NAPA/MAESTRO Interface

Reducing the Level of Effort
for
Ship Structural Design

12/3/2010

Table of Contents

Introduction

This document captures details of the NAPA/MAESTRO Interface (NMI) being developed by Napa Ltd and DRS Defense Solutions, LLC, Advanced Marine Technology Center (DRS). This document includes a brief introduction to the NAPA product and the MAESTRO product and how interfacing the two products can assist in making the structural design process more efficient by leveraging a single 3D structural model. This document also provides a level of effort estimates for two analysis approaches: first, a MAESTRO-only approach, where the model is built and analyzed in MAESTRO, and second, a combined approach where the NAPA model is converted to a MAESTRO model, which is then analyzed in MAESTRO. Finally, a brief description of the current development pursuits and priorities are provided.

Why Create a NAPA/MAESTRO Interface

NAPA Steel is a widely used ship structural design tool used during the early design stages. NAPA is used for various ship design purposes, such as calculation of weights, painting areas, generating data for production planning and cost estimation, and creation of basic drawings (e.g., drawings for Classification submittal and approval). The NAPA model can be converted into a Finite Element Model (FEM) and exported to various FEM systems (e.g., Nastran). The NAPA model can also interface with detail design systems and classification societies' systems, which ensures integration during the whole ship design process.

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 buckling analysis (e.g., at the hull girder level, stiffened panel level, and local member level), and design evaluation.

Interfacing these two products will bring more efficiency to the early stage ship structural design, analysis, and evaluation process. It will do 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.

Level of Effort Comparison for Two Different Approaches

Sample Case Description

A level of effort comparison was undertaken to quantify the potential efficiency of interfacing NAPA data (i.e., the FEM data, loading data, etc.) with MAESTRO. To perform this comparison, a sample data set of a chemical tanker cargo area was evaluated (see Figure 1). Additional assumptions were made to describe a normal scope of work (SOW) for this type of vessel. There were no specific Classification requirements provided, although it was assumed that reports would be generated for Classification review. Further, there was no specification for computing and imposing hydrodynamic loads or performing fatigue analysis. Table 1 provides the assumed SOW for this comparison.

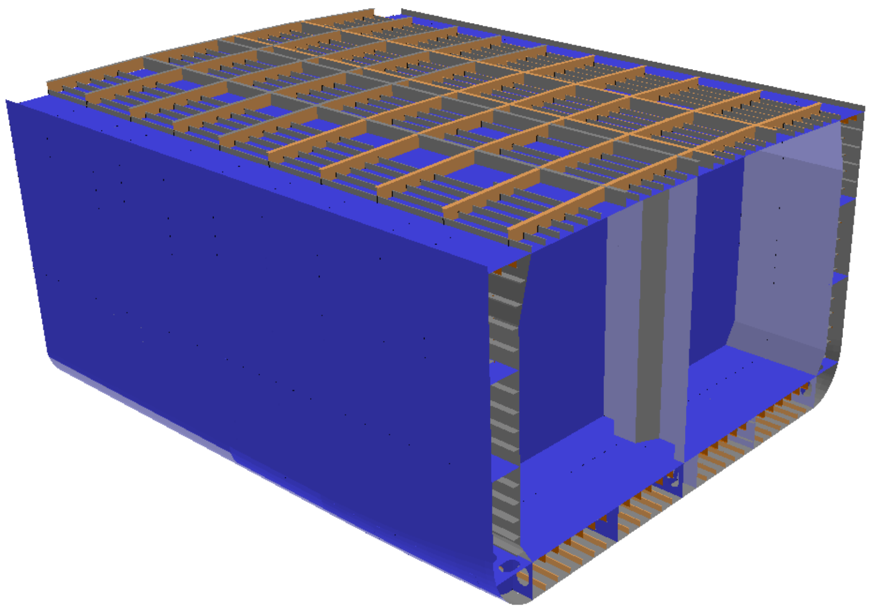


Figure 1 - Chemical Tanker Sample Data

Cargo Area Only	Full Ship
• Develop 3D FEM (only cargo area) • Develop Key Loading Conditions • Perform 3D Response Analysis • Optimize Structure to Reduce Weight • Re-analyze Optimized Structure • Develop 3D Fine Mesh FEM for Critical Areas • Perform 3D Response Analysis on Fine Mesh	• Develop 3D FEM (to capture full ship) • Perform Global Free Vibration Analysis • Perform Local Free Vibration Analysis of Selected Stiffened Panels & Sub-structures

Table 1 - Assumed Scope of Work

Comparison Details

Based on the assumed SOW, activities were developed and sequenced covering the Tasks listed below. Labor hour estimates where allocated to Junior, Senior, and Principal personnel at a distribution of approximately 75%, 20%, and 5% respectively. Although this may be different from organization to organization, it provides insight to the potential efficiencies of using the NMI. The percentages are provided below and represent the savings for using the NMI during the course of the listed activity.

TASK DESCRIPTION	PERCENTAGE SAVED (%)
Task 1.0 Initial Iteration (Cargo Area Only)	**71**
1.1 Develop Mid-level Mesh FEM	89
1.2 Develop Loads/Pre-processing Analysis	67
1.3 Perform Analysis, Post-processing, & Correspondence	0
Task 2.0 Second Iteration (Cargo Area Only)	**52**
2.1 Explore Design Changes to Optimize Weight	0
2.2 Update Mid-level Mesh FEM	85
2.3 Update Loads/Pre-processing Analysis	67
2.4 Perform Analysis, Post-processing, & Correspondence	0
Task 3.0 Third Iteration (Cargo Area Only)	**58**
3.1 Explore Design Changes to Optimize Weight	0
3.2 Update Mid-level Mesh FEM	85
3.3 Develop Fine Mesh FEMs	78
3.4 Update Loads/Pre-processing Analysis	67
3.5 Perform Analysis, Post-processing, & Correspondence	0
Task 4.0 Coarse Mesh Full Ship Analyses	**53**
4.1 Develop Coarse-level Mesh FEM (i.e., fwd & aft of cargo area)	78
4.2 Update Loads/Pre-processing Analysis	78
4.3 Perform Global Free Vibration Analysis	0
4.4 Perform Local Vibration Analysis	0
Task 5.0 Classification Initial Submittal	**0**
5.1 Generate Analysis Report for Submittal	0
5.2 Submit Analysis Report	0
5.3 Respond to Classification Comments	0
Task 6.0 Classification Final Submittal	**0**
6.1 Update Analysis Report for Submittal	0
6.2 Submit Final Report	0
TOTAL SAVINGS	**45**

Table 2 - Percentage Savings using NMI

Comparison Conclusions

As expected, there are particular activities that are not affected by the NMI; therefore, there are no savings for these particular activities. This comparison assumed three analysis/design iterations for the cargo area, while assuming only one iteration for the full ship analysis. Based on these assumptions, it is estimated that a savings of approximately 45% can be realized using the NMI approach. This comparison is obviously based on only one data set. This type of comparison should be re-examined with multiple data sets and their respective Classification rules. Further, this comparison is based on the current state of the NMI development.

Using the NAPA/MAESTRO Interface for Structural Design

The MAESTRO Neutral File

Figure 2, depicts the workflow for using the NMI in a ship structural design process. 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). Napa and DRS AMTC are also working on translating the pertinent loading information. 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). MAESTRO has many different types of ship-based loading patterns, which include the ones listed above; therefore, it is not difficult for MAESTRO to leverage this NAPA defined loading data.

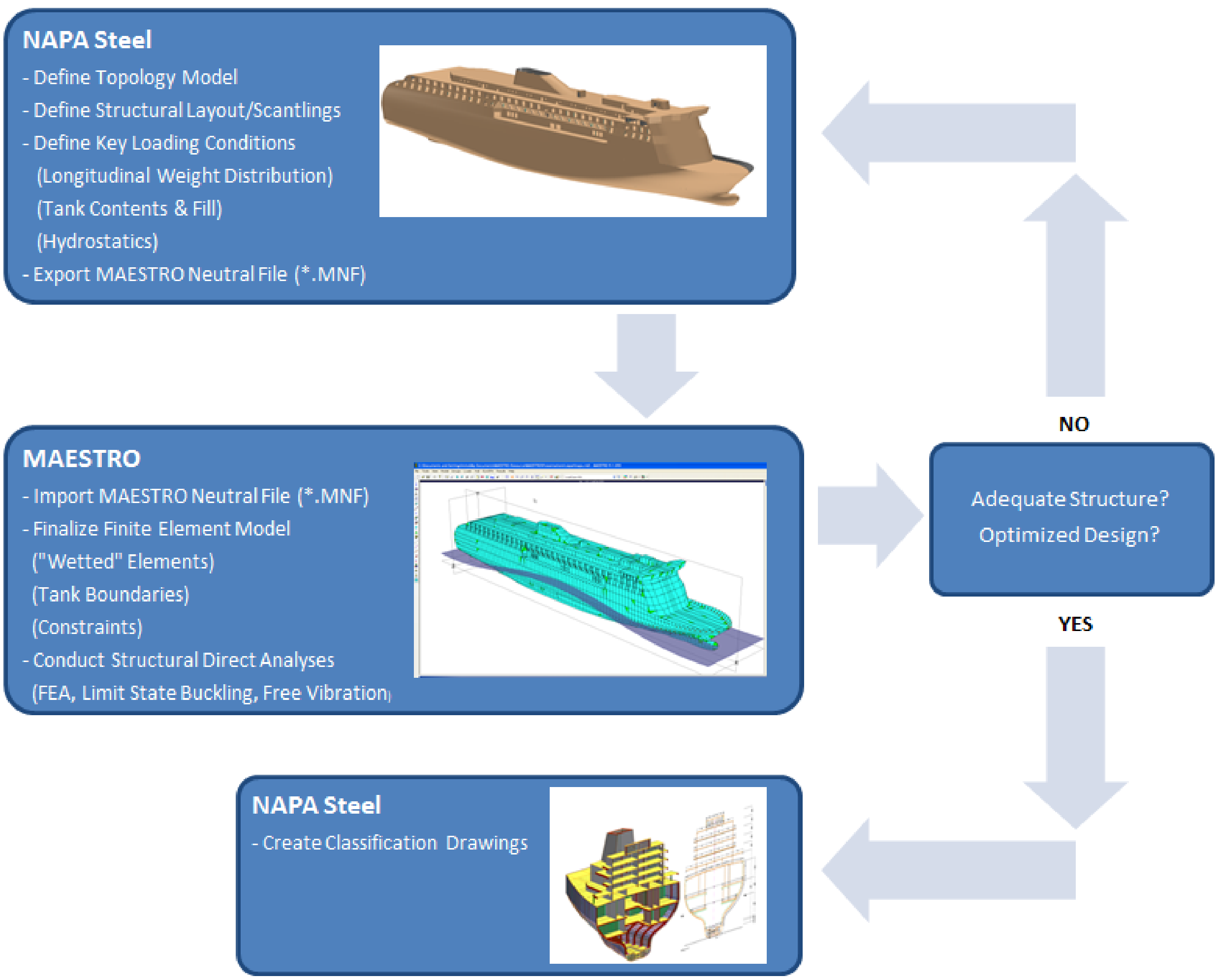
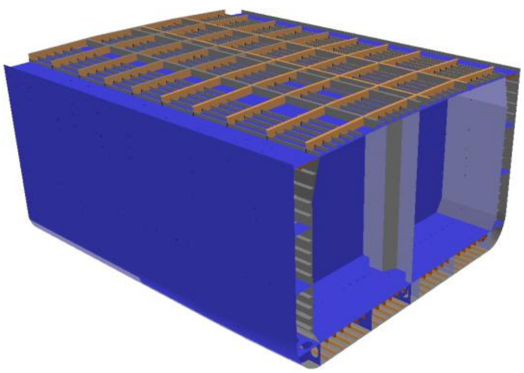


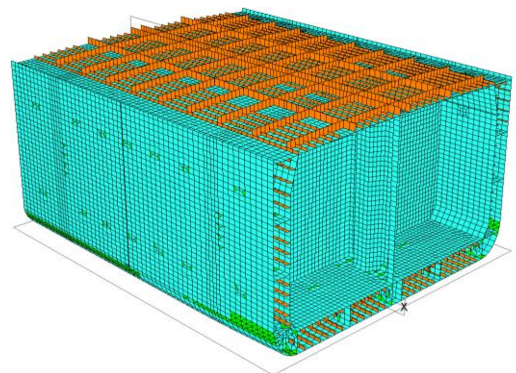
Figure 2 - NAPA/MAESTRO Interface Workflow

Finalizing the MAESTRO Finite Element Model

Once the user has imported the NAPA-generated MAESTRO neutral file, there are two tasks to complete before analysis can be conducted. The first task involves performing integrity checks on the FEM to ensure it is a valid model and ready for analysis, which is titled *Finalize Finite Element Model* in Figure 2. It should be noted that checking the integrity of the FEM is necessary when performing finite element analyses, whether the analyst is building the FEM from scratch or importing it from a 3rd-party system. MAESTRO's integrity checks include: element connectivity, uniform element positive pressure sides, proper *wetted* element definition, and proper tank definition. Figure 3 shows two different imported NAPA models and Figure 4 shows the hull *wetted* element definition, which is important for MAESTRO's ability to properly impose hydrostatic load. MAESTRO has the ability to modify this definition, if necessary, to facilitate proper hydrostatic loading, which is also shown Figure 4. Similar integrity checks can be run to verify that tank boundaries and pressure normals are properly defined, which is shown in Figure 5.



(a) NAPA Steel Model



(b) MAESTRO FEM

Figure 3 - MAESTRO finite element model generated by NAPA Steel

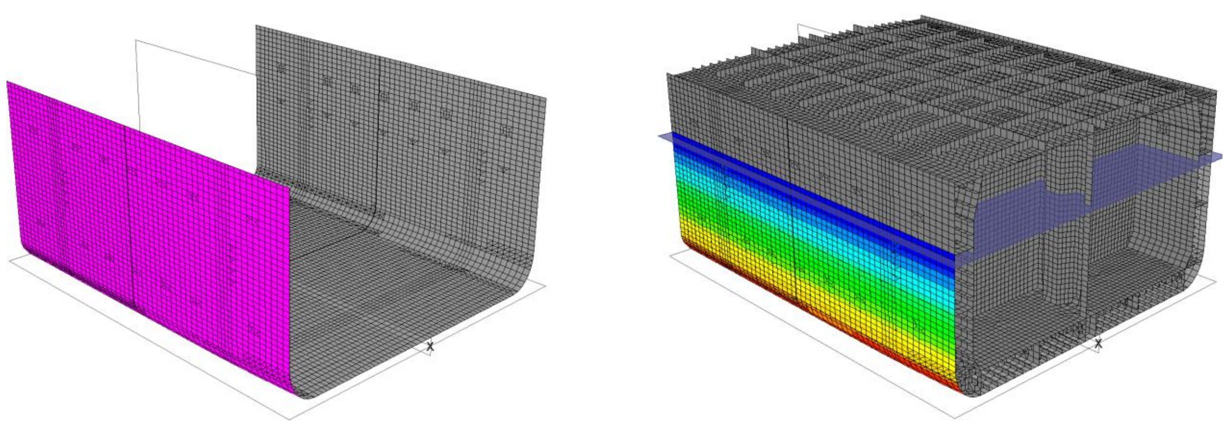


Figure 4 - MAESTRO "wetted" elements for hydrostatic loading

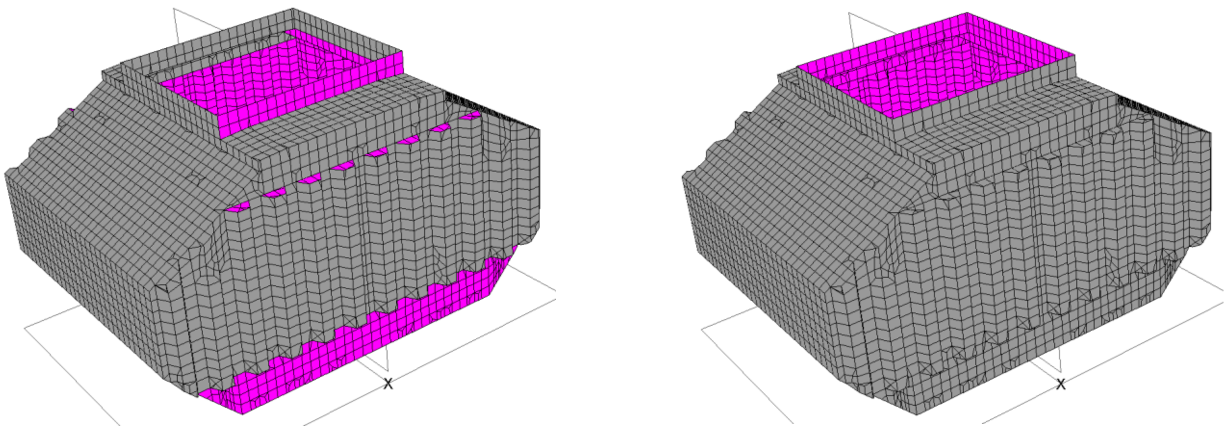


Figure 5 - Tank boundary definition and creating a consistent normal definition

After the model has been checked for integrity, the user can now focus on properly constraining the FEM, which will finalize the model and then it will be ready for analysis. The constraint definition will restrict the model's movement in any of the 3 translational or 3 rotational degrees of freedom.

Conducting Structural Direct Analyses

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 buckling analysis. The limit state buckling analysis includes hull girder collapse analysis, stiffened panel buckling analysis, and local member buckling analysis.

Finite Element Analysis

The first step in comprehensive structural assessment is conducting structural response analysis. This encompasses the computation of deformations and stresses based on finite element analysis methodologies. 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. Figure 6 shows the variety of stress results that can be recovered from the MAESTRO FEM. Stress results can be graphically plotted and dynamically queried, which allows the analyst to effectively post-process the structural response.

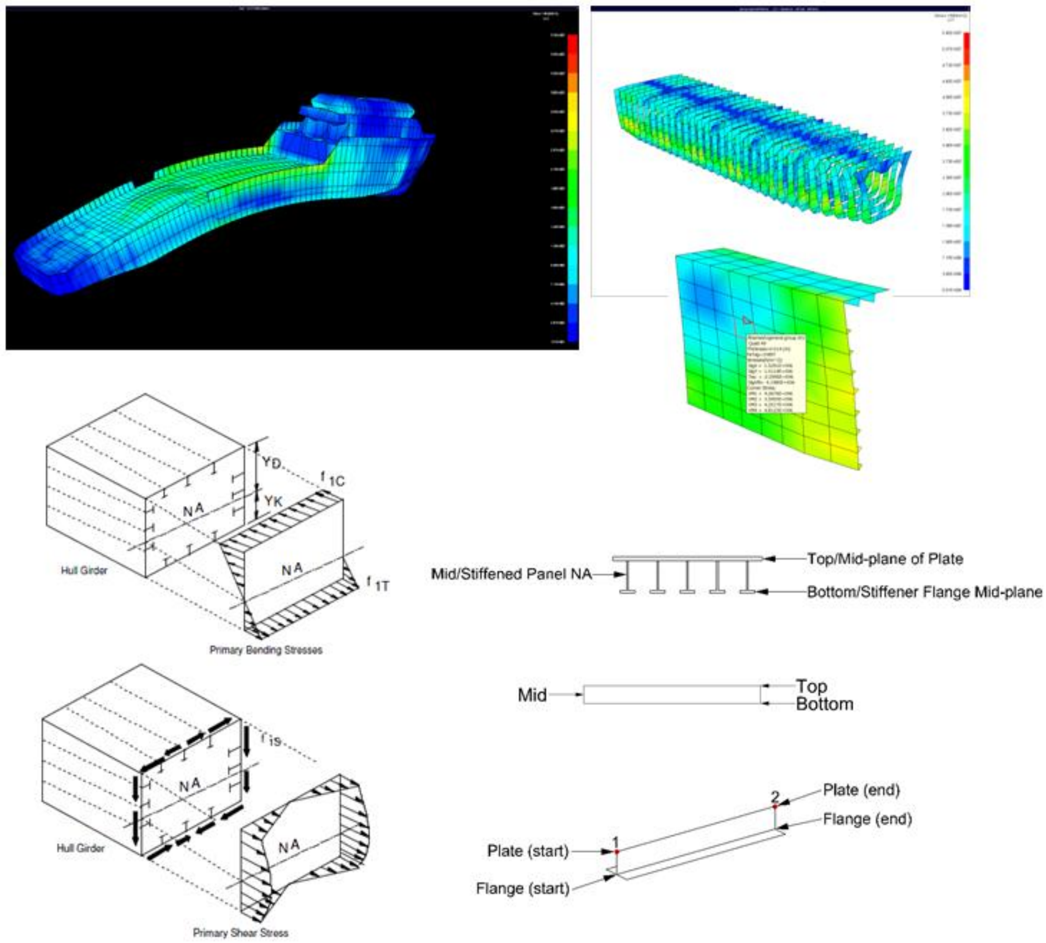


Figure 6 - MAESTRO Response Analysis

Limit State Buckling Analysis

A comprehensive structural design assessment does not end with deformation and stress assessment. Comprehensive structural assessment should include evaluating structural stability and load-carrying capacity. This aspect of structural assessment (i.e., the limit state analysis) has been a core component to MAESTRO from its inception. The formulation of MAESTRO's limit state analysis is covered in two industry standard textbooks: "Ship Structural Analysis and Design" (Hughes and Paik) and "Ultimate Limit State Design of Steel-plated Structures" (Paik & Thayamballi). These textbooks constitute the theoretical manual for MAESTRO's limit state analysis. It should be noted that these limit state formulations are currently being exercised in an International Ship and Offshore Structures Congress (ISSC) and ISO Benchmark Study, and are expected to be adopted as an ISO TS 18072-2 standard for ultimate strength.

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. For ultimate strength of stiffened panels, six collapse modes are evaluated.

These six modes are illustrated in Figure 7 and are categorized as follows:

- 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

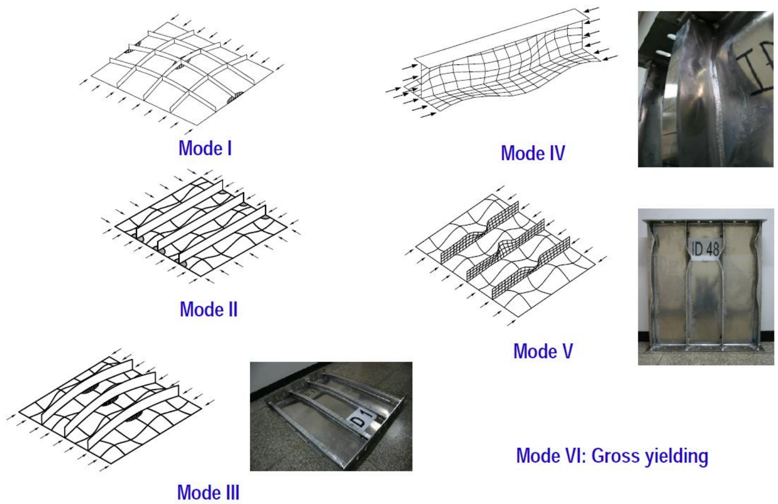


Figure 7 - Ultimate Strength of Stiffened Panels, Collapse Modes

This limit state analysis is done within MAESTRO automatically and comprehensively for the entire FEM and all loading conditions. To properly perform strength assessment, MAESTRO defines the *true stiffened panel* 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 (see Figure 8).

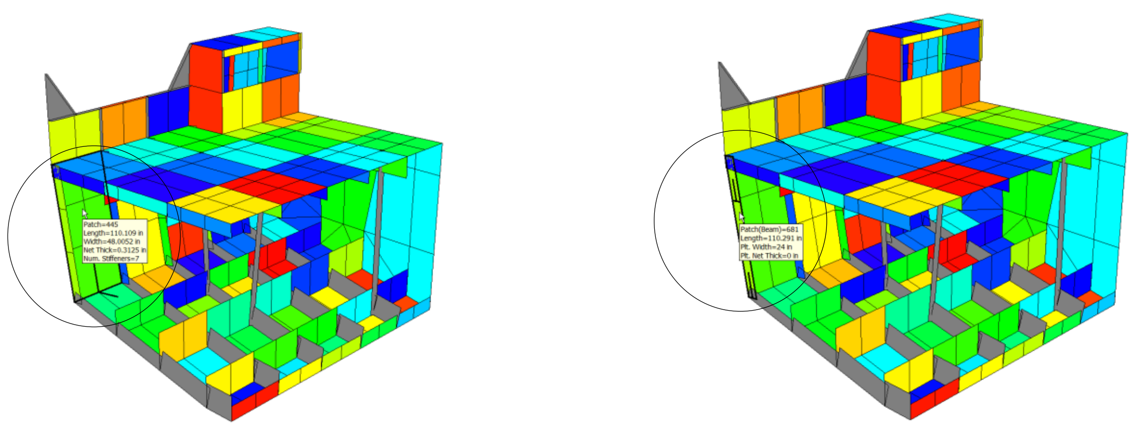


Figure 8 - MAESTRO Evaluation Panels

Figure 9 shows an example of FEM that was imported from NAPA, restrained, automatically balanced, and analyzed in an extreme Sagging and Hogging case (it should be noted that the loads associated with this response are not a real scenario and thus only for demonstrating purposes). MAESTRO successfully conducted a stress analysis and a limit state analysis, which included the creation of true evaluation panels as shown in Figure 10.

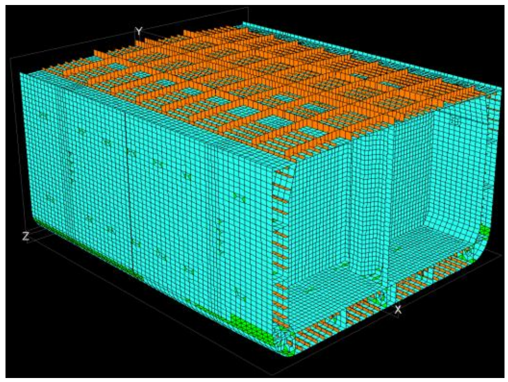
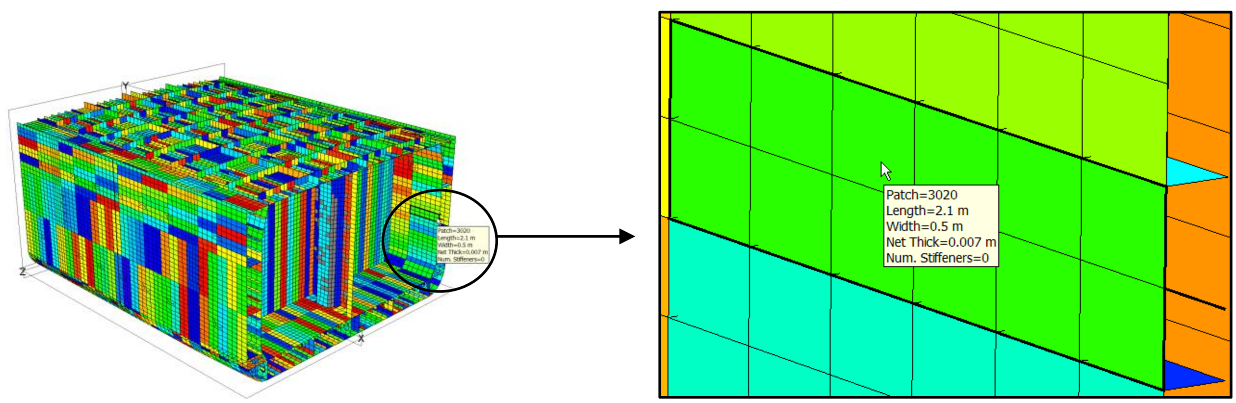


Figure 9 - MAESTRO Response Analysis



(a) Full FEM

(b) Zoomed-in View of Single Panel

Figure 10 - Sample Model Evaluation Panel Creation

Fine Mesh Analysis

The FEM can be refined in areas of interest or concentrated areas of stress using either MAESTRO or NAPA. MAESTRO's fine mesh module allows the user to create refined 3D FEMs of any portion of the MAESTRO model quickly. This is done by creating groups of interest areas and then refining the group based on two different methods: Top-down or Embedded.

Free Vibration Analysis

MAESTRO can compute the natural frequency of the full ship FEM in both a *Dry* mode and a *Wet* mode. The latter automatically computes the added mass associated with the hull *wetted* elements that are below the waterline.

Revising and Reassessing the Structural Design

After conducting the finite element analysis and post-processing the results, the designer can revise the scantlings in NAPA and rerun the analysis. Currently, this is a manual process, but Napa and DRS are exploring ways to automate this feedback loop.

Producing Classification Drawings

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 NAPA model serves as the source for creating these 2D drawings. This utilization of the 3D model is currently being used among NAPA customers in different levels. This leads to a remarkable savings in developing *class* drawings. With the current version of NAPA, the fully automatic drawing creation needs development work from the customer to complete company standard quality drawings.

Interface Development Priorities

The above describes the envisioned workflow and current capability of the NMI. This development and progress has shown great promise thus far. The following items are being pursued and improved upon to increase the robustness of the interface.

Finite Element Modeling

Napa and DRS are working to resolve a few issues related to generating the FEM mesh. The issues are related to the following: Warped Quads, Internal Quad Angles, Free Edges, and Tank Boundary definition.

Loading Conditions

Napa and DRS have identified the pertinent loading data to be transferred to MAESTRO. Formats for transferring this data have been agreed upon and testing of more loading scenarios is necessary.

Analysis

All of the required analysis capability is available in MAESTRO. As described in previous sections, MAESTRO has an existing limit state analysis paradigm. This paradigm includes performing limit state analysis and collecting elements to represent the true stiffened panel. This paradigm is flexible in the sense that other *criteria*, such as those directed by Classification societies, can be implemented if necessary.

Optimization/Feasibility

MAESTRO has the ability to perform optimization of structural scantlings using the Monte Carlo method and the Simulated Annealing method. This capability is in an Alpha stage of development, but can potentially be leveraged to assist the designer in finding an optimized design solution. Napa and DRS are exploring ways to bring this technology to the design process.