) Center Cargo Oil Tank No.4; (4) P-Wing Cargo Oil Tank No.4; (5) S-Wing Cargo Oil Tank No.4; (6) Water Ballast Tank No.4 (P-side); (7) Water Ballast Tank No.4 (S-side)

Another aspect from the finite element point of view is corrosion reduction: a compartment model can be used as a basis for the corrosion reduction. The corrosion reduction, required in CSR (Common Structural Rules), should be applied automatically to surface elements (plate and shell) and line elements (rod and beam) rather than via manual way which requires a lot of man-hours.

Design changes, case study

The purpose of the case study is to show the power in intelligent modelling and the product model based mesh generation. The examples show how big general arrangement level changes can be controlled with topology in product model and structural details with parametric modelling concept. These examples are prepared with NAPA software.

In the example, a reference surface is moved, first from #LONG14 to #LONG13, and further to #LONG12. As seen in the figure below, the model lives with the changed location of the reference surface. Therefore, the following structures move along the reference surface: the longitudinal bulkhead, the girder, the web frame, and the transversal bulkhead.

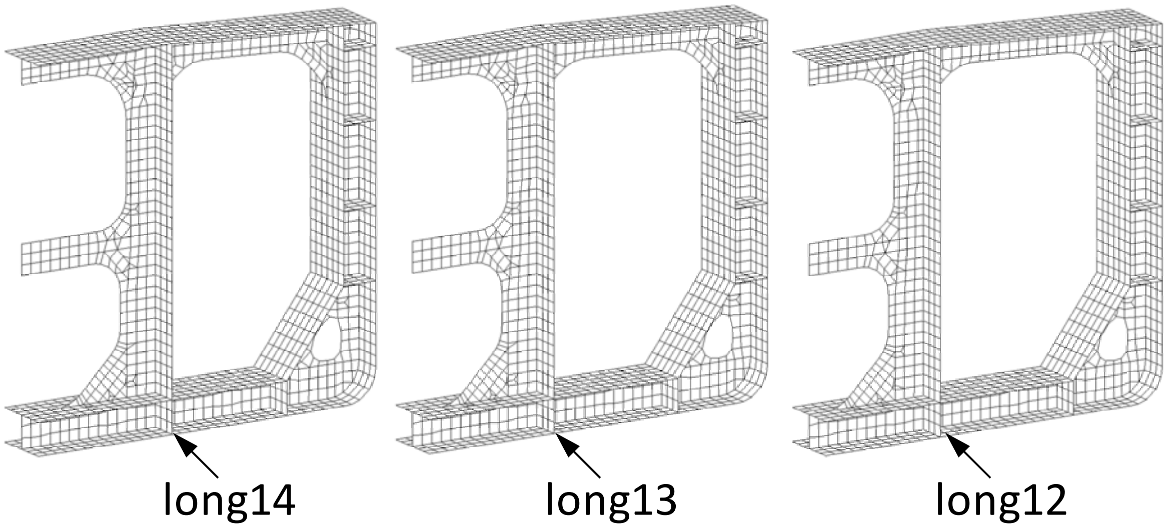

Fig. 5: Left: longitudinal bulkhead on long14. Centre: longitudinal bulkhead on long13, and right: longitudinal bulkhead on long12. Typical web frame, horizontal stringers etc., only port side shown in full-breadth model (by Napa Ltd).

Figs. 6 to 8 show the possibilities to change hopper opening shape and location and how the automatic mesh generation treats different opening geometry and structures. The shown meshes have been coarse and uniform which is suitable for global strength studies. The following chapter shows the possibilities to generate non-uniform meshes and include more detailed geometry for the analysis.

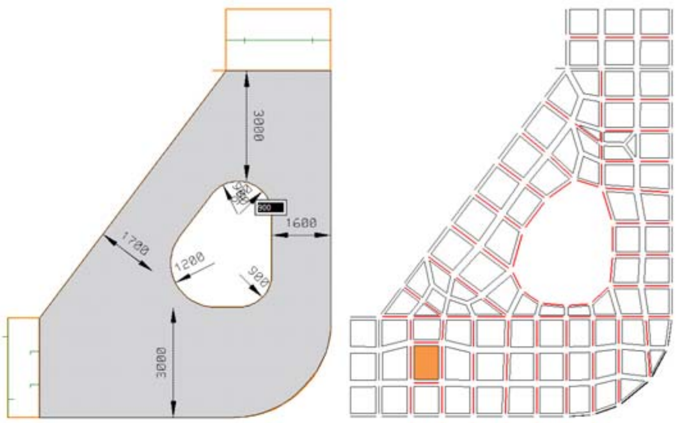

Fig. 6: Initial design of hopper and generated mesh (by Napa Ltd).

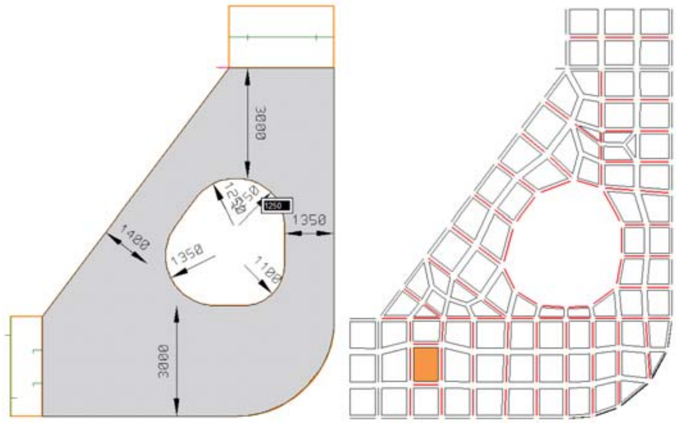

Fig. 7: Modified hopper and updated mesh (by Napa Ltd).

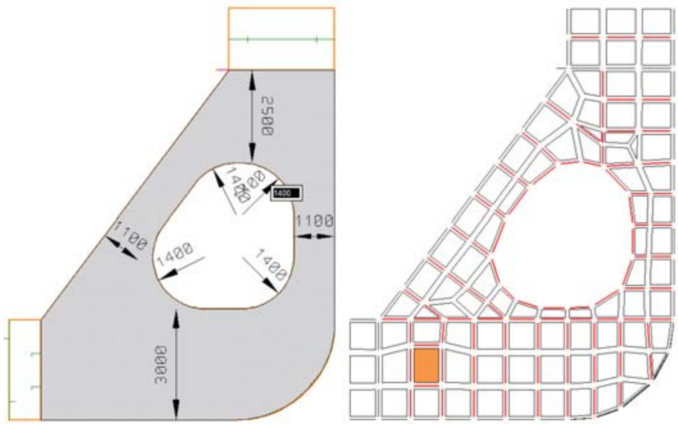

Fig. 8: Further modified hopper and updated mesh (by Napa Ltd).

Structural details and fine mesh areas

As the design evolves, more detailed information on structural details is available for more detailed analysis. Detailed shapes e.g. end-cuts, cut-outs, corner notches, and bracket toes are typical structures that need to be investigated for fatigue analysis. Often the applied mesh size is smaller than 50 mm, or even thickness by thickness mesh is applied to achieve the most accurate estimate for stress results, Common structural rules for oil tankers (2006).

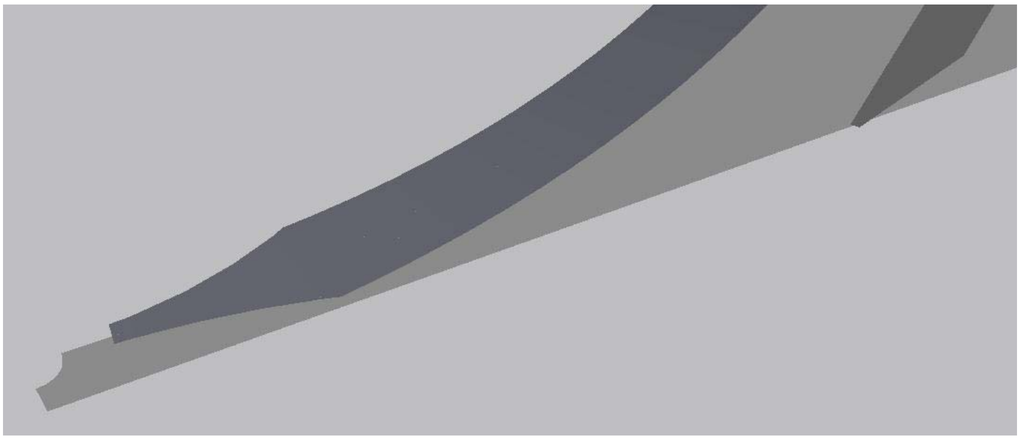

Fig. 9: Bracket toe and faceplate end-cut modelled in NAPA Steel.

The generation of fine mesh areas in NAPA is rather straightforward. In general, there are numerous meshing parameters to choose from and the user has full control of the mesh outcome. Fine mesh areas can be defined and used in several different ways and the system sets no limitations on the creation of fine mesh areas. In practice, the NAPA user has to define only the target mesh size for the refinement and the mesh is generated automatically. See Fig. 10 for JTP requirements for fine mesh areas.

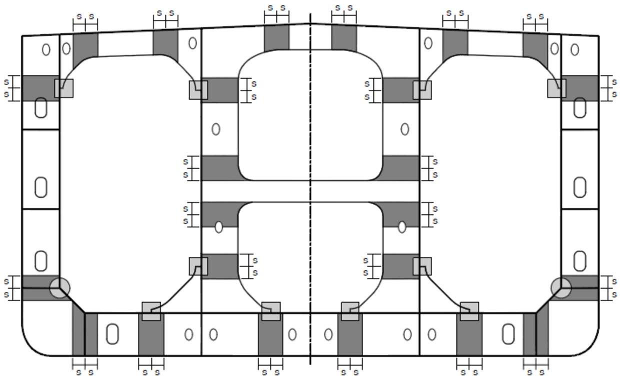

Fig. 10: Fine mesh zones according to JTP requirements.

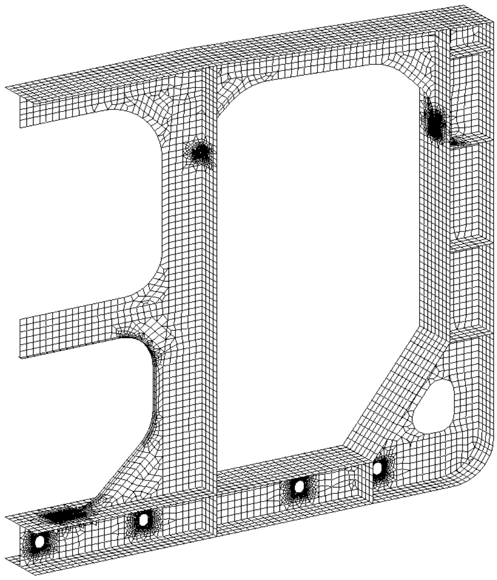

Fig. 11: Web frame mesh with local fine mesh areas (by Napa Ltd).

In most cases, it is desired to apply solely quadratic mesh at the free boundary for potentially high stress areas. See the following mesh example on a tanker model. The applied global mesh size is 0.425 mm, which in this case, results in two elements between the longitudinal stiffeners. The given element size for fine mesh areas is 40 mm. Also, the NAPA user can choose how smoothly the fine mesh area conjoins to the overall (global) mesh. Fig. 11 shows the mesh for the example case. Only the port side of full breadth model is shown. Fig. 12 shows the mesh for a bracket toe. Full-quad mesh is applied at the boundary.

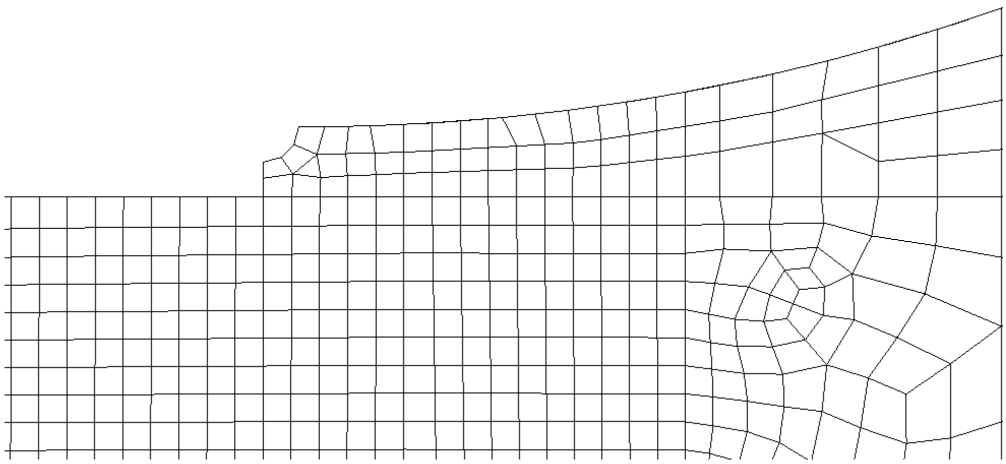

Fig. 12: Bracket toe mesh (by Napa Ltd).

Fig. 13 and Fig. 14 show the meshes for a bracket toe, a faceplate end-cut and an opening. Also, full-quad mesh is applied to the opening boundary, which is a potential area to contain high stress levels, to ensure the best possible estimate for stress results.

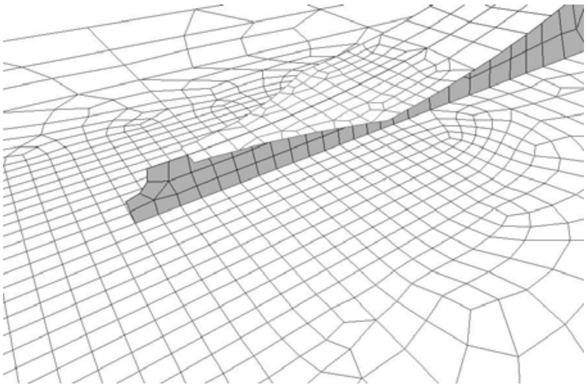

Fig. 13: Bracket toe, innerskin, and faceplate end-cut (by Napa Ltd).

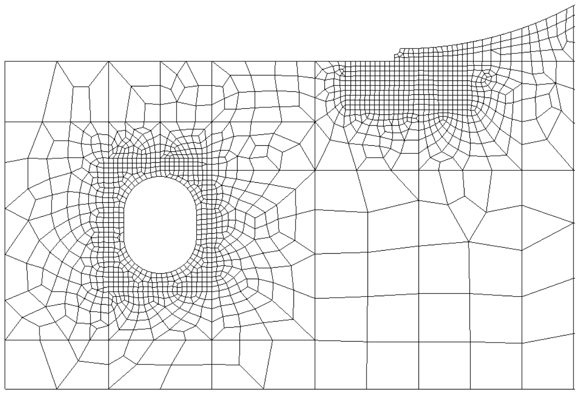

Fig. 14: Mesh refinements for an opening and a bracket (by Napa Ltd).

Conclusions

This paper illustrated some benefits in parametric modelling and automatic mesh generation based on 3D product model. The introduction stated that it is profitable to have a global finite element model as early as possible during the early design and if structural concepts can be evaluated effectively with FEA, these factors yield an optimum situation where the results of the finite element study can be utilized during the design process. This paper presented a method and a system, which help to reach the point where finite element models can be created with very fast pace and good quality up-to-date mesh is available for FEA at any point during the design process.

References

DUMEZ, F.X.; CHEYLAN, C.; BOITEAU, S.; KAMMERER, C.; MOGICATO, E.; DUPAU, T.; BERTRAM, V. (2008), *A Tool for Rapid Ship Hull Modelling and Mesh Generation*, 7st Int. Conf. Computer and IT Appl. Maritime Ind., Liege, pp.6-18

von SELLE, H.; Doerk O.; Scharrer, M. (2009), *Global Strength Analysis of Ship with Special Focus of Hatch Corners*, Proceedings of the 2nd International Conference on Analysis and Design of Marine Structures, March 16-18, 2009.

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., Budapest, pp. 374-389

JANG B.; SUH Y.; KIM E.; LEE, T.; (2008), *Automatic FE modeler using stiffener-based mesh generation algorithm for ship structural analysis*, Marine Structures 21 (2008), pp. 294–325

KUUTTI, I.; UPPALA, V. (2005), *Practical Design Approach to Cost-Optimized Ship Structures*, 12th Int. Conf. on Computer Applications in Shipbuilding, Building

Common structural rules for oil tankers (2006)

Tommi Kurki
Napa Ltd
P.O. Box 470
FI-00181 HELSINKI
FINLAND
Tommi.Kurki@napa.fi