MAESTRO Manual

© 2014 Optimum Structural Design, Inc.

Information included herein is categorized as ECCN EAR99 under the Export Administration Regulations (15 CFR §730-774) issued by the U.S. Department of Commerce. An export license issued by the U.S. Department of Commerce or EAR exception may be required prior to export or transfer of this information to certain parties or end-uses. Public release of this document is not authorized. Diversion contrary to U.S. law is prohibited.

Table of Contents

© 2014 Optimum Structural Design, Inc.

© 2014 Optimum Structural Design, Inc.

© 2014 Optimum Structural Design, Inc.

© 2014 Optimum Structural Design, Inc.

© 2014 Optimum Structural Design, Inc.

Welcome to MAESTRO 11.0.0

1

1 Welcome to MAESTRO 11.0.0

Welcome to MAESTRO 11.0.0, a comprehensive program that leverages the power of Finite Element Analysis and Structural Evaluation for the naval architect.

MAESTRO is a finite element tool providing powerful structural design and analysis capabilities for naval architects.

A robust application, backed up by first class support.

MAESTRO has been designed and created by a group of naval architects that care about your experience with the software. Simply put, we feel successful when our software can leverage your design talents to create better vessels. If you have questions that can't be answered through this Help File or the forum, feel free to contact us at support@maestromarine.com. We enjoy hearing about your projects, your application of MAESTRO, and your challenges, and will do our best to help.

© 2014 Optimum Structural Design, Inc.

Latest Updates

2

2 Latest Updates

Updates and changes to MAESTRO will be described here, listed by release number. You can also read the Release Notes, which are included as part of the installation, to see a complete history of the releases and changes to MAESTRO.

2.1 Version 11.0.0

MAESTRO Release Notes

MAESTRO is a finite element analysis program for rationally-based analysis, evaluation, and structural optimization of ships, offshore structures, and other large complex thin-walled structures.

MAESTRO Requirements

- Operating Systems tested: 32-bit Windows XP and Vista, 32-bit and 64-bit Windows 7
- Operating Systems ***not supported***: Windows ME, 98, 95, NT
- Mac: The Intel Mac with Bootcamp or Parallels ***has not*** been tested
- Microsoft .NET Framework 3.5, SP1.
- Valid license (without this, MAESTRO will operate in the demonstration mode)
- Installation of Sentinel System Driver and Sentinel Protection Server 7.6.6. (Automatically installed with MAESTRO).

Version 11.0

New and Enhanced Modules

The following is a list of the new and enhanced modules found in MAESTRO v11.0. A copy of this list of features can be found in the Release Notes.

- **New Ultimate Limit State (ULS) Heuristic-based Optimization Module:** A new optimization method which uses a variety of heuristic-based optimization methods (i.e., Simulated Annealing, Genetic Algorithms) and MAESTRO's ULS evaluation paradigm has been added to the existing Optimization module.
- **MAESTRO-Wave:** A new time-domain MAESTRO-Wave module has been added. In addition, new wet panel discretization options have been added for the frequency and time domain MAESTRO-Wave calculations including: 2D strip theory using the free surface Green's Function, a high-speed 2.5D Rankine Source Method with forward speed correction, and the Universal RAO method. A new option to speed up computation time has been added to allow MAESTRO to calculate the velocity potential (thus radiation and diffraction pressure) as a constant value for all elements

within the hydrodynamic panel instead of the standard method which calculates them on each FE element individually.

New and Enhanced Features

The following is a list of the new and enhanced features found in MAESTRO v11.0. A copy of this list of features can be found in the Release Notes.

- **Design Wave Creation:** The design wave dialog now allows the user to use a custom sea spectrum definition and also select which wave frequencies to use in determining the extreme design wave.
- **Operating Profiles and Wave Spectra:** A number of pre-defined operating profiles and wave spectra are now available with the MAESTRO distribution.
- **Mean and Extreme Stress Calculations:** A new capability has been added to compute mean stress and extreme stress from a computed RAO database.
- **RAO and Response Spectrum Calculations:** MAESTRO can now calculate motion and stress RAOs at specific locations/elements.
- **Inertia Relief Restraint Method:** A new automatic restraint method has been added in which the user does not have to apply traditional restraints to the model. Instead, MAESTRO automatically creates a fixed node at the model center of gravity which is attached to surrounding structure using soft spring elements. Inertia relief is required to be run prior to solving the model.
- **ABS High Speed Naval Craft Rules Added:** The stress and buckling checks from the ABS High Speed Naval Craft (ABS HSNC) guide have been added to MAESTRO 's limit state framework.
- **Import Mass Distribution from MAESTRO .wet File:** A MAESTRO .wet file can be re-imported into MAESTRO and used for a mass distribution. This allows a user to import a hydrodynamic mesh and full FE model mass distribution to run MAESTRO-Wave for motion and hull girder load RAOs.
- **Hide Elements:** A new command has been added under the View menu that allows a user to hide elements using a variety of selection options.
- **Group Operations and Group Manager:** Two new dialogs have been added allowing the user to more easily manipulate existing groups or create new ones. The Group Operations allows a user to create new groups by copying, adding, subtracting, or finding common elements between one or more existing groups. The Group Manager allows a user to move one or more groups into a new category or subfolder within the groups tree.
- **Bending Moment without Axial Force Contribution:** MAESTRO can now plot the longitudinal bending moment without axial force contributions which makes it easier to compare results to other tools that do not account for the axial force (e.g., section based calculations).
- **Merge Hydrodynamic Panel:** An option has been added to automatically merge evaluation panels into large panels to be used in hydrodynamic (MAESTRO-Wave) calculations.
- **Map Hydrodynamic Patch:** Allows the user to run MAESTRO-Wave with a custom hydrodynamic mesh of a structural model and map the source strengths from the

hydrodynamic mesh to the structural model.

- **Level 1 Evaluation Patch Search:** A new evaluation patch search method has been added allowing the user to define the beam elements representing the patch boundary supports. The patch search/creation can now be performed on the current view part, in addition to the entire model. Also, the patch search functionality has been optimized to reduce computation time.
- **Negative Adequacy Parameter Reporting:** A new feature has been added to report the number of elements and corresponding structural weight of failing structure. All negative adequacy parameters, including minimum adequacy parameter, are now reported from the list elements command.
- **Weight Distribution Load Pattern:** The weight distribution algorithm has been improved to be more efficient.
- **Tank Load Reader:** A new import option allows the user to import a text file of tank loads to create one or more new load case definitions. Sample sources of this data include weight reports, spreadsheets, or a GHS tank load export script.
- **Polygon Mesh File Format Extended:** The polygon mesh file format has been extended to allow users to also import endpoint and strake definitions.
- **Beam Shapes Added:** The following beam shapes have been added: tube, box, and I.
- **Group Selection Options:** A variety of group selection options have been added including selecting by evaluation patches and adding by seed, attached properties, and attached beams.
- **Nastran Import Additions:** MAESTRO now imports concentrated masses and forces from NASTRAN models as well as preserves property names created in FEMAP. FEMAP Neutral (.neu) files can also be imported which preserve FEMAP group information.
- **MAESTRO-Wave Liquid Tank Loads:** New options have been added to the MAESTRO-Wave module to account for tank sloshing loads.
- **Directional Spring and GAP Element Added:** The existing spring element has been expanded to include directional springs and GAP elements.
- **Copy Elements:** A new feature has been added to quickly create one or more new elements in a selected direction by copying existing elements in the model.
- **Split Strake:** A new refinement option has been added allowing the user to “split” a strake into multiple strakes so that all stiffeners are now defined as girders.
- **ALPS/HULL Report:** A new option has been added to the ALPS/HULL results menu allowing the user to create a report of the ALPS/HULL results.
- **Multiple ALPS/HULL Results:** The ability to run and store ALPS/HULL results for more than one ALPS/HULL model has been added.
- **Design Pressure:** The design pressure option has been added back to the evaluation patch dialog allowing a user to specify an additional design pressure to be used for adequacy parameter calculations.
- **SN Curve Added:** The Eurocode 9 SN curve for aluminum has been added for spectral fatigue calculations.

- **Mass Distribution CG Calculator:** A new feature has been added to calculate a mass distribution's VCG and TCG in order to reach a target VCG and TCG for the full load case.

Bug Fixes:

- **Element Directional Display:** This was feature was not working correctly in version 10.
- **Boundary Modules:** Bending moment and shear force loads for boundary modules were previously being applied as full value for a half model. Half models now use the half load value.
- **64-bit Cutting Plane Selection:** A bug with selecting cutting planes in the 64-bit version has been resolved.
- **Inertia Relief All Load Cases:** A bug was fixed that reset the static waterplane to the first load case when running inertia relief balance for all load cases.

Version 10.0.5

- **SFA Analysis with Top-Down Model:** The spectral fatigue analysis can now be run with top-down fine mesh models.
- **Operating Profile Enhancement:** The operating profile used with the ELA and SFA modules can now have unequal probabilities.
- **S-N Curve Parameters Now Visible:** The user is now able to view the parameters for the available S-N curves in MAESTRO.
- **Bug Fix:** MAESTRO now allows multiple SFA groups to be created.

Version 10.0

New and Enhanced Features:

- **MAESTRO-Wave:** New module which provides the ship designer with an integrated frequency-domain computational tool to predict the motions and wave loads of any vessel. To compute these hydrodynamic motions and loads, MAESTRO-Wave first calculates the velocity potentials, source strengths, and flow velocities at the centroids of the hydrodynamic panels for each speed, heading, and frequency requested, and then maps the source strength to the structural panels. The equations of motion are formulated using the structural mesh rather than the hydrodynamic mesh. This approach results in a perfect equilibrium for the structural model. Bending moments, shear forces and torsional moments are automatically in closure. No inertia relief and artificial loads are needed to balance the model. The computation of these hydrodynamic forces is based on 3D potential theory using the zero speed Green's function with a speed correction parameter. A variety of visualizations and output data are available for purposes of post-processing.

© 2014 Optimum Structural Design, Inc.

- **Extreme Load Analysis (ELA):** New module which imports unit wave load data from a hydrodynamic analysis (MAESTRO-Wave or other) via *.smn file format. It then allows the user to calculate hull girder load response RAOs, and provides the necessary short-term and long-term statistical computations to predict extreme values of the maximum loads for a given vessel.
- **Spectral Fatigue Analysis (SFA):** New module which provides the ability to perform global fatigue screening of the vessel. The SFA module introduces additional functionality to the ELA module to compute Stress RAOs, displacement RAOs, define and associate structural groups to SN curves and Stress Concentration Factors (SCFs), and compute fatigue damage based on the Miner cumulative damage principle.
- **MAESTRO/NAPA Steel Interface:** New module which allows the designer to import a NAPA/NAPA Steel 3D model, which includes the finite element model (geometry, scantling properties, and finite elements), loading information (longitudinal weight and bending moment distributions, tank boundary/content/fill definitions, and hydrostatic equilibrium definition), and model hierarchy definitions such as MAESTRO module group. This interface enables shorter design cycle times by using a single 3D structural design model (from NAPA) that can be re-used as the design matures and is ready for analysis in a matter of minutes.
- **VERES Export:** Allows user to export MAESTRO model information to *.MGF, *.M3D, and *.M2D files to support VERES analyses.
- **PRECAL Export:** Allows user to export MAESTRO model information to *.HUL, *.HIN, and *.CND files to support PRECAL analyses.
- **64-bit Version:** Now available in a 64-bit version.
- **ALPS/ULSAP Update:** This module has been updated to version 2011.2.
- **Results View Updates:** When user-defined range is set, elements below that range are set to blue and elements above that range are set to red.
- **Expanded Group Organization Feature:** Allows subfolders to be created underneath the default group types in the groups tree.
- **Drag and Drop Group Creation:** Groups may now be dragged and dropped from one type to another to automatically extract certain elements from an existing group. For example, nodal groups can be created from plate or general group and shell elements may be recovered from node groups or a general group.
- **New Tank Features:** Allows volume groups to be defined as flooded to model flooded or damaged conditions. User can adjust tank permeability and apply deballast pressure.
- **Hull Girder Load Target:** The hull girder load target is a new load pattern that can be used to ensure that either a minimum shear force envelope or a minimum bending moment envelope is met or to add an additional shear force or bending moment on top of all other load patterns.
- **Create Boundary Modules:** Option to assist in applying end moments. A master node is automatically created and joined by RBE2 elements to all nodes at modules ends to ensure that plane sections remain plane. End Moment load is then applied at this master node.
- **Weight and Buoyancy Density Plot:** Option to plot weight and buoyancy per unit

© 2014 Optimum Structural Design, Inc.

length for easier comparison to other hydrostatic programs.

- **Create Module from General Group:** Right-click menu option to create a new module consisting of the elements defined in the selected general group. These elements are removed from their original module.
- **Extract Module:** Option to create a new module by "extracting" elements from an existing module by defining a bounding box. This allows large or full ship imported models to be easily separated into more manageable modules.
- **Define General Group from List of Nodes or Elements:** Select multiple elements/nodes from the grid tab and create a general group using the right-click menu.
- **Mass Distribution Exclude Nodal Groups:** The mass distribution load pattern now allows selected nodal groups to be excluded from receiving the loading.
- **Speed Enhancements:** MAESTRO's loading speed has been increased for models with large numbers of groups defined. The auto-creation of evaluation patches has also been increased to handle larger and finer meshed models.
- **Displacement Direction Plot Added:** Option to select single direction (i.e. X, Y, Z) only to plot deformation. Also added option to plot "true" scale deformation.
- **Cutting Planes added to Right-Click Menu:** The same cutting planes menu that appears in the Tools menu is now added to the menu when you right-click the mouse in the modeling space.

Bug Fixes: A number of minor bugs have been fixed.

Version 9.1.0

New and Enhanced Features:

- **User Defined Extreme Design Wave:** Capability to facilitate Extreme Load Analysis has been added to the MAESTRO main menu. This includes the ability to graphically review an imported load database (via complex Ship Motion file) and the creation of extreme design loads.
- **Ship Motion File Complex Form:** The Ship Motion file has been expanded to allow the importing of Complex displacement, acceleration, and pressure data calculated by 3rd party seakeeping software. This file is backward compatible with the legacy Ship Motion file.
- **Weight Distribution Load Pattern:** Added the ability generate a weight distribution by either entering weights or weight densities at their corresponding longitudinal locations. This load patter can be combined with other existing load patterns.
- **New Midship Section Design Tutorial:** This tutorial will walk through the complete process of creating and analyzing a midship section given a basic set of design parameters and hull geometry.
- **Group Creation & Modification:**
    - Added a method to facilitate the identification of wetted elements

  - Added several methods for creating General groups and Volume groups including an automatic search algorithm
  - Added methods to automatically generate consistent normals within a group
  - Ability to add elements during group creation from the groups view
- **Nastran Import/Export:** The following functionality has been added to the Nastran import/export capability:
  - Significantly improved the importing speed
  - Import Spherical and Cylindrical coordinate
  - Import/Export PBARL/PBEAML cards
  - Export "boundary only" data file (filename.nas.spcd)
  - Import Nastran model as a Group
- **Hull Girder Cross Section Property Calculation:** A new method for defining cross section locations has been added. This functionality allows you to recover hull girder section properties outside of the traditional endpoint locations. This includes recovering hull girder properties from an imported Nastran finite element model.
- **Equation Solver Method:** The mechanism to choose a different Equation Solver method (Sparse, Iterative, or Skyline) has been moved to the File/Preferences dialog. Previously, this mechanism was located in the Analysis/Evaluation dialog.
- **Longitudinal Effectiveness:** Added the ability to assign longitudinal effectiveness to finemesh models and General groups.
- **ALPS/ULSAP:** This module has been significantly updated to the version 2010.4. Additional reference papers have been added to the Help system as well as updated verification models.
- **ALPS/HULL:** This module has been updated to allow the user to impose pressures (i.e., hydrostatic, volume, linpress, etc.) on the defined ALPS/HULL analysis model.
- **Simple Object Mesh Template:** Added the ability to auto-generate the following MAESTRO models:
  - Stiffened Panel with Initial Deflection
  - Sphere
  - Cylinder with/without rings
- **Sample Models:** Security changes have been made in Windows 7. Therefore, if MAESTRO is installed on a Windows 7 OS, the *Models and Samples* directory will be located in C:\ProgramData\MAESTRO directory.
- **Ply-Polygon File Format Import:** MAESTRO now has the ability to import ply-polygon mesh geometry files (*.ply). This feature will create a new module of the quad and triangle elements in the mesh, all with the same plate and material property.

Bug Fixes: A number of minor bugs have been fixed.

Known Issues:

- **Cutting Planes:** Cutting planes deleted when user chooses View/Refresh.
- **Quick Create:** The creation of additional nodes via *Quick Create* fails on the first attempt in some cases.
- **Color Legend:** In some plotting scenarios, the titles and/or values are cutoff. The user can use the View/Options to change the font if necessary.
- **Default Material:** The default material property is deleted when the user creates a new model (via the File/New menu item) in one running instance of MAESTRO.
- **View Refresh:** In plotting different views, MAESTRO will *hang* on the current view. In most cases this can be resolved by the user choosing *View/Element Type* followed by *View/Refresh*.
- **Cylindrical Coordinates:** The cylindrical coordinates were intended for purposes of

inputting the data but not modifying existing Endpoints/Additional Nodes.

© 2014 Optimum Structural Design, Inc.

Version 9.0.8

New and Enhanced Features:

- **Sparse Out-Of-Core (OOC) Solver:** Added Sparse Out-Of-Core (OOC) solver that can solve very large models. The limitation of the solver is based on the computer's hard disk space. Models up to 700,000 degrees of freedom have been tested.
- **Inertia Balance:** Added automated inertia balancing for models having hydrostatic loading. Previously, automated inertia balance was only available for models not having hydrostatic loading. The details of the load balance methodology are available under Load Balance in the Verification and Validation section of the manual.
- **Longitudinal Bending Moments:** Axial forces for longitudinal bending moment calculations are now included. Previously, the bending moment distribution was based on the classical beam theory, where the contributions from axial forces were ignored. In some cases, this resulted in bending moment distributions that did not close at the ends. This only affects the bending moment distribution plot and does not change the model's loads; therefore, this will not change previous model response results. The details of the longitudinal bending moment calculations are available under Load Balance in the Verification and Validation section of the manual.
- **Bending Moment/Shear Plots:** Bending moment and shear force are now plotted at the base line. Previously, these plots were located at the waterline.
- **Load Balancing Verification Documentation:** An extensive explanation and verification of the load balancing methodology is presented in the Help file.

Bug Fixes:

- **Legacy Solver:** Fixed a few synchronization bugs for the legacy solver.
- **Load Creation:** Fixed a bug related to the numbering of load cases. If the user only had two load cases, ID 2 and ID 3, but missing ID 1, a third load case could not be created unless ID 3 was renumber to 1 or 4.
- **Fine Mesh Models and Corrosion:** Fixed a bug when solving *un-corroded* fine mesh models of a *corroded* global model.
- **ALPS/HULL:** Fixed a bug that did not allow the user to execute an ALPS/HULL analysis a second time in a given session.
- **RSpline:** Fixed an RSpline display bug that caused computer crashing when creating/deleting RSplines

Graphical Bugs Fixed:

- **Stress Reporting:** A bug when reporting stress for transversely stiffened elements.
- **Group View Deformation:** Deformation plots when viewing groups were not correct.
- **Adequacy Plots:** Fixed a bug when displaying adequacy parameters of irregular evaluation patches.

© 2014 Optimum Structural Design, Inc.

Version 9.0.7

New and Enhanced Features:

- **Layout Organization:** Many of the menu items and icons have been reorganized, renamed or changed to provide the user with a more consistent and effortless experience. Please see the documentation for a complete description of menus and icons.
- **RBE3:** A new element, RBE3 has been added to MAESTRO. The intended use of the RBE3 is to transmit forces and moments from a primary node to one or more secondary nodes without adding any stiffness to the structure.
- **Transverse Bending Moments:** A newly designed Hull Menu includes options to view transverse plots of bending moments. This is especially useful when designing multi-hulls.
- **Transparency:** This new feature allows the user to choose to set a module or substructure as transparent for easier viewing of the model.
- **Groups Creation/Operation:** New functionality has been added to allow a user to create a new group from existing groups using the Group Operation dialog. Also, a new feature has been added within the groups dialog to add additional elements that are similar to an already selected element. Combined, these features provide the user with a more robust way to create groups.
- **Wetted Group:** To assist in exporting “wetted” elements to FEMAP, new functionality in the Groups menu was added that will automatically create a general group of "wetted" elements.
- **Select by Box:** To expedite the selection of elements, a select-by-box capability has been added.
- **Directional Stress:** A new feature has been added which allows the user to align all element local stress vectors to a global direction. This functionality will enable the presentation of stress in a uniform manner to the analyst can better assess the stress patch in a given direction.
- **Export Stiffness Matrix:** MAESTRO can now export the stiffness matrix in Nastran format through the export Nastran options. This feature will allow advanced users to process analysis runs in Nastran by using the MAESTRO generated stiffness matrix.
- **Batch Execution:** MAESTRO has the ability to perform batch processing, which allows the user to sequentially solve any number of models, each of which may have any number of load cases without having to manually launch each model.
- **AutoSave**: A new auto save feature has been added to MAESTRO. The user can select the number of executed commands in between auto saves from File > AutoSave Frequency...
- **Recover Model:** A new feature has been added under File > Recover, which automatically recovers a model after an unexpected closing of MAESTRO.
- **Documentation:** The documentation has been completely overhauled with a focus more on “how to.” Additionally, all of the documentation is now in a single location, which can be found via the Help menu or help icon.
- **Sentinel Security Drivers/Servers:** The Sentinel System Driver (both for USB) and Sentinel Protection Server (for Network locks) are now integrated into the MAESTRO installation process. Therefore, there is no need for the user to install security

© 2014 Optimum Structural Design, Inc.

drivers/servers separately. The integrated installation checks to determine the necessity of installing the Sentinel Driver/Sentinel Protection Server.

- **Help Menu:** The new Help menu now includes links to the website, forum, support email, and FAQ.
- **FAQ:** Updated FAQ section in the help file and on the website.
- **"Null" Beam and Stiffener Properties:** There are now predefined "null" beam and stiffener layouts when creating a new model. This eliminates the need to define these for an unstiffened panel or unframed strake.
- **Set Current & View Part:** There is now an icon and a right-click menu option in the parts tree to set the current and view part for a module or substructure at the same time. There is also a new icon to set current part and set view part .
- **Background Color:** The new MAESTRO default background color is light gray with gradient north. To change back to a black background, or another color, open the Preferences dialog from File > Preferences.
- **Capture View:** A new feature has been added under the File menu which allows the user to copy a screenshot of the modeling space to their clipboard which can then be pasted into another program.
- **Importing Legacy Results:** A new mechanism has been added to facilitate the importing, i.e. opening *.PLG files, of legacy MAESTRO results.
- **Hull Menu:** A newly designed Hull Menu with options to view horizontal and transverse plots.
- **Solving Fine Mesh Models:** Multiple top down Fine Mesh models can be solved at once from the File > Analysis/Evaluation menu or Analysis/Evaluation dialog.
- **Groups:** A new feature has been added within the groups dialog to add additional elements that are similar to an already selected element.
- **Check & Merge Dialog:** The functionality found in the Check & Merge Dialog (namely Update: FE Tags, Free Edges and Evaluation Patches) has been moved to the Tools > Renumber FE-Tag, View > Edges > Free Edges and Model > Evaluation Patch > Auto-Generate menu items respectively.
- **Model Summary:** Functionality used to report number of elements and nodes has been moved to the Model > Summary menu item.

Bug Fixes:

- **Update FE Tags:** Fixed a bug so that FE Tags are updated when a module or substructure is moved or rotated.
- **Collapsed Quad:** Fixed a bug so that a quad collapsed to a triangular element correctly distributes mass and force to the appropriate nodes.
- **Sort Volume Table:** Fixed a bug so the Volume Table can now be sorted by the Volume Group Name.
- **Deletions Dialog:** Fixed a bug that would cause a hard crash when deleting large amounts of elements from the deletions dialog.
- **Sort Element List by ID:** Fixed a bug that allows elements to be sorted in the Grid tab without information disappearing.
- **Sort Nodes List by ID:** Fixed a bug that caused a hard crash when trying to sorted listed nodes by ID in Grid tab.
- **Warped Quad:** Fixed a bug to take the average face normal of the two triangles making

up a warped quad for area, weight, volume, etc. calculations.

- **Trochoidal Wave:** Fixed a bug so that the trochoidal wave profile definition now uses the formulas provided in DDS-100-6 i.2.
- **Parts Tree in Parts Dialog:** Fixed a bug so that the parts tree within the Modules and Substructures dialog is consistent with the main parts tree.
- **Brackets:** Fixed a bug so that leg lengths are automatically populated for the listed property, and are returned to the automatic lengths if "user defined" is unchecked.
- **Find FE-Tag:** Fixed a bug so a search can now be performed for a nodal or elemental FE-Tag.
- **Deletions Filters:** Fixed a bug so that the filters in the deletions dialog work as expected.
- **Saving/Loading Results:** Fixed a bug so results consistently save and load with a .mdl file.
- **ULSAP Results:** Fixed a bug so that ULSAP results are automatically saved and loaded with a .mdl file.
- **ULSAP Adequacy Parameters:** Fixed a bug so only ULSAP Adequacy parameters are shown in an ULSAP analysis.

Graphical Bugs Fixed:

- **"Show This Thickness Only":** Fixed a bug that allows user to switch back to all "thicknesses" using the dynamic query once the "Show this thickness only" option is selected.
- **Corner Stress:** Fixed a bug so now the check box for "Corner Stress" changes the stress contour plot between nodal and elemental contour plots.
- **Compound Element Effectiveness:** Fixed a bug so now non-transverse compound elements can be defined as longitudinally effective using the dynamic query.
- **Bending Moment when Switching Load Cases:** Fixed a bug so the hull bending moment plot updates when different load cases are selected.
- **Load Case Selection:** Fixed a bug to make sure queried loads are consistent with the load case from the load case selection drop-down menu.
- **Pressure Plots when Switching Load Cases:** Fixed a bug so the color plot and legend updates each time a different load case is selected from the load case drop-down menu.
- **Deformed Model Changes:** Fixed a bug that prevented model from being undeformed if structure was changed in the "deformed" mode.
- **Contour Stress Plot:** Fixed a bug so that the stress legend updates correctly between load cases when viewing a contoured stress plot.
- **Hide Elements Outside Range:** Fixed a bug that now has Hide Elements Outside Range icon functionally properly. This option will now hide elements that are not within the range of displayed values.
- **Stiffeners on Wetted Elements View:** Fixed a bug so that when stiffeners are turned on, they remain in view on the wetted elements when selecting View > Wetted Elements.
- **Mirrored Wetted Elements:** Fixed a bug so the wetted faces of wetted elements in a mirrored module or substructure are shown correct graphically.
- **Adequacy "User Defined" Range Plot:** Fixed a bug so results update correctly when switching between load cases.

© 2014 Optimum Structural Design, Inc.

Introduction

3

3 Introduction

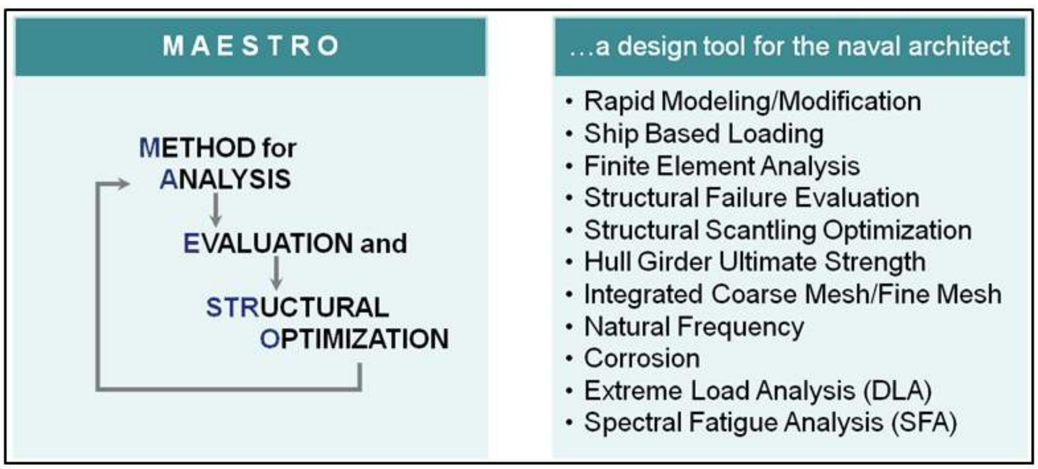
The topics in this section provide some basic information about MAESTRO, what it is for, and what you can do with it.

3.1 About MAESTRO

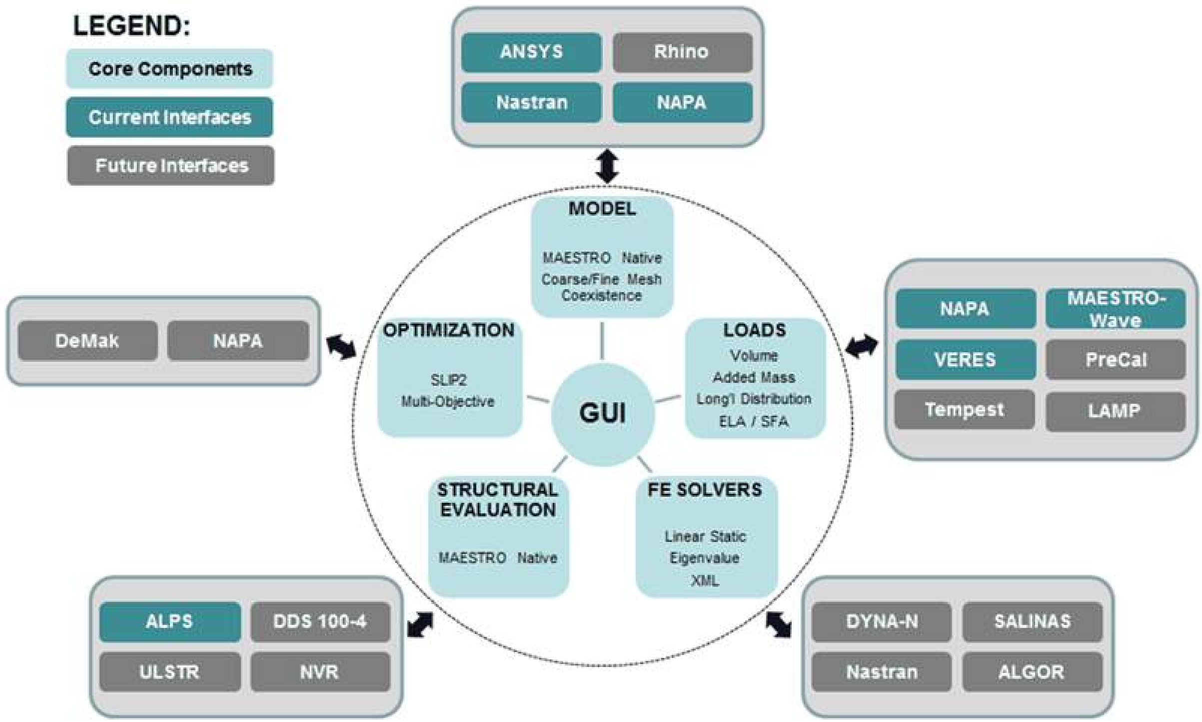
MAESTRO is a design tool specifically tailored to suit naval architects and their finite element analysis and limit-state (failure mode) evaluation needs. The objective of any structural finite element analysis is to accurately determine the response of a structural system that is modeled with finite elements and subjected to given loads. MAESTRO accomplishes this objective through a single Windows-based graphical user interface that completely encompasses the structural modeling (preprocessing), the ship-based loading, the finite element analysis, the limit-state evaluation, and the post-processing.

3.1.1 The Software System

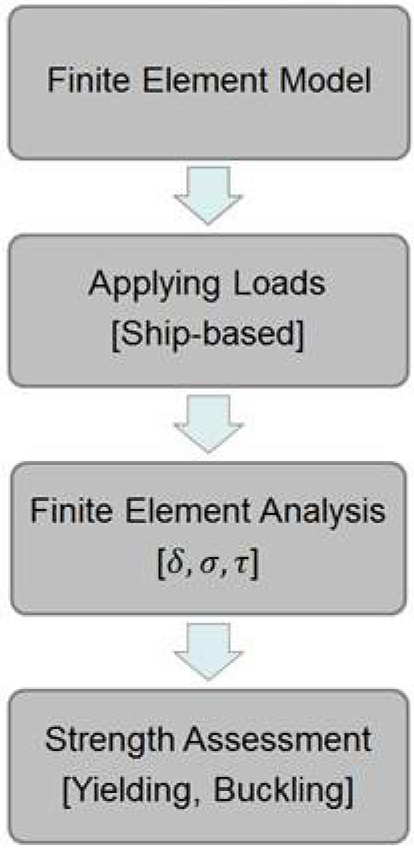
MAESTRO (**M**ethod for **A**nalysis, **E**valuation, and **STR**uctural **O**ptimization) is a design, analysis, and evaluation tool specifically tailored for floating structures and has been fielded as a commercial product for over 20 years and has a world-wide user base. MAESTRO's history is rooted 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 (shown below).



The core capability of MAESTRO is part of a larger *open* product model framework as shown below. This open framework allows the interfacing (i.e., data input and data output) of MAESTRO with a variety of ship structural design and life-cycle assessment technologies for several technical domains including, but not limited to: Structural Life-cycle Assessment, Underwater Shock Assessment, Fatigue Analysis and Assessment, Hydrodynamic Loads Analysis, Ultimate Hull Girder Strength Assessment, and Ship Salvage Assessment. The implementation of the NVR Strength Criteria (i.e., the objective of this panel project) is one example of a particular US Navy toolset or methodology that can be interfaced with this open product model framework.



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



The MAESTRO system is composed of many *modules* that address particular aspects of the ship structural design process.

They are described in the following sections:

Modeling/Analysis/Evaluation

Finemesh Analysis

Natural Frequency

ALPS/ULSAP

ALPS/HULL

Naval Vessel Rules (NVR) Assessment

Nastran Translator

ANSYS Translator

NAPA/MAESTRO Interface

MAESTRO-Wave

Extreme Load Analysis (ELA)

© 2014 Optimum Structural Design, Inc.

Spectral Fatigue Analysis (SFA)

Optimization

3.1.1.1 Modeling Analysis & Evaluation

This module includes the graphical modeler for developing MAESTRO structural models and full post-processing capability. This includes full model viewing capability and generation of loads for input to the MAESTRO Analysis/Evaluation Solver. Post-processing includes such features as graphical display of loads, deflections, stresses, and failure evaluation results from the MAESTRO Analysis/Solver.

MAESTRO's two central operations, analysis and evaluation, are also performed by this module. This module completes a finite element analysis and structural integrity evaluation (failure modes and other limit states) of every member and every load case.

The solver performs three integrated tasks:

- finite element analysis to obtain the actual stresses and deflections throughout the model for all load cases.
- calculation of the failure stresses for all relevant modes of failure, for every member and every load case.
- a complete evaluation of the structural adequacy of every member under each load case, and thereby obtaining a rigorous assessment of the current design of the structure, which includes identifying the most critical failure mode and load case for each member.

3.1.1.2 Finemesh Analysis

The MAESTRO Fine Mesh module allows the user to create refined 3-D FEA models of any portion of the MAESTRO model quickly. The user creates a group made up of elements from the "area of interest" then refines the group based on two options. The first option is called the Top-down mode. In this mode, Fine Mesh Analysis applies displacements from the global analysis to the fine mesh model for boundary conditions. In the Embedded mode (the second option), Fine Mesh Analysis replaces the coarse mesh portion of the model with the fine mesh.

3.1.1.3 Natural Frequency

MAESTRO can compute the natural frequency of the ship in either air or water. When the analysis is done in water, the added mass of the water is automatically applied to the wetted elements.

3.1.1.4 ALPS ULSAP

The features of ALPS/ULSAP include:

- Ultimate limit state evaluation of unstiffened uni-axially stiffened panels and cross-stiffened panels.

© 2014 Optimum Structural Design, Inc.

- Any combination of load components, namely longitudinal axial compression or tension, transverse axial compression or tension, edge shear, longitudinal in-plane bending, transverse-in-plane bending-and lateral pressure can be applied.
- Either steel or aluminum alloy material can be dealt with, considering the softening effect in heat affected zone caused by welding.
- Initial imperfections in form of initial deflections and welding residual stresses are dealt with as parameters of influence.
- Various types of structural degradation such as corrosion wastage (general or pitting), fatigue cracking-and local denting are dealt with as parameters of influence.
- Impact pressure action arising from sloshing, slamming and green water can be analyzed for providing permanent set in terms of panel deflection.

3.1.1.5 ALPS HULL

The features of ALPS/HULL include:

- Progressive collapse analysis of ship hulls until and after the ultimate limit state is reached, using simplified nonlinear finite element method (idealized structural unit method).
- Any combination of hull girder load components, namely vertical bending, horizontal bending, sectional shear and torsion can be applied.
- Either steel or aluminum alloy ship hulls can be dealt with, considering the softening effect in heat affected zone caused by welding.
- Initial imperfections in form of initial deflections and welding residual stresses are dealt with as parameters of influence.
- Various types of structural degradation, e.g., corrosion wastage (general or pitting), fatigue cracking-and local denting are dealt with as parameters of influence.

3.1.1.6 Naval Vessel Rules Assessment

The NVR Assessment module provides structural assessment based on the direct analysis methodologies described in the ABS Rules for Building and Classing Naval Vessels. The NVRs are governed by a “DISTRIBUTION STATEMENT D” statement, which authorizes only U.S. Department of Defense (DoD) and U.S. DoD contractors the ability to gain access to the NVRs and their implementation into MAESTRO. Please contact support@maestromarine.com for guidance regarding proper DoD authorization.

3.1.1.7 Nastran Translator

This module translates either the entire MAESTRO finite element model or a portion thereof into Nastran, including loads. Alternatively, Nastran models can be translated into MAESTRO (only Nastran geometry).

© 2014 Optimum Structural Design, Inc.

3.1.1.8 ANSYS Translator

This module automatically translates either the entire MAESTRO finite element model or a portion thereof into ANSYS.

3.1.1.9 NAPA MAESTRO Interface

The NAPA/MAESTRO Interface (NMI) module allows the designer to import a NAPA/NAPA Steel 3D model, which includes the finite element model (geometry, scantling properties, and finite elements), loading information (longitudinal weight and bending moment distributions, tank boundary/content/fill definitions, and hydrostatic equilibrium definition), and model hierarchy definitions such as MAESTRO module group. This interface enables shorter design cycle times by using a single 3D structural design model (from NAPA) that can be re-used as the design matures and is ready for analysis in a matter of minutes.

3.1.1.10 MAESTRO-Wave

The MAESTRO-Wave module provides the ship designer with an integrated frequency-domain and time-domain computational tool to predict the motions and wave loads of any vessel. To compute these hydrodynamic motions and loads, MAESTRO-Wave first calculates the velocity potentials, source strengths, and flow velocities at the centroids of the hydrodynamic panels for each speed, heading, and frequency requested, and then maps the source strength to the structural panels. The equations of motion are formulated using the structural mesh rather than the hydrodynamic mesh. This approach results in a perfect equilibrium for the structural model. Bending moments, shear forces and torsional moments are automatically in closure. No inertia relief and artificial loads are needed to balance the model. The computation of these hydrodynamic forces is based on 3D potential theory using the zero speed Green’s function with a speed correction parameter. A variety of visualizations and output data are available for purposes of post-processing.

3.1.1.11 Extreme Load Analysis

The Extreme Load Analysis (ELA) module imports unit wave load data from a hydrodynamic analysis (MAESTRO-Wave or other) via *.smn file format. It then allows the user to calculate hull girder load response RAOs, and provides the necessary short-term and long-term statistical computations to predict extreme values of the maximum loads for a given vessel. This includes the ability to define or import wave scatter diagrams, operational profiles, and wave spectra as well as compute hull girder RAOs for user-defined dominant load parameters (e.g., vertical bending moment). Finally, extreme equivalent regular waves (equivalent design waves) are internally computed and selected for assessment of extreme global loads. The user has a variety of options to add still water loads to the wave-induced wave loads and re-balance these components.

3.1.1.12 Spectral Fatigue Analysis

The Spectral Fatigue Analysis (SFA) module provides the ability to perform global fatigue screening of the vessel. The SFA module introduces additional functionality to the ELA module to compute Stress RAOs, displacement RAOs, define and associate structural groups to SN curves and Stress Concentration Factors (SCFs), and compute fatigue damage based on the Miner cumulative damage principle.

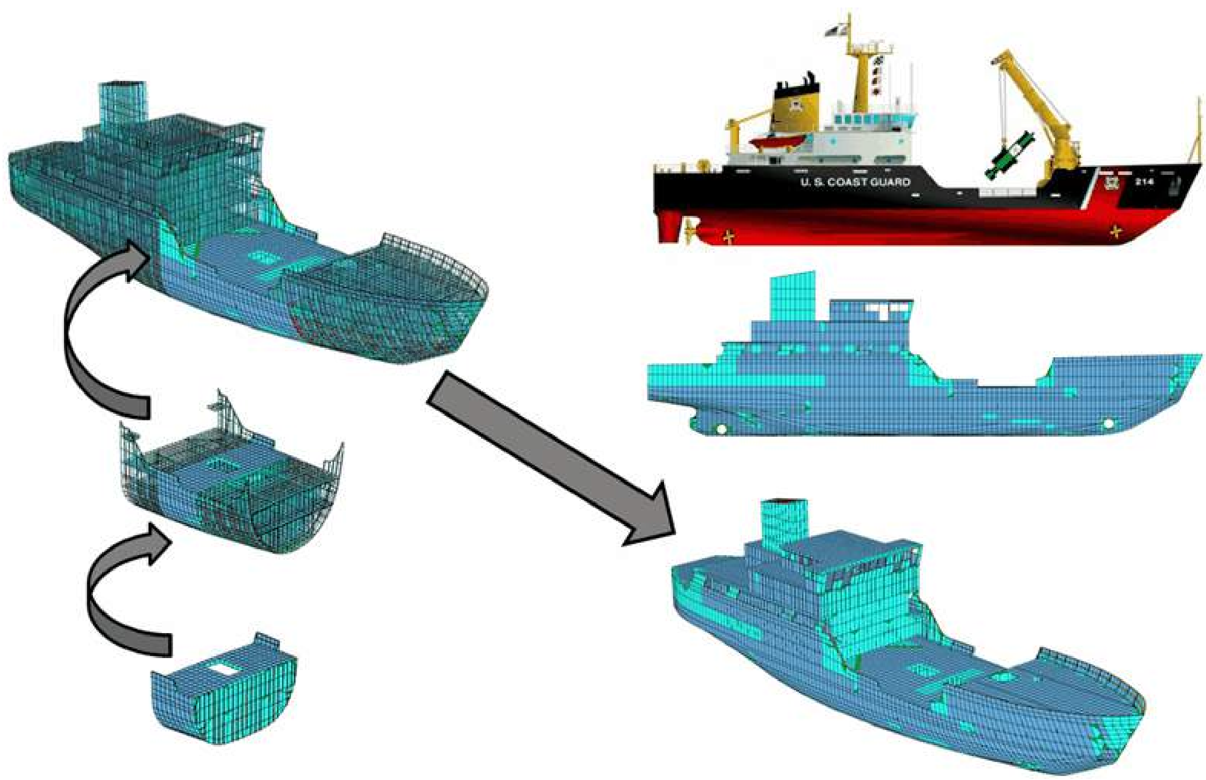
© 2014 Optimum Structural Design, Inc.

3.1.1.13 Optimization

The MAESTRO Optimization module uses sequential linear programming to redesign the structure. This optimization eliminates any structural inadequacies while achieving an optimum design based on user specified objectives (goals), which may be least weight or least cost or even both of these, in a weighted non-dimensional combination. In its optimization mode MAESTRO iterates the structure through design cycles in which it revises scantlings, reruns the finite element analysis, and reevaluates the structural adequacy of each member for all failure modes and load cases. This iterative process continues until the structure has converged to an optimum design that has no structural inadequacies.

3.1.2 Finite Element Modeling

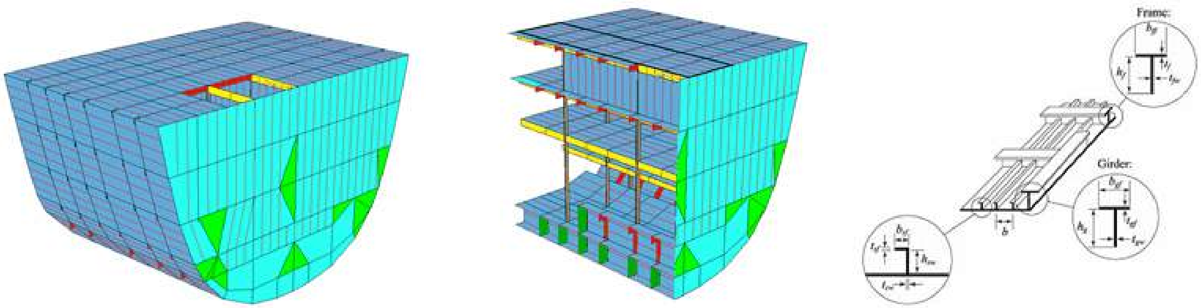
The first step in the FEA process is creating the model, which consist of nodes and finite elements. In the design of large structures such as ships, it is usually advisable to divide the task into a few distinct subtasks in order to maintain a good overview and control of the model generation. Most large structures can be reduced to several levels of component structures for which the design and analysis is relatively independent. Such a structure can best be modeled by subdividing it into a hierarchy of parts, down to the module level, and then constructing each module using a three-dimensional mesh of nodes and appropriate groupings of finite elements. As shown in the figure below, the MAESTRO structural modeling is organized in two levels: modules and substructures.



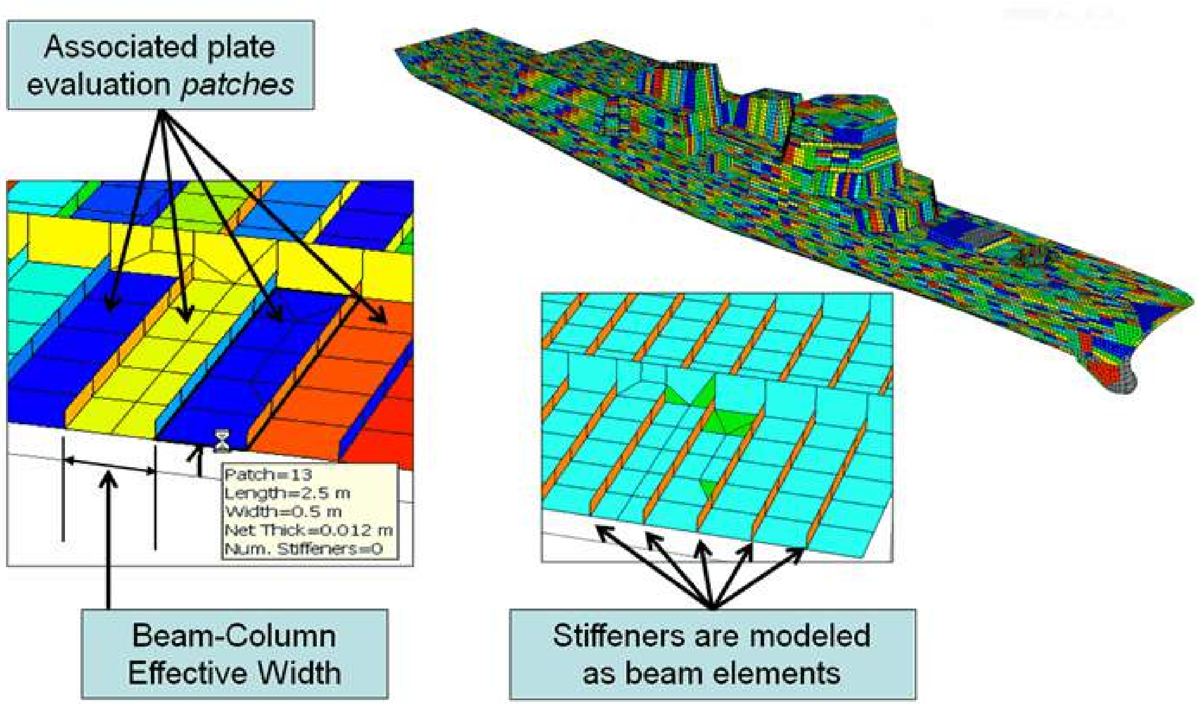
Of particular interest in this model hierarchy, is the composition of the module. The module

© 2014 Optimum Structural Design, Inc.

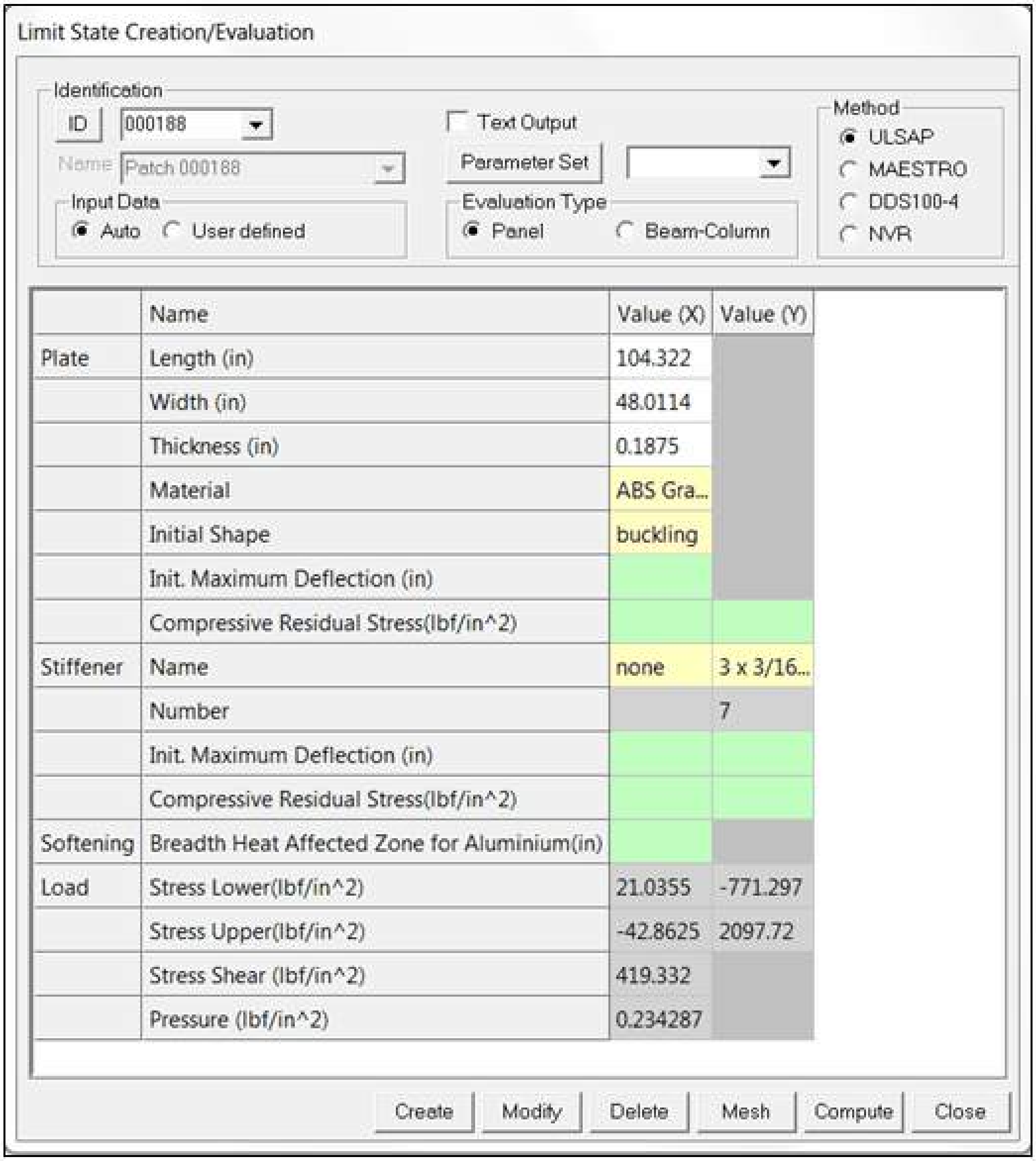
typically consists of a distinct segment of the ship (e.g., cargo holds in commercial ships or watertight subdivisions in naval ships). The definition of the module is helpful for this because they are ideal high-level building blocks and, in the parallel midbody of a tanker, bulker, or submarine, only one module (i.e., one cargo hold or compartment) needs to be built and then copied. Finally, within this module, the definition of the strake (see the figure below) is used to capture and retain critical information (e.g., breadth between stiffeners, span of longitudinals, frames, or girders) for the process of strength assessment. This concept of a strake, and the recovery/retaining of the strength assessment parameters are further extended to the concept of a MAESTRO Evaluation Patch.



The Evaluation Patch (E-patch) is a collection of elements with its boundary supported by bulkheads or beams. An E-patch can also be a single element. In traditional MAESTRO, an E-patch was a lengthwise strake panel or a few strake panels, if the section per bay was defined as greater than one. The evolution of the E-patch in MAESTRO eliminated this strict definition, freeing the analyst to create more refined FEMs, as sometimes is required. These E-patches are defined automatically (see the quilted colors in the figure below; each color represents an automatically generated E-patch) by MAESTRO or can be defined manually by the user.



Each E-patch can be subsequently reviewed for the recovered strength assessment parameters (see the figure below). Later sections will discuss how stresses are utilized in this E-patch paradigm.



3.1.3 Applying Loads (Ship-based)

The next step in the FEA process is the application of loads. Loads are a prerequisite to structural response analysis and design. 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., Basic Loads-Live Loads/Dead Loads/Liquid Tank Loads, Sea Environment-Hull Girder Loads/Sea Loads/Ship Motion Loads, Operational Loads-Slamming/Flooding/Docking, and Combat Environment-Shock/Air blast/Weapons Effects). Although MAESTRO has the capability to accurately model these different load patterns, it should be noted that the analyst must use sound engineering judgment to determine what loads (e.g., "rule based" versus "first-principle based") can be combined in a particular load case. Rule based loads must be distinguished as such and not combined

© 2014 Optimum Structural Design, Inc.

with first-principle loads. Loads that are not generated within (e.g., computed hydrodynamic loads) MAESTRO can be imported and used for analyzing extreme global load structural responses.

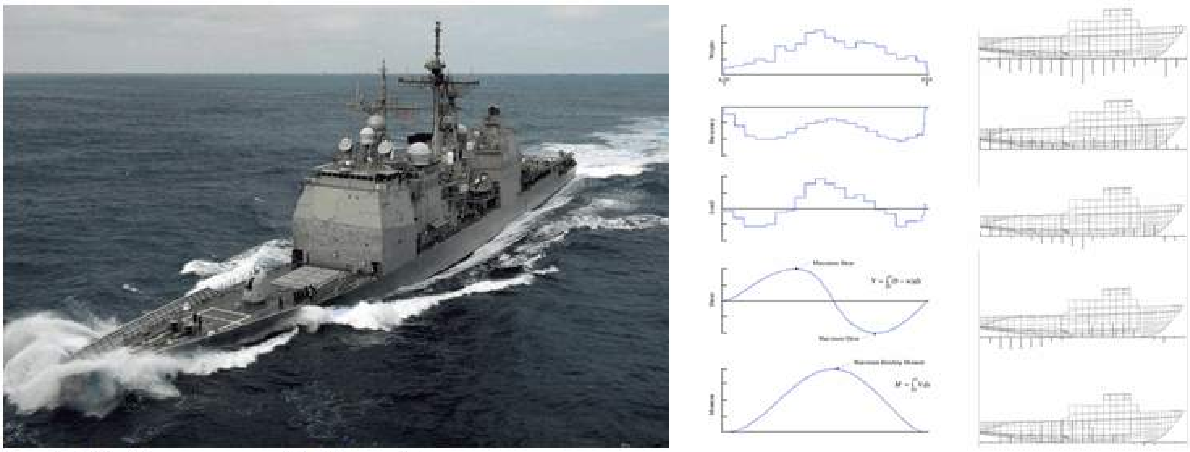
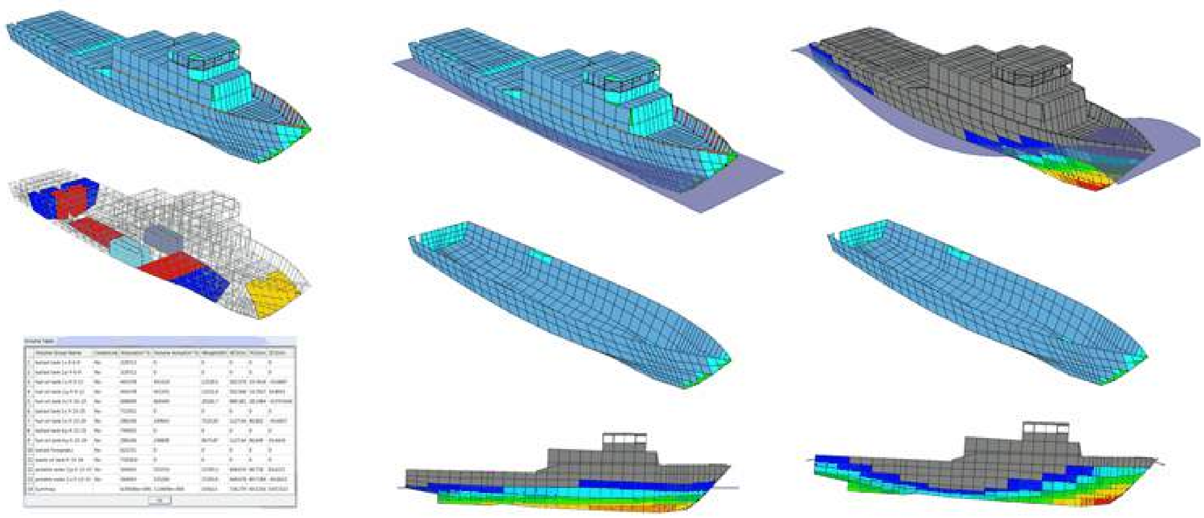


Photo Citation: *www.globalsecurity.org*



3.1.4 Checking Your Model

At each stage of the FEA process, you receive graphical verification of your progress. MAESTRO provides tools for checking your model before and after analysis, which gives you confidence that you have correctly modeled the structure.

MAESTRO makes extensive use of dynamic querying, which allows the user to make changes graphically while checking the model. Dynamic query is a powerful tool that is used in all phases of the FEA process. The dynamic query functionality is intended to be used in conjunction with particular MAESTRO menus, e.g. the *View*, *Loads*, *Hull*, and *Results* menus. In combination with these particular menus, the user can make changes ("on the fly") while modeling (i.e., thickness, material, pressure sides, etc.).

In addition to graphically presenting model properties such as element thickness (plate, beam web and beam flange), positive pressure sides and beam properties (to name a few), MAESTRO provides built in integrity checks for element aspect ratios, disconnected elements, warped (twisted) quads, element internal angles and overlapped elements. These integrity checks can be executed at any stage of the process.

3.1.5 Analyzing Your Model

When the modeling stage is complete, MAESTRO provides interfaces to solvers to perform both finite element analysis and structural evaluation. Regarding structural evaluation, MAESTRO provides three failure mode strength assessments, which are discussed briefly.

MAESTRO Failure Evaluation

The first is the MAESTRO failure evaluation, which is based on the theory presented in Ship Structural Design (Hughes). The following are the modes of failure examined by the MAESTRO evaluation method:

Panels

- Panel Collapse (stiffener flexure, combined buckling, membrane yield and stiffener bucking)
- Panel Yield (tension/compression in the stiffener flange and tension/compression in the plate)
- Panel Serviceability due to local plate bending longitudinal/transverse
- Panel Failure due to local plate buckling;

Girders

- Girder Collapse (tripping and tension/compression in flange)
- Girder Yield (tension/compression in the plate/flange)

Frames

- Frame Collapse (plastic hinge)
- Frame Yield (tension/compression in the plate/flange)

ALPS/ULSAP Failure Evaluation

The second is the ALPS/ULSAP failure evaluation, which is based heavily on the theory presented in Ultimate Limit State Design of Steel-Plated Structures (Paik and Thayamballi). The following are the modes of failure examined by the ALPS/ULSAP evaluation method:

© 2014 Optimum Structural Design, Inc.

- Overall collapse of plating and stiffeners as a unit
- Collapse under predominantly biaxial compression
- Beam-column type collapse
- Local buckling of stiffener web
- Tripping of stiffener
- Gross

ALPS/HULL Strength Assessment

Finally, the progressive collapse analysis of ships under hull girder loads can be assessed in MAESTRO, using the optional ALPS/HULL module. The theory behind ALPS/HULL is presented in Ultimate Limit State Design of Steel-Plated Structures (Paik and Thayamballi).

3.1.6 Post-processing

After the analysis (structural response and/or structural evaluation), MAESTRO provides visualization features that enable you to quickly interpret the results, verify loading conditions, and record results. Deformation plots, stress plots, contour plots, and animations of each loading condition are available in MAESTRO.

As previously mentioned, MAESTRO makes extensive use of dynamic querying not only in the model stage of the FEA process, but the post-processing stage as well. For example, while the user is graphically post-processing with dynamic query turned on, a pop-up window displays the exact information that is being used to create the graphics (i.e., stresses, deformations, structural adequacy). Also, this information can be echoed, in text format, to the output window located at the bottom of the GUI.

3.2 Getting Help

Using this help file:

This help is designed to be used on-screen. It is extensively cross-linked so that you can find more relevant information to any subject from any location. If you prefer reading printed manuals a PDF version of the entire help is installed in the \Help subdirectory, located in the directory where you installed MAESTRO (by default, ...\MAESTRO\Help). This may be useful as a reference but you will probably find that the active hyperlinks, cross-references and active index make the on-screen electronic version of the help much more useful.

Getting Started

Start by studying the About MAESTRO and General sections.

Using the help while you're working

As far as possible the help is separated by the basic steps of finite element

© 2014 Optimum Structural Design, Inc.

modeling and analysis. This makes easier to find the answer to your question based on your stage of the finite element analysis process.

- **To find information on a specific topic, navigate to the appropriate section:**

  General

  Geometry/Finite Element Modeling

  Checking The Model

  Importing and Exporting

  Loading The Model

  Analyzing and Post-Processing

  Advanced Processing and Programming

  Extreme Load Analysis

  NAPA-MAESTRO Interface

  Verification and Validation

- **When you're frustrated,** use the Index and Search functions as well as check out the Frequently Asked Questions section.

Tutorials:

- See the Tutorials section for some basic tutorials to get you started with using MAESTRO.
- You will find some of the models used in the tutorials in the MAESTRO/Models & Samples/Tutorial Models folder.
- Full ship sample models can be found in MAESTRO/Models & Samples/Full Ship Models folder.

Getting a printed user manual:

Please don't try to print the HTML Help version of the help from the Microsoft help viewer; it would look terrible. You will find a formatted PDF version of the entire documentation designed for printing in the MAESTRO\Help folder or from Start > All Programs > MAESTRO

Email Support:

support@maestromarine.com

Telephone Support:

(410) 604-8000

Website:

www.maestromarine.com

Forum:

MAESTRO Forum

Other MAESTRO Representatives:

In addition to the options above, you may also contact a local representative from the list below for MAESTRO technical support:

Europe/Middle East/Russia

Design Systems & Technologies

Phone: +33-4-92-91-13-24

Fax: +33-4-92-91-13-38

ds-t@ds-t.com

3.3 How to buy MAESTRO

MAESTRO can be purchased by visiting the online purchasing page and filling out the required information and submitting the form. Once the form is received, you will be contacted by a MAESTRO sales representative and a quote will be generated for the requested modules or maintenance and support extension. You may also contact the DRS AMTC directly:

160 Sallitt Drive, Suite 200

Stevensville, MD 21666

Phone:	410-604-8000
Fax:	410-643-5370

© 2014 Optimum Structural Design, Inc.

Email: support@maestromarine.com

3.4 Software Maintenance and Support

STANDARD MAINTENANCE SERVICES

Scope of Service: During the Period of Performance, M&S services for MAESTRO ("the Software") include:

- Periodic updates of the Software that may incorporate (i) correction of any substantial defects, (ii) fixes of any minor bugs, and (iii) at the sole discretion of AMTC, enhancements to the Software. These updates will be available electronically or via CD.

- Ten (10) hours technical support consisting of telephone, fax, and electronic mail consultation, during AMTC normal business hours, and frequently asked questions (FAQ) through the MAESTRO website, to assist Customer in the operation and application of the Software. Additional customized technical support during the Period of Performance will be billed to Customer at the AMTC's current billing rates. This ten hour limit does not apply to technical support related to software bug fixes. This allotted time amount of customized technical support will expire at the end of the Period of Performance.

- Maintenance at AMTC offices of a test version, for the three most recent versions of the Software.

Services Not Included: M&S services do not include:

- Charged-for-Enhancements that are offered, at AMTC's sole discretion, to Customers upon payment of a license fee;

- Custom programming services;

- On-site support;

- Training; and

- Hardware and related supplies

© 2014 Optimum Structural Design, Inc.

Maintenance and Support (M&S) Period of Performance Expiration and Re-Establishing M&S

- A 12 month M&S fee is required to be purchased at the time of purchasing MAESTRO software. M&S is priced at 20% of the price of the software per year. After 12 months, the Software licensee has the option to purchase an additional 12 months of M&S for the next year. If the M&S agreement has expired, the licensee will no longer receive or have access to the services described in the Annual M&S Scope of Services. To re-establish MAESTRO M&S, the licensee must first pay for any lapsed M&S period, in order to become current. In addition to any lapsed M&S, M&S would also be due for the current year. Lapsed M&S is capped at 2-1/2 years (30 months); therefore, a user whose license is 2-1/2 years or more out of M&S would pay 50% (20% * 2.5 years) of the list price of the software, plus M&S for the current year.

Software License and Terms: The MAESTRO Software License Agreement governs the terms and conditions of use of the Software.

Reprogramming the Hardware Lock: Please execute the following steps:

1. Remove all other hardware locks from your computer. The MAESTRO hardware lock should be the only device attached to your computer for this operation.

2. Select Start > All Programs > MAESTRO x.x.x> MAESTRO Fast Lock (where *x.x.x* is the current version number)

3. Select: Edit/Set Passwords.

4. Enter the new passwords which are shown above. The new Support Date will be shown.

Downloading the Latest Version of the Software: The most recent MAESTRO software can be downloaded from our website, at ***http://www.maestromarine.com***. Please follow the MAESTRO software installation instructions as noted below:

1. Download the self-extracting zip file into a temporary folder or your desktop, using the "MAESTRO Installation File" link.

2. Double-click on the file to run it; this will automatically extract the files and run setup.exe and install the necessary security drives.

3. Follow additional instructions provided on-screen to complete the installation.

Support Contact: Please contact us with any questions or concerns at: ***support@maestromarine.com***.

© 2014 Optimum Structural Design, Inc.

3.5 Getting Started

The current release of MAESTRO can be downloaded by sending an email to sales@maestromarine.com requesting a download link.

Once the download is complete, double-click the self-extracting file to start the installation process.

Follow the on-screen instructions to complete the installation. With MAESTRO version 9.0 and later, the installation will automatically ensure the appropriate drivers and servers (for network licenses) are installed.

Once the MAESTRO installation is complete, verify that the license dongle (USB or parallel port) is attached to the local machine where MAESTRO is installed, or if it is a network license that the dongle is attached to the host computer and the Sentinel Protection Service is started on the host computer. This can be verified by opening the Control Panel, clicking on Administrative Tools, and then Services. If the service is not already started, start the service before launching MAESTRO to ensure the network lock is found.

If you do not have a license or your current license is expired, MAESTRO will run in Demo mode with limited functionality. **If you would like to purchase a license, or renew an existing license, you can go to http://www.maestromarine.com/purchase.php to fill out a form for a quotation or email directly to support@maestromarine.com with your inquiry.**

If you experience any problems during or after the installation, click **Help > Technical Support** from the menu to send an email to support@maestromarine.com.

Note for Network Locks

If you are troubleshooting issues with a network lock, you may refer to the ReadMe.pdf file located in the MAESTRO installation directory under *MAESTRO/System/Sentinel* or see the Security Devices section of the help manual.

© 2014 Optimum Structural Design, Inc.

General

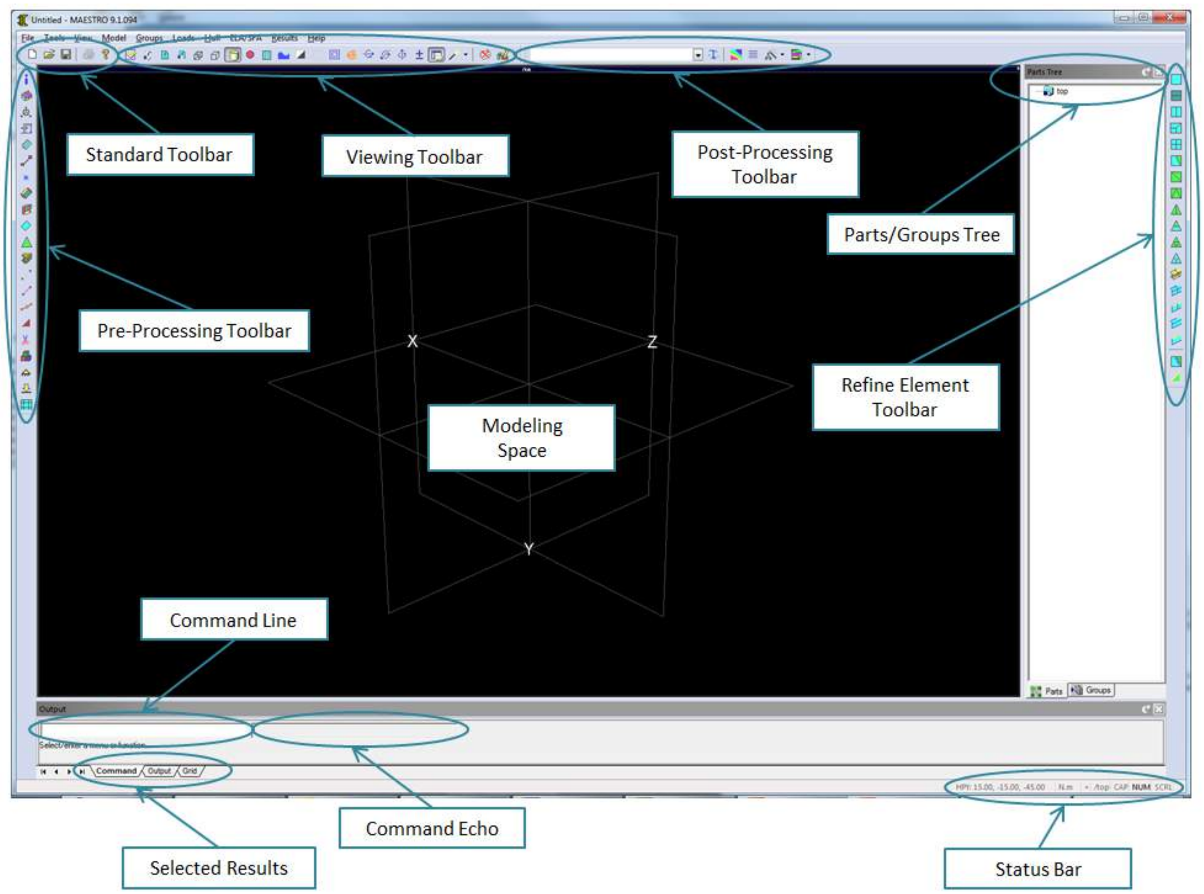
4

4 General

The topics in this section provide an overview of the MAESTRO program user interface.

4.1 Workspace Layout

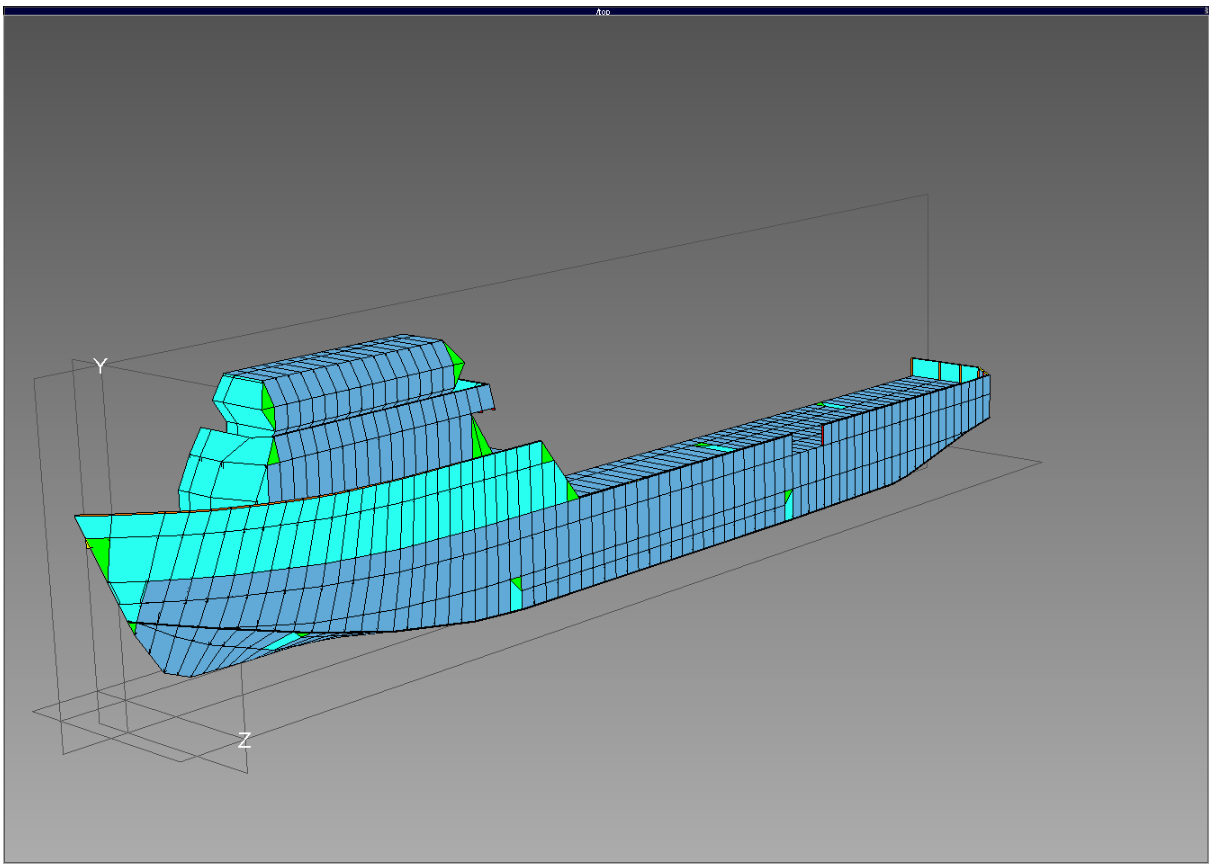
The MAESTRO Modeler is a tool for creating a structural model through a graphical user interface (GUI). With the help of various menu options and toolbars, the user can rapidly generate a large, yet accurate model. The layout of the MAESTRO GUI is shown below.



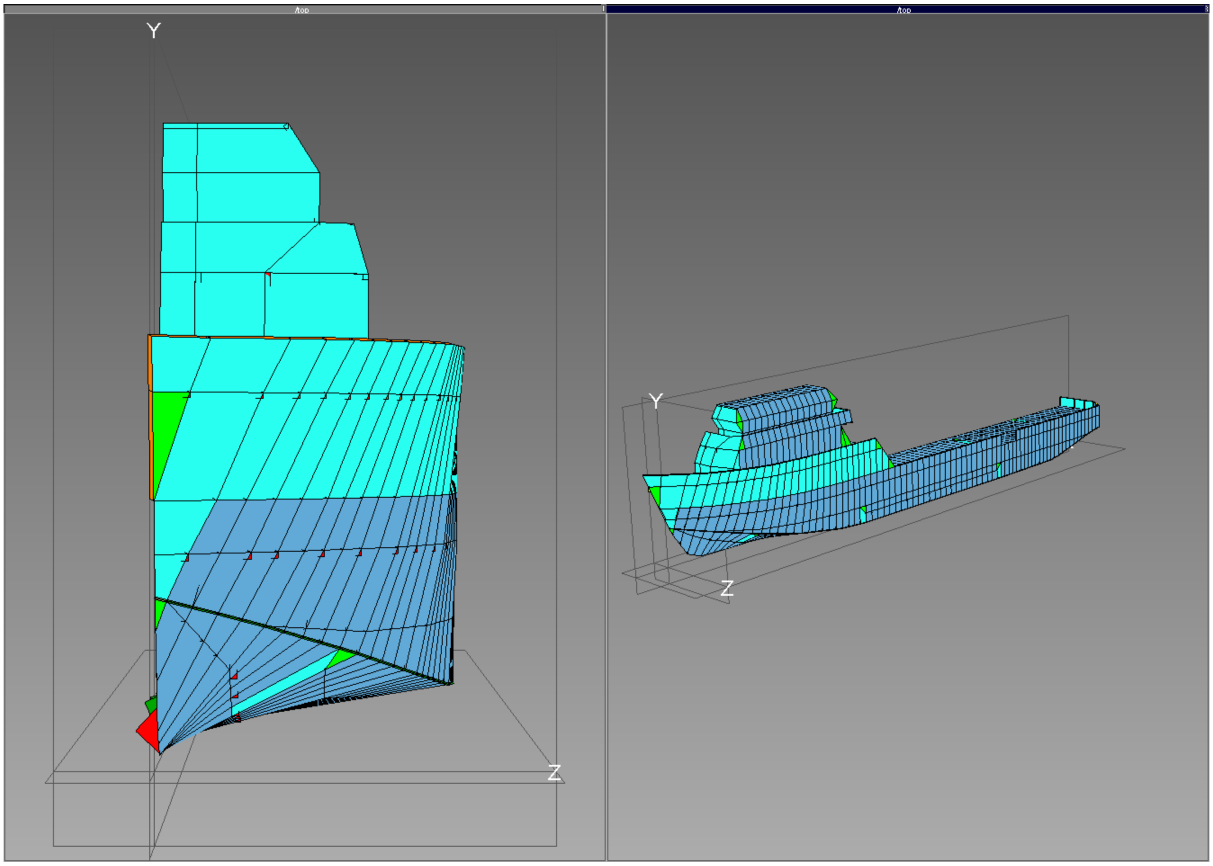
The main display area contains the graphical representation of the model geometry. It is used for viewing either the entire or a portion of the current structural model, and is often used to make interactive selections. The main display can have several layouts including one, two, or four viewports. The number in the upper right corner of each viewport indicates the ID of that viewport, and the current view part is displayed in the window border of each viewport. The active viewport's window border is highlighted in dark blue.

© 2014 Optimum Structural Design, Inc.

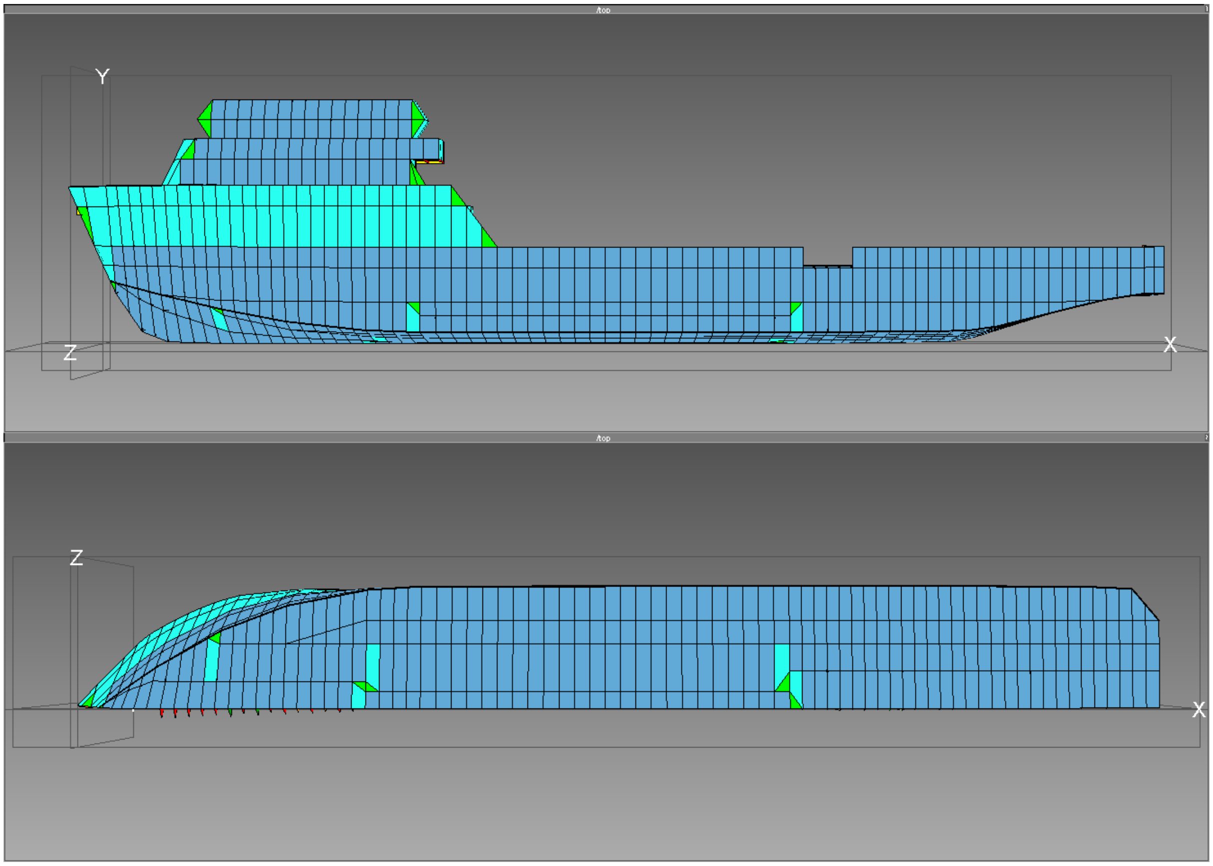
The user can control how and what is seen in the main display through the commands in the View menu. Commands to rotate the current view can also be accessed with a single mouse click via the Viewing toolbar. Several more commonly used viewing commands can be accessed by clicking the right mouse button anywhere in the main display area. This results in a pop-up menu being displayed from which the user can perform operations such as zooming in or out, fitting the view, panning the view, toggling to the previous view, setting any of the five standard views, and toggling perspective projection on and off.



Single View

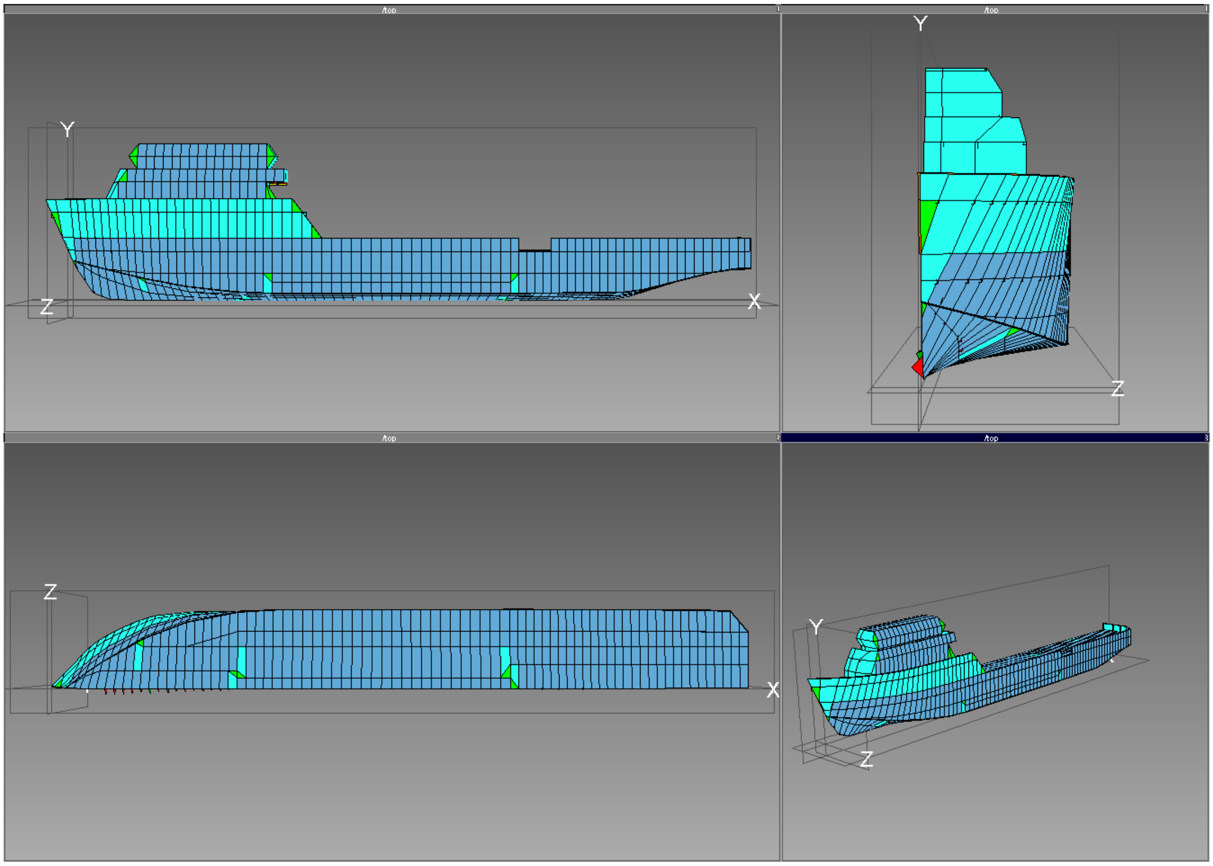


Split Vertical View



Split Horizontal View

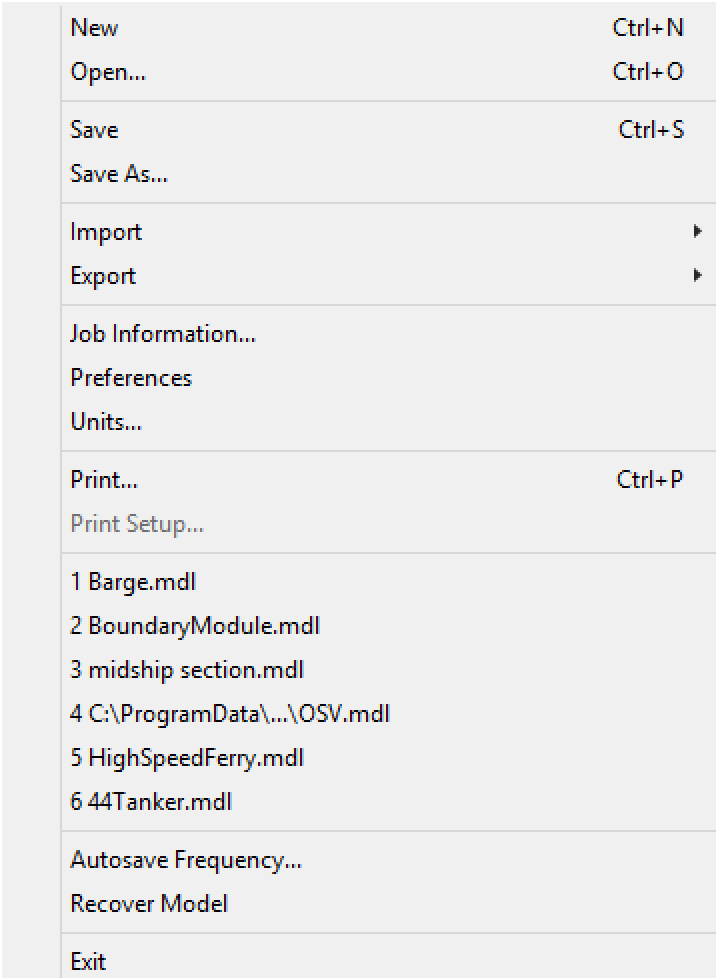
© 2014 Optimum Structural Design, Inc.



4 Views

4.2 File Menu

The File menu provides several common functions within MAESTRO. A brief description of each option is discussed below.



New

This option will open and new MAESTRO file.

Open...

This option allows the user to select and open a previously saved MAESTRO .mdl file.

Save

This option will save the current MAESTRO model file.

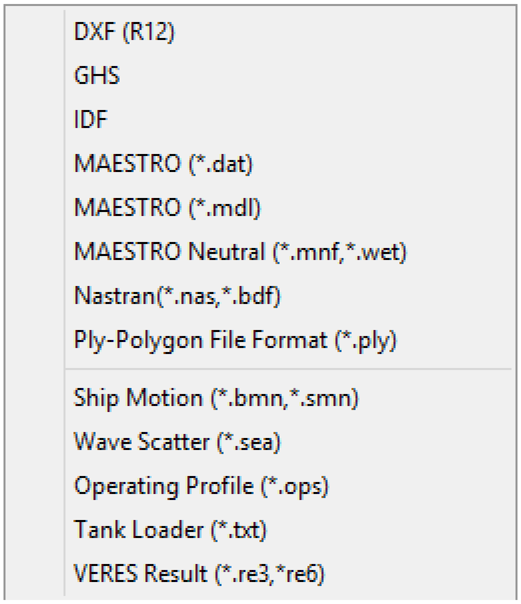
Save As...

This option will allow the user to choose the location and save the current MAESTRO model file under a new file name.

© 2014 Optimum Structural Design, Inc.

Import >

This option will open a fly-out menu of file types for MAESTRO to import.



See Importing Geometry for details of the DXF, GHS, PLY and IDF import options.

See Importing an Analysis Model for details of the MAESTRO (*.dat), MAESTRO (*.mdl), and Nastran import options.

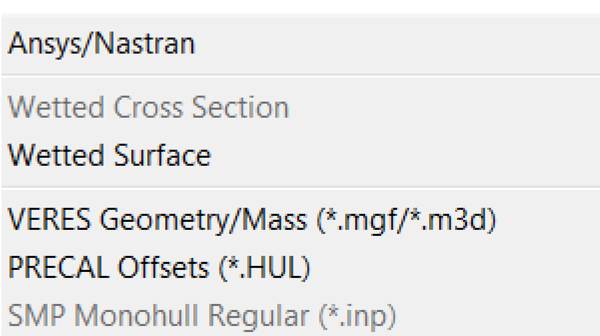
See Importing Hydro Loads for details of the Ship Motion (*.smn), Wave Scatter (*.sea), and the Operating Profile (*.opf) import options.

See NAPA-MAESTRO Interface for details of the MAESTRO Neutral File (*.mnf).

See Tank Loader for details on importing tank loads (*.txt).

Export >

This option will open a fly-out menu of file types for MAESTRO to export.



Job Information...

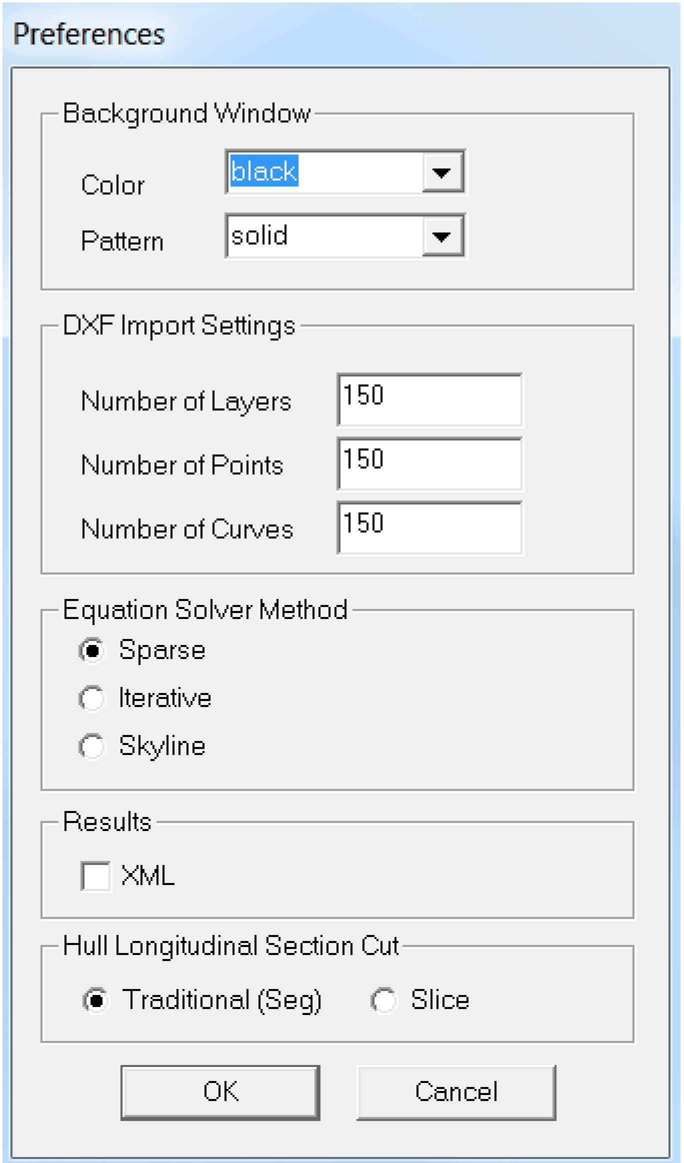
This option will open the Job Information dialog.

Preferences

This option will open the preferences dialog where background settings and DXF Import settings can be changed. The Equation Solver Method section allows the user to choose the Equation Solver Method: Sparse, Iterative, Skyline. The Results section allows the user to export the results file into XML format.

The Hull Longitudinal Section Cut allows the user to select whether the cross section properties are recovered at the MAESTRO endpoints or at the exact longitudinal location specified in the Job Information dialog. For more information, please see the Hull > View Longitudinal > Properties section.

© 2014 Optimum Structural Design, Inc.



Units...

This option will open the Units dialog.

Print...

This option will open the Print dialog.

Print Setup...

This option will open the Print Setup dialog.

Quick Reference Files

The next section of the File menu provides the previously opened model files for quick reference.

Autosave Frequency...

This option allows the autosave frequency to be changed.

Recover Model

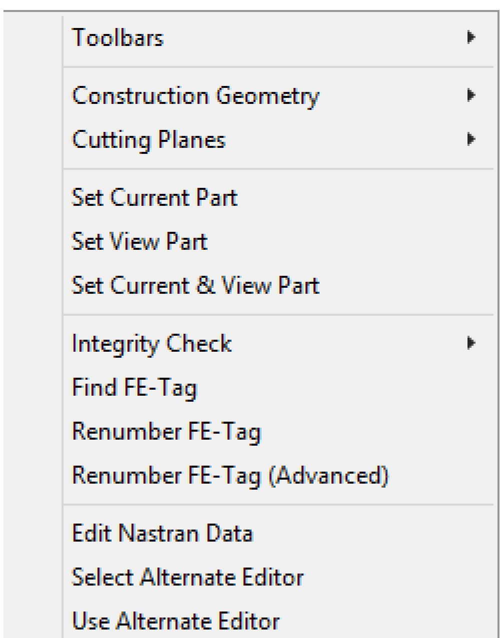
This option will recover the model if a crash or other error occurs.

Exit

This option will exit MAESTRO.

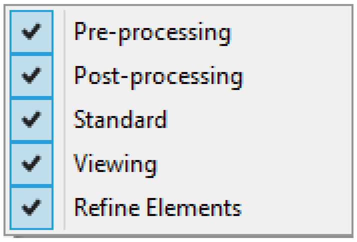
4.3 Tools Menu

The tools menu provides several options to assist the user throughout the finite element analysis process. A brief description of each option is discussed below.



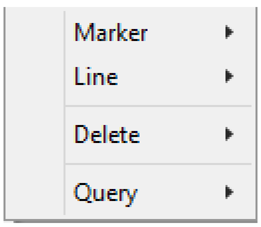
Toolbars >

This option allows the user to toggle on and off the view of the MAESTRO toolbars.



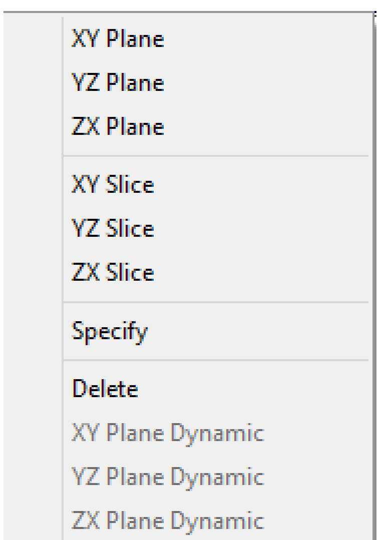
Construction Geometry >

This option opens the Construction Geometry menu which allows the user to create and delete construction lines and markers.



Cutting Planes >

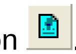
This option allows the user to define a cutting plane or delete an existing one.



Set Current Part

This option allows the user to click a part in the model space to set as the current part. This is the same as clicking the Set Current Part icon .

Set View Part

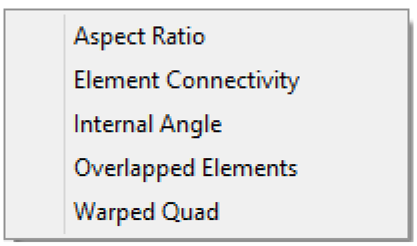
This option allows the user to click a part in the model space to set as the view part, which will zoom fit the window to this part. This is the same as clicking the Set View Part icon .

Set Current & View Part

This option allows the user to click a part in the model space to set as the current and view part. This combines the function of the *Set Current Part* and *Set View Part* options. This can also be done by clicking the Set Current & View Part icon .

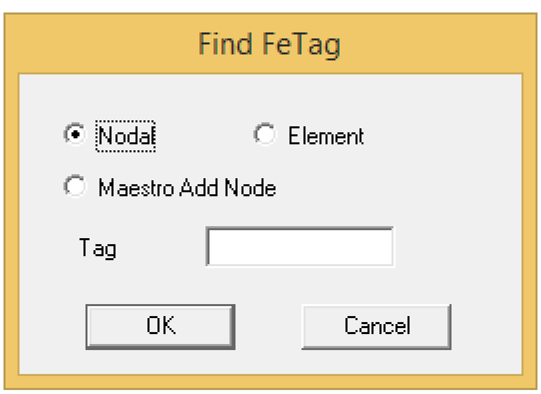
Integrity Check >

This option opens the integrity check options menu allowing the user to choose one of the functions to check the model.



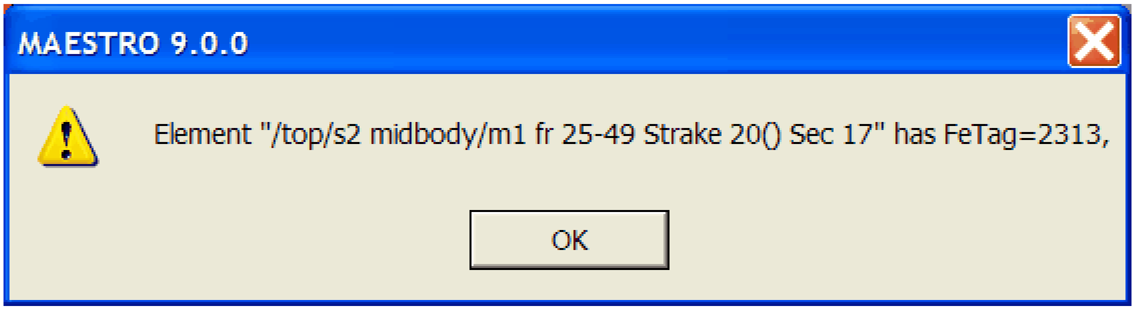
Find FE-Tag

This option will launch the Find FE-Tag to locate a node or element by it's FETag.



Click the radio button to choose between "Nodal" and "Element" and type the FeTag to search. A dialog will list the node or element location for the enter FeTag. This location will be echoed to the Output tab.

© 2014 Optimum Structural Design, Inc.



Renumber FE-Tag

This option will renumber all Nodal and Element FE-Tags in the model to remove any gaps in FE-Tag numbers.

Renumber FE-Tag (Advanced)

This option is similar to the "Renumber FE-Tag" functionality except is provides the user with the ability to specify the "starting" value of the FE-Tag.

Edit Nastran Data

This option allows the user to open and edit the Nastran *.nas file (if applicable) in the MAESTRO Output tab or a separate editor program if the *Use Non-MAESTRO Editor* option is checked.

Select Alternate Editor

This option allows the user to select a text editor program to use for *.nas files outside of MAESTRO.

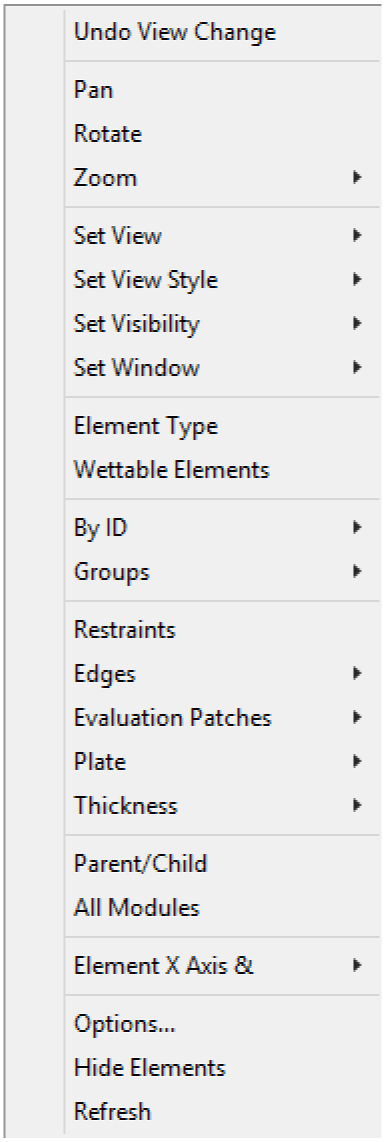
Use Alternate Editor

This option allows the user to toggle on or off the option to use an alternate editor for Nastran data. A check mark will appear next to the option in the menu if an alternate editor is to be used, otherwise the Nastran data will open in the Output tab.

4.4 View Menu

The View menu provides several options for viewing the model that help in the FEA modeling process. A brief descriptio

© 2014 Optimum Structural Design, Inc.



Undo View Change

This option will undo the last change to the model view made. For example, if the model is being viewed in the profile view and is changed to body plan, choosing this option will return the view to profile.

Pan

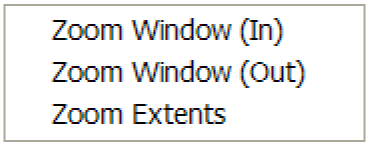
This option allows the user to pan the model with the mouse. This can also be done by holding the *Shift* key and clicking and holding the mouse wheel button and moving the mouse to pan the model.

© 2014 Optimum Structural Design, Inc.

Rotate

This option allows the user to rotate the model with the mouse. This can also be done by clicking and holding the mouse wheel button and moving the model as desired.

Zoom >



Zoom Window (In)

This option will allow the user to select an area of the model by a user-defined box and zoom in. This can also be done by right-clicking in the model space and selecting *Zoom In.*

Zoom Window (Out)

This option will allow the user to select an area of the model by a user-defined box and zoom out. This can also be done by right-clicking in the model space and selecting *Zoom Out.*

Zoom Extents

This option allows the user to select to zoom to the extents of the model. This can also be done by right-clicking in the model space and selecting *Fit.*

Set View >

This option allows the user to select from the standard views or a named view.

Specify

This option will turn on the rotate function allowing the user to click the left mouse button and spin the model interactively.

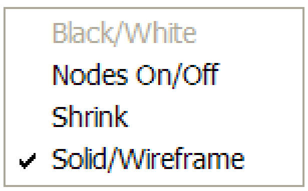
Store

This option allows the user to "store" the current view to be recalled at a later time.

Recall

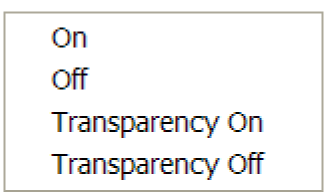
This option allows the user to "recall" the last stored view.

Set View Style >



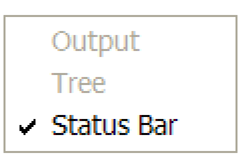
This allows the user to change the view between a black or white background (if current background is black or white), show nodes on or off, shrink or unshrink elements, and solid or wireframe view. These options can also be set in the View Options dialog or from the View Toolbar.

Set Visibility >



This option allows the user to set a module display on or off and transparent or not transparent. These options can also be set in the Parts Tree.

Set Window >



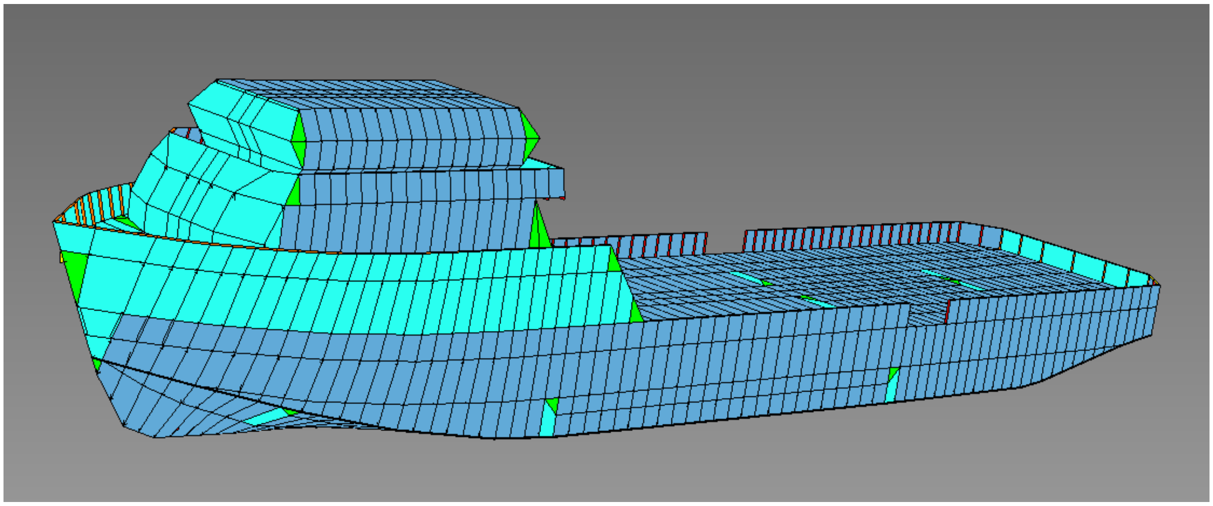
© 2014 Optimum Structural Design, Inc.

This option allows the user to show or hide the parts/groups tree, the output section and the status bar.

Element Type

This option is the default MAESTRO view where each type of element has a different color.

In conjunction with the Element Type view, the user can dynamically query and element to change it from "wetted" to "unwetted" and vice-versa, reverse the element normal, and rotate the local x-axis of the element. To use this functionality, the user must first change to the Element Type view, toggle the Dynamic Query icon, move the mouse cursor over a particular element, right-click the mouse, and finally select the menu item.

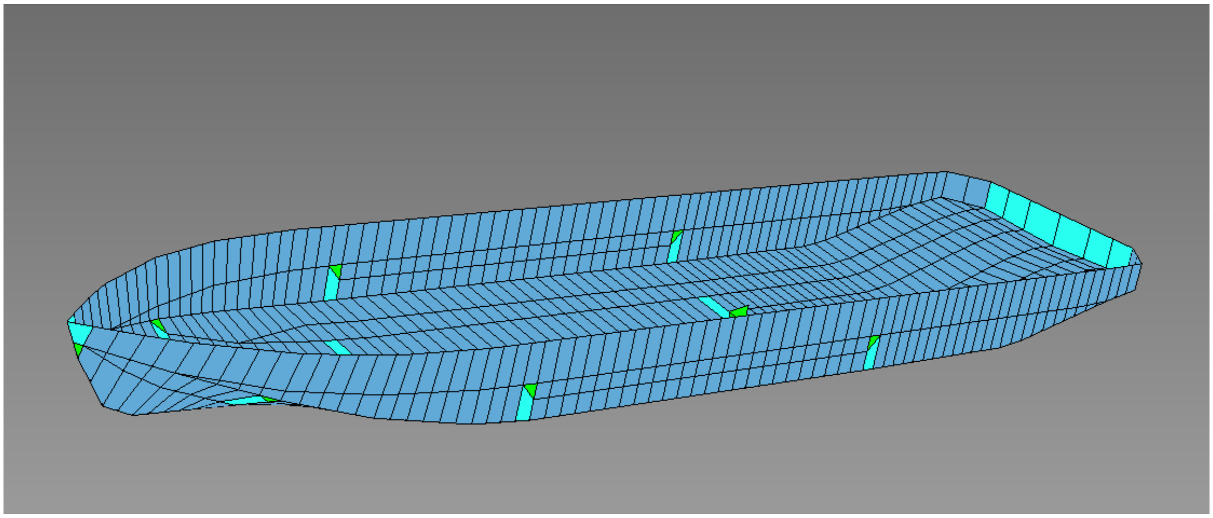


Wettable Elements

This option will change the view to display on the elements that are defined as "wettable".

In conjunction with the Wettable Elements view, the user can dynamically query an element to change it from "wettable" to "unwettable" and vice versa. To use this functionality, the user must first change to the Element Type view, toggle the Dynamic Query icon, move the mouse cursor over a particular element, right-click the mouse, and finally select the menu item.

To return to viewing all elements, you must select **View > Wettable Elements** to uncheck the option.



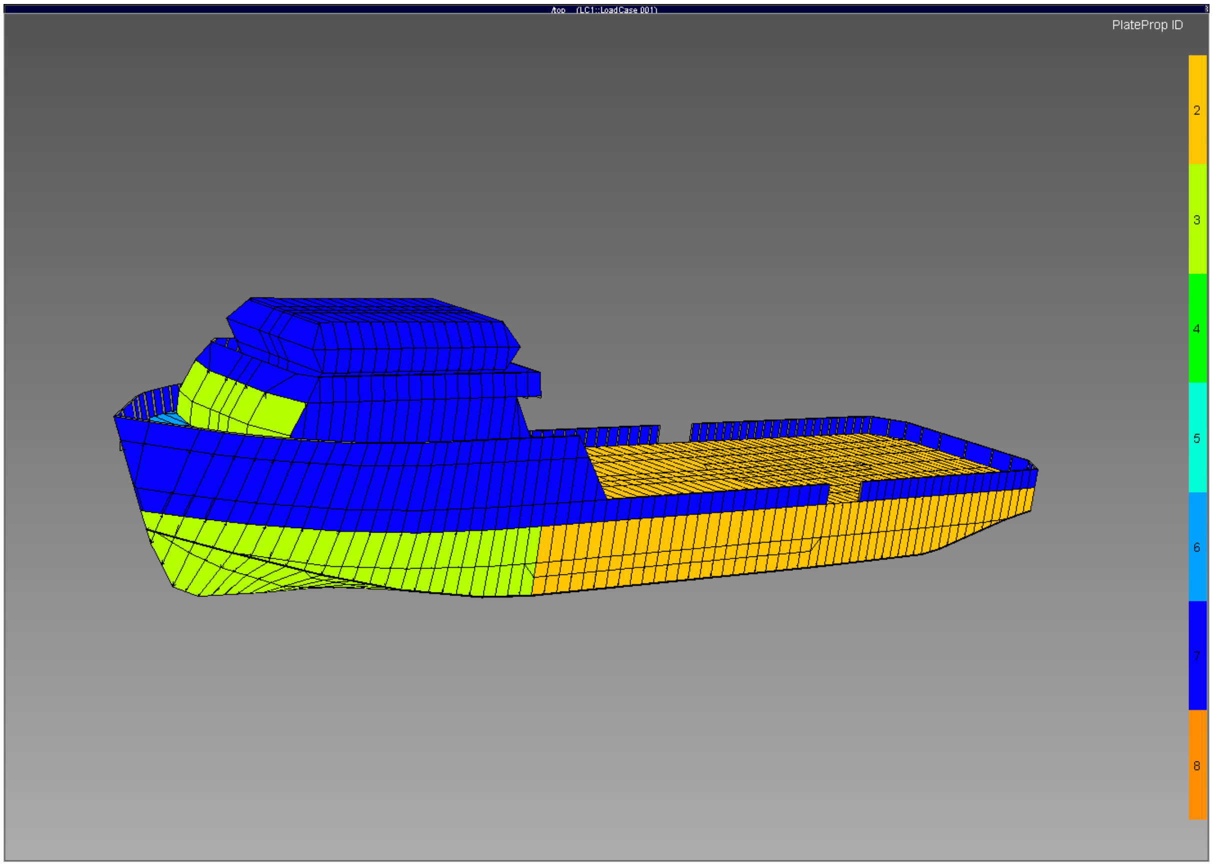
By ID

Material

Beam Property
Plate Property
Rod Property
Stiffener Layout

This option allows the user to view the model by material ID, plate property ID, beam property ID, rod property ID, or stiffener layout ID.

The dynamic query function can be used to highlight an element and change the beam, rod, or stiffener layout property by right-clicking on the element in the corresponding view.



Groups

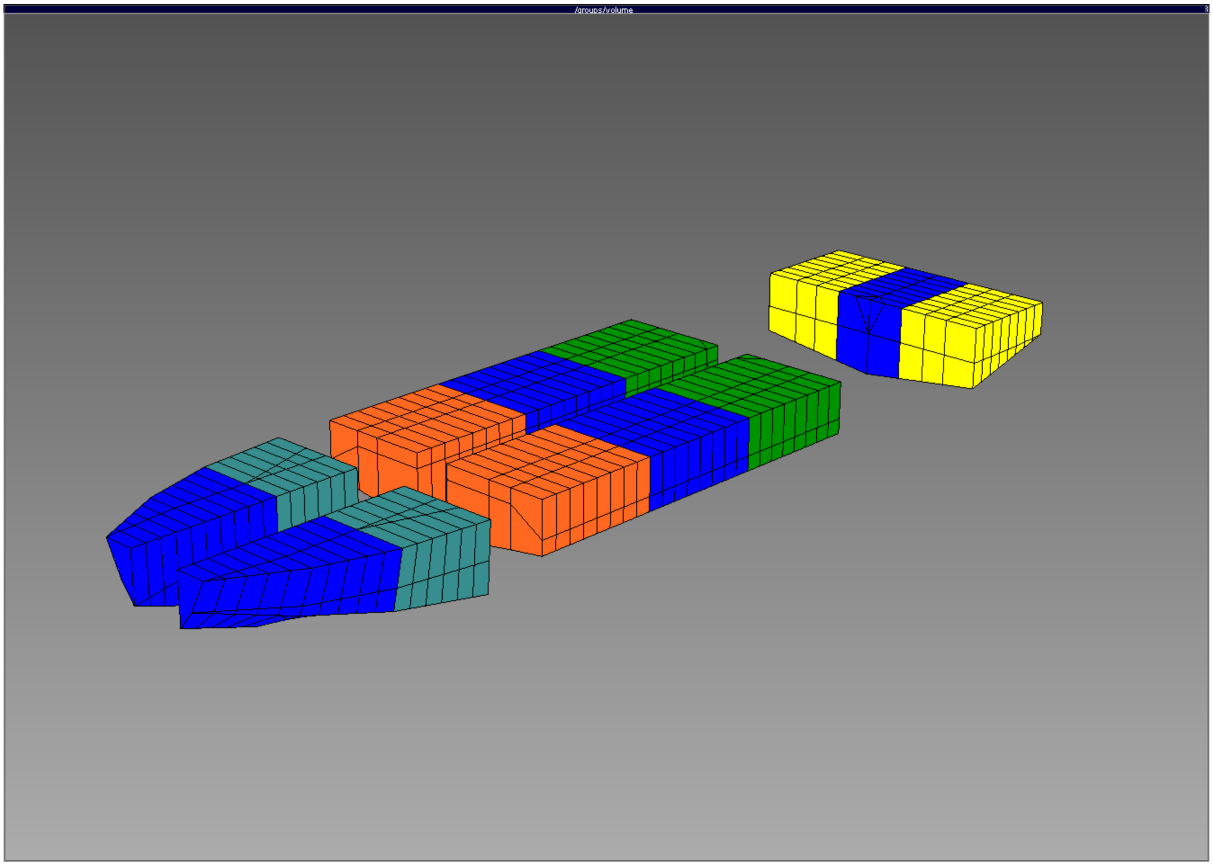
- Corrosion Side
- User-defined Color
- Volume/Plate Pressure Side

Corrosion Side

This option allows the user to view the side of the element which corrosion is applied to, shown as pink, if a corrosion group is defined.

User-defined Color

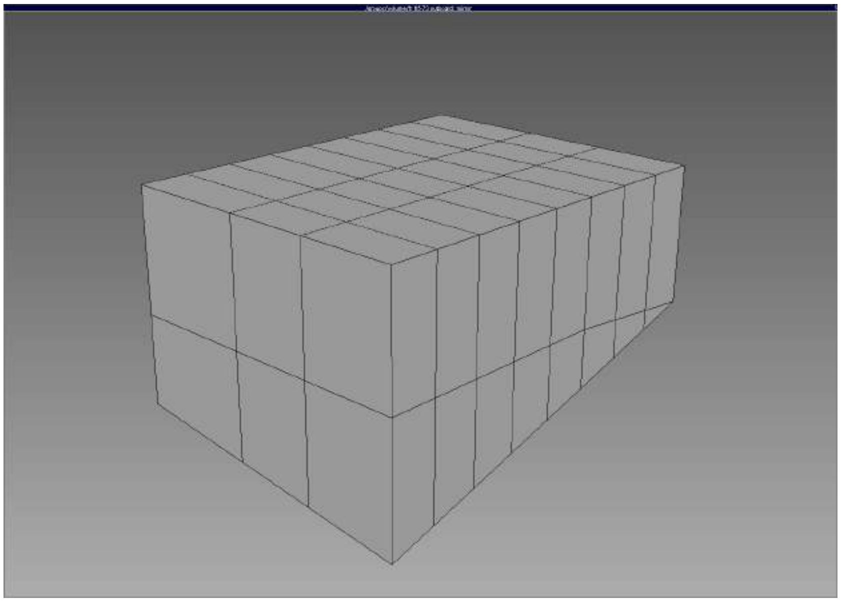
This option will display the currently viewed groups as the user-defined color set in the groups dialog.

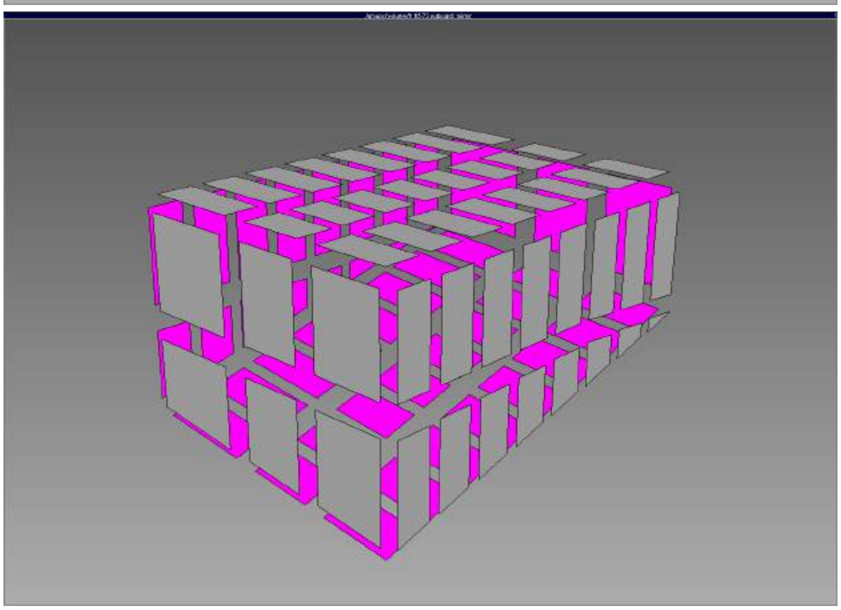


Volume/Plate Pressure Side

As mentioned above, MAESTRO distinguishes between a positive and negative pressure face by a pink and gray face respectively. The pink face of the element will receive the positive pressure. As seen below, a typical tank will have the pressure side on the "inside" of the tank. This is very important when defining tanks.

© 2014 Optimum Structural Design, Inc.

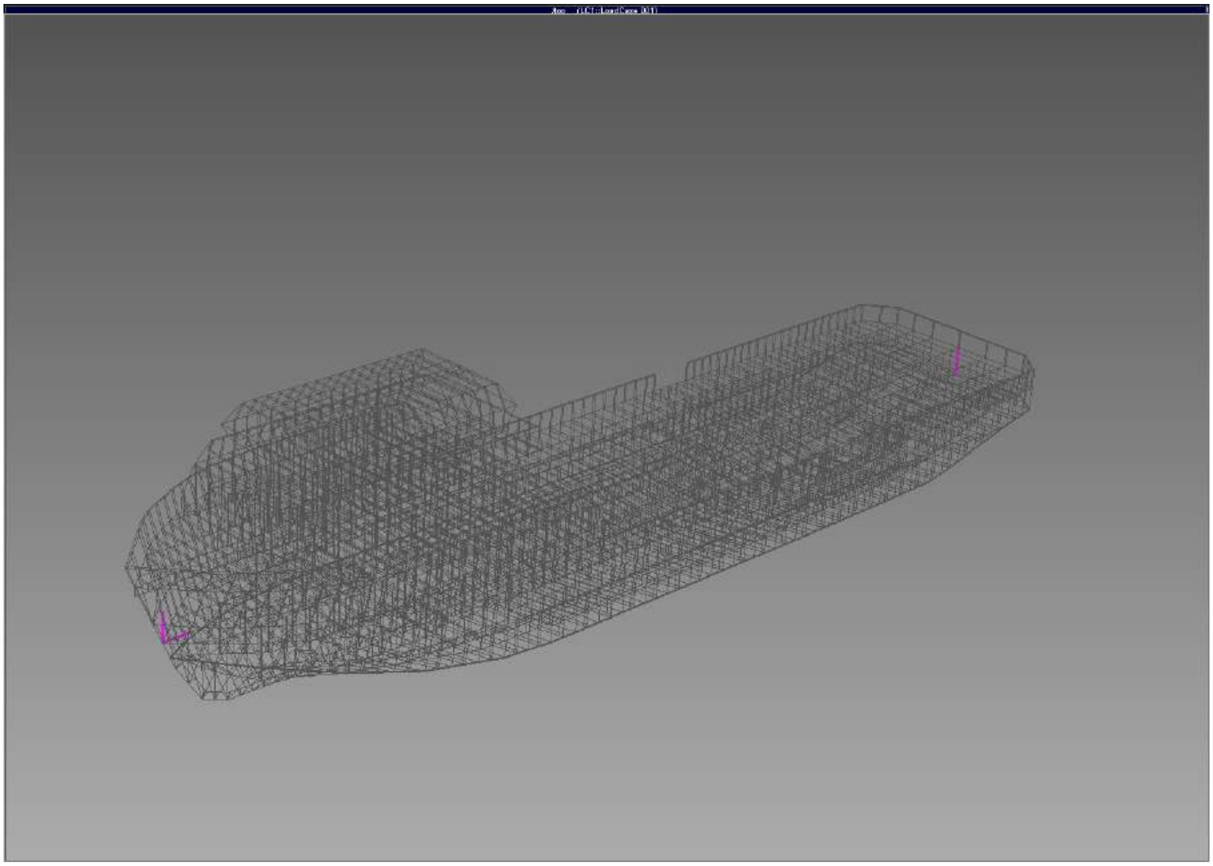




In conjunction with the Volume/Plate Pressure Side view, the user can use the Dynamic Query tool, which can be initiated via the  icon, to Flip Pressure Side, define the volume group as a Centerline Group, and Remove an element associated with the group. When using this functionality, the user can also choose to switch between the Element Pressure Side view or the Tank/Group Pressure Side view. This aids in the building of tanks. To use this functionality the user must first change to the Volume/Plate Pressure Side view, toggle the Dynamic Query icon, move the mouse cursor over a particular element, right-click the mouse, and finally select the menu item.

Restraints

This option displays the model in wireframe view showing the nodes that are constrained and in which direction.



Edges

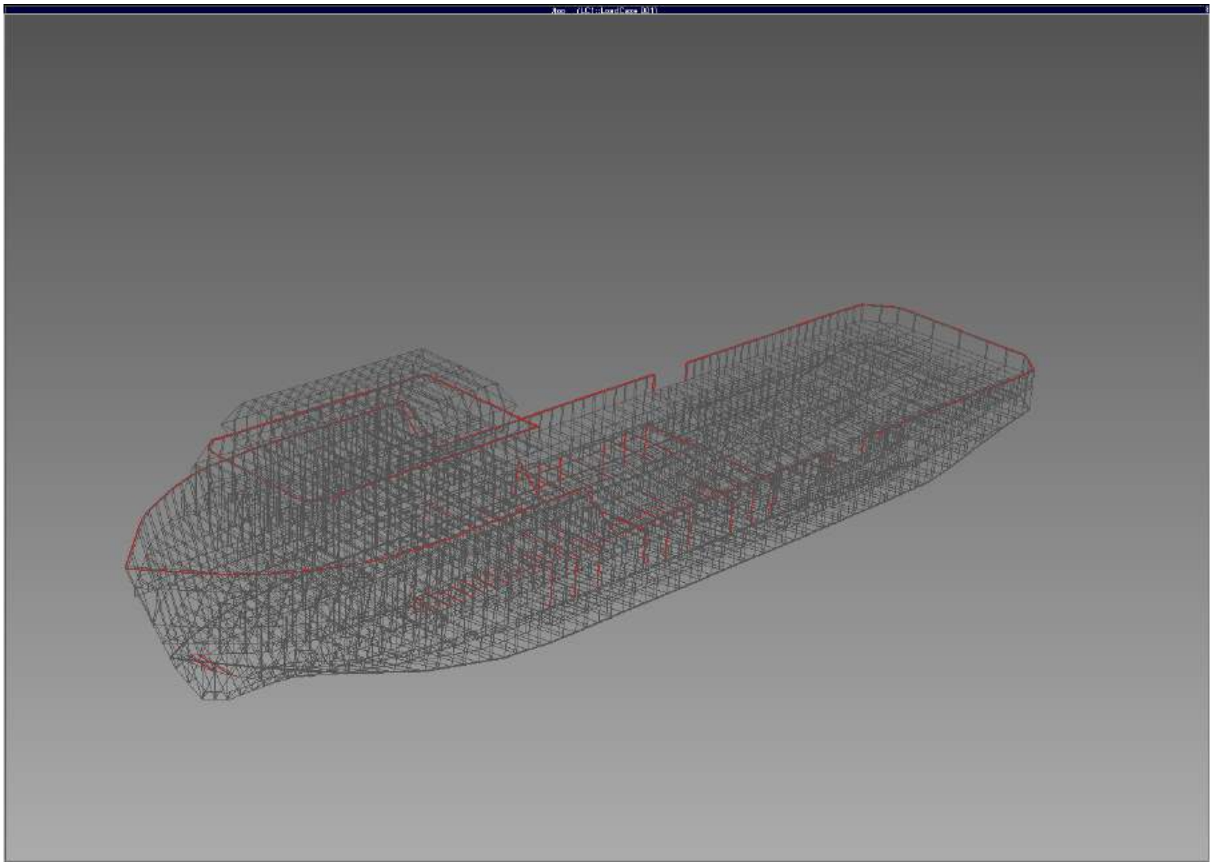
Free Edges

3 Edges
4 or More

Free Edges

This option will check the model for free edges and can create a group of potential free edge errors if desired. It is easiest to see the free edges in red when the model is in wireframe view.

© 2014 Optimum Structural Design, Inc.



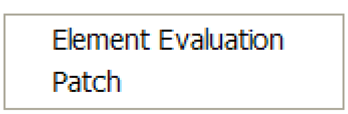
3 Edges

This option checks the model for edges where 3 elements meet and can create a general group of the 3 edge elements if desired. It is easiest to see the edges in red when the model is in wireframe view.

4 or More

This option checks the model for edges where 4 or more elements meet and can create a general group of the 4 or more edge elements if desired. It is easiest to see the edges in red when the model is in wireframe view.

Evaluation Patches



Included Elements

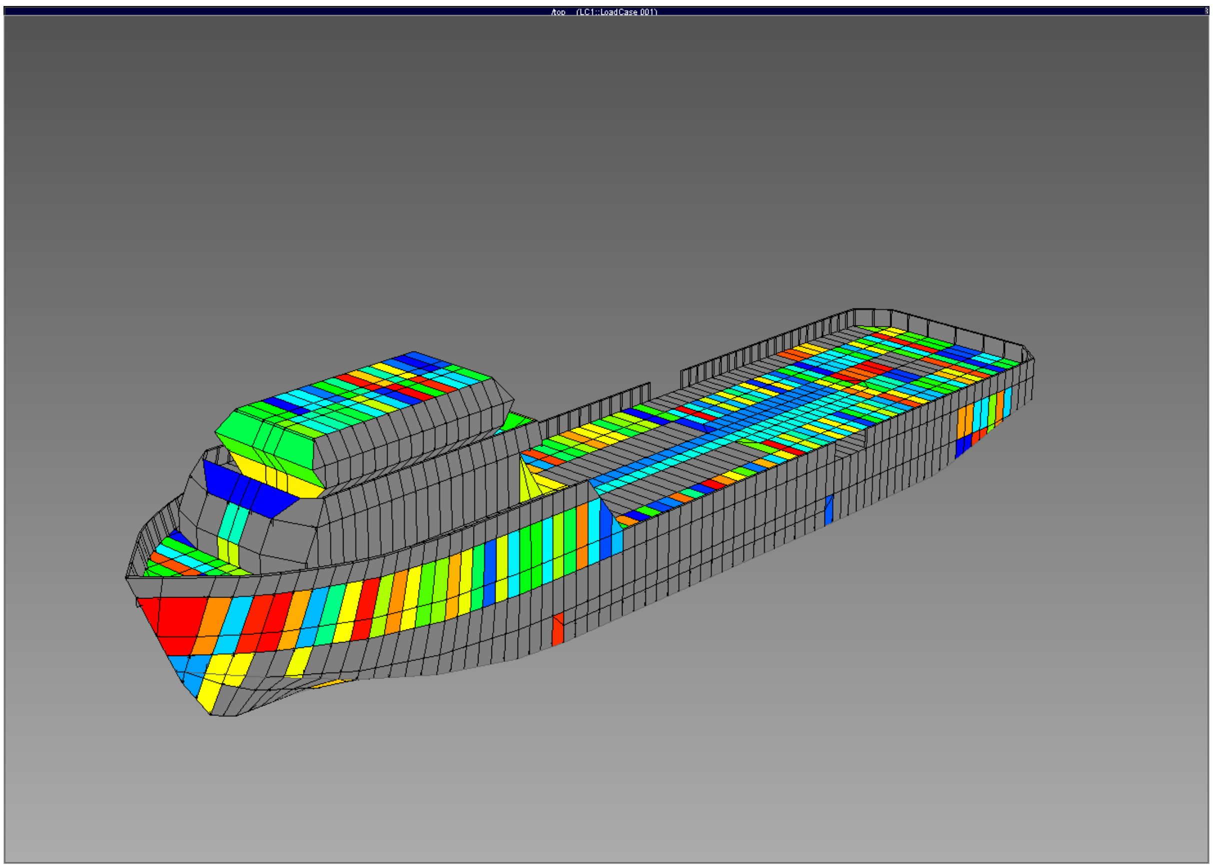
This option will display the elements that make up the evaluation patches that will be analyzed. This includes single panel elements that are bordered by beam elements on all

© 2014 Optimum Structural Design, Inc.

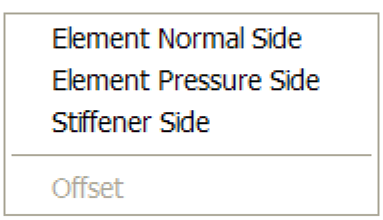
edges. The dynamic query icon  (with patch option checked) can be used to highlight an element or patch and select *Evaluation Off.*

Patch

This option will display the defined patches for the model. They dynamic query icon  can be used to highlight a patch and retrieve information about it by clicking the icon and selecting *Patch* from the drop down menu.



Plate



Element Normal Side

The figure below shows how MAESTRO distinguishes between a positive normal and negative normal face. The pink face of the element will receive the positive normal side.

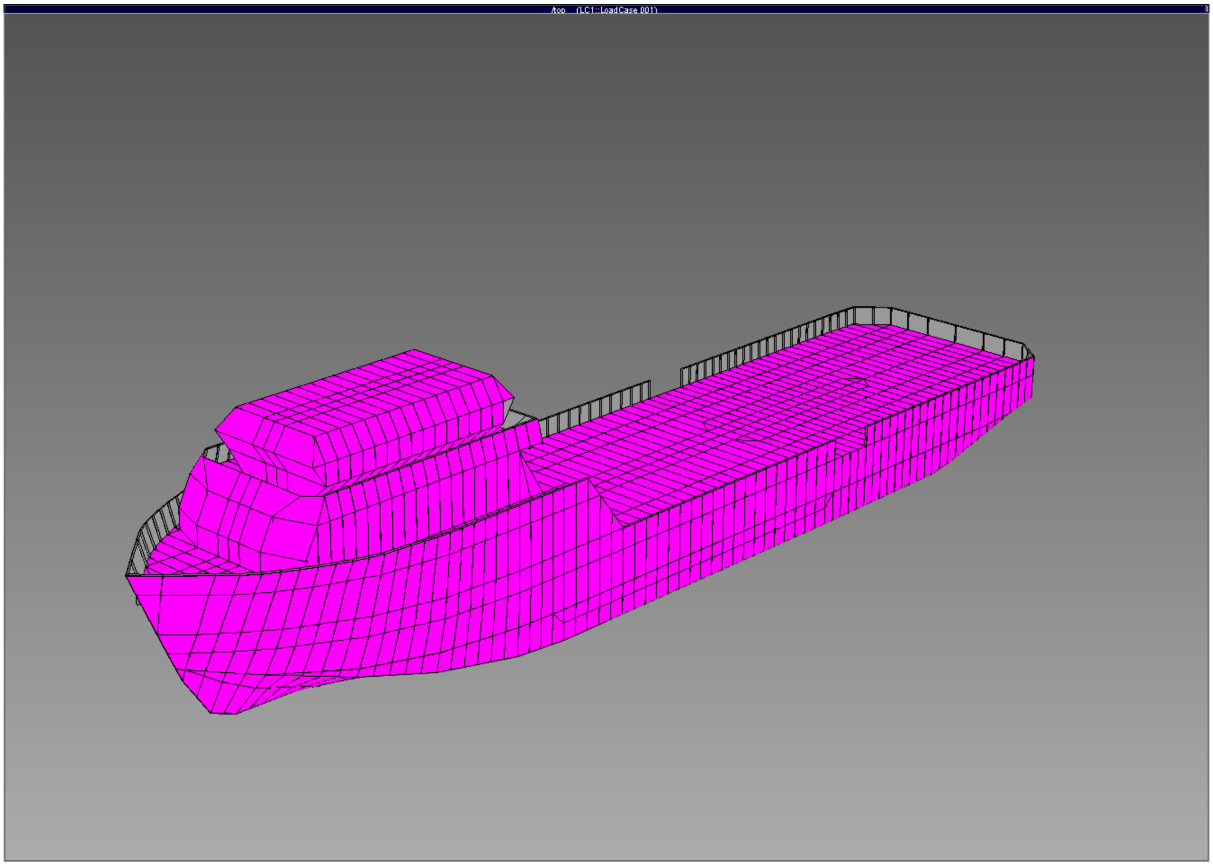
In conjunction with the Element Normal Side view, the user can use the Dynamic Query tool, which can be initiated via the icon, to reverse the element normal and consequently flip the pressure side. To use this functionality the user must first change to the Element Normal Side view, toggle the Dynamic Query icon, move the mouse cursor over a particular element, right-click the mouse, and finally select the menu item.

NOTE: *Element Normal Side* cannot be flipped for Strake Elements. Only the flipping of the pressure side can occur for strakes resulting in the flipping of the Strake Frame web orientation.

Element Pressure Side

The figure below shows how MAESTRO distinguishes between a positive and negative pressure face. The pink face of the element will receive the positive pressure.

In conjunction with the Element Pressure Side view, the user can use the Dynamic Query tool, which can be initiated via the  icon, to reverse the element pressure side. This will flip the element pressure side. To use this functionality the user must first change to the Element Pressure Side view, toggle the Dynamic Query icon, move the mouse cursor over a particular element, right-click the mouse, and finally select the menu item.



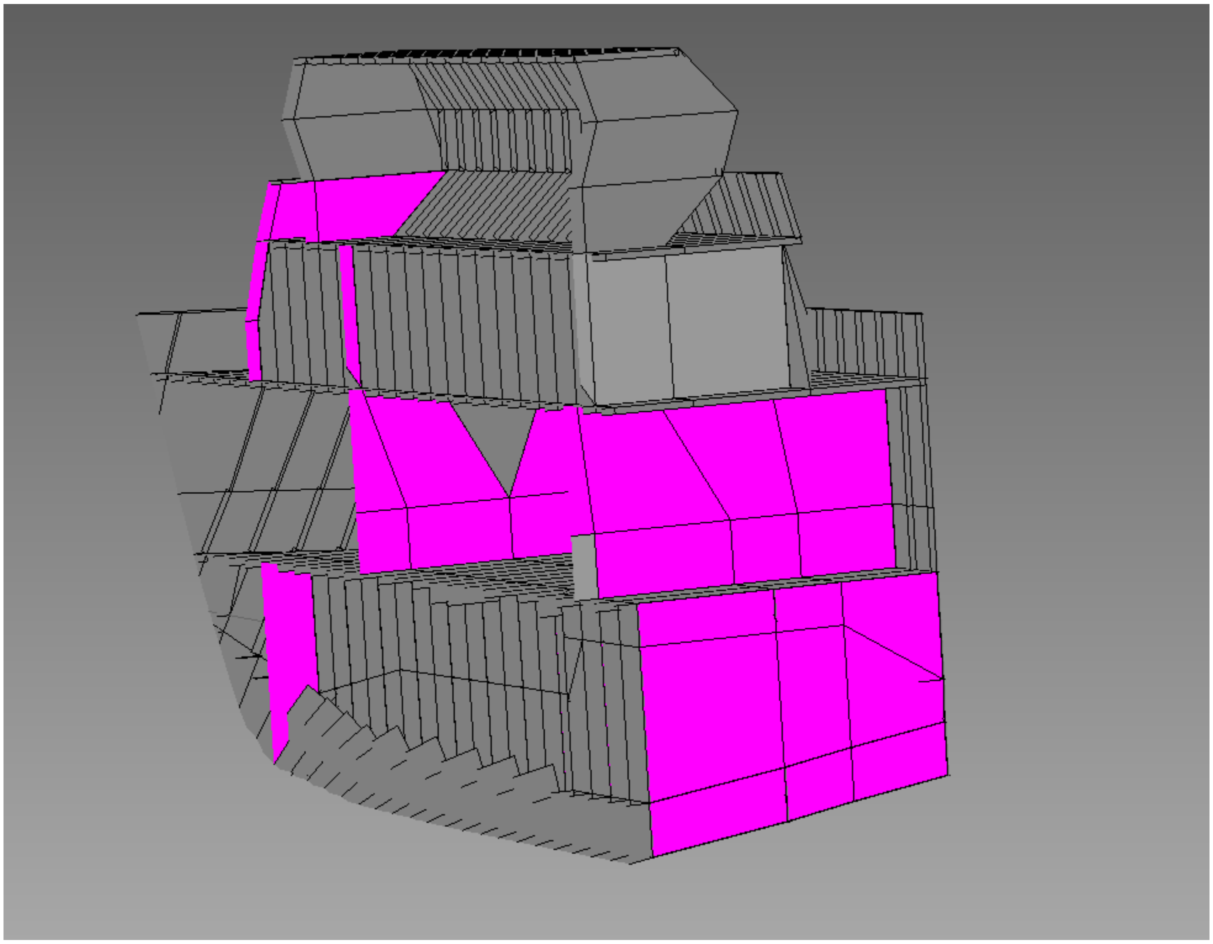
The pressure side is plotted in pink on the side of the element a pressure load would be applied in order to have the net force vector pointing in the direction of the elements positive normal using the right-hand rule. The pressure side of a quad or tri element is the same as the normal side, whereas the pressure side of a strake is always opposite to the frame side.

Stiffener Side

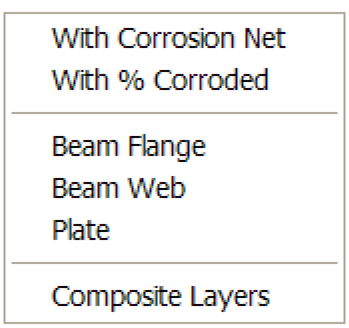
The pink face designates the stiffener side of the plate element.

In conjunction with the Stiffener Side view, the user can use the Dynamic Query tool, which can be initiated via the  icon, to Flip Stiffener Side. With this view displaying, toggle the Dynamic Query icon, move the mouse cursor over a particular element, right-click the mouse, and finally select Flip Stiffener Side.

© 2014 Optimum Structural Design, Inc.



Thickness



With Corrosion Net

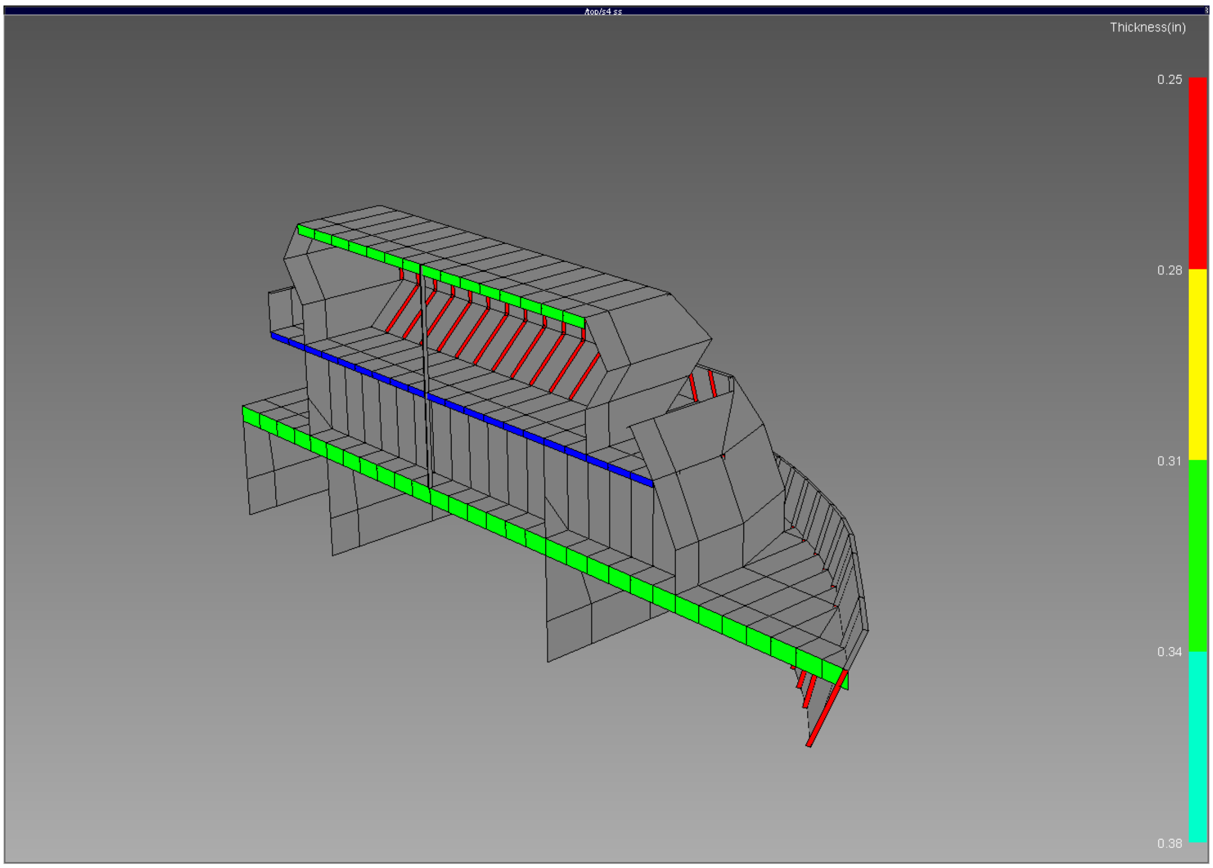
This option is used to toggle the view to display the thickness of Plate, Beam Web, or Beam Flange with or without net corrosion and is used in conjunction with the Plate, Beam Web, or Beam Flange menu option.

With % Corroded

This option is used to toggle the view to display the thickness of Plate, Beam Web, or Beam Flange with or without % corroded and is used in conjunction with the Plate, Beam Web, or Beam Flange menu option.

Beam Flange

The Beam Flange view, under the Thickness menu, displays the flange thickness of all beam elements in the model. This helps the user to identify mistakes in the model.

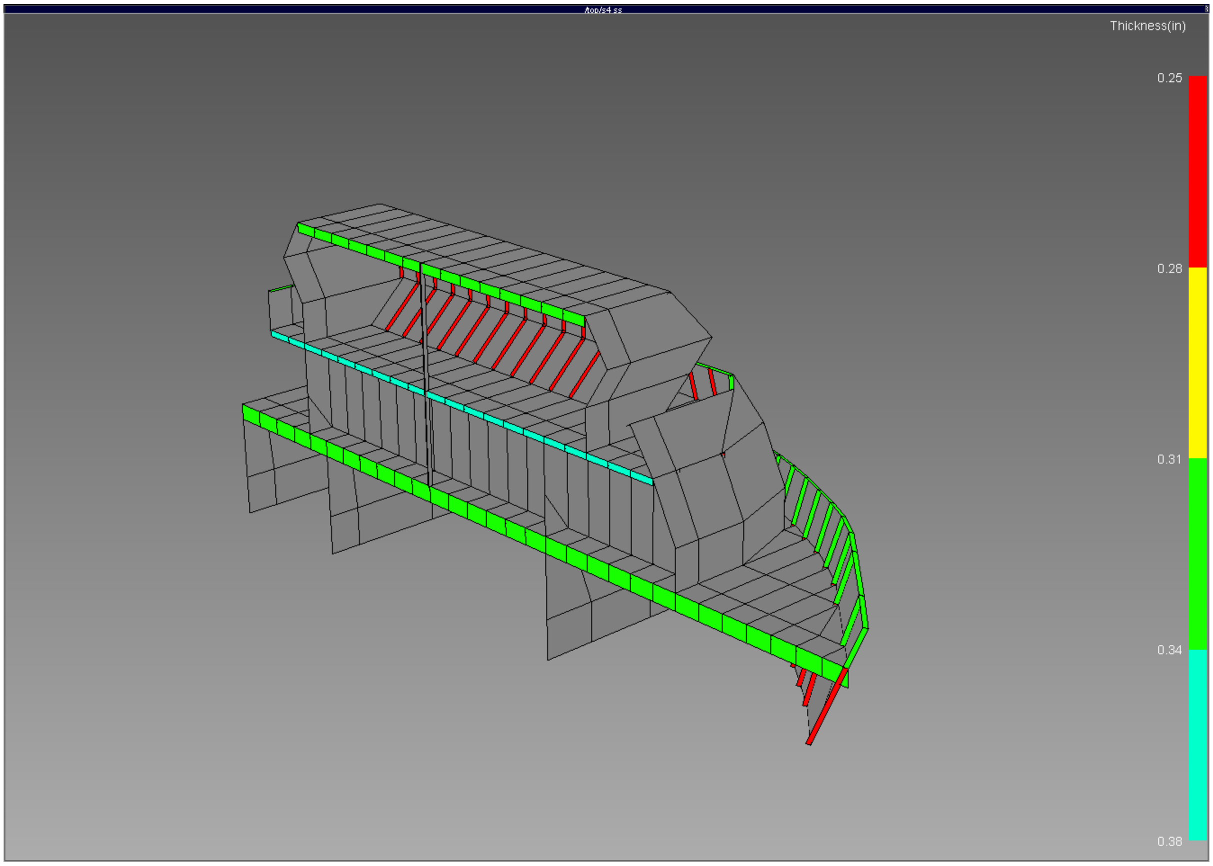


In conjunction with the Beam Flange thickness view, the user can use the Dynamic Query tool, which can be initiated via the  icon, to change the thickness in a particular element. To use this functionality the user must first change to the Beam Flange view, toggle the Dynamic Query icon, move the mouse cursor over a particular element, right-click the mouse, and finally select the thickness of choice.

Beam Web

The Beam Web view, under the Thickness menu, displays the web thickness of all beam elements in the model. This helps the user to identify mistakes in the model.

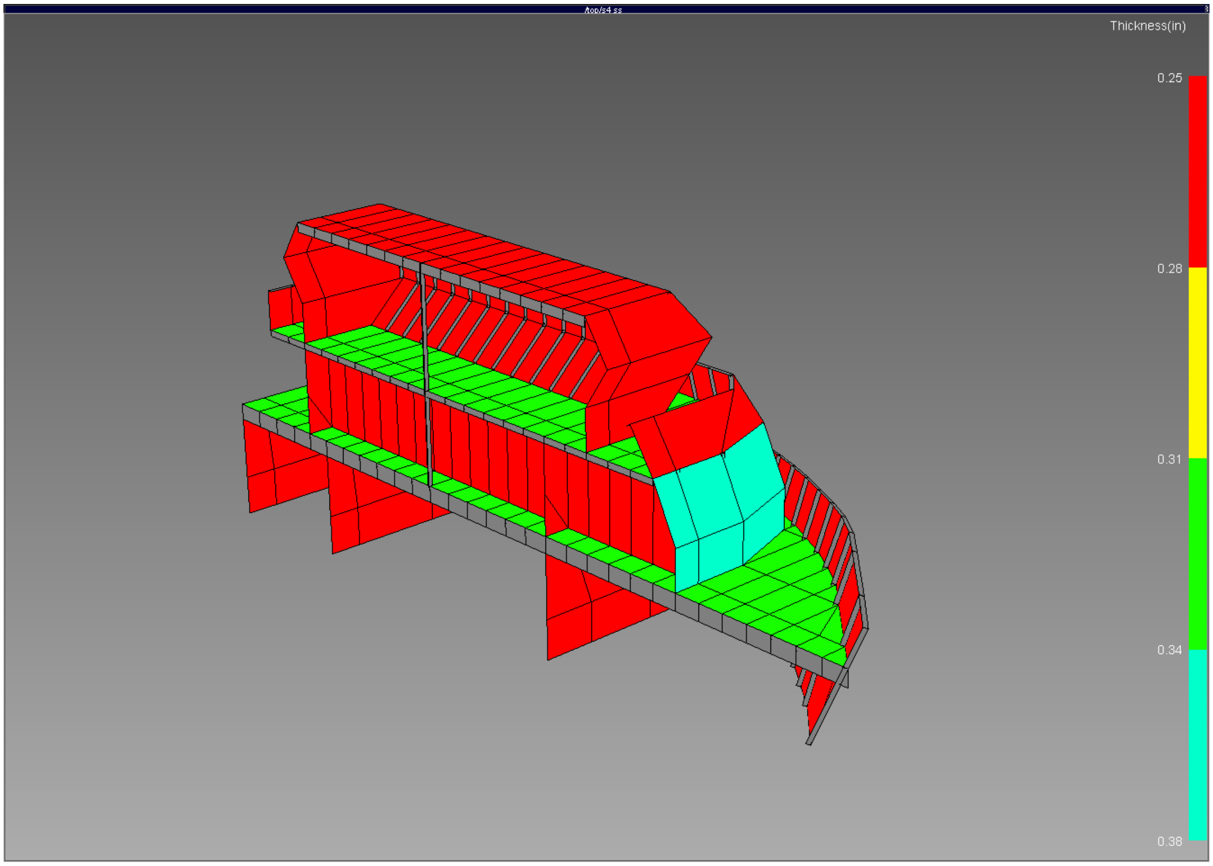
© 2014 Optimum Structural Design, Inc.



In conjunction with the Beam Web thickness view, the user can use the Dynamic Query tool , which can be initiated via the  icon, to change the thickness in a particular element. To use this functionality the user must first change to the Beam Web view, toggle the Dynamic Query icon, move the mouse cursor over a particular element, right-click the mouse, and finally select the thickness of choice.

Plate

The Plate view, under the Thickness menu, displays the plate element thickness in the model. This helps the user to identify mistakes in the model.



In conjunction with the Plate thickness view, the user can use the Dynamic Query tool, which can be initiated via the  icon, to change the thickness in a particular element. To use this functionality the user must first change to the Plate view, toggle the Dynamic Query icon, move the mouse cursor over a particular element, right-click the mouse, and finally select the thickness of choice.

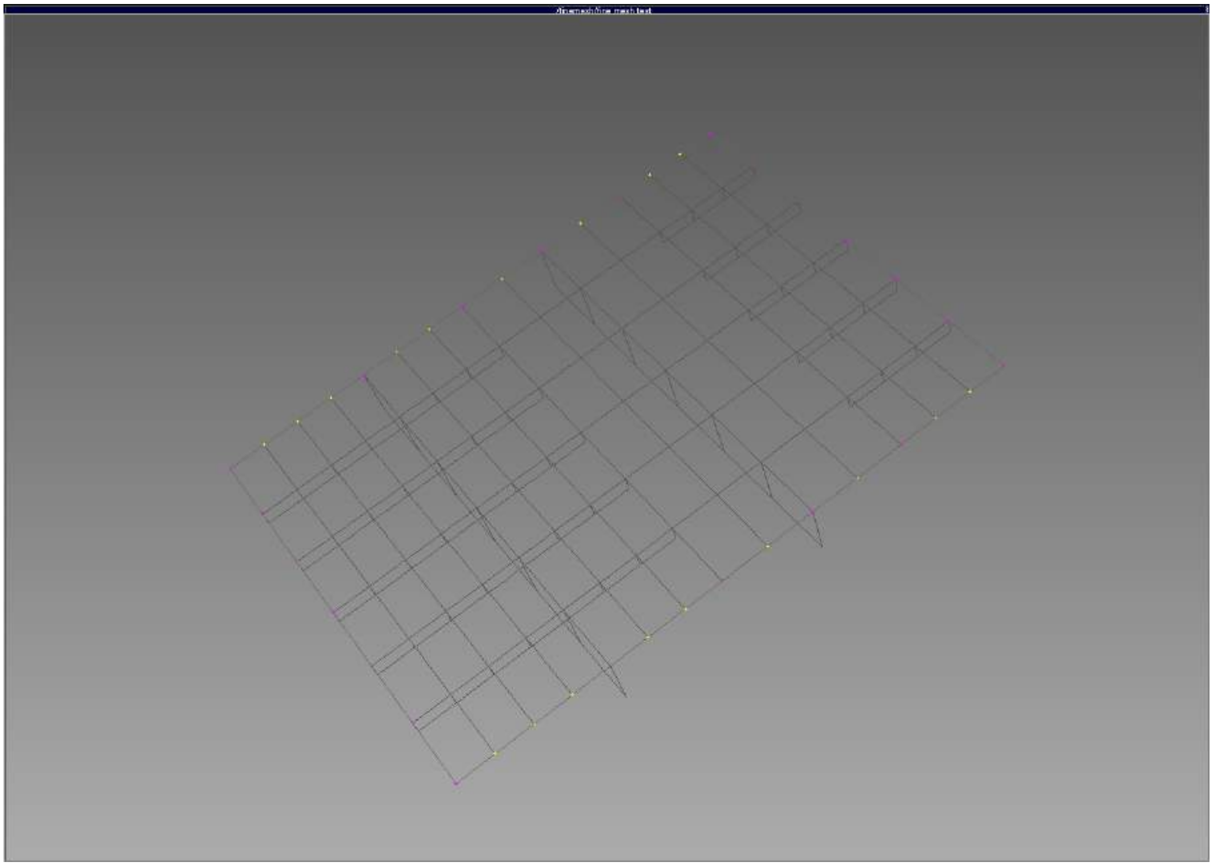
Composite Layers

The Composite Layers view displays the number of layers associated with all composite elements in the model.

Parent/Child

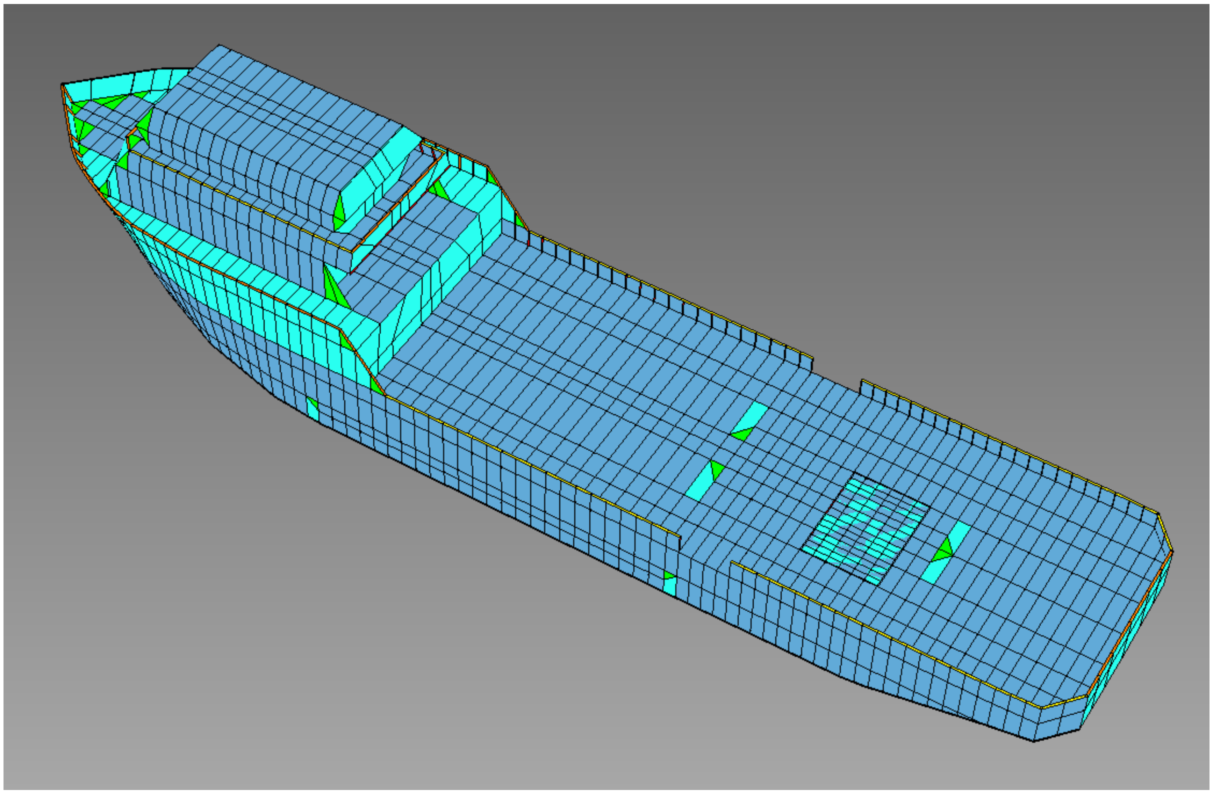
This option will display the master and secondary nodes of a fine mesh model. The nodes that correspond to global nodes will be shown in pink and the fine mesh secondary nodes will be shown in yellow. In other words this option will display the master(pink) and secondary(yellow) nodes for the R-Spline elements that translate the displacements between the global and fine mesh models.

© 2014 Optimum Structural Design, Inc.



All Modules

This option is used to view the global, coarse mesh model with the fine mesh model(s) shown on top of the coarse mesh elements.

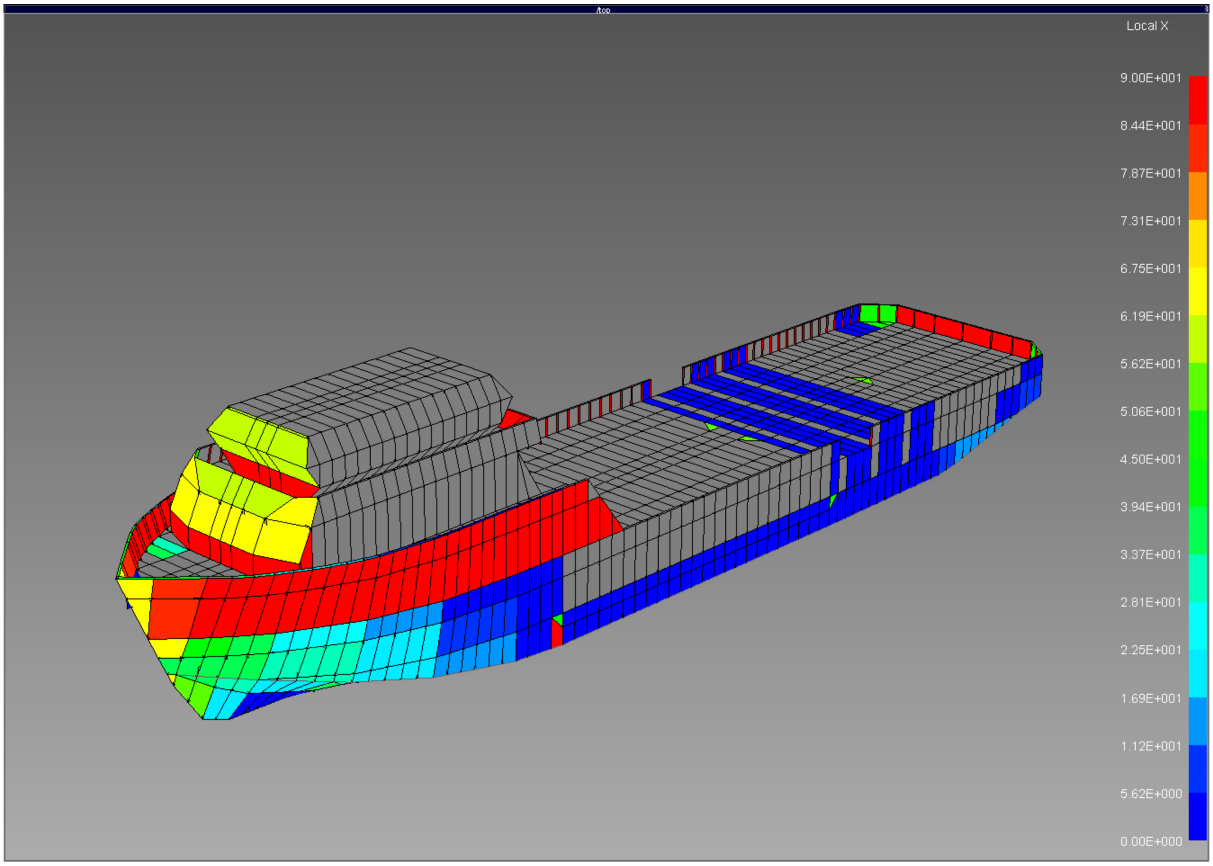


Element X Axis &

Global X
Global Y
Global Z

Global X

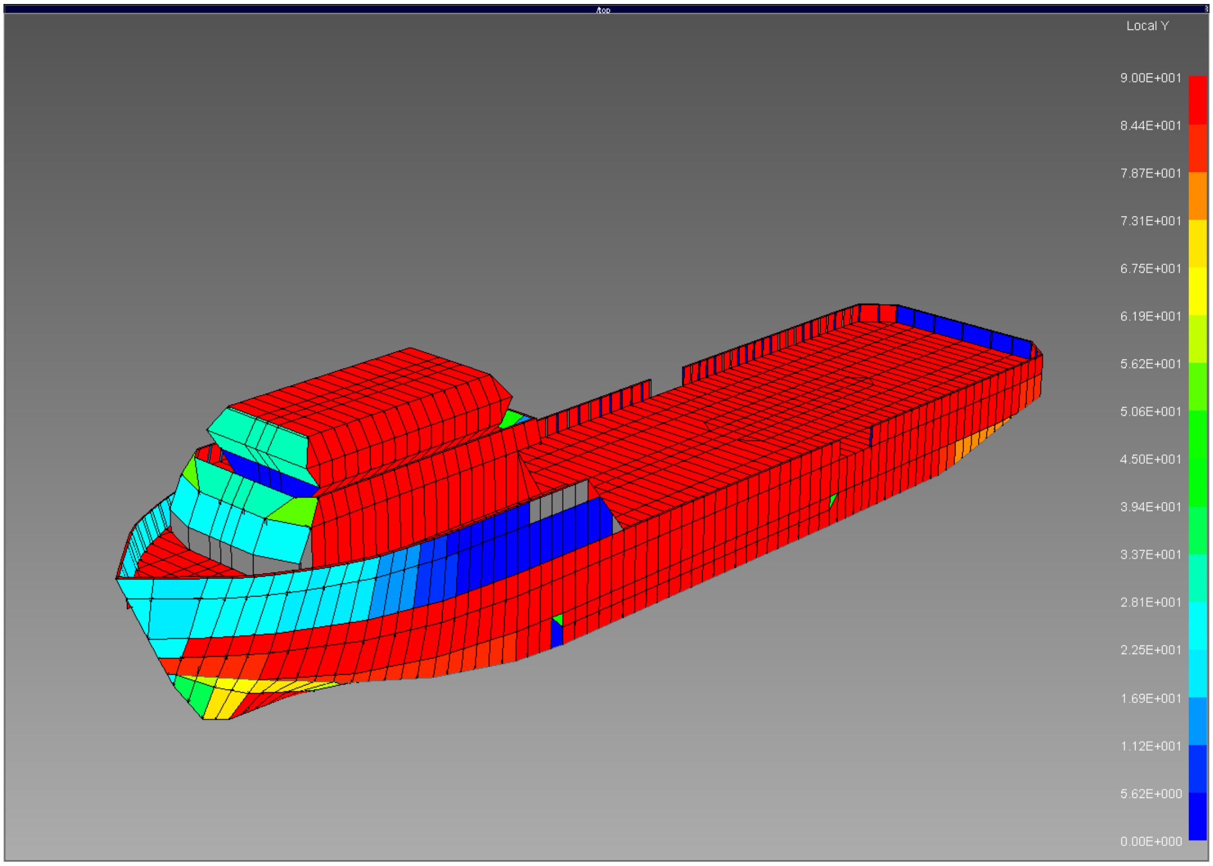
This option will color the elements corresponding to the angle that the element's local x-axis makes with the global x-axis.



Global Y

This option will color the elements corresponding to the angle that the element's local x-axis makes with the global y-axis.

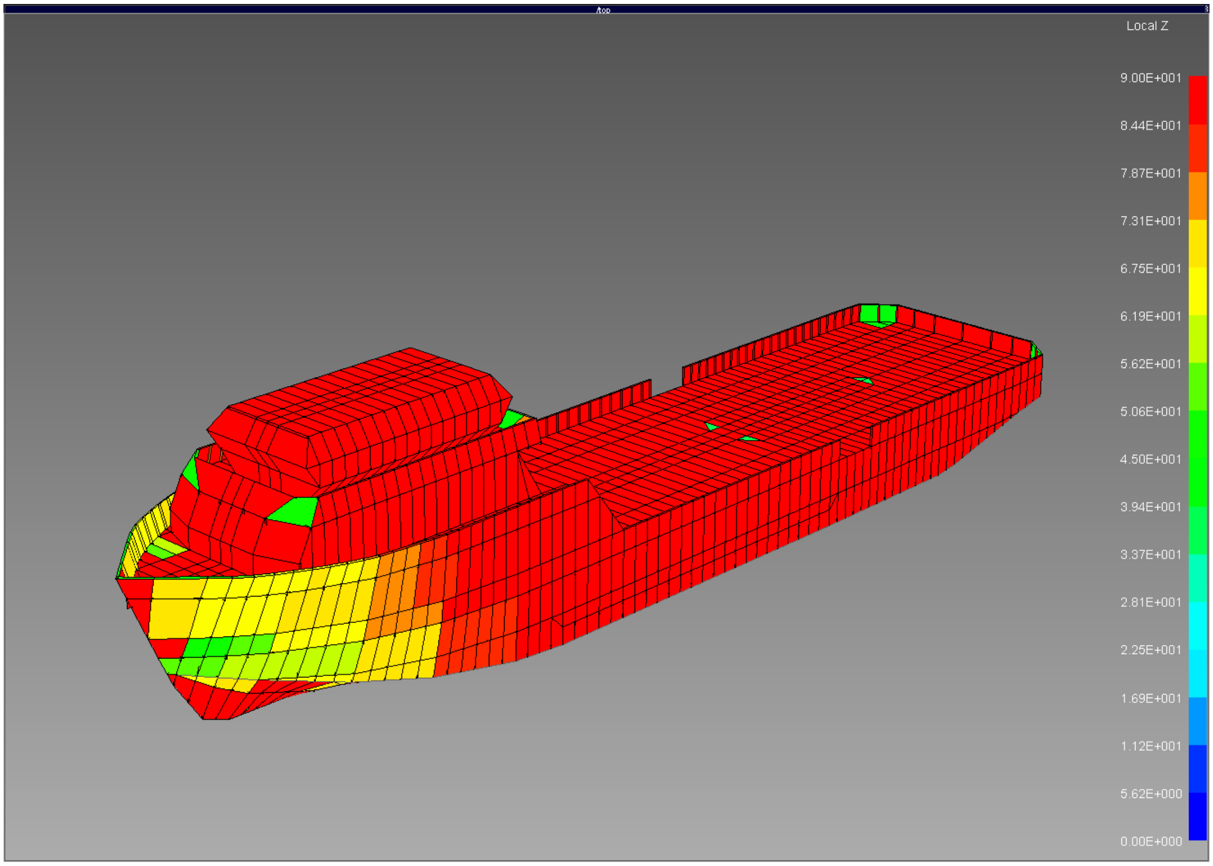
© 2014 Optimum Structural Design, Inc.



Global Z

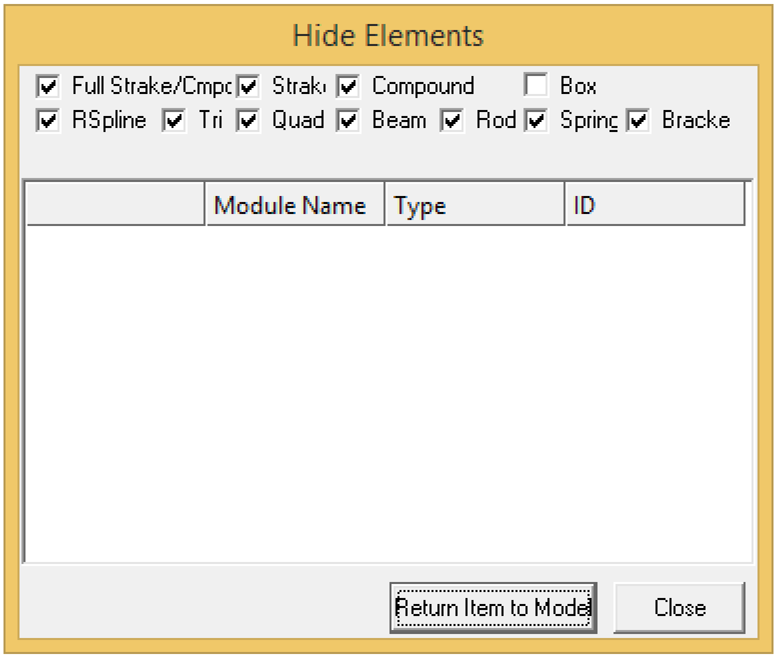
This option will color the elements corresponding to the angle that the element's local x-axis makes with the global z-axis.

© 2014 Optimum Structural Design, Inc.



Hide Elements

This option will open a dialog allowing the user to select elements to hide from the current view. There are a number of right-click selection options after an element is added to the dialog.



Options...

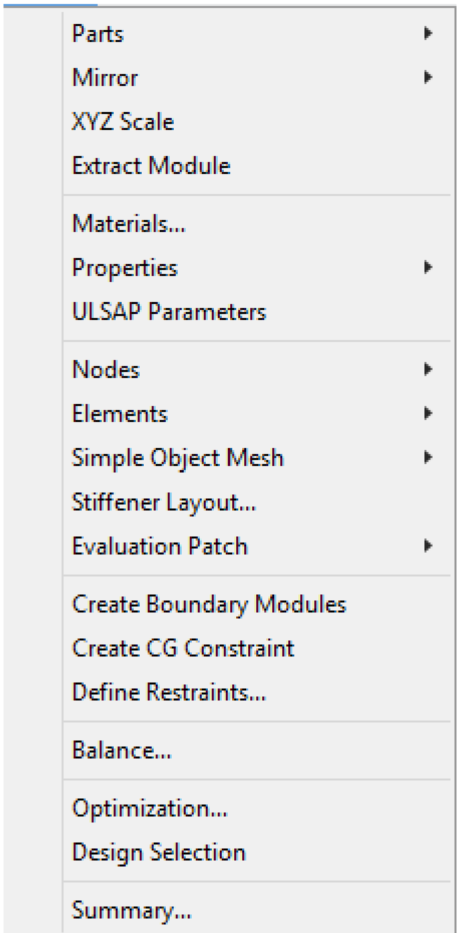
This option will launch the View Options dialog.

Refresh

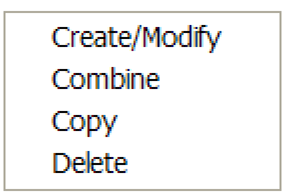
This option will allow the user to refresh the graphics.

4.5 Model Menu

The Model menu provides options to build, constrain and balance the model. A brief description of each option is discussed below.



Parts >



Create/Modify

This option will open the Parts dialog, where a structure or module can be created or modified. This is the same as clicking the parts icon .

Combine

This option will move the selected module beneath the substructure that is set as the current part. The module is selecting by clicking on it with the mouse in the modeling space. Note: a substructure must be set as the current part for this option to be allowed since a module cannot be added beneath another module.

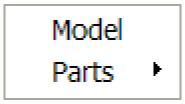
Copy

This option will create a copy of the selected part and allow the user to name and set the location of the new part. The part is selected by clicking on it in the modeling space.

Delete

This option is used to select a part in the modeling space to delete. This can also be done by right-clicking on a part in the parts tree and selecting *Delete.*

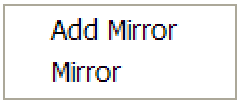
Mirror >



Model

This option will open the Mirror dialog which is used to mirror the model or add a mirror.

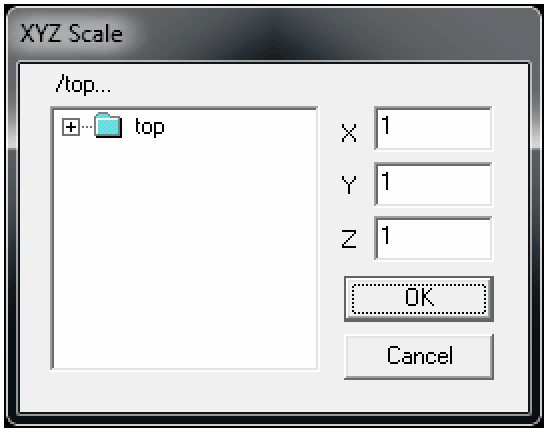
Parts >



This menu is used to mirror or add mirror a specific part. Once the option is selected from the menu, click on the part to mirror or add mirror and follow the prompts in the command tab.

XYZ Scale

This option allows one or more modules to be scaled in the X, Y, Z, or a combination of directions. This option will not affect material and element properties, it will simply scale the node locations in the direction(s) specified. Note: if the module or modules has a rotation defined, the scaled results may not be correct; this scaling is only performed in the translational directions.



The tree on the left side of the dialog allows the user to select the module, substructure or full model to scale. The X, Y and Z boxes are the scale factors for the selected part. Leaving the scale factor "1" will not affect the model in that direction.

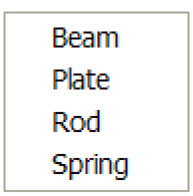
Extract Module

The *Extract Module* command will enable a module to be split into multiple pieces. The user must first define a bounding box in which the enclosed elements will be extracted. For more information see the section on extracting a module with the Extract Module command from imported NASTRAN geometry.

Materials...

This option will open the Materials dialog where materials can be created, modified and deleted.

Properties >



Beam

This option will open the properties dialog to the beam properties tab.

Plate

This option will open the properties dialog to the plate properties tab.

Rod

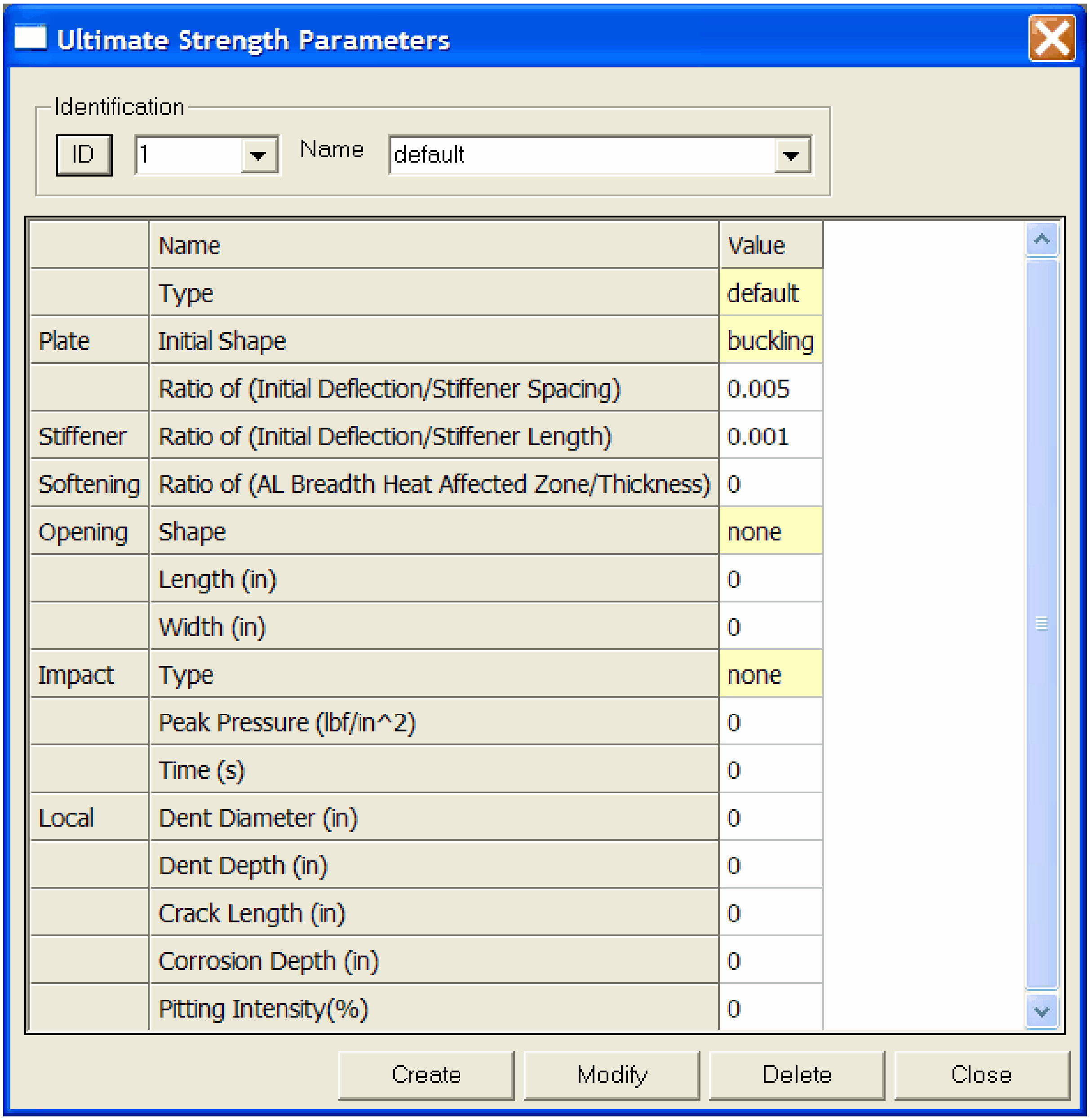
This option will open the properties dialog to the rod properties tab.

Spring

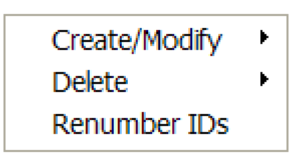
This option will open the properties dialog to the spring properties tab.

ULSAP Parameters

This option will open the Ultimate Strength Parameters dialog where sets of parameters can be created, modified and deleted.



Nodes >



Create/Modify >

This menu option is used to create or modify endpoints or additional nodes by opening the appropriate dialog box.

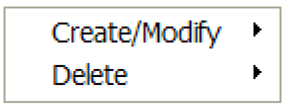
Delete >

This menu option is used to delete endpoints and additional nodes. Note: a module must be set as the current part in order to use this functionality. Once endpoint or additional node is selected, the user will be prompted to specify the ID of the endpoint or additional node to delete. Selecting unused nodes will delete any endpoints or additional nodes not being used to define an element.

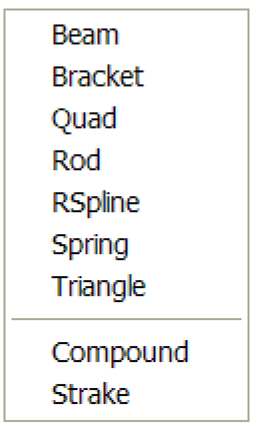
Renumber IDs

This option is used to renumber the node ID of the current modules endpoints and additional nodes. This will eliminate any gaps in ID numbers if nodes are deleted.

Elements >

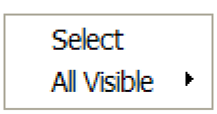


Create/Modify >



This menu is used to create or modify an additional element, compound, or strake and will open the appropriate dialog.

Delete >

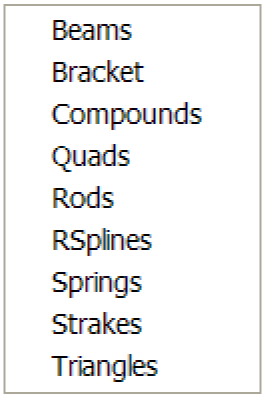


Select

This option will open the Deletions dialog.

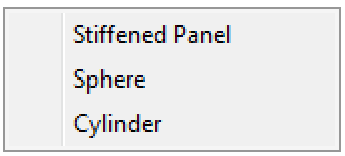
© 2014 Optimum Structural Design, Inc.

All Visible >

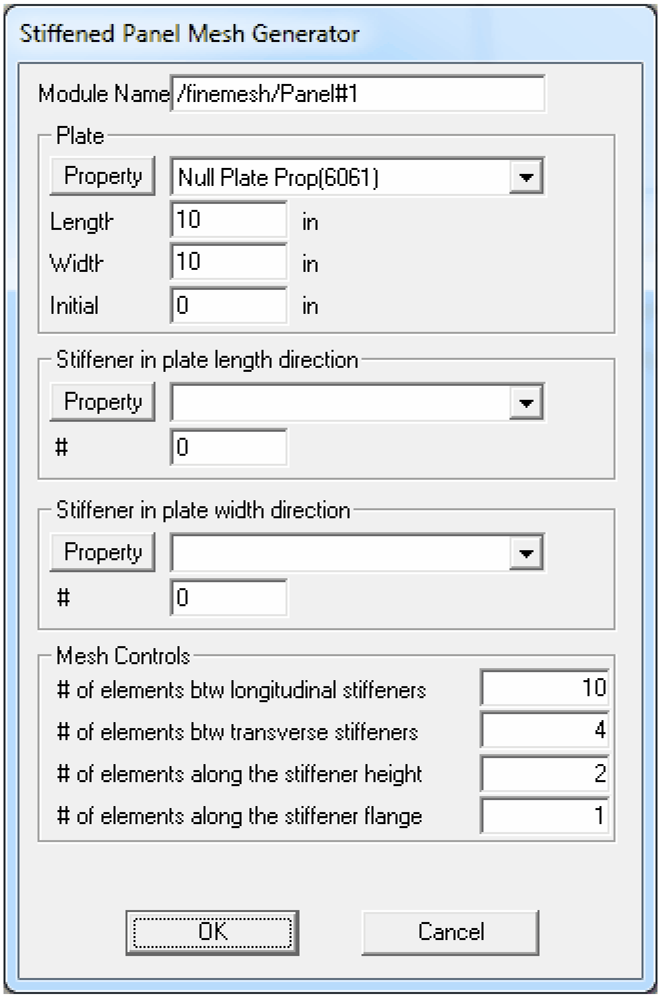


This option allows the user to delete all visible elements of a specific type by choosing the desired element from the menu.

Simple Object Mesh



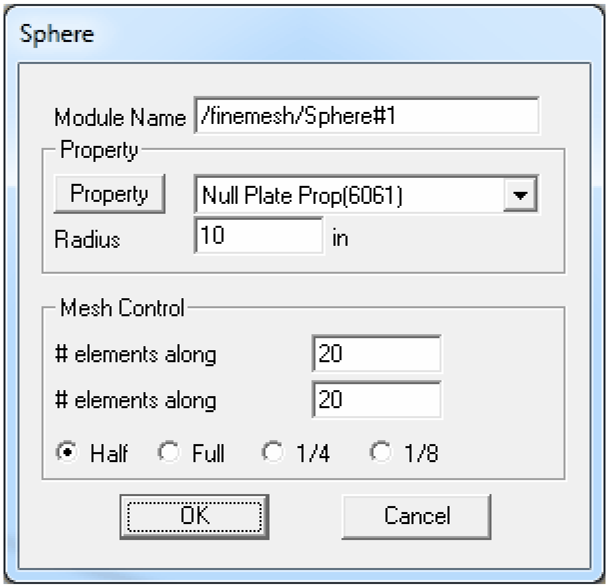
Stiffened Panel



This option allows the user to define and auto generate a stiffened panel. Options allow for the length, width, thickness and material of the plate as well as the stiffener property and number in each primary direction. Mesh controls also allow the user to define the number of elements between stiffeners as well as the number of elements to define the stiffener webs and flanges.

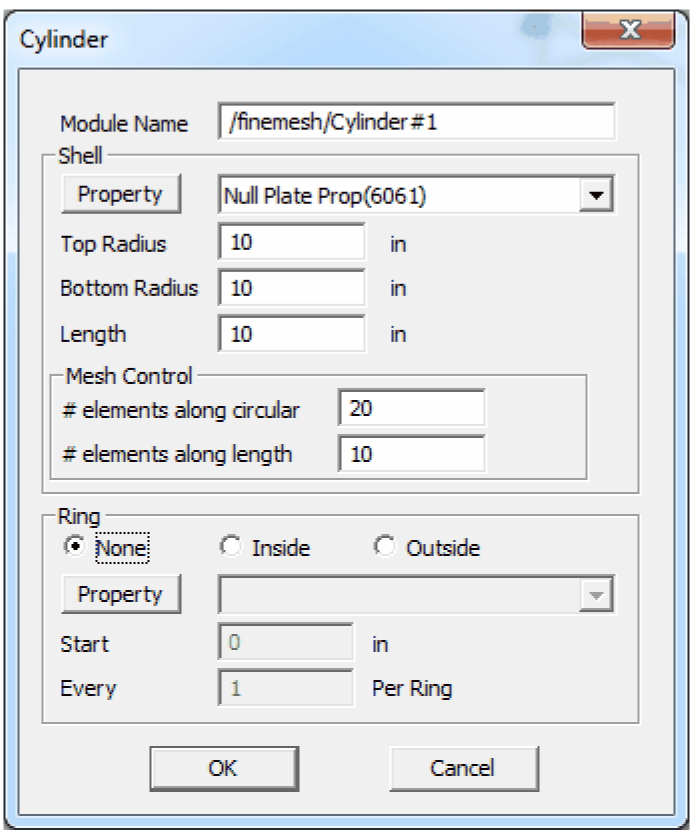
Sphere

© 2014 Optimum Structural Design, Inc.



This option allows the user to create a full, half, 1/4 or 1/8 sphere mesh with a given radius, material property and number of elements defining the mesh shape.

Cylinder



This option allows the user to create a cylinder mesh with a given property, top and bottom radii, length and mesh controls. The user can also add rings on the inside or outside of the cylinder at a given spacing.

Stiffener Layout...

This option will open the Stiffener Layout dialog where stiffener layouts can be created, modified and deleted.

Evaluation Patch >

Auto-Generate
Create/Evaluate
Delete All

Auto-Generate

This option will automatically create evaluation patches from the model's elements.

Create/Evaluate

This option will launch the Evaluation Patch dialog.

Delete All

This option will delete all evaluation patches defined for the model.

Create Boundary Modules

This option allows the user to automatically create a master node at the neutral axis connected to all the module's end nodes by RBE2 elements in support of End Moment load patterns. This will ensure that the plane sections remain plane. The End Moment load is now applied at the master node versus spreading the imposed forces and moments to the existing nodes. Note: this option is only valid with a "Cut" model.

Create CG Restraint

This option will automatically create a fixed restraint at the model's CG connected to surrounding structure with soft springs.

Define Restraints...

This option opens the Restraints dialog.

© 2014 Optimum Structural Design, Inc.

Balance...

This option opens the Balance dialog.

Optimization...

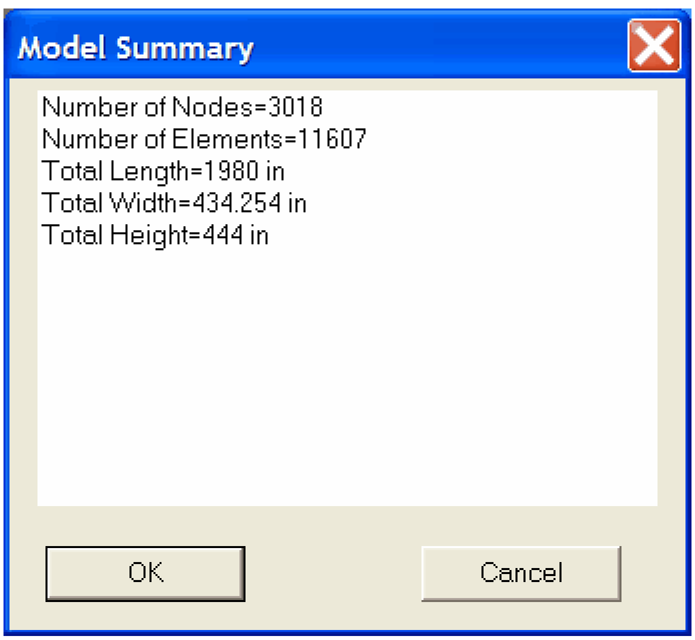
This option will open the ULS Optimization dialog.

Design Selection...

This option will show the list of optimized design results and allow the user to select one, if available.

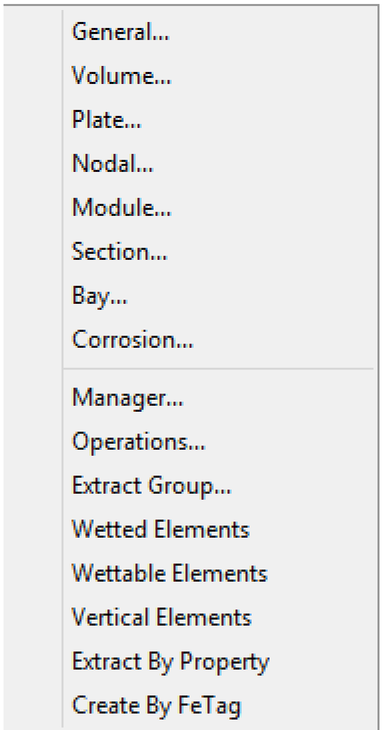
Summary...

This options opens the Model Summary dialog providing the total number of nodes and elements as well as the total modeled length, width, and height.



4.6 Groups Menu

The Groups menu provides options to create, modify and delete groups as well as perform operations on existing groups.



General...

This option opens the groups dialog to the General tab.

Volume...

This option opens the groups dialog to the Volume tab.

Plate...

This option opens the groups dialog to the Plate tab.

Nodal...

This option opens the groups dialog to the Node tab.

Module...

This option opens the groups dialog to the Module tab.

Section...

© 2014 Optimum Structural Design, Inc.

This option opens the groups dialog to the Section tab.

Bay...

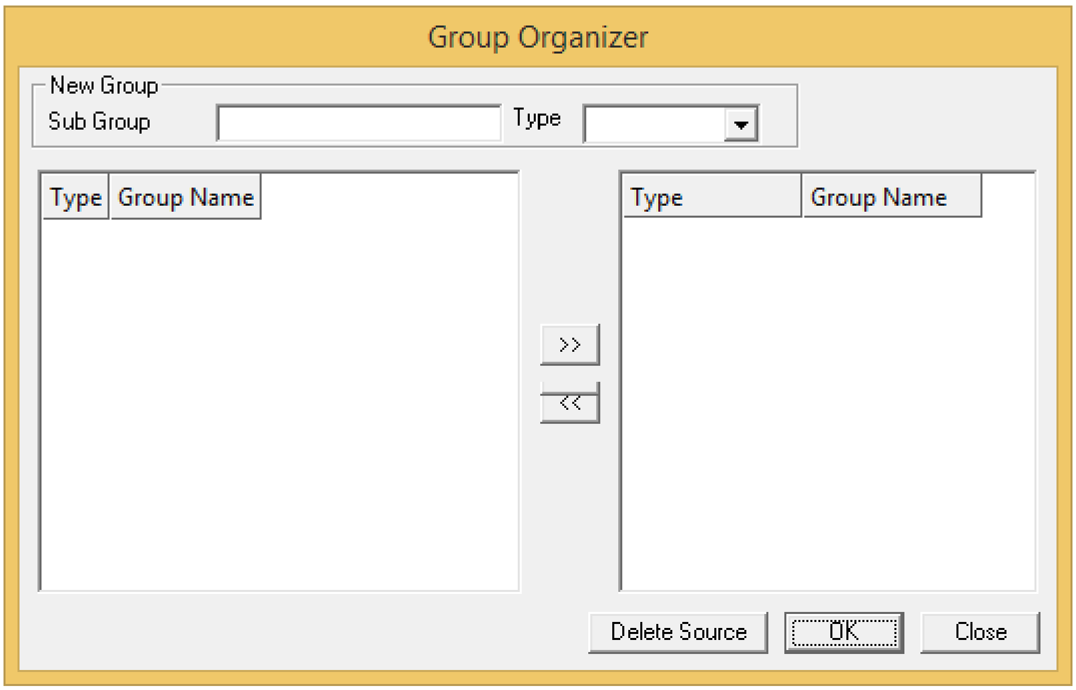
This option opens the groups dialog to the Bay tab.

Corrosion...

This option opens the groups dialog to the Corrosion tab.

Manager...

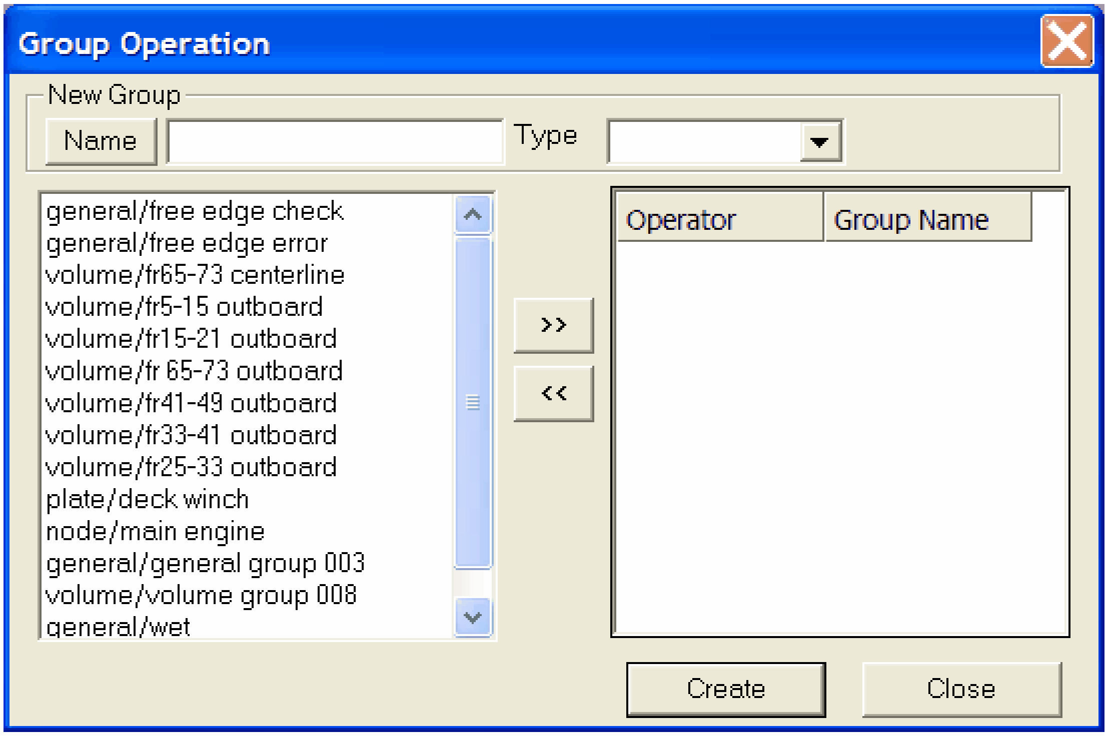
This option will open the Group Manager dialog.



The Group Manager will list all of the groups and their type in the left hand side. Groups can be moved into new subgroups or have their type changed by selecting options from the new group section and moving the existing groups to the right-hand side of the dialog.

Operations...

This option will open the Group Operation dialog.



Existing groups can be combined and subtracted to create new groups using this dialog. A new group of shared elements between existing groups can also be created.

1. Move the groups of interest into the right column by clicking the group and then clicking the >> button.
2. Select the operator for each group added.

+: this will add the elements of the selected group to the new group (no elements will be double counted)

-: this will exclude the elements of the selected group from the new group

&: this will add the common elements of the selected groups to the new group

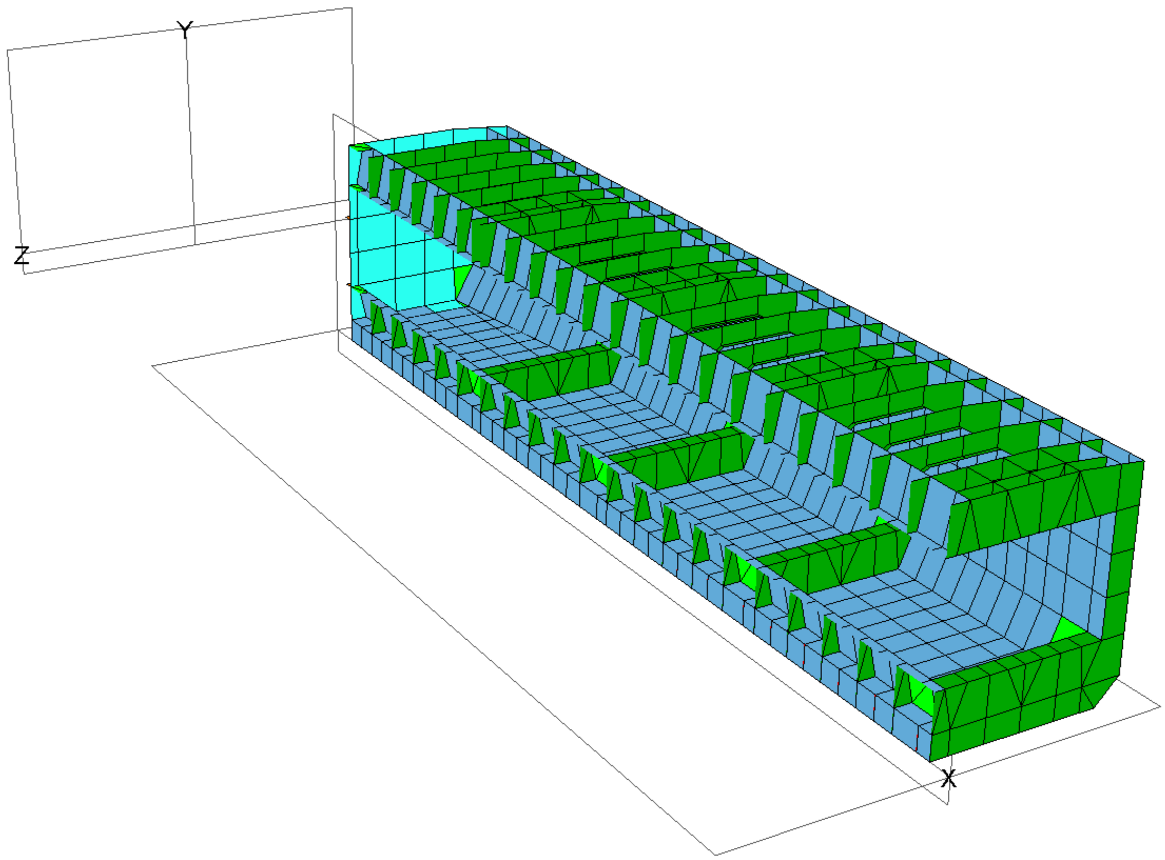
Note: The "-" sign should not be taken literally as a subtraction. If a group is subtracted from the new group and then a group containing elements that were previously subtracted is added, the elements originally excluded by selecting the "-" operator will not be added subsequently.

3. Select the group type from the drop-down menu.
4. Give the new group a name.
5. Click Create.

© 2014 Optimum Structural Design, Inc.

Example: Creating Group of Parallel Midbody Stripped of Transverse Structure

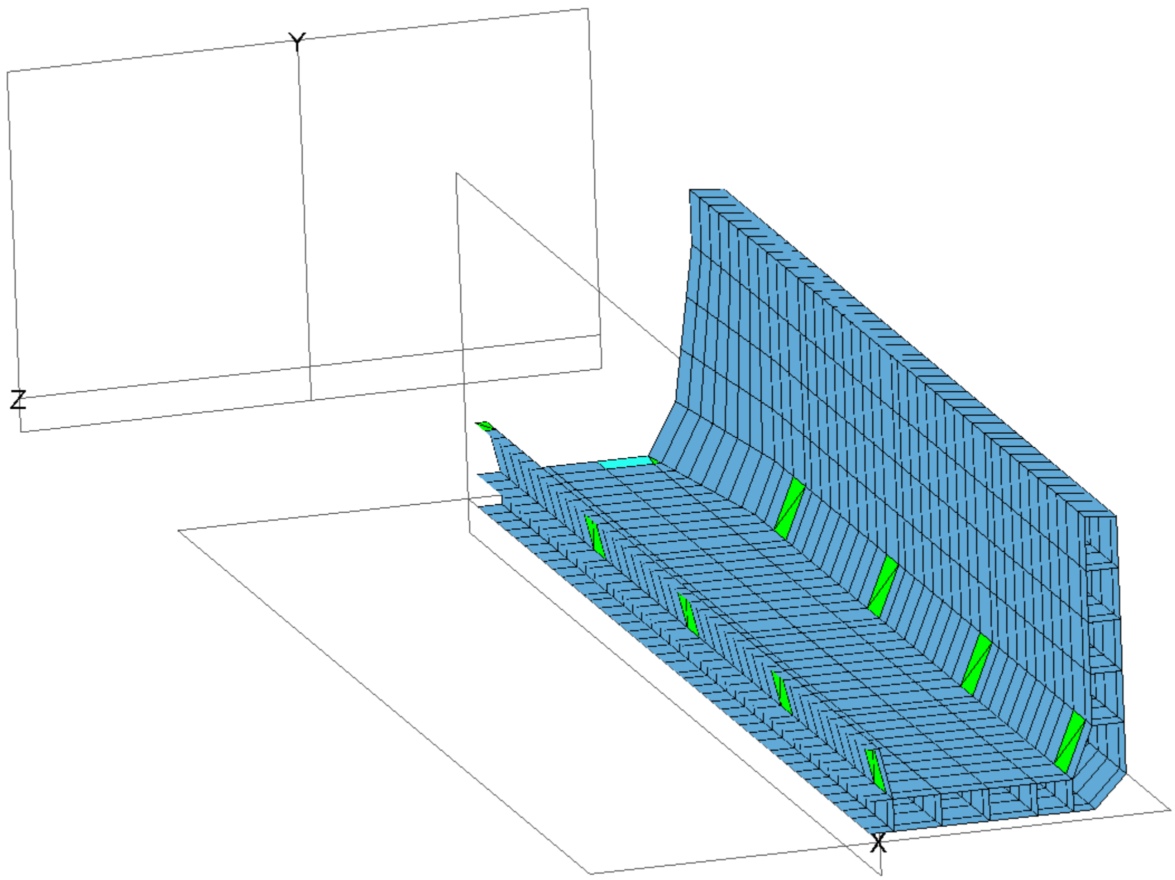
Say, for example, a general group consists of all parallel midbody and the goal is to preserve the inner bottom and inner hull ballast tanks but to strip the module entirely of transverse framing and bulkheads. The initial module group looks like the following, which is a half hull with deck removed to show transverse structure:



Again, the goal is to remove web frames, transverse frames, and the transverse bulkhead. As opposed to deleting all compound elements from the general group, it would be easier to create a general group or a plate group of all transverse structure. This can be done simply by entering the groups dialog and checking the "Full Strake/Full Compound" option in either the general groups tab or the plate groups tab. Click an element of the transverse structure which will be indicated by green elements. Right-click in the dialog and select "Add By Type". All compound elements, including the transverse frames and web frames should be included in this new group with type "cmpd". Set the current view to the group containing all modules that make up the parallel mid-body. The aft-most station contains a transverse bulkhead that needs to be eliminated. Create either a plate group or general group consisting of the quad elements which can be all selected using the right-click "Add By Type" option. Since all of the shell plating and inner bottom is modeled with quad strakes, only the true quad elements of the bulkhead will be selected. Individually select any remaining bar elements or tri elements that are part of the bulkhead.

© 2014 Optimum Structural Design, Inc.

The group operations dialog can now be used to exclude all transverse structure from the above picture. Select Groups > Operations... and the Group Operation dialog will appear. The left hand column lists all groups that are eligible for group operations. The right hand column will be populated with the groups and operations explained in the Operations section. The main group group containing all structure in the midbody should be given a "+" sign to add it to the operations. The transverse bulkhead and structure groups should be excluded by assigning a "-" to each of them. The dialog should appear as the following. The final group, devoid of all transverse structure, should appear as the following.



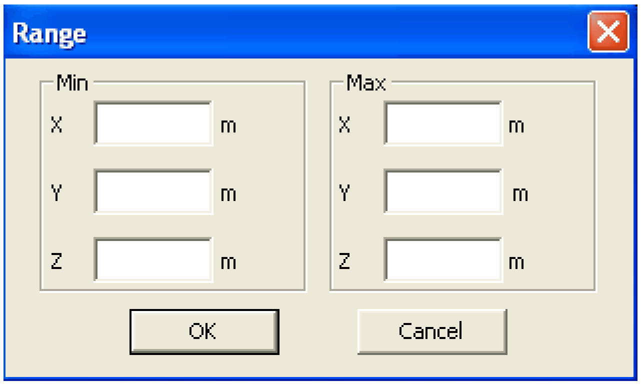
Example: Create a Nodal Group of Nodes Associated with a Plate Group

The group operations dialog can be used to add groups together or exclude selected elements of compatible group types (ie. plate group and general group addition). The operations dialog can alternatively be used to gather nodes from elements into a group. Say, there exists a plate group or a general group of some elements. The goal is to extract the nodes that are associated with the vertices of each element into a separate nodal group. This can be done simply by adding the plate or general group to the right hand column and giving it a plus sign. ID the group as type "Nodal" and the result will be a nodal group of all nodes associated with those elements. This can be leveraged to assist the loading of the model.

© 2014 Optimum Structural Design, Inc.

Extract Group...

This option allows the user to define a region in space in which all elements will be extracted into a general group called "extract". The extract group option will only extract elements from the current view and visible elements. The default is to extract all visible elements and can be selected by entering nothing in the range for extraction and then clicking "OK". The extract group dialog where bounds for extraction are defined looks like the following:



Wetted Elements

This option will create a general group named "wetted#lc**x**" (where **x** is the load case number) of all shell elements defined as wettable that will receive hydrostatic pressure from the given load case's load pattern.

Wettable Elements

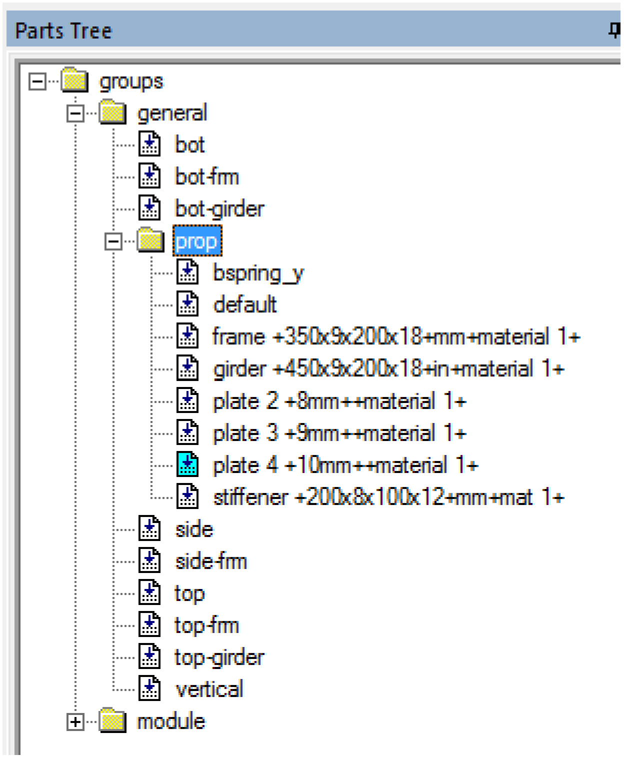
This option will create a general group named "wettable" of all the shell elements that are defined as wettable.

Vertical Elements

This option will automatically create a general group of the "vertical" plate elements defined as being within the threshold value of the vertical, or MAESTRO Y-axis. This option provides a quick way to select well supported ship structure that can be loaded using the Target Load loading pattern. This is especially useful with finer mesh models that may have several plate elements between frames.

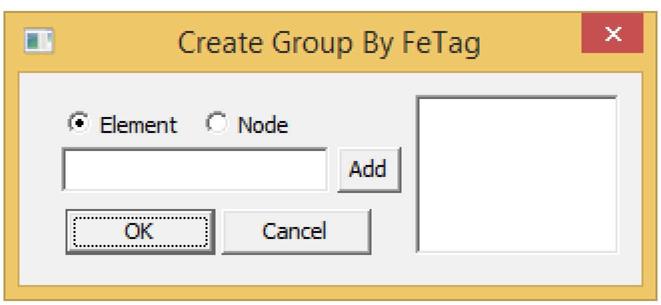
Extract By Property

This option will automatically create general groups of all the element properties defined and those elements which are assigned that property.



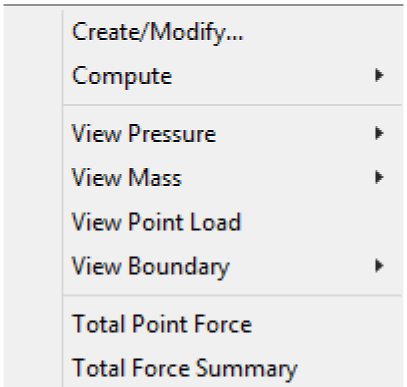
Create by FeTag

This option will open a dialog allowing a user to create a group by element or nodal FE Tag. Multiple FE Tags can be entered at once separated by commas.



4.7 Loads Menu

The Loads menu allows the user to create or modify a load case, as well as view the resulting pressure and forces for the selected load case. A brief description of each option is discussed below.

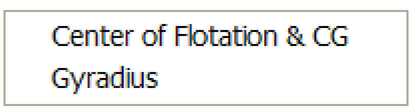


Create/Modify...

This option will launch the Loads dialog where load cases can be created, modified, and deleted.

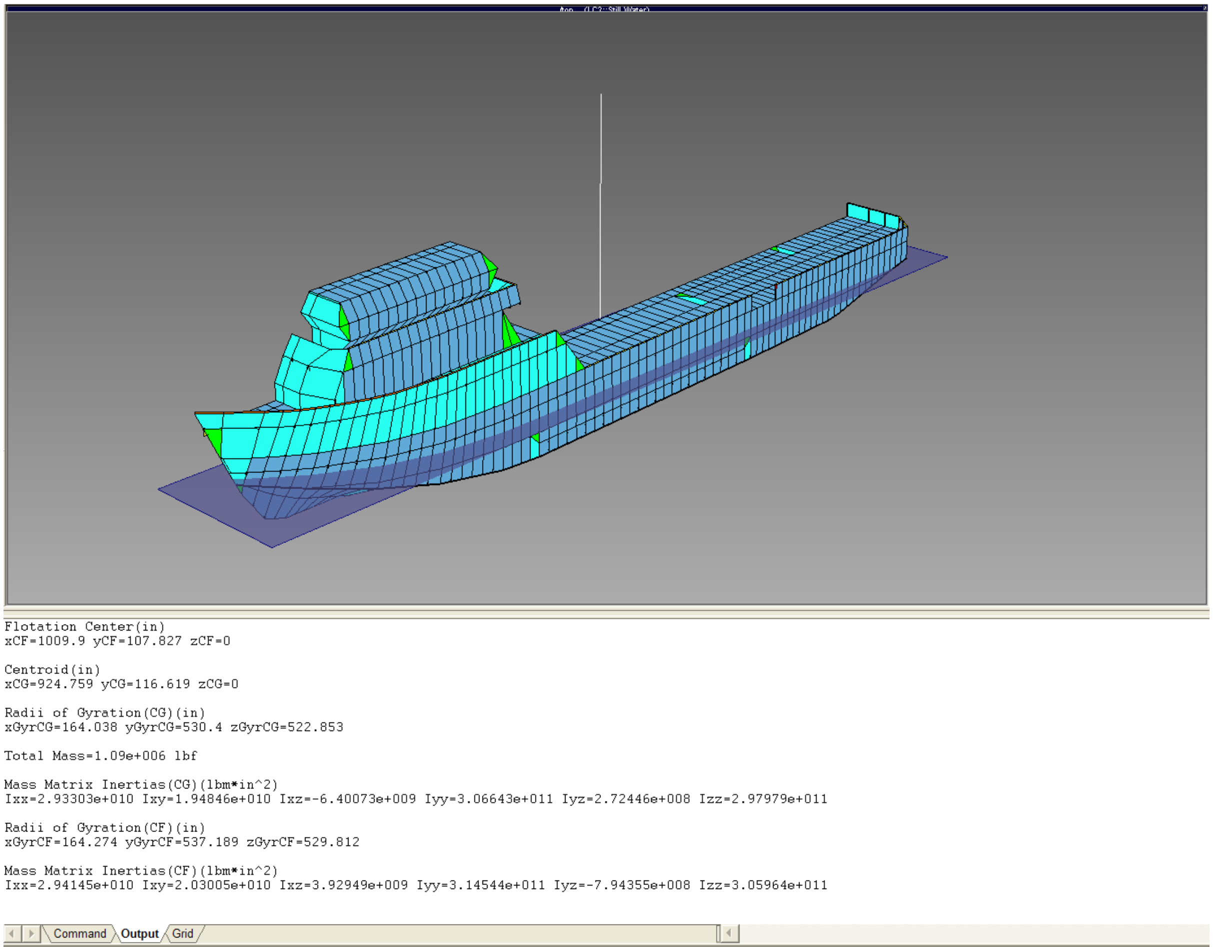
Compute >

The Compute menu has two options to calculate parameters of the modeled structure.   A detailed description of each of these options is below:



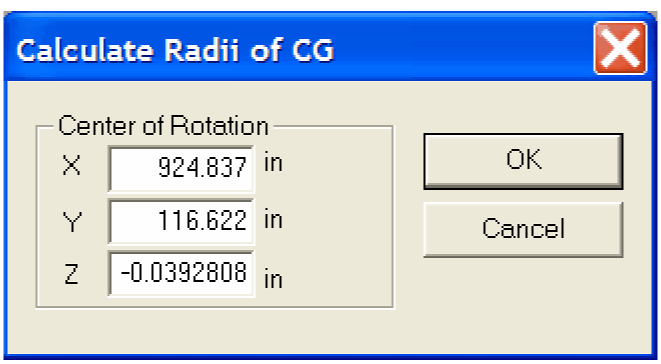
Center of Flotation & CG

This option will calculate and output the center of flotation and center of gravity to the output tab as well as graphically display the waterplane and CG and CF locations.



Gyradius

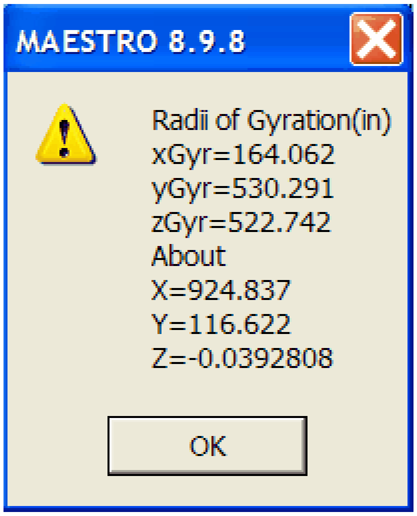
This option will compute the radius of gyration about the user-defined center of rotation. Once selected a dialog will launch prompting the user to input the center of rotation.



After the center of rotation is defined, click OK. The results will be displayed in a new dialog

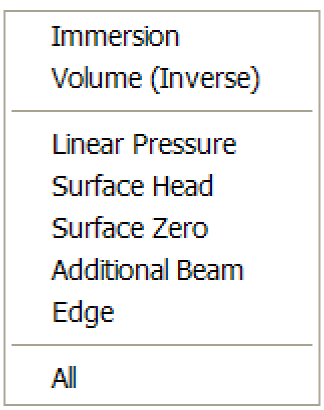
© 2014 Optimum Structural Design, Inc.

box as well as be echoed to the Output tab.



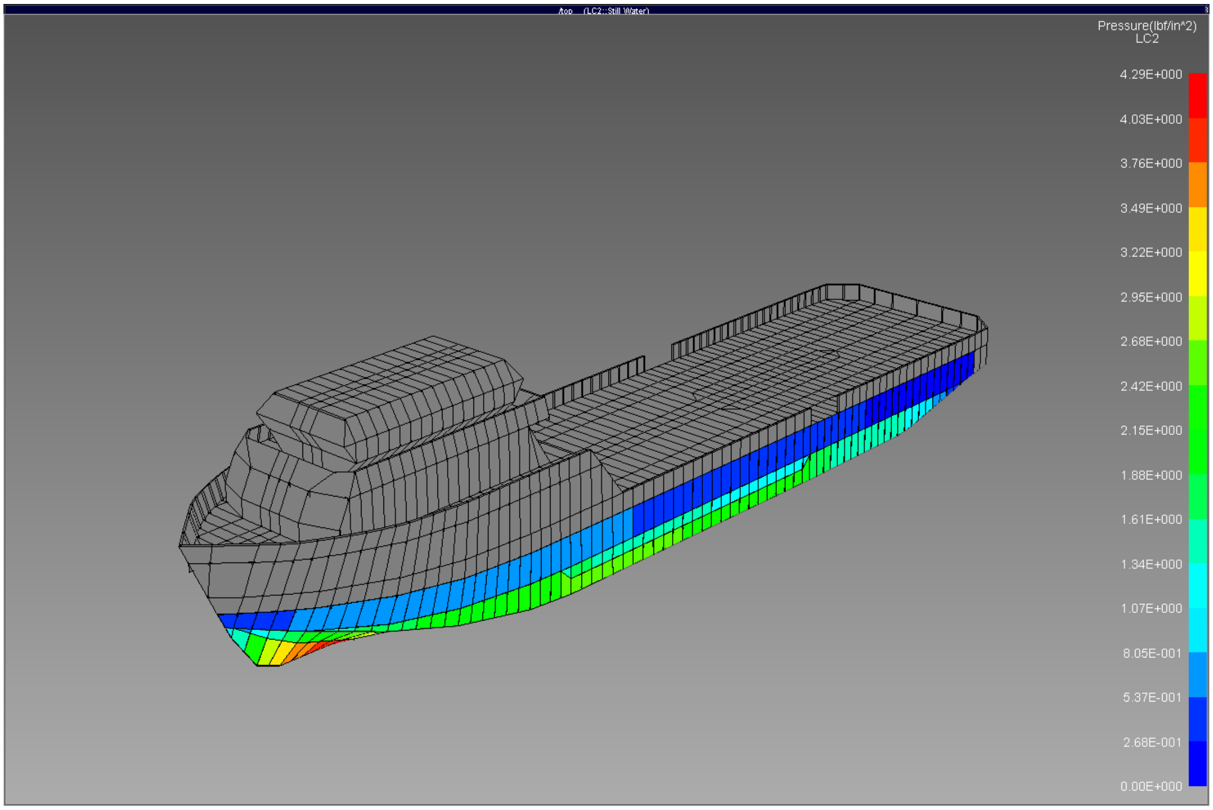
View Pressure >

The sub-menu allows the user to choose to view the pressure due to various loading options, or all, for the currently selected load case. Note: the scale for the graphical representation of pressure will remain the same if switching between load cases, unless the maximum pressure value is higher than the current scale's maximum value. To view the load case specific legend, turn off the pressure view and then turn it on again.



Immersion

This option will display the pressure on the "wetted" panel elements based on the initial location of the model's waterplane for the currently selected load case.



The dynamic query icon  can be used to query an element to recover the pressure and double-clicked to echo the information to the output tab. (Note: While quad elements are defined with nodes 1, 2, 3, and 4, the dynamic query will present Pressure[0], Pressure [1], Pressure[2], and Pressure[4]. Node 1 corresponds to Pressure[0] and so forth.)

Note well that if the pressure is zero at a node, then the query will not display the pressure value for that node.

Volume (Inverse)

This option will display the pressure due to all volume loads for the active load case on the inverse side of the plate so that the pressure display can be seen easily. The dynamic query icon  can be used to query an element to recover the pressure and double-clicked to echo the information to the output tab.

Linear Pressure

This option will display the pressure of the panel elements for which a linear pressure is defined for the currently selected load case. The dynamic query icon  can be used to query an element to recover the pressure and double-clicked to echo the information to the output tab.

Surface Head

This option will display the pressure of the panel elements for which a surface head is defined for the currently selected load case. The dynamic query icon can be used to query an element to recover the pressure and double-clicked to echo the information to the output tab.

Surface Zero

This option will display the pressure of the panel elements for which a surface zero is defined for the currently selected load case. The dynamic query icon can be used to query an element to recover the pressure and double-clicked to echo the information to the output tab.

Additional Beam

This option will display the pressure of the additional beam elements for which a pressure is imposed for the currently selected load case. The dynamic query icon can be used to query a beam element to recover the pressure and double-clicked to echo the information to the output tab.

Edge

This option will highlight the edge of elements where an additional stress is applied. Similar to the free edges view, the edges where a stress is applied will show up in red.

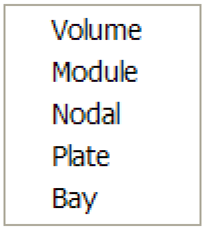
All

This option will display the net pressure on the elements for the currently selected load case. The dynamic query icon can be used to query an element to recover the pressure and double-clicked to echo the information to the output tab.

View Mass >

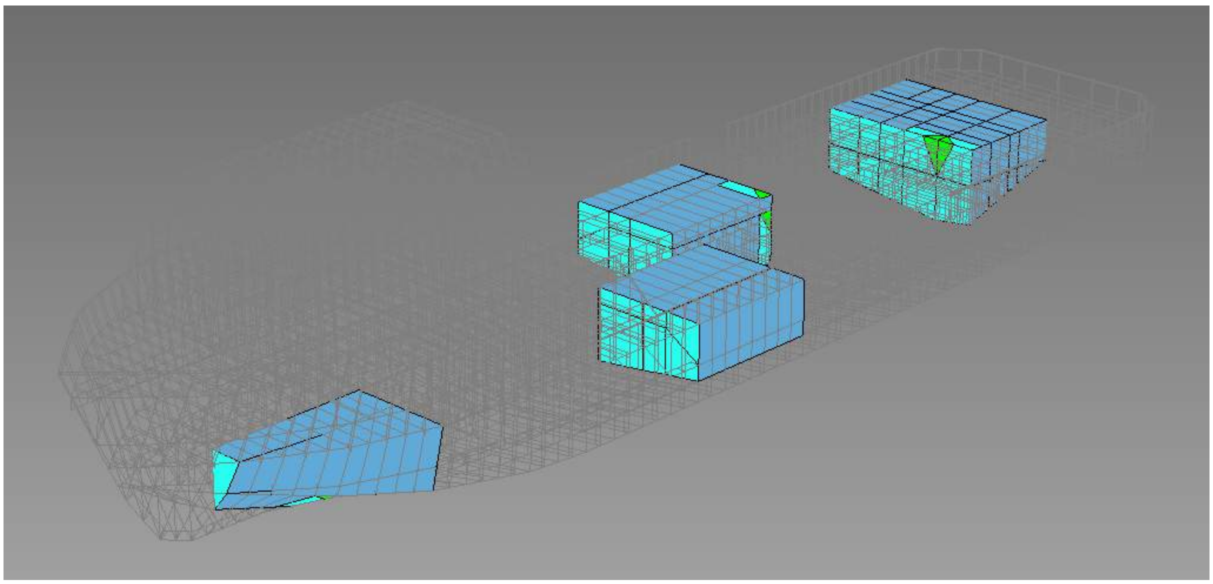
This sub-menu allows the user to view the different mass loads that may be imposed on the model for the currently selected load case.

© 2014 Optimum Structural Design, Inc.



Volume

This option will display the volume groups that are loaded for the currently selected load case.



Module

This option will show the modules and their corresponding elements which are added to the load case either as a structural weight or a scaled weight for the currently selected load case.

Nodal

This option will display the nodes which are applied a mass in the currently selected load case.

View Point Load

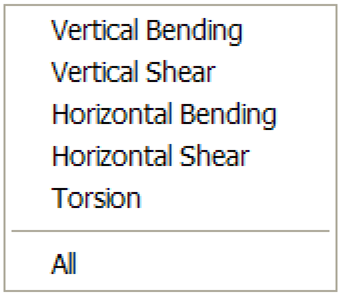
This option will give a graphic representation of the directional load applied at each node(s).

The dynamic query  icon can be used to query the force in each direction at the selected node.

© 2014 Optimum Structural Design, Inc.

View Boundary

Note: this option is for cut models only.



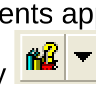
Vertical Bending

This option will graphically display the boundary nodal forces and moments applied to simulate the user-defined vertical end moments. The dynamic query  icon can be used to query the force and moments at individual nodes.

Vertical Shear

This option will graphically display the boundary nodal forces and moments applied to simulate the user-defined vertical shear force or the vertical shear force calculated by MAESTRO to balance the model. The dynamic query  icon can be used to query the force and moments at individual nodes.

Horizontal Bending

This option will graphically display the boundary nodal forces and moments applied to simulate the user-defined horizontal end moments. The dynamic query  icon can be used to query the force and moments at individual nodes. Note: this is only applicable if *transverse symmetry* is not checked in the Job Information dialog.

Horizontal Shear

This option will graphically display the boundary nodal forces and moments applied to simulate the user-defined horizontal shear force or the horizontal shear force calculated by MAESTRO to balance the model. The dynamic query  icon can be used to query the force and moments at individual nodes. Note: this is only applicable if *transverse symmetry* is not checked in the Job Information dialog.

Torsion

© 2014 Optimum Structural Design, Inc.

This option will graphically display the boundary nodal forces and moments applied to simulate the user-defined torsional moments. The dynamic query  icon can be used to query the force and moments at individual nodes. Note: this is only applicable if *transverse symmetry* is not checked in the Job Information dialog.

All

This option will graphically display all the boundary nodal forces and moments applied to the model representing the vertical bending, vertical shear, horizontal bending, horizontal shear, and torsion. The dynamic query  icon can be used to query the force and moments at individual nodes.

Total Point Force

This option provides a graphical representation of the equivalent nodal force for the currently selected load case. The dynamic query  icon can be used to query a specific node and that node can be double-clicked to echo the information to the Output tab.

Total Force Summary

This option will print out the total combined point loads for the current view part in the Output tab.

4.8 Hull Menu

The Hull menu is used to display various model properties, which can then be queried. This provides the user with a graphical method to quickly and accurately identify trends. Because graphics alone are not sufficient, a majority of these menu items produce distribution data, in the form of text in the Output window for further scrutiny. A brief description of each option is discussed below.

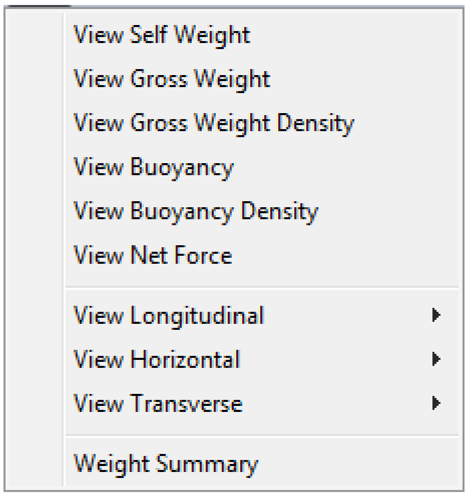
Note:

If transverse symmetry is checked in the Job Information dialog, the results shown

© 2014 Optimum Structural Design, Inc.

will be for the full model, not just the half modeled.

In conjunction with the Hull menu, the user can use the Dynamic Query tool, which can be initiated via the  icon, to query the distribution graph. To use this functionality the user must select the Hull menu item of choice, toggle the Dynamic Query icon, and move the mouse cursor over a particular portion of the graph. This will produce a text box with data for the highlighted section of the distribution. Further, the user can echo this information to the Output window by double-clicking the left mouse button while hovering over the graph entity of interest.



View Self Weight

View Gross Weight

View Gross Weight Density

View Buoyancy

View Buoyancy Density

View Net Force

View Longitudinal >

View Horizontal >

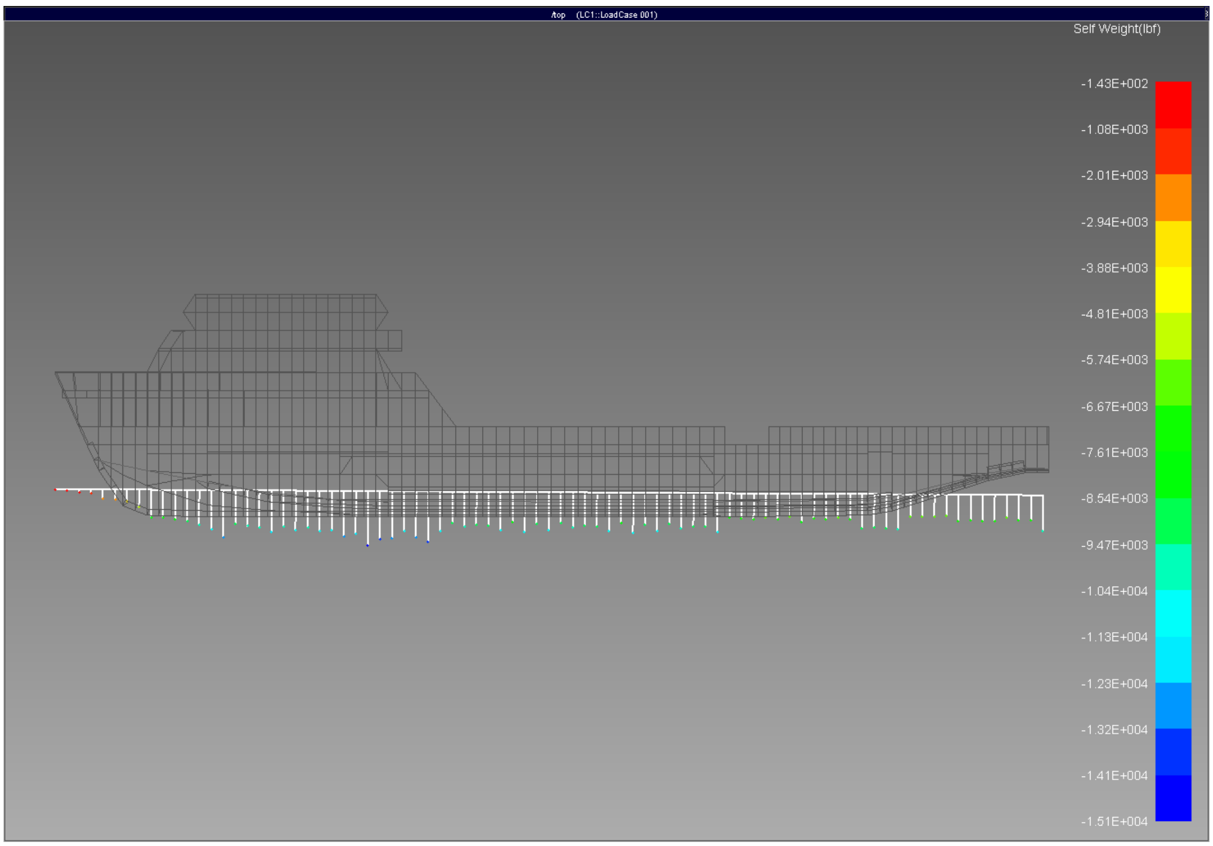
View Transverse >

Weight Summary

View Self Weight

The **View Self Weight** command under the the Hull menu is used to display the MAESTRO calculated "modeled" weight. The term "modeled" weight refers to the weight calculated by MAESTRO based on the materials and elements that make up the FE model. As shown below, MAESTRO produces a display of this weight distribution. Further, MAESTRO echoes this distribution to the Output window, which is found at the bottom of the MAESTRO GUI.

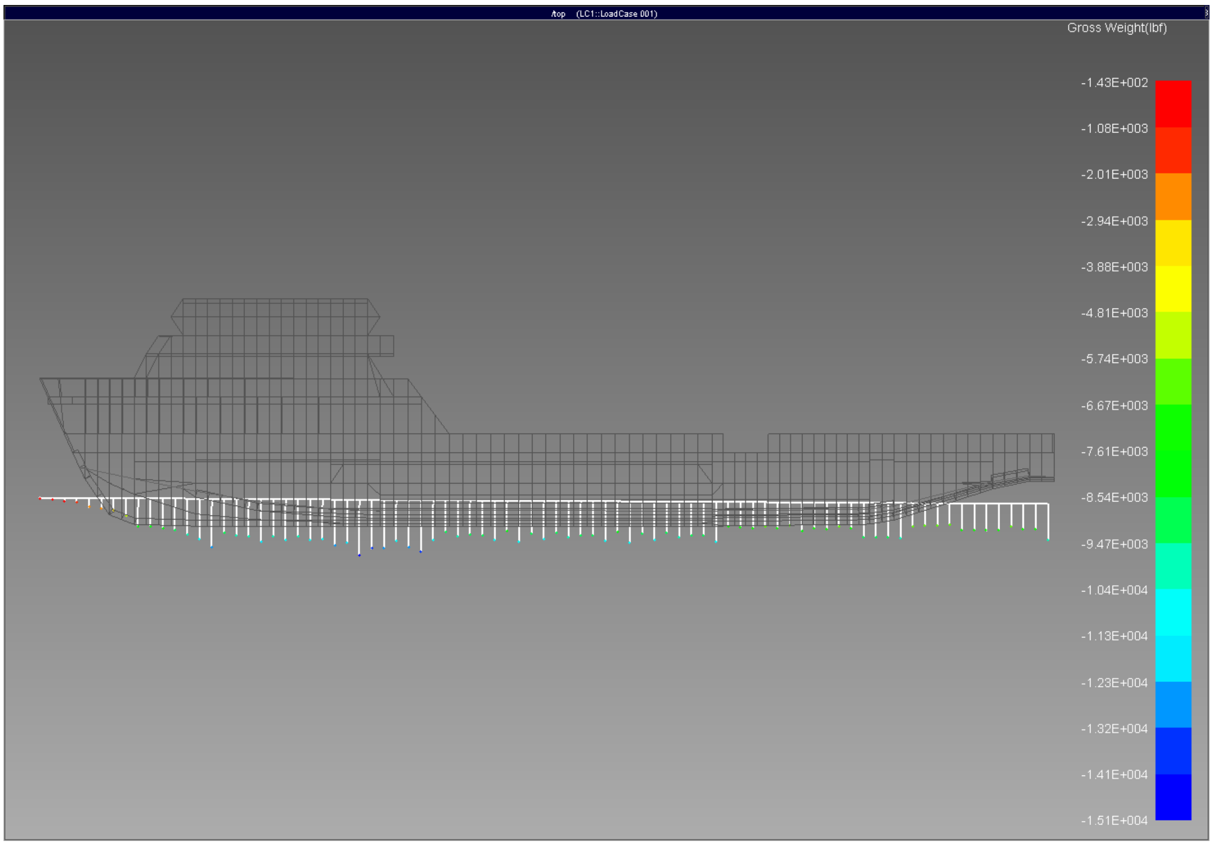
In conjunction with the View Self Weight command, the user can use the Dynamic Query tool, which can be initiated via the  icon, to query the distribution graph. To use this functionality the user must select the View Self Weight command, toggle the Dynamic Query icon, and move the mouse cursor over a particular portion of the graph. This will produce a text box with graph data. The user can echo this information to the Output window by double-clicking the left mouse button while hovering over the graph entity of interest.



View Gross Weight

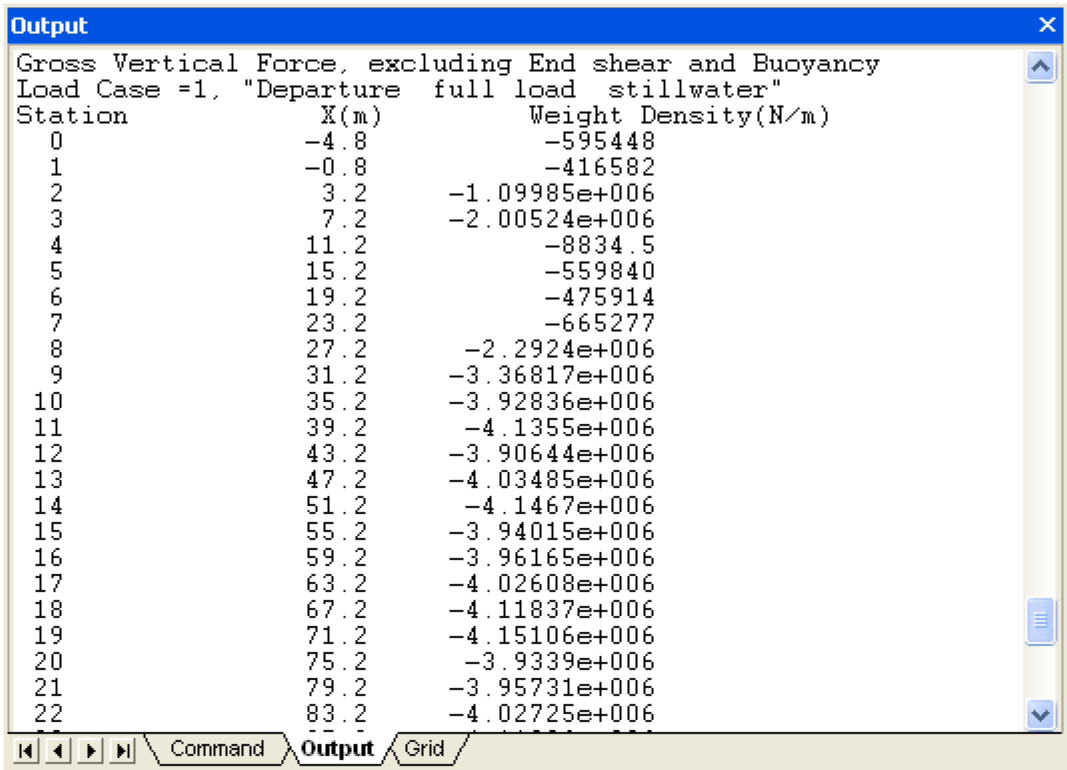
The **View Gross Weight** command under the the Hull menu is used to display the FE model's gross weight for the selected load case. As shown below, MAESTRO produces a display of this weight distribution. Further, MAESTRO echoes this distribution to the Output window, which is found at the bottom of the MAESTRO GUI.

In conjunction with the View Gross Weight command, the user can use the Dynamic Query tool, which can be initiated via the  icon, to query the distribution graph. To use this functionality the user must select the View Gross Weight command, toggle the Dynamic Query icon, and move the mouse cursor over a particular portion of the graph. This will produce a text box with graph data. The user can echo this information to the Output window by double-clicking the left mouse button while hovering over the graph entity of interest.



View Gross Weight Density

The **View Gross Weight Density** command under the Hull menu is used to show the weight per unit length. The output in the output tab looks like the following with a weight density at each station:



Output

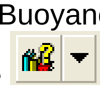
Gross Vertical Force, excluding End shear and Buoyancy
Load Case =1, "Departure full load stillwater"

Station	X(m)	Weight Density(N/m)
0	-4.8	-595448
1	-0.8	-416582
2	3.2	-1.09985e+006
3	7.2	-2.00524e+006
4	11.2	-8834.5
5	15.2	-559840
6	19.2	-475914
7	23.2	-665277
8	27.2	-2.2924e+006
9	31.2	-3.36817e+006
10	35.2	-3.92836e+006
11	39.2	-4.1355e+006
12	43.2	-3.90644e+006
13	47.2	-4.03485e+006
14	51.2	-4.1467e+006
15	55.2	-3.94015e+006
16	59.2	-3.96165e+006
17	63.2	-4.02608e+006
18	67.2	-4.11837e+006
19	71.2	-4.15106e+006
20	75.2	-3.9339e+006
21	79.2	-3.95731e+006
22	83.2	-4.02725e+006

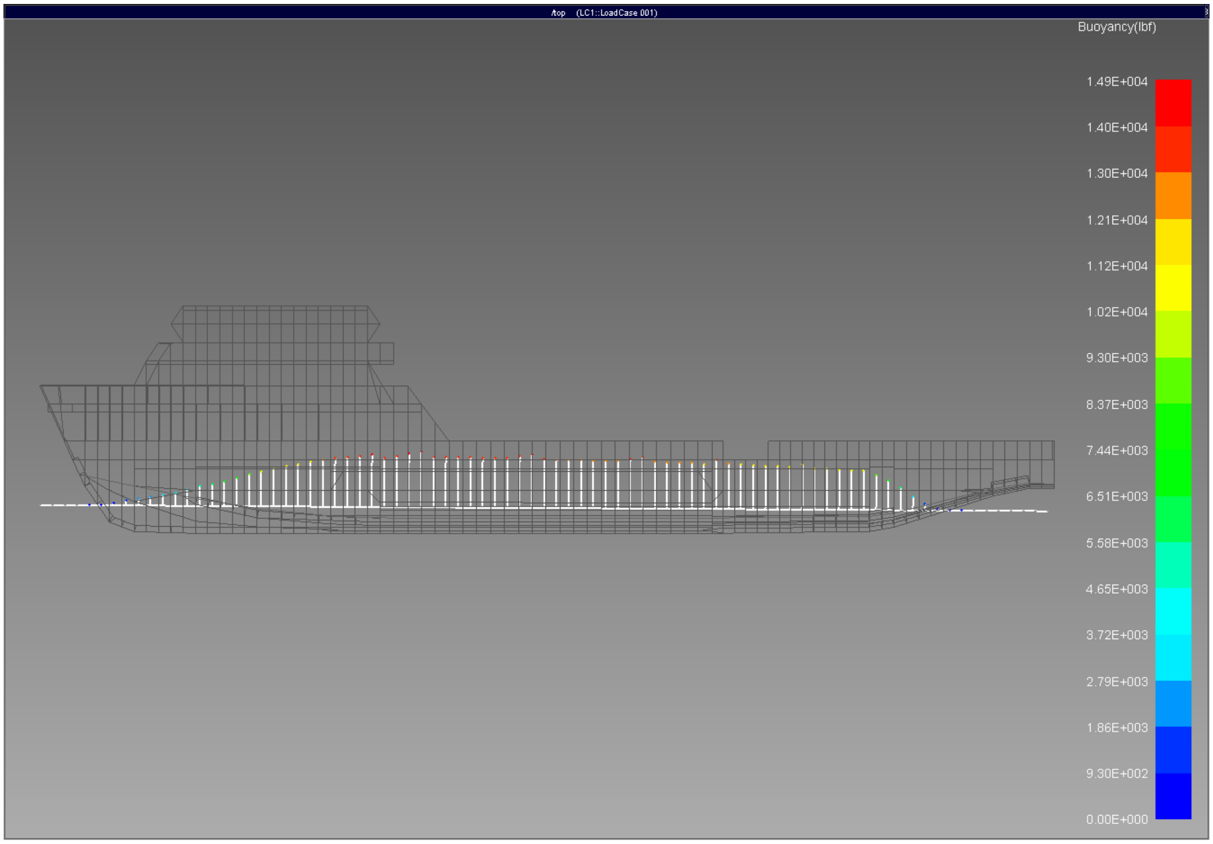
Command Output Grid

View Buoyancy

The **View Buoyancy** command under the the Hull menu is used to display the FE model's buoyancy distribution for the selected load case, as shown below. Further, MAESTRO echoes this distribution to the Output window, which is found at the bottom of the MAESTRO GUI.

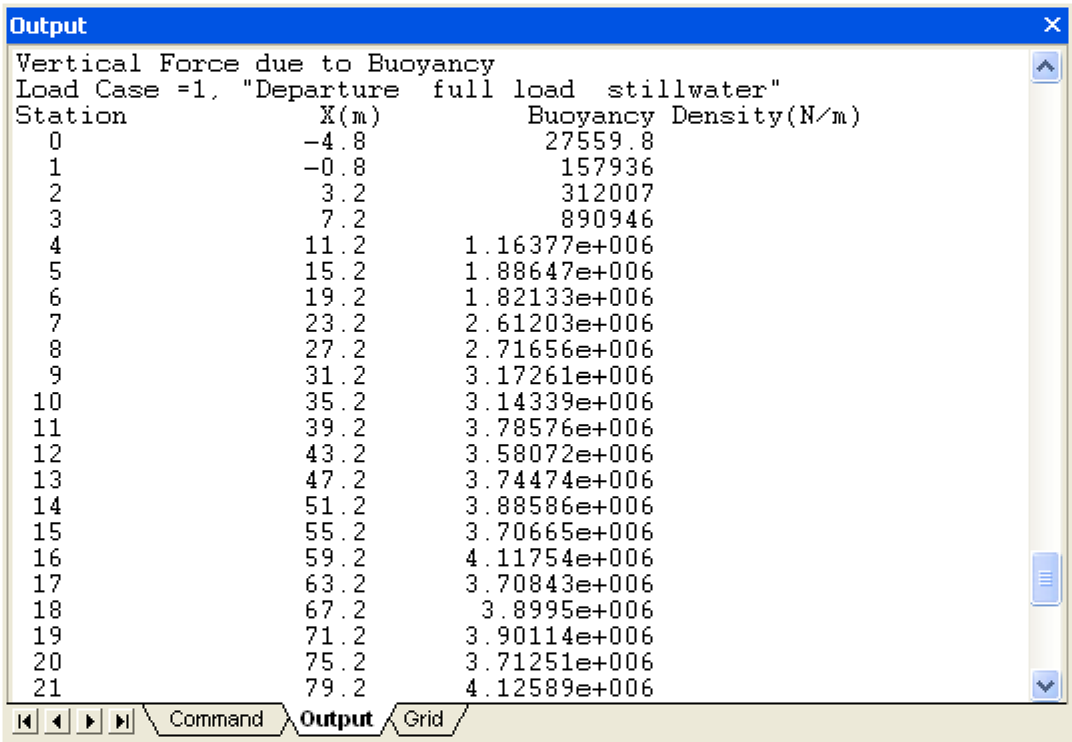
In conjunction with the View Buoyancy command, the user can use the Dynamic Query tool, which can be initiated via the  icon, to query the distribution graph. To use this functionality the user must select the View Buoyancy command, toggle the Dynamic Query icon, and move the mouse cursor over a particular portion of the graph. This will produce a text box with graph data. The user can echo this information to the Output window by double-clicking the left mouse button while hovering over the graph entity of interest.

© 2014 Optimum Structural Design, Inc.



View Buoyancy Density

The **View Buoyancy Density** command under the Hull menu is used to show the buoyancy per linear distance. The output in the output tab looks like the following with a buoyancy density at each station:

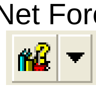

Output

Vertical Force due to Buoyancy
Load Case =1, "Departure full load stillwater"

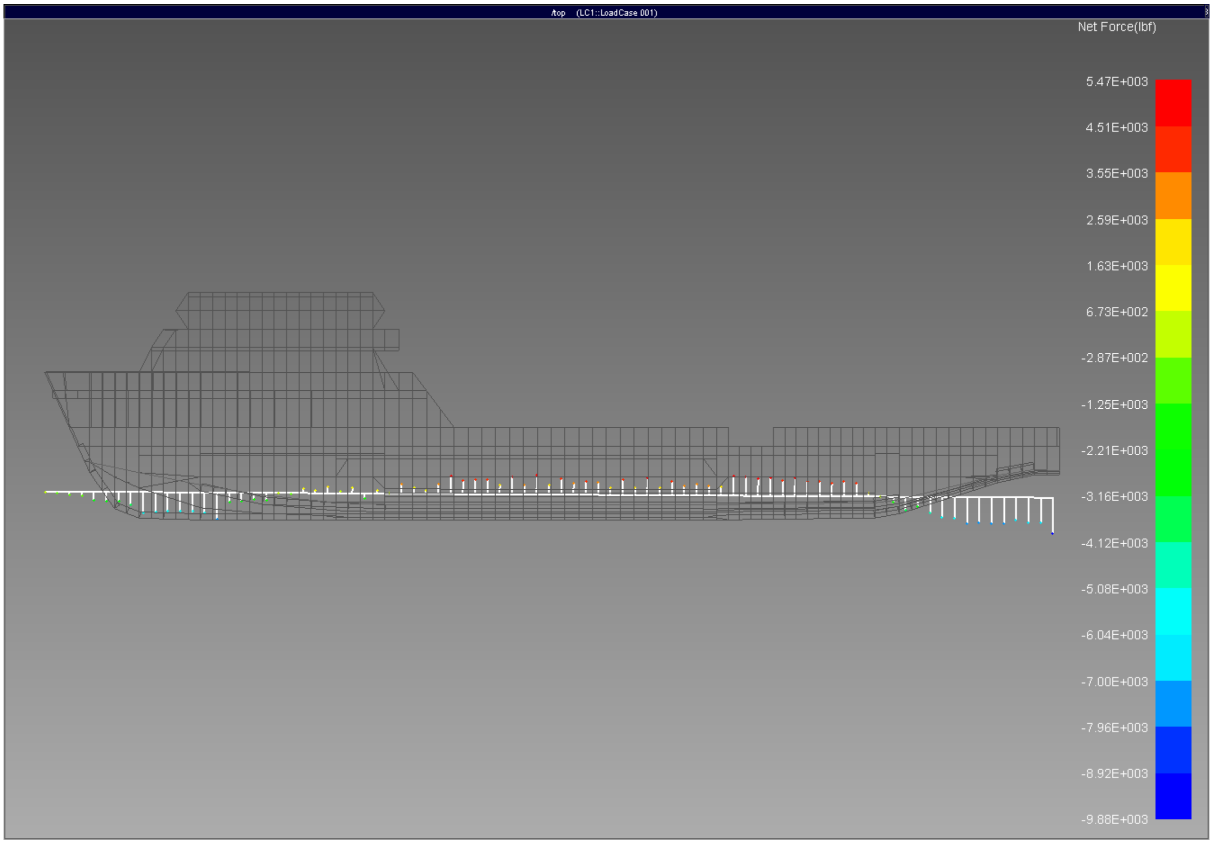
Station	X(m)	Buoyancy Density(N/m)
0	-4.8	27559.8
1	-0.8	157936
2	3.2	312007
3	7.2	890946
4	11.2	1.16377e+006
5	15.2	1.88647e+006
6	19.2	1.82133e+006
7	23.2	2.61203e+006
8	27.2	2.71656e+006
9	31.2	3.17261e+006
10	35.2	3.14339e+006
11	39.2	3.78576e+006
12	43.2	3.58072e+006
13	47.2	3.74474e+006
14	51.2	3.88586e+006
15	55.2	3.70665e+006
16	59.2	4.11754e+006
17	63.2	3.70843e+006
18	67.2	3.8995e+006
19	71.2	3.90114e+006
20	75.2	3.71251e+006
21	79.2	4.12589e+006

View Net Force

The **View Net Force** command under the the Hull menu is used to display the FE model's net force distribution for the selected load case, as shown below. MAESTRO also echoes this distribution to the Output window, which is found at the bottom of the MAESTRO GUI.

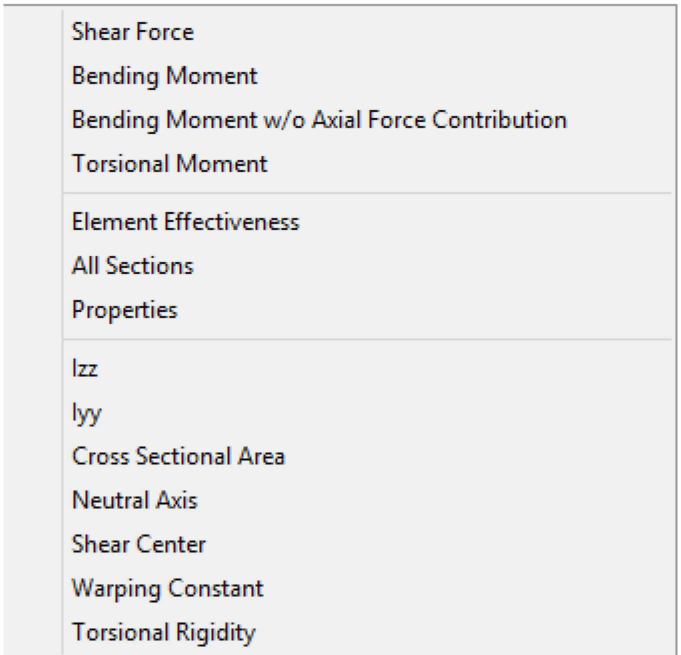
In conjunction with the View Net Force command, the user can use the Dynamic Query tool, which can be initiated via the  icon, to query the distribution graph. To use this functionality the user must select the View Net Force command, toggle the Dynamic Query icon, and move the mouse cursor over a particular portion of the graph. This will produce a text box with graph data. The user can echo this information to the Output window by double-clicking the left mouse button while hovering over the graph entity of interest.

© 2014 Optimum Structural Design, Inc.



View Longitudinal >

The View Longitudinal menu has several options to display the longitudinal properties of the modeled structure.   A detailed description of each of these options is below:



Shear Force

Bending Moment

Bending Moment w/o Axial Force Contribution

Torsional Moment

Element Effectiveness

All Sections

Properties

Izz

Iyy

Cross Sectional Area

Neutral Axis

Shear Center

Warping Constant

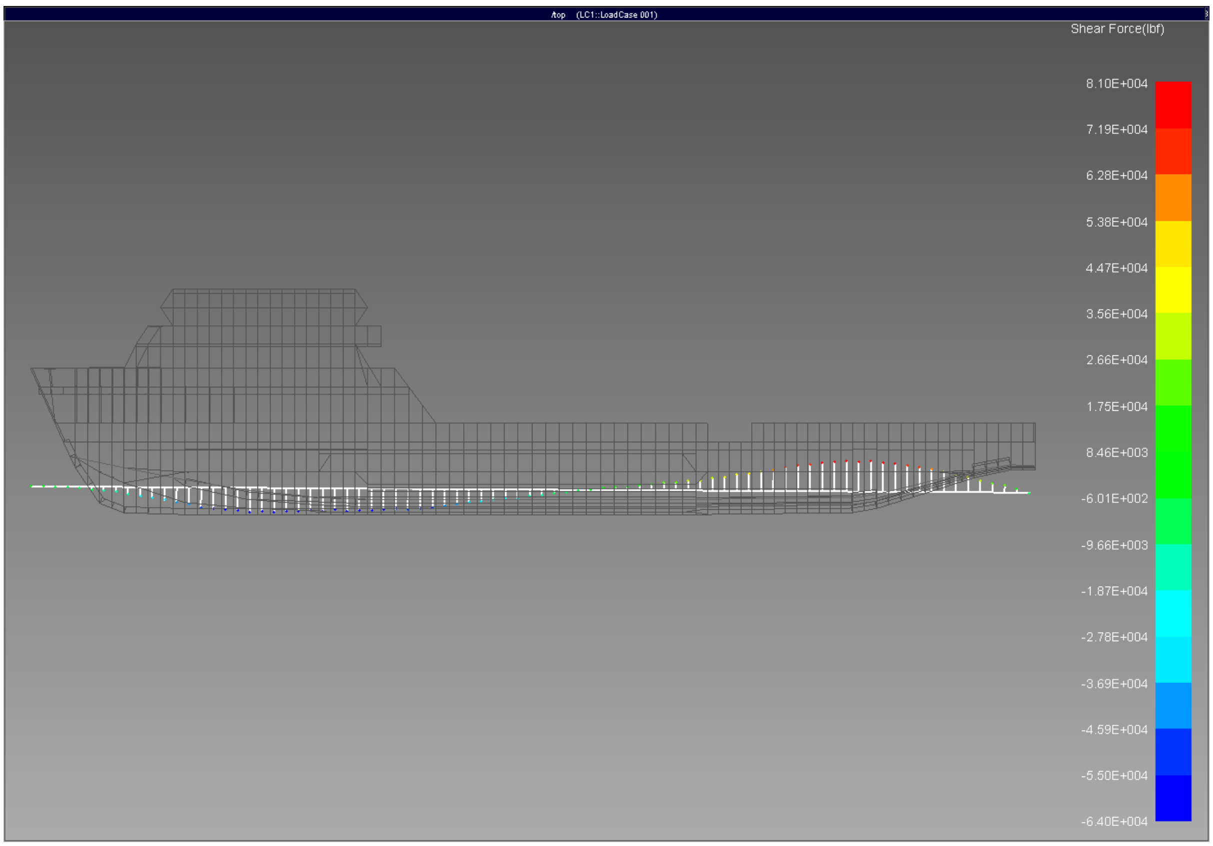
Torsional Rigidity

Shear Force

© 2014 Optimum Structural Design, Inc.

The **View Longitudinal > Shear Force** command under the the Hull menu is used to display the FE model's longitudinal shear force distribution, as shown below. MAESTRO also echoes this distribution to the Output window, which is found at the bottom of the MAESTRO GUI.

In conjunction with the View Longitudinal Shear Force command, the user can use the Dynamic Query, which can be initiated via the  icon, to query the distribution graph. To use this functionality the user must select the View Shear Force command, toggle the Dynamic Query icon, and move the mouse cursor over a particular portion of the graph. This will produce a text box with graph data. Further, the user can echo this information to the Output window by double-clicking the left mouse button while hovering over the graph entity of interest.

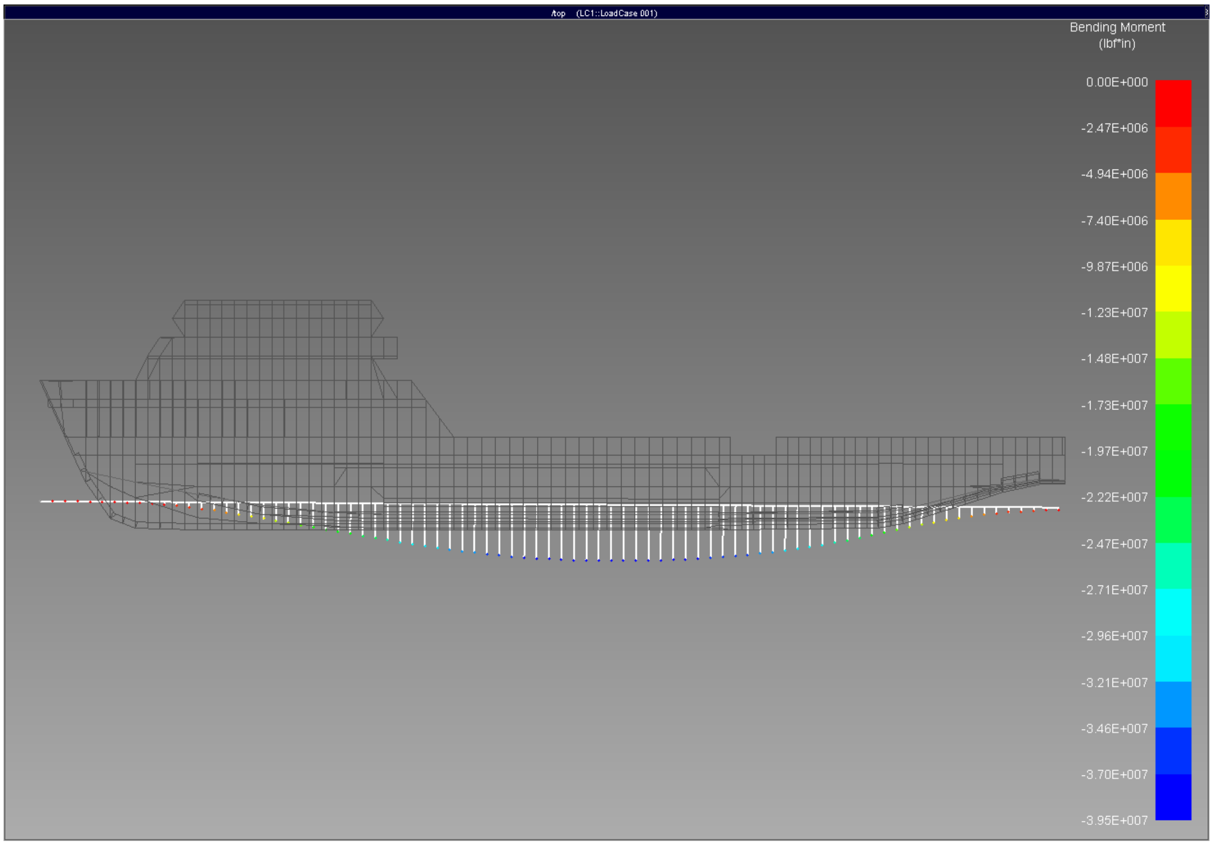


Bending Moment

The **View Longitudinal > Bending Moment** command under the the Hull menu is used to display the FE model's longitudinal bending moment distribution, as shown below. MAESTRO also echoes this distribution to the Output window, which is found at the bottom of the MAESTRO GUI.

In conjunction with the View Longitudinal Bending Moment command, the user can use the

Dynamic Query, which can be initiated via the  icon, to query the distribution graph. To use this functionality the user must select the View Bending Moment command, toggle the Dynamic Query icon, and move the mouse cursor over a particular portion of the graph. This will produce a text box with graph data. Further, the user can echo this information to the Output window by double-clicking the left mouse button while hovering over the graph entity of interest.



Bending Moment w/o Axial Force Contribution

The **View Longitudinal > Bending Moment w/o Axial Force Contribution** command under the the Hull menu is used to display the FE model's longitudinal bending moment distribution ignoring the contribution from axial forces (e.g., hydrostatic forces on a flat stern). MAESTRO also echoes this distribution to the Output window, which is found at the bottom of the MAESTRO GUI.

In conjunction with the View Longitudinal Bending Moment w/o Axial Force Contribution command, the user can use the Dynamic Query, which can be initiated via the  icon, to query the distribution graph. To use this functionality the user must select the View Bending Moment command, toggle the Dynamic Query icon, and move the mouse cursor over a particular portion of the graph. This will produce a text box with graph data. Further, the user can echo this information to the Output window by double-clicking the left mouse

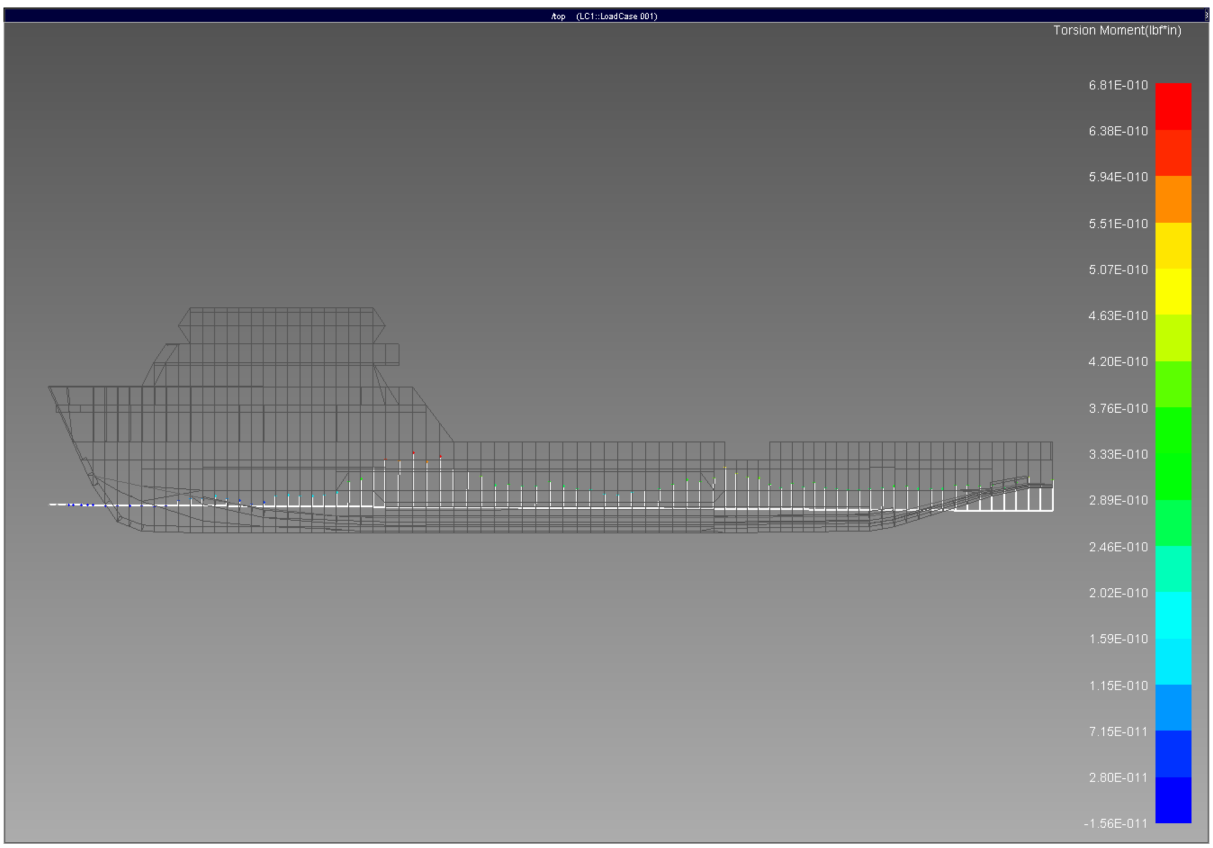
© 2014 Optimum Structural Design, Inc.

button while hovering over the graph entity of interest.

Torsional Moment

The **View Longitudinal > Torsional Moment** command under the the Hull menu is used to display the FE model's longitudinal torsional moment distribution, as shown below. MAESTRO also echoes this distribution to the Output window, which is found at the bottom of the MAESTRO GUI.

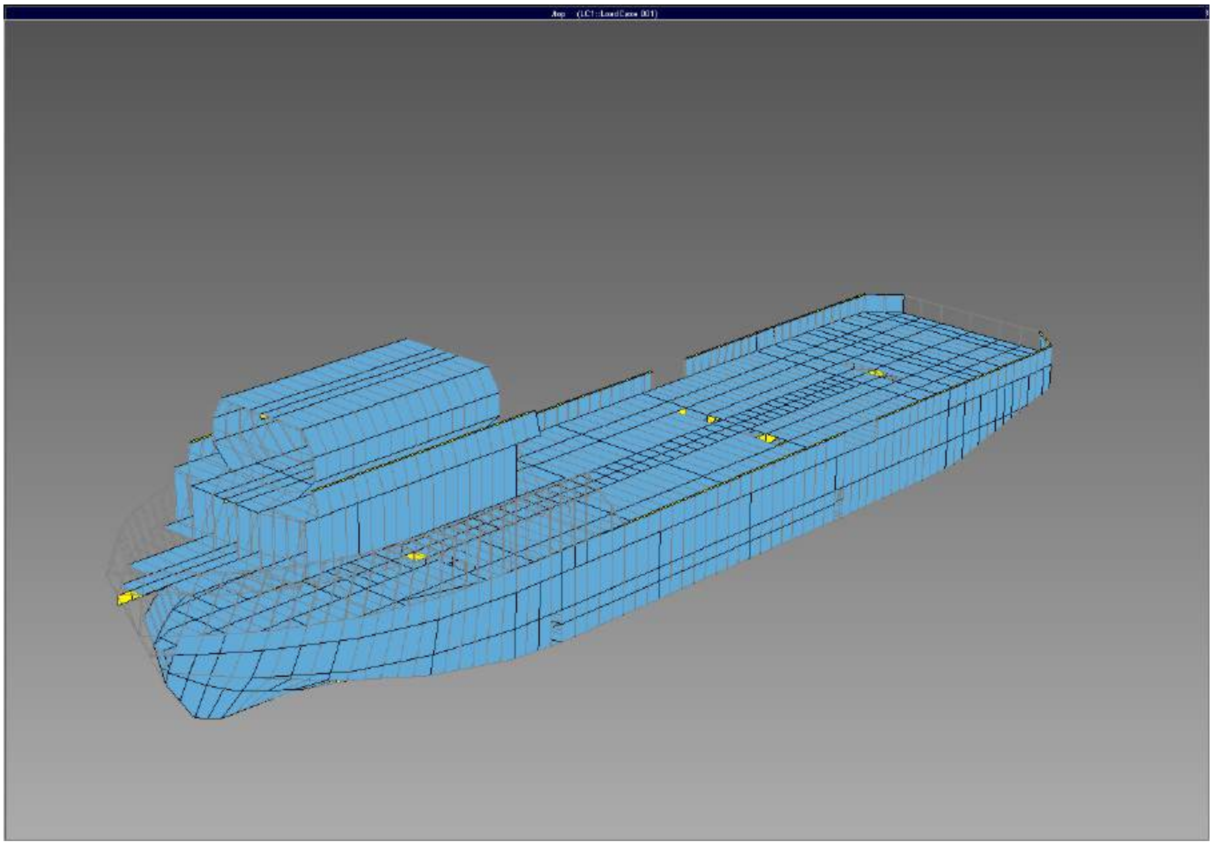
In conjunction with the View Longitudinal Torsional Moment command, the user can use the Dynamic Query, which can be initiated via the  icon, to query the distribution graph. To use this functionality the user must select the View Longitudinal Torsional Moment command, toggle the Dynamic Query icon, and move the mouse cursor over a particular portion of the graph. This will produce a text box with graph data. Further, the user can echo this information to the Output window by double-clicking the left mouse button while hovering over the graph entity of interest.



Element Effectiveness

The **View Longitudinal > Element Effectiveness** command under the hull menu shows a graphical representation of which elements have longitudinal effectiveness.

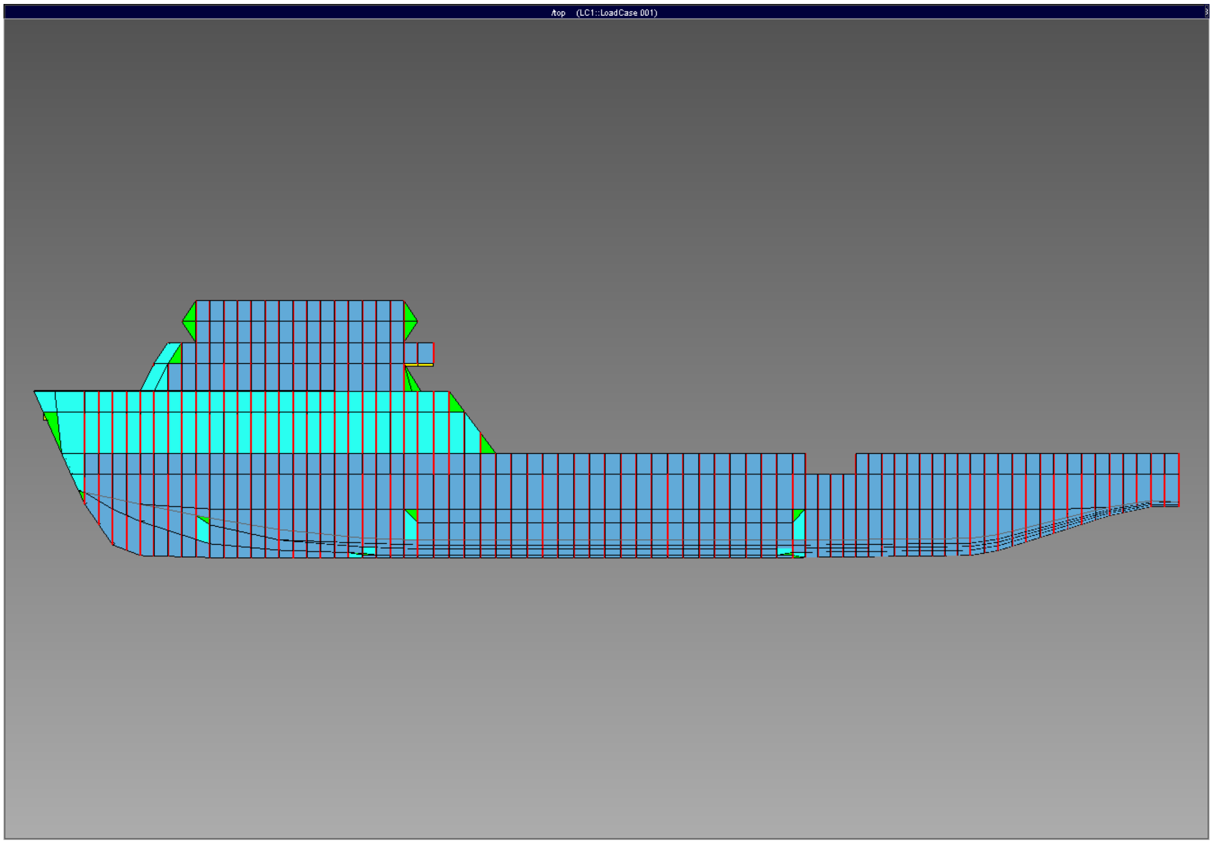
The Dynamic Query  function can be used to highlight an element and toggle on or off whether the element is longitudinally effective.



All Sections

The **View Longitudinal > All Sections** command under the Hull menu will display all the sections of the model. This is helpful if you are viewing specific sections and would like to return to the full model view.

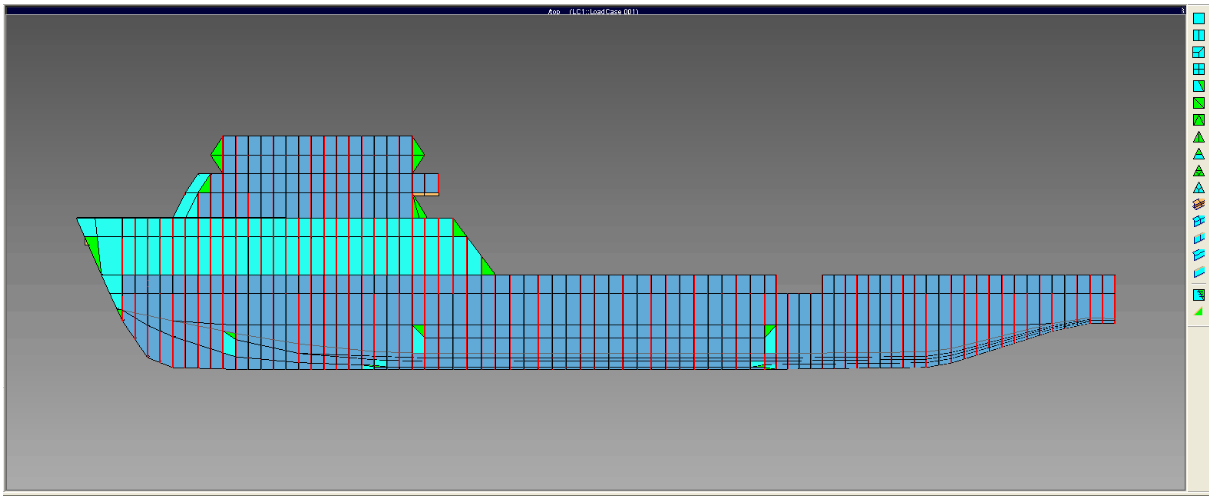
© 2014 Optimum Structural Design, Inc.



Properties

The **View Longitudinal > Properties** command under the Hull menu displays the hull girder properties for each section in table form in the Grid tab. Within the grid, the user can select individual sections to view, or copy the results and paste them into another program such as Microsoft Excel.

© 2014 Optimum Structural Design, Inc.

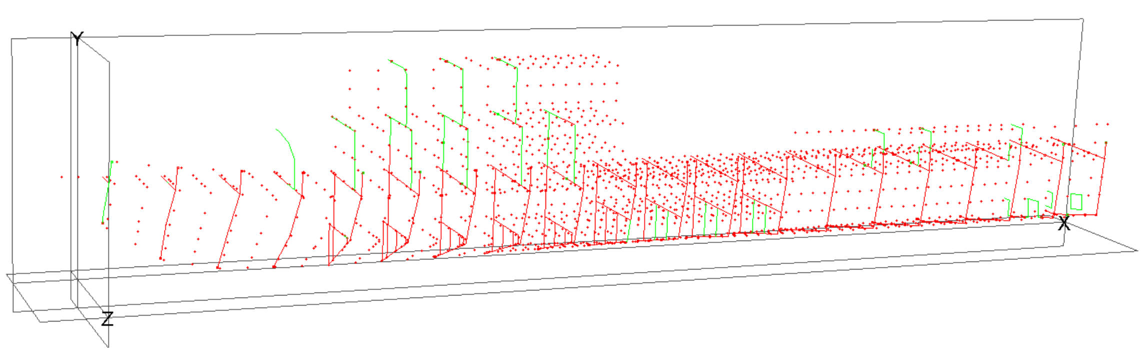


	Segment	xLoc(in)	Status	Izz(V)(in^4)	Iyy(H)(in^4)	J(in^4)	Iww(in^6)	Area(in^2)	AreaZ(in^2)	AreaY(in^2)	Neutral Axis Z(in)	Neutral Axis Y(in)	Shear Center Z(in)	Shear Center Y(in)	Y(DK2NA)(in)	Y(BL2N
12	6	96	OK	1.51733e+006	622631	1.78333e+006	7.89501e+009	255.603	155.71	123.733	2.07077e-016	127.523	-3.40781e-014	221.011	160.477	11
13	7	96	OK	1.51733e+006	622631	1.78333e+006	7.89501e+009	255.603	155.71	123.733	-6.21232e-016	127.523	-3.23479e-014	221.011	160.477	11
14	7	120	OK	1.71415e+006	991160	2.51948e+006	5.0312e+009	280.672	178.335	126.177	-1.32007e-015	122.366	4.46395e-014	190.612	165.634	11
15	8	120	OK	1.77062e+006	1.13208e+006	3.18361e+006	2.8012e+010	348.614	185.112	194.119	-1.67012e-015	119.428	-4.34387e-013	265.863	168.572	11
16	8	144	OK	1.84362e+006	1.49087e+006	4.23441e+006	6.01782e+010	370.111	203.634	197.094	-4.29031e-016	117.395	4.05697e-013	282.753	218.605	11
17	9	144	OK	5.69752e+006	2.35859e+006	4.13137e+006	2.8594e+009	542.377	291.9	284.703	-0.168011	173.353	0.0345482	162.801	162.647	17
18	9	168	OK	6.09895e+006	2.80144e+006	6.80121e+006	7.8984e+009	570.162	316.226	288.162	-1.11399e-015	171.099	6.60503e-014	218.79	200.901	16
19	10	168	OK	7.85837e+006	4.33535e+006	6.80121e+006	4.01499e+009	668.842	372.906	341.842	-1.58273e-015	188.478	-5.24646e-015	194.27	183.522	18
20	10	192	OK	8.23971e+006	4.9153e+006	7.64531e+006	4.74314e+009	691.969	392.093	345.782	9.17897e-016	184.497	4.22546e-014	186.226	223.503	18
21	11	192	OK	8.84784e+006	5.32129e+006	7.64531e+006	4.78391e+009	713.15	395.931	365.626	6.23444e-015	189.595	3.06075e-014	183.794	218.405	18
22	11	216	OK	9.29461e+006	6.01364e+006	8.57483e+006	5.49961e+009	736.89	415.213	370.083	3.87873e-015	185.49	-6.72182e-015	176.193	258.51	18
23	12	216	OK	1.2545e+007	6.25389e+006	1.05784e+007	2.21879e+010	784.447	431.931	404.575	-5.39791e-016	229.068	4.69204e-014	189.5	214.932	22

This data can be copied and pasted into a program such as Excel by following these steps:

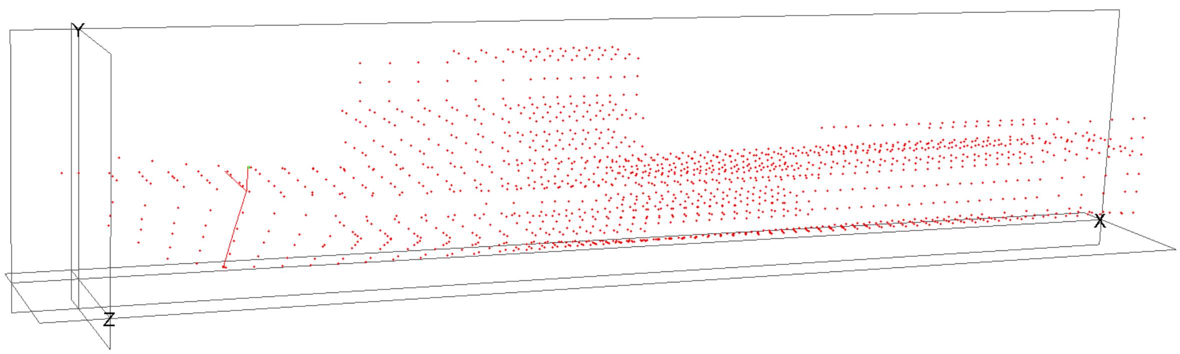
1. Right-click inside the grid and select "Change to Fixed Rows."
2. Change fixed row number from 1 to 0.
3. Copy the data.
4. Paste the data into the desired program.

If the "Slice" option is selected in the Hull Longitudinal Section Cut area of the Preferences dialog, the defined sections will be displayed as below:



The sections shown in red represent longitudinally effective elements while the green sections represent longitudinally ineffective elements.

The same properties will be recovered in the Grid tab, but the user can now right-click and select "Show this Section" will show just the section cut selected. (Selecting "Show all Sections" will return to the full ship sections view.)



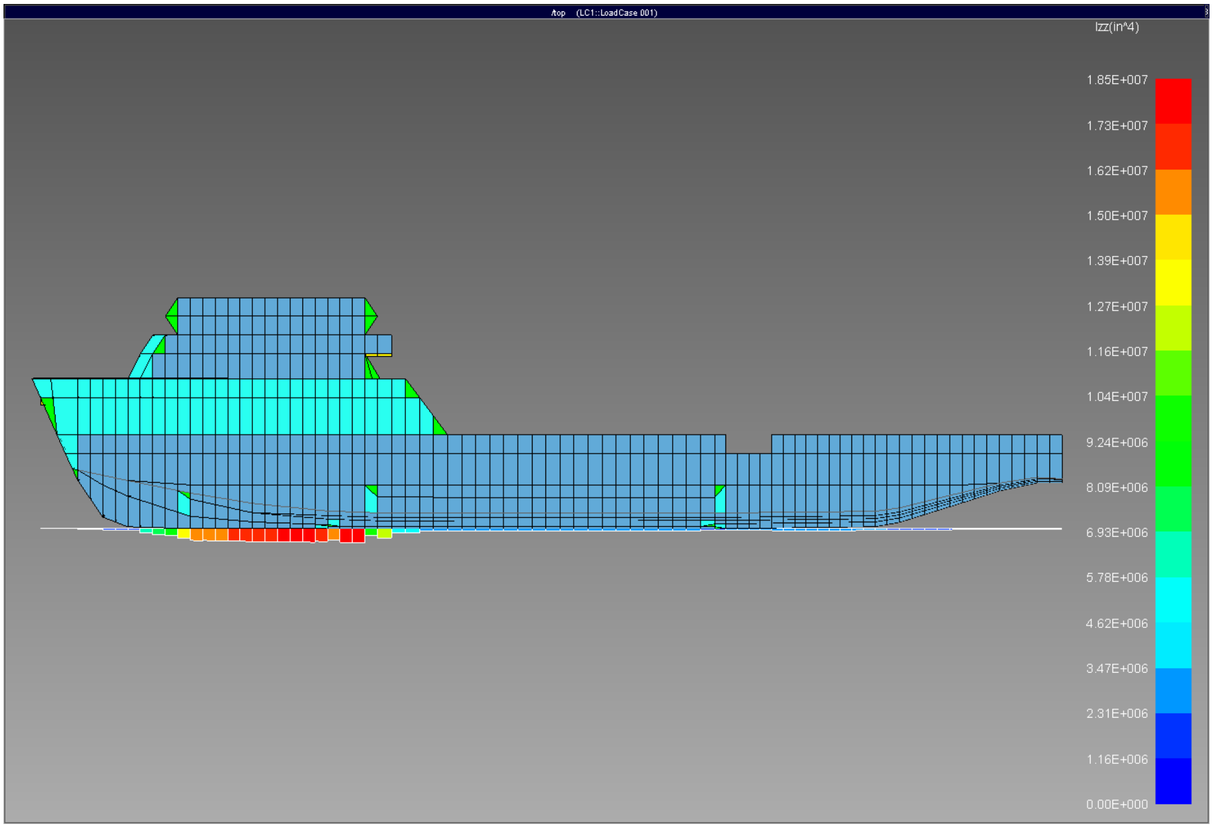
Right-clicking and selecting "Export this Section" or "Export All Sections" will display the section node IDs (note: this is not the finite element node ID) and their locations on the Output tab. This will also list the thickness of the plate for each edge defined between two section nodes. The "smeared thickness" includes any defined stiffeners between those two nodes. The longitudinal and shear effectiveness columns list whether that edge is associated with a longitudinally effective element. The last section of the output lists the section nodes and their total area which includes the rod, beam and bar area contributions.

Izz

The **View Longitudinal > Izz** command under the the Hull menu is used to display the FE model's inertia properties about the z-axis for each section, as shown below.

In conjunction with the View Longitudinal Izz command, the user can use the Dynamic Query, which can be initiated via the  icon, to query the distribution graph. To use this functionality the user must select the View Izz command, toggle the Dynamic Query icon, and move the mouse cursor over a particular portion of the graph. This will produce a text box with graph data. Further, the user can echo this information to the Output window by double-clicking the left mouse button while hovering over the graph entity of interest.

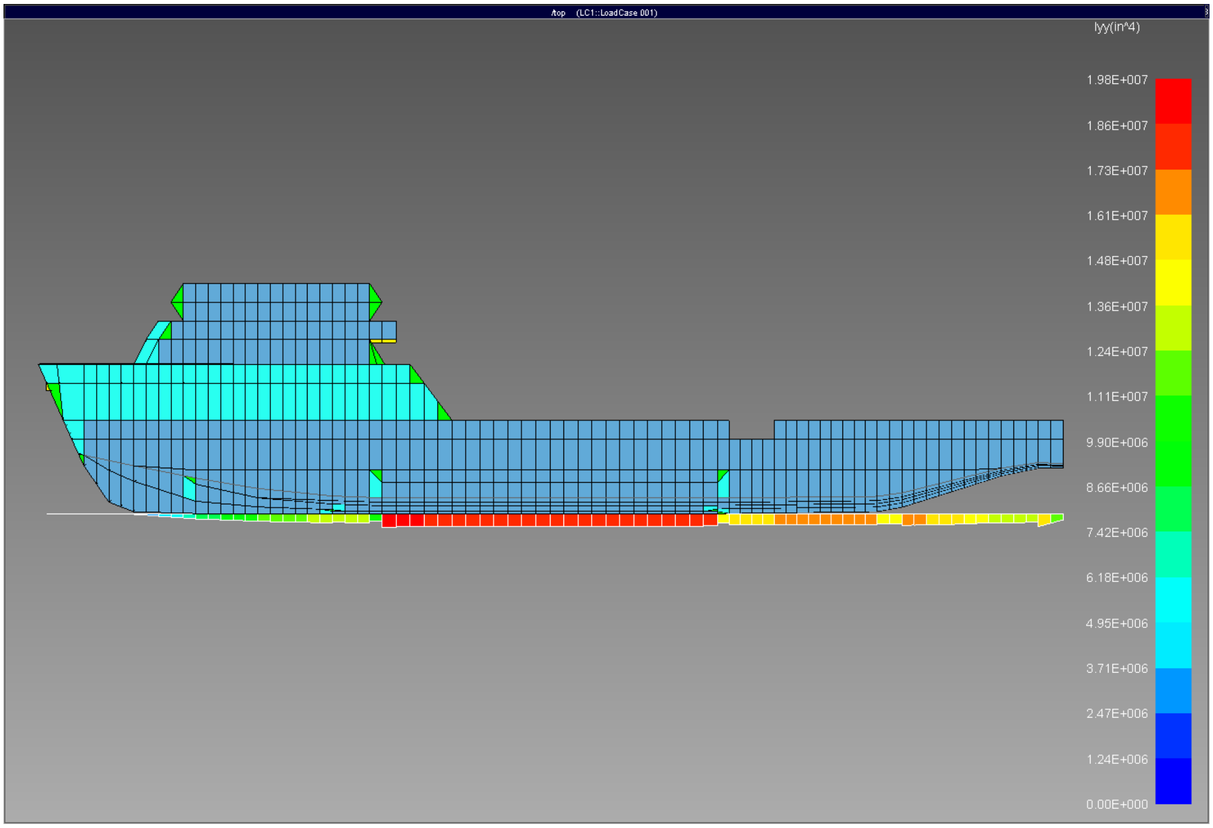
© 2014 Optimum Structural Design, Inc.



Iyy

The **View Longitudinal > Iyy** command under the the Hull menu is used to display the FE model's inertia properties about the y-axis for each section, as shown below.

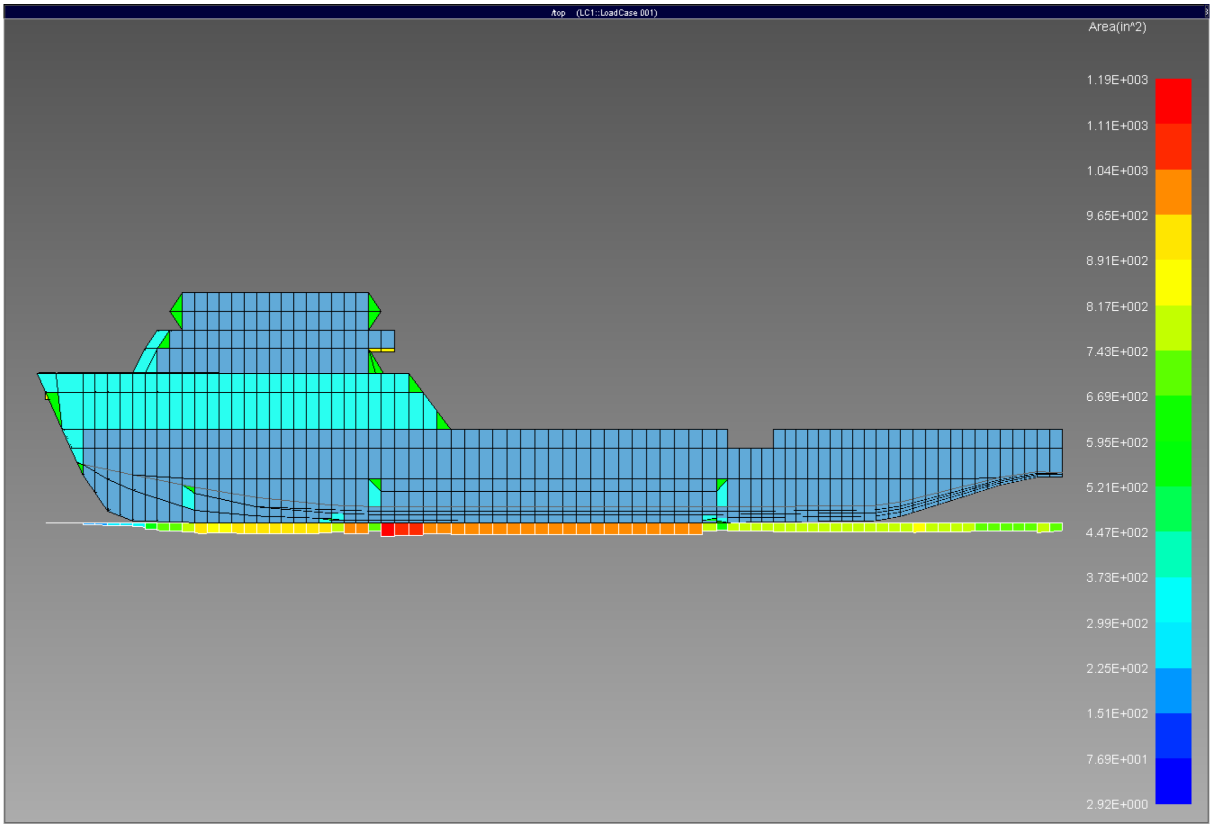
In conjunction with the View Longitudinal Iyy command, the user can use the Dynamic Query, which can be initiated via the  icon, to query the distribution graph. To use this functionality the user must select the View Iyy command, toggle the Dynamic Query icon, and move the mouse cursor over a particular portion of the graph. This will produce a text box with graph data. Further, the user can echo this information to the Output window by double-clicking the left mouse button while hovering over the graph entity of interest.



Cross Sectional Area

The **View Longitudinal > Cross Sectional Area** command under the the Hull menu is used to display the FE model's cross sectional area properties, as shown below.

In conjunction with the View Longitudinal Cross Sectional Area command, the user can use the Dynamic Query, which can be initiated via the  icon, to query the distribution graph. To use this functionality the user must select the View Cross Sectional Area command, toggle the Dynamic Query icon, and move the mouse cursor over a particular portion of the graph. This will produce a text box with graph data. Further, the user can echo this information to the Output window by double-clicking the left mouse button while hovering over the graph entity of interest.

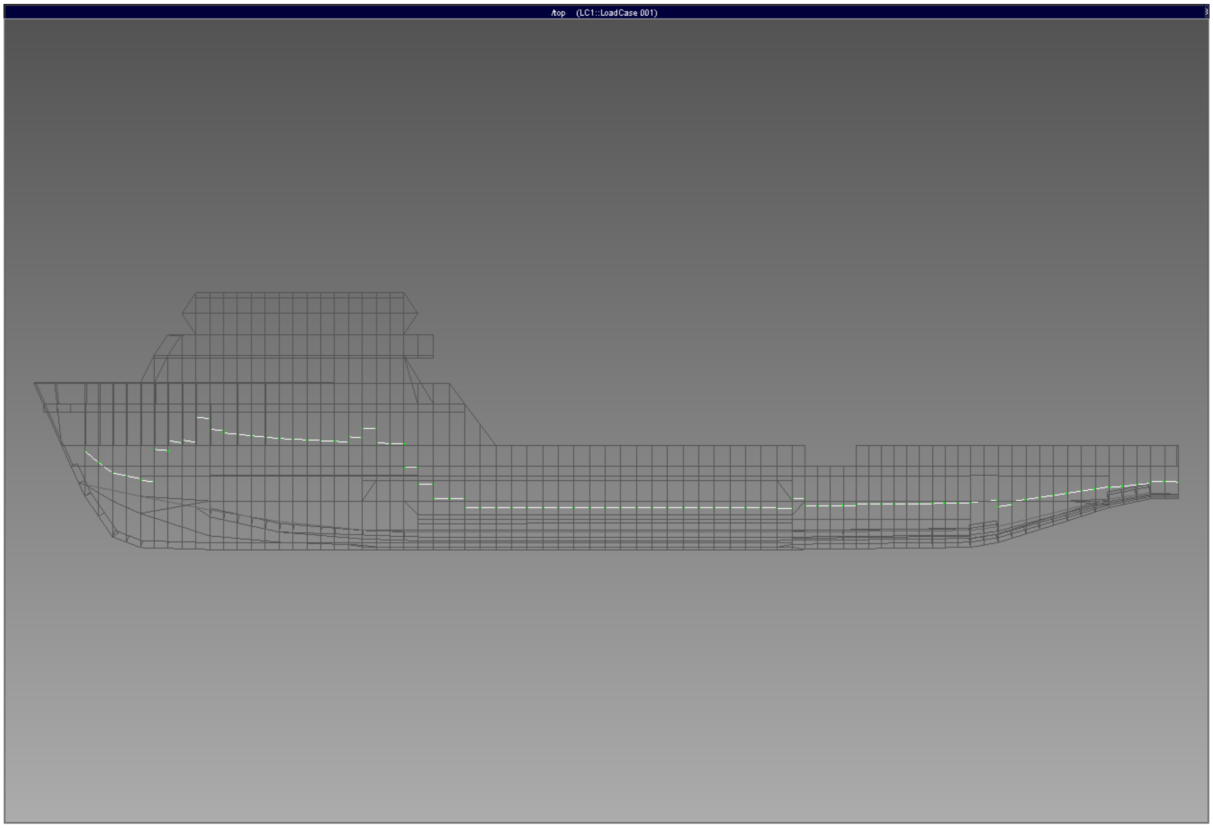


Neutral Axis

The **View Longitudinal > Neutral Axis** command under the the Hull menu is used to display the FE model's neutral center, as shown below.

In conjunction with the View Longitudinal Neutral Axis command, the user can use the Dynamic Query, which can be initiated via the  icon, to query the graph. To use this functionality the user must select the View Neutral Axis command, toggle the Dynamic Query icon, and move the mouse cursor over a particular portion of the graph. This will produce a text box with graph data. Further, the user can echo this information to the Output window by double-clicking the left mouse button while hovering over the graph entity of interest.

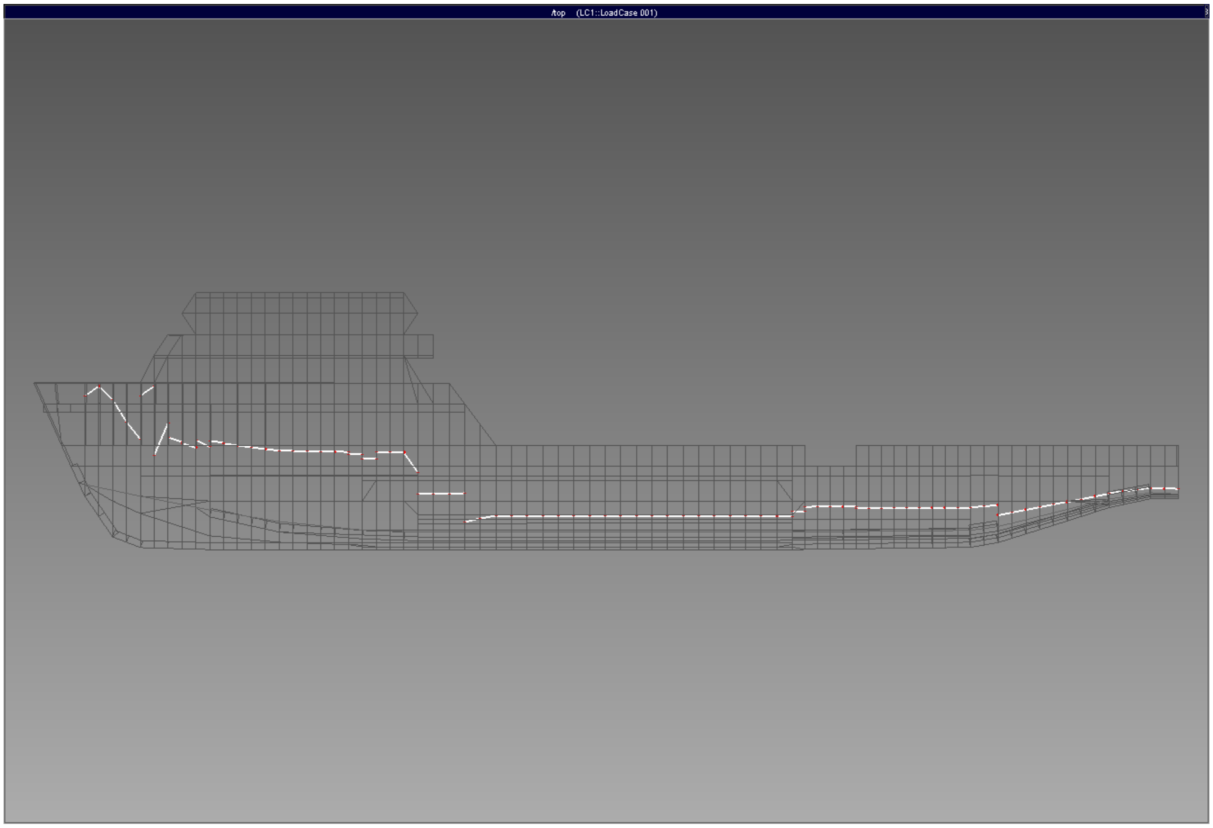
© 2014 Optimum Structural Design, Inc.



Shear Center

The **View Longitudinal > Shear Center** command under the the Hull menu is used to display the FE model's shear center, as shown below.

In conjunction with the View Shear Center command, the user can use the Dynamic Query, which can be initiated via the  icon, to query the graph. To use this functionality the user must select the View Shear Center command, toggle the Dynamic Query icon, and move the mouse cursor over a particular portion of the graph. This will produce a text box with graph data. Further, the user can echo this information to the Output window by double-clicking the left mouse button while hovering over the graph entity of interest.

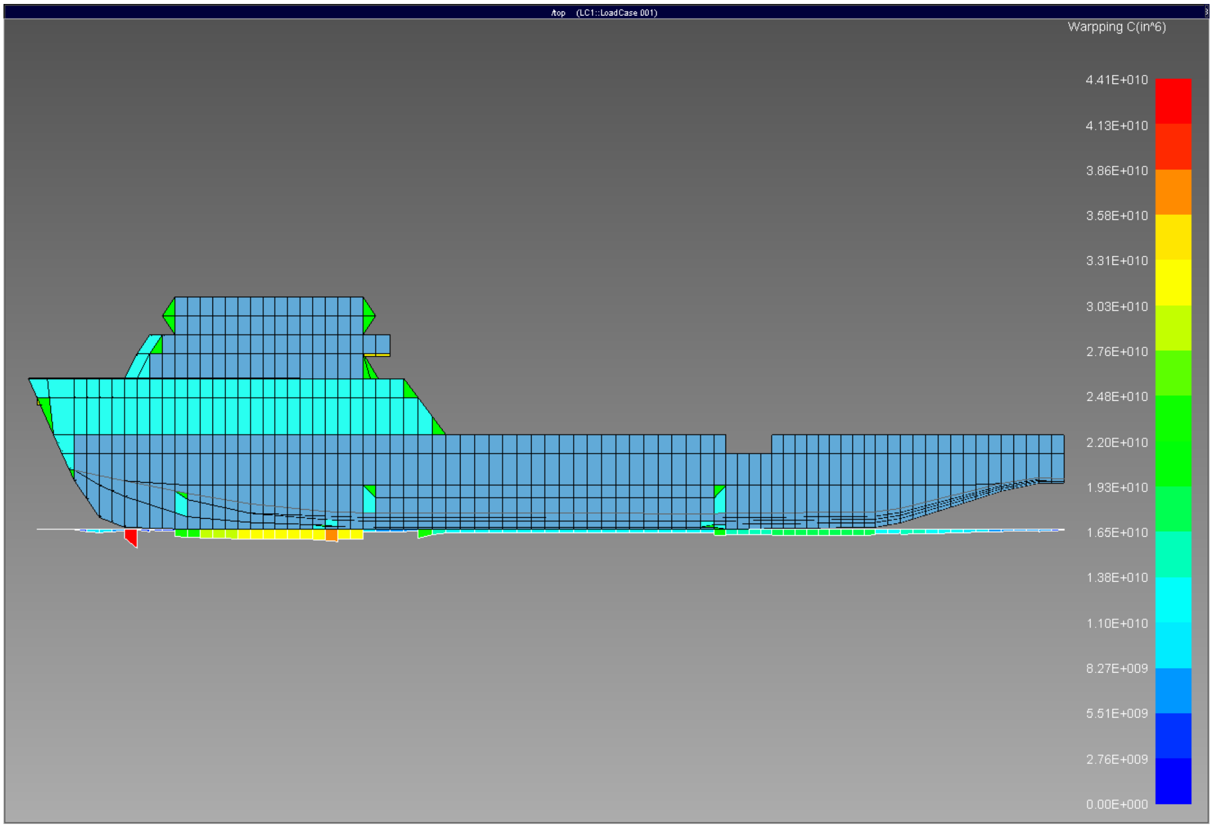


Warping Constant

The **View Longitudinal > Warping Constant** command under the the Hull menu is used to display the FE model's warping properties, as shown below.

In conjunction with the View Longitudinal Warping Constant command, the user can use the Dynamic Query, which can be initiated via the icon, to query the distribution graph. To use this functionality the user must select the View Warping Constant command, toggle the Dynamic Query icon, and move the mouse cursor over a particular portion of the graph. This will produce a text box with graph data. Further, the user can echo this information to the Output window by double-clicking the left mouse button while hovering over the graph entity of interest.

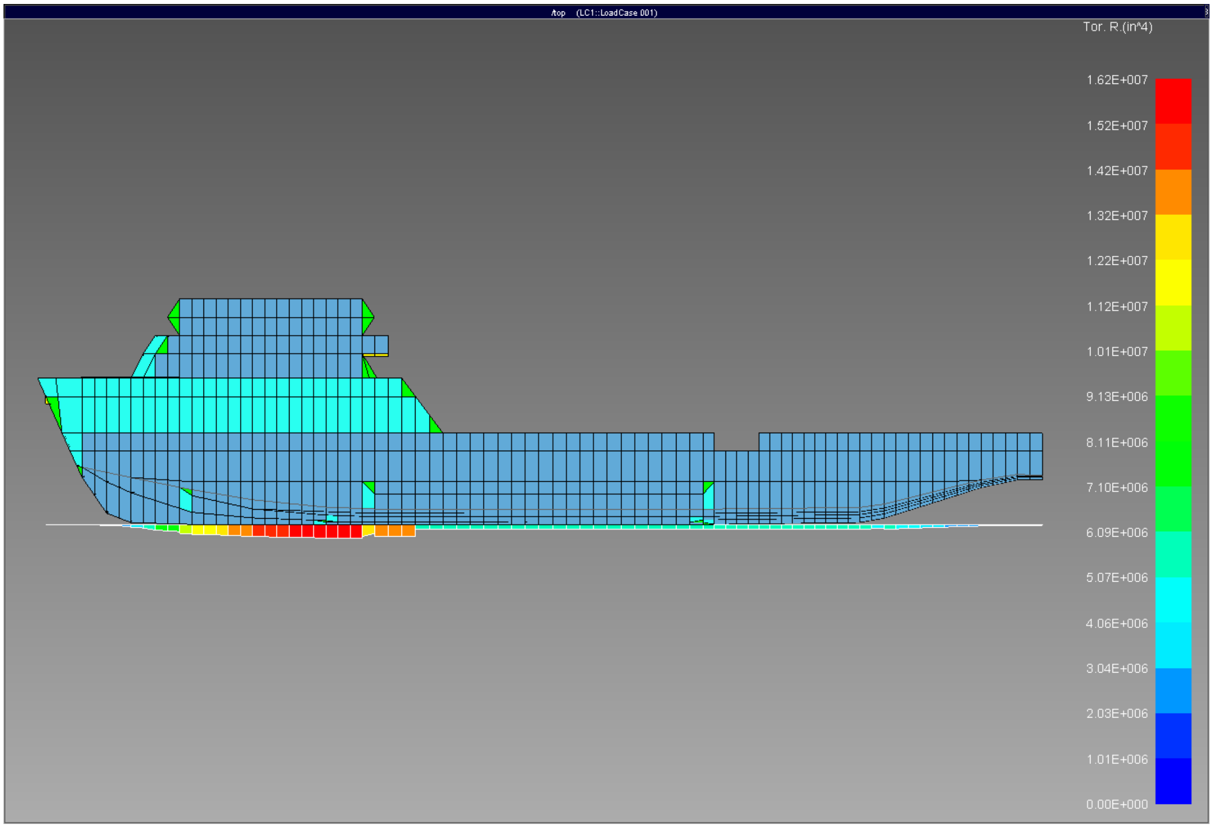
© 2014 Optimum Structural Design, Inc.



Torsional Rigidity

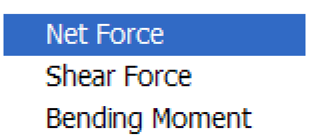
The **View Longitudinal > Torsional Rigidity** command under the the Hull menu is used to display the FE model's torsional rigidity properties, as shown below.

In conjunction with the View Longitudinal Torsional Rigidity command, the user can use the Dynamic Query, which can be initiated via the  icon, to query the distribution graph. To use this functionality the user must select the View Longitudinal Torsional Rigidity command, toggle the Dynamic Query icon, and move the mouse cursor over a particular portion of the graph. This will produce a text box with graph data. Further, the user can echo this information to the Output window by double-clicking the left mouse button while hovering over the graph entity of interest.



View Horizontal >

The View Horizontal menu has several options to display the horizontal properties of the modeled structure.   A detailed description of each of these options is below:



Net Force

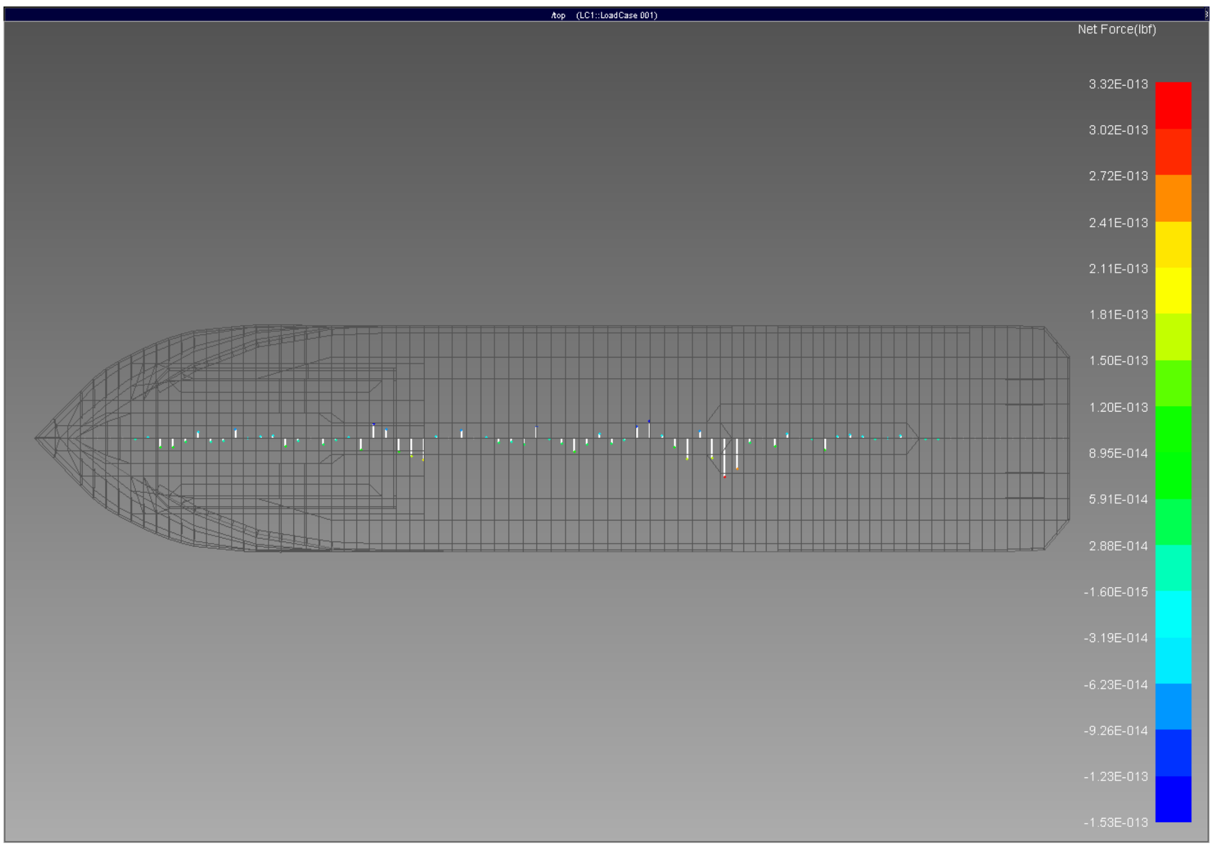
Shear Force

Bending Moment

Net Force

The **View Horizontal > Net Force** option is used to display the FE Model's horizontal net force distribution. Further, MAESTRO echoes this distribution to the Output window, which is found at the bottom of the MAESTRO GUI.

In conjunction with the View Horizontal Net Force command, the user can use the Dynamic Query, which can be initiated via the  icon, to query the distribution graph. To use this functionality the user must select the View Horizontal Net Force command, toggle the Dynamic Query icon, and move the mouse cursor over a particular portion of the graph. This will produce a text box with graph data. Further, the user can echo this information to the Output window by double-clicking the left mouse button while hovering over the graph entity of interest.

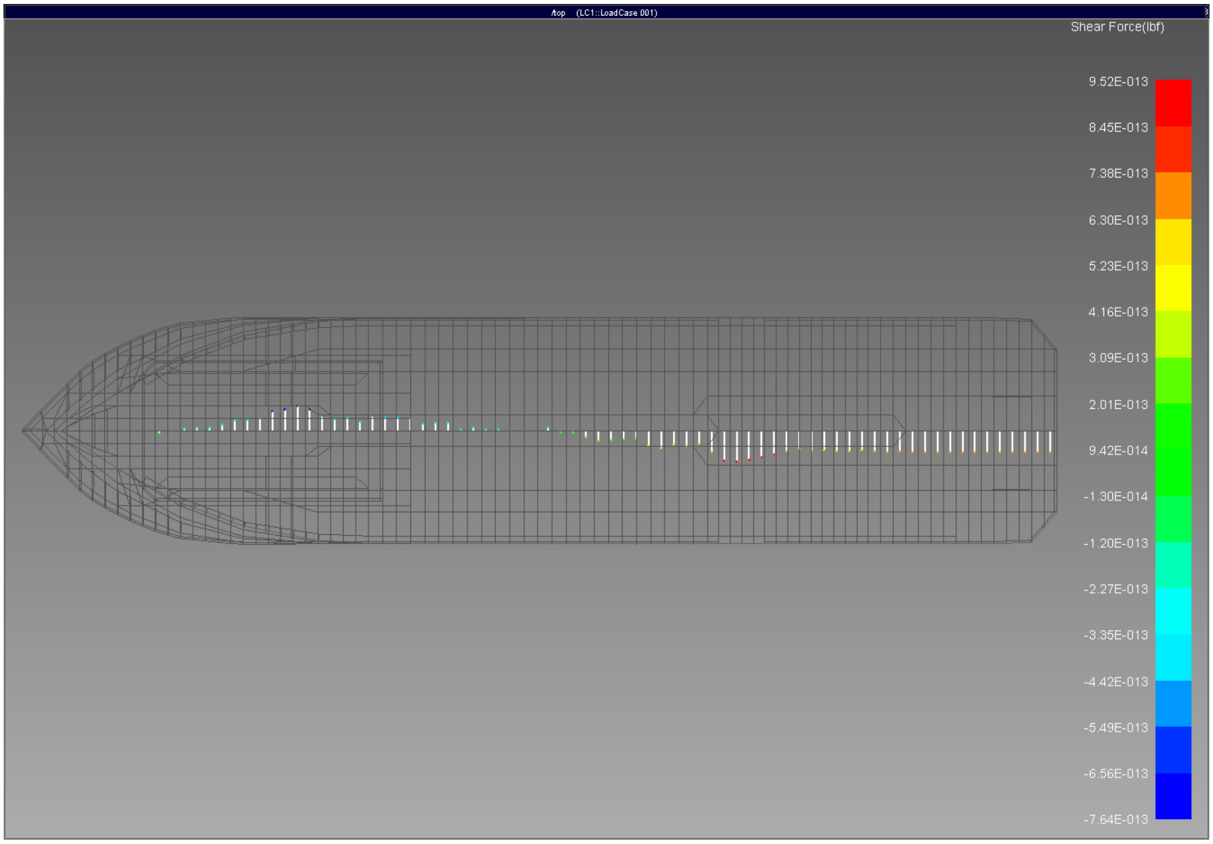


Shear Force

The **View Horizontal > Shear Force** option is used to display the FE Model's horizontal shear force distribution. Further, MAESTRO echoes this distribution to the Output window, which is found at the bottom of the MAESTRO GUI.

In conjunction with the View Horizontal Shear Force command, the user can use the Dynamic Query, which can be initiated via the  icon, to query the distribution graph.

To use this functionality the user must select the View Horizontal Shear Force command, toggle the Dynamic Query icon, and move the mouse cursor over a particular portion of the graph. This will produce a text box with graph data. Further, the user can echo this information to the Output window by double-clicking the left mouse button while hovering over the graph entity of interest.

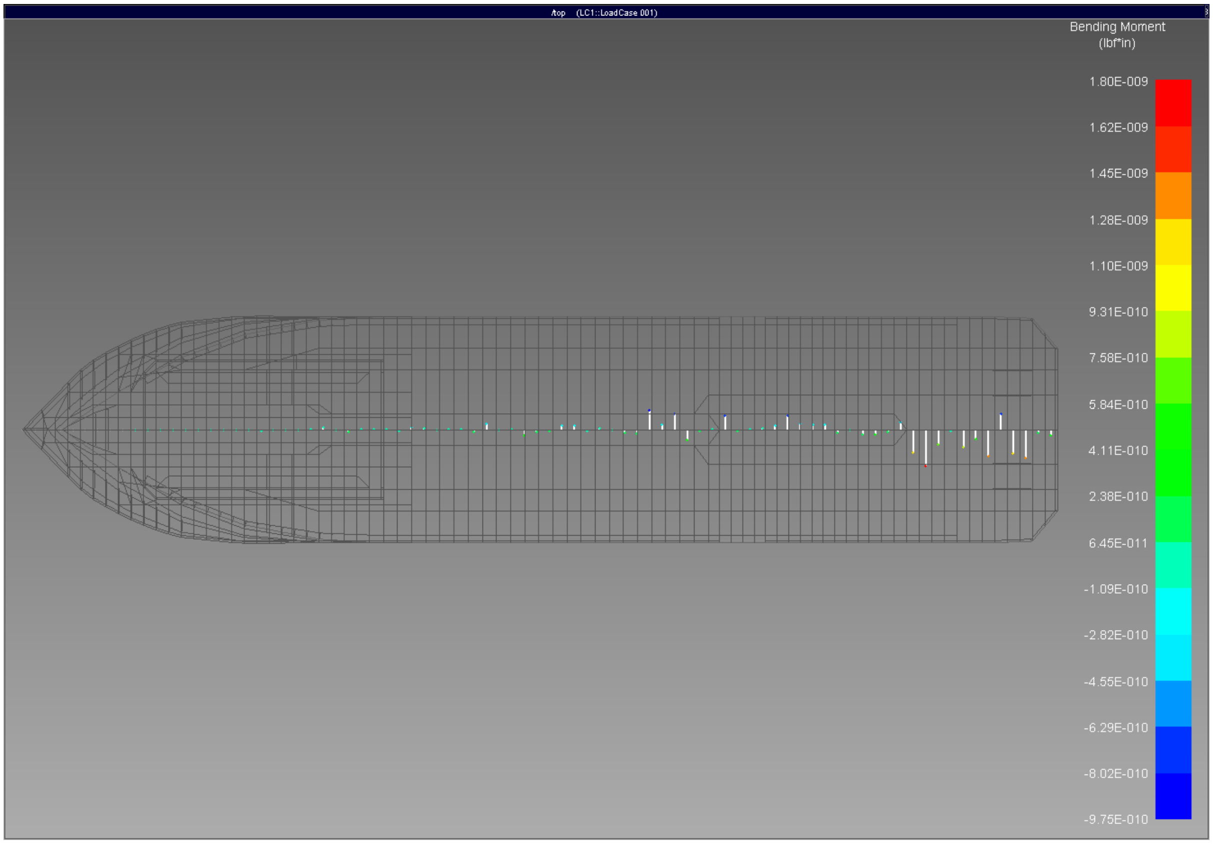


Bending Moment

The **View Horizontal > Bending Moment** option is used to display the FE Model's horizontal bending moment distribution. Further, MAESTRO echoes this distribution to the Output window, which is found at the bottom of the MAESTRO GUI.

In conjunction with the View Horizontal Bending Moment command, the user can use the Dynamic Query, which can be initiated via the  icon, to query the distribution graph. To use this functionality the user must select the View Horizontal Bending Moment command, toggle the Dynamic Query icon, and move the mouse cursor over a particular portion of the graph. This will produce a text box with graph data. Further, the user can echo this information to the Output window by double-clicking the left mouse button while hovering over the graph entity of interest.

© 2014 Optimum Structural Design, Inc.



View Transverse >

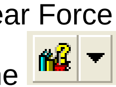
[Shear Force](#)

[Bending Moment](#)

[Torsional Moment](#)

Shear Force

The **View Transverse > Shear Force** option is used to display the FE Model's transverse shear force distribution. Further, MAESTRO echoes this distribution to the Output window, which is found at the bottom of the MAESTRO GUI.

In conjunction with the View Transverse Shear Force command, the user can use the Dynamic Query, which can be initiated via the  icon, to query the distribution graph. To use this functionality the user must select the View Transverse Shear Force command, toggle the Dynamic Query icon, and move the mouse cursor over a particular portion of the

© 2014 Optimum Structural Design, Inc.

graph. This will produce a text box with graph data. Further, the user can echo this information to the Output window by double-clicking the left mouse button while hovering over the graph entity of interest.

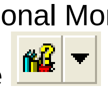
Bending Moment

The **View Transverse > Bending Moment** option is used to display the FE Model's transverse bending moment distribution. Further, MAESTRO echoes this distribution to the Output window, which is found at the bottom of the MAESTRO GUI.

In conjunction with the View Transverse Bending Moment command, the user can use the Dynamic Query, which can be initiated via the  icon, to query the distribution graph. To use this functionality the user must select the View Trasnverse Bending Moment command, toggle the Dynamic Query icon, and move the mouse cursor over a particular portion of the graph. This will produce a text box with graph data. Further, the user can echo this information to the Output window by double-clicking the left mouse button while hovering over the graph entity of interest.

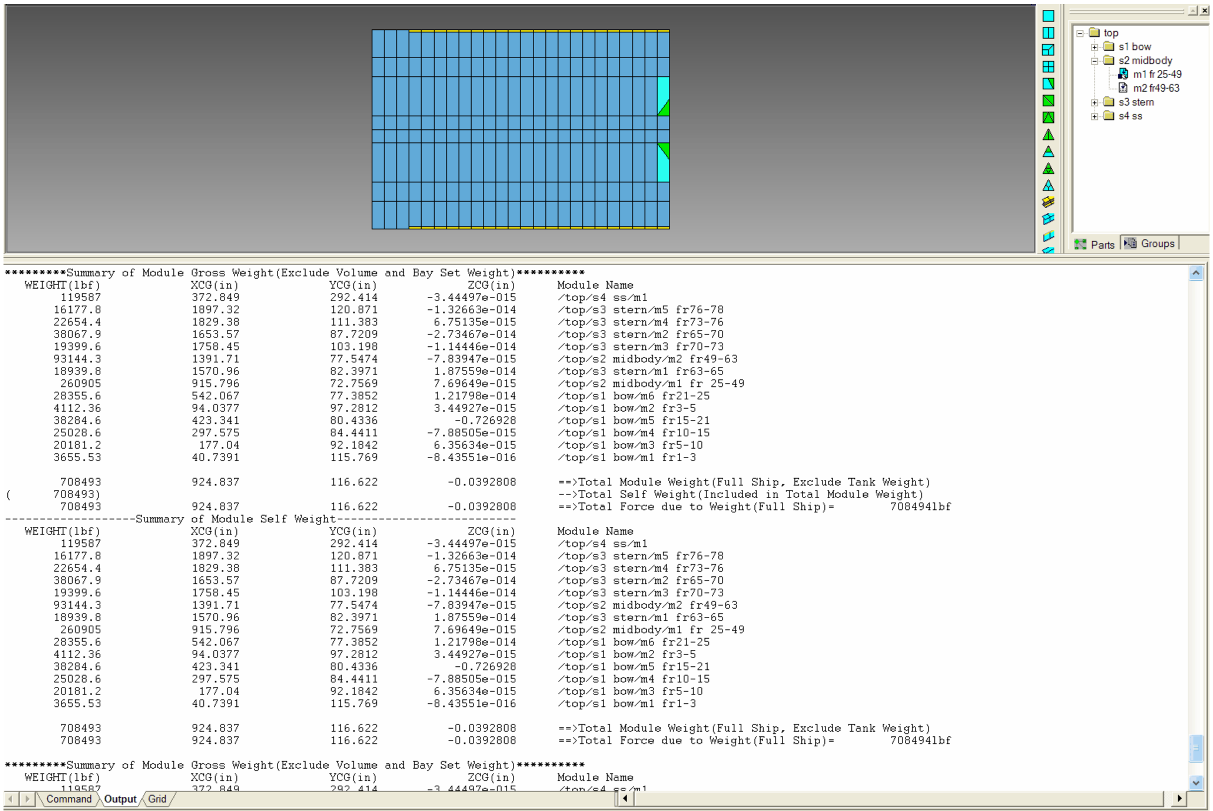
Torsional Moment

The **View Transverse > Torsional Moment** command under the the Hull menu is used to display the FE model's transverse torsional moment distribution, as shown below. MAESTRO also echoes this distribution to the Output window, which is found at the bottom of the MAESTRO GUI.

In conjunction with the View Transverse Torsional Moment command, the user can use the Dynamic Query, which can be initiated via the  icon, to query the distribution graph. To use this functionality the user must select the View Transverse Torsional Moment command, toggle the Dynamic Query icon, and move the mouse cursor over a particular portion of the graph. This will produce a text box with graph data. Further, the user can echo this information to the Output window by double-clicking the left mouse button while hovering over the graph entity of interest.

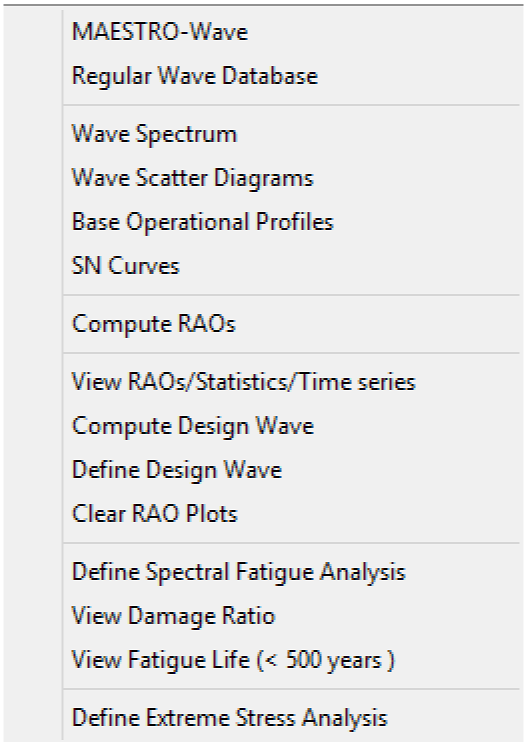
Weight Summary

The **Weight Summary** command under the the Hull menu is used to produce weight summary tables in the Output window, as shown below.



4.9 Wave Menu

The Wave Menu is used to exercise the MAESTRO-Wave, Extreme Load Analysis, and Spectral Fatigue Analysis modules and all of their options.



MAESTRO-Wave

This option will open the MAESTRO-Wave dialog to create and run MAESTRO-Wave cases.

Regular Wave Database

This option will open the regular wave database and show results from any imported MAESTRO-Wave .bmn/.smn files.

Wave Spectrum

This option allows the user to create new theoretical or user defined wave spectra.

Wave Scatter Diagrams

This option allows users to view and modify wave scatter diagrams.

Base Operational Profiles

This option allows users to view and modify operational profiles.

© 2014 Optimum Structural Design, Inc.

SN Curves

This option allows users to view, modify, and create new SN curves.

Compute RAOs

This option allows users to compute RAOs from any imported .bmn/.smn files.

View RAOs/Statistics/Time Series

This option allows users to view results from any imported .bmn/.smn files.

Compute Design Wave

This option allows users to compute a design wave using short or long term statistics from any .bmn/.smn files.

Define Design Wave

This option allows users to define a custom design wave from any .bmn/.smn files.

Clear RAO Plots

This option will clear any current RAO plots from the MAESTRO modeling space.

Define Spectral Fatigue Analysis

This option allows the user to define a wave spectral fatigue analysis or slamming fatigue analysis load case.

View Damage Ratio

This option allows the user to view the damage ratio results after running a fatigue load case.

View Fatigue Life (< 500 Years)

This option allows the user to view the fatigue life results after running a fatigue load case.

Define Extreme Stress Analysis

This option allows the user to calculate mean and extreme stress load cases from any imported .bmn/.smn files and computed stress RAOs.

© 2014 Optimum Structural Design, Inc.

4.10 Analyze Menu

The Analyze Menu is used to run global and local finite element analysis, limit state analysis, and hull girder ultimate strength.

Global & Local FEA

This option will launch the Analysis/Evaluation dialog with the "Solve All Fine Mesh Models" checked.

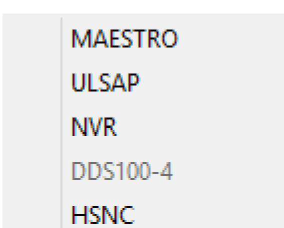
Global FEA

This option will launch the Analysis/Evaluation dialog.

Local FEA

This option will analyze any fine mesh models that exist in the current file.

Limit State Evaluation (Post FEA) >



These options will run only the limit state analysis for the selected limit states. Note: finite element analysis results are required to run just the limit state calculations.

© 2014 Optimum Structural Design, Inc.

Individual Panel Evaluation

This option will launch the standalone panel evaluation dialog.

Hull Girder Collapse

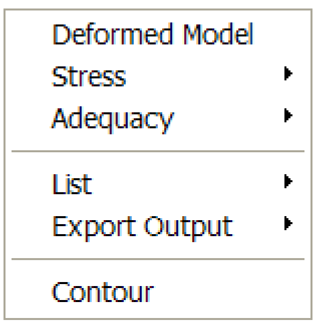
This option will run the ALPS/HULL analysis if an ALPS/HULL model exists in the current file.

Legacy Version of MAESTRO >

These menu options are for legacy users of MAESTRO and should only be accessed by experienced users.

4.11 Results Menu

The Results menu allows the user several options to display and post-process the results from the finite element analysis.



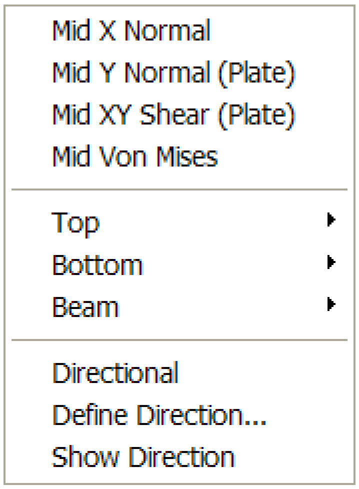
Deformed Model

This option will toggle on and off whether the model is displayed as deformed or undeformed. A check mark will appear next to this option in the menu if the model is in the deformed view. The Deformation Scale can be adjusted in the View Options dialog under the Post-Processing section.

Stress >

For each load condition, MAESTRO calculates stresses for all finite elements. The menu items below provide access to recovering stresses in rods, beams, bare plates and stiffened panels. An important distinction between each element type's stress recovery will be discussed in the Stress Results sections, as well as additional details for each of the stress recovery options. The dynamic query icon  can be used to highlight and recover a specific element's analysis results. Double-clicking the element will echo the results to the Output tab.

Note: the scale for the graphical representation of stress will remain the same if switching between load cases, unless the maximum stress value is higher than the current scale's maximum value. To view the load case specific legend, turn off the stress view and then turn it on again.



Directional

This option will toggle on or off the directional stress option.

Define Direction...

This option will launch the Define Stress Direction dialog.

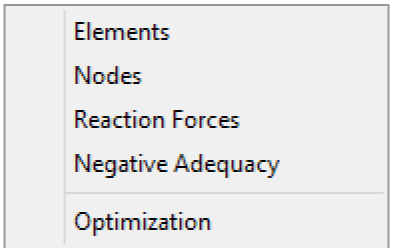
Show Direction

This option will toggle on or off whether small quills are placed on each element defining the element's local X-direction.

Adequacy >

This option will provide the menu for the particular failure mode evaluation chosen during the analysis. For more information on each adequacy parameter and how to perform a failure mode evaluation, please see the Failure Mode Evaluation section.

List >



Elements

This option will populate the Grid tab with a list of elements, their stresses, and adequacy parameter results (if applicable). The cells can be copied and pasted into a database program, like Microsoft Excel, if desired.

Nodes

This option will populate the Grid tab with a list of the element nodes, their location, and their displacement. The cells can be copied and pasted into a database program, like Microsoft Excel, if desired.

Reaction Forces

This option will list the reaction forces and moments in the Output tab for each constrained node and total each force and moment direction at the bottom.

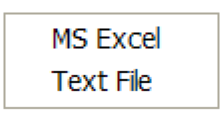
Negative Adequacy

This option will list the number of elements with a negative adequacy parameter and those elements percentage of the total structural weight for the selected load case.

Optimization

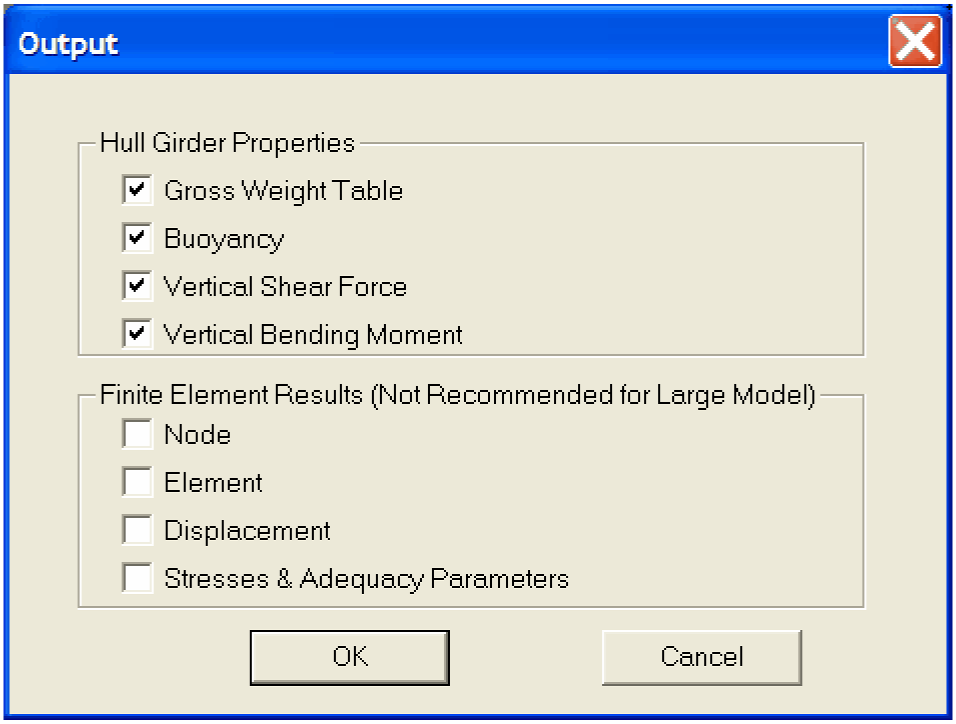
This option will list the results of the previous optimization run, if available.

Export Output >



MS Excel

This option will open the Output dialog so the user can select which properties and results to output. MAESTRO will then automatically create a Microsoft Excel file of the selected output data.



Text File

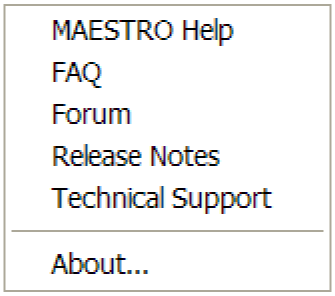
This option will open a dialog allowing the user to save a text file of the elements, materials, and results.

Contour

This option toggles on the stress contour view. A check mark will appear next to the option in the menu when contour plot is turned on.

4.12 Help Menu

The Help menu provides several resources to the user for help with MAESTRO. A brief description of each option is discussed below.



MAESTRO Help

This option will launch the MAESTRO help file. This can also be done by clicking the MAESTRO Help icon .

FAQ

This option will launch the MAESTRO website FAQ section. These FAQ are the same as the ones found in the MAESTRO help file.

Forum

This option will launch the MAESTRO forum where MAESTRO users and technical support personnel post tips, tricks, and technical support questions and responses.

Release Notes

This option will open the MAESTRO release notes providing a summary of new features and fixed bugs.

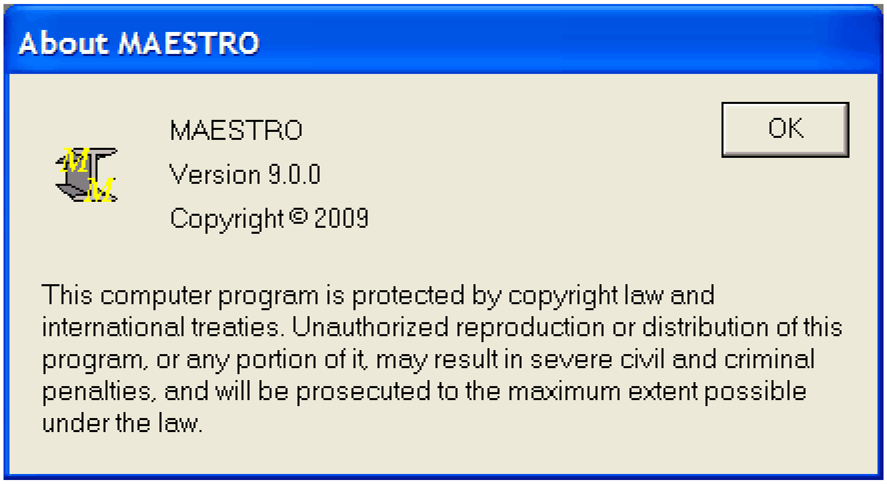
Technical Support

This option will launch your desktop email program with an email pre-addressed to support@maestro.com.

About...

This option will open the About MAESTRO dialog providing the currently installed version of MAESTRO and copyright information.

© 2014 Optimum Structural Design, Inc.



4.13 Toolbars

The MAESTRO interface contains five toolbars to allow the user to access often used commands with a single mouse click. These include the Standard toolbar, the Pre-Processing toolbar, the View toolbar, the Post-Processing toolbar, and the Refine Element toolbar. All commands associated with toolbar buttons can also be accessed via the main menu.

Please see the GUI Interface figure to see the toolbar layout.

Standard Toolbar

Pre-Processing Toolbar

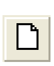
Viewing Toolbar

Post-Processing Toolbar

Refine Element Toolbar

Standard Toolbar

The Standard toolbar which is located at the upper left part of the interface just below the main menu provides a fast and easy means of performing some common tasks. Most of the icons on the toolbar correspond to a specific file menu command. A brief description of the action performed by each toolbar icon is given below:

 This icon creates a new modeler file.

 This icon launches a dialog prompting the user to select the existing modeler file they wish to open.

 This icon will save the current modeler file over the existing file name. If no file name is defined, it will prompt the user to name the file and then save.

 This icon will launch the print dialog allowing the user to print the a hardcopy of the contents of the main display.

 This icon will launch the MAESTRO help file.

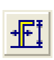
Pre-Processing Toolbar

The Pre-Processing toolbar which is located on the left side of the interface provides fast and easy means of accessing the functions necessary to construct and load a finite element model.

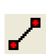
 This icon launches the Job Information dialog. It is the same as selecting **File > Job Information...** from the menu.

 This icon launches the Parts dialog. It is the same as selecting **Model > Parts > Create/Modify** from the menu.

 This icon launches the Materials dialog. It is the same as selecting **Model > Materials...** from the menu.

 This icon launches the Properties dialog. It is the same as selecting **Model > Properties** from the menu.

 This icon launches the Stiffener Layout dialog. It is the same as selecting **Model > Stiffener Layout...** from the menu.

 This icon launches the Reference Points dialog. It is the same as selecting **Model > Nodes > Create/Modify > EndPoint** from the menu.

 This icon launches the Additional Nodes dialog. It is the same as selecting **Model > Nodes > Create/Modify > Additional Node** from the menu.

 This icon launches the Strakes dialog. It is the same as selecting **Model > Elements > Create/Modify > Strake** from the menu.

 This icon launches the Compounds dialog. It is the same as selecting **Model >**

Elements > Create/Modify > Compound from the menu.

 This icon launches the Finite Element Quad dialog. It is the same as selecting **Model > Elements > Create/Modify > Quad** from the menu.

 This icon launches the Finite Element Triangle dialog. It is the same as selecting **Model > Elements > Create/Modify > Triangle** from the menu.

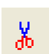
 This icon launches the Finite Element Beam dialog. It is the same as selecting **Model > Elements > Create/Modify > Beam** from the menu.

 This icon launches the Finite Element Rod dialog. It is the same as selecting **Model > Elements > Create/Modify > Rod** from the menu.

 This icon launches the Finite Element Spring dialog. It is the same as selecting **Model > Elements > Create/Modify > Spring** from the menu.

 This icon launches the Finite Element RSpline dialog. It is the same as selecting **Model > Elements > Create/Modify > RSpline** from the menu.

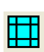
 This icon launches the Finite Element Bracket dialog. It is the same as selecting **Model > Elements > Create/Modify > Bracket** from the menu.

 This icon launches the Deletion dialog. It is the same as selecting **Tools > Deletions > Select** from the menu.

 This icon launches the Groups dialog. It is the same as selecting one of the group types from the **Groups** menu.

 This icon launches the Restraints dialog. It is the same as selecting **Model > Define Restraints...** from the menu.

 This icon launches the Loads dialog. It is the same as selecting **Loads > Create/Modify...** from the menu.

 This icon launches the Evaluation Patch dialog. It is the same as selecting **Model > Evaluation Patch > Create/Evaluate** from the menu.

Viewing Toolbar

The Viewing toolbar which is located at the top of the modeling space provides the various viewing options for the model. These options can be helpful from constructing the model all the way through post processing.

This icon launches the View Options dialog. It is the same as selecting **View > Options...** from the menu.

This icon allows the user to click a part in the model space to set it as the current part. It is the same as highlighting a part in the parts tree and right-clicking and selecting *Set Current Part* or selecting **Tools > Set Current Part** from the menu.

This icon allows the user to click a part in the model space to set it as the current view. It is the same as highlighting a part in the parts tree and right-clicking and selecting *Set View Part* or selecting **Tools > Set View Part** from the menu.

This icon allows the user to click a part in the model space to set it as the current part and current view. It is the same as highlighting a part in the parts tree and right-clicking and selecting *Set Current & View Part* or selecting **Tools > Set Current & View Part** from the menu.

This icon allows the user to click a module in the model space and set it as a Transparent view. It is the same as highlighting a part in the parts tree and right-clicking and selecting *Set Transparency On.*

This icon allows the user to click a transparent view module and return it to the standard element type view. It is the same as highlighting a part in the parts tree and right-clicking and selecting *Set Transparency Off.*

This icon toggles the view between solid and wireframe.

This icon toggles node visibility as on or off.

This icon will toggle on and off stiffeners. This is the same as opening the View Options dialog and checking "Stiffeners".

This icon will toggle on and off the water plane. This is the same as opening the View Options dialog and checking "Water Plane". Note, to turn on the CG and CF markers, you must still open the View Options dialog.

This icon toggles the background color between black and white.

This icon will hide the elements outside of the defined range for the current plot.

This icon will toggle shrink elements on or off.

This icon enables dynamic rotation of the model. After selecting this toolbar button, use the left mouse button to rotate the model. Use a left down-click to rotate the model, and release the mouse button to set the new view angle. The command equivalent of this item is Set-View-Angles.

NOTE: The user also has the choice of using the wheel button (if available) to invoke the dynamic rotation functionality. Simply move the mouse while simultaneously holding the wheel button down.

 This icon rotates the model about the global X-axis each time it is clicked.

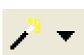
 This icon rotates the model about the global Z-axis each time it is clicked.

 This icon rotates the model about the global Y-axis each time it is clicked.

 This icon will toggle the direction of any sign sensitive command. Rotations will occur in a positive direction and zoom command will magnify views when the "+" is enabled and conversely, rotations will be in a negative sense and zooming will zoom out when the "-" is enabled.

NOTE: Icons only flip/rotate the model in constant increments of 15 degrees. To rotate the model to a specific angle, execute **View > Set View > Specify** from the menu. The user is then prompted to supply the angle of rotation as an argument in the Command Line. Specific angles can also be achieved dynamically through the dynamic rotation toolbar button.

 This icon will toggle the view of the Parts Tree on or off.

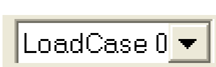
 This icon allows the user to select an element type to apply the Quick Create functionality to.

 This icon is for the "Cancel" command. Clicking on this toolbar button will cancel any action that is currently being executed and also clears the command stack. This is the same as pressing the *Esc* key.

 This icon allows the user to query the model, dependent on the current view, by moving the mouse cursor over an element. To see more uses, read about the view options in the View Menu, the Loads Menu, and the dynamic query's ability with Creating and Evaluating Evaluation Patches, and Post-Processing of Failure Mode Analysis.

Post-Processing Toolbar

The Post-Processing toolbar which is located at the top of the modeling space provides quick access to functions to balance, analyze and post-process the model.



This drop-down menu allows the user to quickly switch between defined load cases and also shows which load case the current results shown are for.

© 2014 Optimum Structural Design, Inc.

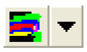
 This icon launches the Balance dialog. It is the same as selecting **Model > Balance...** from the menu.

 This icon toggles the Contour results plot on and off.

 This icon is used with composite layers analysis.

 This icon launches the Animation dialog. It is then used to toggle the animation on or off.

 This icon will force any animation that is running to stop.

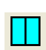
 This icon launches the Analysis dialog. Clicking the drop-down arrow allows the user to select the analysis type.

Refine Element Toolbar

The Refine Element toolbar which is located on the right side of the modeling space is used to quickly and easily convert coarse mesh elements into finer mesh elements.

 This icon allows the user to double-click a strake panel and MAESTRO will automatically convert it to a quad element.

 This icon allows the user to double-click a full strake and split it into two identical strakes.

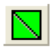
 This icon allows the user to double-click a quad element and MAESTRO will automatically split the quad into two quads.

 This icon allows the user to double-click a quad element and MAESTRO will automatically split the quad into three quads.

 This icon allows the user to double-click a quad element and MAESTRO will automatically split the quad into four quads.

 This icon allows the user to double-click a quad element and MAESTRO will automatically split the quad into a quad and a triangle.

 This icon allows the user to double-click a quad element and MAESTRO will automatically split the quad into two quads and a triangle.

 This icon allows the user to double-click a quad element and MAESTRO will automatically split the quad into two triangles.

 This icon allows the user to double-click a quad element and MAESTRO will automatically split the quad into three triangles.

 This icon allows the user to double-click a triangle element and MAESTRO will automatically split the triangle into two triangles.

 This icon allows the user to double-click a triangle element and MAESTRO will automatically split the triangle into a triangle and a quad.

 This icon allows the user to double-click a triangle element and MAESTRO will automatically split the triangle into four triangles.

 This icon allows the user to double-click a triangle element and MAESTRO will automatically split the triangle into three quads.

 This icon allows the user to double-click a rod or beam element and MAESTRO will automatically split the rod or beam into two elements.

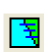
 This icon allows the user to double--click a beam element and MAESTRO will automatically split the beam element into two quads for the beam web and 2 quads for the beam flange (if applicable).

 This icon allows the user to double-click a beam element and MAESTRO will automatically split the beam element into two quads for the beam web and 2 rods for the beam flange (if applicable).

 This icon allows the user to double-click a beam element and MAESTRO will automatically convert the beam web to a quad and the flange to a quad (if applicable).

 This icon allows the user to double-click a beam element and MAESTRO will automatically convert the beam web to a quad and the flange to a rod (if applicable).

 This icon allows the user to double-click a stiffened strake or quad element and MAESTRO will automatically convert the internal stiffeners to beam elements and split the plate element into multiple elements.

 This icon allows the user to merge two elements into one by clicking the first element and then double-clicking the second element to be merged.

 This icon allows the user to double-click a bracket element and MAESTRO will automatically convert the bracket to a triangle and a rod.

© 2014 Optimum Structural Design, Inc.

4.14 Standard Views

MAESTRO has several default views that will automatically orient the model for viewing. These views can be accessed from the **View > Set View >** menu, or by right-clicking in the modeling space and selecting from the menu. MAESTRO also allows the user to create Named View, which can be saved and called up like a default view.

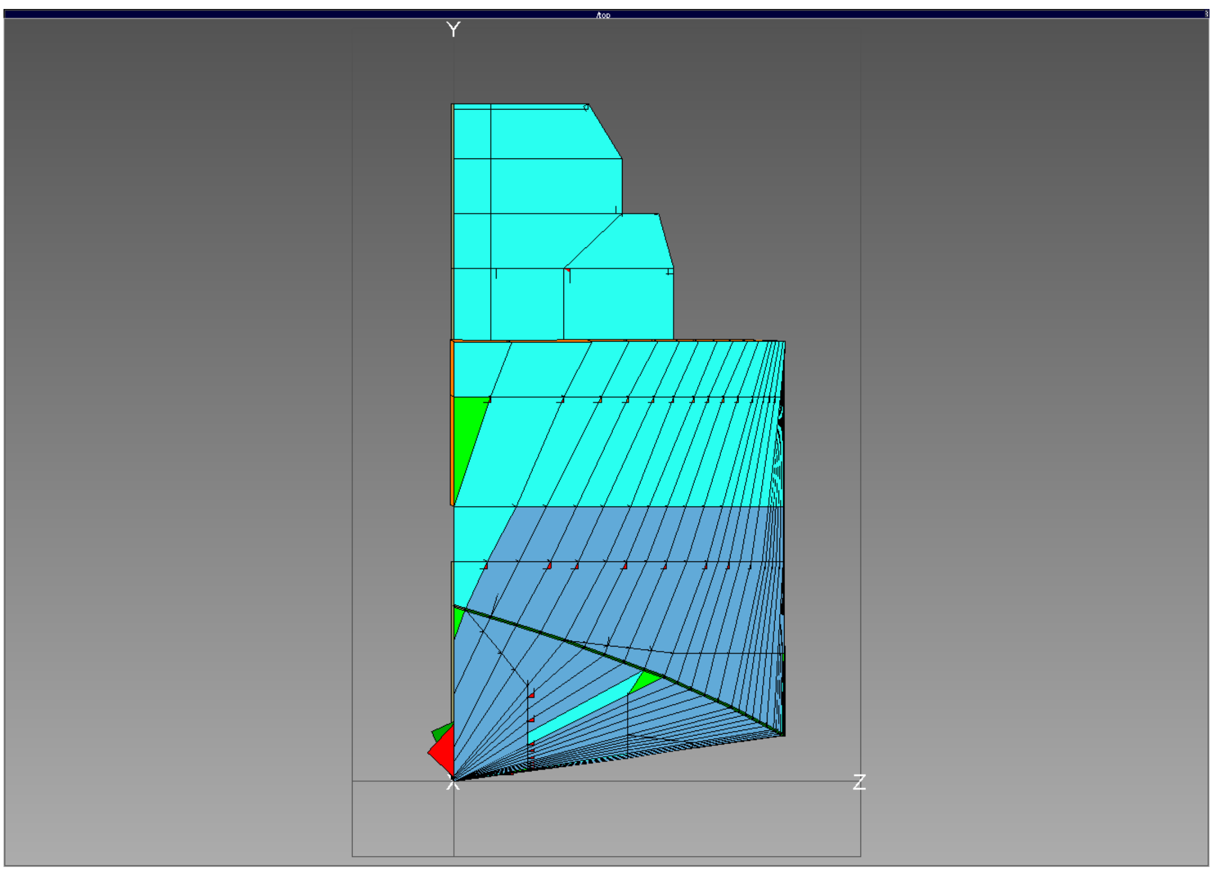
Body Plan

Profile View

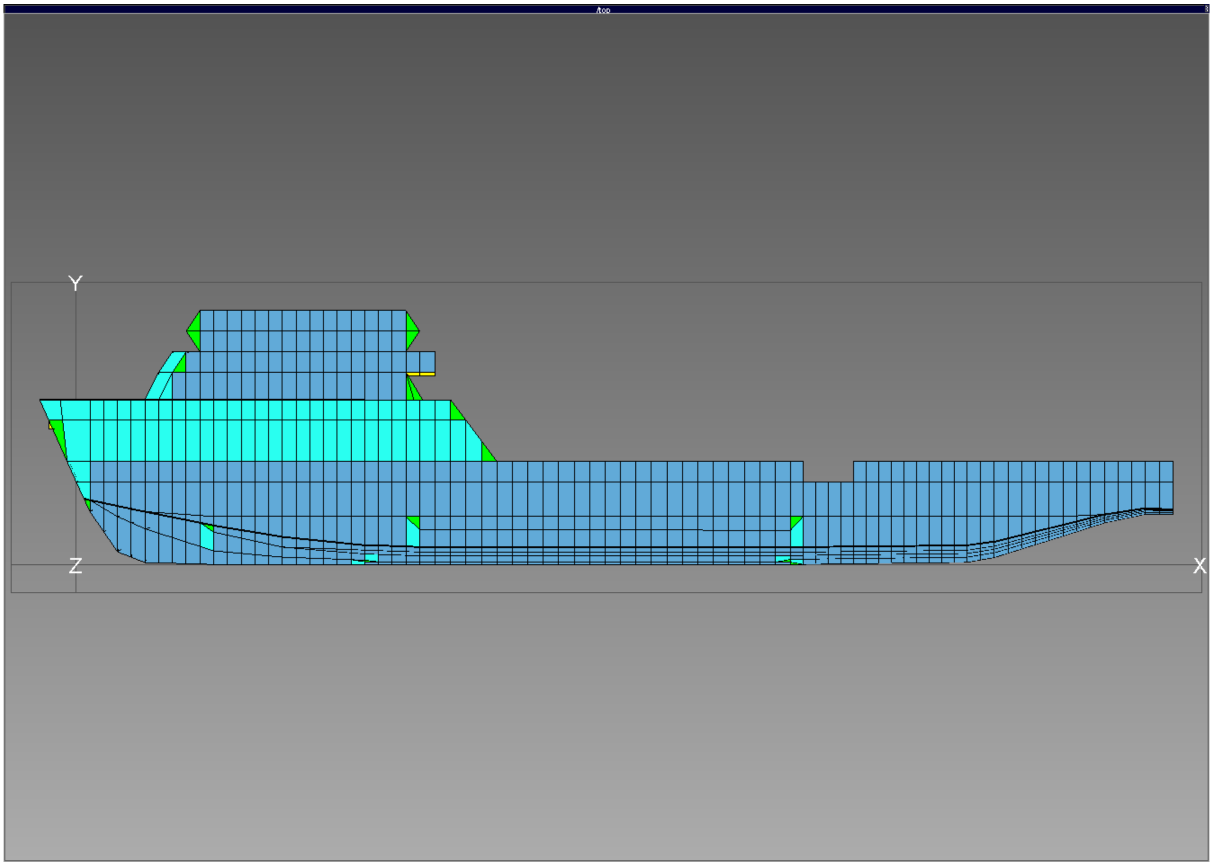
Plan View

Isometric Views

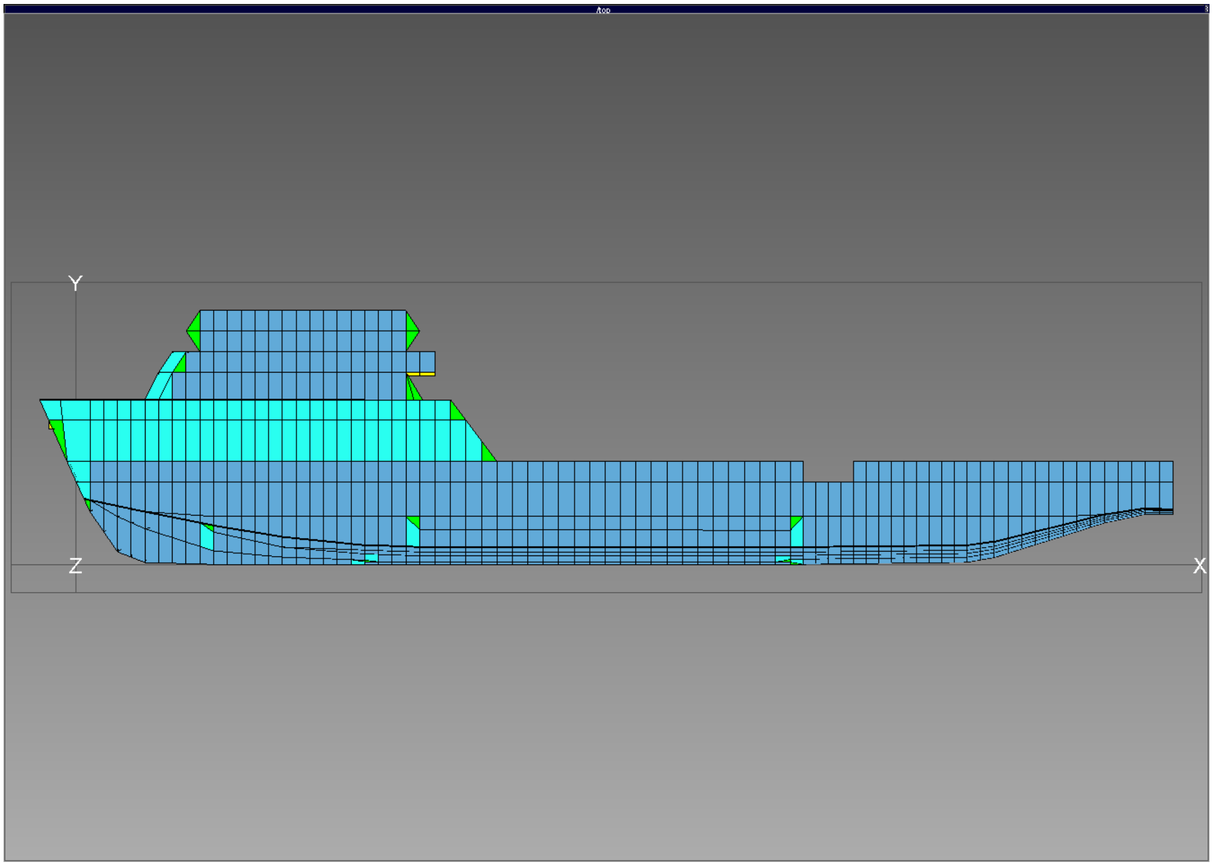
Body Plan



Profile View

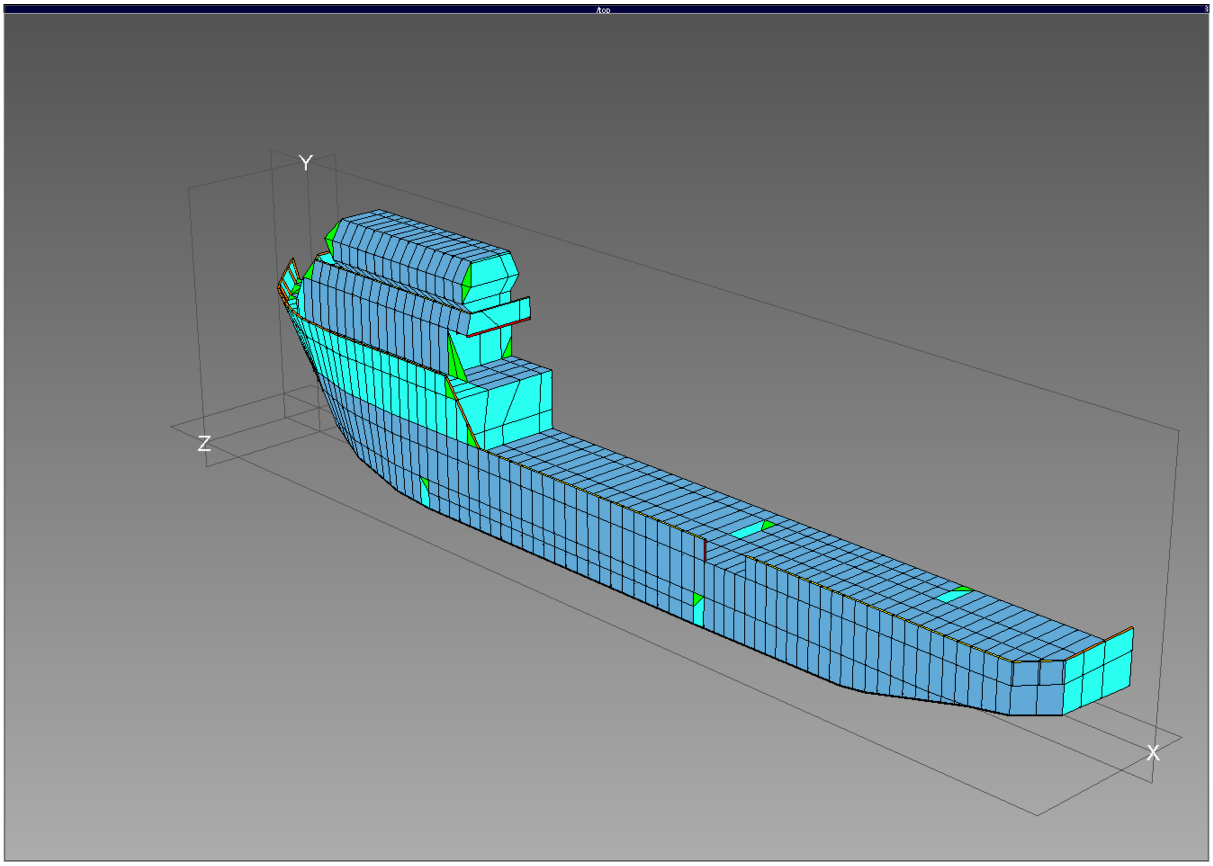


Plan View

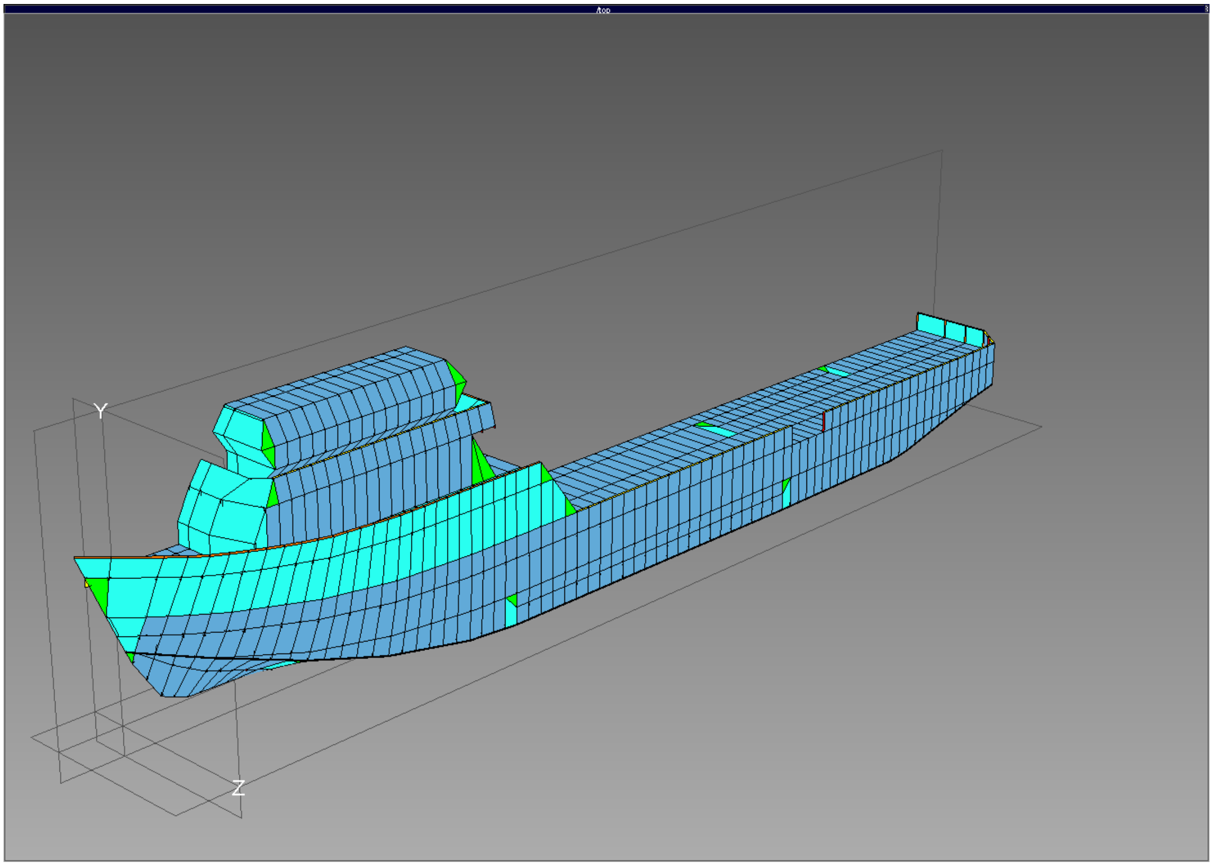


Isometric Views (SouthEast, SouthWest, NorthEast, NorthWest)

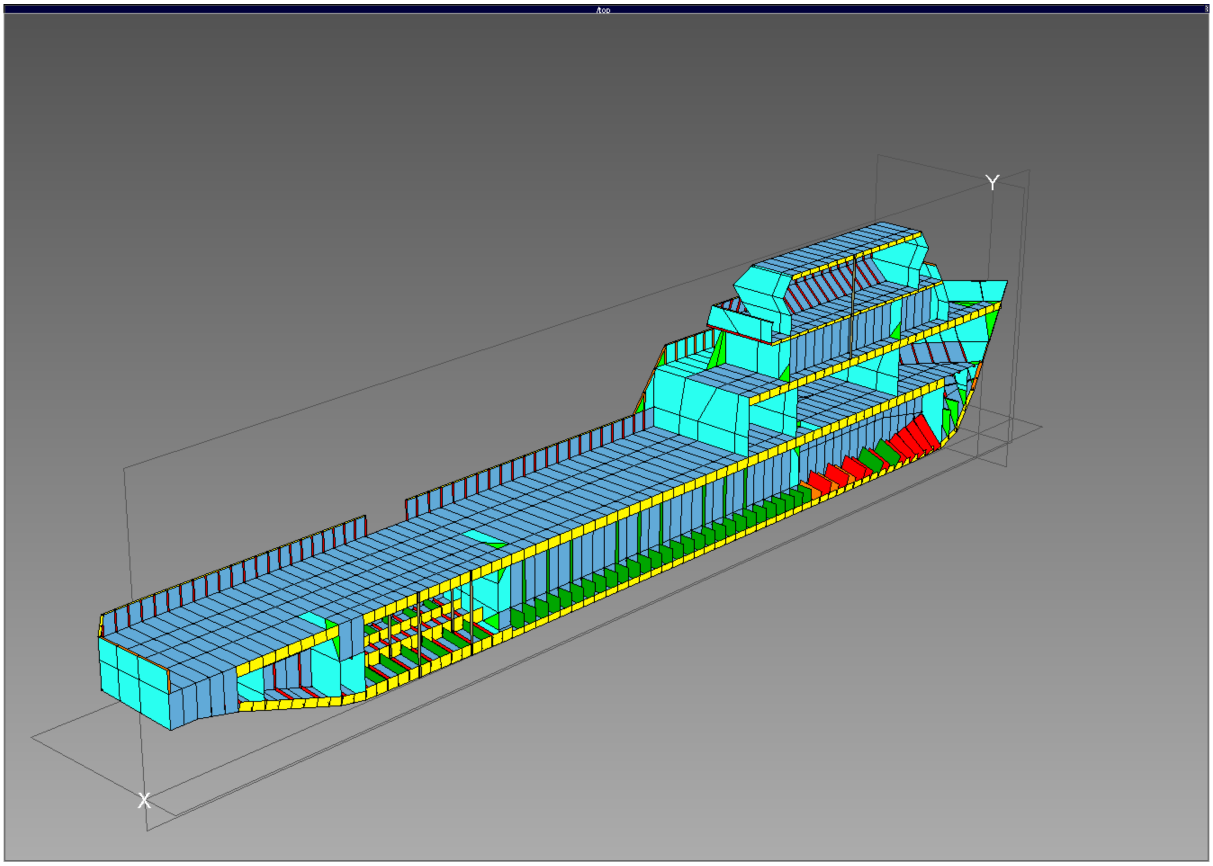
SouthEast



SouthWest

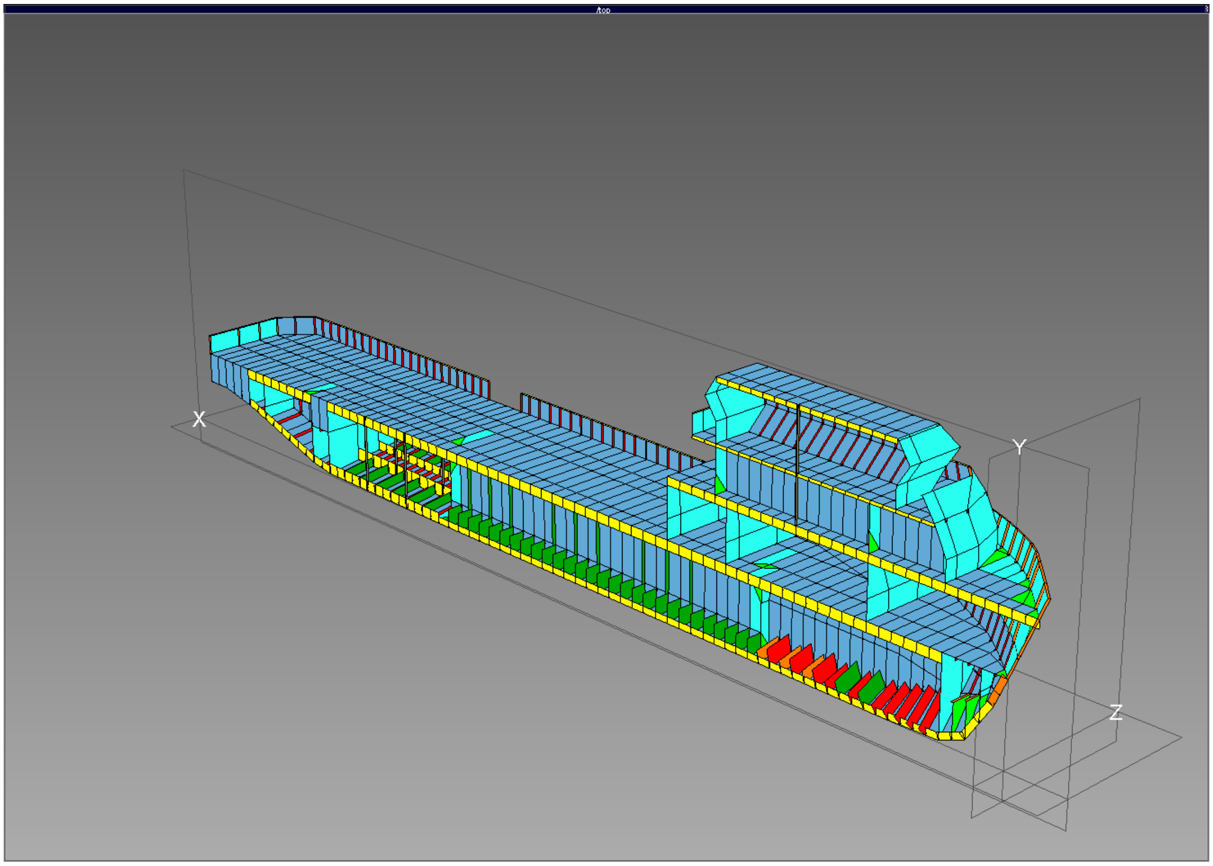


NorthEast



NorthWest

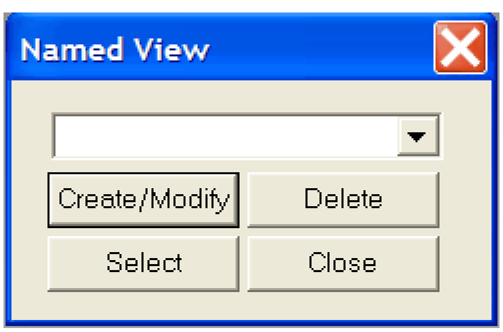
© 2014 Optimum Structural Design, Inc.



4.15 Named Views

MAESTRO allows the user to create their own model orientation view and save this view to be loaded later. In addition to saving the model orientation, the Named View will save the View Options, user-defined range, and view type (plate thickness, stress plot, element type).

The Named View menu can be launched from the **View > Set View > Named View...** menu or by right-clicking in the modeling space and selecting *Named View*.



Creating a New Named View

1. Set the model to the orientation and options desired.
2. Open the Named View dialog.
3. Type a name into the drop-down menu.
4. Click the *Create/Modify* button.

Modifying a Named View

1. Set the model to the new orientation and options desired.
2. Open the Named View dialog.
3. Select the Named View from the drop-down menu to modify.
4. Click the *Create/Modify* button.

Selecting a Named View

1. Open the Named View dialog.
2. Select the desired view from the drop-down menu.
3. Click the *Select* button.

Deleting a Named View

1. Open the Named View dialog.
2. Select the desired view from the drop-down menu.
3. Click the *Delete* button.

© 2014 Optimum Structural Design, Inc.

4.16 Using the Mouse & Shortcut Keys

[Left Mouse Button]

[Right Mouse Button]

[Mouse Wheel]

[Keyboard Shortcuts]

Left Mouse Button

The left button is used to select any menu item, toolbar item, parts tree item, and groups tree item. The left button is also used to interact with MAESTRO's many dialog boxes. In conjunction with Finite Element, Reference Points, Compounds, and Strakes dialog boxes, the user can select the closest entity. It is important to note that the user must first click in the ID field of the dialog box to select the entity of interest.

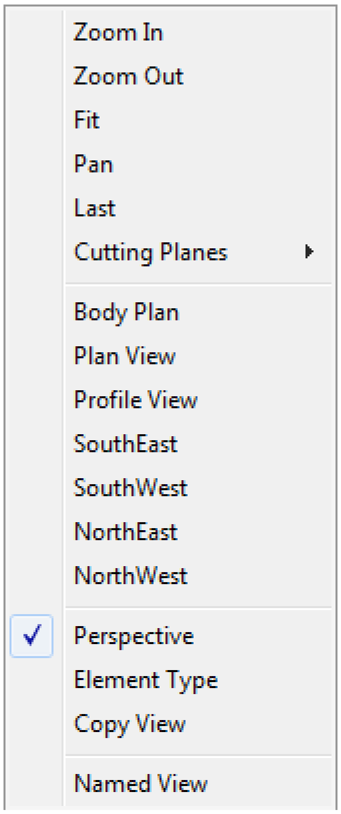
© 2014 Optimum Structural Design, Inc.

Right Mouse Button

The right button is used to invoke various quick access menus. The quick access menu will differ depending on where the cursor is located in the GUI.

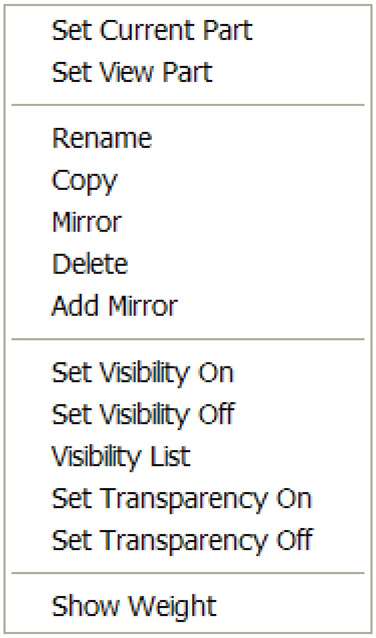
Main Display Area

When the mouse cursor is located in the main display area, the user, with the click of the right button, will access the menu shown below. This menu item allows the user to set the current view projection. Here, the user can choose from a list of viewing operations and standard views. This menu also allows the user to toggle a perspective projection on or off as well as copy the current display.



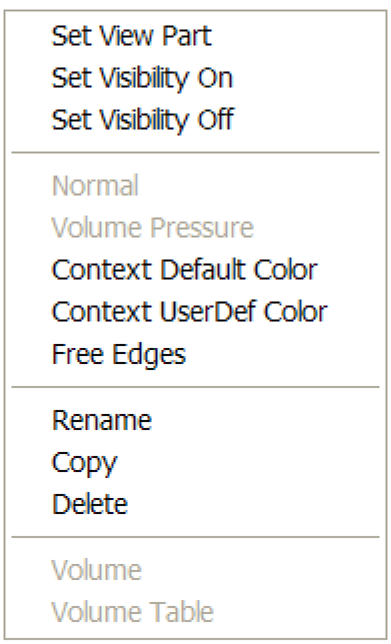
Parts Tree Area

When the cursor is located in the Parts Tree area, the user can access a parts quick menu. This menu allows the user to set part attributes, modify parts, and query parts, as shown below.



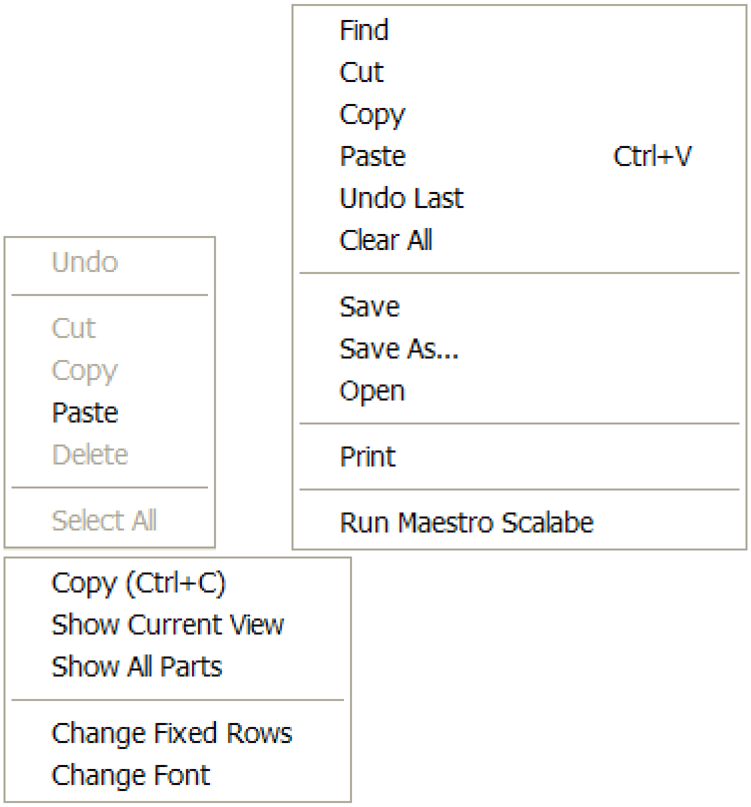
Groups Tree Area

When the cursor is located in the Groups area, the user can access a groups quick menu. This menu allows the user to set group attributes, modify groups, and query groups, as shown below.



Command Line, Output Window, and Grid Window

When the cursor is inside the area of one of these tabs, which are located at the bottom of the MAESTRO GUI, the right-click will access various quick menus. These quick menus are shown below.



Mouse Wheel

The wheel button is used to zoom and pan in your modeling space without using any MAESTRO commands.

Roll Wheel

Rotate the wheel forward to zoom in and backwards to zoom out. Use the Shift and CTRL key to accelerate zooming.

Hold Wheel

Hold down the wheel button and drag the mouse to dynamically spin the model around.

Hold Wheel + Hold Shift

© 2014 Optimum Structural Design, Inc.

Hold down the Shift key and the wheel button, then drag the mouse to dynamically pan the model.

Hold CTRL + Single Click Wheel

Hold the CTRL key down and then single click the wheel button, this combination invokes the Cutting Planes command.

Keyboard Shortcuts:

- *<Esc> = key invokes the cancel command*
- *<Ctrl + n> = Endpoints and Additional Nodes dialog*
- *<Ctrl + e> = Finite Element dialog*
- *<Ctrl + k> = Strake dialog*
- *<Ctrl + d> = Deletion dialog*
- *<Ctrl + g> = Groups dialog*
- *<Ctrl + r> = Restraints dialog*
- *<Ctrl + l> = Loads dialog*

4.17 Dialog Box Conventions

The MAESTRO interface makes extensive use of dialog boxes to guide the user in developing the structural model. Detailed descriptions of how to use these dialog boxes are provided in later sections.

Dialog boxes in MAESTRO are one of two types; *modal* or *modeless*. Modal dialog boxes require the user to complete interaction within the dialog and close it before continuing with any further interaction with the program. They are used for operations that are performed infrequently and which can be completed in a single train of thought. Examples of operations in MAESTRO which employ modal dialogs include setting the units system, setting MAESTRO job information, and setting viewing options. *Modeless* dialog boxes, on the other hand, allow the user to interact with the any part of the program while they are open. They are used for repetitive operations and for operations where the user needs to interact with the display area while the dialog box is open. For example, when creating many structural elements, the user would not want to open the Finite Elements dialog box to create each element. Furthermore, while the Finite Elements dialog is open, the user might like to select a node in the Main Display area. This is only possible with a modeless dialog box. Most of MAESTRO's dialog boxes are modeless including the node dialog, element dialog, strake dialog, compound dialog, materials dialog, properties dialog, and stiffener layout dialog. In addition to the aforementioned benefits, modeless dialog boxes allow the user to switch between creating nodes, elements, and strakes without having to individually open and close each dialog.

Modeless dialog boxes that contain references to other data (e.g., material and/or scantling properties) have buttons that provide quick access to the dialogs used to define this data. For example, the dialog for creating Quadrilateral elements contains buttons labeled Property and Layout, which relate to the plating and the stiffeners, respectively. Clicking these buttons will invoke the appropriate dialog box to allow the user to review the values associated with a particular data item, modify that item if desired, or define a new item. For example, plating has two properties: thickness and material type. If the user was creating a new quad element and found that the desired combination of thickness and material type was not in the Property drop-down list because it had not yet been defined, he/she could click the Property button to define the new combination.

4.18 GUI Conventions

MAESTRO's graphical user interface is compatible with Microsoft’s Windows® operating systems. As such, most of the interface conventions will already be familiar to the user. The purpose of this section is to document those interface conventions which are peculiar to MAESTRO such as initiating interactive agents and getting the next available ID.

The first convention worth noting relates to use of the mouse. It is assumed that a two-button + "wheel" Windows-compatible mouse is available. In practically all instances where a reference is made to a mouse click or a selection with the mouse, it is intended that the user click the left mouse button. In any case where it is necessary to deviate from this convention, a special note will be made.

Part of the power of MAESTRO lies in its ability to link complex descriptions of structural items to a simple visual description in the graphics display. Take the simple case of a strake. Strakes contain a multitude of data describing their behavior in the structure as a whole. It is useful for the user to simply point at a strake of interest in the graphical display in order to query that element for its structural data. Once the user has selected the strake of interest, he/she may want to change one of the strake’s defining endpoints by clicking on a new endpoint in the display. Performing these operations involves initiating an interactive agent which informs the program that you are about to select something in the graphics display and you wish the program to do something in response. In MAESTRO the mechanism used to initiate interactive agents from a dialog box is to set the focus in the field in question. For the strake example, the user clicks (with the left mouse button) in the ID field to initiate the agent and then clicks in the graphics display on the strake of interest. If the selection is successful, all of the fields in the strake dialog box will be updated with the data for the selected strake. If the user now wishes to define this strake’s endpoint 1, click in the endpoint 1 field to initiate the agent then select the desired endpoint in the graphics display. This convention of setting the cursor focus to a field to initiate an interactive agent occurs in all of the dialog boxes where such operations are logical.

Another consistent interface convention in MAESTRO relates to specifying item identification numbers (ID’s) for various structural items. Many of the data items found in MAESTRO occur in large lists such as elements, nodes, properties, materials, etc. It is therefore necessary to have a unique means of identifying these items for them to be useful. In MAESTRO this unique identification takes the form of a numerical ID. ID values must be positive integers. Most times when creating a new item from a dialog box the user is not concerned with what ID an item has but simply wishes to get a unique ID. To accomplish this in any of MAESTRO’s dialog boxes the user merely clicks the ID button. This will get the next available ID in the particular list in question. Note that any numbering holes will be ignored. If the user has 10 properties defined numbered 1 through 10, deletes property number 5, then clicks the ID button, the ID field is filled with 11 not 5.

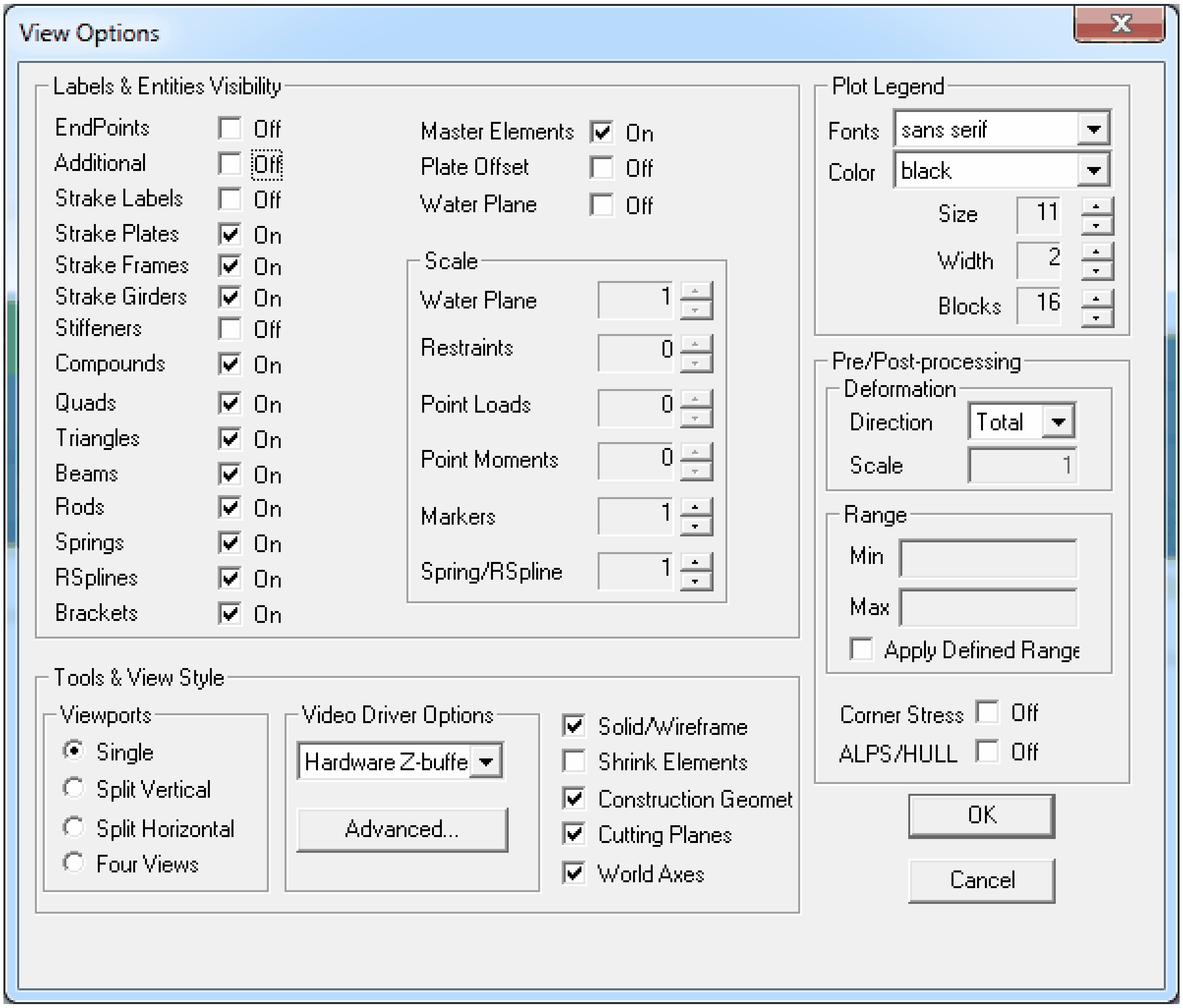
Although all structural items in MAESTRO must have an ID many of them can also have a name. Named items include materials, properties, stiffener layouts, strakes, compounds, and of course modules and substructures. In each of these cases (with the exception of modules and substructures) the names are optional, meaning that they can be left blank. However, it is often very convenient to name items for easy reference later. The clearest example of this is material names. It might be difficult for the user to remember that material 13 was supposed to be aluminum, but by naming this material it would be obvious later, when specifying a plate's material, which material was to be selected. The names of like items must be unique within a particular list. The user could not have two materials named aluminum with slightly different properties. In general the user should make item names as detailed as necessary to avoid future confusion about what is contained in that item.

4.19 View Options

Toolbar	
Menu	**View > Options...**

The View Options dialog allows the user to control a wide variety of viewing options including element/node visibility, rendering algorithms, viewport layout, etc.

The View Options dialog can be launched from the toolbar, or from the **View > Options...** menu.



Labels & Entities Visibility

The view of the elements listed in the dialog can be toggled *On* and *Off* by clicking the check box. Several of the elements can also be toggled to *On w/label* which will show element IDs, Water Plane CGs, or Strake IDs.

Scale

The Scale section of the View Options dialog allows the user to scale the graphic view of several entities. For example, the Restraints scale can be used to lengthen the graphic representation of the restraints from the **View > Restraints** menu option.

Tools & View Style

Viewports

This section of the View Options dialog allows the user to switch between a single, split, or four view workspace. For more detailed information on these views, please see the Workspace Layout section.

Video Driver Options

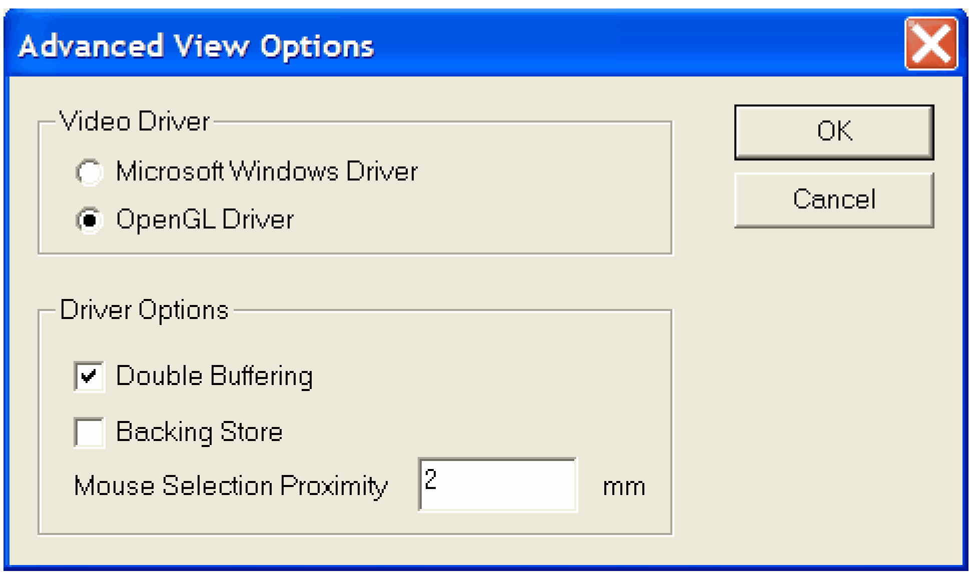
The recommend settings in the View Options dialog are:

Rendering Options: Hardware Z-buffer

Video Driver: OpenGL Driver

Driver Options: Double Buffering

The Video Driver and Driver Options can be found by clicking the *Advanced...* tab in the View Options dialog.



The Tools & View Style sections also allows the user to select between a variety of viewing options and whether construction geometry, cutting planes and the world axes are visible.

Plot Legend

This section allows the user to change the default settings of the legend when plots are shown.

Pre/Post-processing

Deformation

Direction: The direction option allows the user to plot the deformation in all or only one direction (i.e. X, Y, Z) at a time.

Scale: The deformation scale allows the user to adjust the magnitude (e.g. "1" is the true deformation scale) of the deformed model view. Note: the model must be in a deformed view to adjust the deformation scale.

Range

When a view is selected with a legend of values, this range is automatically populated with the maximum and minimum values. The Min and Max can be overridden with user-defined values by checking the *Apply Defined Range* box and inputting new numbers.

Corner Stress

When this option is checked, the stress contour plot is based on elemental nodal stress. If it is unchecked, the stress contour plot is based on the average elemental centroidal stress.

ALPS/HULL

When ALPS/HULL results are available, this option allows the user to turn on results, turn on results with the moment curve displayed, or turn off the results.

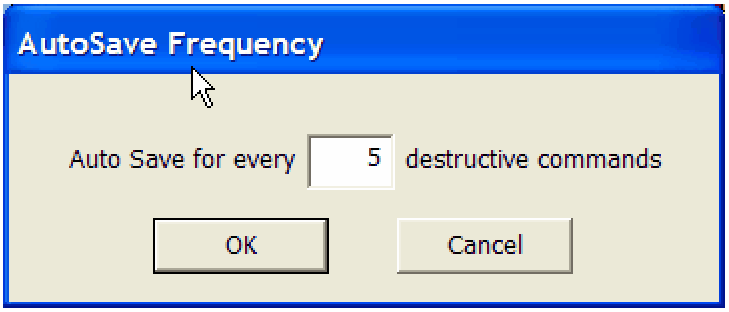
4.20 Auto Save and Recover Model

Auto Save

MAESTRO has an auto save function, which will automatically backup the model after a set number of *hard* commands. A "hard" command is any command where something in the model is created, modified, or deleted. For example, creating a strake or modifying a material property is considered a "hard" command. Changes to the graphics view, i.e. model view orientation, is not considered a "hard" command.

To set the frequency of the automatic backup, select **File > Autosave...** from the menu.

This will open the AutoSave Frequency dialog box.



Input the number of "hard" commands before each auto backup and click *OK* to save. An *#autosav.rcv* file will automatically be generated in the MAESTRO/System directory.

Model Recovery

Option #1

In the event that MAESTRO does not respond, or the program crashes, you can recover your model by selecting **File > Recover Model** from the menu. This will access the *#autosav.rcv* file automatically generated during Auto Save (as described above). The *#autosav.rcv* file is created based on the user-defined AutoSave Frequency and is stored in the MAESTRO/System directory.

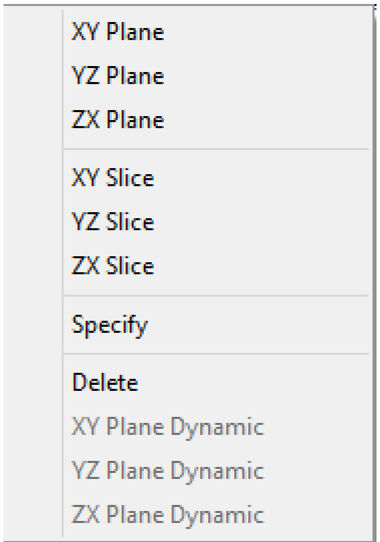
Option #2

Alternatively, a *filename.*bck file is created and saved in the location of the current .mdl file. Open the location where the original model was saved. You will see a *filename*.mdl and *filename*.bck. The *filename*.bck file can be opened directly by MAESTRO; this is your recovered modeler file.

4.21 Cutting Planes

The cutting plane functionality can be found under the **Tools > Cutting Planes**. Cutting planes can also be found by right clicking and selecting **Cutting Planes.**

© 2014 Optimum Structural Design, Inc.



Cutting planes are planes in space which delineate visible and invisible regions of space. A user can insert a cutting plane into the model and specify which side is visible and which is invisible. This can be very useful at times, such as when wishing to view only the interior of a full hull model. By placing a longitudinal cutting plane at the origin and hiding the port side, the interior of the starboard model is made visible. The user can create cutting planes aligned with any of the principal axes or define a cutting plane with arbitrary orientation. In addition, the user can specify a cutting slice in which two closely spaced cutting planes are placed in the model with their visible side being the region between them.

As with all of the View commands, Cutting Planes only alters the view on the display; it does not actually change the model. The user can add any number of cutting planes to the model and they are stored with the particular part in which they are defined. Cutting planes do not persist in the model if it is saved and retrieved.

YZ Plane Places a cutting plane in the YZ plane. YZ Plane issues a prompt to specify first a point on the cutting plane and second a point on the visible side of the cutting plane. These points may be specified by clicking on a point in the modeling space or entering the coordinates in the form of (x, y, z), at the command line.

ZX Plane Places a cutting plane in the ZX plane. ZX Plane issues a prompt to specify first a point on the cutting plane and second a point on the visible side of the cutting plane. These points may be specified by clicking on a point in the modeling space or entering the coordinates in the form of (x, y, z), at the command line.

© 2014 Optimum Structural Design, Inc.

XY Plane	Places a cutting plane in the XY plane. XY Plane issues a prompt to specify first a point on the cutting plane and second a point on the visible side of the cutting plane. These points may be specified by clicking on a point in the modeling space or entering the coordinates in the form of (x, y, z), at the command line.
YZ Slice	Erases everything from the display not lying in the specified YZ plane (plus the thickness of the slice). YZ Slice issues a prompt to specify a point on the desired plane. This point may be specified by clicking on a point in the modeling space or entering the coordinates in the form of (x, y, z), at the command line. The default slice thickness is defined internally in the Modeler as 10 cm.
ZX Slice	Erases everything from the display not lying in the specified ZX plane (plus the thickness of the slice). ZX Slice issues a prompt to specify a point on the desired plane. This point may be specified by clicking on a point in the modeling space or entering the coordinates in the form of (x, y, z), at the command line. The default slice thickness is defined internally in the Modeler as 10 cm.
XY Slice	Erases everything from the display not lying in the specified XY plane (plus the thickness of the slice). XY Slice issues a prompt to specify a point on the desired plane. This point may be specified by clicking on a point in the modeling space or entering the coordinates in the form of (x, y, z), at the command line. The default slice thickness is defined internally in the Modeler as 10 cm.
Specify	Cutting planes can have any arbitrary orientation in space. This command places a cutting plane with a user-specified orientation. The user is prompted to specify a vector normal to the plane, a point on the plane, and a point on the visible side of the plane. These points may be specified by clicking on a point in the modeling space or entering the coordinates in the form of (x, y, z) at the command line. For example, to generate a cutting plane that is parallel to the Y axis, but is at a 45° angle to both the X and Z axes, specify a normal vector of (1,0,1), an origin of (0,0,0), and the visible side by clicking on a point on the model.
Delete	Removes all cutting planes from the model and restores the active viewport.
Dynamic XY	A dynamic XY cutting plane allows for the user to click and dynamically cut into the hull, starting at centerline and cutting into either port or starboard by dragging the mouse in the respective direction.
Dynamic YZ	The dynamic YZ plane cuts longitudinally from the x position that the user clicks along the vessel. For most accurate results for initial cut point try cutting in profile view.
Dynamic ZX	The dynamic ZX plane cuts vertically from the y position that the user clicks above the baseline. For most accurate results for initial cut point try cutting in body plan view.

© 2014 Optimum Structural Design, Inc.

4.22 Security Devices

The topics in this section provide details of how to updated the security device, the Fast Lock utility and details regarding the use of network security devices.

4.22.1 Updating Security Device

It is very important you first have the two passwords available that were provided by the distributors. If you do not have these passwords, you should contact the distributors before continuing. These passwords will allow the proper operation of MAESTRO and other optional modules.

If you are experiencing any problems with your security device please contact support@maestromarine.com.

Procedure for Updating the Security Device

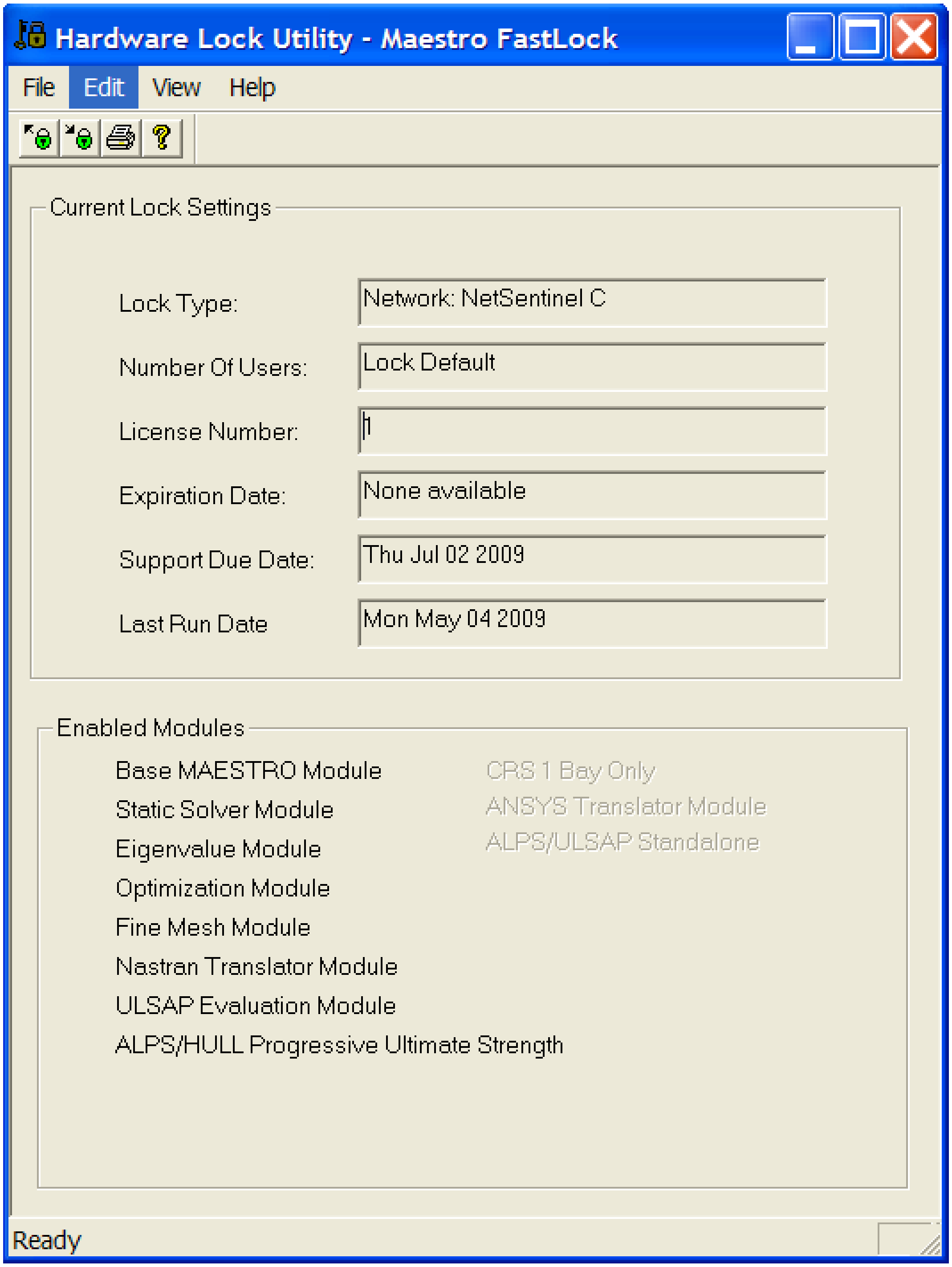
To update your security device, perform the following steps:

1. Locate passwords that have been provide by the distributors or your MAESTRO dealer. If you do not have passwords, you will only be able to view the security device information. Please contact support@maestromarine.com regarding passwords.

2. Attach the security device to the USB port.

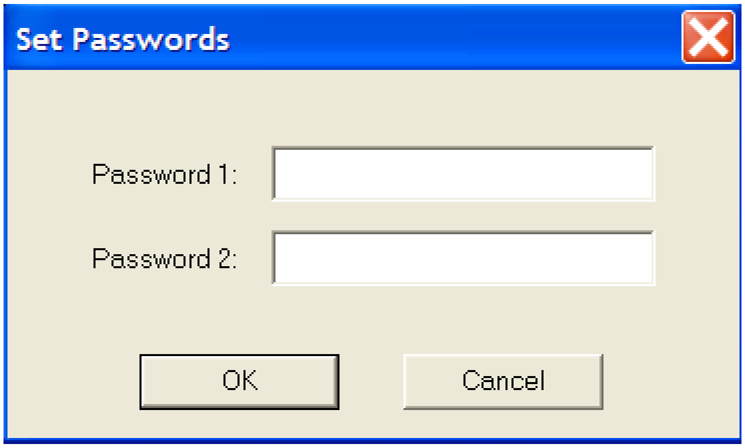
NOTE: PARALLEL PRINTER PORT SECURITY DEVICES ARE NO LONGER SUPPORTED.

3. Click *MAESTRO Fast Lock* from the **Start > Program Files > MAESTRO** menu. The dialog below will appear.



4. Choose *Read Lock* from the File menu. At this point the Fast-Lock utility will report your current options.

5. Choose *Set Passwords...* from the Edit menu. The dialog below will appear.



6. Enter both Password 1 and Password 2.

7. Click OK. The lock is now updated.

4.22.2 Fast Lock - Security Device Utility

Fast Lock is a utility that allows the user to update and maintain their MAESTRO security device. This utility provides the capability to remotely update the user's maintenance and support as well as update the user's optional modules. Outside of updating the security device, the Fast Lock utility is used to inform the user of the device's current configuration. If you are experiencing any problems with your security device please contact technical support at support@maestromarine.com.

Interface

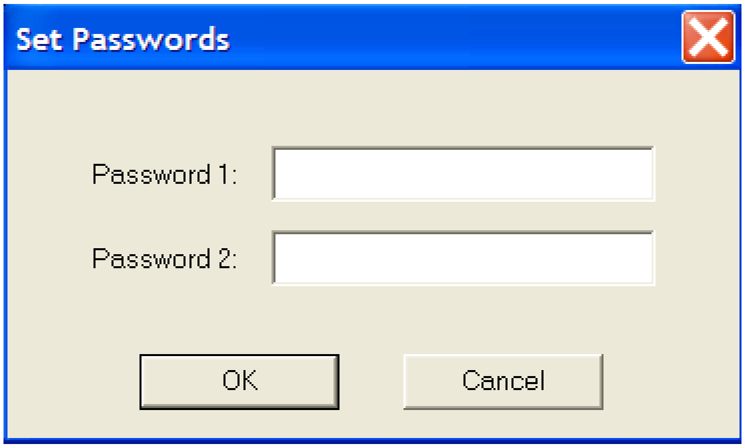
The Fast Lock interface is very easy to use and understand (nearly self-explanatory); a brief description follows.

File Menu

Using the File menu, the user can execute the Read Lock, Print, Print Setup, and Exit commands.

Edit Menu

The user can execute the *Set Passwords...* command using this menu item. The Set Passwords dialog, seen below, will then prompt the user to enter two passwords, which is provided by the distributors or your local MAESTRO dealer.



View Menu

Using this menu, the user can toggle on/off the Toolbar an Status bar.

Help Menu

This menu provides Fast Lock version information.

Current Lock Settings

This area of the Fast Lock interface displays security device information. This information includes Lock Type, Number of Users, License Number, Expiration Date, Support Due Date, and Last Run Date.

Enabled Modules

This area of the Fast Lock interface displays modules that are enabled.

4.22.3 SafeNet Network Lock

MAESTRO relies on a hardware lock to provide license security. MAESTRO can be run in a network environment in which MAESTRO is installed and used on many client workstations with security handled by a single hardware lock installed on a central server machine. Interaction between MAESTRO and the hardware lock is handled by a security server program that must be running on the server machine where the lock is installed. The system that handles network security is known as Sentinel Protection Server and is developed by SafeNet, Inc.

The installation of all security components, including the Sentinel Protection Server, is bundled into the installation of MAESTRO. Please refer to the SafeNet *ReadMe.pdf* file, which can be found in the following directory: *...\MAESTRO\System\Sentinel*

NOTE:

MAESTRO has a mechanism to release a network license 10 minutes after a crash occurs. Alternatively, a user with Administrator computer privileges can *Restart* the Sentinel Protection Server manually via the MS Windows *Services* dialog. See the *About Sentinel License Monitor* section of the SafeNet *ReadMe.pdf* for further troubleshooting tips.

4.23 Installation Directory and Sample Files

By default, MAESTRO is installed in a *MAESTRO* directory located in the user's *Program Files*. Upon a successful installation of MAESTRO, the following two directories will be created: *Help* and *System*. A third directory *Models and Samples* will be created in the user's *ProgramData* folder.

Help Directory

The Help directory contains the electronic manual in both Compiled HTML Help File (*.chm) and Adobe Acrobat (*.pdf) formats. Supporting manuals, i.e. MAESTRO Basic Features Tutorial, Legacy Data Prep Manual, and MAESTRO Release Notes, are also included in this directory.

System Directory

The System directory contains all of the necessary files that allow MAESTRO to function properly. These files should not be removed or modified. The System directory also includes support files and documentation for the Sentinel security device in the event troubleshooting is required.

Models and Samples Directory

The Models and Samples directory contains a variety of MAESTRO example models, verification models, sample programs (C++/C#) and other data to assist the user in learning and understanding MAESTRO. All of the data found in this directory is appropriately referenced throughout the documentation. If you are running Windows Vista or Windows 7, your Models and Samples directory will be installed under the user's *Program Data* folder. Note: this folder's view may be set to "hidden."

Geometry/Finite Element Modeling

5

5 Geometry/Finite Element Modeling

The topics in this section provide detailed information on the MAESTRO functionality used during the Finite Element Modeling stage of an analysis.

5.1 Model Organization

In the design of large structures it is usually advisable to divide the task into a few distinct subtasks in order to maintain a good overview and control of the design. Most large structures can be reduced to several levels of component structures for which the design and analysis is relatively independent. Such a structure can best be modeled by subdividing it into a hierarchy of parts, down to the module level, and then constructing each module using a three-dimensional mesh of nodes and appropriate groupings of finite elements. As shown below in Figure 1, the MAESTRO structural modeling is organized into four levels: members (elements), strakes, modules, and substructures.

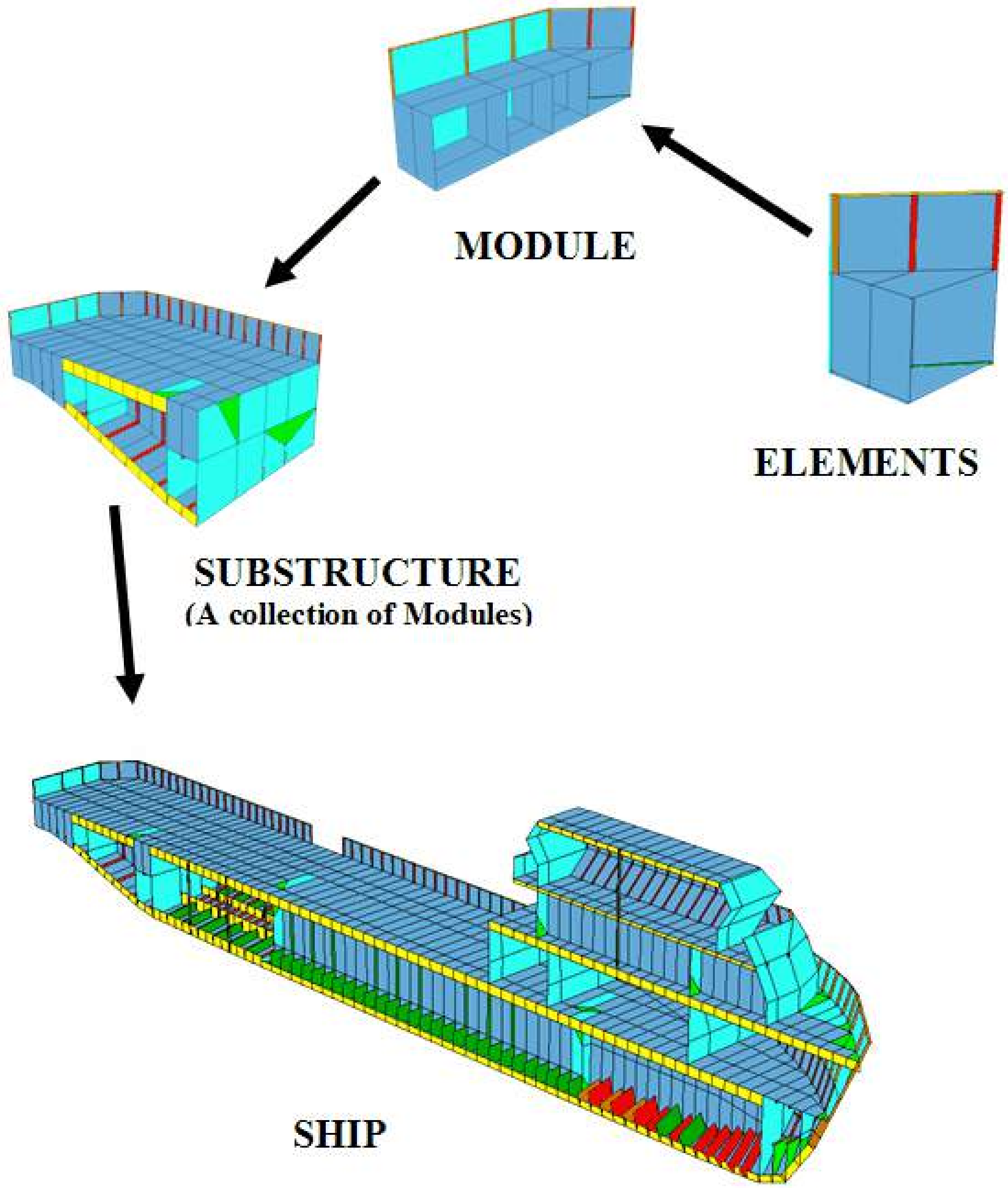


Figure 1 Hierarchy of Structural Modeling

MAESTRO Elements

The basic unit of structural modeling is a principal member such as a transverse frame (red beam), stiffened panel (dark blue), girder (yellow beam), rod (brown line), etc., as shown

© 2014 Optimum Structural Design, Inc.

below in Figure 2. In order to have an efficient interaction between the finite element analysis and the optimization, the finite elements in MAESTRO are in most cases the same as the principal members. The elements are therefore relatively large; e.g. a complete panel from one deck to another and from one frame to the next, or a corresponding segment of a transverse frame or longitudinal girder. The finite element types most commonly used in MAESTRO are described below.

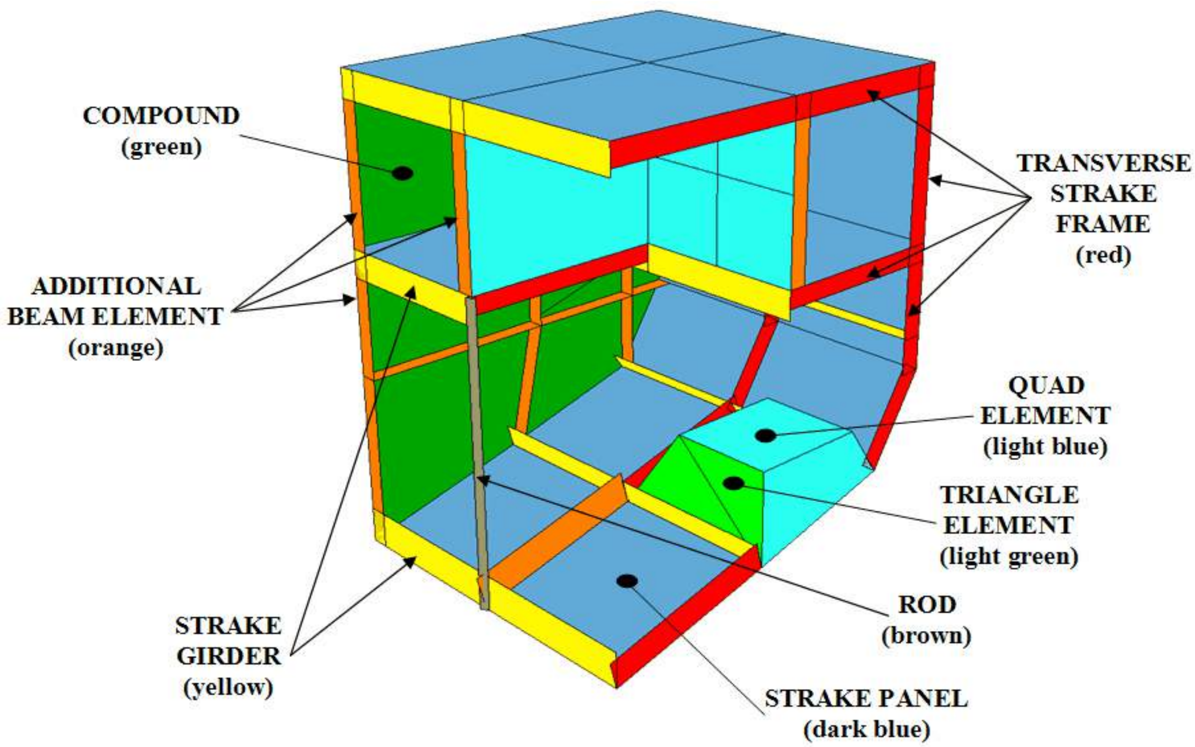


Figure 2 MAESTRO Elements

CQuadR Stiffened Panel

This is the most general and hence most useful panel element in MAESTRO, and it is the default element for all panels (strakes and quads). It is an orthotropic shell quadrilateral that automatically includes the bending stiffness if the stiffeners in either the longitudinal or the transverse direction. It is the same element as in the NASTRAN program, but in MAESTRO the (quite large) task of calculating the orthotropic properties is automated. The CQuadR element is a 4 noded flat shell element with each node having 6 degrees of freedom.

Hybrid CBAR Beam Element

© 2014 Optimum Structural Design, Inc.

This element has been developed for modeling of beams (frames, girders, additional beams) attached to plating. The plating acts as one of the flanges of the beam, and MAESTRO takes this into account in calculating the beam's flexural properties, using a user-specified effective breadth of this plate flange. In contrast, the axial stiffness is for the beam only (which is why element is called "hybrid") because the in-plane stiffness of the plating is accounted for by the panel element. The neutral axis location and the moment of inertia of each hybrid beam are automatically calculated and printed. In a finite element model, a beam element is nothing more than a "line" element, joining two nodes and setting up a relationship between their displacements and rotations. For this reason most finite element programs display a beam element simply as a line. In that case they do not look like real beams, and they can be difficult to see, especially in a large model. In order to make the beam elements more visible and more realistic, MAESTRO displays them as "web panels"; that is it displays the web of the beam, but not the flange.

CTRIR Triangle Element

This is a flat, constant strain element that can be placed between any three nodes in a module.

CRod

The rod element is a pin-jointed bar which can connect any two nodes in a module. The rod element can carry axial force and axial torsion only.

Compound

This consists of any user-defined assemblage of in-plane membrane elements (bar, beam, triangle and quadrilateral elements) in any transverse section of any module.

CELAS2 Spring Element

The spring element is a linearly elastic spring which can connect any two nodes in a module. These elements can carry either force or moment loads. Forces in the spring element will cause translational displacement and moments will cause rotational displacement.

Restraint Elements

MAESTRO has three types of restraint elements. The RBE2 element is a rigid element, while the RBE3 and RSPLINE element are interpolation elements. RSpline elements can act as a boundary element when conducting a fine mesh model. The three restraint elements offered by MAESTRO are described below:

RSpline: Transmits displacements to a group of nodes located between the two defined reference nodes. These translations are prescribed from the slope and displacement of a flexible, tubular beam element connecting the two reference nodes. This is the type

© 2014 Optimum Structural Design, Inc.

of RSpline element used to transfer displacement from coarse mesh nodes to fine mesh nodes.

RBE2: Used to create a rigid body connection between a group of dependent nodes and one independent reference node. An RBE2 element is usually used to model areas that are very stiff compared to the adjacent structure.

RBE3: Used to transmit motion between a group of nodes and one reference node. The motion at the reference node is the least square weighted average of the motions at the other nodes. An RBE3 element is used to transfer forces and moments from the reference node to several dependent nodes without adding stiffness to the structure.

Strakes and Modules

Figure 3 below, shows an example of a module, which is a portion of structure having the following properties:

- It has one direction (regarded as its "lengthwise" direction) along which there are regularly spaced transverse planes or sections, which constitute either the locations or the boundaries of some or all of the individual structural members.
- Within each transverse section the layout of the members is similar, and in the lengthwise direction the geometry of each section is either constant or linearly varying depending on the endpoint definitions.
- Members that occur repeatedly tend to have the same local structural dimensions; that is, plate thickness and flange and web widths and thicknesses. (These are sometimes referred to as "scantlings").

Because of a module’s geometric regularity and uniformity of scantlings in the lengthwise direction, it is convenient to regard a module as being made up of "strakes". As shown in Figure 3, a strake is a lengthwise row of stiffened or un-stiffened panels, frame segments and, optionally, a longitudinal girder along edge one of the strake.

The location of beams and panels is not limited to strakes; they can also be inserted individually at other locations and at various orientations.

© 2014 Optimum Structural Design, Inc.

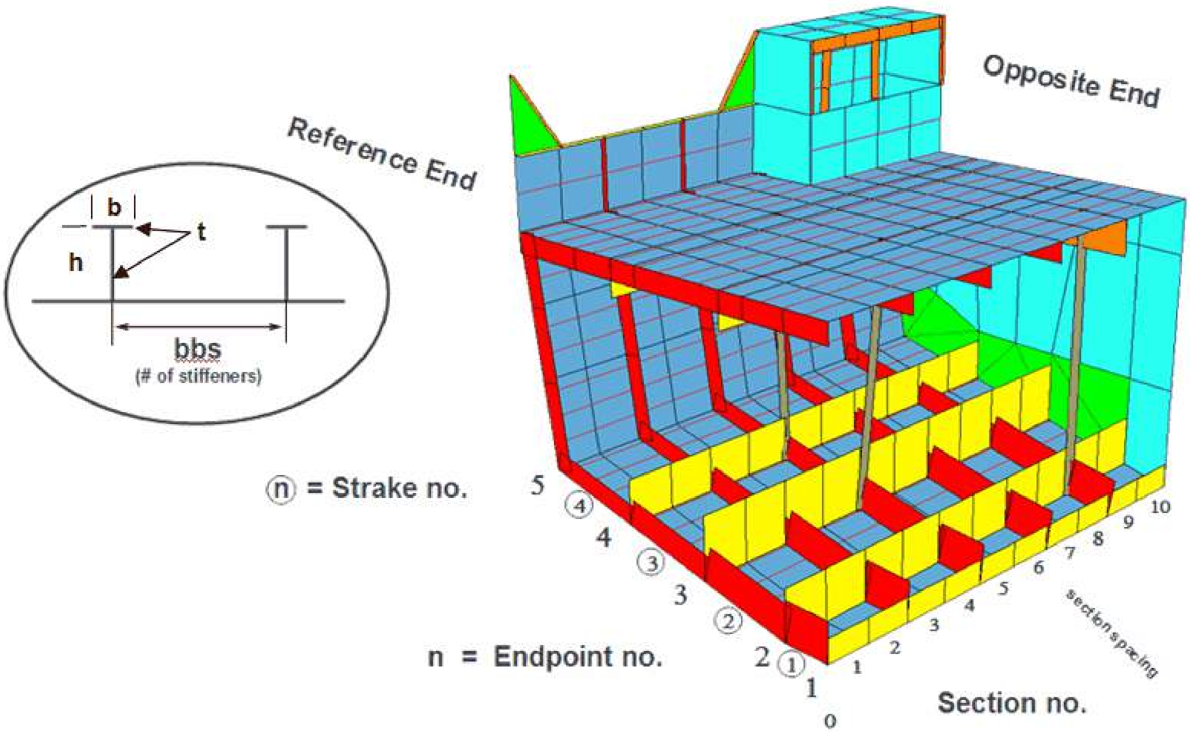


Figure 3 MAESTRO Terminology

Each module has its own three-dimensional nodal mesh defined within the module's reference system. This mesh is generated by specifying the location of endpoints in a transverse plane at both ends of the module. These end planes are called the reference plane and the opposite plane. The program then creates a node at each user-defined section location between the reference and opposite endpoint.

Once the nodal mesh has been defined, strakes are then created by specifying the pair of endpoints that are in line with the sides of the strake. Strakes and sections are numbered sequentially, as are the intervals between the sections. The program uses the terms "strake", "endpoint", "section" and "module" to refer to locations within the structural model. For example, the first panel in the keel strake is located at "strake 1, section interval 1". Each strake extends for the full length of the module and has uniform scantlings. However, shorter strakes and changes in scantlings can be obtained by assigning two (or more) strakes to the same pair of endpoints and then selectively deleting portions of each. Also, various combinations of scantlings can be obtained by deleting the unwanted strake elements and inserting "additional" elements in their place. MAESTRO provides an element deletion feature that makes this relatively easy.

Frames need not occur at every section; they can occur at every other section, specific sections only, or not at all. The total length between frames is referred to as a bay. This means that each structural panel, which by definition extends from one frame to the next can be comprised of two panel elements or three panel elements, or any number (n). The only limitation is that n must divide evenly into the highest section number in the module. Note that section numbering begins with zero.

Any of the nodes lying in the two end planes of a module can be moved lengthwise, out of their plane, by any (differing) amount, thus allowing the modeling of complex end shapes and the joining of modules at arbitrary angles or orientations. Note that this does not move the end sections or change the end values or section spacing.

Substructures

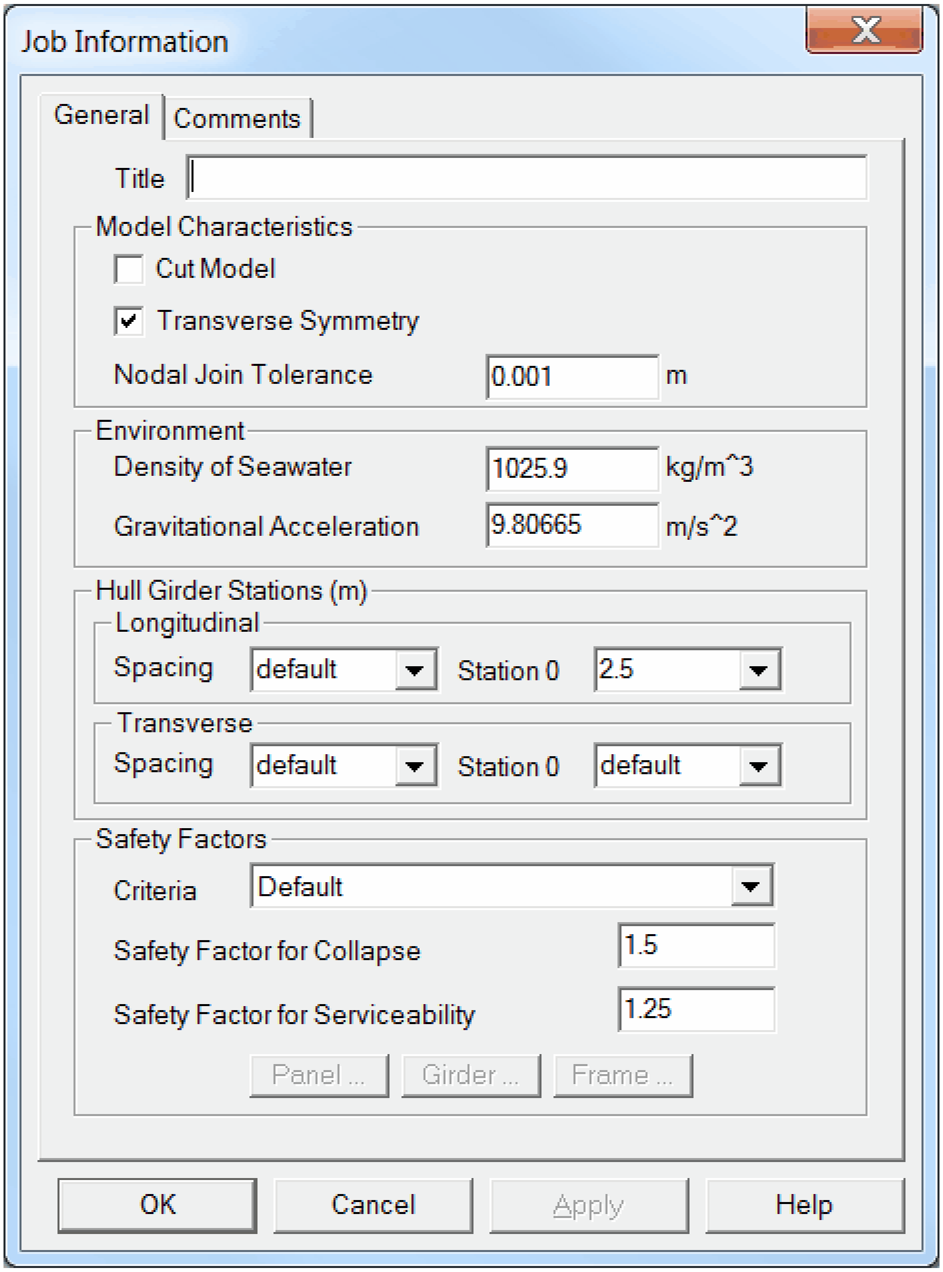
A *substructure* is a set of modules which are generally associated with each other, e.g. , bow, midbody, and stern. The substructures are merely a convenient means of grouping modules together; there are no specific functions or operations associated with substructures. A substructure's origin is defined with respect to the global origin and may, like modules, be oriented in any direction about the X, Y, and Z axes. Module origins and axes are defined relative to their parent substructure's origin and axes. For example, if the midbody substructure origin is defined as being at X = 55m, Y = 0m and Z = 0m with respect to the global origin, then a module within this substructure which starts 75m forward of the global origin, or 20m forward of the substructure's origin, would be defined with a module origin of X=20m, Y=0m, and Z=0m.

5.2 Defining Job Information

Toolbar	
Menu	**File > Job Information...**

The MAESTRO Job Information dialog can be accessed from the **File > Job Information...** menu, or from the toolbar. This is where global settings and the job title can be defined.

© 2014 Optimum Structural Design, Inc.



Title - provides the user with a location to associate a Job Title or description

Model Characteristics
The Cut Model option provides the ability to apply bending moments at bounding module locations. This is useful if it is the intent to not generate a complete model (only 0.4L amidships for example) and apply bending moments at the ends.

The transverse symmetry option notifies MAESTRO that it is the intent to construct a "half model" and thus MAESTRO will automatically mirror the model (internally) about the centerline for analysis. This will also automatically restrain (internal to the calculation) the model on the centerline.

The Nodal Join Tolerance field sets the value, in the currently selected units, in which the nodes will be automatically joined by the modeler and solver. Setting too small a value (less than 5 mm or so) may cause some nodes to not join or free edges to be mistakenly calculated. Conversely, setting the value too high may cause inadvertent nodal joining. The seawater density and gravity values can be changed as desired.

Environment - here the user can define the model's environment. For example, the user can set the water density and gravitation acceleration in the unit system defined. It is important to understand that these parameters affect all aspects of the analysis so caution should be taken if changing these values.

Hull Girder Stations - The Hull Girder Stations allows the user to change the default values of the station spacing and 0 location for longitudinal and transverse hull plots.

The station spacing is defined for plotting the summed vertical loads (via the Hull menu). The word *DEFAULT* sets the spacing equal to the section spacing of module 1, substructure 1. If another value is desired enter it here in the current length units. Station spacing is merely a convenient interval along the length (or athwartship) of the structure at which the program reports the total upward, downward and net vertical and horizontal loads that act on the structure within each (structure X axis or Z axis) interval for each load case.

The stations defined in "Station Spacing" are spaced along the structure X or Z axis, and their numbering starts at zero and increases in the positive X direction (or Z direction in the case of *transverse*).

- Default. Station 0 is located at Xmin, the lowest (smallest positive or largest negative) value of X (or Z) within the structure.
- (User input value). Entering a value, in the current length units, specifying an overriding value of X (or Z) for station 0.

NOTE: If the location of station 0 is such that it is within the structure, then all of the loads that occur from Xmin to X0 will be allocated to station 0.

Safety Factors - Safety factors for panel, girder and frame elements can be set by clicking on the associated button.

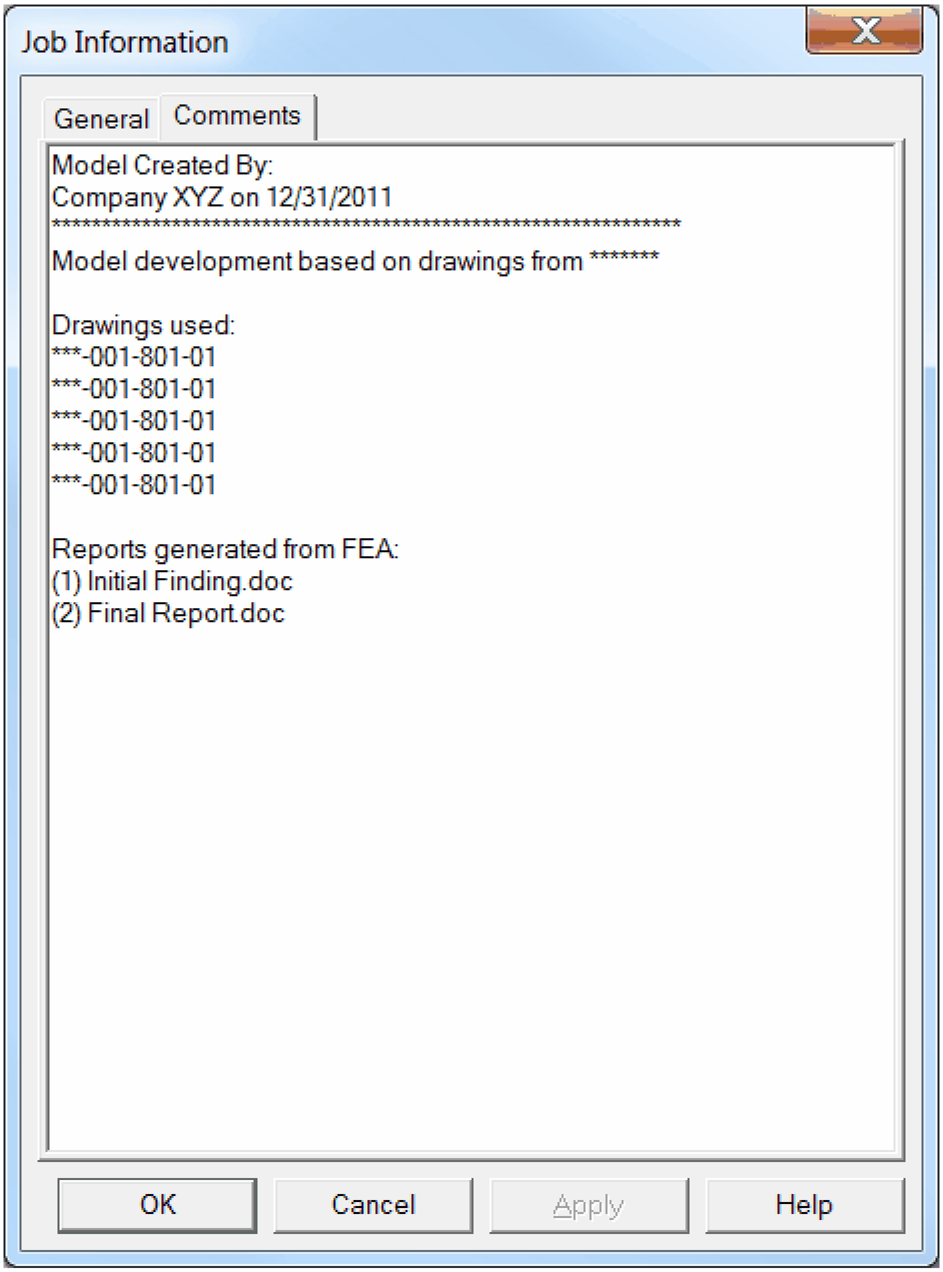
The *Criteria* drop-down has three options: *Default*, *Explicit*, and *Forensic*.

- <u>*Default*</u> selects that the two default values of safety factor - one for *Collapse* and one for *Serviceability* that are selected are considered appropriate for all limit states of each type, and there is no need for any explicit definitions of individual values. Therefore, for this option, the individual panel, girder, and frame elements buttons are grayed-out and not accessible.
- <u>*Explicit*</u> allows the user to specify an individual safety factor for each limit state. The individual limit states are accessed via the Panel…, Girder…, and Frame… buttons which are active for this criteria only. The user is cautioned to read Structural Evaluation and

© 2014 Optimum Structural Design, Inc.

Partial Safety Factors prior to using the Explicit criteria.

- *Forensic* sets all of the safety factors to 1.0, i.e., the purpose of this job is to determine whether any structural failures would actually occur for the loads that are specified in this job.



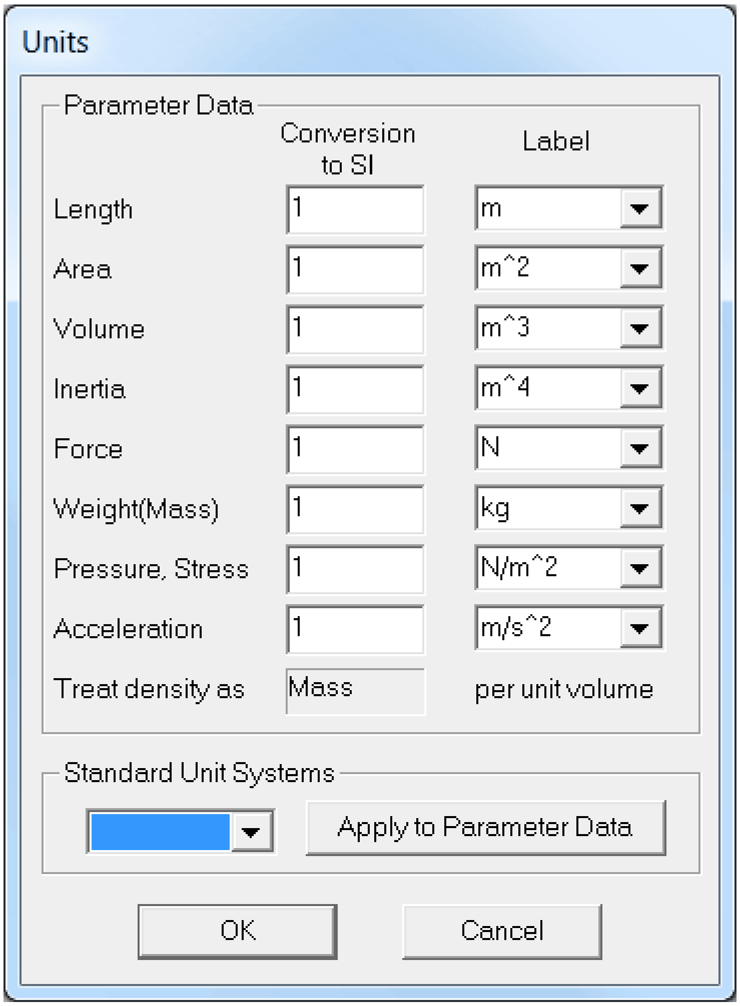
The Comments tab allows the user to attach comments to the model.

© 2014 Optimum Structural Design, Inc.

5.3 Defining Units

Toolbar	N/A
Menu	**File > Units...**

The Units dialog box can be accessed from the **File > Units...** menu.



Individual parameters can be changed using the Label drop down boxes or a Standard Unit System can be chosen from the drop down box. The *Set Parameter Data* box must be clicked for the units to change.

The seven Standard Unit Systems available are:

- SI (N, m) - Newtons and meters

© 2014 Optimum Structural Design, Inc.

- SI (N, mm) - Newtons and millimeters
- SI (MN, m) - MegaNewtons and meters
- fps - Feet, pound-force and seconds
- mks - Meters, kilogram-force and seconds
- cks - Centimeters, kilogram-force and seconds
- ips - Inches, pound-force and seconds

THE USER CAN SWITCH BETWEEN UNITS AT ANY TIME. The model data is stored in SI units (N,m) and is converted to the current selected units. Regardless of what units are chosen, the underlying structural model does not physically change. All that changes is how dimensional values are displayed to the user throughout the interface. For example, if a structural model is 100 meters long and the user changes the length units to feet, the model is still 100 meters long but it would be reported to the user as 328.08 feet long. This allows a user to be switch to another system, say IPS units, enter data that is best address in the temporary system, and then switch back to the analysis's standard units. This is particularly helpful when building a mixed system model.

5.4 Defining Materials & Properties

Quick Reference:

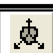
[Defining Materials](#)

[Creating a Plate Property](#)

[Creating a Beam Property](#)

[Creating a Rod Property](#)

[Creating a Spring Property](#)

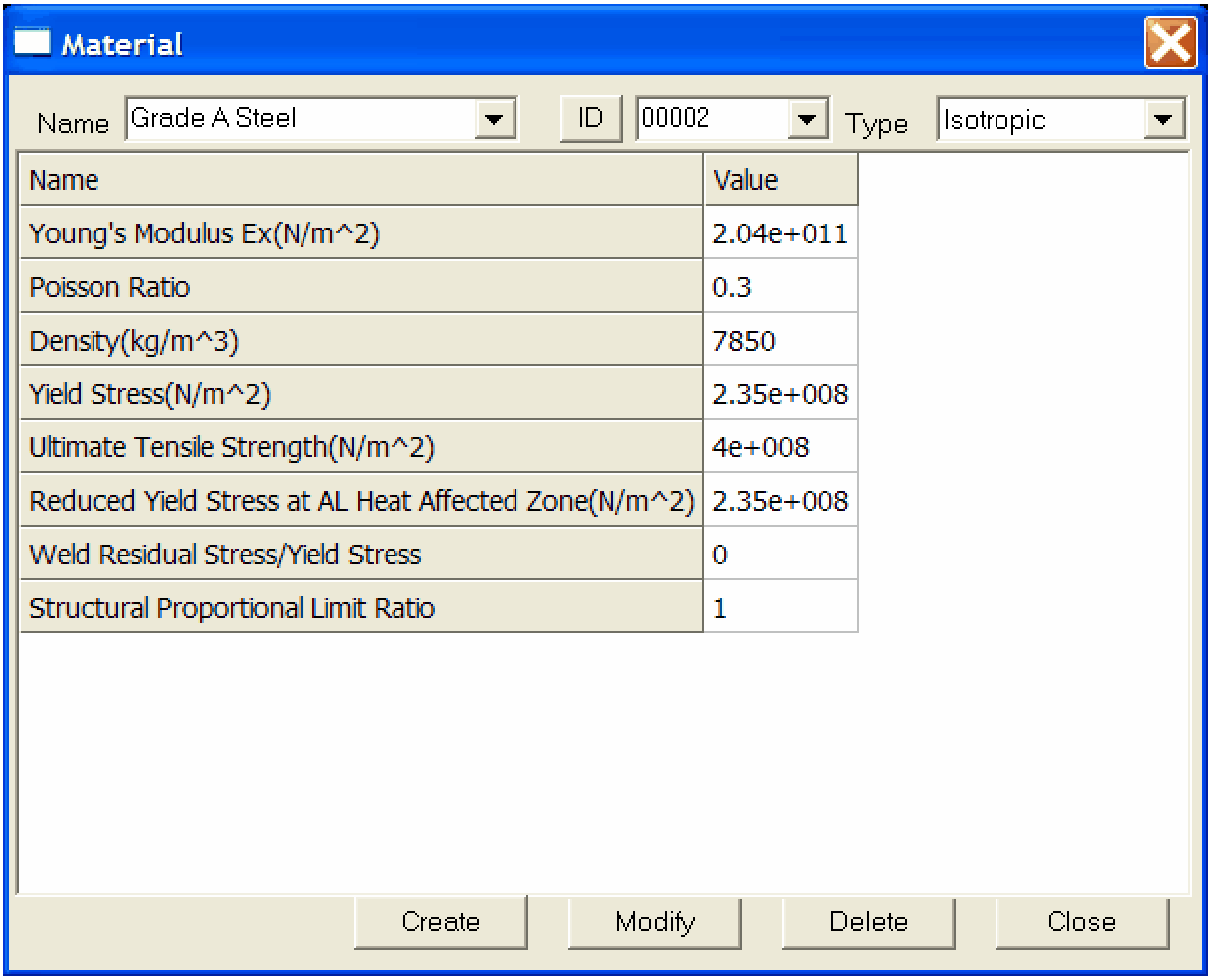
Toolbar	
Menu	**Model > Materials...**

The materials defined in the dialog box are global and are used as reference for element properties and therefore should be defined prior to creating an element property.

This tutorial shows the procedure for creating a new material.

© 2014 Optimum Structural Design, Inc.

1. Begin by opening the Materials dialog box from the **Model > Materials...** menu, or from the toolbar.



2. Click the ID button to get the next unique ID for the new material.

3. Type in a descriptive Name for the material and select the type: Isotropic, Orthotropic, or Compound.

4. Fill in the material parameter values in the units specified.

5. Click the Create button.

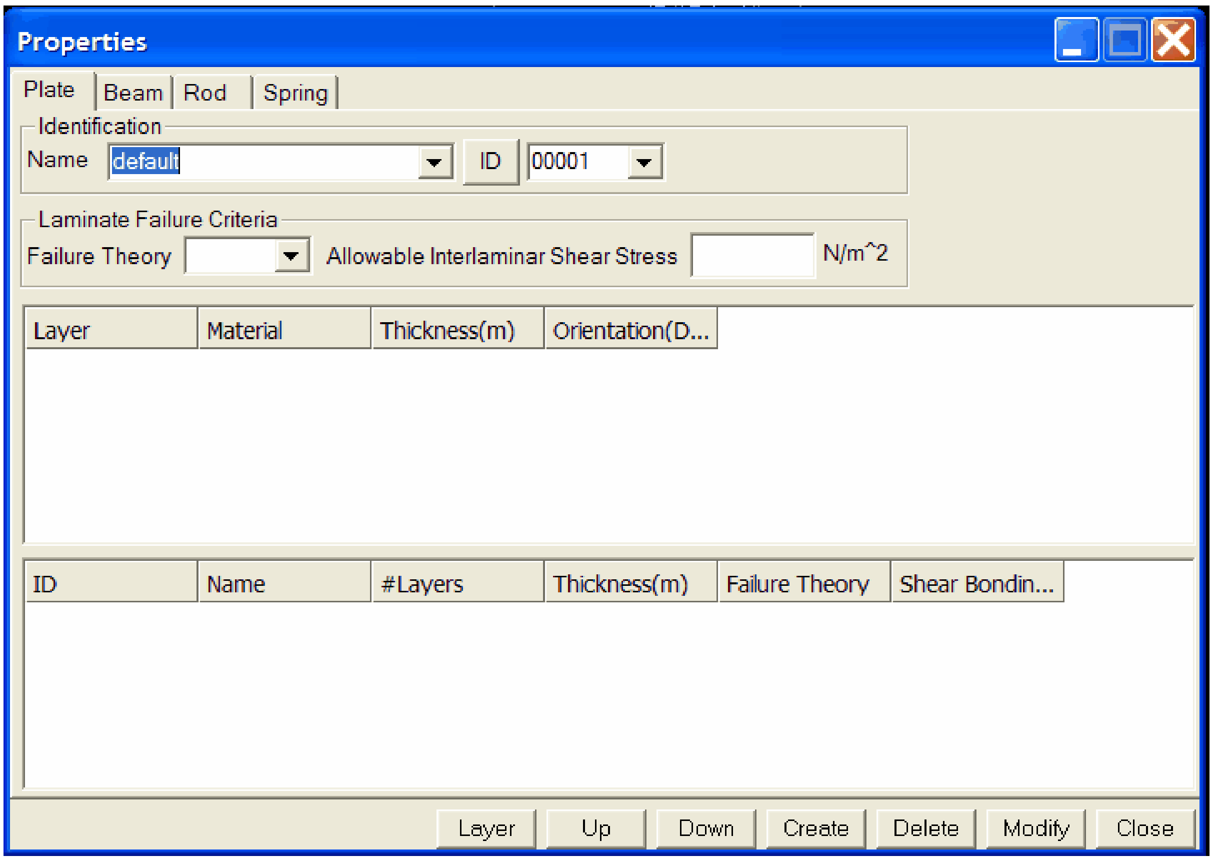
This procedure can be repeated to create additional materials. It can be helpful to create all the anticipated materials prior to creating the model.

Toolbar	

Menu	Model > Properties >

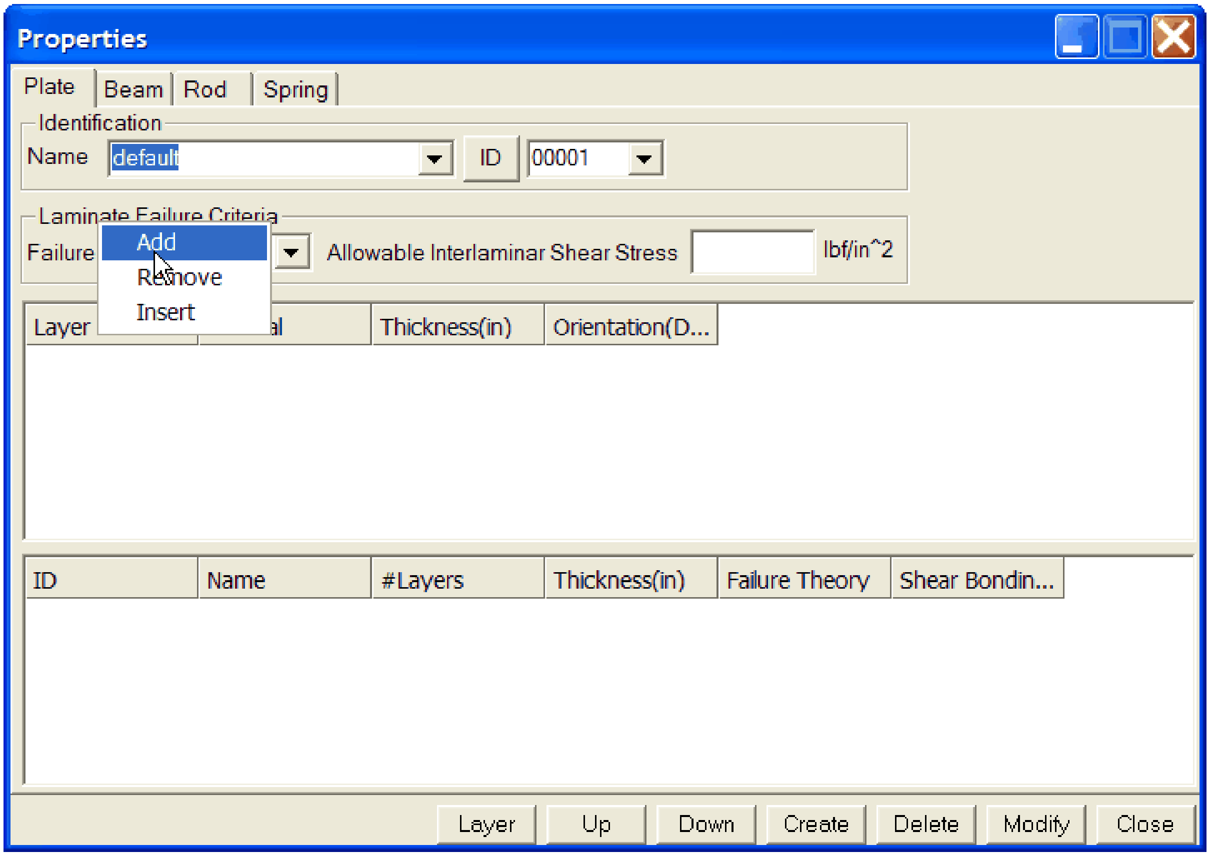
Once a material(s) is defined, the Properties dialog box is used to create Plate, Beam, Rod, and Spring element properties.

Begin by opening the Properties dialog box from the **Model > Properties>** menu, or from the toolbar.



This tutorial shows the procedure for creating a new plate element property.

1. Click the Plate tab.
2. Click the ID button to assign a unique ID to the plate element property.
3. Give the plate element property a descriptive name.
4. Right-click in the first large white space of the dialog and click Add. This will add a layer to the new property.



5. Select the material for the layer from the drop-down menu. Define a thickness in the units shown.

6. Click Create.

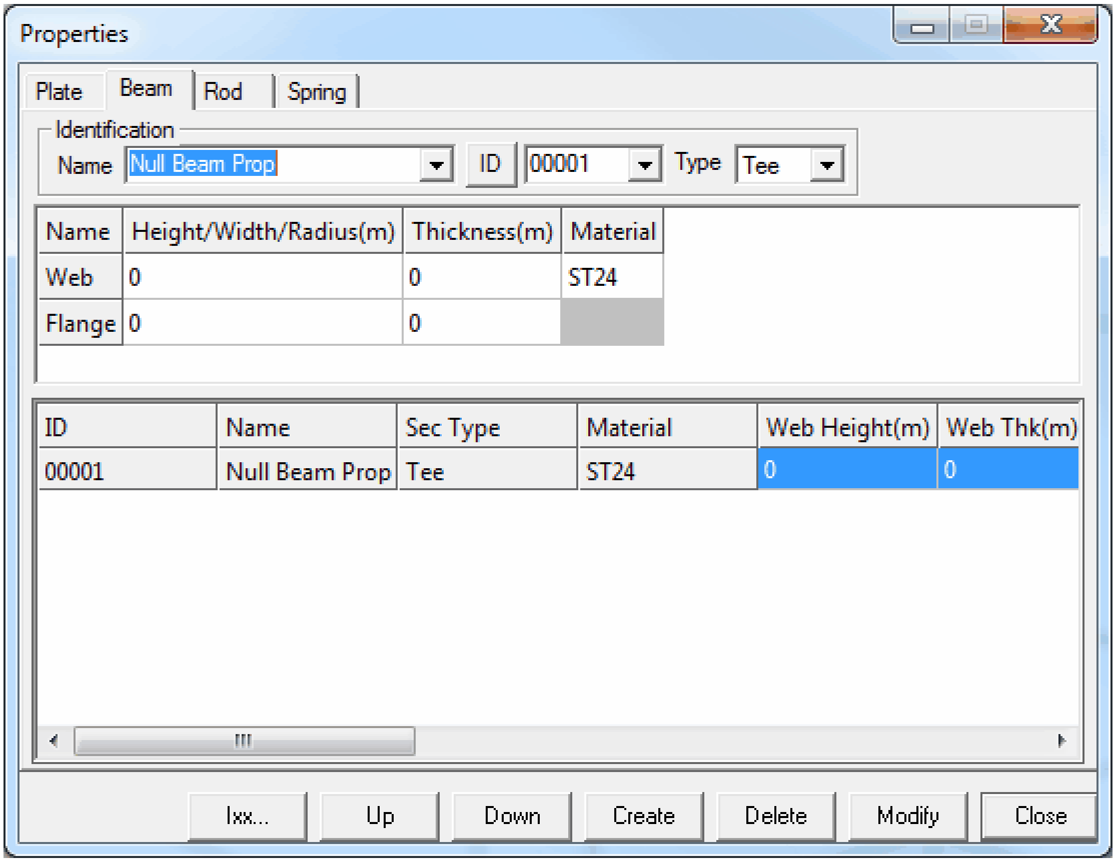
Composite Structures

MAESTRO can create composite structures for strake, quad and tri shell elements. In order to create a composite structure, follow the steps for a plate property definition but now additional layers can be added and fiber orientations set for each layer. Input the appropriate Laminate Failure Critera for the composite structure. Stresses can be recovered for each layer of a composite material. See the Recovering Composite Layer Stresses section for details.

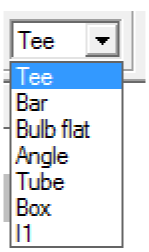
This procedure can be repeated to create additional plate element properties.

This tutorial shows the procedure for creating a new beam property.

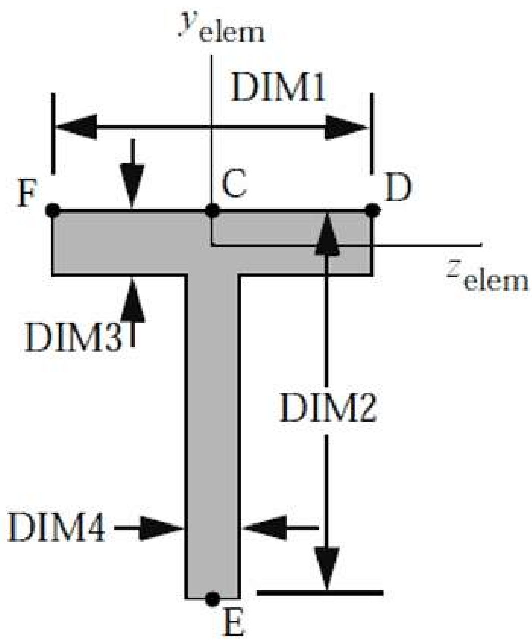
1. Click the Beam tab.



2. Click the ID button to assign a unique ID to the beam element property.
3. Give the beam element property a descriptive name.
4. Select the beam type from the drop-down menu.

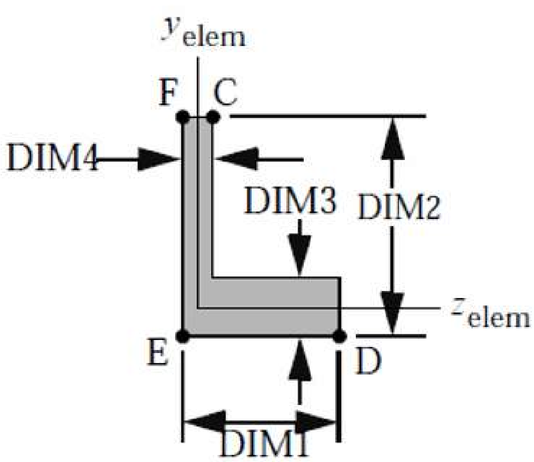


5. Define the Web and Flange (if applicable) Height/Width/Inner Radius(for Tube Beams) and thickness. The definitions for each type of beam's dimensions are shown below (Bf: flange width, Tf: flange thickness, Hw: web height, Tw: web thickness):



TYPE="T"

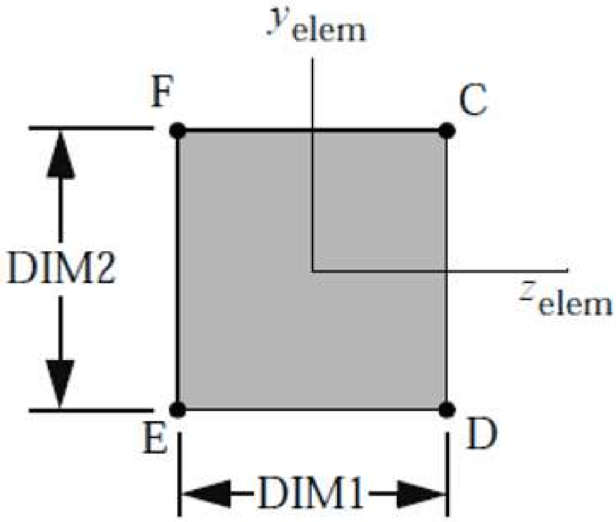
$$\begin{cases} b_f = dim1 \\ t_f = dim3 \\ t_w = dim4 \\ h_w = dim2 - dim3 \end{cases}$$



TYPE="L"

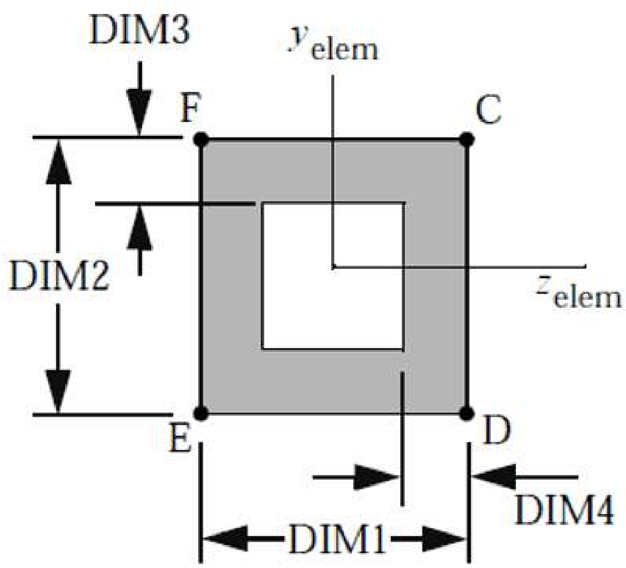
$$\begin{cases} b_f = dim1 \\ t_f = dim3 \\ t_w = dim4 \\ h_w = dim2 - dim3 \end{cases}$$

© 2014 Optimum Structural Design, Inc.



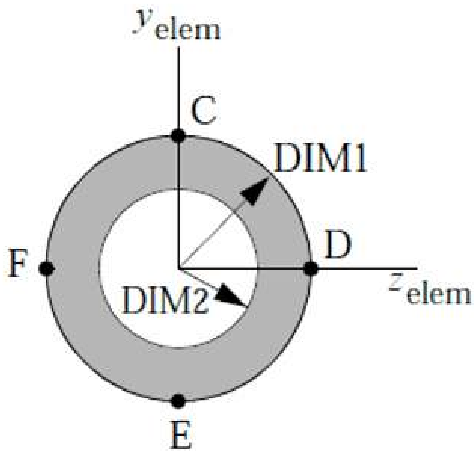
$$\begin{cases} b_f = 0 \\ t_f = 0 \\ t_w = dim1 \\ h_w = dim2 \end{cases}$$

TYPE="BAR"



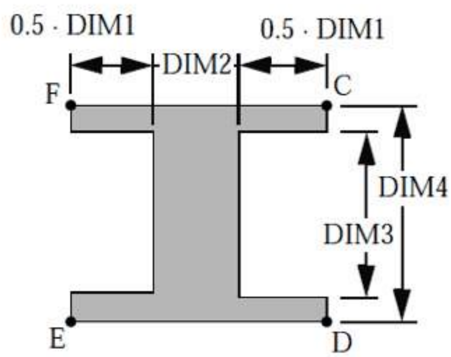
$$\begin{cases} b_f = dim1 \\ t_f = dim3 \\ t_w = dim4 \\ h_w = dim2 \end{cases}$$

TYPE="BOX"



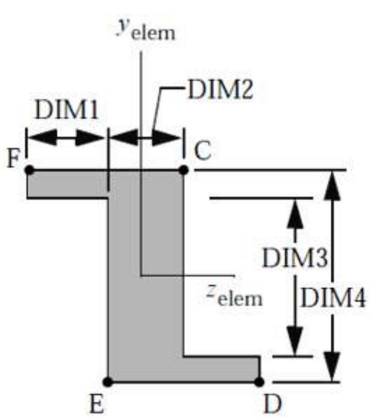
TYPE="TUBE"

$$\begin{cases} b_f = 0 \\ t_f = 0 \\ t_w = dim1 - dim2 \\ h_w = dim1 \end{cases}$$



TYPE="I1"

$$\begin{cases} b_f = dim1 + \mathrm{dim2} \\ t_f = (dim4 - dim3)/2 \\ t_w = dim2 \\ h_w = dim3 \end{cases}$$



TYPE="Z"

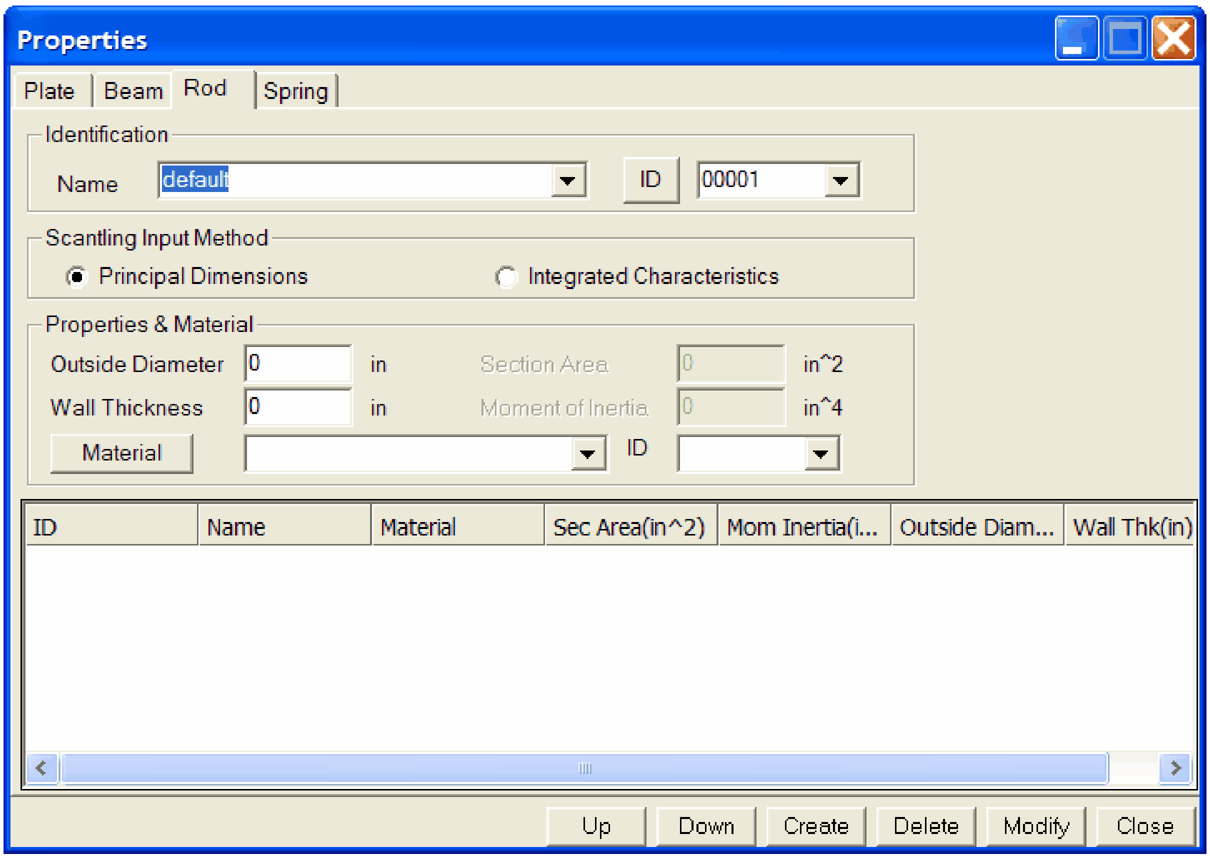
6. Select the material.

7. Click Create.

Note, the beam properties list can become expansive and it may be helpful to sort the beam properties in the dialog box and this order will be maintained when selecting a frame, girder or beam property.

This tutorial shows the procedure for creating a new rod element property.

1. Click the Rod tab.



2. Click the ID button to assign a unique ID to the rod element property.

3. Give the rod element property a descriptive name.

4. Define the rod by its Principal Dimensions (Outside Diameter and Wall Thickness) or its Integrated Characteristics (Section Area and Moment of Inertia).

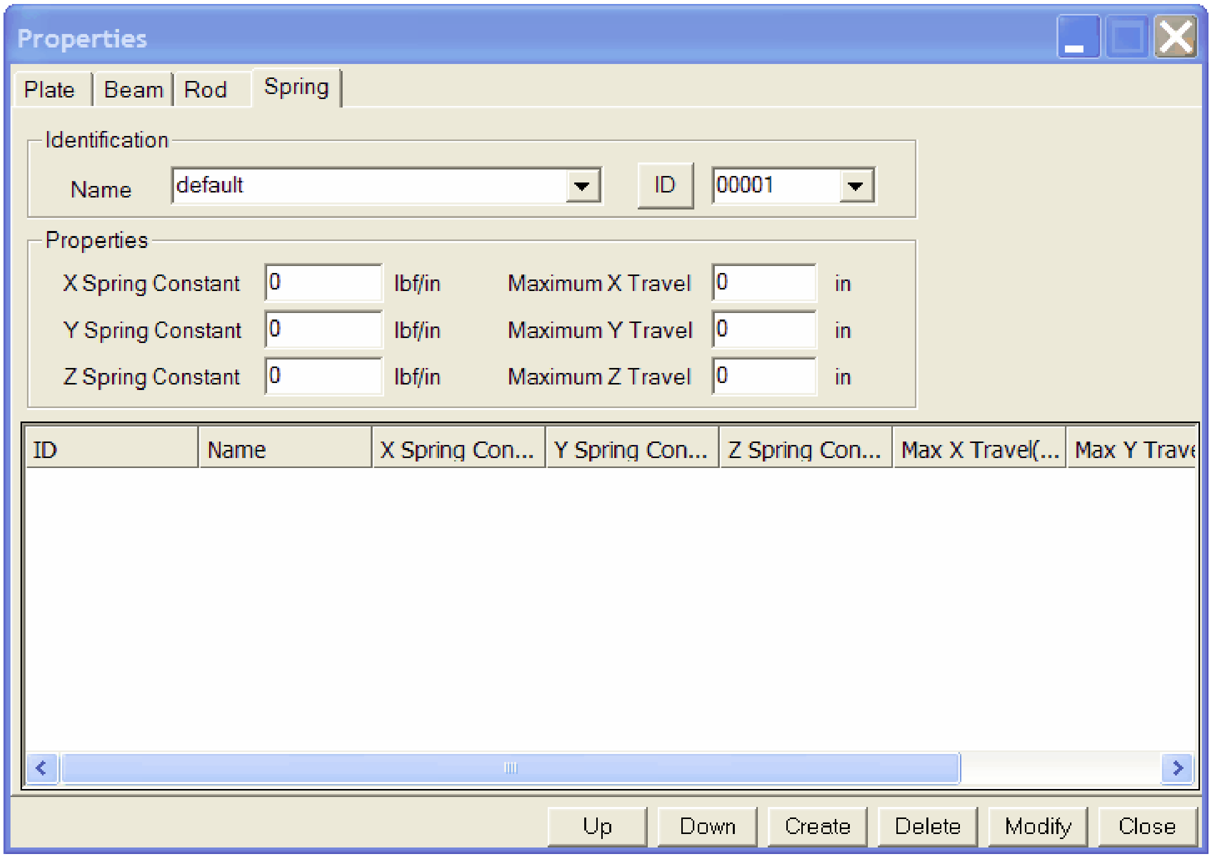
5. Select the material for the rod from the drop down menu.

6. Click Create.

This tutorial shows the procedure for creating a new spring element property.

© 2014 Optimum Structural Design, Inc.

1. Click the Spring tab.



2. Click the ID button to assign a unique ID to the spring element property.

3. Give the spring element property a descriptive name.

4. The spring properties are defined by the spring constant in the X,Y, and Z directions as well as the permissible travel permitted in each direction.

This travel distance is assumed to be the restriction in stretching and compressing the spring.

5. Click Create.

© 2014 Optimum Structural Design, Inc.

5.5 Defining Parts

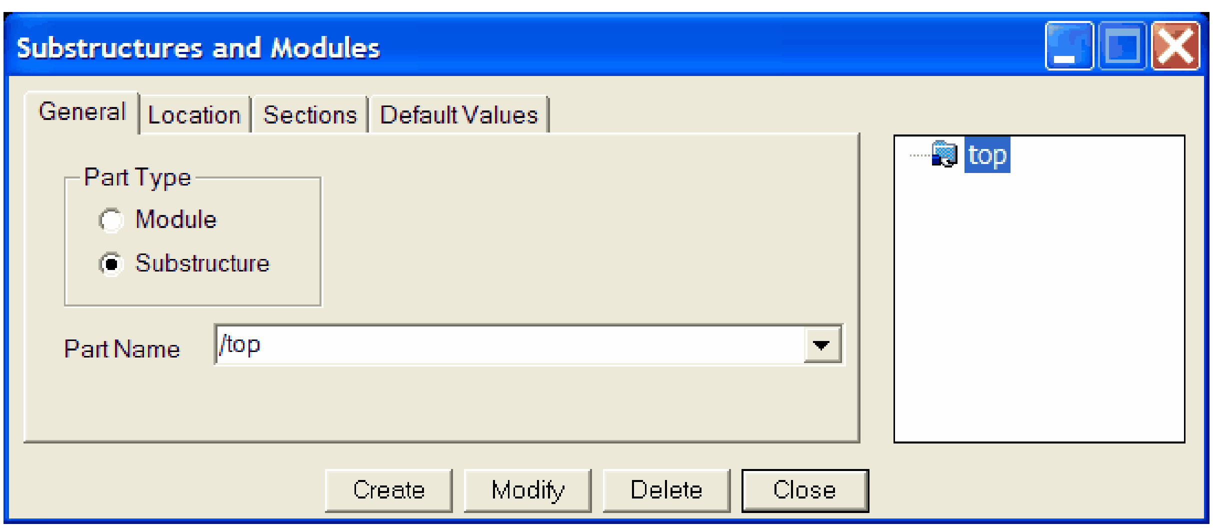
Toolbar	
Menu	**Model > Parts > Create/Modify**

A MAESTRO model is made up of parts, or more specifically substructures and modules. In the model hierarchy, modules make up substructures, which make up the full model. Before you begin modeling, it is a good idea to first review your structural drawings and plan out your Substructure and Module breakdown.

This dialog contains four tabs labeled General, Location, Sections, and Default Values. In addition the window on the right side of the dialog displays the current parts tree structure. The parts tree is interactive and allows the user, via right-button clicking, to set the current part, the view part, and rename parts. The process of creating a new part involves entering the required data in the fields of each tabbed page and clicking the Create button. Modifying an existing part involves selecting the desired part from the *Part Name* drop-down list in the General tab or from the parts tree at the right, changing the desired values and clicking the Modify button. Note that whenever a part is modified from any of the Substructures and Modules tabs that part becomes the Viewed Part. Each of the tabs in the Substructures and Modules dialog are discussed more fully in the following sections.

This tutorial shows the procedure for creating a substructure and an accompanying module.

1. Begin by opening the Substructures and Modules dialog box from the **Model > Parts > Create/Modify** menu, or from the toolbar.



2. Select *top* from the local parts tree; the radio button should be selected for Substructure

© 2014 Optimum Structural Design, Inc.

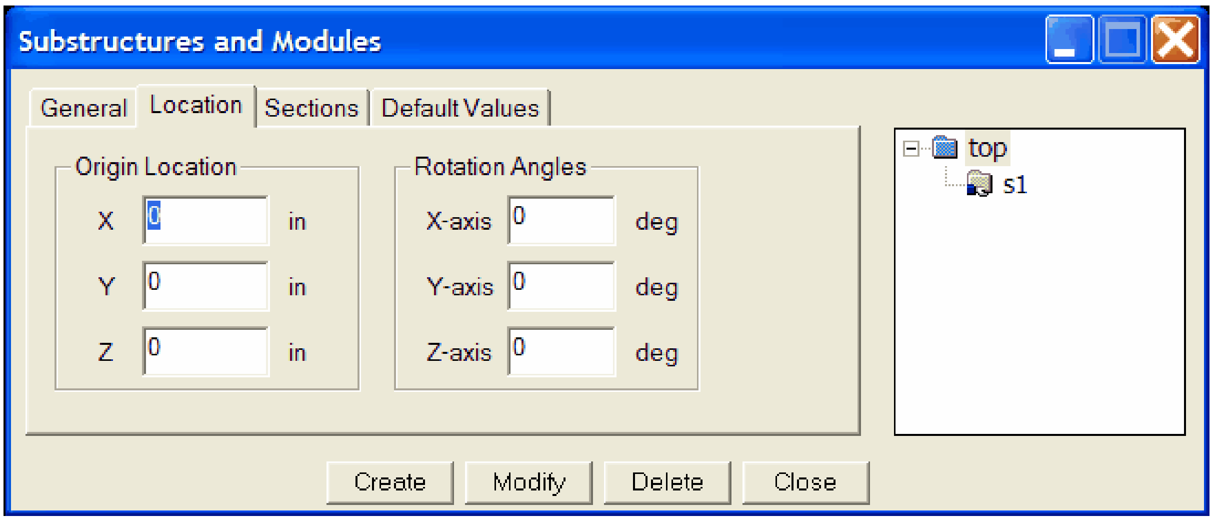
under Part Type.

3. In the Part Name box, type a forward slash after *top* and give a descriptive name for the substructure; click Create.

Note: if you click the "+" next to *top*, the substructure just created will be shown along with any other substructures created.

4. In the Location tab, set the Origin Location and Rotation Angles for the substructure and click Modify.

Note: the origin location of the substructures is relative to the top origin location, which by default is (0,0,0).

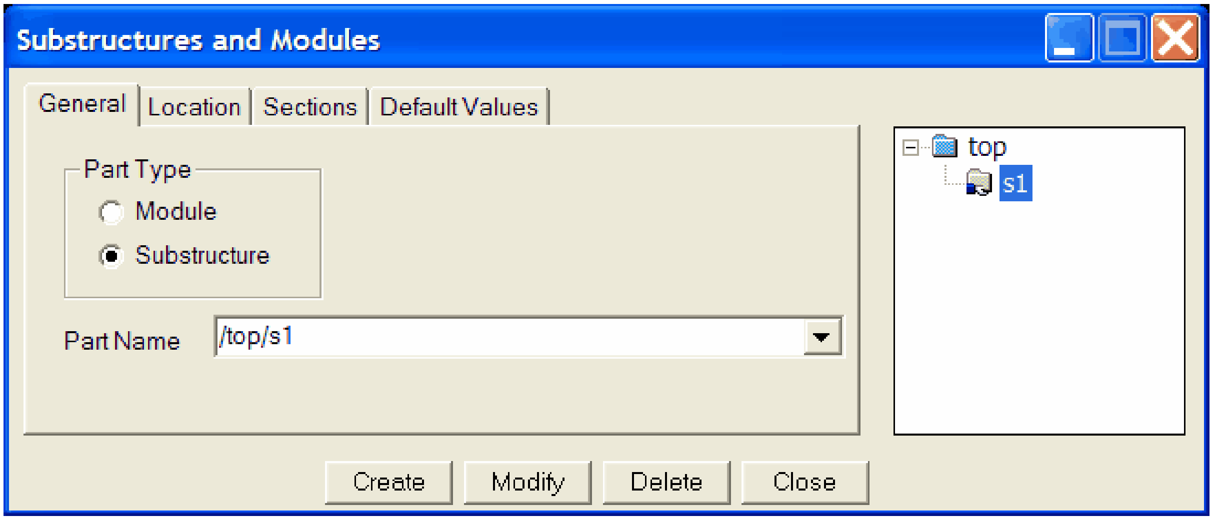


5. The Sections and Default Values tabs only apply to modules and thus are ignored for substructures.

Additional substructures can be created in this same manner. Now we are ready to create a module within our newly created substructure.

6. Click to the General tab and select the substructure from the local parts tree on the right.

© 2014 Optimum Structural Design, Inc.



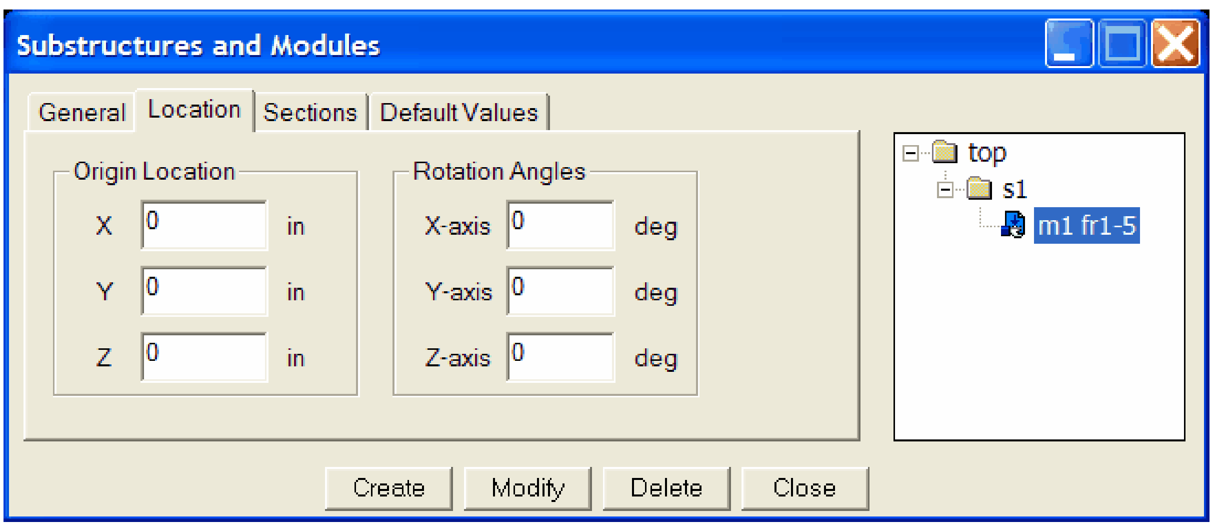
Note: the Part Name box now lists the substructure name after *top*.

7. After the substructure name, type a forward slash and a descriptive name for the module.

8. Click the radio button for Module and then click Create.

Note: clicking the "+" next to the substructure name will now display the modules that make up that substructure.

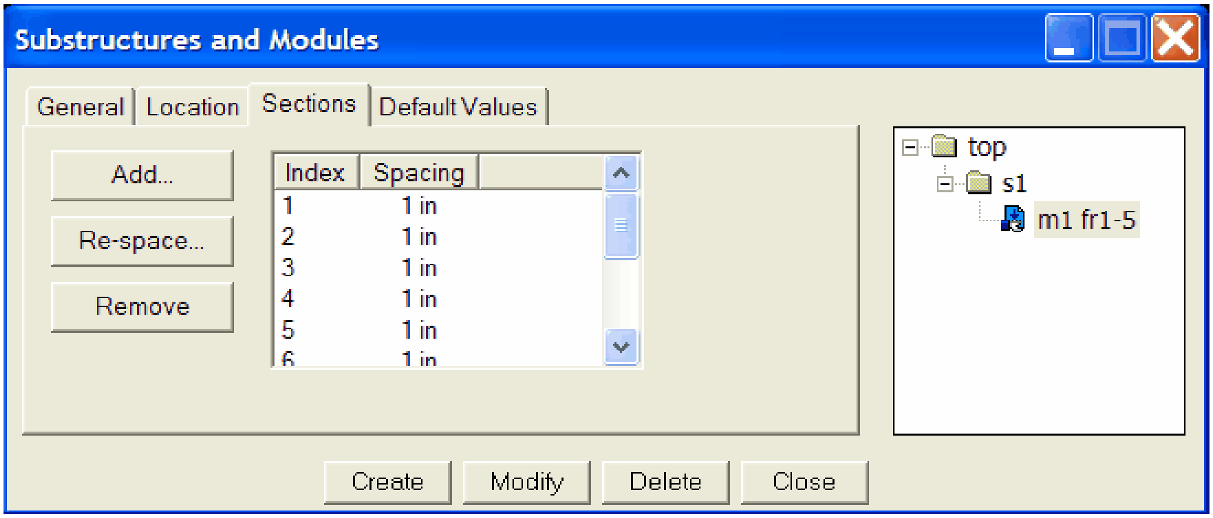
9. After making sure the new module is highlighted in the local parts tree and the name shows up in the Part Name box, select the Location tab. The Part Name box and Parts tree specify which part is currently "active" and can be modified.



10. Define the Origin Location and Rotation Angles for the module and click Modify.

Note: these values are applied within the current part's reference system. For example if your substructure X origin is set at 120 inches, then setting the X origin of the module to 0 inches is equivalent to 120 inches in the global reference system.

11. Select the Sections tab. The default sections is 10, with 1 unit spacing.

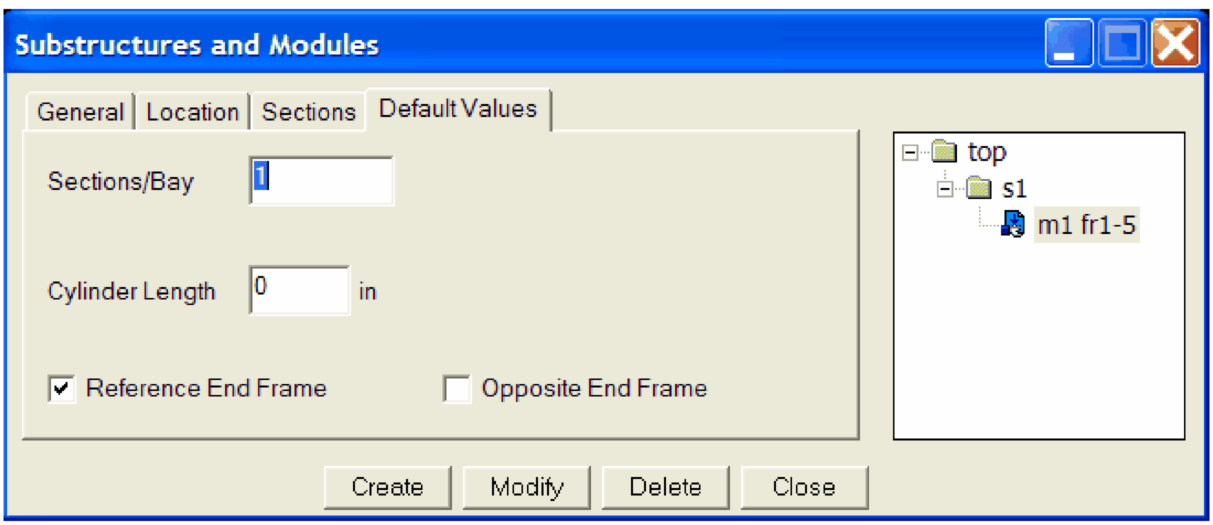


12. These sections can be re-spaced, or deleted and new sections can be created. To do this, highligh the sections and click Remove.

13. Click Add... and define the number of sections and their spacing. Multiple sections can be added with different spacing.

14. Click the Modify button.

12. Select the Default Values tab.



13. The Sections/Bay and Cylinder Length can be defined here. The sections per bay defines the number of strake sections per structural bay. This allows for the transverse frames to be placed every second section, or third section or any other regular spacing. The Cylinder Length parameter is for a specific application to cylindrical structures. This parameter defines the length between bulkheads (circular diaphragms) in the cylinder. This parameter is ONLY applicable to strakes of type "cylinder."

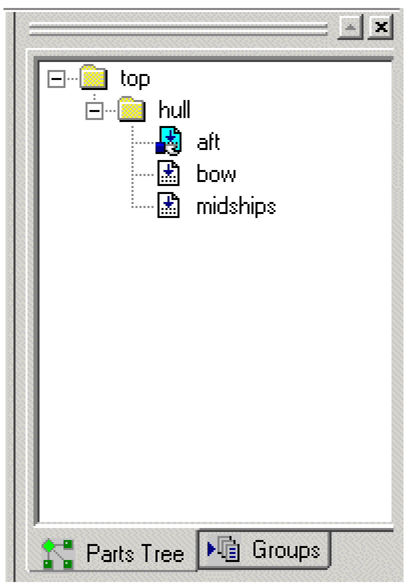
15. The Reference End and Opposite End Frame options defines whether a transverse frame will be located at either end of the module's strakes. This check box acts as a toggle, turning Frames On/Off at the extreme ends of the modules.

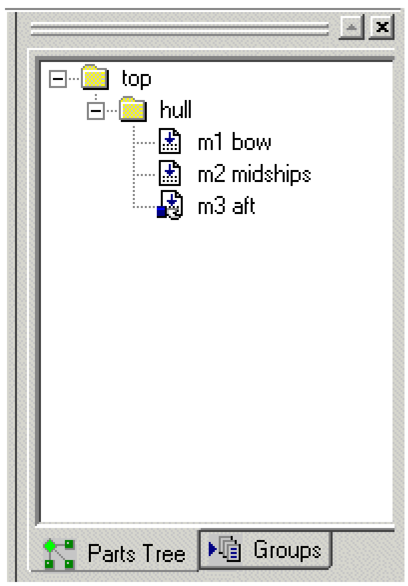
14. Click Modify.

Additional modules can be created in the same manner for the current substructure or other created substructures.

NOTE:

MAESTRO's parts tree, by default, orders the substructures and their respective modules alphabetically. Thus a model with a substructure called "hull" and modules called "bow", "midships", and "aft" would be displayed as shown on the left. To maintain a specific order to the substructures and/or modules it is recommended that the name include an abbreviated numbering system (as shown below).



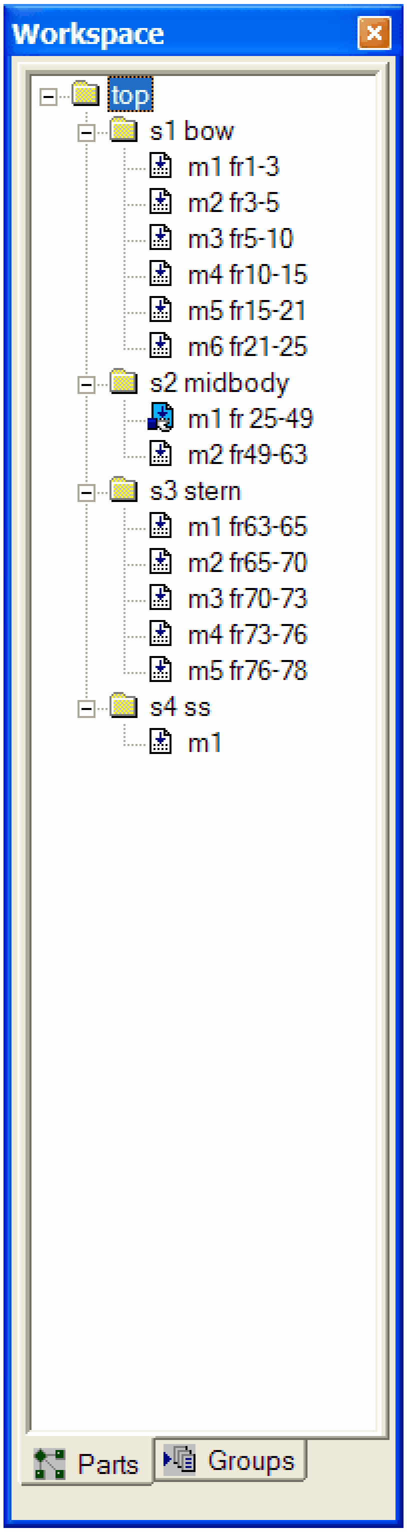


5.6 Parts Tree

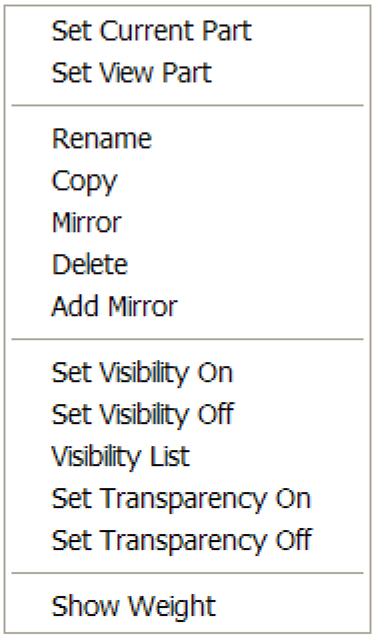
The parts tree pane displays the hierarchy of substructures and modules in the model, in a folder format directory. The parts tree can be manipulated just as a directory tree can, but the user must remember that modules must be contained in substructures.

© 2014 Optimum Structural Design, Inc.

The tree can be shown or hidden using the parts tree icon .



The number and organization of the parts may be adjusted either through the Parts command on the main menu, or directly in the parts tree. Clicking the right mouse button launches a popup menu offering the user several options.



Set Current Part

This will set the highlighted module or substructure as the current part.

Set View Part

This will set the highlighted module or substructure as the current view. If a substructure is selected, all the modules in that substructure will be visible. To view the entire model, highlight and right-click *top* and select *Set View Part*.

Set Current & View Part

This will set the highlighted module or substructure as the current part and the current view.

Rename

This will allow the highlighted module or substructure to be renamed within the parts tree.

Copy

This will create a copy of the highlighted module or substructure.

Mirror

This will launch the mirror dialog and allow the user to *Mirror* or *Add Mirror* for the highlighted part.

Delete

This will delete the highlighted module or substructure. If a substructure is deleted, all of its modules will be deleted as well.

Add Mirror

This will launch the mirror dialog and allow the user to *Mirror* or *Add Mirror* for the highlighted part.

Set Visibility On

This will toggle the visibility of the highlighted part to *On*.

Set Visibility Off

This will toggle the visibility of the highlighted part to *Off*. If a substructure's visibility is set to off, all of the modules within that substructure will have their visibility set to off.

Visibility List

This will launch the Visibility List dialog which looks the same as the parts tree, but allows the user to toggle the visibility on/off with one click for each module or substructure.

Set Transparency On

This will set the highlighted module or substructure as transparent.

Set Transparency Off

This will set the highlighted module or substructure as not transparent.

Show Weight

This will show a dialog providing the weight of the highlighted module of substructure. If a

© 2014 Optimum Structural Design, Inc.

substructure is selected, the weight will be for all of the modules within that substructure combined.

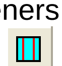
5.7 Defining Stiffener Layouts

Toolbar	
Menu	**Model > Stiffener Layout...**

Stiffener layouts allow the user to define the number of stiffeners or breadth between stiffeners for a panel element. Stiffener properties are defined in the beam properties dialog box.

Stiffeners are defined as beam elements, but they are not actual finite elements in a coarse mesh model. Instead, they are treated as additional stiffness in the defined direction for the panel element they are defined on, thus converting the material to orthotropic. In a coarse mesh model, all stiffeners are treated as "internal" regardless of their defined location. The number of stiffeners, whether defined as internal or edge, is the key to how the mechanical properties of the panel element are changed. However, the location is relevant when creating a fine mesh model from the coarse mesh model. At this point, an actual beam element will be created representing the properties of the stiffener at the defined location. As a result of MAESTRO's treatment of stiffeners, there are a couple caveats when defining stiffened panels:

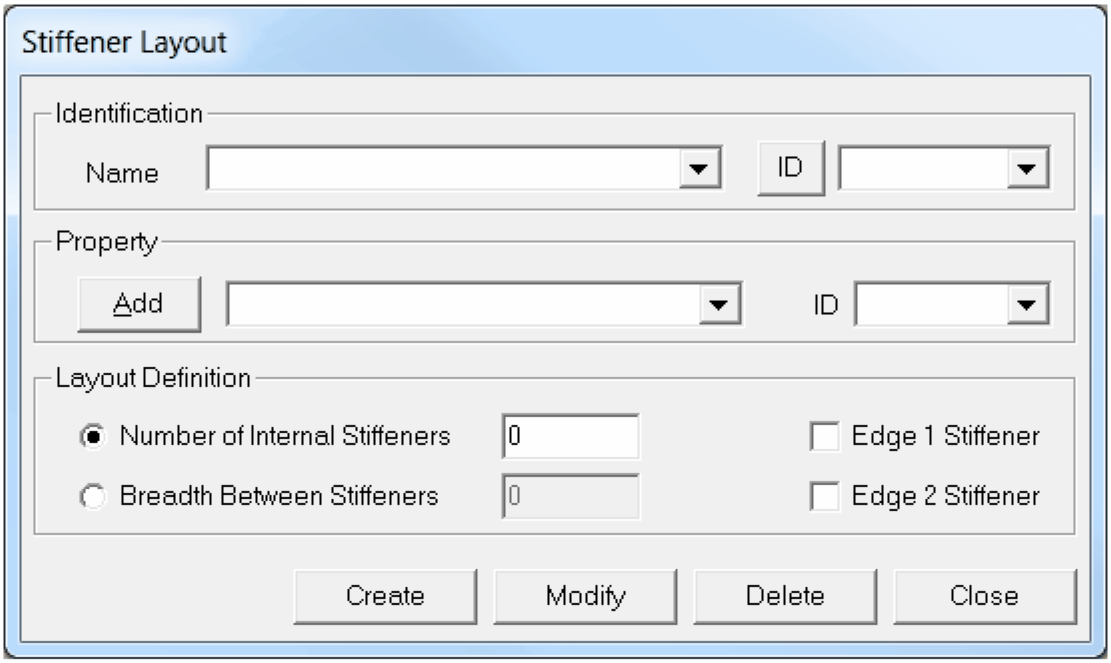
- An edge stiffener defined in a coarse mesh model on the centerline will not be flagged as a centerline element. Instead it will be treated as if there are two stiffeners with identical properties, thus affecting the total stiffness, cross sectional area, etc.
- Similar to the point above, stiffeners may be modeled as "overlapped" and MAESTRO's integrity check will not recognize this. For example, two consecutive strakes may each have a stiffener on their shared edge or a transverse stiffener layout with edge 1 and edge 2 stiffeners defined could be applied to a strake, thus creating overlapped stiffeners. Again this will be treated as if there are two stiffeners with identical properties, thus affecting the total stiffness, cross sectional area, etc.

In the coarse mesh model, stiffeners are shown graphically with a red line representing their location and orientation. Stiffeners can be toggled on or off in the View Options dialog or with the Stiffeners On/Off icon .

This tutorial shows the procedure for defining stiffener layouts that can be applied to a strake panel or an additional quad element.

© 2014 Optimum Structural Design, Inc.

1. Begin by opening the Stiffener Layout dialog box from the **Model > Stiffener Layout...** menu, or from the toolbar. The dialog can also be accessed from within the Strakes dialog box or the Fine Elements dialog on the Quad tab.



2. Click ID to create a new layout.

3. Give the layout a descriptive name.

4. Select the property of the stiffener from the drop-down box. If the property is not yet defined, click the *Property* box and the Beam Property dialog will open.

5. MAESTRO allows the user to define either the number of internal stiffeners or the Breadth between stiffeners. Select the radio button for the option you desire and fill in the number of distance.

6. MAESTRO also allows for an Edge 1 or Edge 2 stiffener. The Edge 1 and 2 corresponds to a longitudinal stiffener layout applied to a strake. In this case, Edge 1 is the edge defined by EndPoint 1 and Edge 2 is defined by EndPoint 2.

7. Click Create to save the stiffener layout. This layout will now be listed in the drop-down box in the Strakes and Quad dialog boxes.

© 2014 Optimum Structural Design, Inc.

5.8 Importing Geometry

Toolbar	N/A
Menu	**File > Import >**

Curves can be imported into MAESTRO to serve as construction geometry to assist in model generation. The idea is to import curves at strategic locations, such as the Reference and Opposite ends of the Parts definition. This will allow the user to "snap" to these locations of interest using MAESTRO's construction geometry. See the Construction Geometry section for more information on MAESTRO's construction geometry.

Importing a . PLY File

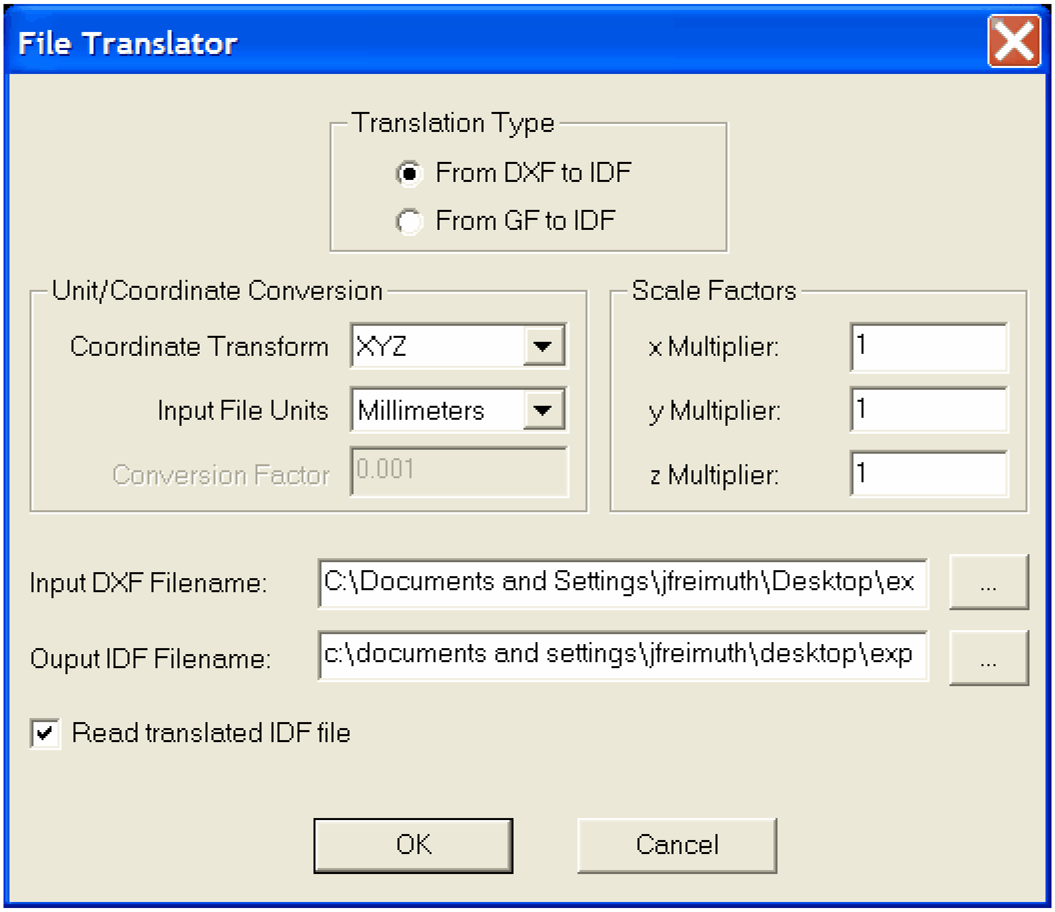
Defining Surface Normal and Wettable Elements

This tutorial shows the procedure for importing geometry as an IDF, DXF, or GF format.

1. Make sure that *top* is set to the current part.
2. Select **File > Import >** from the menu. Choose the file type for the input.
3. Select your geometry file from it's saved location and click Open.

Note: MAESTRO reads in IDF format, but can also convert and read in DXF and GF files using the File Translator dialog box.

© 2014 Optimum Structural Design, Inc.



4. Select the Translation Type and any unit or coordinate conversions. MAESTRO can also scale the model in any of the three Cartesian coordinate directions.

5. Select the Read translated IDF file option if you would like MAESTRO to automatically load the converted file. Click OK.

6. A prompt will open asking to insert marks? Selecting yes will place a construction marker at any curve end points or intersections.

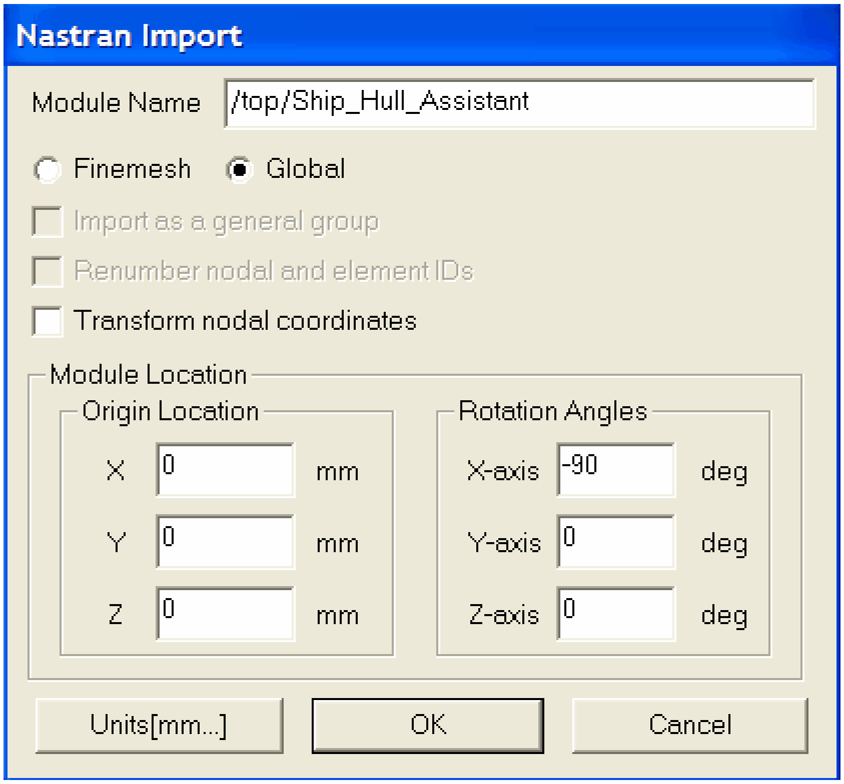
Importing a .PLY File

This tutorial shows the procedure for importing geometry as a Polygon Mesh File Format (*.ply). The polygon mesh file supports individual quad and triangle elements as well as endpoint and strake definitions. Strakes will be given default properties but can be modified individually or using groups. A sample (PlyImport.txt) file can be found in the **Models and Samples > Import** directory.

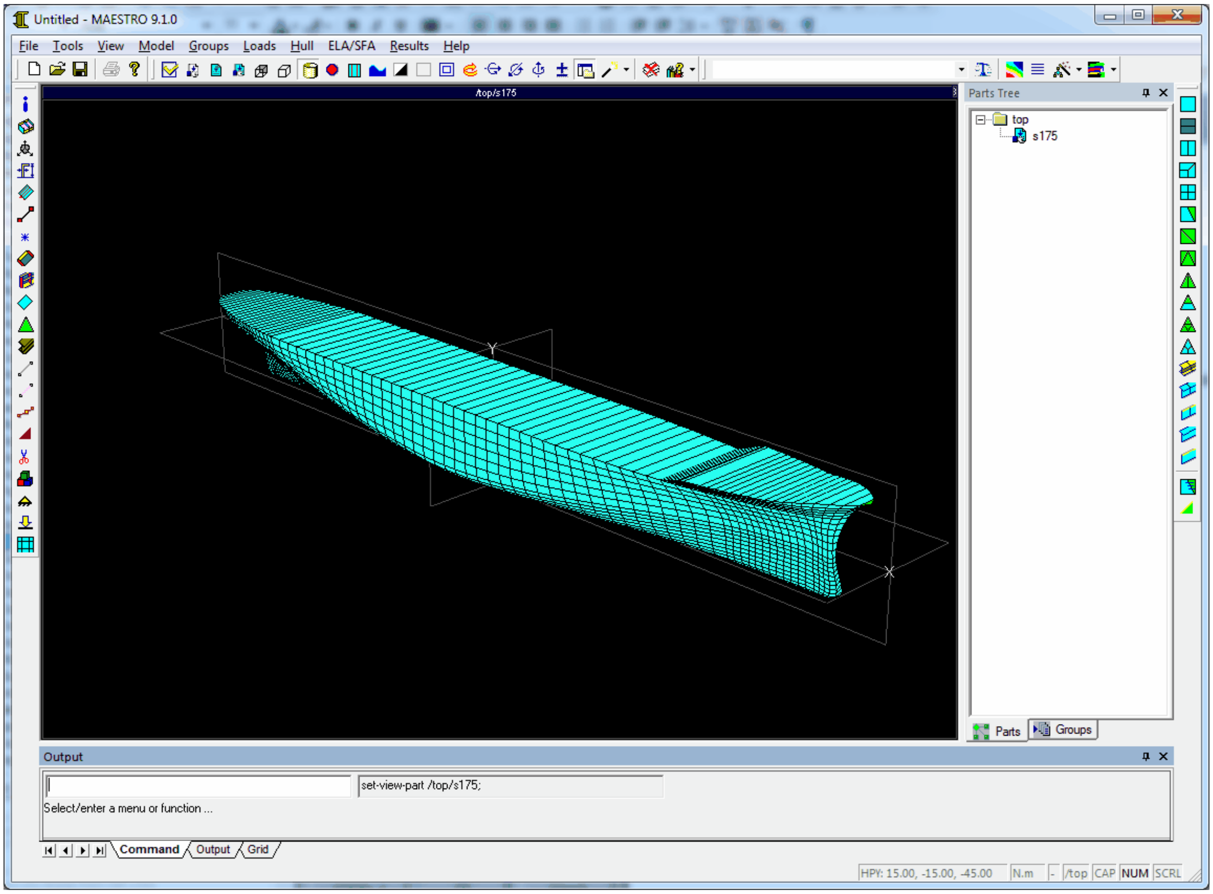
1. Select **File > Import > Ply-Polygon File Format (*.ply)** from the menu.

2. Select the geometry file from it's saved location and click Open.

© 2014 Optimum Structural Design, Inc.



3. Select the options from the dialog and click OK. These options are similar to importing a Nastran model. MAESTRO does not detect the previous coordinate orientation so we must enter the correct rotations about the x, y, and z axes to transform the model from the Rhino coordinate system to the MAESTRO coordinate system. The "Transform nodal coordinates" option not only transforms the geometry but also the old coordinate system, so that it is possible to work from MAESTRO coordinates. If this item is not checked and the user needs to enter coordinates, they will need to be entered in the old coordinate system.



The importing polygon mesh file will automatically create a new module of the mesh elements. All quad and triangle elements will be automatically assigned the same plate and material property. These can be changed individually or all at once by creating a group and changing the group's property.

This sample model (S175.ply) can be found in the **Models and Samples > Import** directory.

For guidelines on creating a mesh in Rhino, see the following section.

© 2014 Optimum Structural Design, Inc.

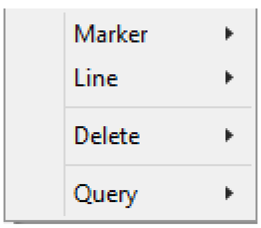
5.9 Creating Construction Geometry

Toolbar	**N/A**
Menu	**Tools > Construction Geometry >**
Mouse	**Hold Ctrl key & Right-Click**

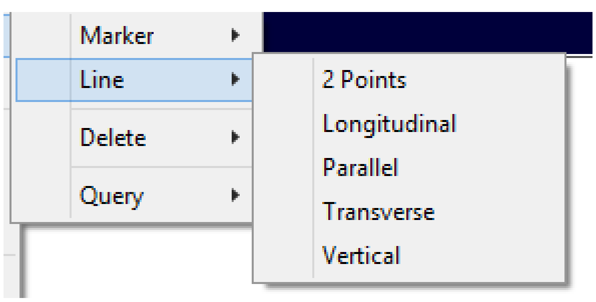
MAESTRO's construction geometry can be very helpful in defining EndPoints and additional nodes.

This tutorial shows the procedure for creating two construction lines and a construction marker at their intersection. A description of each type of construction geometry can be found below.

1. Open the construction geometry dialog box from the **Tools > Construction Geometry >** menu or by holding the **Ctrl key** and right-clicking in the modeling space.



2. Hold the mouse over CLine and the construction line options will fly out from the menu.



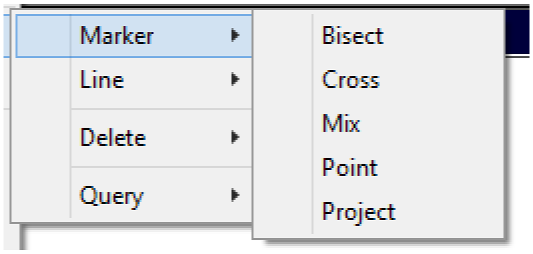
3. Select one of the options to create a construction line.

4. Follow the instructions in the command tab at the bottom of the screen to define the construction line.

5. Repeat this process to create an additional construction line that intersects the first one.

6. Access the construction geometry menu and hold the mouse over CMarker and the construction marker options will fly out from the menu.

© 2014 Optimum Structural Design, Inc.



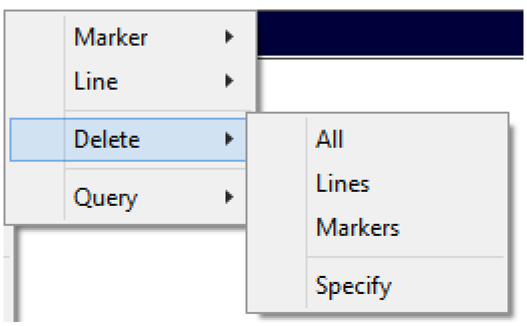
7. Select the Cross option.

8. Click on one of the construction lines near the intersection point. Then click on the second construction line.

A construction marker will appear at the intersection. This point can be used to "snap" EndPoints or additional nodes to.

The Repeat command can be used to create multiple construction lines or markers by selecting the construction geometry to create and then clicking on repeat in the menu.

Construction geometry can be deleted by holding the mouse over the Delete CGeom option and the delete options will fly out from the menu.



Construction Markers

Point: This command allows the user to create a construction marker at a specified location in space either interactively with the mouse or through the command line.

Bisect: This command allows the user to create a construction marker at a specified fraction along a particular line segment. To enter a specified fraction, delete the "{" in the command line and type the fraction in decimal form (i.e. 0.25).

Project: This command allows the user to create a construction marker at the projection of a specified point on an indicated line

Cross: This command allows the user to create a construction marker at the crossing of two indicated lines. If the two lines (or their extensions) do not physically intersect in space, the marker is placed at the closest point on the primary line (the first line selected) to the secondary line (the second line selected). The two lines are selected using the mouse.

Mix: This command is similar to bisect, but instead the user selects two points of a line segment.

Construction Lines

2 Points: This option will create a construction line in space between two selected points.

Parallel: This option will create a construction line in space passing through a specified point and parallel to the specified direction vector. The direction vector is often specified by using the mouse to select another line segment in the model parallel to the desired vector.

Longitudinal: This option allows the user to create a construction line in space through a specified point in the longitudinal (i.e. module’s local X-dir) direction.

Transverse: This option allows the user to create a construction line in space through a specified point in the transverse (i.e. module’s local Z-dir) direction.

Vertical: This option allows the user to create a construction line in space through a specified point in the vertical (i.e. module’s local Y-dir) direction.

Delete Construction Geometry

All: This option will delete all existing construction geometry.

CMarkers: This option will delete all existing construction markers.

CLines: This option will delete all existing construction lines.

© 2014 Optimum Structural Design, Inc.

Specify: This option allows the user to select which construction geometry to delete using the mouse.

Query

Point: This option options a dialog allowing the user to click on nodes in the model and report their XYZ coordinates.

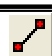
Distance: This option opens a dialog allowing the user to click on nodes and recover their distance from the global origin.

5.10 Creating EndPoints & Additonal Nodes

Quick Reference:

Creating EndPoints

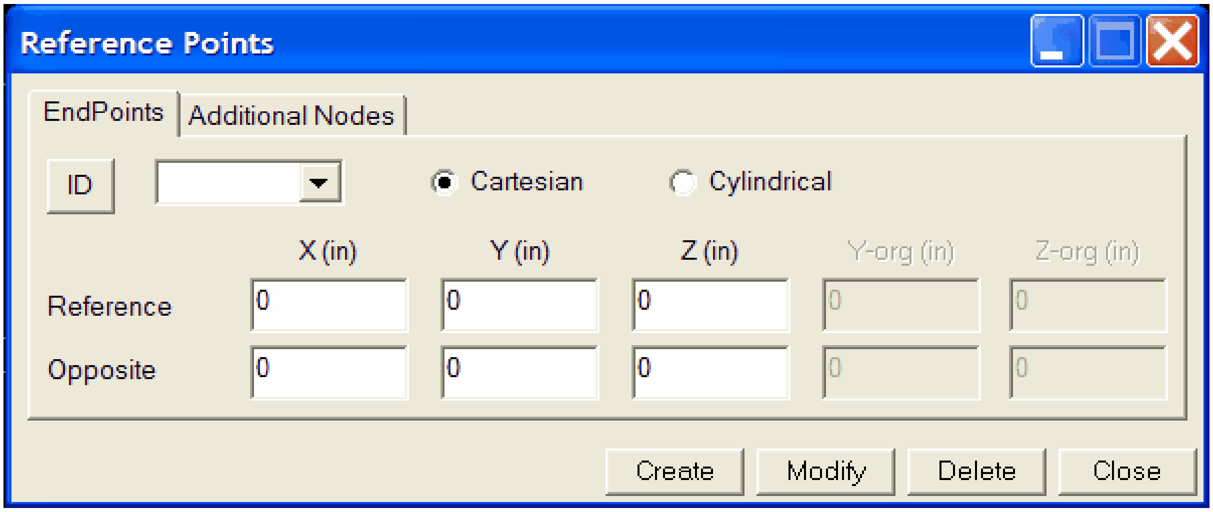
Creating Additional Nodes

Toolbar	
Menu	**Model > Nodes > Create/Modify > EndPoint**
Keyboard	**<Ctrl + n>**

EndPoints are defined by a Reference and Opposite end of the current module. A set of nodes is created along the line between these two points at the module spacing defined in the Parts dialog.

This tutorial shows the procedure for creating a set of EndPoints.

1. Make sure the appropriate module is set as the current part and launch the EndPoints dialog from the **Model > Nodes > Create/Modify > EndPoint** menu, or from the toolbar.



2. Click the ID button to get the next unique ID for the new EndPoints.

The Reference and Opposite X values will be automatically filled according to the module location and sections definition in the parts dialog.

3. The Y and Z values can be typed in by the user, or "snapped" to a point in the model by clicking in one of the coordinate value boxes and then clicking a point in the model.

Values can be "snapped" to other EndPoints, Additional Nodes, or Construction Markers.

4. Click the Create button.

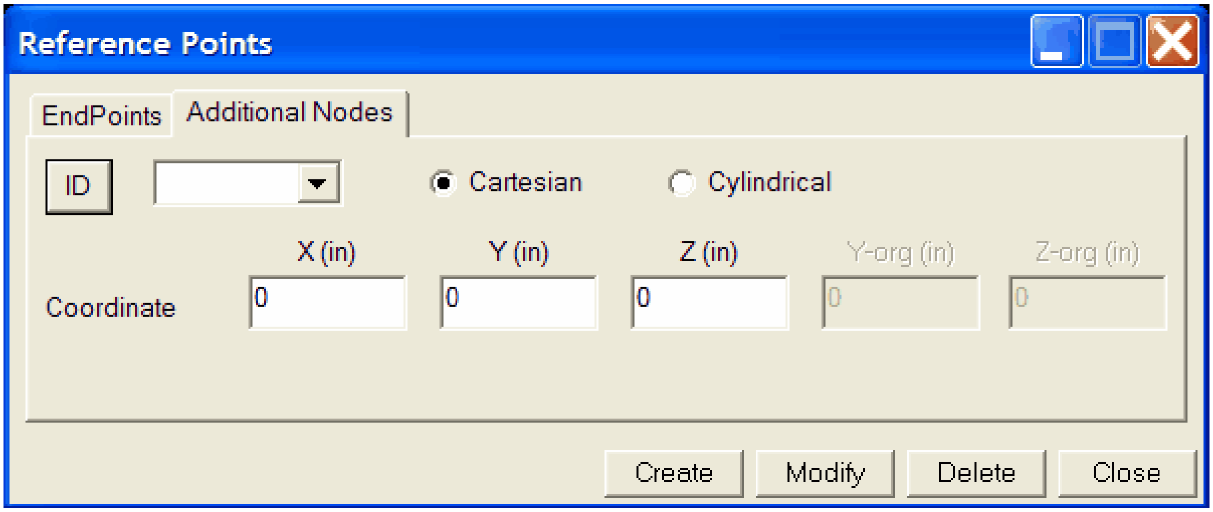
This same procedure can be repeated for additional EndPoints.

Toolbar	
Menu	**Model > Nodes > Create/Modify > Additional Node**
Keyboard	**<Ctrl + n>**

Additional nodes allow the user to insert individual nodes into the model in addition to the sets created with EndPoints.

This tutorial shows the procedure for creating Additional Nodes.

1. Make sure the appropriate module is set as the current part and launch the Additional Nodes dialog from the **Model > Nodes > Create/Modify > Add'l Node** menu, or from the toolbar.



2. Click the ID button to get the next unique ID for the new additional node.

3. Type in the coordinate values or "snap" to a point in the model by clicking in one of the coordinate boxes and the clicking the point in the model.

4. Click the Create button.

This same procedure can be repeated for more additional nodes.

5.11 Creating Strakes

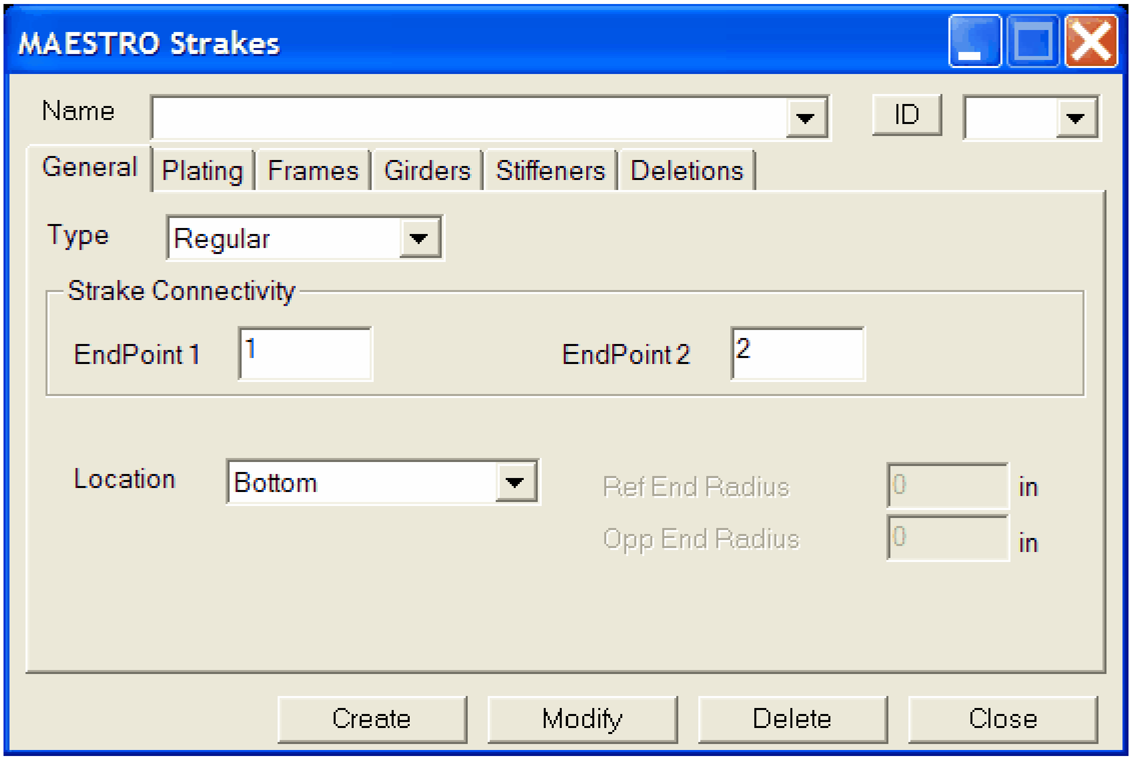
Toolbar	
Menu	**Model > Elements > Create/Modify > Strake**
Keyboard	**<Ctrl + k>**

Strakes are a convenient way of creating all of the structural elements between two sets of end points. The strake data is defined in the MAESTRO Strakes dialog box via the General, Plating, Frames, Girders, Stiffeners, and Deletions tabs.

This tutorial shows the procedure for creating a strake between a set of defined end points.

© 2014 Optimum Structural Design, Inc.

1. Begin by opening the MAESTRO Strakes dialog box from the **Model > Elements > Create/Modify > Strake** menu, or from the toolbar.



2. Make sure the correct module is set to the current part and click the ID button to assign the strake a unique ID.

Note: Frames can be defined as I-beams by selecting "Second Flange" as the strake type. This will use the selected T-beam property for the frame and automatically add a second flange using the T-beam flange properties. See the Second Flange verification section for examples.

3. Click in the EndPoint 1 box and then click the first end then the second end point defining the strake. The two boxes should automatically update with the end point numbers.

Note: you may click anywhere along the line of nodes making up the end point.

4. Select the location of the strake: Bottom, Side, Deck, or Other. This is important for MAESTRO to define the elements as wetted or not. Bottom and Side will assign the shell elements as "wetted".

5. Click the Plating tab and select the plate property from the drop down menu. A new property can be created by clicking the Property box.

6. Click the Frames tab and select the frame property and Frame Web Orientation. If there are no frames on this section, there should be a property with no frames defined. A new property can be created by clicking the Property box.

Frame Web Orientation

Frames in the transverse direction will be in-line with the YZ plane, whereas frames in the normal direction will be normal to the strake plating.

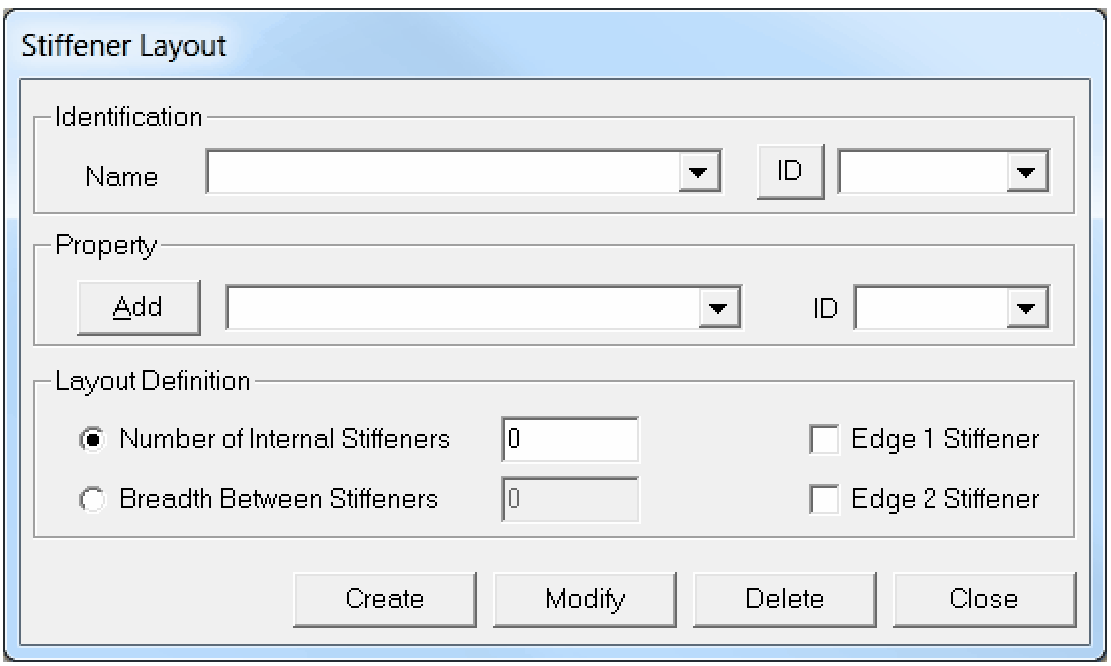
Sign Convention for Normal and Transverse Options

The Normal and Transverse options involve a Positive/Negative choice, and the positive direction is defined by a right hand rule.

To determine the positive direction, the index finger should point from Endpoint 1 to Endpoint 2. The middle finger should point in the +X direction and the thumb will then be pointing in the positive direction.

7. Click the Girders tab and if applicable, click the Enable Girder box and select the girder property and Angle in degrees. If there is no girder to be included in the current strake, make sure the Enable Girder box is unchecked.

8. Click the Stiffeners tab and if applicable choose the stiffener layout desired. If no layouts are defined or to create a new layout, click the Layout button or the Stiffener Layout icon  from the toolbar. See the Stiffener Layouts section for more information.



9. Click the ID button to assign a unique ID and enter a descriptive name for the stiffener layout.

Note: the first stiffener layout should be a null property. This gives the option to define an unstiffened strake or quad.

10. Select the stiffener property from the drop down menu. If the beam element property desired is not already defined, a new one can be created by clicking the Property button.

11. The stiffener layout can be defined by number of internal stiffeners or by defining the breadth between stiffeners.

12. The user can also define if there is an Edge 1 or Edge 2 stiffener.

13. Click Create and then click Close to return to the MAESTRO Strakes dialog box.

© 2014 Optimum Structural Design, Inc.

14. Select the new Stiffener Layout from the drop down menu.

15. Click the Deletions tab and check if any of the strake sections should have deleted plate, frame or girder elements.

16. Click Create.

5.12 Creating Additional Elements

Quick Reference:

Creating Quads

Creating Triangles

Creating Beams

Creating Rods

Creating Springs

Creating RSplines

Creating Brackets

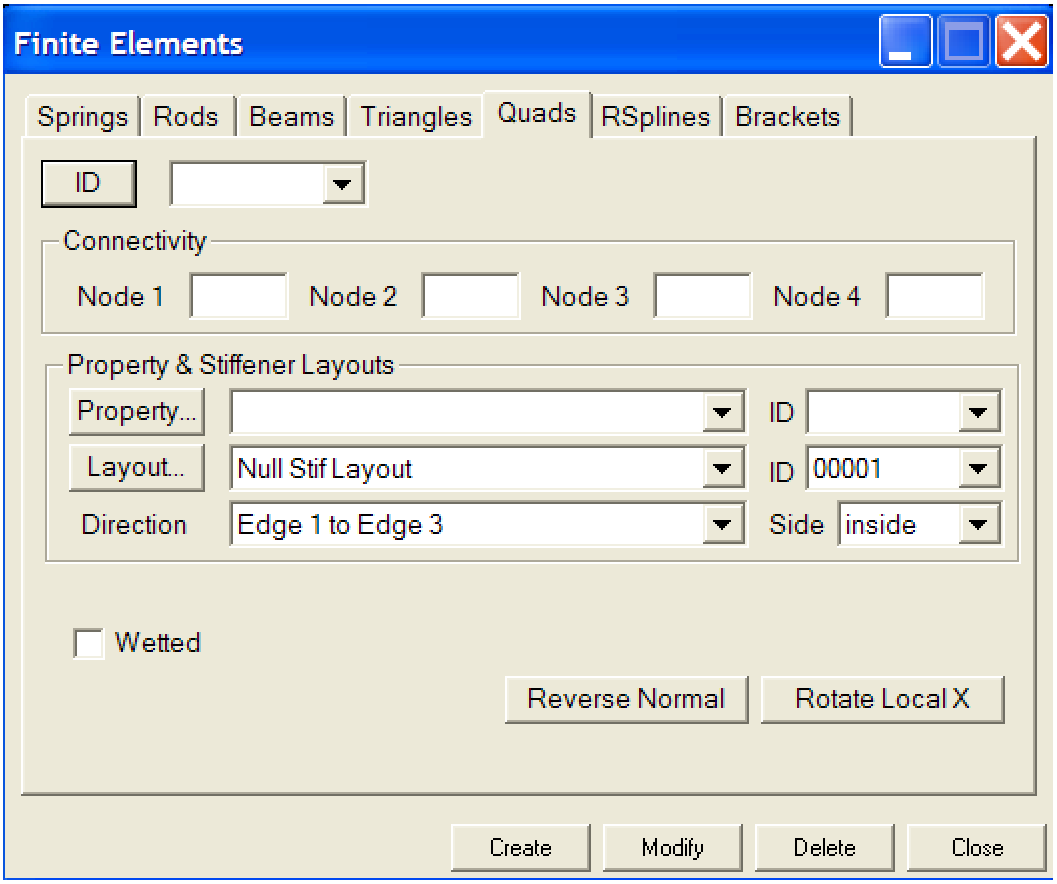
In addition to strakes, MAESTRO has the ability to create individual panel elements as triangles or quads (depending on the number of nodes used to define the element), beam elements, and rod elements.

Toolbar	
Menu	**Model > Elements > Create/Modify > Quad**
Keyboard	**<Ctrl + e>**

This tutorial shows the procedure for creating an individual quad element.

1. Begin by opening the Finite Element Quad dialog box using the **Model > Elements > Create/Modify > Quad** menu, or from the toolbar.

© 2014 Optimum Structural Design, Inc.



2. Make sure the appropriate module is set as the current part and click the ID button to assign a unique ID to the element.

3. Click inside the Node 1 box and then click the four nodes that will define the quad element in the model.

4. Select the plate property from the Property drop down menu. If the property is not defined, click the Property button to define a new plate element property.

5. Select the stiffener layout, if applicable.

6. If a stiffener layout is chosen, chose the orientation of the stiffeners in the Direction drop down menu. The Side is defined as inside:opposite of the pressure side, outside:same as the pressure side.

Note: Edge 1 is defined as the edge between nodes 1 and 2, Edge 2 between nodes 2 and 3, Edge 3 between nodes 3 and 4, and Edge 4 between nodes 4 and 1.

7. Check the Wetted box if the element is part of the bottom or side shell.

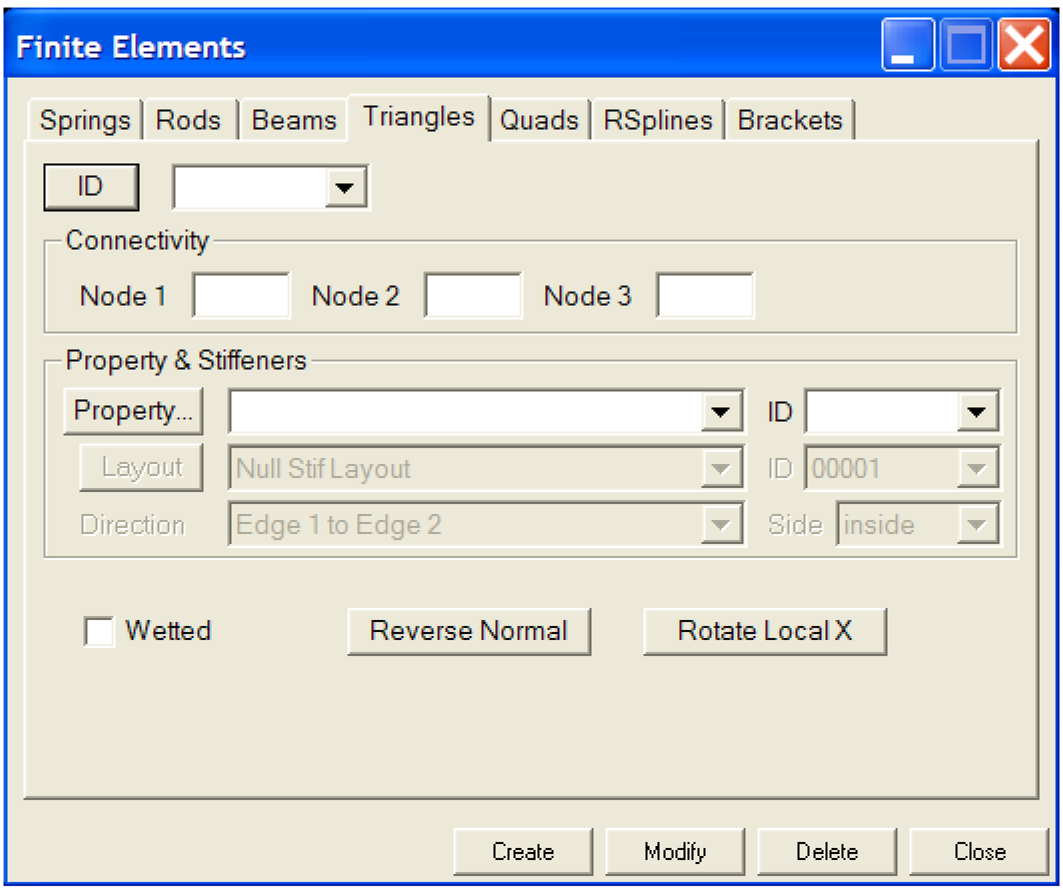
8. Click Create.

This procedure can be repeated to create additional quad elements.

Toolbar	
Menu	**Model > Elements > Create/Modify > Triangle**
Keyboard	**<Ctrl +e>**

This tutorial shows the procedure for creating an individual triangle element.

1. Begin by opening the Finite Element Triangle dialog box using the **Model > Elements > Create/Modify > Triangle** menu, the toolbar, or clicking the Tri tab in the Finite Element dialog box.



2. Make sure the appropriate module is set as the current part and click the ID button to assign a unique ID to the element.

3. Click in the Node 1 box and click the three nodes that make up the triangle element in the model.

4. Select the plate property from the drop down menu. If the property is not defined, click the Property button to create a new plate element property.

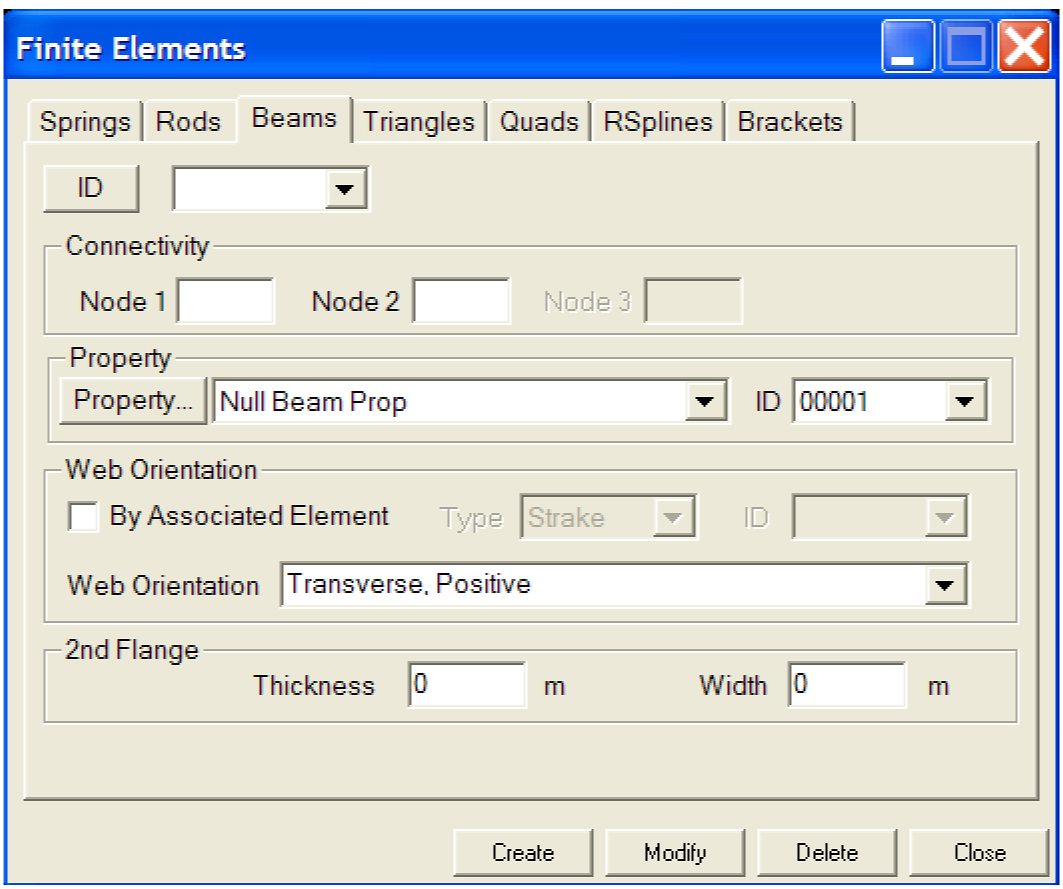
5. Check the Wetted box if the element is part of the bottom or side shell.
6. Click Create.

This procedure can be repeated to create additional triangle elements.

Toolbar	
Menu	**Model > Elements > Create/Modify > Beam**
Keyboard	**<Ctrl +e>**

This tutorial shows the procedure for creating an individual beam element.

1. Begin by opening the Finite Element Beam dialog using the **Model > Elements > Create/ Modify > Beam** menu, the toolbar, or clicking the Beam tab in the Finite Elements dialog box.



© 2014 Optimum Structural Design, Inc.

2. Make sure the appropriate module is set as the current part and click the ID button to assign a unique ID to the element.

3. Click in the Node 1 box and click the two nodes that make up the beam element in the model. Node 1 and Node 2 should now be populated with values representing the ID (and Section number for Endpoints) of the Node and/or Endpoint.

4. Select the beam property from the drop down menu. If the property is not defined, click the Property button to create a new beam element property.

5. The next step is to define the Web Orientation of the beam element. There are a total of four ways to define the orientation of the beam element's web; two methods use a reference plane to define the web orientation while two methods define the web orientation more directly. For the *reference plane* method, the reference plane can be: (1) the plane of an Associated 2D Element or (2) the plane formed by three nodes in space. Alternatively, the web orientation can be defined directly using: (1) the Transverse (YZ) plane of the module or (2) defining a vector (e.g., $\{V_1, V_2, V_3\}$).

Reference Plane Methods

1. Associated Element – In this option the reference plane is the plane of a particular 2D element (i.e., a strake panel, a triangle, or a quad element).

(a) To use this option, first check the *By Associated Element* box in the Web Orientation section of the Additional Element dialog. If this option is selected, the Node 3 field for the Third Node option is grayed out.

(b) Next, the user can choose what type of element and what element ID should be used as the reference plane. For example, the user can choose Type *Strake* followed by selecting (via the ID drop-down or graphically selecting with a mouse left-click) a panel in the strake.

(c) Now that a reference plane has been defined, the user can define the web orientation as being normal to that plane using the Normal (Positive/Negative) option or, for a non-normal orientation, by specifying the angle (in degrees) **away from** the normal. For example, specifying 30 would place the web at 30 degrees from the normal, or 60 degrees from the reference plane.

2. By Defining a Third Node - Alternatively, the reference plane is the plane passing through three nodes: Node 1 and Node 2 at the ends of the beam, and a third node (Node 3) chosen by the user.

(a) To use this option, first ensure that the *By Associated Element* check box IS NOT checked. When this check box IS NOT checked, the Node 3 field should be editable (i.e., un-grayed and ready to receive user input).

(b) Populate the Node 3 field by left-clicking in the Node 3 field (to move the mouse focus here) followed by clicking on any node that, together with Node 1 and Node 2, forms the desired plane. The value in this field represents the ID (and Section number for Endpoints) of Node 3.

(c) Now that a reference plane has been defined, the user can define the web orientation as being normal to that plane using the Normal (Positive/Negative) option or, for a non-normal orientation, by specifying the angle (in degrees) **away from** the normal.

© 2014 Optimum Structural Design, Inc.

For example, specifying 30 would place the web at 30 degrees from the normal, or 60 degrees from the reference plane.

Direct Methods

1. Using the Transverse (YZ) Plane of the Module – The first direct method places the web directly **in** the transverse (YZ) plane of the module and points the web in a *positive* or *negative* direction. This is done by selecting Transverse (Positive/Negative). This option can only be used when the two nodes defining the beam lie in the YZ plane of the module, which forms a *beam vector*.

2. Using a Vector to Directly Define Web Orientation – Finally, there is a second direct method that defines the web direction in terms of a vector in the X, Y and Z coordinates of the module. The three components are placed inside curly braces and separated by commas $\{V_X, V_Y, V_Z\}$. The components can be any length; they do not have to define a unit vector.

Sign Convention for Normal and Transverse Options

The Normal and Transverse options involve a Positive/Negative choice, and the positive direction is defined by a right hand rule.

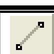
In the Normal option the positive normal is determined by a right hand rule based on three nodes: either the first three nodes of the Associated Element, or nodes 1 and 2 of the beam and the specified Node 3. The index finger points from node 1 to node 2. The middle finger points toward node 3 and both fingers are in the reference plane. Then the thumb points in the positive direction.

In the Transverse option the index finger points from node 1 to node 2 of the beam (defining the *beam vector*), and middle fingers points in the module's positive X direction. The thumb now points in the module's transverse (YZ) plane in the positive direction.

6. Enter values in the 2nd Flange Thickness and Width fields in order to simulate an I-beam by way of using a T-beam property plus a second flange. Please see the Second Flange verification section for examples.

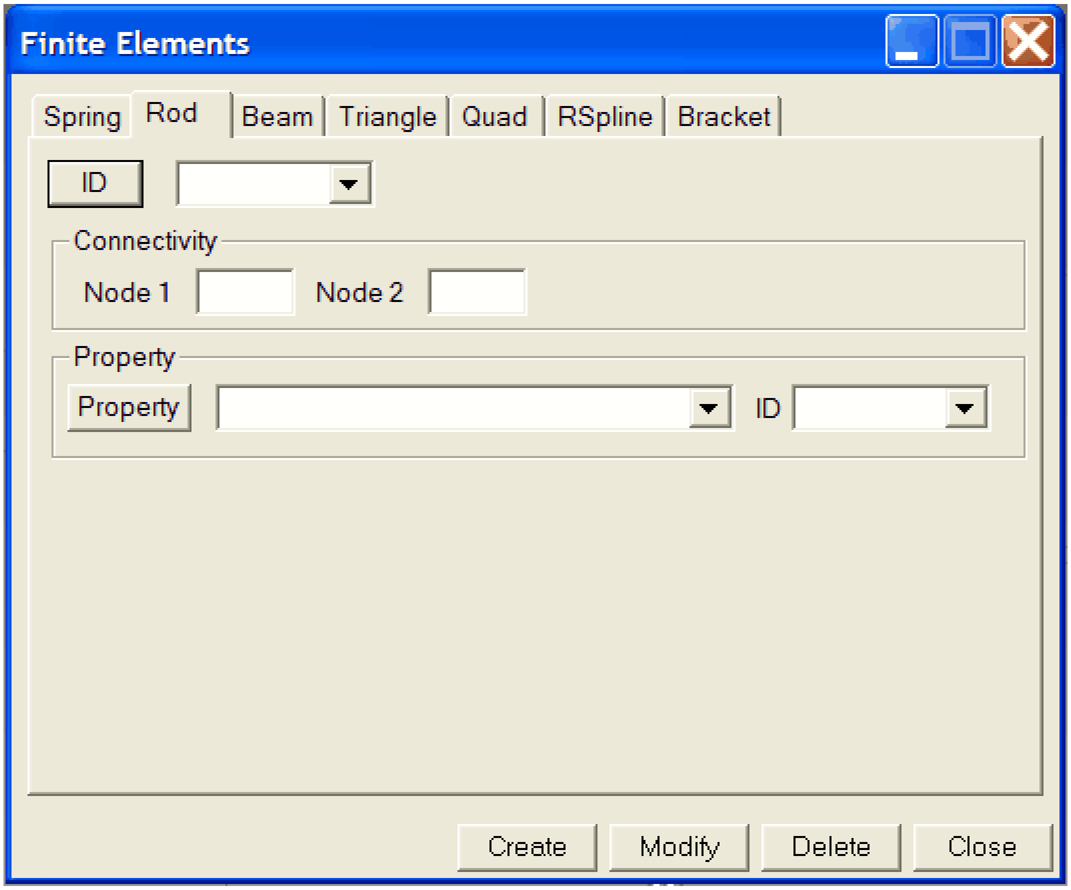
7. Click Create.

This procedure can be repeated to create additional beam elements.

Toolbar	
Menu	**Model > Elements > Create/Modify > Rod**
Keyboard	**<Ctrl +e>**

This tutorial shows the procedure for creating an individual beam element.

1. Begin by opening the Finite Element Rod dialog using the **Model > Elements > Create/ Modify > Rod** menu, the toolbar, or clicking the Rod tab in the Finite Elements dialog box.



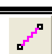
2. Make sure the appropriate module is set as the current part and click the ID button to assign a unique ID to the element.

3. Click in the Node 1 box and click the two nodes that make up the rod element in the model.

4. Select the rod property from the drop down menu. If the property is not defined, click the Property button to create a new rod element property.

5. Click Create.

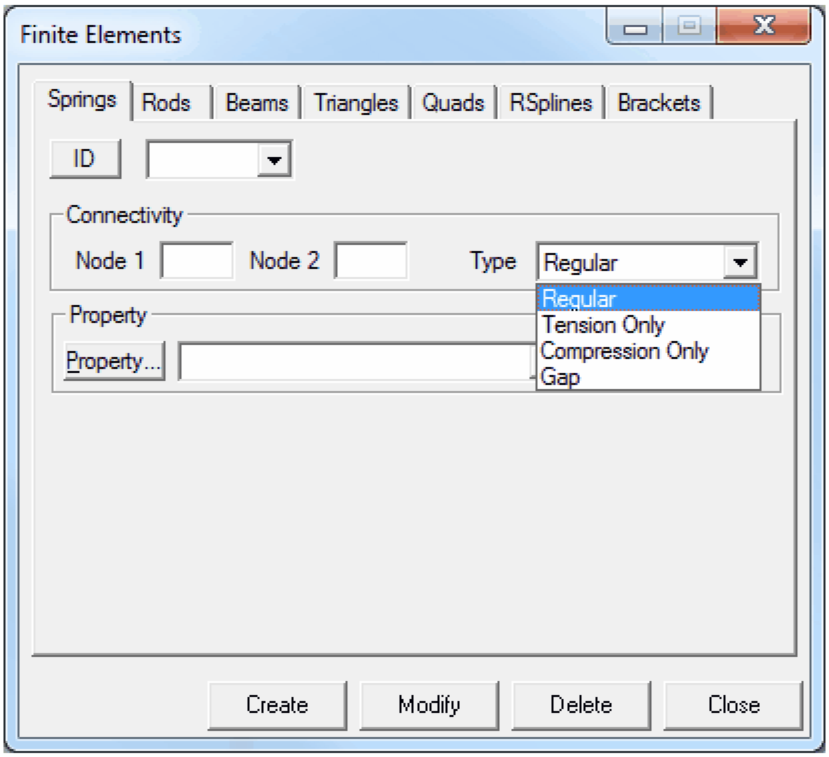
This procedure can be repeated to create additional rod elements.

Toolbar	
Menu	**Model > Elements > Create/Modify > Spring**

Keyboard	<Ctrl +e>

This tutorial shows the procedure for creating a spring element.

1. Begin by opening the Finite Element Spring dialog using the **Model > Elements > Create/Modify > Spring** menu, the toolbar, or clicking the Spring tab in the Finite Elements dialog box.



2. Make sure the appropriate module is set as the current part and click the ID button to assign a unique ID to the element.

3. Click in the Node 1 box and click the two nodes that make up the spring element in the model.

4. Choose the type of spring:

Regular: The spring acts as a regular spring element in tension and compression.

Tension Only: The spring only provides stiffness when it is in tension.

Compression Only: The spring only provides stiffness when it is in compression.

Gap: The spring only adds stiffness when the force on the spring nodes is enough to displace the spring nodes more than the original distance between them.

** See the Verification section of the manual for sample models and comparison results.

© 2014 Optimum Structural Design, Inc.

5. Select the spring property from the drop down menu. If the property is not defined, click the Property button to create a new spring element property.

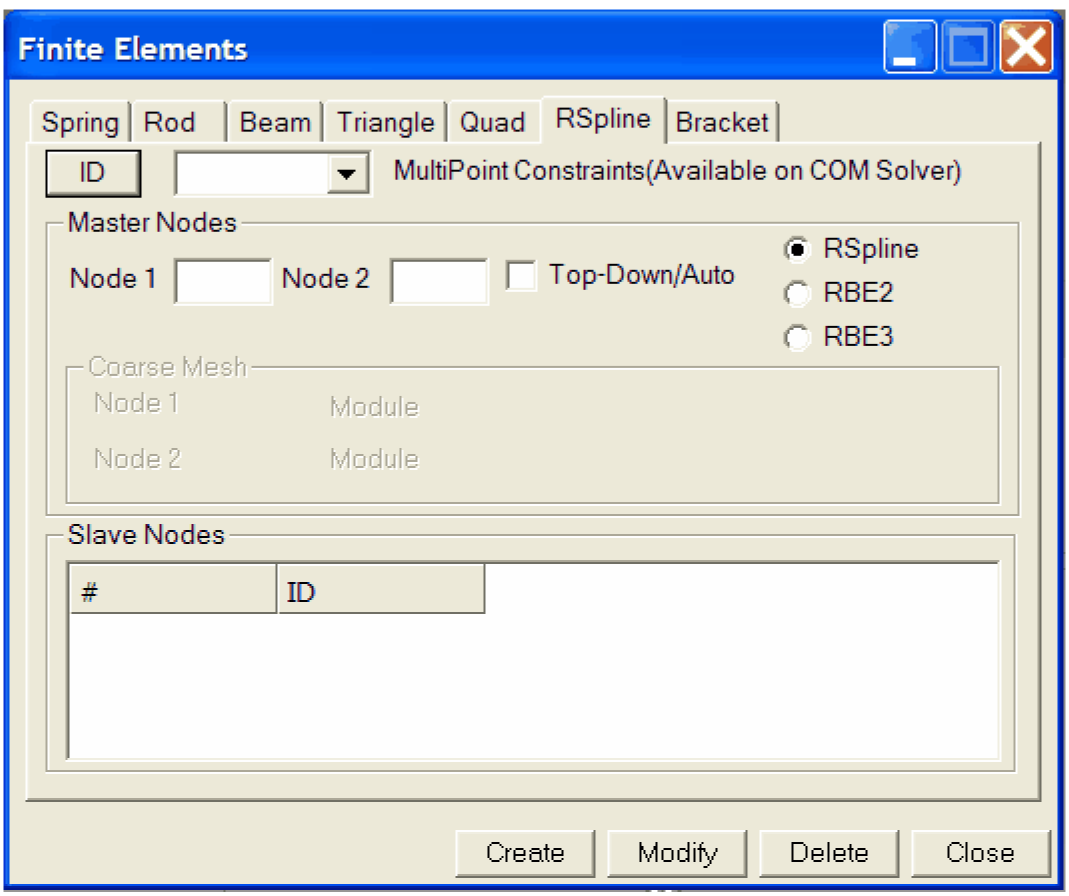
6. Click Create.

This procedure can be repeated to create additional spring elements.

Toolbar	
Menu	**Model > Elements > Create/Modify > RSpline**
Keyboard	**<Ctrl + e>**

This tutorial shows the procedure for creating an rigid spline element (RSpline).

1. Begin by opening the Finite Element RSpline dialog using the **Model > Elements > Create/Modify > RSpline** menu, the toolbar, or clicking the RSpline tab in the Finite Elements dialog box.



2. Make sure the appropriate module is set as the current part and click the ID button to

assign a unique ID to the element.

3. Select the type of RSpline element. For a description of the three types, please see the Model Organization section.

4. Click in the Node box and click the node that makes up the first part of the rigid spline element in the model.

5. Checking Top-Down can be used to create a rigid link between a coarse mesh and fine mesh model. If Top-Down/Auto is selected, the slave nodes of the fine mesh model between the two master nodes in the coarse mesh model will be automatically added.

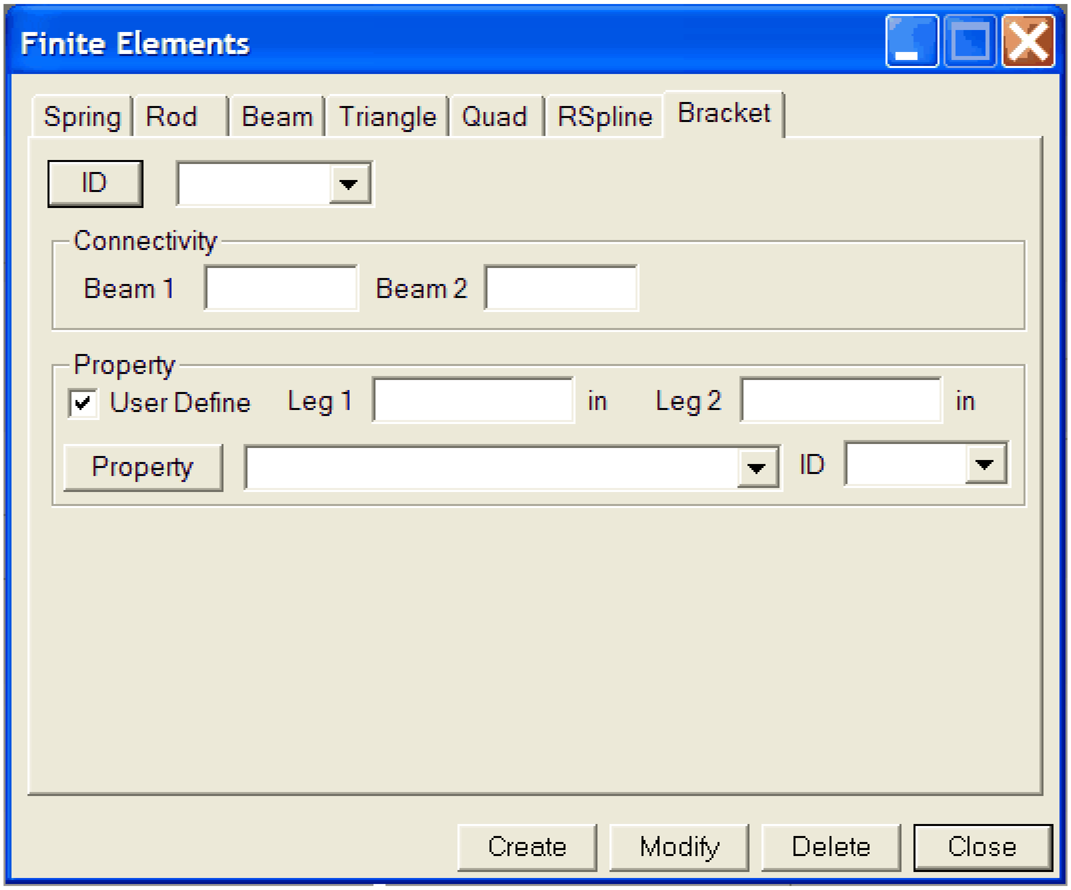
6. Click Create.

This procedure can be repeated to create additional rigid spline elements.

Toolbar	
Menu	**Model > Elements > Create/Modify > Bracket**
Keyboard	**<Ctrl + e>**

This tutorial shows the procedure for creating a bracket element.

1. Begin by opening the Finite Element Bracket dialog using the **Model > Elements > Create/Modify > Bracket** menu, the toolbar, or clicking the Bracket tab in the Finite Elements dialog box.



2. Make sure the appropriate module is set as the current part and click the ID button to assign a unique ID to the element.

3. Click in the Beam 1 box and click the two beams that will be connected by the bracket element.

4. Select a property from the drop-down box, or click *Property* to define a new one.

5. Check the *User Define* box if you want to define the length of Leg 1 and Leg 2 of the bracket. Otherwise MAESTRO will automatically calculate these values.

6. Click Create.

This procedure can be repeated to create additional bracket elements. For more information on brackets, see the Bracket Verification section.

5.13 Creating Compounds

Toolbar	

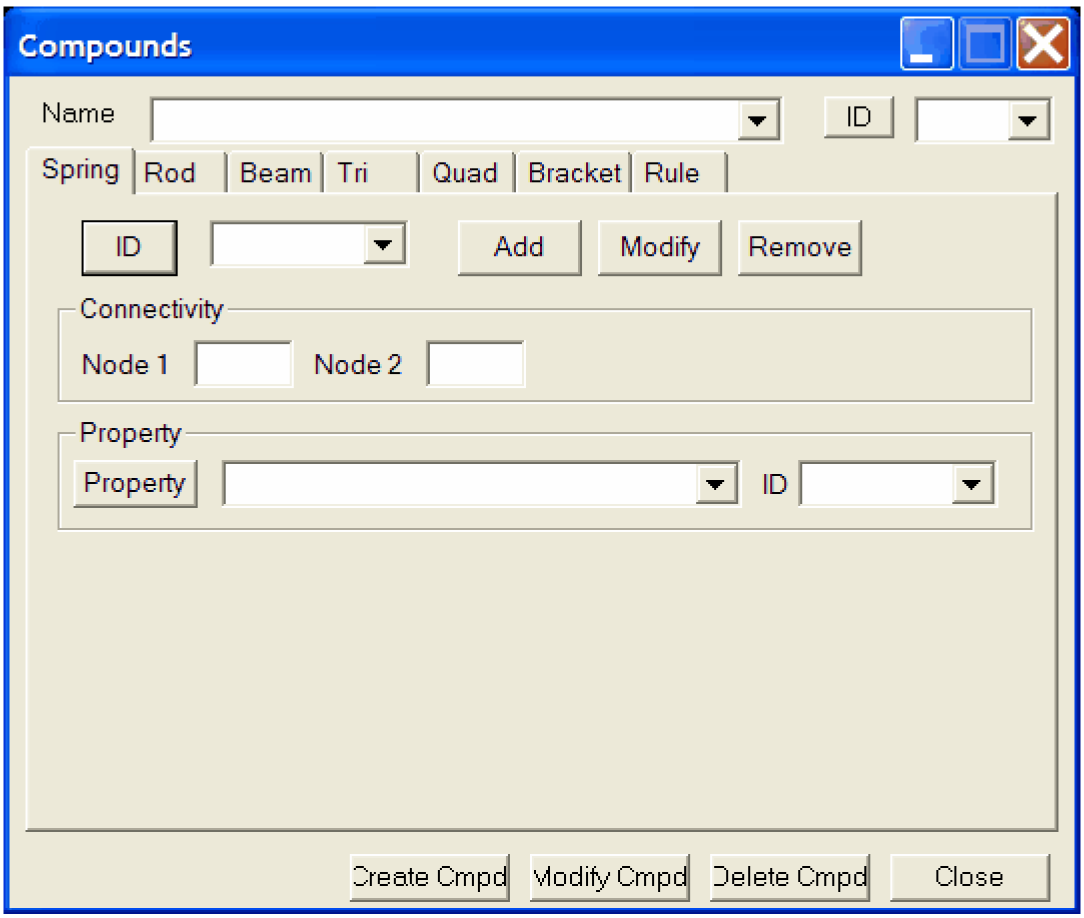
© 2014 Optimum Structural Design, Inc.

Menu	Model > Elements > Create/Modify > Compound

Compound elements provide a convenient method for quickly creating repetitive transverse structure. Compounds are made up of spring, rod, beam, and plate elements. A prototype is created once and can be repeated along a module's longitudinal direction using the replication rule.

This tutorial shows the procedure for creating a compound prototype and setting the replication rule.

1. Begin by opening the Compounds dialog box using the **Model > Elements > Create/ Modify > Compound** menu, or from the toolbar.



2. Make sure the appropriate module is set as the current part and click the ID button at the top of the dialog to assign a unique ID to the compound.

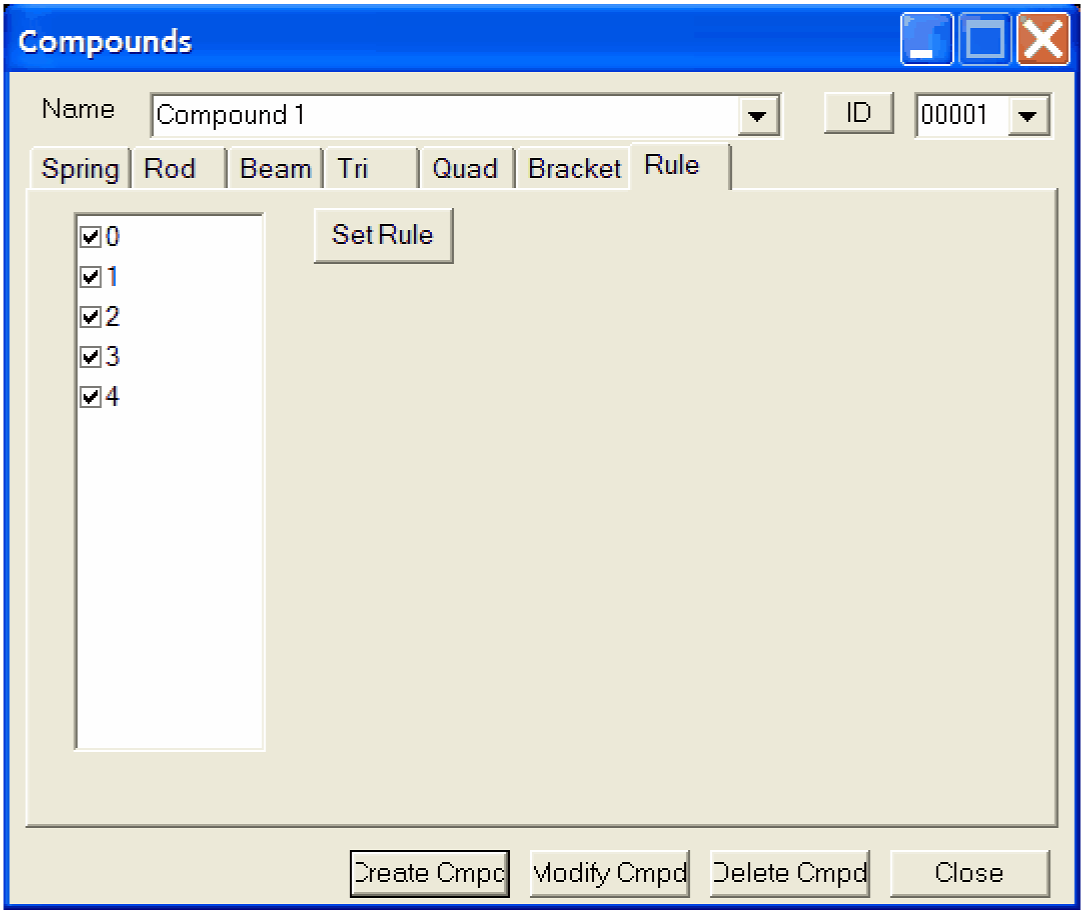
3. Type in a descriptive name for the compound in the Name box and click Create Cmpd.

4. Create Springs, Rods, Beams, Tris, and Quads that make up the compound in the same procedure as creating an additional element, but instead of clicking Create, click Add in the specific element tab.

© 2014 Optimum Structural Design, Inc.

It is a good convention to define all elements on the reference end and set them for additional sections using the Rule tab.

5. Click the Rule tab.



The Rule tab allows the user to set at which sections within the module the compound will be created. The *0* represents the reference end and the last number represents the opposite end. Checking the box for each section will apply the defined compound at that section.

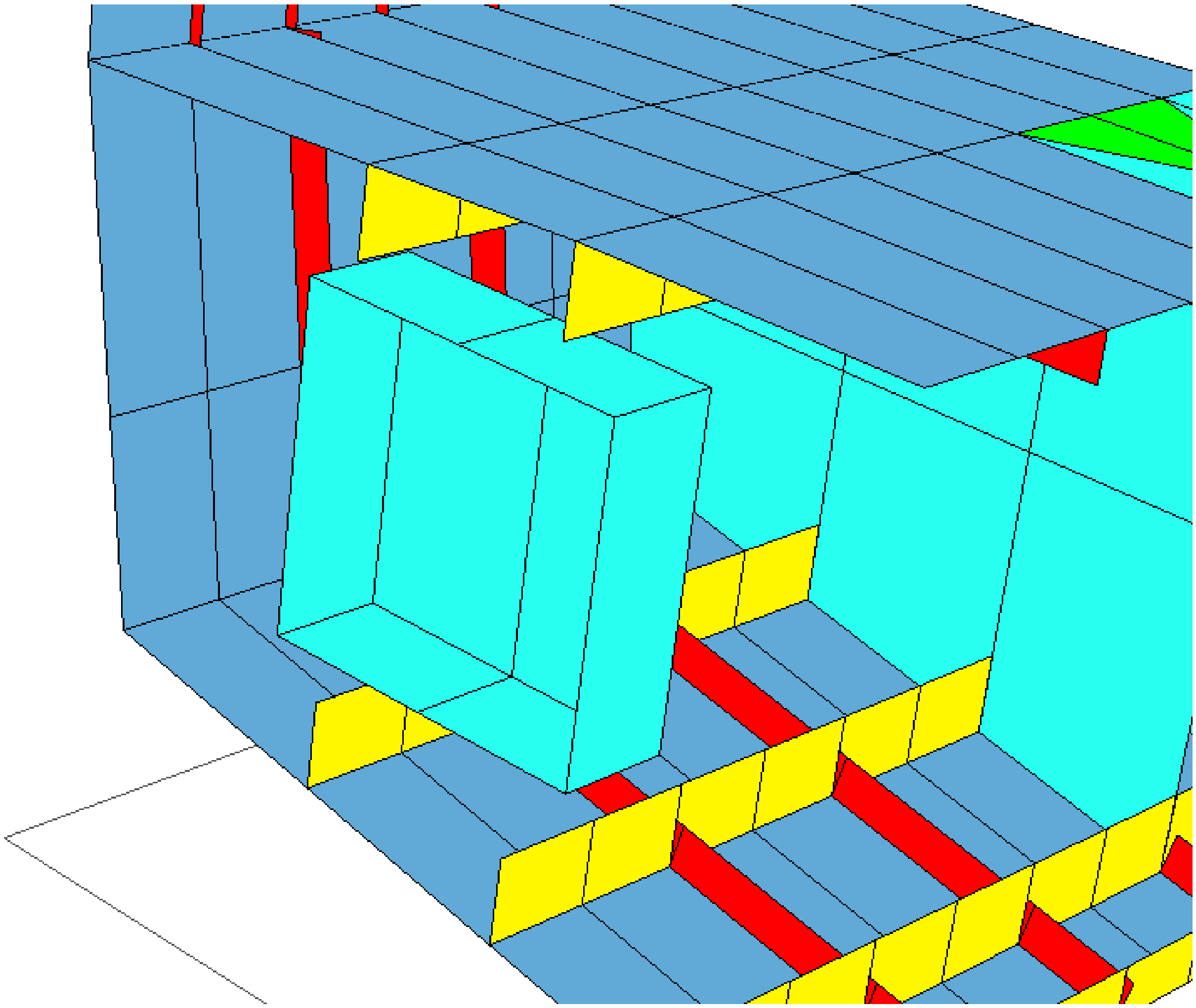
6. Click the Set Rule button.

This procedure can be repeated to create additional compounds.

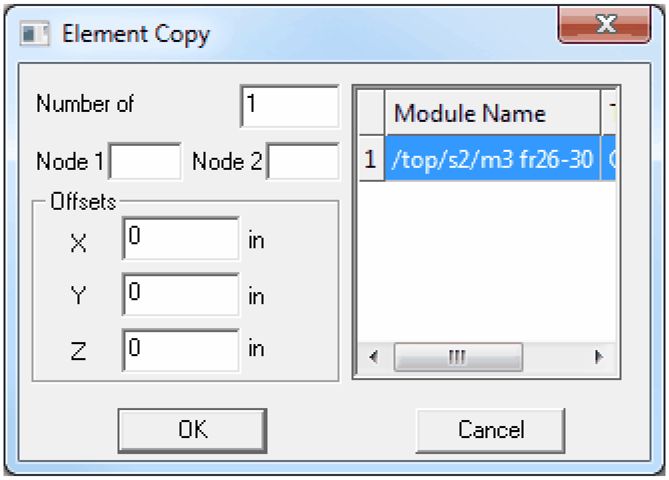
Compounds can have elements added or deleted at any time by adding or deleting elements in their specific tabs. The Modify Cmpd button is only used to modify the name of the compound. The entire compound can be deleted using the Delete Cmpd button.

5.14 Copying Elements

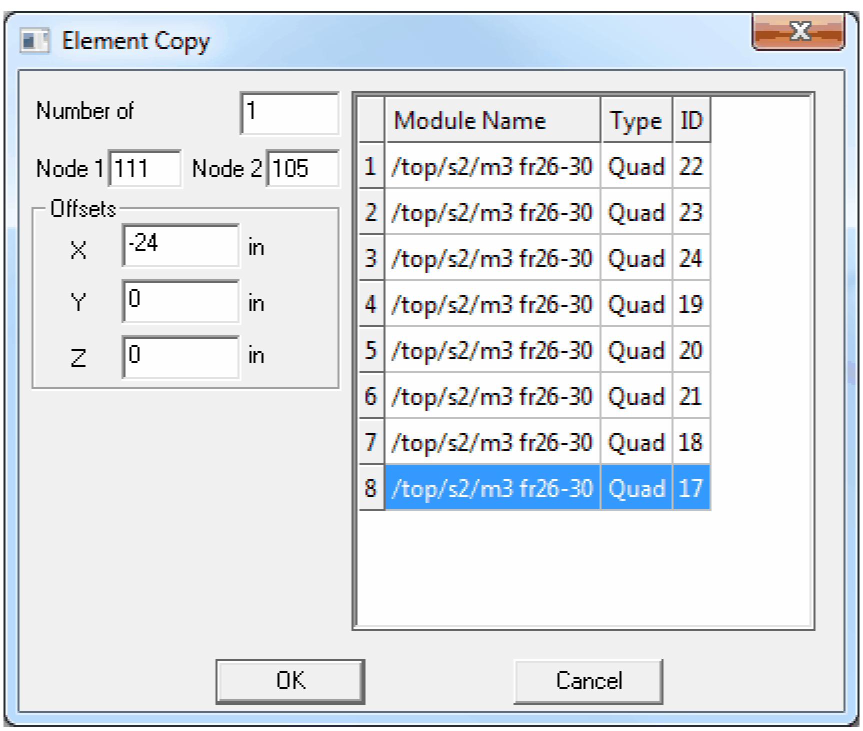
Similar to the module copy function, the element copy provides a method to quickly create structure by leveraging existing elements. As an example, we'll show how the tank structure below can be quickly expanded in the longitudinal direction.



The copy element command can be launched from the Model > Elements > Copy menu option, or by dynamically querying an element and right-clicking and selecting copy. This command will open the Element Copy dialog:

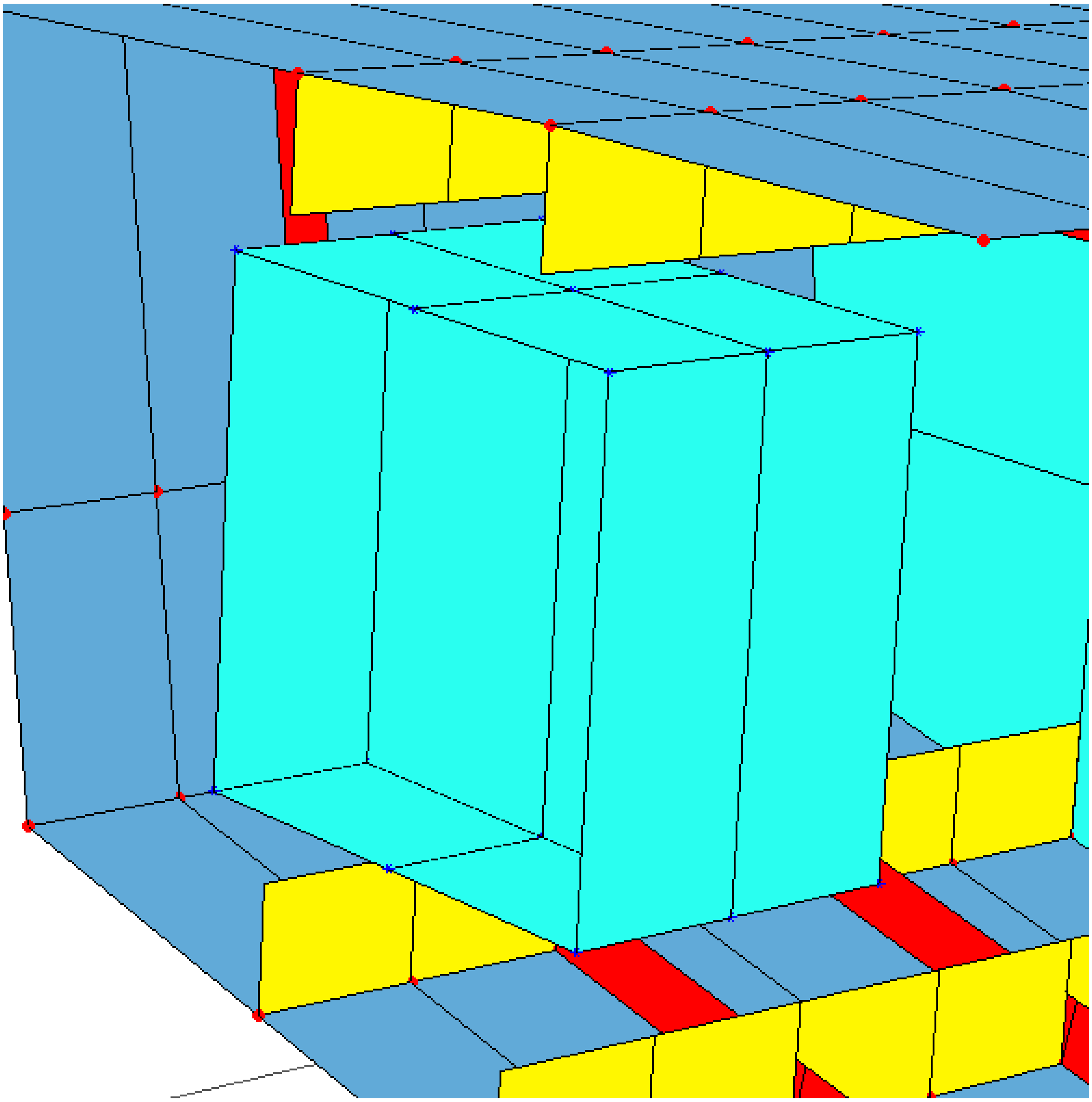


The right-hand side of the dialog will show a list of the elements to be copied. To add more elements, click once in the list white space and then click the elements in the model to copy. In the case of the example, we want to click all of the elements making up the tank. Next we want to set the number of copies to make. In this example we just want to extend our tank one endpoint spacing so we will leave this at 1. The last information we need to specify is the offset to copy the elements by. This can be set by clicking on two nodes in the model to automatically populate the Offsets input, or the user can manually enter these values. In this example, we can click on two nodes on the top of the tank in the direction we want to offset the copied elements. The Element Copy dialog should now look like this:



© 2014 Optimum Structural Design, Inc.

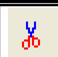
Next, Click OK and the new copied elements (and necessary nodes) are automatically created.



To complete the example, the old end shell elements (which are now internal to the tank) can be deleted.

© 2014 Optimum Structural Design, Inc.

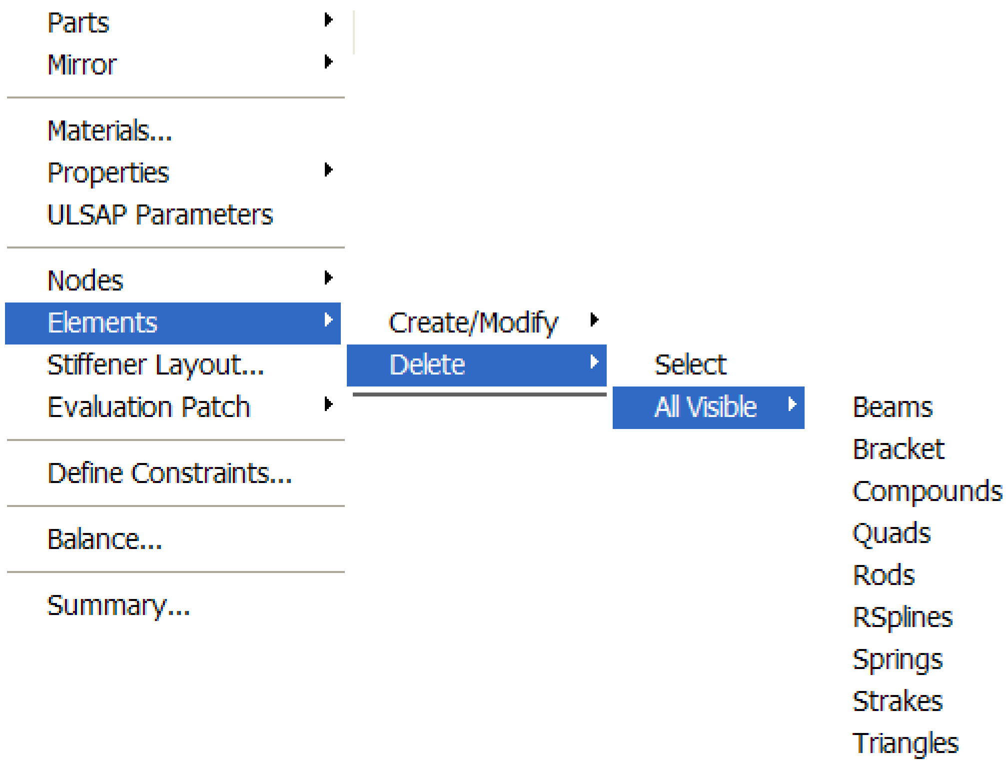
5.15 Deleting Elements

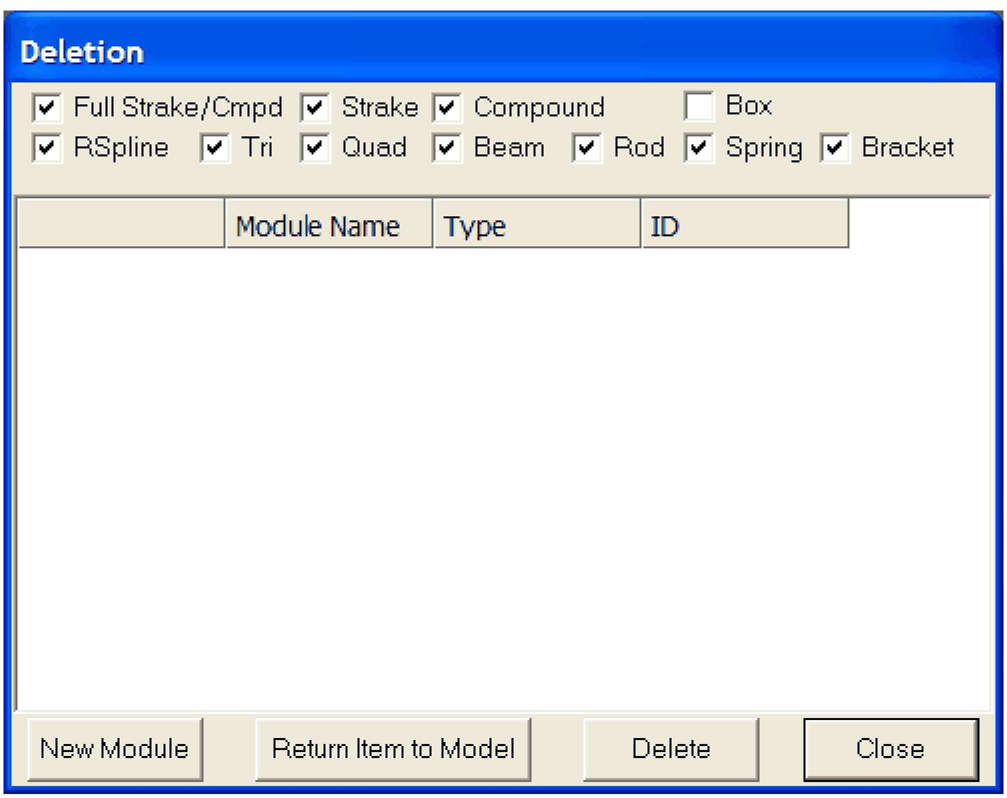
Toolbar	
Menu	**Tools > Deletions > Select** **Tools > Deletions > All Visible >** **Model > Elements > Delete >**
Keyboard	**<Ctrl + d>**

In addition to deleting elements in the Strakes, Additional Elements, or Compounds dialog boxes, the user can delete specific elements by selecting them with the mouse or by ID or delete all of one element type at once.

This tutorial shows the different ways an element or elements can be deleted.

1. An element can be deleted using the **Model > Elements > Delete >** menu, the **Tools > Deletions > Select, Tools > Deletions > All Visible >** or from the toolbar.





2. The Model menu allows the user to select the type of element to be deleted. A specific element can be selected using the mouse or by the element ID.

3. The Deletion dialog box allows the user to select multiple elements to delete by clicking on them with the mouse.

The check boxes at the top of the screen serve as a filter for which types of elements may be deleted. Checking "Box" allows the user to click in the deletions dialog and then select elements to delete by drawing a box around them. Note: the element filters will not hold when using the select by box option.

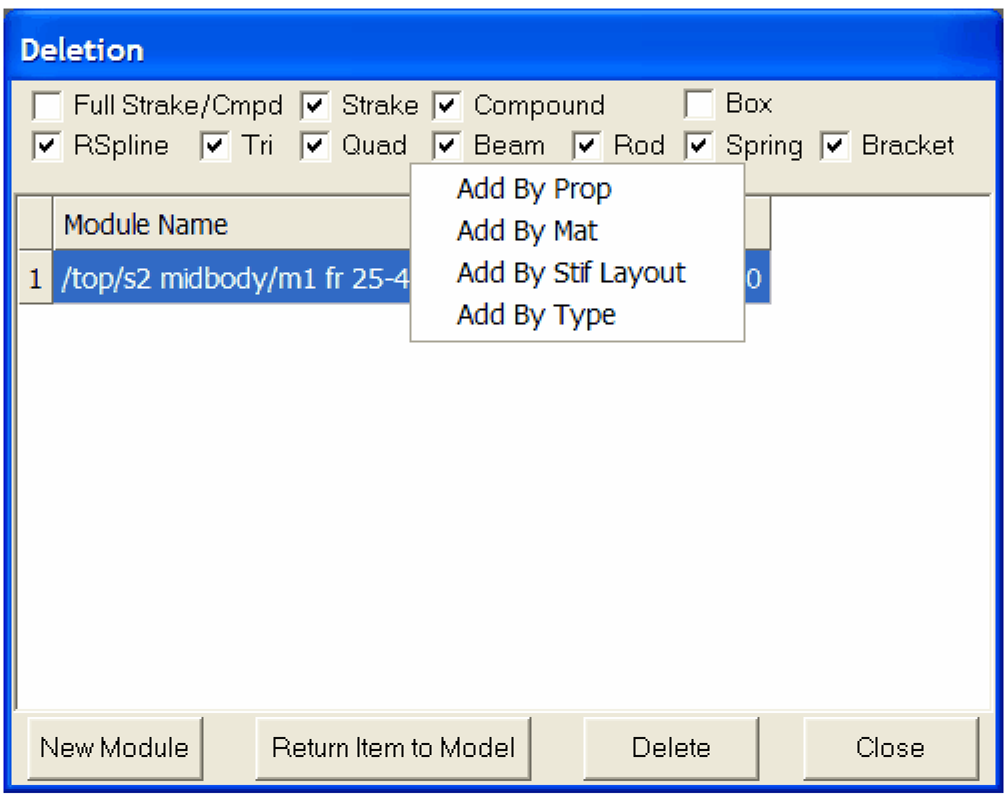
4. The selected elements will appear in a list with their Module Name, Type and ID. The selected elements will also be removed in the model view.

To undelete an item, select it from the list and click the Recover Item button.

5. Click Delete to permanently delete the selected elements.

6. All visible elements can be deleted from the **Tools > Deletions > All Visible >** menu. This allows the user to delete all visible elements of a specific type.

Once an element is added to the Deletion dialog, the user can right click on the element line and a new menu will appear with additional deletions options.



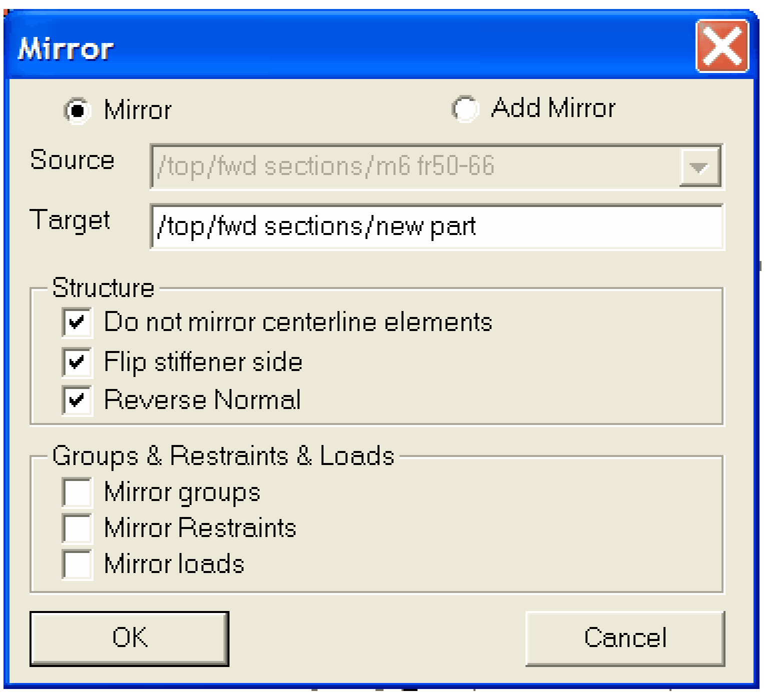
These options will add the elements from the current view part that have the either the same property, material, stiffener layout, or type as the currently selected element from the dialog box. Note: using the right-click menu does not take into account the selected element filters.

The *New Module* button will create a new module consisting of the elements added to the deletions dialog.

5.16 Mirroring a Model

There are two options when mirroring a part or the full model: Mirror and Add Mirror.

The mirror dialog can be opened by right clicking on a part in the parts tree and selecting Mirror or Add Mirror. The dialog can also be launched from the **Model > Mirror > Parts >** or **Model > Mirror > Model** menu.



Mirror

The mirror function will create a mirror of the selected part(s) with a new name defined by the user. The user also selects the new location, or Target of the mirrored part(s).

Add Mirror

© 2014 Optimum Structural Design, Inc.

The add mirror function will create a mirror of the selected part(s), but includes the mirrored parts in the original part(s).

The location of the new part(s) is such that the origin is at the same location as the original part but all nodes and elements are reflected through the local X-Y plane of the original part.

Structure Options

Do not mirror centerline elements: By checking this box, MAESTRO will assume all centerline elements were modeled with their full properties and thus not create a copy.

Flip Stiffener Side: By checking this box, MAESTRO will flip the stiffener side of the mirrored elements such that the stiffener orientation matches with the original part(s).

Reverse Normal: By checking this box, MAESTRO will reverse the normal direction of the mirrored elements such that the element normal side is preserved in the original part(s).

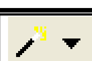
Groups & Restraints & Loads Options

Mirror Groups: By checking this box, any groups created in the original part(s) will be mirrored in the new part(s). Groups tagged as centerline groups will automatically combine the two groups into a single group.

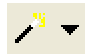
Mirror Restraints: By checking this box, any restraints not on the centerline will be mirrored and applied to the new part(s).

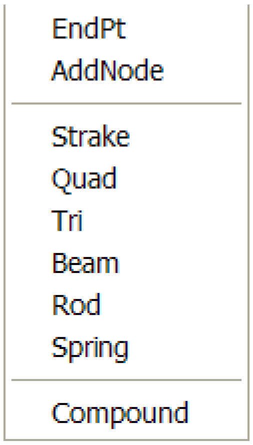
Mirror Loads: By checking this box, any defined loads will be mirrored and applied to the new part(s).

5.17 Quick Create

Toolbar	

Menu	**N/A**

The Quick Element Creation icon  is composed of a toggle button and a drop-down button.



This capability is a powerful tool used during the modeling building phase in the FEA process. Toggling this command On, the user can create *endpoints, nodes, strakes, quads, triangles, beams, rods, springs,* and *compounds* without the use of dialog boxes. The use of this functionality will be discussed below.

EndPoint

Additional Node

Strake

Quad

Triangle

Beam

Rod

Spring

Compound

EndPoint Creation

© 2014 Optimum Structural Design, Inc.

To create an Endpoint using the Quick Element Creation method, it is important to note that the user must first define construction markers. Without construction markers, this functionality will not work.

Select the Endpoint option from the drop-down menu. The user can now single click on the first construction marker in the model that is to become the endpoint's reference end and then double click on the construction marker that will become the endpoint's opposite end. Repeat this process for as many endpoints as you wish.

After completing the Quick Creation sequence there are two important things to remember. The first is to toggle off the quick creation icon and second, the Quick Creation process does not update the ID numbers in the dialog box. Therefore if you are going to continue using the dialog box you must close the Reference Points Dialog Box and the reopen it.

Additional Node Creation

To create an Additional Node using the Quick Element Creation method, it is important to note that the user must first define construction markers. Without construction markers, this functionality will not work.

Select the AddNode option from the drop-down menu. The user can now double click on the construction marker in the model. Repeat this process for as many additional nodes as you wish.

After completing the Quick Creation sequence there are two important things to remember. The first is to toggle off the quick creation icon and second, the Quick Creation process does not update the ID numbers in the dialog box. Therefore if you are going to continue using the dialog box you must close the Reference Points Dialog Box and the reopen it.

Strake Creation

The user must first create a "prototype" strake, in the usual manner, for the other strakes. This will serve to define all of the future strake properties except for their endpoint connectivity. After creating this "prototype", select the Strake option from the drop-down menu. The user can now single click on the first endpoint of the strake and then double-click the second endpoint of the strake. Repeat this process for as many strakes as you wish, but remember these new strakes are based on the properties of the "prototype" strake.

After completing the Quick Creation sequence there are two important things to remember. The first is to toggle off the quick creation icon and second, the Quick Creation process

© 2014 Optimum Structural Design, Inc.

does not update the ID numbers in the dialog box. Therefore if you are going to continue using the dialog box you must close the Strake Dialog Box and the reopen it.

Quad Creation

The user must create a "prototype" quad, in the usual manner, for the other elements. This will serve to define all of the properties except for their node locations. After creating this "prototype", select the Quad option from the drop-down menu. The user can now single click on the first three nodes in the model that are to become the element nodes 1, 2, and 3 in the new quad, and then double click on the fourth node. Repeat this process for as many quad elements as you wish, but remember these new quads are based on the properties of the "prototype" quad.

After completing the Quick Creation sequence there are two important things to remember. The first is to toggle off the quick creation icon and second, the Quick Creation process does not update the ID numbers in the dialog box. Therefore if you are going to continue using the dialog box you must close the Finite Element Dialog Box and the reopen it.

Triangle Creation

The user must create a "prototype" triangle element, in the usual manner, for the other elements. This will serve to define all of the properties except for the node locations. After creating this "prototype" triangle, select the Triangle option from the drop-down menu. The user can now single click on the first two nodes in the model that are to become the element nodes 1 and 2 in the new triangle, and then double click on the third node. Repeat this process for as many triangle elements as you wish, but remember these new triangles are based on the properties of the "prototype" triangle.

After completing the Quick Creation sequence there are two important things to remember. The first is to toggle off the quick creation icon and second, the Quick Creation process does not update the ID numbers in the dialog box. Therefore if you are going to continue using the dialog box you must close the Finite Element Dialog Box and the reopen it.

Beam Creation

The user must create a "prototype" beam element, in the usual manner, for the other elements. This will serve to define all of the properties except for the node locations. After creating this "prototype" beam, select the Beam option from the drop-down menu. The user can now single click on the first node in the model that is to become the element node 1 in the new beam, and then double click on the second node. Repeat this process for as many beam elements as you wish, but remember these new beams are based on the "prototype"

© 2014 Optimum Structural Design, Inc.

beam.

After completing the Quick Creation sequence there are two important things to remember. The first is to toggle off the quick creation icon and second, the Quick Creation process does not update the ID numbers in the dialog box. Therefore if you are going to continue using the dialog box you must close the Finite Element Dialog Box and the reopen it.

Rod Creation

The user must create a "prototype" rod element, in the usual manner, for the other elements. This will serve to define all of the properties except for the node locations. After creating this "prototype" rod, select the Rod option from the drop-down menu. The user can now single click on the first node in the model that is to become the element node 1 in the new rod, and then double click on the second node. Repeat this process for as many rod elements as you wish, but remember these new rods are based on the properties of the "prototype" rod.

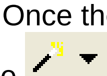
After completing the Quick Creation sequence there are two important things to remember. The first is to toggle off the quick creation icon and second, the Quick Creation process does not update the ID numbers in the dialog box. Therefore if you are going to continue using the dialog box you must close the Finite Element Dialog Box and the reopen it.

Spring Creation

The user must create a "prototype" spring element, in the usual manner, for the other elements. This will serve to define all of the properties except for the node locations. After creating this "prototype" spring, select the Spring option from the drop-down menu. The user can now single click on the first node in the model that is to become the element node 1 in the new spring, and then double click on the second node. Repeat this process for as many spring elements as you wish, but remember these new springs are based on the properties of the "prototype" spring.

After completing the Quick Creation sequence there are two important things to remember. The first is to toggle off the quick creation icon and second, the Quick Creation process does not update the ID numbers in the dialog box. Therefore if you are going to continue using the dialog box you must close the Finite Element Dialog Box and the reopen it.

Compound

The compound quick create functionality works similar to the other quick create options. The *Compounds* menu option is a toggle on or off and will show a check mark when it is on. To use the compound quick create, you must first create a compound. Once the compound is created, you can toggle on the compound quick create option from the  icon. Next,

© 2014 Optimum Structural Design, Inc.

create a "prototype" element in the compound, and then follow the procedure for creating additional elements of that type after selecting the type from the quick create menu.

After completing the Quick Creation sequence there are two important things to remember. The first is to toggle off the quick creation icon and compound option and second, the Quick Creation process does not update the ID numbers in the dialog box. Therefore if you are going to continue using the dialog box you must close the Compound Dialog Box and the reopen it.

5.18 Defining Restraints

Toolbar	
Menu	**Model > Define Restraints...**
Keyboard	**<Ctrl + r>**

Restraints are used to restrict the model's movement in any of the 3 translational or 3 rotational degrees of freedom.

Objectives in Restraining a Model

Define Restraints Through the Model > Define Restraints Menu

Automatic Inertia Relief Restraints Method

Defining Module End Restraints (Create Boundary Modules Command)

The following tutorial shows how to define restraints for a model. In addition to the **Model > Define Restraints** dropdown, which is addressed below, the dynamic query option may be used to query a node and edit the restraints. Right-click while dynamically querying a node and select **Restraints**. This option will allow you to restrict the node in any combination of the three rotations and translations. The **Fixed** option will set all rotational and translational restraints. The **Pinned** option will allow all rotations but no translations. Springs can be set by choosing **Spring** and then one of six directions to place the second node which is fixed. Refer to Spring Element Properties for more information on springs. The **Remove** option will remove all constrains from a node, but will not remove springs.

Objectives in Restraining a Model

To solve the structures problem in FEA, adequate restraints must be modeled. The problem may be singular or without a solution if proper restraints are not modeled. Alternatively, the deformation and stresses may be associated with a rigid body translation if proper restraints are not modeled. To ensure none of the above occurs, all six degrees of freedom must be constrained. There are several methods in place to accomplish this task. Constraining the model twice transversely at different x values will prevent translation in the z-direction and rotation about the y-axis. Constraining the model at each end section in the y-direction will prevent y translation and rotation about the z-axis. Constraining the model twice in the transverse direction at different y values will prevent rotation about the x-axis. Alternatively, the user could constrain the model twice in the y-direction at two different transverse locations.

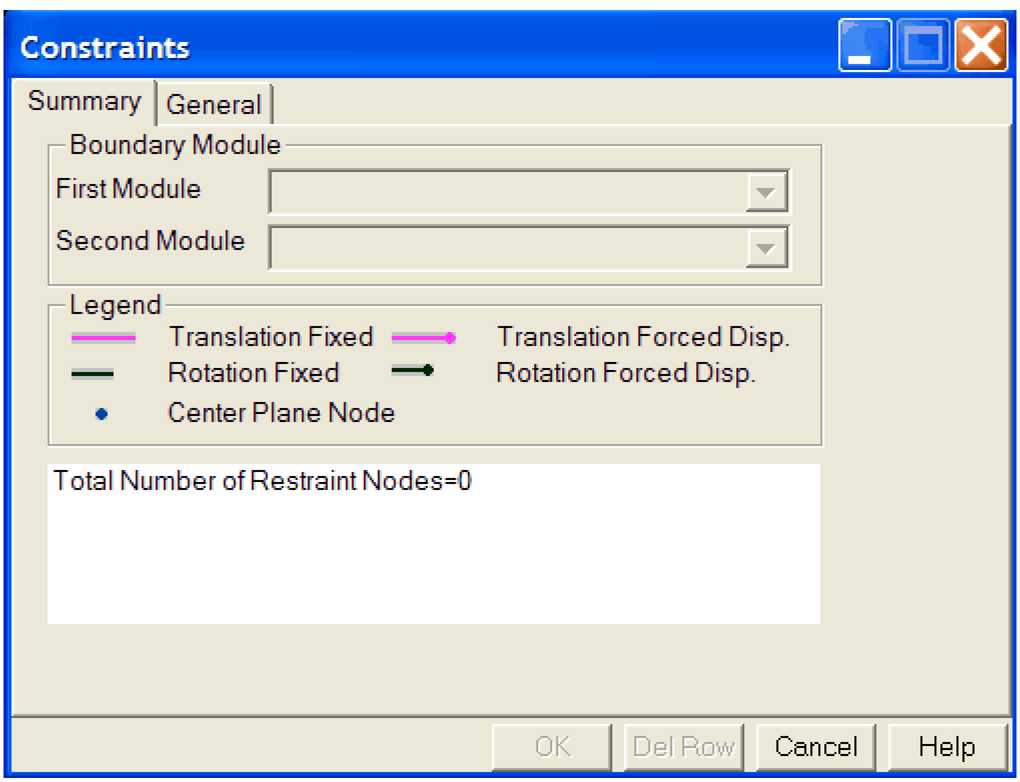
In addition to using user defined restraints, MAESTRO will constrain a model in the z-direction and about the x- and y-axes if the model is marked as transverse symmetric.

Lastly, another way of constraining the model is by using spring elements. These tend be to "softer" in that they allow motion at the structural nodes to which they are attached, but may be used to reduced stresses and abnormal local deformations that usually occur at hard restraints.

To read more about defining restraints continue to the below section "The Define Restraints Menu." To look at several examples of constraining the FEA model and the viewing the results see the below section "Defining Module End Restraints."

The Define Restraints Menu

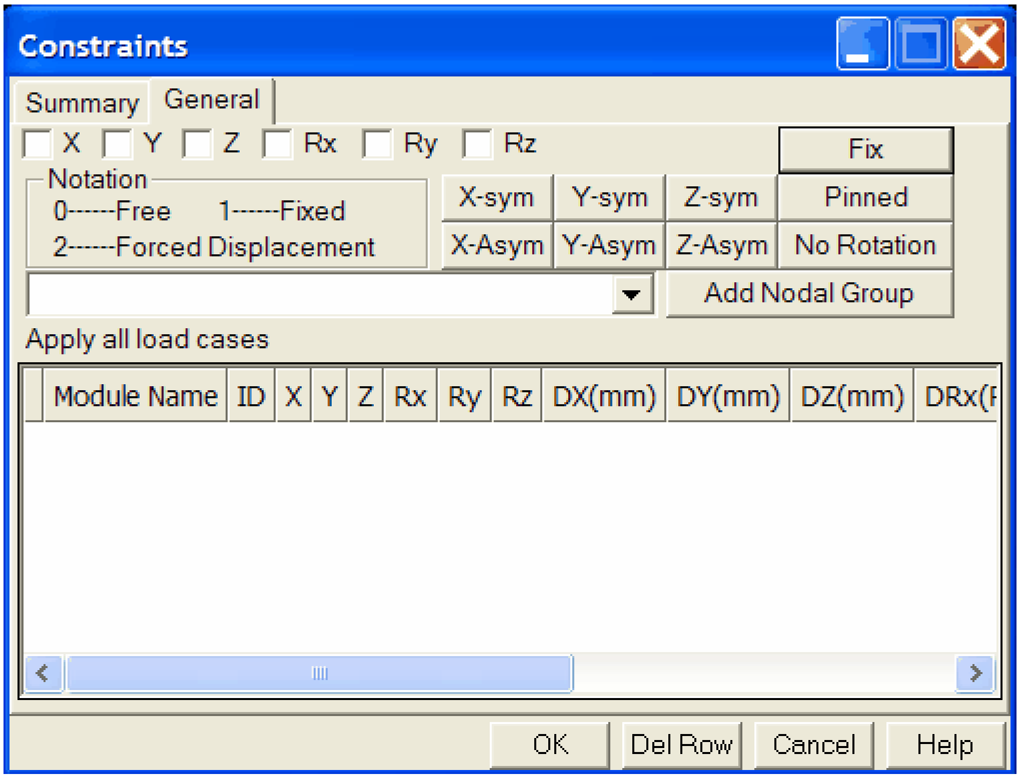
1. Begin by opening the restraints dialog by using the **Model > Define Restraints...** menu option, or from the toolbar.



The first tab in the dialog gives a summary of the restraints. The legend gives the graphical representation when the view restraints option is selected. This can be chosen from the **View > Restraints** menu option. The magnitude of the restraint marker can be adjusted in the View Options dialog.

If only a small portion of the ship has been modeled, the user can define boundary conditions at the vessels "ends" by selecting the First Module and Second Module. This is useful for preliminary evaluation of the ship's structure.

2. Click the General tab.



3. Click in the large white space and then select a node from the model to constrain.

4. The module name and ID will automatically populate.

5. The X, Y ,Z , Rx, Ry, and Rz boxes will be automatically filled with 0s meaning a free condition. Double click in the box to apply a fixed condition (1) or forced displacement (2) per the Notation key. The DX, DY, DZ, DRx, DRy, and DRz values specify the value of forced displacement if any.

6. Repeat steps 3-5 to add additional restraints.

7. Select OK to save.

Automatic Inertia Relief Restraints Method

The idea of the inertia relief restraint methodology is to automatically place a single restraint

© 2014 Optimum Structural Design, Inc.

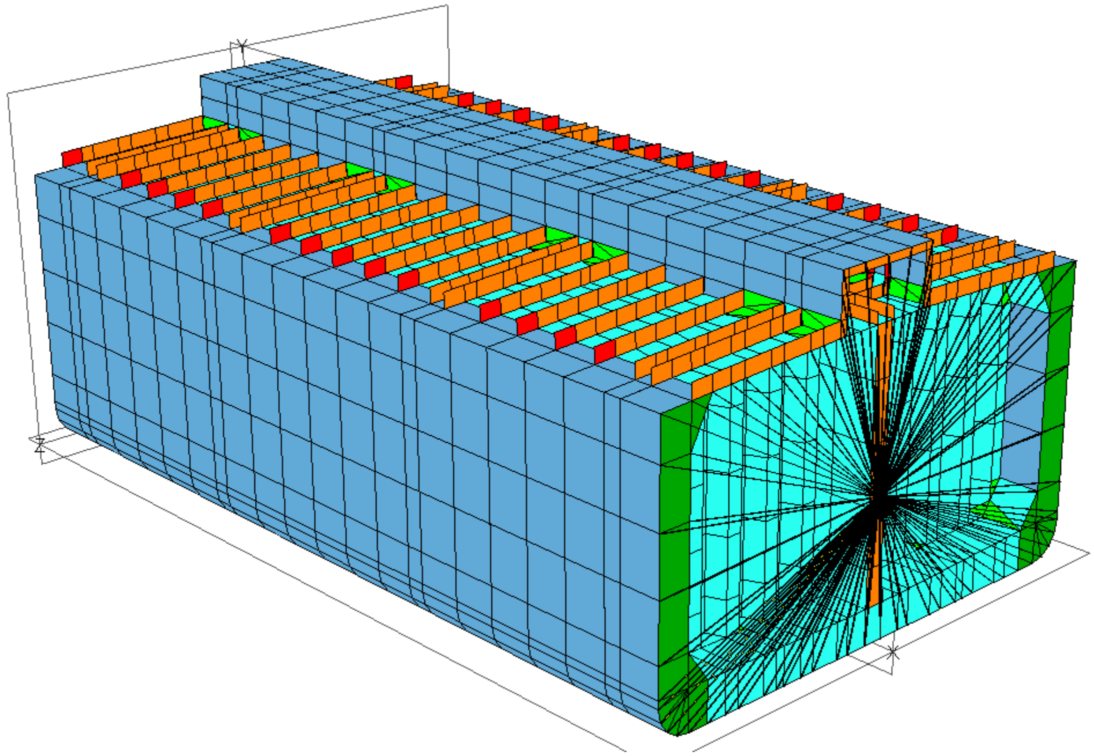
point at the model CG and connect to surrounding structure using soft springs. If the model is fully balanced, this single restraint is sufficient to prevent the model's rigid body motion. In order to fully balance a model in a quasi-static wave, the hydrostatic balance is first applied. However, since the hydrostatic based equilibrium can only balance the force in heave, and moments in roll and pitch, additional adjustment is need for forces in surge and sway, and moment in yaw. This adjustment is done by using “inertia relief” (i.e., to artificially impose very small accelerations such that the model is fully balanced in all 6 directions). This methodology applies to static loading as well as for models with seakeeping loads applied.

The procedure for using this restraint approach is:
1. Create load cases in the model. Do not apply traditional restraints.
2. Run the hydrostatic balance for all load cases.
3. Run the inertia relief balance for all load cases.
4. Solve the model.

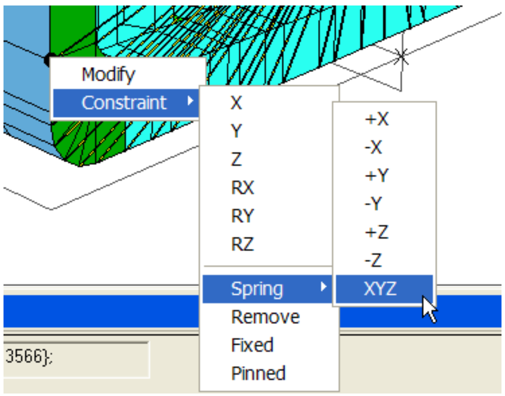
Defining Module End Restraints (Create Boundary Modules Command)

This feature enables the user to automatically constrain "end sections" within a cut model such that the end sections remain plane during bending (i.e., rotating about the section's neutral axis). The approach is well documented in class societies' literature (e.g., ABS, DNV, and Lloyd's). Specifically, "Appendix B: Structural Strength Assessment" of the Common Structural Rules for Double Hull Oil Tankers (IACS, July 2010) provides a detailed description of class requirements and the use of this approach (see 2.5.4 and 2.6).

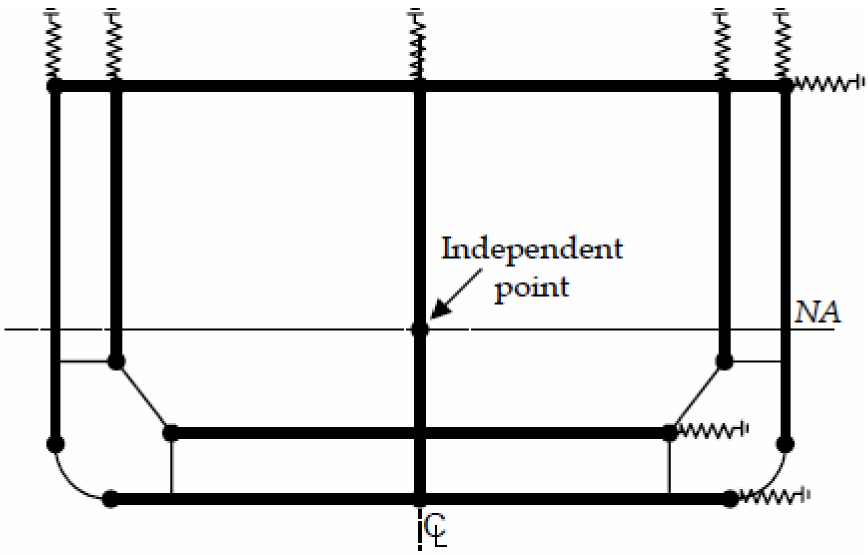
The Create Boundary Modules command can be found under the Model dropdown menu. The feature is to be used only with cut models. The command joins a master node located on the centerline at the height of the neutral axis to each slave node which is exposed at the end cross section. A rigid link, RBE2 element, joins each slave node to the master at each end of the cut model. The master node is located at an independent point where the vertical neutral axis and the horizontal neutral axis intersect. The results look like the following:



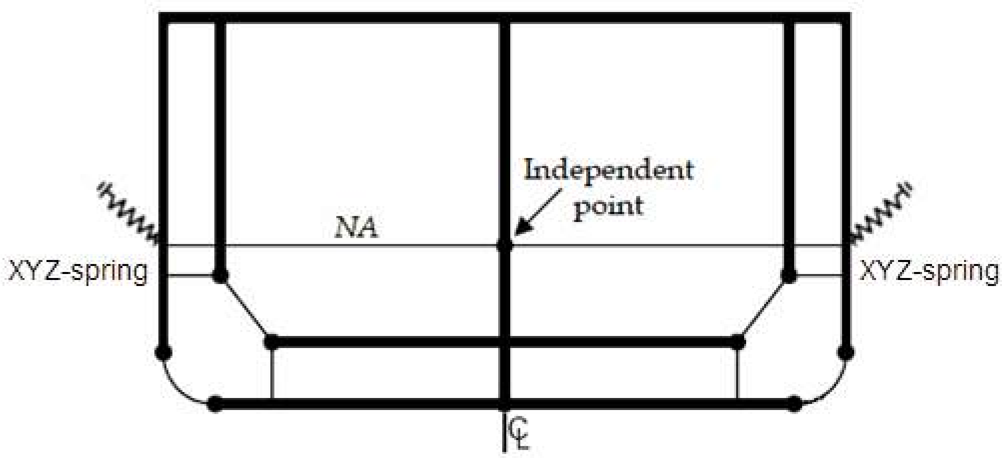
When applying restraints in this manner, RBE2 elements must be permitted to move at both ends. This method of applying the moment about an independent point on the neutral axis is effectively prescribing motion at all nodes on the end section (opposite end of the RBE2's than the independent node). Thus, a spring element must be used instead of a rigid restraint. These can easily be added by using the dynamic query for a node, right-click, select *Restraints*, and then select *Spring*. The right-click during node query menu looks like the following:



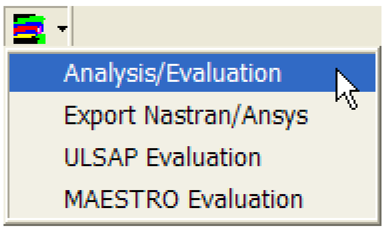
Section 2.6 of "Appendix B: Structural Strength Assessment" of the Common Structural Rules for Double Hull Oil Tankers (IACS, July 2010) requires boundary restraints at the deck with stiffness in the vertical direction and restraints at the bottom and inner bottom (stiffness in only the global transverse direction). The "free" end of the spring should be constrained in the 3 degrees of translation and the 3 degrees of rotation. The CSR required boundary constrains are summarized in the following schematic.



However, as will be shown below, an alternative constraining approach can yield very smiilar results (e.g., nodal displacements and stresses). Below is a diagram of alternative boundary restraints.

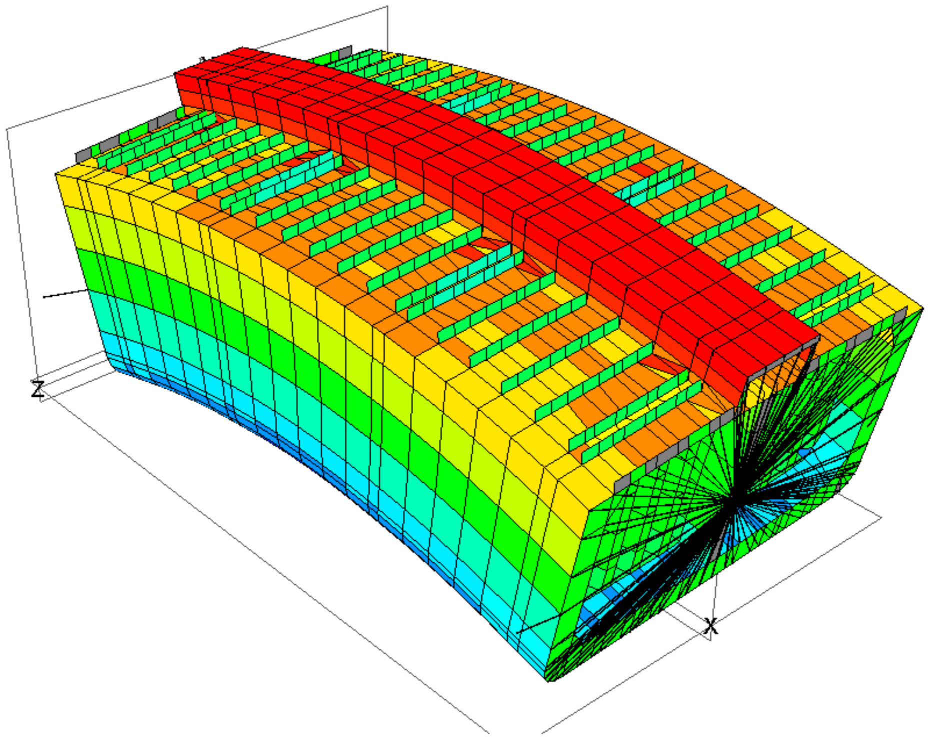


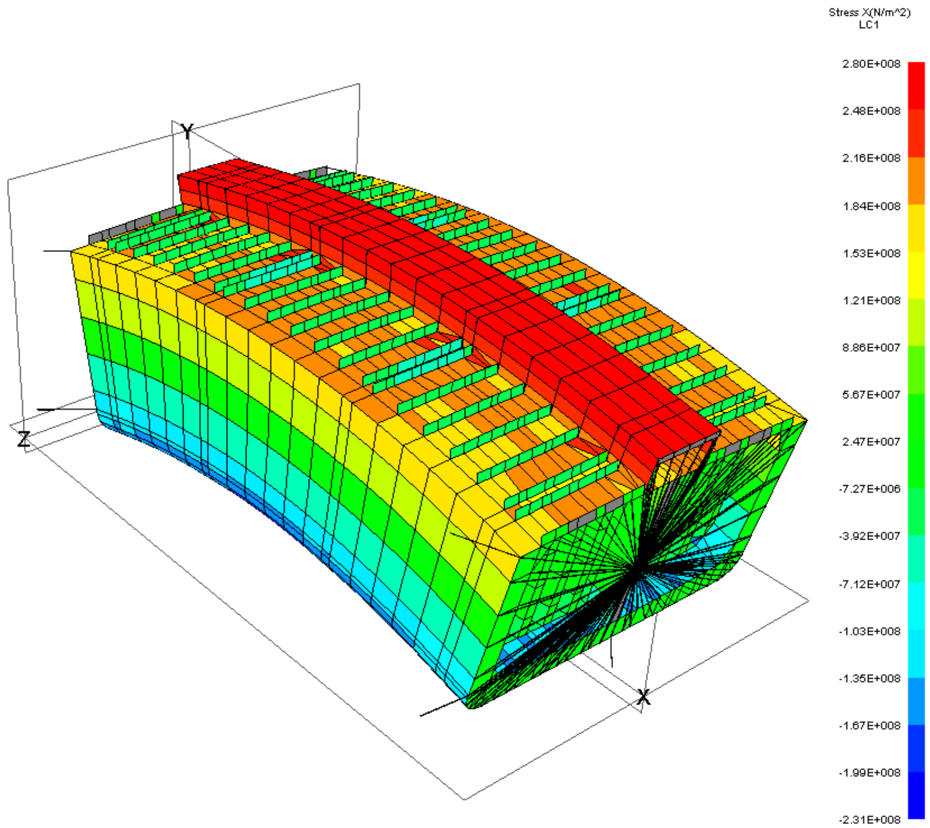
The next step is to load the model with an end moment and solve the finite element model through Analysis/Evaluation.



Note: If *boundary modules* were explicitly defined in the model, any end moment applied to the structure will be applied about the master node (independent point) of the *boundary module*. To apply the moment in the traditional sense (distributed across all nodes of the cross section), the *boundary modules* must be deleted.

Two solved FEA models are shown below to demonstrate the similarity in mid-plane normal stress in the x-direction between the two above restraint approaches. The solved model on the right is the result of the CSR restraints mentioned above. The solved model on the left is the result of the alternative restraint approach mentioned above.

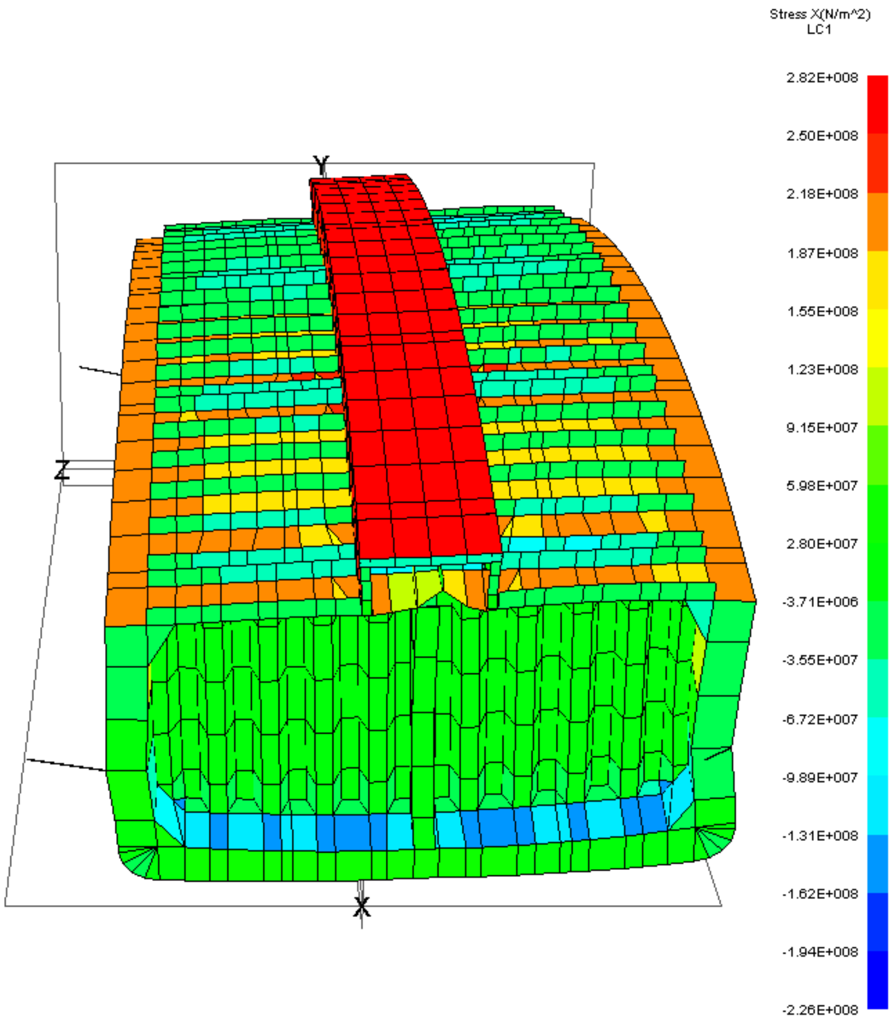


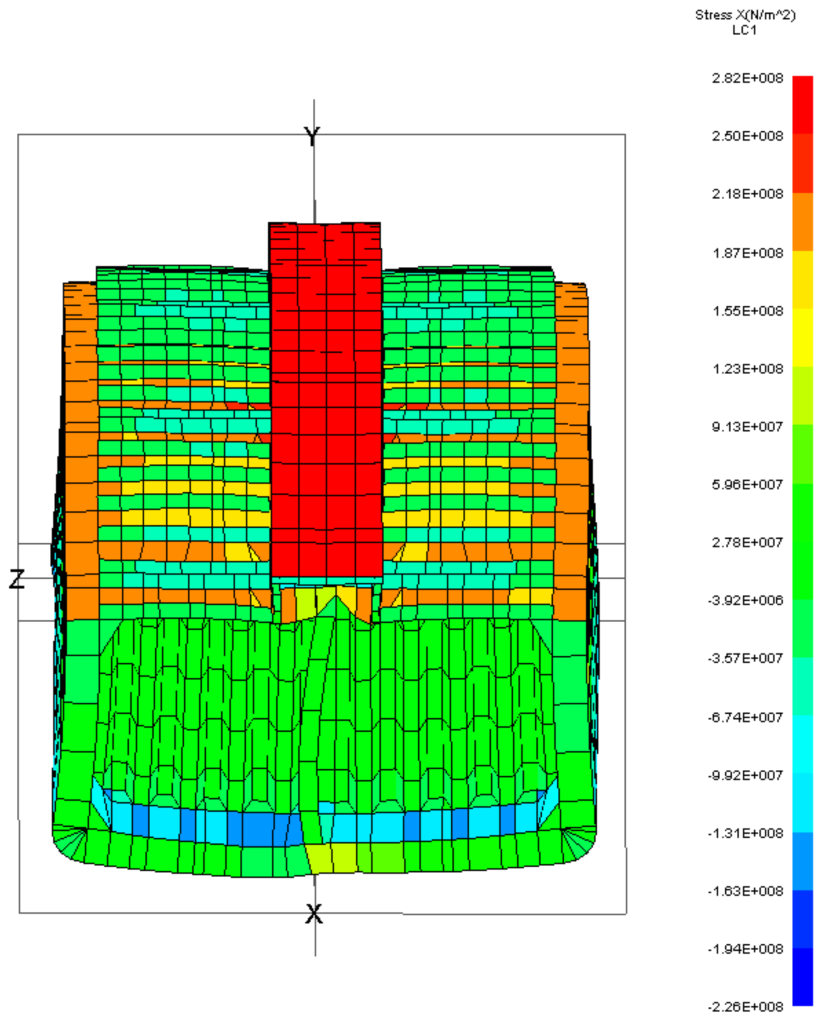


The mid-plane normal stress in the x-direction is virtually identical when comparing the above two models.

In addition to the restraint methods described above, examine what happens if an end moment is applied in the traditional sense, meaning that it is not applied to the master node (there isn't one) but is applied across all nodes on the end section. The solved FE models below show the affect of excluding the boundary module from the FEA. The solved model on the left is constrained with 4 springs at nodes near the horizontal neutral axis with stiffness in X, Y, and Z directions. Use of springs makes the model prone to translate in space when the nodal forces are distributed across the model. Noticeable out of plane deformations can be observed near the deck but the longitudinal bulkhead along centerline is not affected as much as in the below right hand model.

The model on the right is constrained with three rigid restraints at the aft end, two Y-direction translational restraints at nodes near the horizontal neutral axis and one Z-direction translational restraint at a node on the double bottom on centerline. The fore end (which is viewed below) of the cut model is fixed in translation in the X and Y directions at two nodes near the neutral axis. A third restraint is placed at a node on the double bottom on centerline; this node is fixed in Z translation. This method of restraint yields significant out of plane bending both on deck and along the longitudinal bulkhead. Although the resultant stresses are similar to the above analyses, this is not characteristic of the true state of deformation of a full ship. These four scenarios serve as four examples of how the structural problem could be solved.




Stress X(N/m^2)
LC1
2.82E+008
2.50E+008
2.18E+008
1.87E+008
1.55E+008
1.23E+008
9.13E+007
5.96E+007
2.78E+007
-3.92E+006
-3.57E+007
-6.74E+007
-9.92E+007
-1.31E+008
-1.63E+008
-1.94E+008
-2.26E+008
Y
Z
X

Checking The Model

6

6 Checking The Model

The topics in this section provide detailed information on the MAESTRO functionality used during the model checks stage of an analysis.

6.1 Model Integrity Checks

Quick Reference:

Wetted Elements Check
Element Pressure Side Check
Aspect Ratio Check
Internal Angle Check
Warped Quad Check
Overlapped Elements Check
Free Edges Check
Element Connectivity Check

Routine model integrity checks are a simple and time saving process. It can be beneficial to conduct integrity checks after each module is completed, instead of trying to check the entire model at once when it is completed.

The following tutorials show how to exercise several common model integrity checks.

Toolbar	N/A
Menu	**View > Wetted Elements**

1. This integrity check can be performed from the **View > Wetted Elements** menu option.

2. All elements in the current view not defined as wetted, or strakes not defined as side or bottom will disappear.

3. Elements improperly defined can be changed from wetted to not wetted, or vice versa by clicking the dynamic query icon, .

4. Highlight the element to change and right-click with the mouse and check or uncheck *Wet* accordingly.

5. Uncheck **View > Wetted Elements** from the menu to return to the element type view.

Toolbar	N/A
Menu	**View > Plate > Element Pressure Side**

1. This integrity check can be performed from the **View > Plate > Element Pressure Side** menu option.

2. All elements in the current view will be given a pink (pressure) side and gray (non-pressure) side.

3. This provides a quick graphical check on the hull pressure side as well as to verify a common convention was used for defining interior structures.

4. The dynamic query, , can be used to switch the pressure side of an element by right-clicking and selecting Normal Reverse.

5. Select **View > Element Type** from the menu to return to the element type view.

See Plate for additional details regarding Pressure Side conventions.

Toolbar	N/A
Menu	**Tools > Integrity Check > Aspect Ratio**

1. This integrity check can be performed from the **Tools > Integrity Check > Aspect Ratio** menu option.

2. A dialog box opens allowing the user to set the maximum allowable aspect ratio; the default value is 4.0.

3. If there are no elements exceeding this aspect ratio a dialog box will notify the user of this; otherwise the elements exceeding the aspect ratio will be identified with a color corresponding to the aspect ratio as defined in the legend on the right of the screen. Elements that do not exceed the aspect ratio will be gray.

The Output tab at the bottom of the screen will also list the elements exceeding the aspect ratio and their corresponding aspect ratio.

4. Select **View > Element Type** from the menu to return to the element type view.

Toolbar	N/A

© 2014 Optimum Structural Design, Inc.

Menu	**Tools > Integrity Check > Internal Angle**

1. This integrity check can be performed from the **Tools > Integrity Check > Internal Angle** menu option.

2. A dialog box opens allowing the user to set the maximum allowable internal angle; the default value is 150.0 degrees.

3. If there are no elements exceeding this internal angle a dialog box will notify the user of this; otherwise the elements exceeding the internal angle will be identified with a color corresponding to the internal angle as defined in the legend on the right of the screen. Elements that do not exceed the internal angle will be gray.

The Output tab at the bottom of the screen will also list the elements exceeding the internal angle and their corresponding internal angle.

4. Select **View > Element Type** from the menu to return to the element type view.

Toolbar	N/A
Menu	**Tools > Integrity Check > Warped Quad**

1. This integrity check can be performed from the **Tools > Integrity Check > Warped Quad** menu option.

2. A dialog box will open allowing the user to set the maximum allowable quad warping in degrees; the default is 20.0.

3. If there are no warped quads in the model, a dialog box will notify the user of this; otherwise the quad exceeding the warped angle will be identified with a color corresponding to the warped angle as defined in the legend on the right of the screen. All other elements will be gray.

4. Select **View > Element Type** from the menu to return to the element type view.

Toolbar	N/A
Menu	**Tools > Integrity Check > Overlapped Elements**

1. This integrity check can be performed from the **Tools > Integrity Check > Overlapped Elements** menu option.

© 2014 Optimum Structural Design, Inc.

2. If there are no overlapped elements in the model, a dialog box will open notifying the user of this; otherwise the overlapped elements will be grouped and shown on the screen.

3. A dialog box will open asking if the user would like to use the deletion dialog box to delete the overlapped elements.

It is recommended that the user investigate further to determine which is the true "overlapped element" before deleting.

4. Select **View > Element Type** from the menu to return to the element type view.

Toolbar	N/A
Menu	**View > Edges > Free Edges**

1. This integrity check can be performed from the **View > Edges > Free Edges** menu option.
2. A dialog box will open asking the user if they would like MAESTRO to create a group of the elements with free edges and if MAESTRO should check for free edge errors. If the user answers *Yes* to the first question, a general group will be created and displayed with its free edges in red on the screen. If the user answers *No*, the entire model will be displayed with its free edges in red.

It is easiest to see the free edges in wireframe mode , with stiffeners and end points turned off.
3. Select **View > Element Type** from the menu to return to element type view.

Toolbar	N/A
Menu	**Tools > Integrity Check > Element Connectivity**

1. This integrity check can be performed from the **Tools > Integrity Check > Element Connectivity** menu option.
2. A dialog box will open reporting the number of unconnected elements, if any.
3. If there are any unconnected elements, they will be listed in the Output tab at the bottom of the screen.

© 2014 Optimum Structural Design, Inc.

Importing and Exporting

7

7 Importing and Exporting

MAESTRO has the ability to import and export different data to support different aspects and types of analysis related to the ship design process.

7.1 Importing Models and Data

MAESTRO has the ability to import different types of data files to support different aspects of modeling and analysis as well as MAESTRO legacy data files. The following describes each of these types and provides the context in which a user would utilize them.

Geometry Import

MAESTRO (*.dat)

MAESTRO (*.mdl)

MAESTRO (*.mnf)

Nastran/FEMAP Neutral (.nas/.dat/.neu)

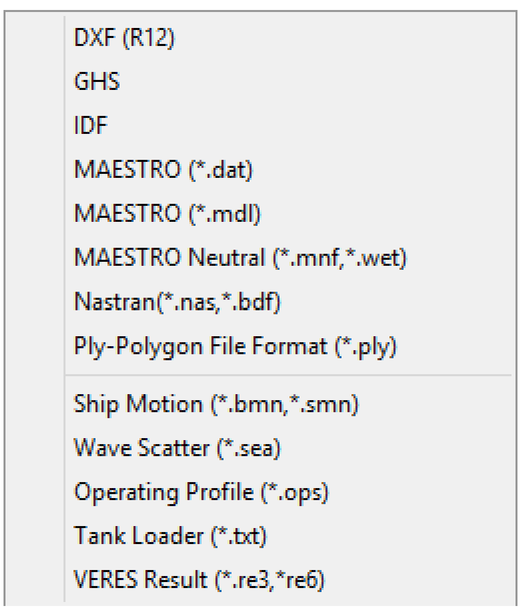
Ply-Polygon (*.ply)

Ship Motion (*.smn)

Wave Scatter (*.sea)

Operating Profile (*.opf)

Tank Loader (*.txt)



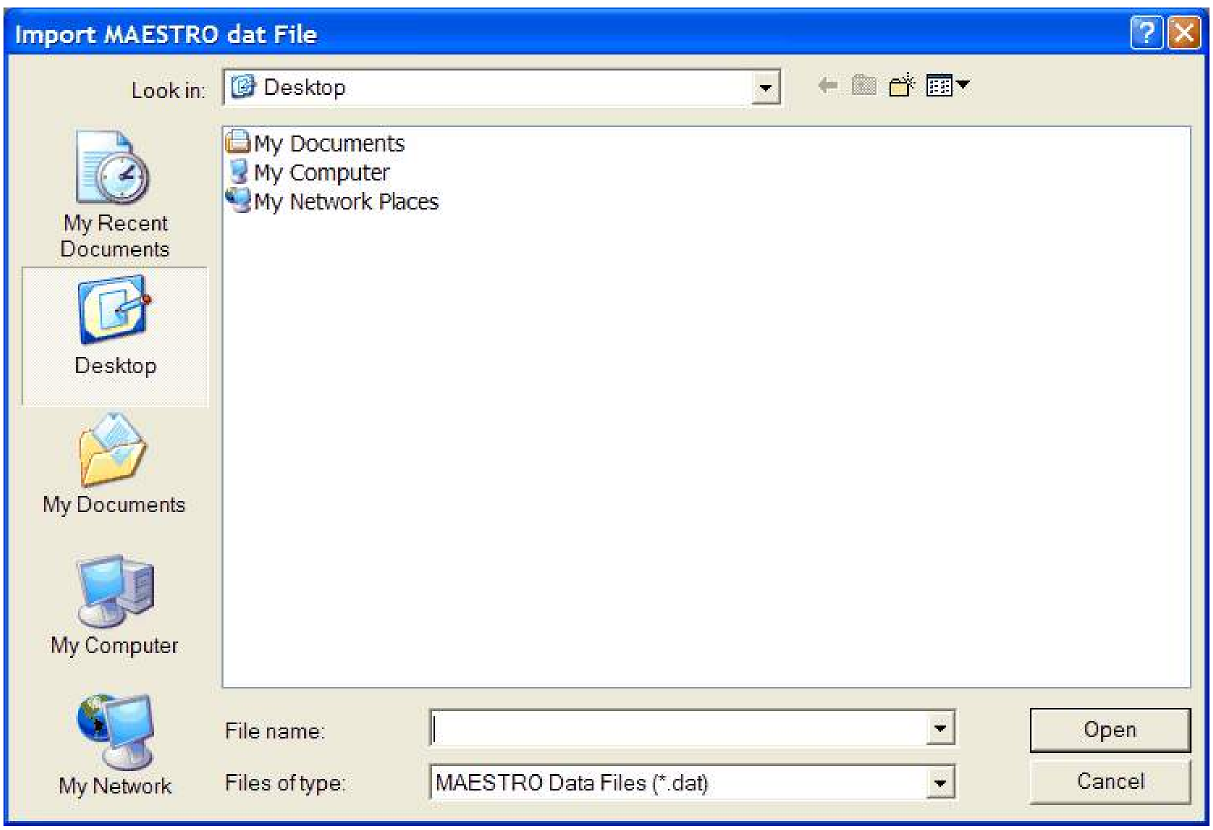
© 2014 Optimum Structural Design, Inc.

Geometry Import (DXF (R12)/GHS/IDF)

Curves can be imported into MAESTRO to serve as construction geometry to assist in model generation. The idea is to import curves at strategic locations, such as the Reference and Opposite ends of the Parts definition. This will allow the user to "snap" to these locations of interest using MAESTRO's construction geometry. See the Construction Geometry section for more information on MAESTRO's construction geometry. See Importing Geometry for a details of the DXF, GHS, and IDF import options.

MAESTRO (*.dat)

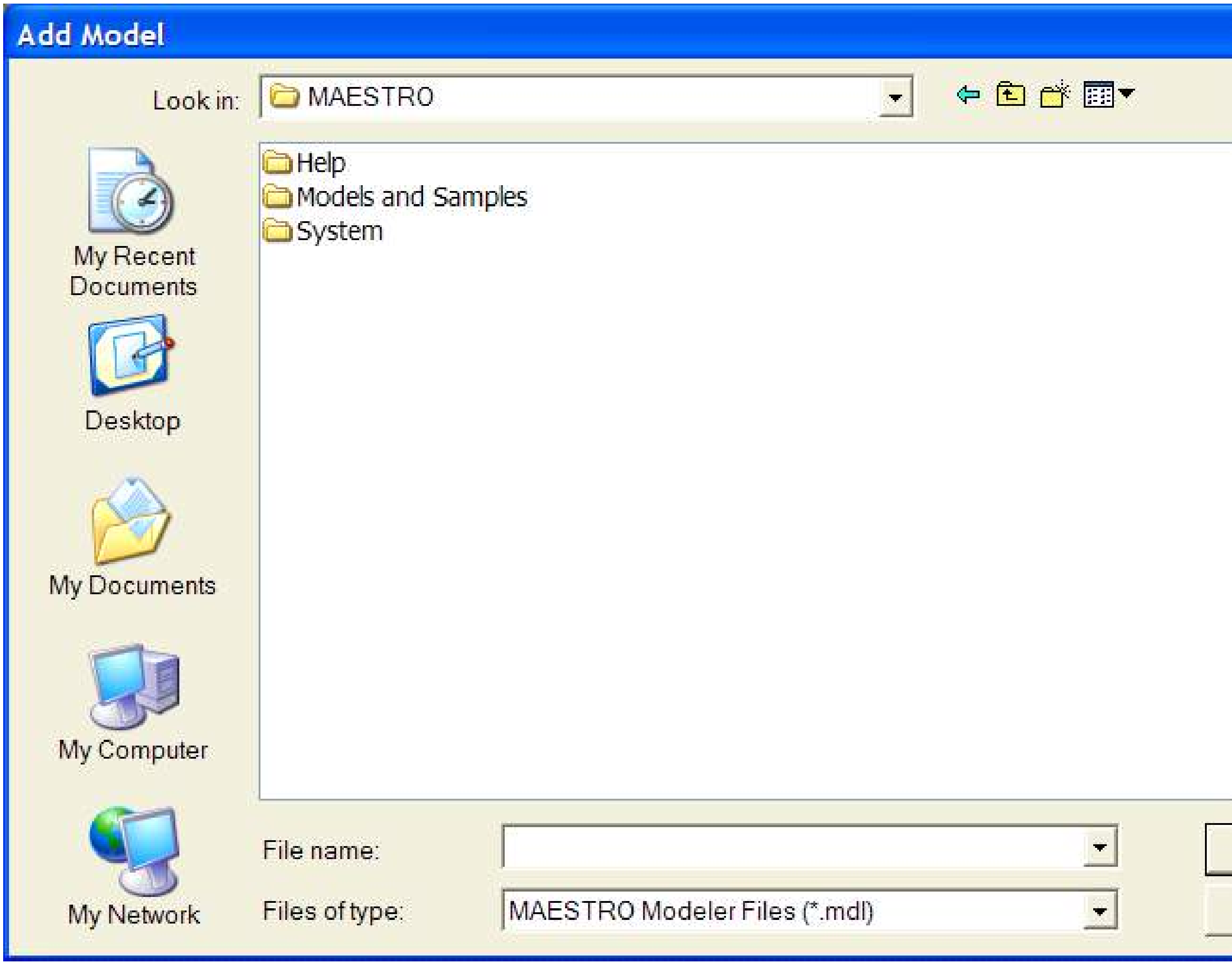
The Import > MAESTRO (*.dat) allows user to import a legacy data input file and converts it to an equivalent *.mdl file. For a complete description of the *.dat file, see the Legacy Data Preparation Manual.



MAESTRO (*.mdl)

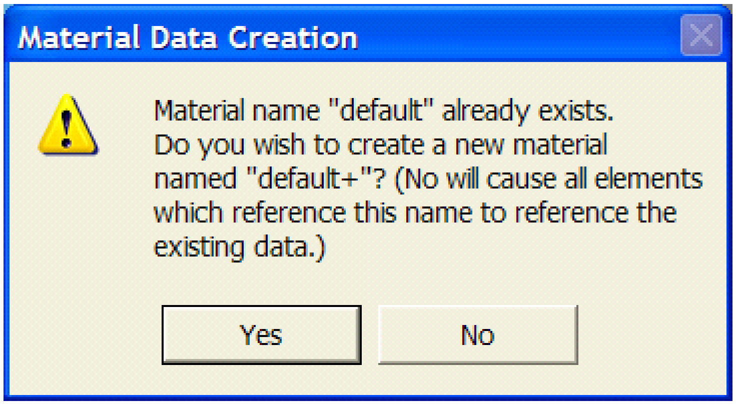
Two existing MAESTRO models can be combined into one using the Import > MAESTRO (*.mdl) menu option. This functionality can be useful when two different users are creating different parts of a model and want to combine them. In order to combine the two files, open one of the files and choose File > Import > MAESTRO (*.mdl) from the menu.

© 2014 Optimum Structural Design, Inc.



Choose the MAESTRO file to be imported and click Open.

If there are duplicates between the two models, separate dialogs will open and give the option to use existing properties and materials or create new ones.



MAESTRO Neutral File (*.mnf)

The MAESTRO Neutral File is used when interfacing with NAPA data. See the NAPA-MAESTRO Interface section for more details.

Nastran

MAESTRO has the ability to import a Nastran analysis model or FEMAP neutral file from File > Import > Nastran menu time. FEMAP nuetral files allows users to import group definitions from FEMAP into MAESTRO as general groups. On import MAESTRO persists the Nastran generated FE Tags and FEMAP generated property names..

MAESTRO imports Nastran fixed format small field of: MAT1, MAT2, MAT8, PBAR, PBARL, PBEAM, PCOMP, PSHELL, CTRIA3, CQUAD4, CBAR, CBEAM, CROD, CTRIAR and CQUADR.

MAESTRO translates from Nastran as follows:

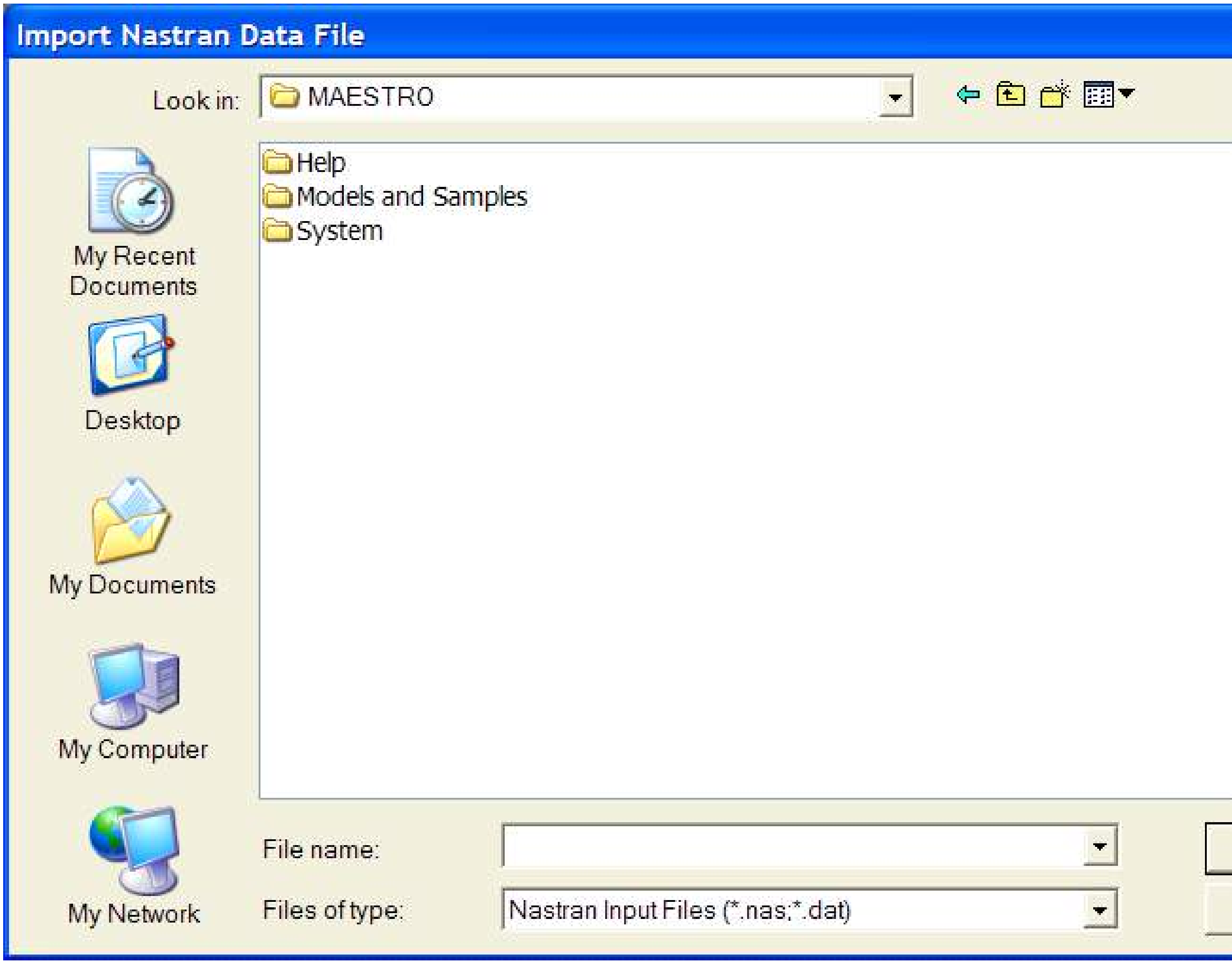
Quad-->CQUADR
Triangle and Compound Triangle ->CTRIR
Frame, Girder and Additional Beam ->CBAR
Rod->CROD
Spring->CELAS2
RSpline (2 node)->MPC
RSpline (1 node)->RBE2
Isotropic Material->MAT1
Composite Material->MAT8
Fixed Restraints and Enforced Displacement ->SPC, SPC1

Concentrated Mass--> CONM2

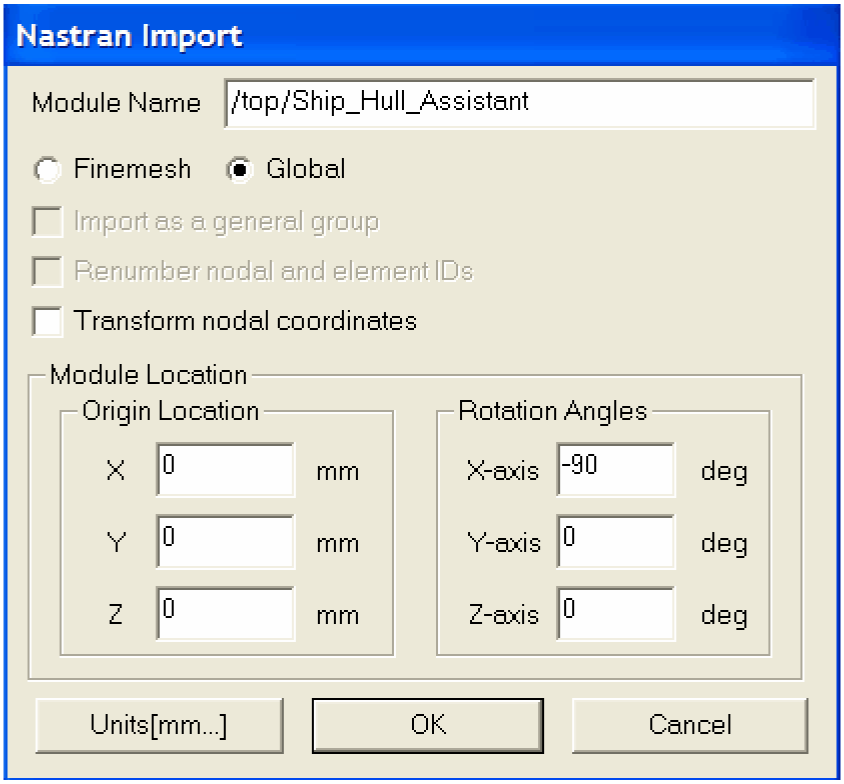
Forces--> FORCE

Coordinate Frames--> CORD2C, CORD2R, Cord2S, CORD1C, CORD1R, CORD1S

The user will be presented with the following dialog after selecting *Nastran* from the File > Import menu.



Select the *.nas or *.dat file to import and click Open. A Nastran Import dialog will open allowing the user to set the module location and units.



**When applying rotation to the imported model, the order the rotation is applied is roll (X-axis), pitch (Z-axis), yaw (Y-axis).

This option will import the finite elements only, and not the loads. The model can be synchronized to the global model using RSpline elements by right-clicking on the new module in the parts tree and selecting Synchronize. This will automatically place RSplines connecting the global model nodes with the imported Nastran model nodes.

Extract Module Command

To split the imported NASTRAN file into modules, use the Extract Module command under Model > Extract Module. The extract module command allows the user to split the module by defining upper and lower bounds for form a box that will be extracted. The extracted contents are placed within a new module under the "top" folder called "extract#".

Ply-Polygon (*.ply)

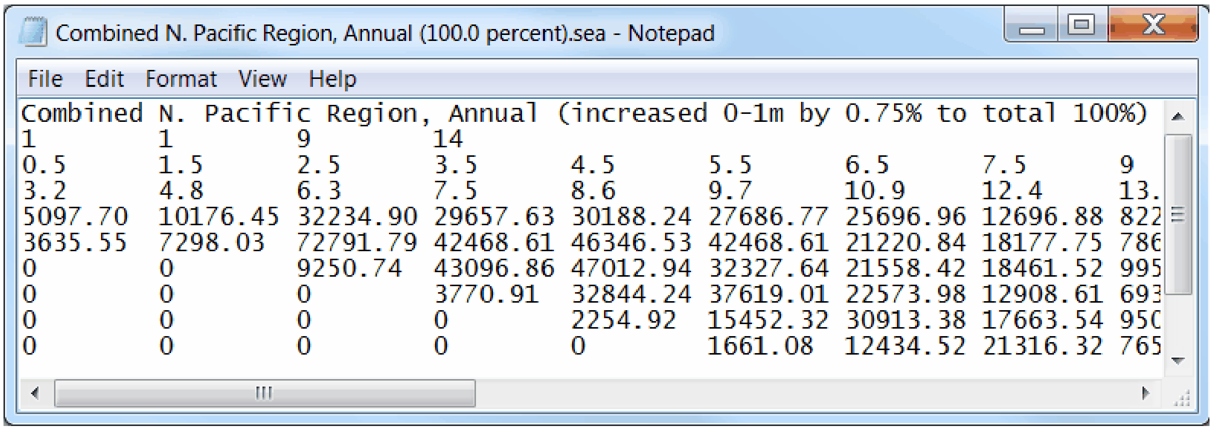
MAESTRO has the ability to import ply-polygon mesh geometry files (*.ply). This feature will create a new module of the quad and triangle elements in the mesh, all with the same plate and material property. See the Importing Geometry and Creating a Mesh In Rhino sections for more detail.

Ship Motion (*.smn)

See the Importing Hydrodynamic Loads section.

Wave Scatter (*.sea)

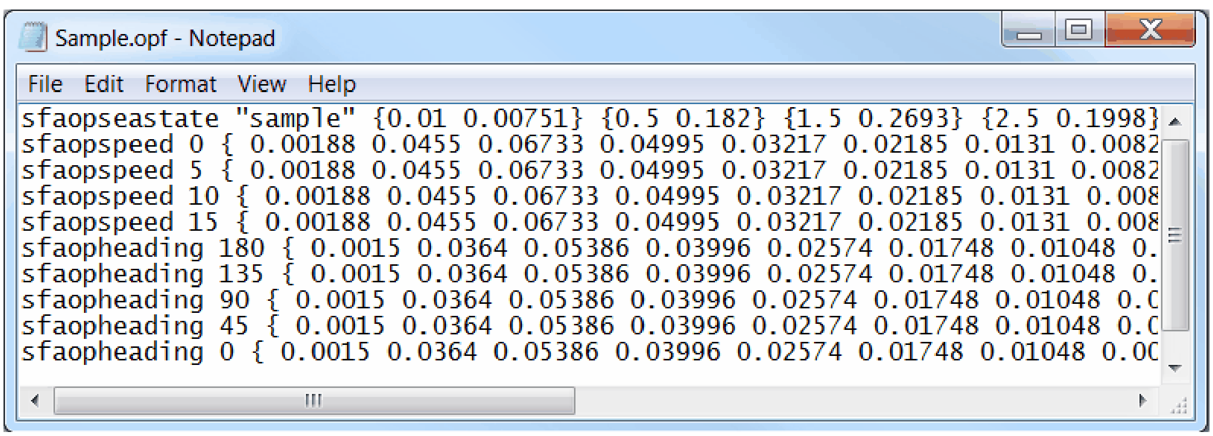
To support computations for the Extreme Load Analysis and Spectral Fatigue Analysis process, a user can import a wave scatter diagram using this option. A sample *.sea file can be found in the *Import Hydro Loads* directory in the *Models and Samples* folder.



```
Combined N. Pacific Region, Annual (increased 0-1m by 0.75% to total 100%)
1        1        9        14
0.5      1.5      2.5      3.5      4.5      5.5      6.5      7.5      9
3.2      4.8      6.3      7.5      8.6      9.7      10.9     12.4     13.
5097.70  10176.45 32234.90 29657.63 30188.24 27686.77 25696.96 12696.88 822
3635.55  7298.03  72791.79 42468.61 46346.53 42468.61 21220.84 18177.75 786
0        0        9250.74  43096.86 47012.94 32327.64 21558.42 18461.52 995
0        0        0        3770.91  32844.24 37619.01 22573.98 12908.61 693
0        0        0        0        2254.92  15452.32 30913.38 17663.54 950
0        0        0        0        0        1661.08  12434.52 21316.32 765
```

Operating Profile (*.opf)

To support computations for the Extreme Load Analysis and Spectral Fatigue Analysis process, a user can import an operating profile using this option. A sample *.opf file can be found in the *Import Hydro Loads* directory in the *Models and Samples* folder.

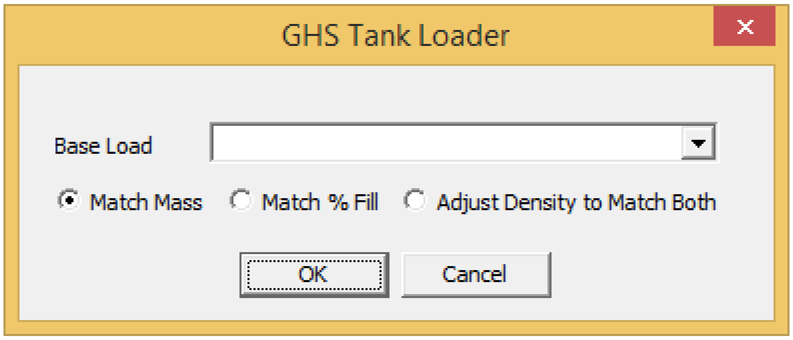


```
sfaopseastate "sample" {0.01 0.00751} {0.5 0.182} {1.5 0.2693} {2.5 0.1998}
sfaopspeed 0 { 0.00188 0.0455 0.06733 0.04995 0.03217 0.02185 0.0131 0.0082
sfaopspeed 5 { 0.00188 0.0455 0.06733 0.04995 0.03217 0.02185 0.0131 0.0082
sfaopspeed 10 { 0.00188 0.0455 0.06733 0.04995 0.03217 0.02185 0.0131 0.008
sfaopspeed 15 { 0.00188 0.0455 0.06733 0.04995 0.03217 0.02185 0.0131 0.008
sfaopheading 180 { 0.0015 0.0364 0.05386 0.03996 0.02574 0.01748 0.01048 0.
sfaopheading 135 { 0.0015 0.0364 0.05386 0.03996 0.02574 0.01748 0.01048 0.
sfaopheading 90 { 0.0015 0.0364 0.05386 0.03996 0.02574 0.01748 0.01048 0.0
sfaopheading 45 { 0.0015 0.0364 0.05386 0.03996 0.02574 0.01748 0.01048 0.0
sfaopheading 0 { 0.0015 0.0364 0.05386 0.03996 0.02574 0.01748 0.01048 0.00
```

Tank Loader (*.txt)

The tank loader allows a user to import a text file to automatically create one or more new load cases in MAESTRO with the tank load definitions.

© 2014 Optimum Structural Design, Inc.



The user can specify a base load case (e.g., lightship condition). Note: the tank names in the text file and MAESTRO must match. Also, if multiple load cases are imported in a single text file, the newly created MAESTRO load cases will all use the same base load case. In the case there are any slight differences in the tank geometry between the source file and MAESTRO, the user can select to match percent filled, match mass, or let MAESTRO automatically adjust the density to match the percent filled and mass. This will attempt to preserve the tank center of gravity assuming no major differences in tank geometry.

A sample of the text file input is below. "loadcase" is a keyword used to separate the different load cases in the file.

```
PART, LOAD, SPGR, WEIGHT
LOADCASE "abc1"
1I.P, 0.000, 1.025, 0.000
1I.S, 0.000, 1.025, 0.000
LOADCASE "abc2"
1I.P, 0.000, 1.025, 0.000
1I.S, 0.000, 1.025, 0.000
```

7.2 Exporting Models and Data

MAESTRO has the ability to export different types of data files to support different aspects of analysis processes. The following describes each of these types and provides the context in which a user would utilize them.

Ansys/Nastran

Wetted Cross Section

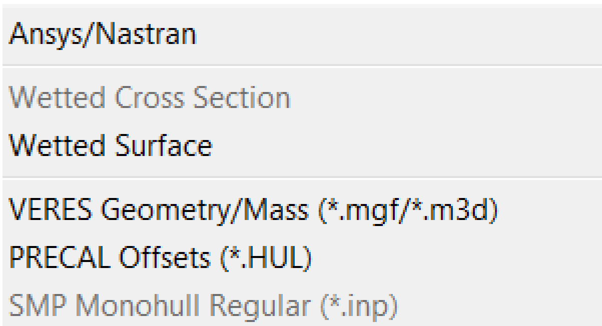
Wetted Surface

VERES

© 2014 Optimum Structural Design, Inc.

PRECAL

To export a model, select **File > Export** from the main menu and then the type of file to export.



Ansys/Nastran

This option will launch the Nastran/Ansys Export Options dialog.

To facilitate a way of effectively exporting the parts of the model of interest, please refer to the Create Module (from general group) command in the Groups Tree section.

MAESTRO export to Nastran as follows:

ELEMENTS & MATERIAL
Quad-->CQUADR
Triangle and Compound Triangle ->CTRIR
Frame, Girder and Additional Beam ->CBAR
Rod->CROD
Spring->CELAS2
RSpline (2 node)->MPC
RSpline (1 node)->RBE2
Isotropic Material->MAT1
Composite Material->MAT8
Fixed Restraints and Enforced Displacement ->SPC

LOADS

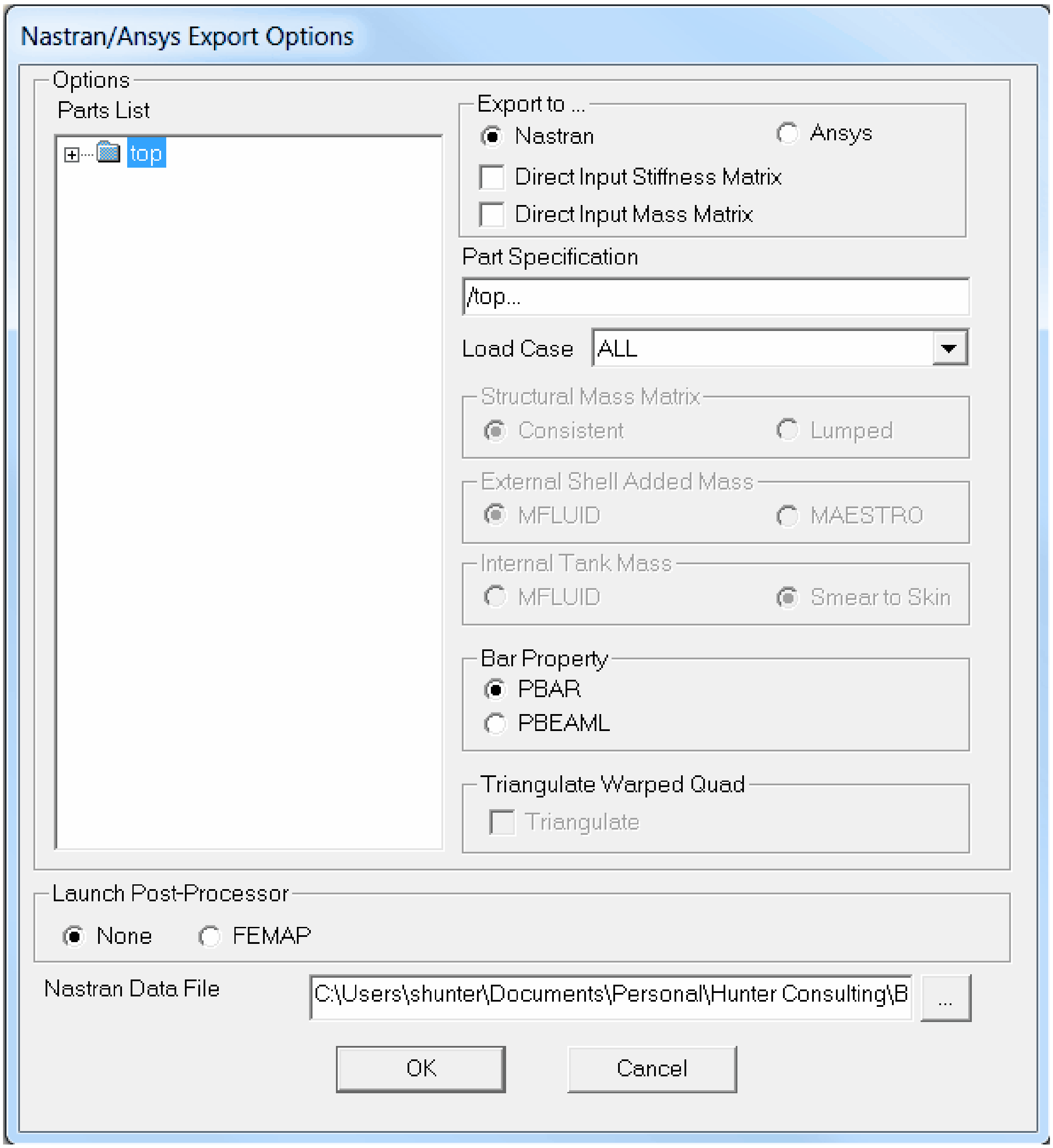
Stiffness Matrix

MAESTRO can output the stiffness matrix in Nastran format by clicking the box next to this option. A Setting button will appear which can be clicked and a dialog will open to change the material, pshell, and pbar properties.

Mass Matrix

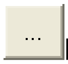
MAESTRO can output the mass matrix in Nastran format by clicking the box next to this option.

Static Analysis



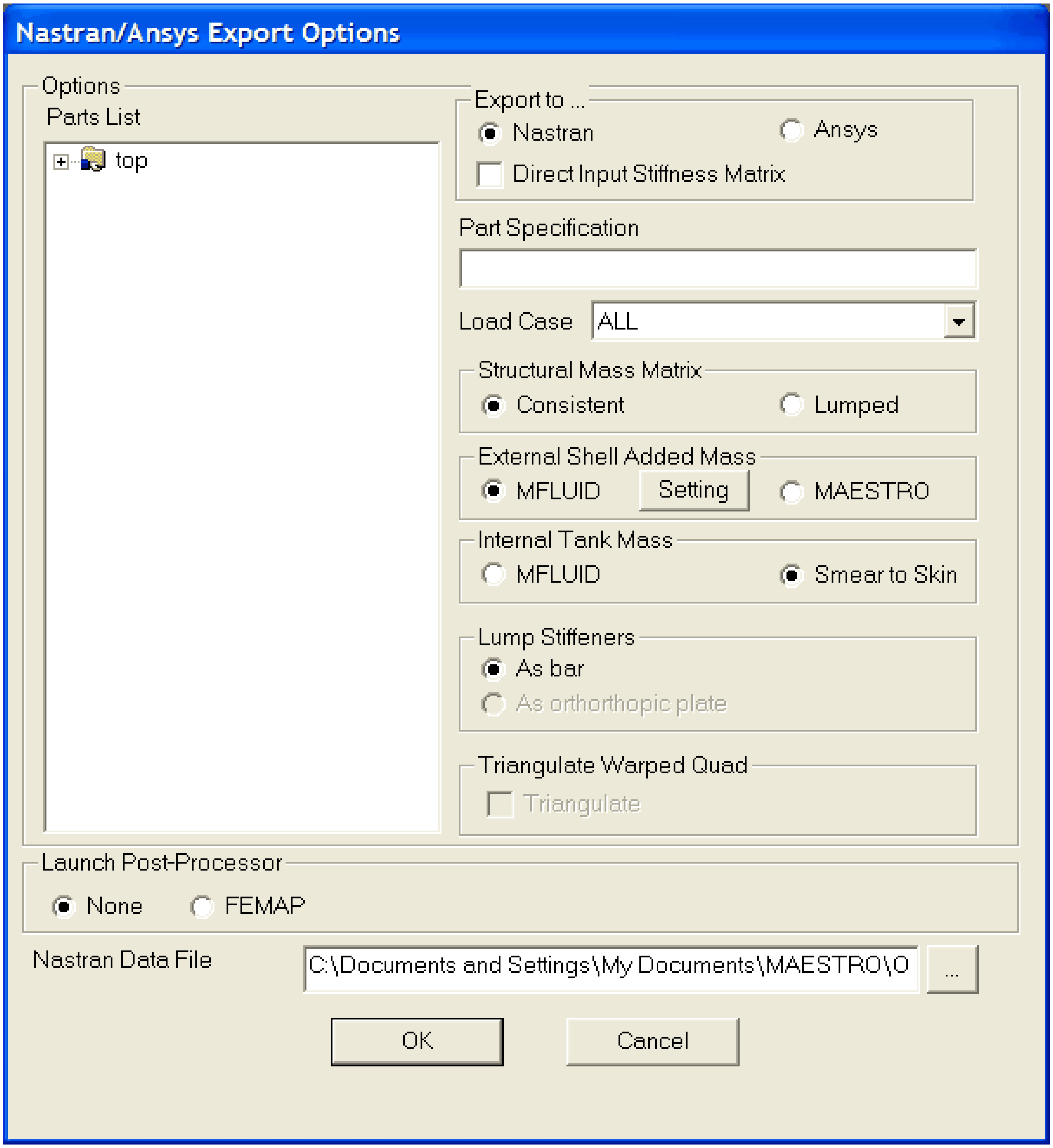
The Parts List allows the user to select the part(s) or group(s) to export to the Nastran export files. To export the entire model, select "top". Which parts or groups are to be exported is shown under "Part Specification" on the right of the dialog box.

FEMAP can be automatically launched once the file is created by clicking the radio button.

The Nastran export creates two files: a Nastran data deck file (*.nas), and a boundary only data (i.e. enforced displacement) file (*.nas.spcd). These files are automatically given the same name as the *.mdl file and saved in the same location, but can be changed by clicking the 

button.

Modal Analysis



The procedure for this type of analysis is the same as for a static analysis, except there are now more options available.

Structural Mass Matrix

There are two options for dealing with the structural mass matrix. MAESTRO can either export the densities and let Nastran calculate the mass, the Consistent option, or MAESTRO can lump the mass at the nodes and export these values, the Lumped option.

External Shell Added Mass

The two options for exporting the external shell added mass are to have MAESTRO write the MFLUID card to be read in Nastran, or MAESTRO can use it's own functionality to calculate the added mass and apply this mass to the nodes of the external shell elements. The Setting button is used to change the MFluid card settings; default values are used otherwise.

Internal Tank Mass

The two options for exporting the internal tank mass is have MAESTRO write the MFLUID card to be read in Nastran, or MAESTRO can lump the mass into the nodes defining the tank skin elements so that the mass and center of gravity of the fluid is matched.

Wetted Cross Section

This option will launch the Export Wetted Cross Section dialog which allows the user to select the location to save the *.wcs file. The newly created file will contain the section locations and wetted cross sectional areas as well as coordinates for the points defining the cross section based on the defined wetted elements in the model.

Wetted Surface

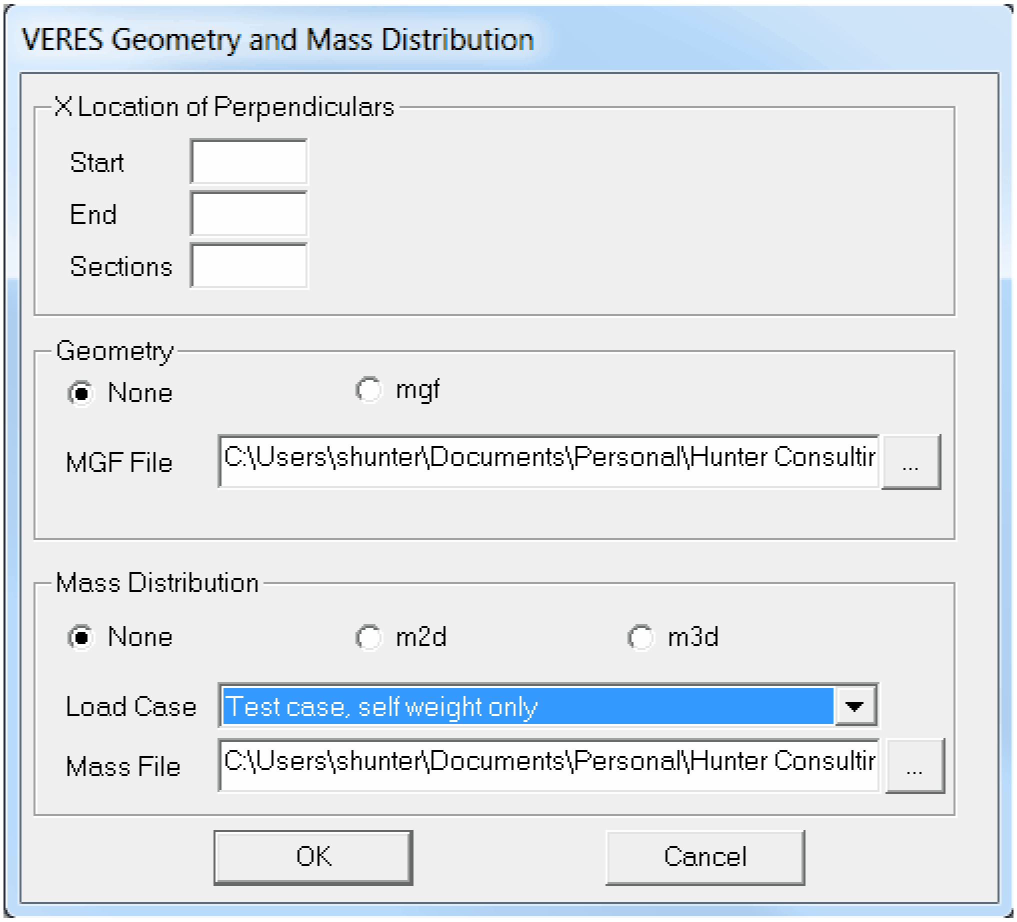
This option will launch the Export Wetted Surface dialog which allows the user to select the location to save the *.wet file. The newly created file will contain the node locations and elements for the elements defined as wetted in the model.

VERES

This option will generate a VERES *MGF, M3D,* or *M2D* file. The *Start* and *End* fields are the X (longitudinal) locations of the hydrodynamic model sections and the *Sections* field is the total number of sections wanted by the user. This is the basis for the *MGF* file, which is the VERES hydrodynamic sectional geometry.

The remaining options are are described below.

© 2014 Optimum Structural Design, Inc.



.M3D

The *.M3D file exports the model's nodal mass distribution. This option is only valid for models with less than 5,000 nodes.

.M2D

The *.M2D file exports the model's sectional mass distribution. This option is intended for larger models with more than 5,000 nodes.

PRECAL

This option will generate a PRECAL *HUL, HIN, CND*, or *INP* file. The following are brief descriptions of the PRECAL file:

.HUL

The *.HUL file exports the model's geometry.

.HIN

The *.HIN file exports the mass properties of the model.

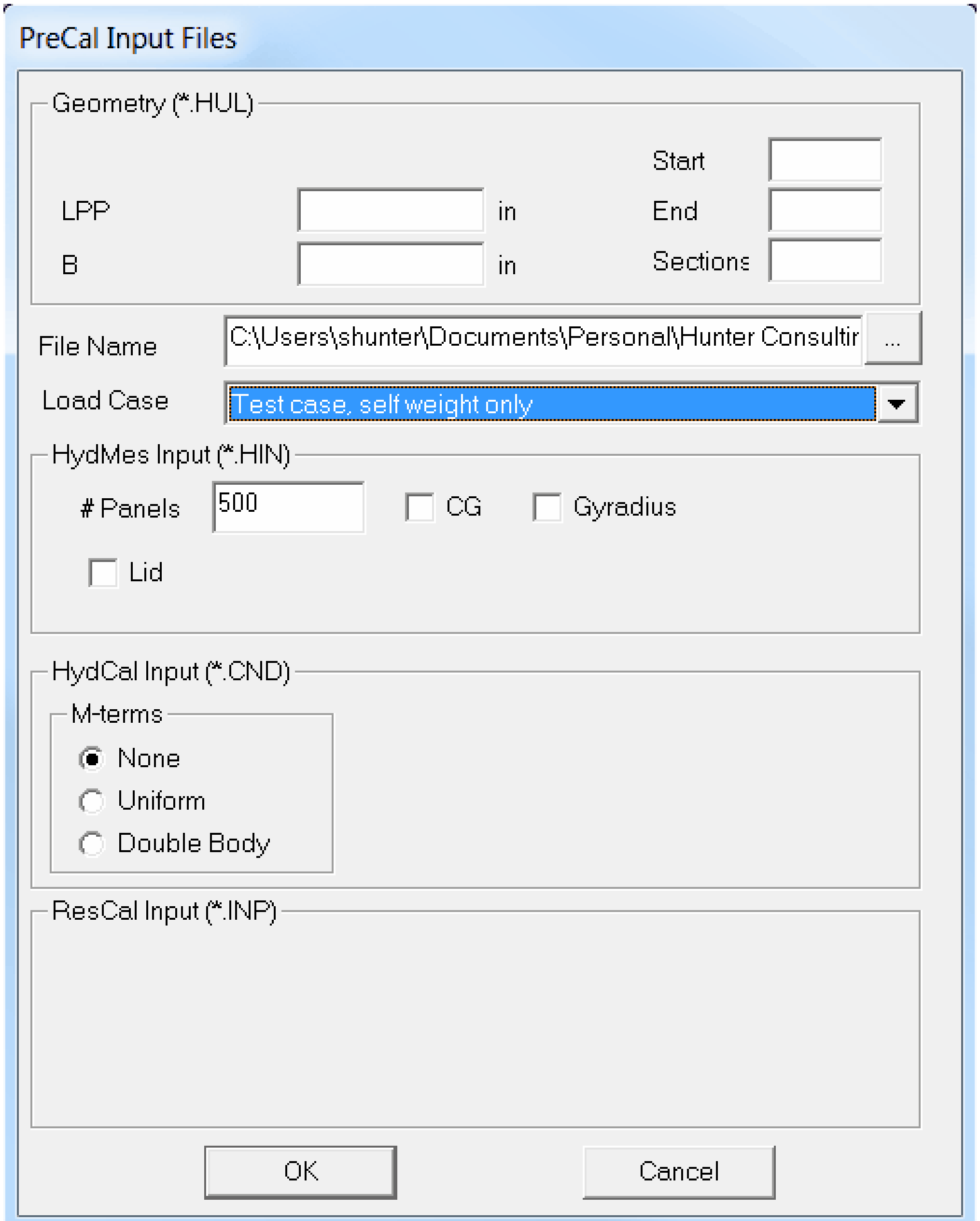
.CND

The *.CND file exports the conditions (i.e., speed and frequency) for the model.

.INP

The *.INP file exports the user specified options to calculate the vessel motions and wave loads.

© 2014 Optimum Structural Design, Inc.


PreCal Input Files
Geometry (*.HUL)
Start
LPP
in
End
B
in
Sections
File Name
C:\Users\shunter\Documents\Personal\Hunter Consultir
...
Load Case
Test case, self weight only
HydMes Input (*.HIN)
Panels
500
CG
Gyradius
Lid
HydCal Input (*.CND)
M-terms
None
Uniform
Double Body
ResCal Input (*.INP)
OK
Cancel

NAPA-MAESTRO Interface

8

8 NAPA-MAESTRO Interface

Table of Contents

8.1 Introduction

The following documentation 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.

8.2 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.

8.3 Level of Effort Comparison for Two Different Approaches

The following section is organized as follows:

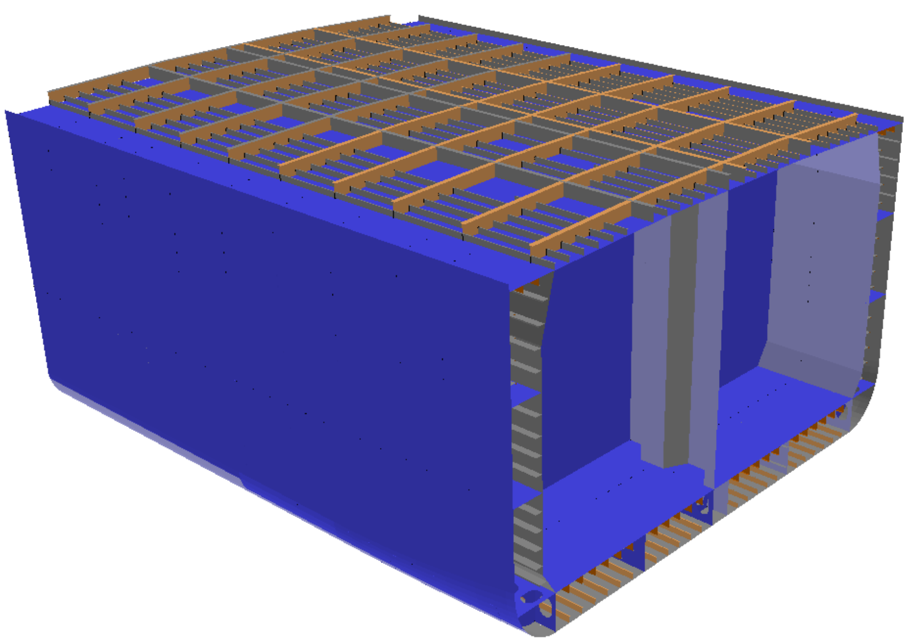
Sample Case Description

Comparison Details

Comparison Conclusions

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 tanker cargo area, which is shown in below. Additional input was established to describe a normal scope of work (SOW) for this type of vessel and analysis. There were no specific Classification requirements established, although it was assumed that reports would be generated for submittal to review authorities. Further, there was no specification for computing and imposing hydrodynamic loads or performing fatigue analysis. The table below provides the assumed SOW for this comparison.



© 2014 Optimum Structural Design, Inc.

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

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 NAPA/MAESTRO Interface (NMI). The percentages are provided below and represent the savings for using the NMI during the course of the listed activity.

© 2014 Optimum Structural Design, Inc.

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**

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 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.

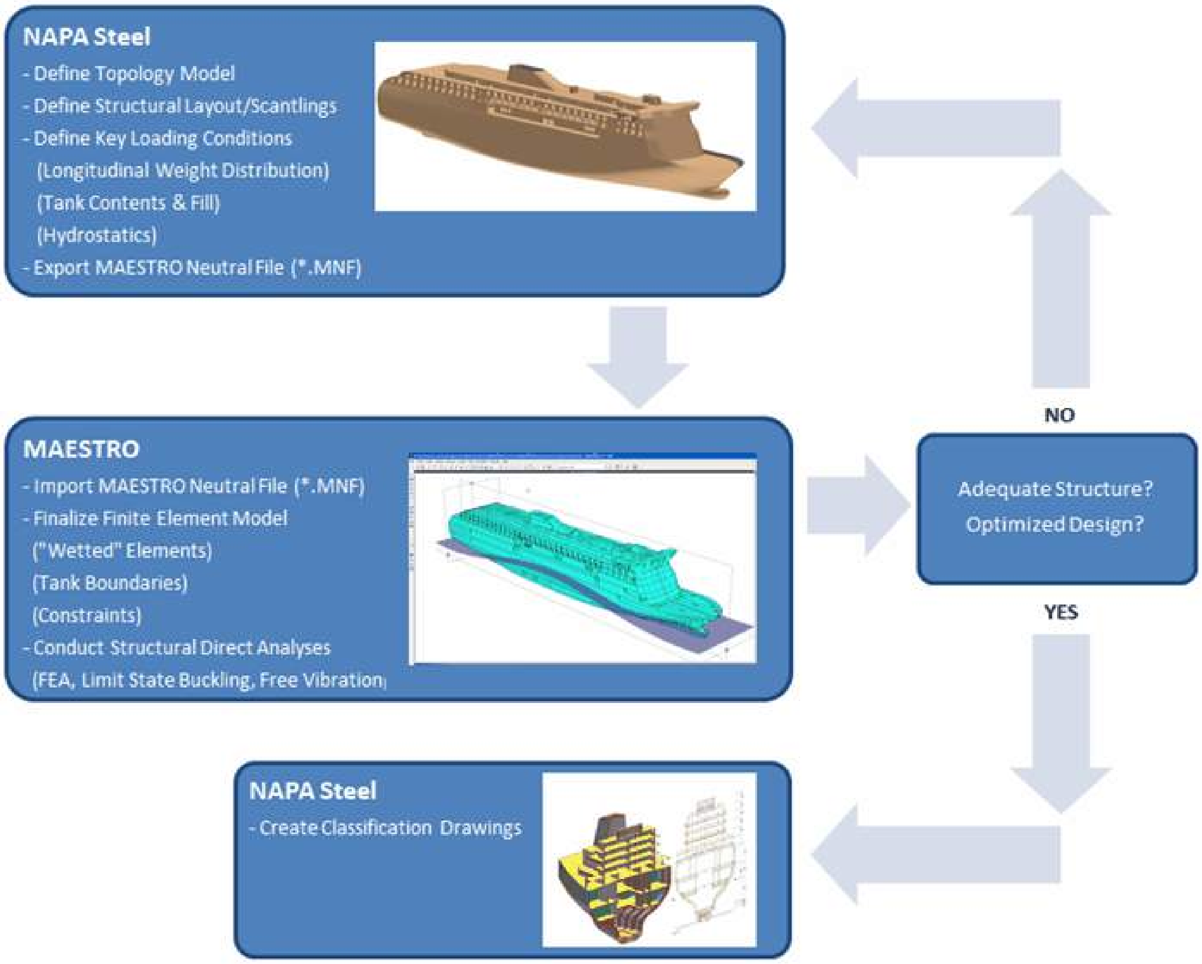
8.4 Using the NAPA/MAESTRO Interface for Structural Design

The following section is organized as follows:

The MAESTRO Neutral File

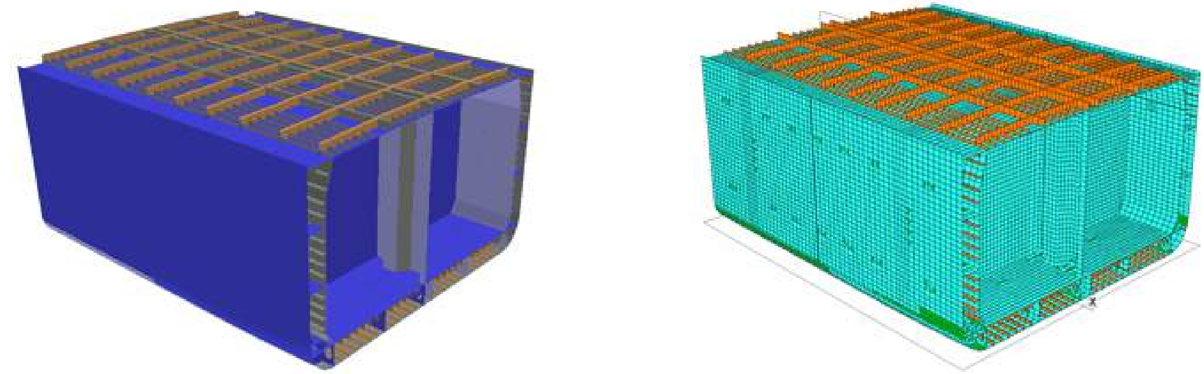
The figure below depicts the workflow for using the NAPA/MAESTRO Interface (NMI) in a ship structural design process. At the core of the interface is the MAESTRO Neutral File, which contains the NAPA generated data that is pertinent for creating and analyzing the MAESTRO finite element model. Napa and 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 have also implemented a translator for pertinent loading information. The loading data includes: 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.

© 2014 Optimum Structural Design, Inc.



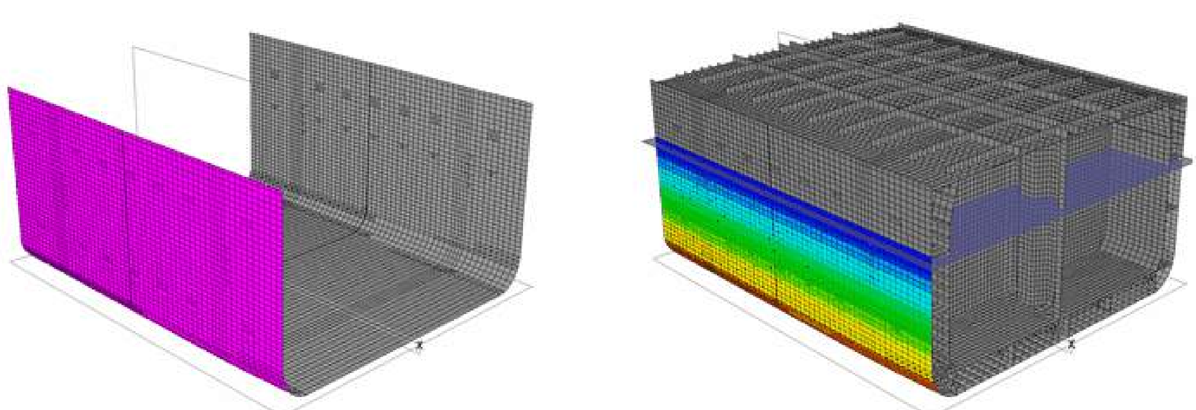
Finalizing the MAESTRO Finite Element Model

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 the figure above. 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. The first figure below shows the imported NAPA models. The second figure below 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. Similar integrity checks can be run to verify that tank boundaries and pressure normals are properly defined, which is shown in the third figure below.

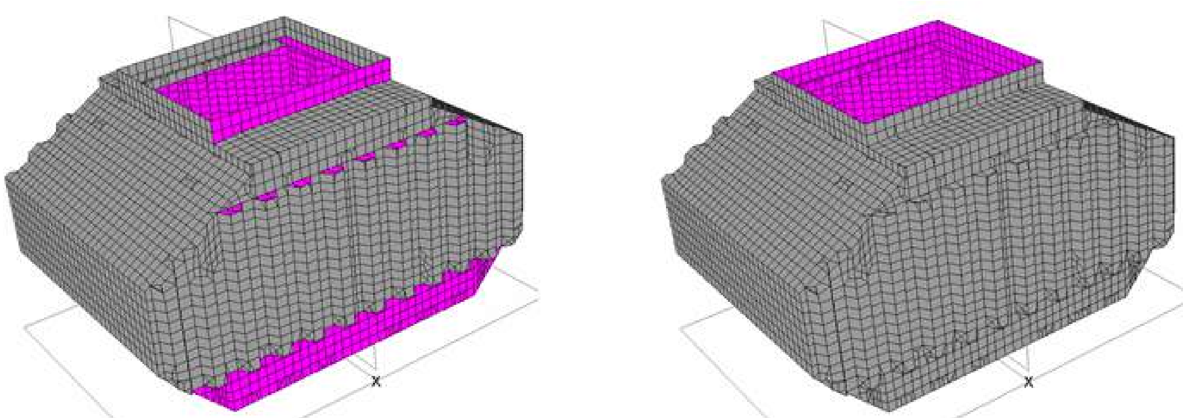


(a) NAPA Steel Model (b) MAESTRO FEM

Imported FEM from NAPA Steel to MAESTRO



MAESTRO "wetted" Elements for Hydrostatic Loading



Tank Boundary Definition and Creating Consistent Normal Definition

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.

Review the following sections for details of MAESTRO structural assessment capabilities:

Finite Element Analysis

Limit State Buckling Analysis

Fine Mesh Analysis

Free Vibration Analysis

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.

Production 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.

8.5 Interface Development Priorities

The following sections describe the envisioned workflow and current capability of the NAPA/MAESTRO Interface (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.

© 2014 Optimum Structural Design, Inc.

Loading The Model

9

9 Loading The Model

The topics in this section provide detailed information on the MAESTRO functionality used while loading the model for an analysis. It describes the different methods for loading the model as well as options to graphically view the loads applied.

9.1 Groups Tree

The groups tree displays each group which has been created in the model, in a folder format directory. Each group appears as its own directory, and cannot be a subdirectory of another group. However, individual groups within a folder may be organized into subfolders.The following group types are all accessible from the Groups menu. The following groups each have a tab in the groups dialogue as well.

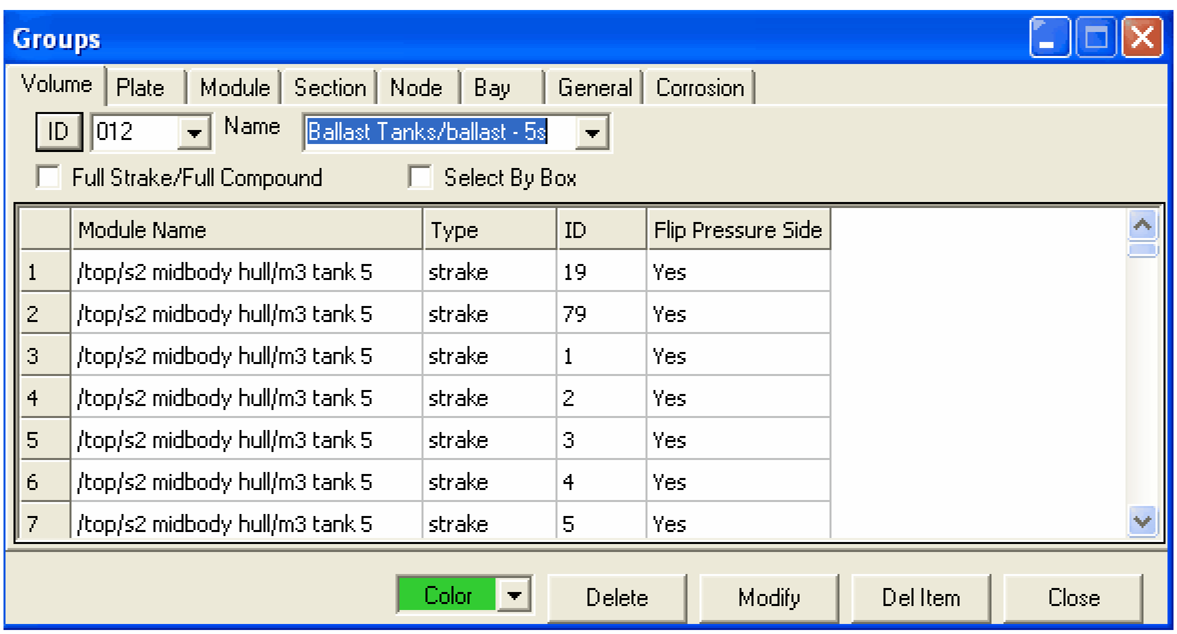
General Groups

Volume Groups

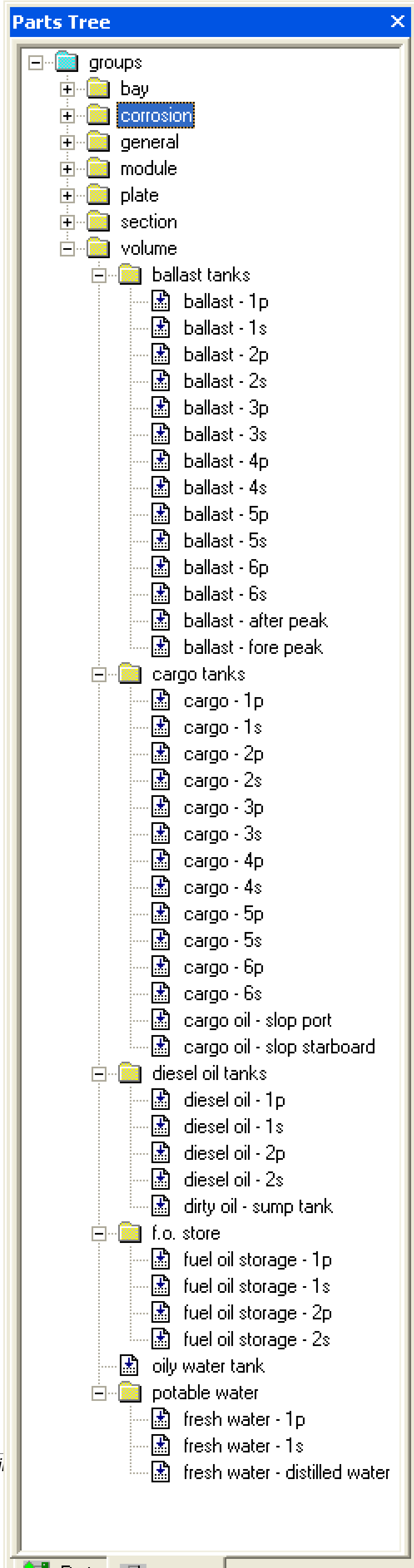
Plate Groups

Nodal Groups, Module Groups, Section Groups, Bay Groups, and Corrosion Groups

To create a subfolder, select the appropriate group ID or Name, type in the subfolder name before the group name followed by a forward slash, and click modify. Bring similar tanks into the subfolder by dragging and dropping into the folder.



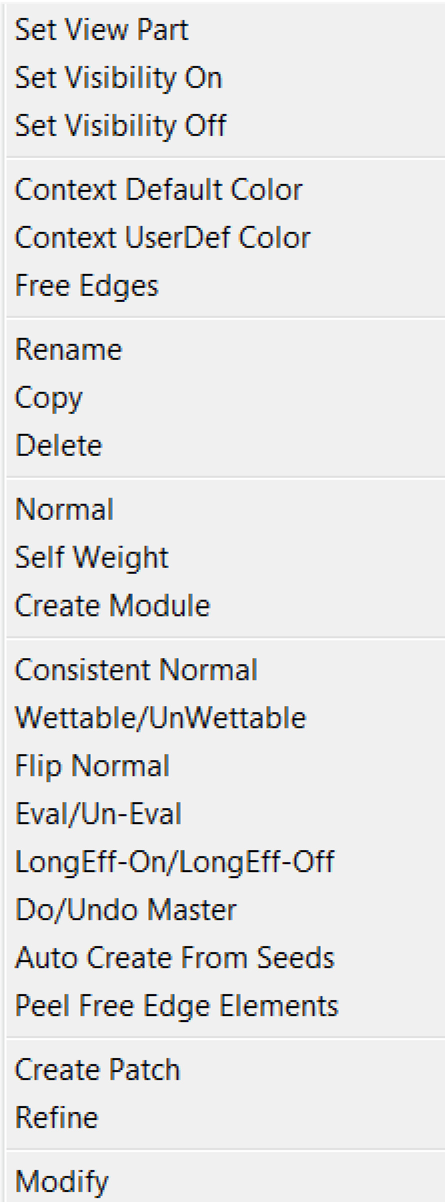
© 2014 Optimum Structural Design, Inc.


Parts Tree
groups
bay
corrosion
general
module
plate
section
volume
ballast tanks
ballast - 1p
ballast - 1s
ballast - 2p
ballast - 2s
ballast - 3p
ballast - 3s
ballast - 4p
ballast - 4s
ballast - 5p
ballast - 5s
ballast - 6p
ballast - 6s
ballast - after peak
ballast - fore peak
cargo tanks
cargo - 1p
cargo - 1s
cargo - 2p
cargo - 2s
cargo - 3p
cargo - 3s
cargo - 4p
cargo - 4s
cargo - 5p
cargo - 5s
cargo - 6p
cargo - 6s
cargo oil - slop port
cargo oil - slop starboard
diesel oil tanks
diesel oil - 1p
diesel oil - 1s
diesel oil - 2p
diesel oil - 2s
dirty oil - sump tank
f.o. store
fuel oil storage - 1p
fuel oil storage - 1s
fuel oil storage - 2p
fuel oil storage - 2s
oily water tank
potable water
fresh water - 1p
fresh water - 1s
fresh water - distilled water

© 2014 Opti

Any group can be copied, deleted or renamed in the group window, using the popup menu launched by right-clicking the mouse.

The General Groups right-click menu is:



Set View Part

This will set the highlighted group as the current view.

Set Visibility On

This will toggle the visibility on for the highlighted group.

Set Visibility Off

This will toggle the visibility off for the highlighted group.

Context Default Color

This will display the highlighted group with the default MAESTRO element colors in the context of the entire model; the un-highlighted groups will be represented in a wire-frame view.

Context UserDef Color

This will display the highlighted group with the user-defined colors selected in the Groups dialog. The highlighted group will be displayed in the context of the entire model, where the un-highlighted groups will be represented in a wire-frame view.

Free Edges

This will check the highlighted group for free edges.

Rename

This will allow the group to be renamed within the groups tree.

Copy

This will create a copy of the highlighted group.

Delete

This will delete the highlighted group.

Normal

This will display the highlighted group with the normal side of elements colored.

Create Module

Right-clicking general groups and selecting "Create Module" will split the module from current module and place it within a new module under a folder named "general". "General" will be located under the "top" directory and within it will be the group, with parts split into its

© 2014 Optimum Structural Design, Inc.

own module titled "extract#".

Consistent Normal

Please see the Defining Consistent Normals section.

Wettable/Unwettable

This will toggle the plate elements of the highlighted general group as wet or unwet.

Flip Normal

This will flip the normal direction for the elements of the highlighted general group.

Eval/Un-Eval

This will flag the highlighted general group for allowing/ignoring it during limit state evaluation.

Do/Undo Master

This will make the highlighted general group's elements master elements so they do not show up when you display the coarse and fine mesh models at the same time. When you view all modules, the coarse elements used to create the fine mesh model will not be displayed overlapping the new fine mesh elements.

Auto Create From Seeds

Please see the Creating Groups from Seed Elements section.

Peel Free Edge Elements

This functionality supports the creation of closed group (e.g., volume groups, plate groups, etc.). Once this function is executed on a particular general group, MAESTRO will search the elements within this group and eliminate (or *peel)* elements that have a free edge. The user can use this function in combination with *dragging* and *dropping* the general group into a plate group to eliminate unwanted beam elements. Ultimately the user's goal is to be left with a closed group without any free edges, which can then be converted to a volume group for loading.

Create Patch

This will create an evaluation patch using the elements of the highlighted general group.

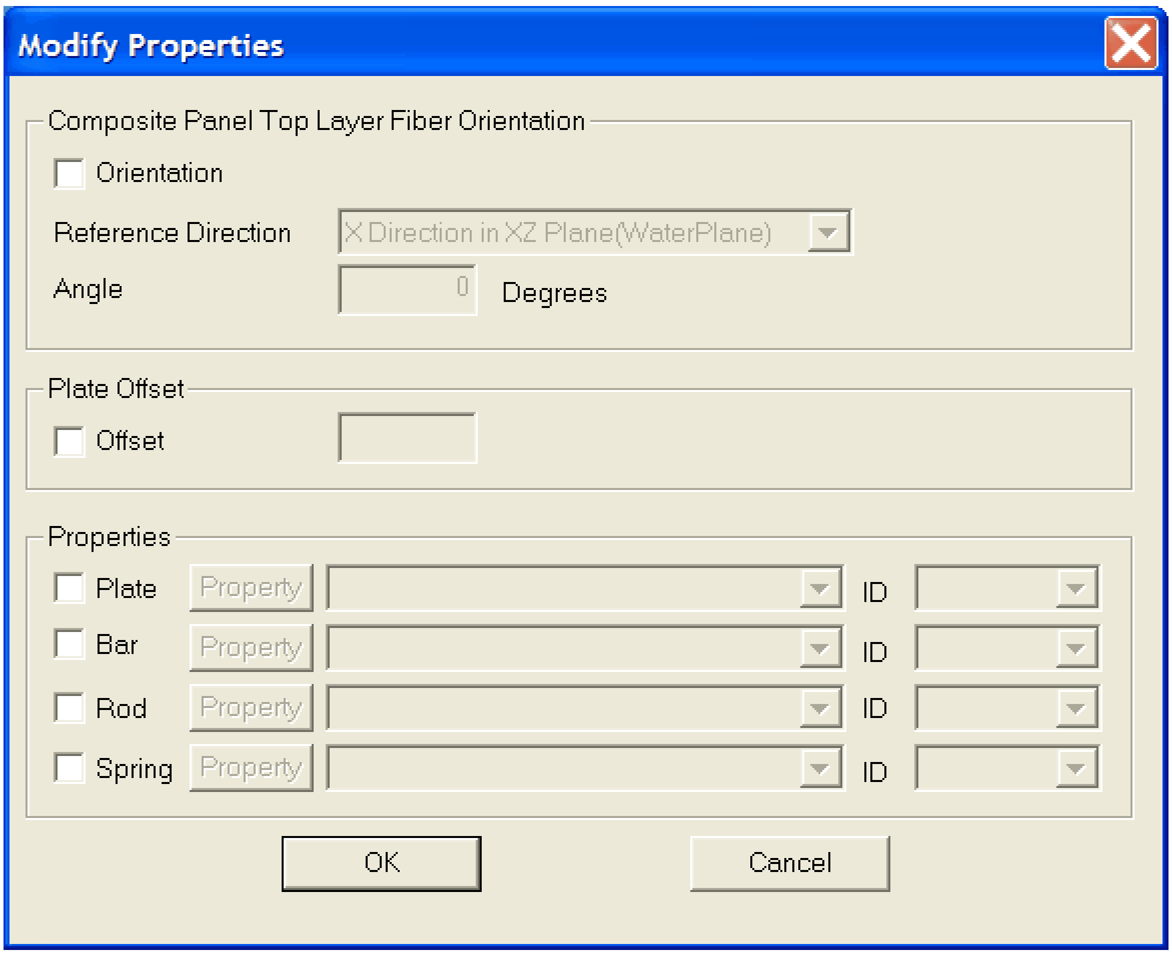
© 2014 Optimum Structural Design, Inc.

Refine

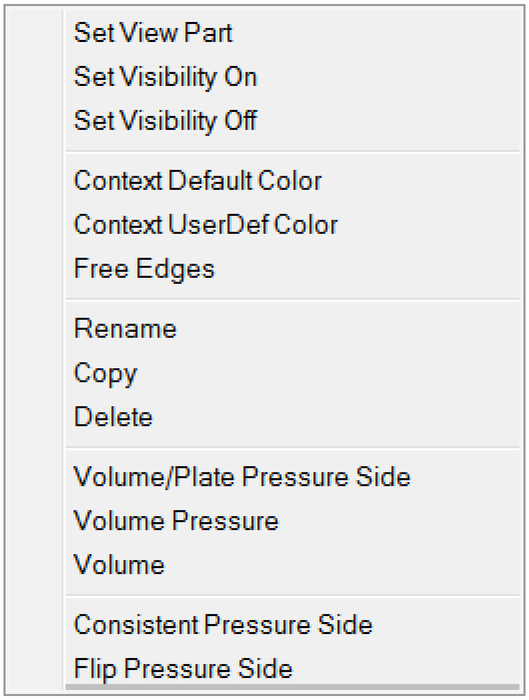
This will launch the Refine dialog and allow the user to create a fine mesh model of the elements in the highlighted general group.

Modify

This will launch the Modify Properties dialog which allows the user to modify the highlighted general group's settings. These settings will apply to all applicable elements of the general group. For instance changing the plate property will change all plate elements in the group to the specified property.



The Volume Groups right-click menu is:



*** Please see the General Groups right-click menu definitions for menu options not listed below.*

Volume/Plate Pressure Side

This will display a graphical representation of which side of the plate elements the pressure load is being applied to.

Volume Pressure

This will display a graphical representation of the volume pressure for the selected group if defined in the currently selected load case.

Volume

This will report the volume of the group, if applicable.

Consistent Pressure Side

This will make all of the elements pressure side in the volume group uniform.

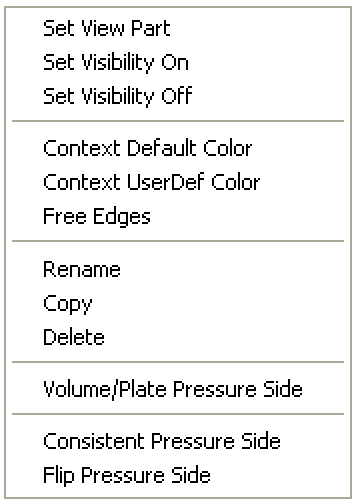
Flip Pressure Side

This will flip the pressure side of all elements in the volume group.

Volume Table

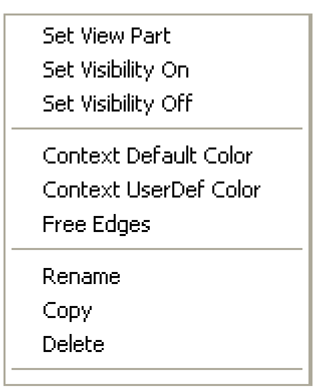
This will produce a table of tank volumes and masses based on the currently selected load case. Note this option is only available when right-clicking on the "parent" volume group item in the groups tree.

The Plate Groups right-click menu is:



*** Please see the General Groups and Volume Groups right-click menu definitions for menu options not listed below.*

The Nodal Groups, Module Groups, Section Groups, Bay Groups, and Corrosion Groups right-click menu is:



*** Please see the General Groups and Volume Groups right-click menu definitions for menu options not listed below.*

9.2 Creating Groups

Toolbar	
Menu	**Groups >**
Keyboard	**<Ctrl + g>**

The groups dialog allows for the creation of 8 different types of groups. New groups are automatically added to the groups tree under its appropriate heading. Groups are often used to load the model as shown in the figure, as well as for post-processing. For details on using the group operations dialog, please see the Groups Menu section.

A more detailed description and steps to create each group are below:

General

Volume

Plate

General/Volume/Plate Group Selection Options

Module

Section

Nodal

Bay

Corrosion

Wetted

Organizing Groups

Defining General Groups from an Elements List or Nodes List

Defining General Groups from Seed Elements

Defining Wetted Elements from General Groups

Defining Volume Groups from General Groups

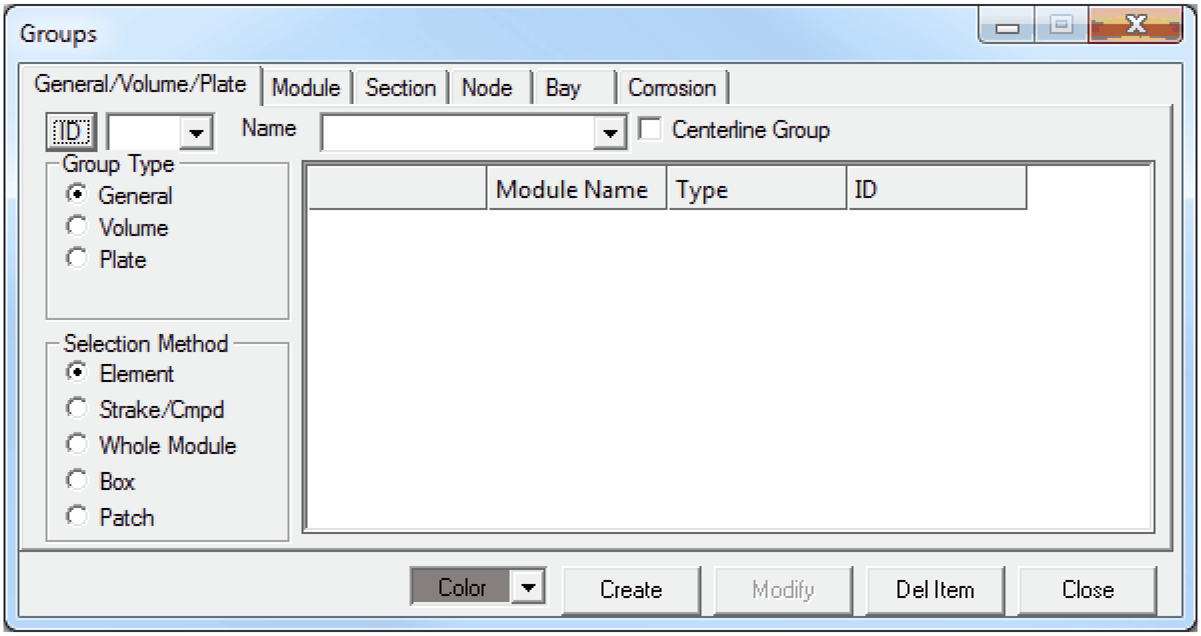
© 2014 Optimum Structural Design, Inc.

[Defining Consistent Normals for Volume, Plate, and General Groups](#)

General Group

The General groups dialog is a convenient way for the user to create a collection of elements for viewing "areas of interest." The General groups is also used create a General group which can then be refined for fine meshing or simply used for viewing this particular "area of interest." A general group can include any type of elements.

1. Begin by opening the groups dialog box using the **Groups > General...** menu option, or from the toolbar.



2. Click the ID button to assign a unique ID to the general group.

3. Type a descriptive name into the Name box and select Select under the Group Type.

4. Check the *Centerline Group* if the general group spans the centerline of a half model. This will automatically combine the mirrored group into one single group if the model is mirrored.

5. Checking the *Full Strake/Full Compound* box will include the entire strake or compound as the part of the general group by clicking on any part of it.

6. Checking the *Whole Module* box will include all elements of the module by clicking any part of it.

7. Click inside the main white part of the dialog box and then select the elements that make up general group. The *Select By Box* option can be checked to use a box window to add all the elements within that select box.

Once an element is added to the group, right-clicking on the element will bring up a menu allowing the user to flip normal side, or add all elements with the same property, material,

© 2014 Optimum Structural Design, Inc.

stiffener layout, or type. This will apply to all elements in the current view part.

8. Select a color from the drop down menu to give the general group a unique color.
9. Click the Create button.

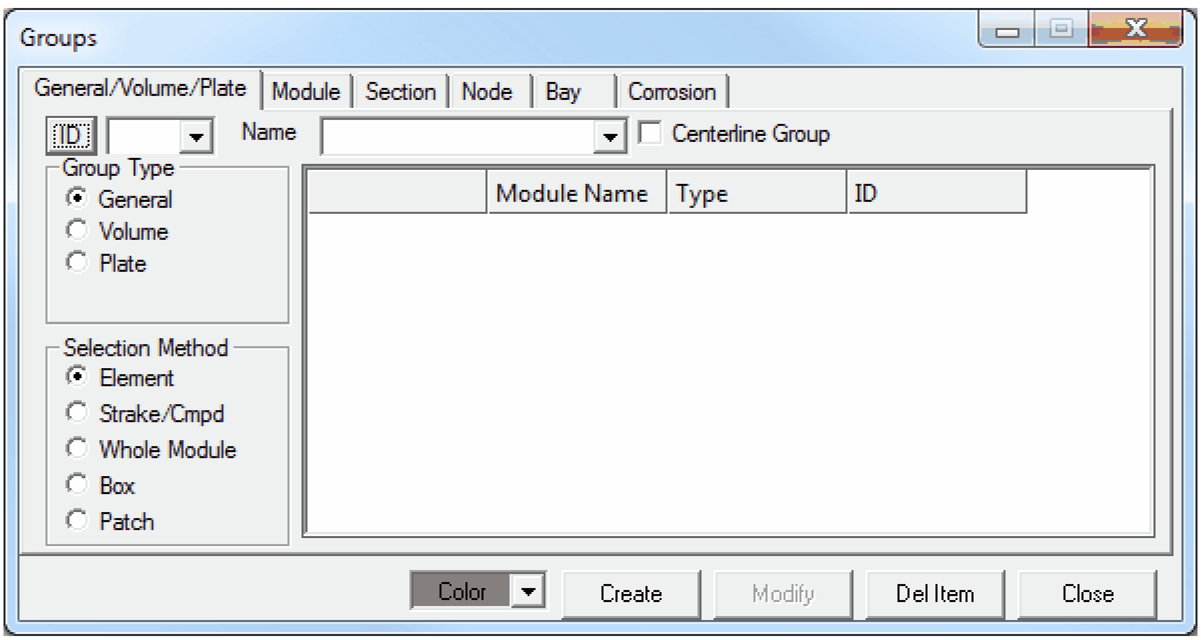
The new general group will appear in the Groups Tree as one of the General groups, with whatever name was given in the Groups dialog box.

Volume Group

Tanks can be easily created in MAESTRO by defining a volume group by clicking the elements that make up the boundary faces of the tank.

This tutorial shows the procedure for creating a tank defined by a volume group.

1. Begin by opening the groups dialog box using the **Groups > Volume...** menu option, or from the toolbar.



2. Click the ID button to assign a unique ID to the tank.
3. Type a descriptive name into the Name box and select Volume under the Group Type.
4. Check the *Centerline Group* if the tank spans the centerline of a half model. This will automatically combine the mirrored group into one single group if the model is mirrored.
5. Checking the *Full Strake/Full Compound* box will include the entire strake or compound as the tank boundary by clicking on any part.

6. Click inside the main white part of the dialog box and then select the elements that make up the tank boundary faces. The *Select By Box* option can be checked to use a box window to add all the plate elements within that select box.

Once an element is added to the group, right-clicking on the element will bring up a menu allowing the user to flip normal side, or add all elements with the same property, material, stiffener layout, or type. This will apply to all elements in the current view part.

7. The Normal of the element can also be flipped by double clicking in the *Flip Normal* column to change between *Yes* and *No.*

Once the Flip Normal is changed to *Yes* or *No*, it will remain the same as new elements are added to the definition.

8. Select a color from the drop down menu to give the tank a unique color. You can view the tank in your user defined color by right-clicking the group in the groups tree and selecting *Context UserDef Color*

9. Click the Create button.

The new tank will appear under the group tab of the parts tree under the Volume folder as the name given in the Groups dialog box.

10. Right-click the tank name and select *Set View Part.*

11. Select **View > Plate > Volume/Plate Pressure Side** from the menu.

This will show the pressure side of the tank due to its volume contents. In order for MAESTRO to treat the tank load properly, the tank plate elements should all have their volume pressure side on the inside of the tank.

12. While still in the Volume/Plate Pressure Side view, the dynamic query can be used to highlight an element and right-click and select *Flip Pressure Side* to change the pressure side. Similarly, the user can right-click on a volume group and select *Consistent Pressure Side*. This will automatically assign all of the inside or outside elements to the same Volume/Plate Pressure side. If the outside of the tank is the pressure side, the user can right-click on the volume group and select *Flip Pressure Side* to automatically flip all the pressure sides.

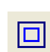
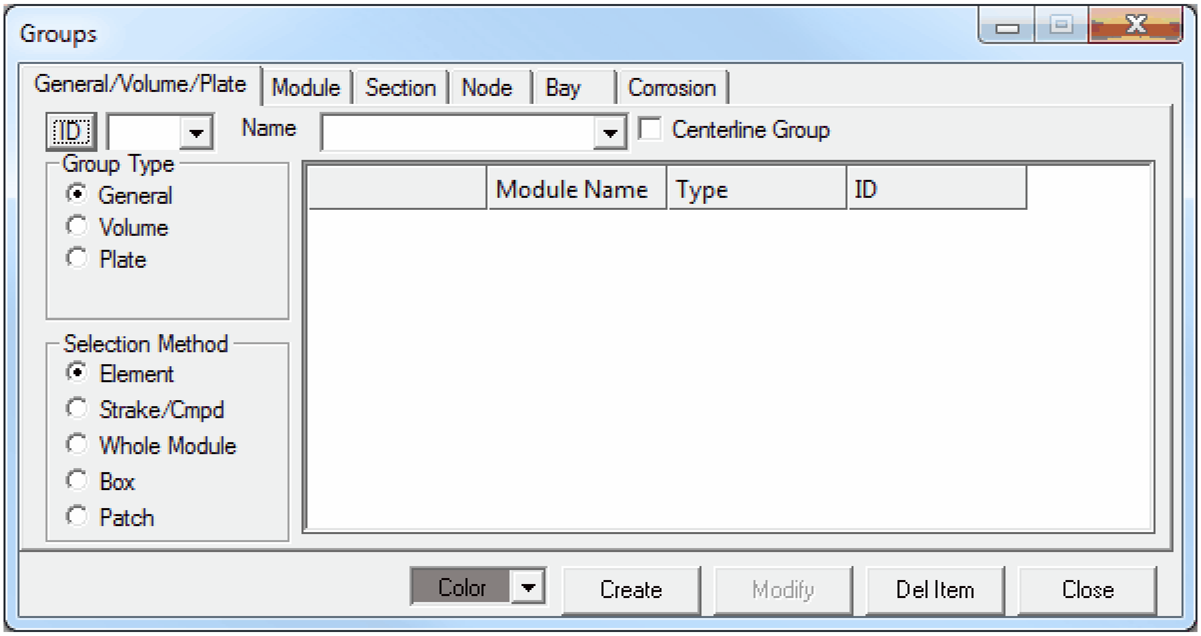
It is helpful to use the shrink elements view,  to verify the pink pressure side is on the inside of the tank.

Plate Group

A plate group is a group of panel elements that can be used to apply a mass or pressure load to.

1. Begin by opening the groups dialog box using the **Groups > Plate...** menu option, or from the toolbar.

© 2014 Optimum Structural Design, Inc.



2. Click the ID button to assign a unique ID to the plate group.

3. Type a descriptive name into the Name box and select *Plate* under the Group Type.

4. Check the *Centerline Group* if the plate group spans the centerline of a half model. This will automatically combine the mirrored group into one single group if the model is mirrored.

5. Checking the *Full Strake/Full Compound* box will include the entire strake or compound as the part of the plate group by clicking on any part of the strake or compound panels.

6. Click inside the main white part of the dialog box and then select the elements that make up the tank boundary faces. The *Select By Box* option can be checked to use a box window to add all the plate elements within that select box.

Once an element is added to the group, right-clicking on the element will bring up a menu allowing the user to flip normal side, or add all elements with the same property, material, stiffener layout, or type. This will apply to all elements in the current view part.

7. The Normal of the element can also be flipped by double clicking in the *Flip Normal* column to change between *Yes* and *No.*

Once the Flip Normal is changed to *Yes* or *No*, it will remain the same as new elements are added to the definition.

8. The mass of the plate group can be defined here and will be used if the plate group is added to a load case as a mass.

9. Select a color from the drop down menu to give the plate group a unique color.

10. Click the Create button.

The new plate group will appear under the group tab of the parts tree under the plate folder as the name given in the Groups dialog box.

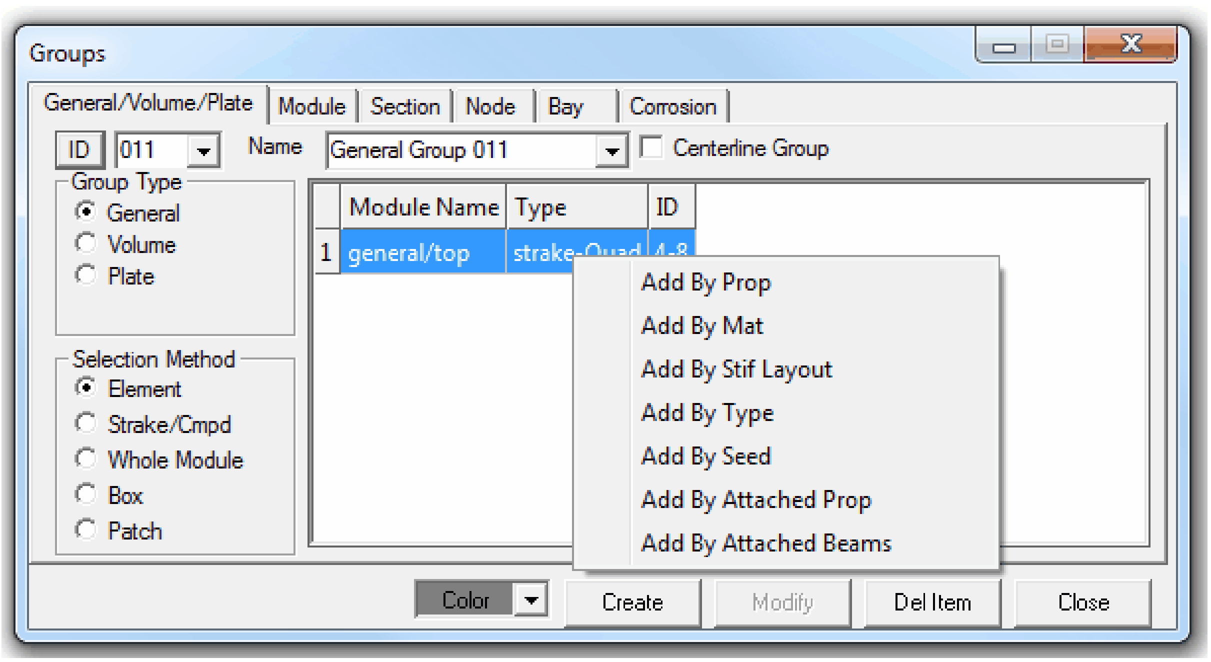
© 2014 Optimum Structural Design, Inc.

General/Volume/Plate Selection Options

When adding elements to a general/volume/plate group, the element selection options are:

- Element: the user manually clicks each element to add to the group.
- Strake/Cmpd: the user clicks a single element of a strake or compound and that entire strake or compound is added to the group.
- Whole Module: all of the selected module's elements are added to the group.
- Box: the user is able to create a box with the mouse and add all of the visible elements within the box to the group.
- Patch: the user can add elements by selecting on evaluation patches. This will add all of the elements making up the patch.

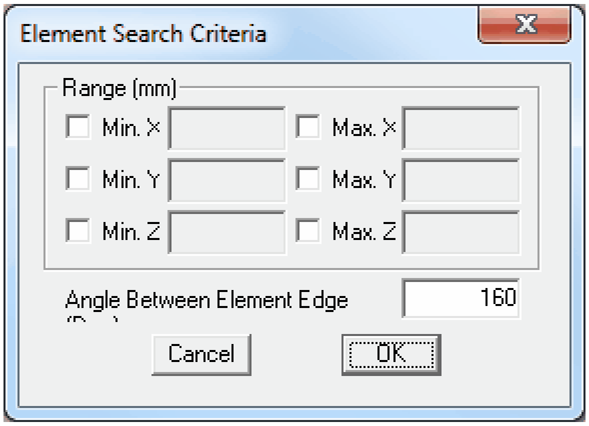
In addition to these selection methods, there are several options available from the right-click menu once an element is added to the group:



- Add By Prop: This option will automatically add any other visible elements with the same property (i.e., plate, beam, rod, spring property) as the element selected.
- Add By Mat: This option will automatically add any other visible elements with the same material property as the element selected.
- Add by Stif Layout: This option will automatically add any other panel elements with the same stiffener layout as the element selected.
- Add By Type: This option will automatically add any other visible elements with the same type (e.g., strake quad, strake beam, additional quad, etc.) as the element selected.

© 2014 Optimum Structural Design, Inc.

- Add By Seed: This option will use the selected element, or elements, as seed elements in order to search for other elements. This option will open the Element Search Criteria dialog and follows the same process as described here.

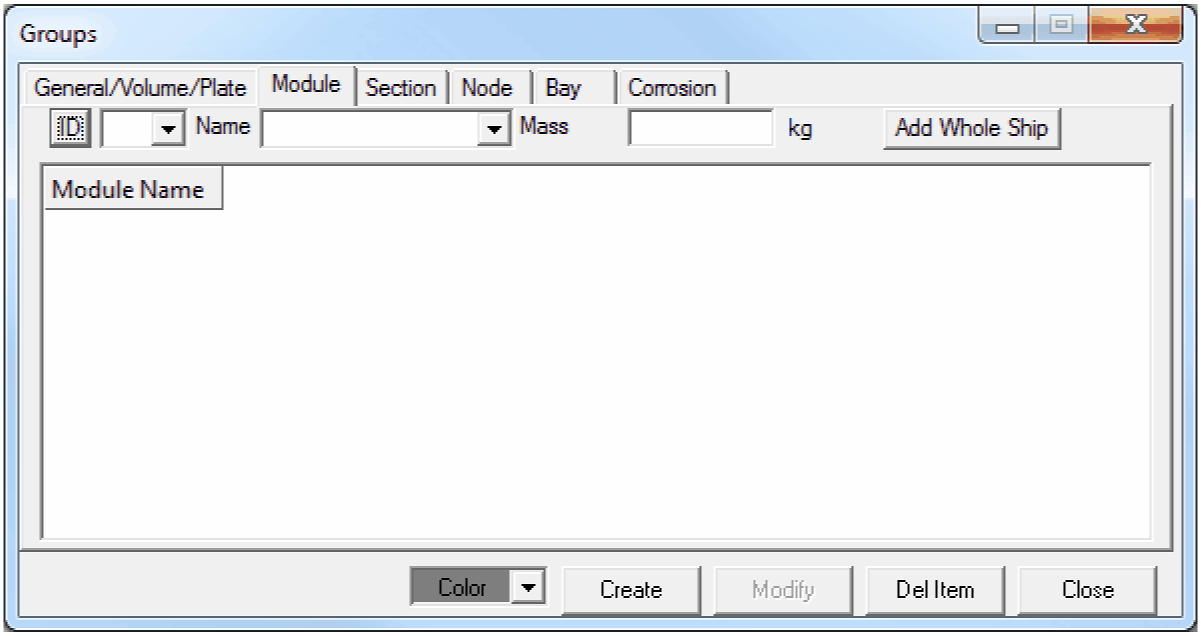


- Add By Attached Prop: This option will automatically add any attached elements that share the same property as the selected element.
- Add By Attached Beams: This option will automatically add any visible beam elements connected to the selected plate element.

Module Group

A module group is a group of module(s) used to scale the structural weight of a model. This group is used to define a non-structural mass whose spatial distribution closely approximates the surrounding structure. The mass is distributed among the structural nodes in the same proportion as the structural mass, and can represent items such as furniture, paneling, auxiliary machinery, or any additional structural weight. This can also be used as a tool to match a known weight distribution.

1. Begin by opening the groups dialog box using the **Groups > Module...** menu option, or from the toolbar.



2. Click the ID button to assign a unique ID to the module group.

3. Type a descriptive name into the Name box.

4. Click in the white space and then click on the module(s) to add to the group.

5. Assign the modules a scaled mass/weight value. This mass/weight is assumed to be symmetric, and should be a half value for half-width models. Note the field is dependent on the Units used; Mass (when working with SI Units) or Weight (when working with fps, mks, cks, and ips Units).

6. The *Add Whole Ship* option can be clicked to add all modules. In this case, the mass/ weight assigned to these modules is the full value, regardless of whether the modules are a half-ship.

7. Select a color from the drop down menu to give the module group a unique color.

8. Click the Create button.

The new module group will appear under the group tab of the parts tree under the module folder as the name given in the Groups dialog box.

Section Group

In the design of long structures such as ships, where overall bending is the dominant load effect, the lengthwise distribution of mass must be accounted for early in the design, and should be modeled as accurately as possible at all stages of the design. Ideally in a three-dimensional model the masses should be placed at their actual locations, using whichever of the methods presented here (volumes, bays, module, nodal, plate) is most appropriate for each type of mass.

But at early stages of design some masses may be known (or estimated) only in the form of

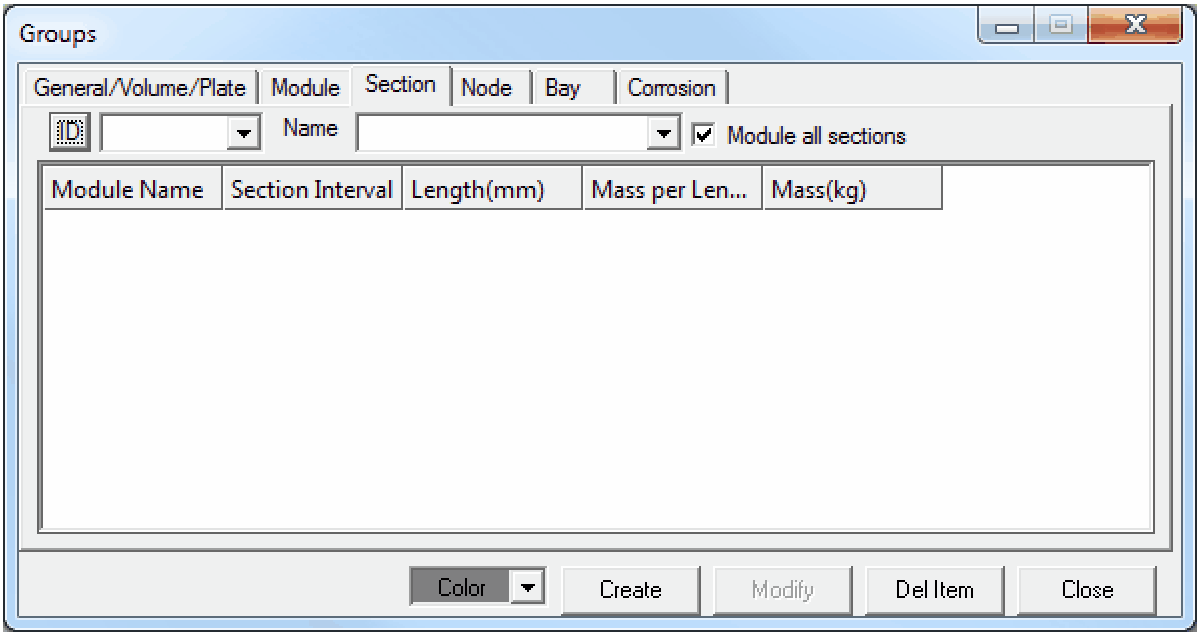
© 2014 Optimum Structural Design, Inc.

a one-dimensional distribution along the length of the structure, and the Section definition option is intended for this (and only this, because it is very approximate). In the Section option, the one-dimensional mass distribution is specified as a target value of mass per length for each section interval of selected modules. For each section interval, the program converts this target value of distributed mass into point masses at the endpoint-generated nodes in each of the two sectional planes at the boundaries of that section interval. Hence the selected modules must be orientated lengthwise in the structure. Transverse modules (such as a transverse bulkhead) should not be included. With this option there is no flexibility in choosing the nodes; within each section (YZ plane) the program uses all of the endpoint-generated nodes, and it allocates the point masses in proportion to the structural mass of the strake-based elements (panels, girder segments and frame segments) attached to those nodes.

The target values of distributed mass are specified (and may differ) for each section interval. The target values refer only to the modules for which they are defined. In the simplest case, when each module constitutes a complete transverse section of the structure, the target values correspond to the full cross section of the structure (or to a half section for a half model). However, if some of the modules are in parallel (such as a hull module surmounted by a superstructure module) then the target values of distributed mass must be divided and apportioned among those modules. In a half model all specified values of mass should be half values.

A sections group is used to define a non-structural mass which is distributed among the sections of a module. The additional mass on the module can be either equal for all sections, or different for each section. Within each section, each endpoint-generated node carries the same mass.

1. Begin by opening the groups dialog box using the **Groups > Section...** menu option, or from the toolbar.



2. Click the ID button to assign a unique ID to the section group.

3. Type a descriptive name into the Name box.

4. Click in the white space and then click on the module(s) to add to the group. Clicking any element of a module will add the entire module to the group.

If the *Module all sections* box is checked, when a module is added, it's mass/weight per length will apply to all sections of that module.

5. Define the mass/weight per length for each section or entire module.

The mass will automatically update based on the length of the section.

6. Select a color from the drop down menu to give the section group a unique color.

7. Click the Create button.

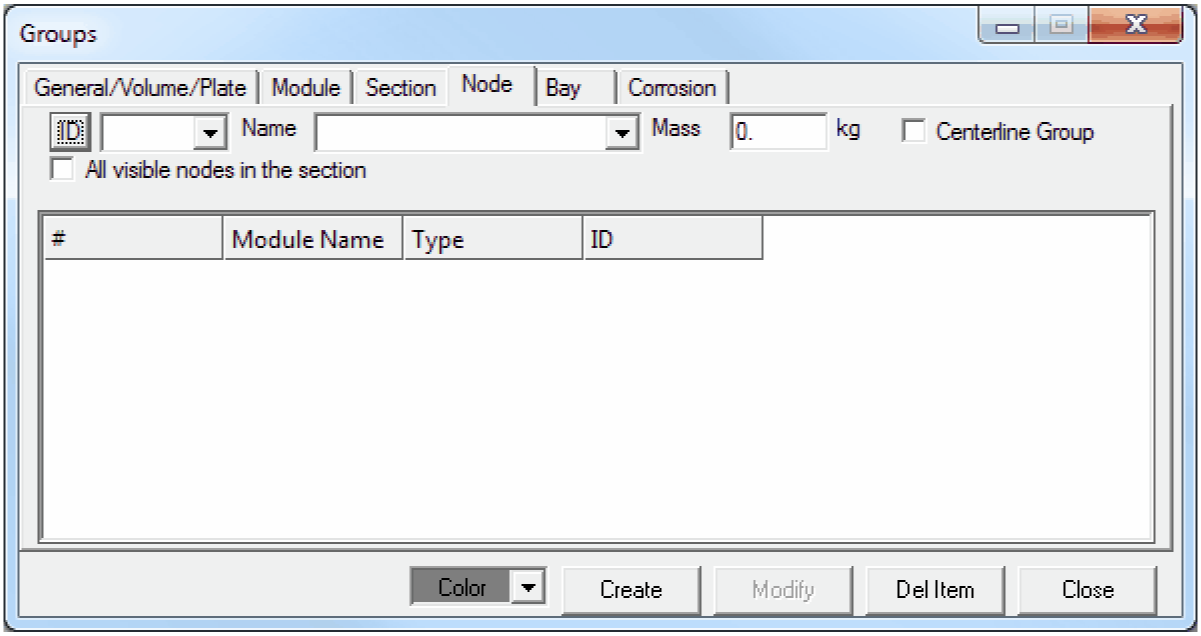
The new section group will appear under the group tab of the parts tree under the section folder as the name given in the Groups dialog box.

Node Group

A node group is used to define an additional mass which is equally divided among a collection of nodes. The nodes may be of any type (endpoint-generated or additional), and can be located anywhere in the model.

1. Begin by opening the groups dialog box using the **Groups > Node...** menu option, or from the toolbar.

© 2014 Optimum Structural Design, Inc.



2. Click the ID button to assign a unique ID to the node group.

3. Type a descriptive name into the Name box.

4. Click in the white space and then click the nodes to be added to the group. Checking the *All visible nodes in the section* will add all the nodes in the section of the next clicked endpoint node. Note: this must be unchecked to add an additional node to the group.

5. Check the *Centerline Group* if the node group spans the centerline of a half model. This will automatically combine the mirrored group into one single group if the model is mirrored.

6. Define the mass/weight of the node group. Each node will receive an equal portion of the total mass/weight. Note: if the nodal group is made up of only centerline nodes, the mass/weight should be assigned the full value regardless of whether is is a half model.

7. Select a color from the drop down menu to give the section group a unique color.

8. Click the Create button.

The new node group will appear under the group tab of the parts tree under the Node folder as the name given in the Groups dialog box.

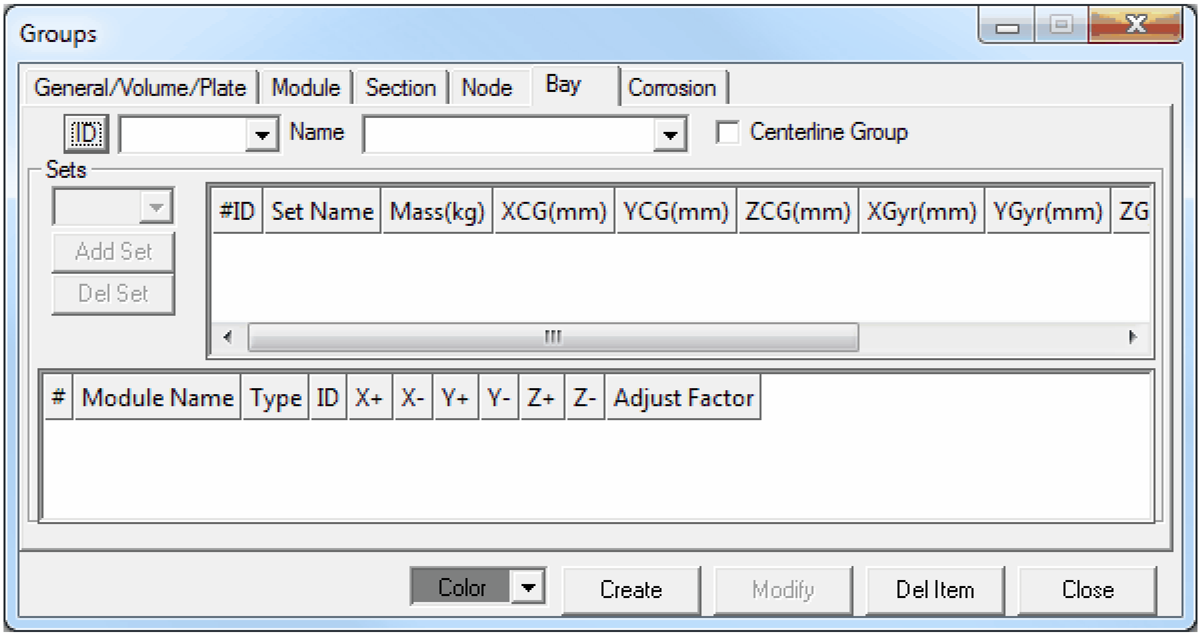
Bay Group

This group option is intended for large solid masses that are supported at several nodes and whose center of mass is at an appreciable distance from these nodes. Common examples are bays of containers, main engines, and un-modeled portions of structure such as masts. Bays, and the sets of masses within them, are selected individually by using the groups dialog. In addition, the way a support node handles mass may change between load cases. This simulates the case of containers, particularly, which are constrained by cell guides and one-way attachment devices.

© 2014 Optimum Structural Design, Inc.

NOTE: The defined centroid and gyradii are not carried over in a Natural Frequency Analysis.

1. Begin by opening the groups dialog box using the **Groups > Bay...** menu option, or from the toolbar.



2. Click the ID button to assign a unique ID to the bay group.

3. Type a descriptive name into the Name box.

4. Click Create.

5. Click Add Set. You can then give the set a name, mass, centers of gravity and radii of gyration.

6. Click in the lower window, and then select nodes from the model to add to the set.

7. The columns for X+, X-, Y+, Y-, Z+, and Z- are used to define whether the corresponding node can receive force in that direction due to an additional acceleration, designated either with a "Y" for yes or "N" for no.

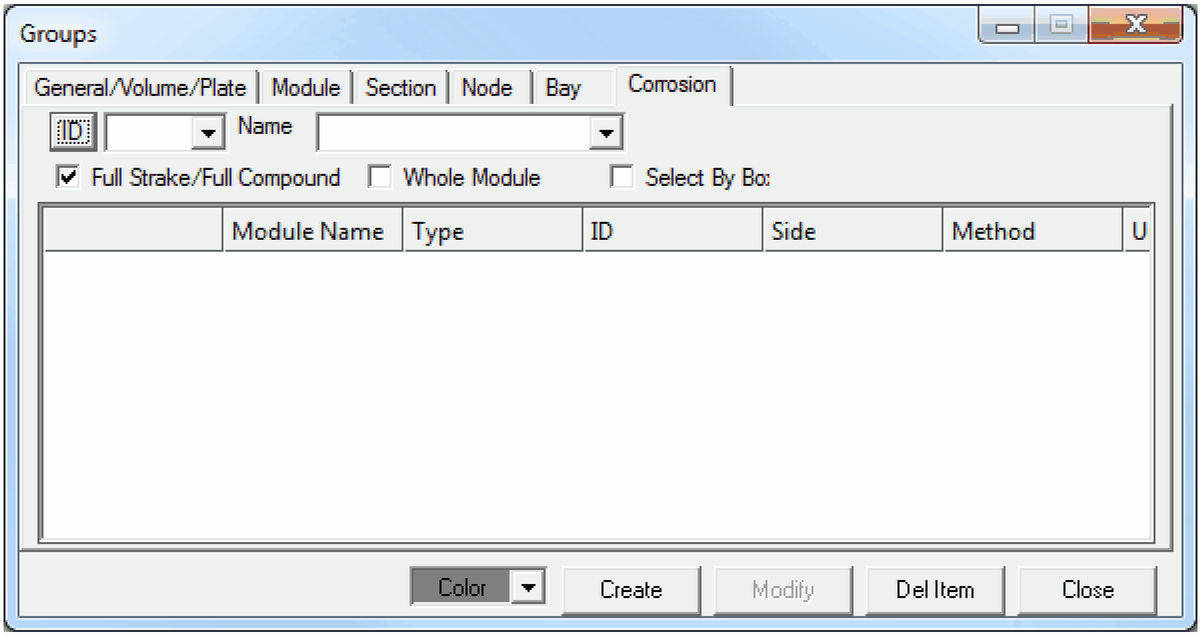
8. Click Modify to save the changes.

Sample bay set input data can be found in the *Models and Samples* MAESTRO installation directory as BayAcceleration.mdl and BayBalance.mld.

Corrosion Group

This group option allows the user to define a group that will consider corrosion effects. The user can define which side (stiffener, plate, or both) the corrosion effects occur. Further, the user can choose to apply corrosion by a percentage or as a net. The user can apply corrosion to the plate, web, or flange associated with the corrosion group.

1. Begin by opening the groups dialog box using the **Groups > Corrosion...** menu option, or from the toolbar.



2. Click the ID button to assign a unique ID to the corrosion group.

3. Type a descriptive name into the Name box.

4. Check the *Centerline Group* if the general group spans the centerline of a half model. This will automatically combine the mirrored group into one single group if the model is mirrored.

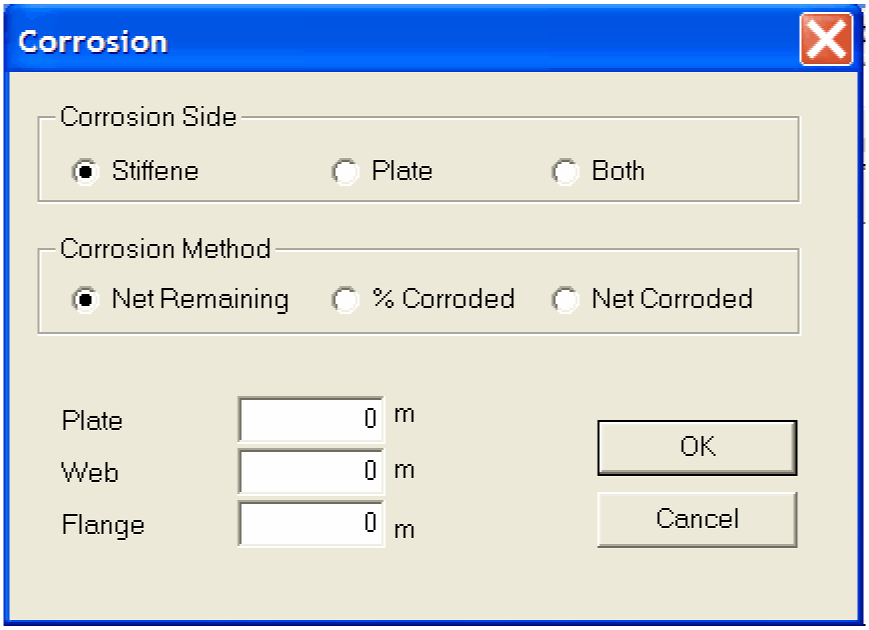
5. Checking the *Full Strake/Full Compound* box will include the entire strake or compound as the part of the general group by clicking on any part of it.

6. Checking the *Whole Module* box will include all elements of the module by clicking any part of it.

7. Click inside the main white part of the dialog box and then select the elements that make up the tank boundary faces. The *Select By Box* option can be checked to use a box window to add all the plate elements within that select box.

Once an element is added to the group, right-clicking on the element will bring up a menu allowing the user to flip normal side, or add all elements with the same property, material, stiffener layout, or type. This will apply to all elements in the current view part.

If the *Whole Module* box is checked when selecting elements, an additional dialog box will open.



This allows the user to define the corrosion side and method for defining the corrosion. This will be applied to all elements in the selected module.

8. Select a color from the drop down menu to give the corrosion group a unique color.

9. Click the Create button.

The new corrosion group will appear under the group tab of the parts tree under the Corrosion folder as the name given in the Groups dialog box.

Group Wetted Elements

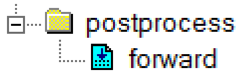
A general group of all defined "wetted" plate elements can be created using the **Groups > Wetted Elements** menu. This will automatically create the general group "wet" and does not involve using the Groups dialog. Elements are defined as "wetted" by checking the box for quads and triangles or by selecting "side" or "bottom" as the *location* in the strakes dialog.

Expanded Group Organization Feature

Subfolders can be created under any group types. The subfolder's icon will be a folder, while the group itself will have an icon matching that of a part. To create a subfolder and group within this subfolder, the user must use the "/" (forward slash) character when naming the group.

For example, to create a subfolder and group called "postprocessing" and "forward" respectively, the user would enter the following in the name field: "postprocessing/forward". The resulting group in the Group Tree would look like the following:

© 2014 Optimum Structural Design, Inc.



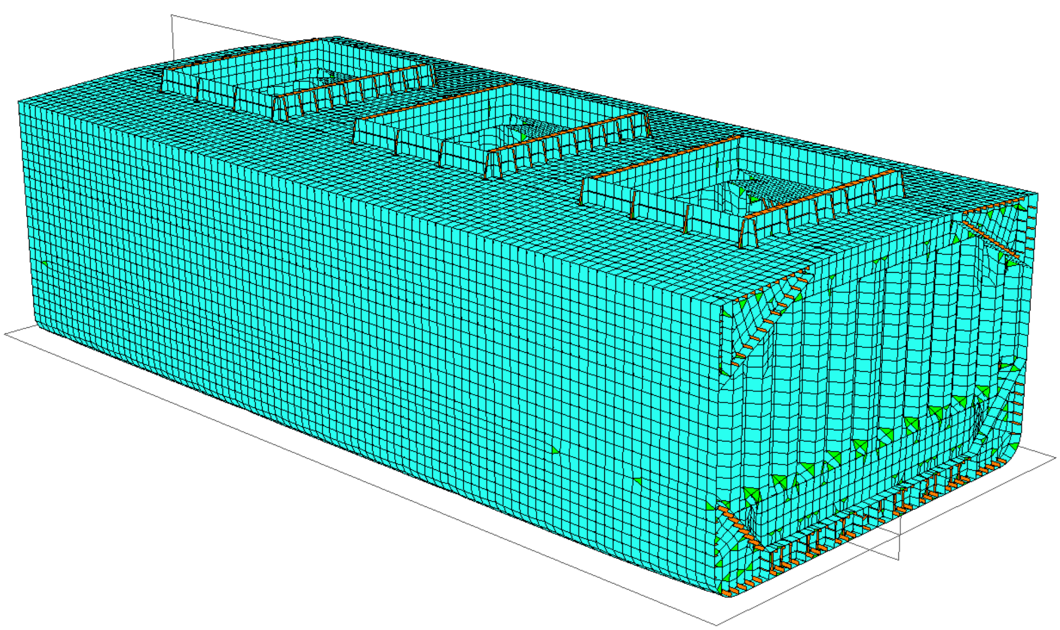
Once multi-level groups are created, the user can simply *drag and drop* the group tree objects into any folder.

Defining General Groups from an Elements List or Nodes List

A list of elements or nodes can be output to the grid tab at the bottom of the MAESTRO interface by selecting the group or module in which you are interested and clicking **Results > List > Elements** or **Results > List > Nodes.** The individual elements or nodes can be made into a general group by right-clicking and selecting *Create General Group*. Multiple elements or nodes can be selected as well and made into a general group. To refresh the list and view only elements or nodes that are visible, right-click and select *Show Current View.*

General Groups from Seed Elements

There are many cases where the user would like to generate large general groups. These general groups are used for post-processing, conversion to other group types, etc. The following example demonstrates how creating general groups from *seed* elements can make this process much quicker than other MAESTRO group creation methods. This method is especially useful when importing Nastran (or other 3rd-party finite element models) models that need the hull *wetted* elements generated for proper hydrostatic/hydrodynamic loading.



The following example creates a general group of the hull envelope using *seed* elements

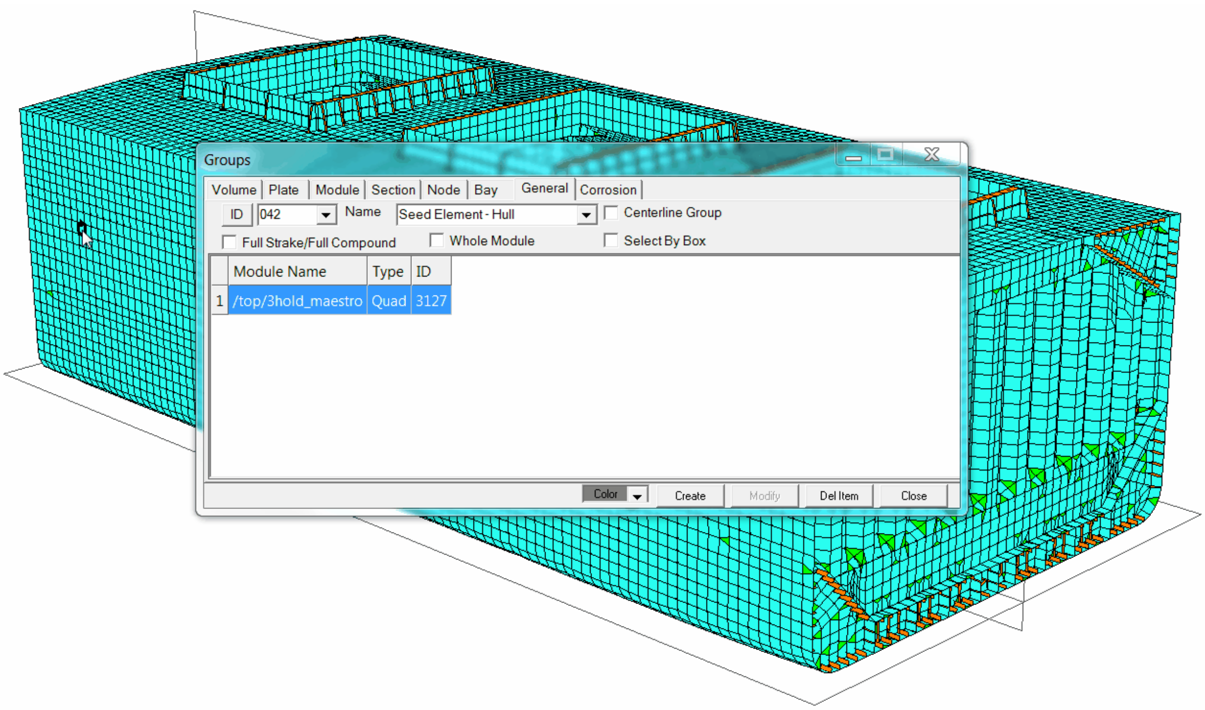
1. Begin by opening the groups dialog box using the **Groups > General...** menu option, or from the toolbar.

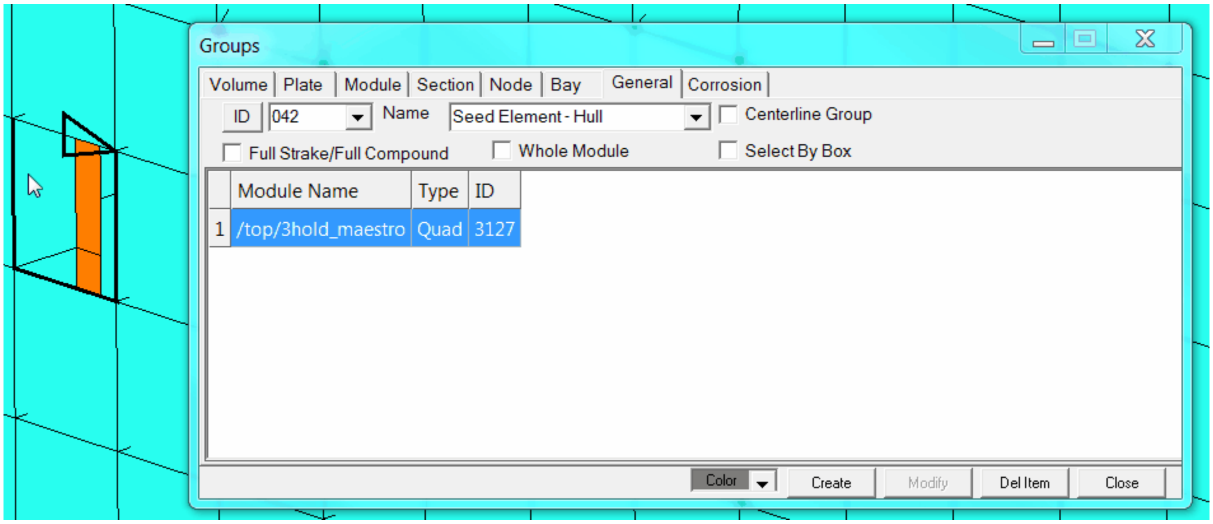
2. Click the ID button to assign a unique ID to the general group.

3. Type a descriptive name into the Name box.

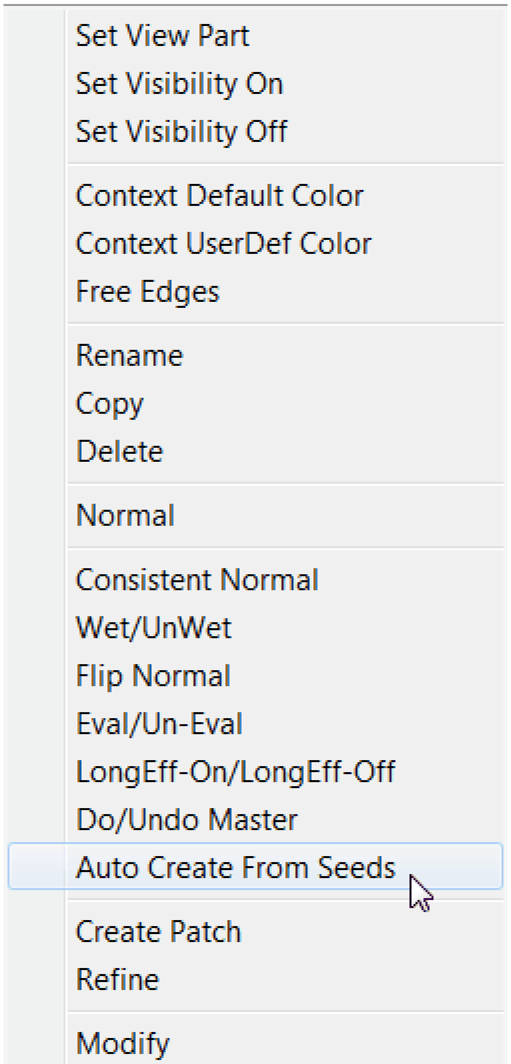
4. Click inside the main white part of the dialog box and then select one or more *seed* elements that make up the hull boundary.

5. Click Create. A general group, consisting of the identified *seed* elements has now been created and can be viewed in the Groups tree.

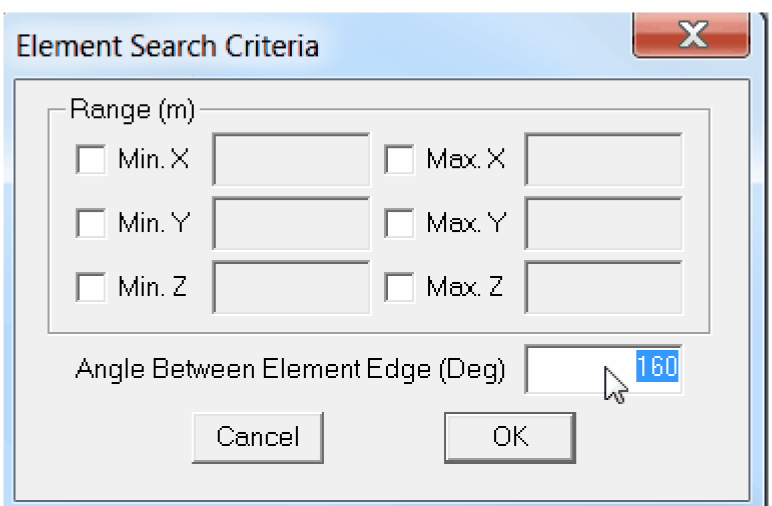




6. From the group tree, select previously defined seed group, right mouse click to select "Auto Create from Seeds" menu item



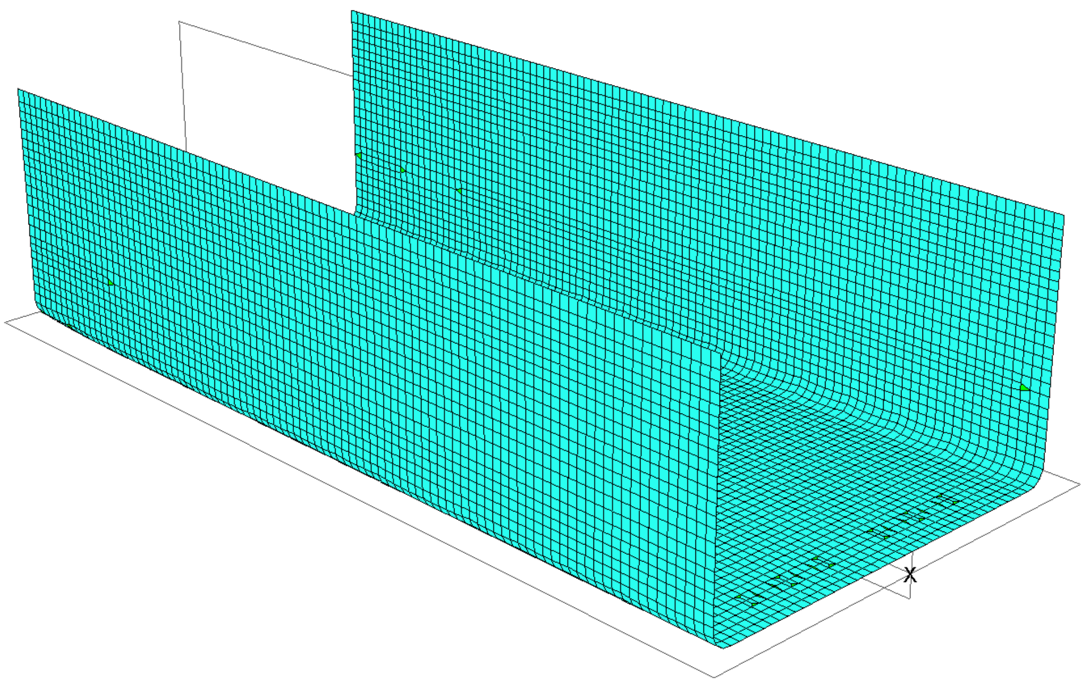
7. The Element Search Criteria dialog box now appears. Define the criteria and click OK.



NOTE:

Here the user has the ability to define the searching criteria by specifying a bounding box range (e.g., Min. X, Max X, Min. Y, etc). The searching criteria is further refined by specifying the threshold angle of neighboring elements. For example, if the user defines 160 degrees, any element whose edge face normal is connected to the *seed* element having an element normal angle greater than 160 degrees relative to the *seed* element edge face normal will be added to the group. This new element, just added to the group will then perform the same criteria check, thus adding the element if it is within the threshold angle and within the bounded box. This process continues until all elements, within the specified criteria are added to the group.

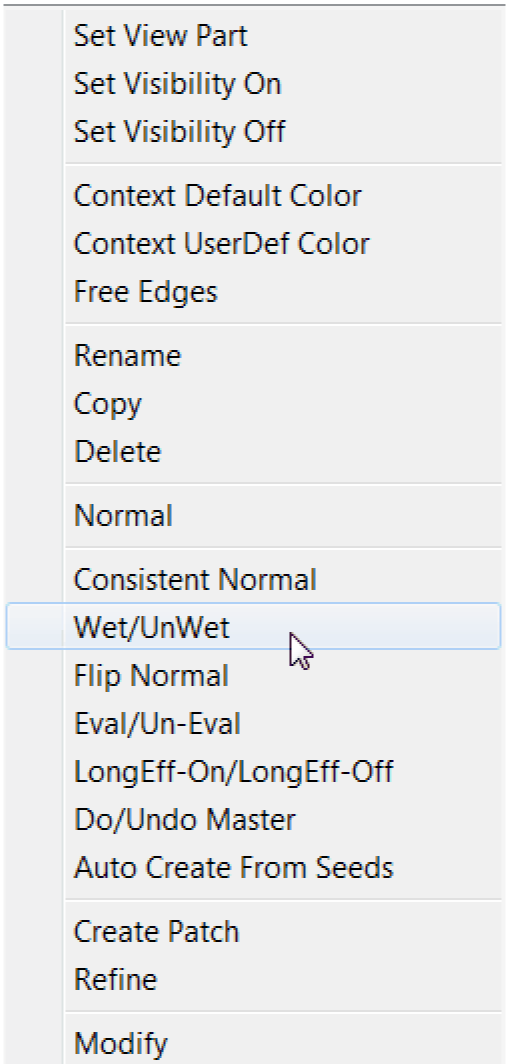
8. The new General Group has been added to the groups tree. View the new group to confirm the definition. In this example, we've created the elements associated with the hull envelope. Now, it is the task to make these elements *wetteable* in order for MAESTRO to apply hydrostatic loading. See the next section for details on how to accomplish this.



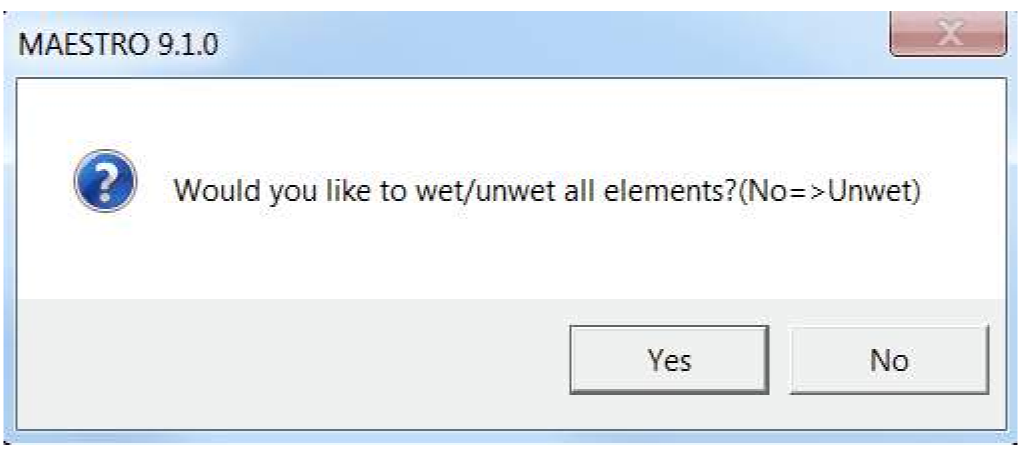
Defining Wettable Elements from General Groups

The following tutorial provides details regarding the definition of *wettable* elements from a general group; therefore, it is assumed you have successfully created a general group.

1. From the groups tree, right-click on the general group of interest and select the "Wettable/UnWettable" menu item.



2. Next, you are asked whether you want to make all elements wettable or unwettable. Yes = make all elements in group wettable elements / No = unwettable elements



3. Using the Parts tree, view the entire model and then choose **View > Wettable Elements**

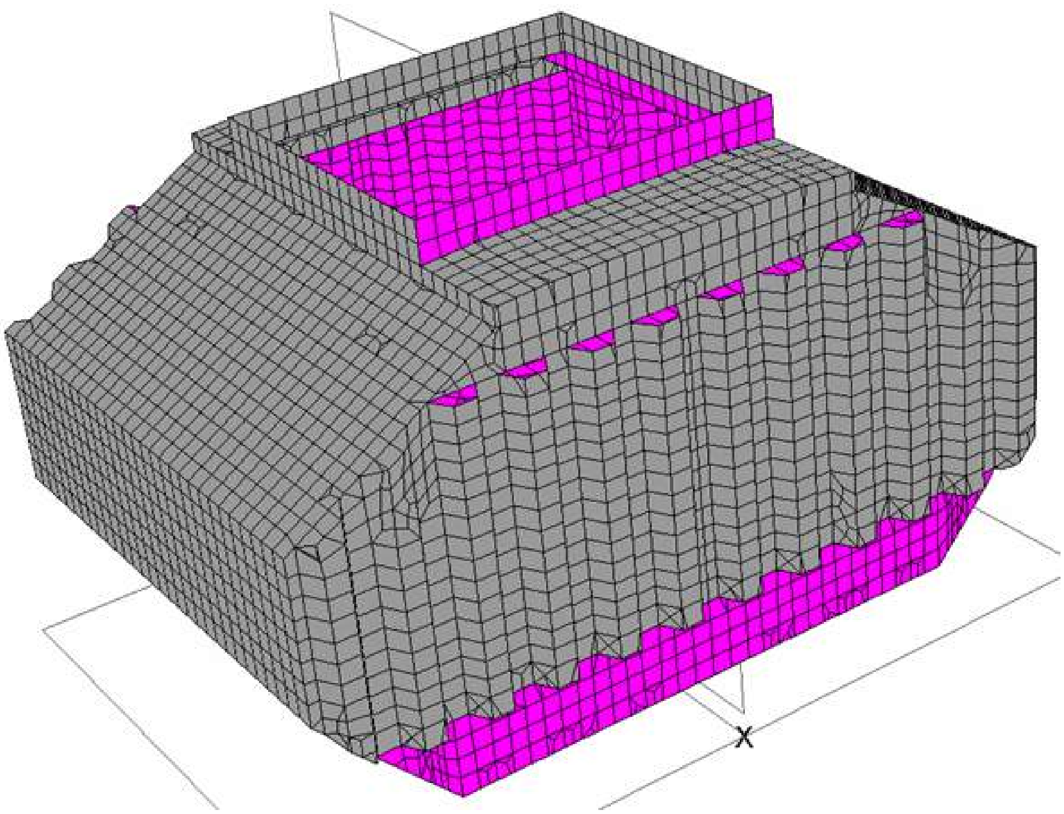
to confirm the elements have been defined correctly.

Defining Volume Groups from General Groups

Volume groups can be created from existing general groups using the group operations dialog. This will convert all of the shell elements from the general group into a new volume group. In addition to this method, a general group can be right-clicked to reveal the *Peel Free Edge Elements* command. Wetted elements can be subtracted from general group as explained in the group operations dialog. Furthermore, free edges can be peeled to reveal a tank within the module. Free edge elements that have been peeled from the general group are preserved as a new general group. They can be dragged and dropped back into the main general group or discarded. When the volume group is all that remains, it can be dragged and dropped under the volume group folder.

Defining Consistent Normals for Volume, Plate, and General Groups

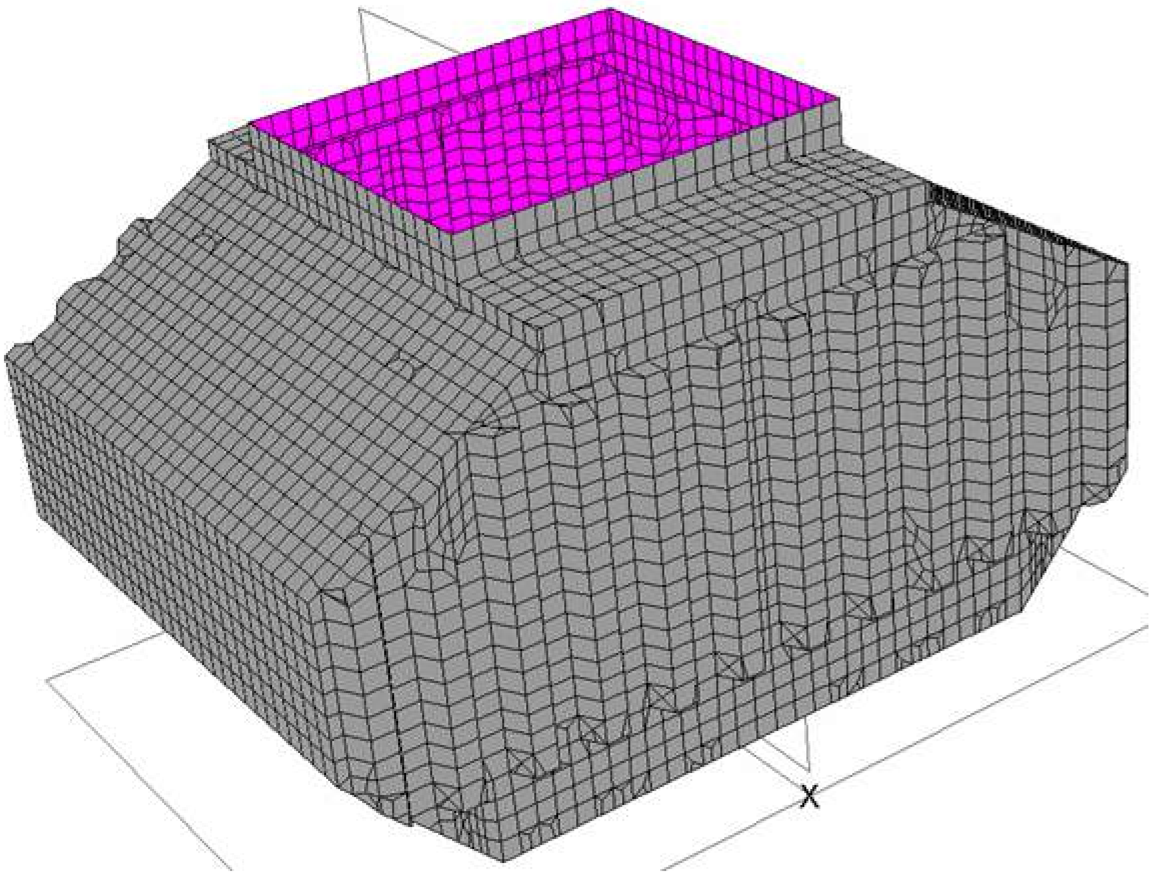
The following procedure will facilitate defining consistent normals for volume and plate groups. This is especially helpful when importing finite element models from 3rd-party tools, where a consistent normal definition is not present (such as the image below)



1. Begin by right-clicking on the volume (and/or plate and/or general) group of interest and

select the "Consistent Normal" menu item.

2. At this point, although the normals are consistent, they still may need to be flipped, which can be done either by dynamically querying individual elements or flipping the normal for the entire group. Click "Flip Pressure Side" to flip the normal for the entire group.



9.3 Defining Loads

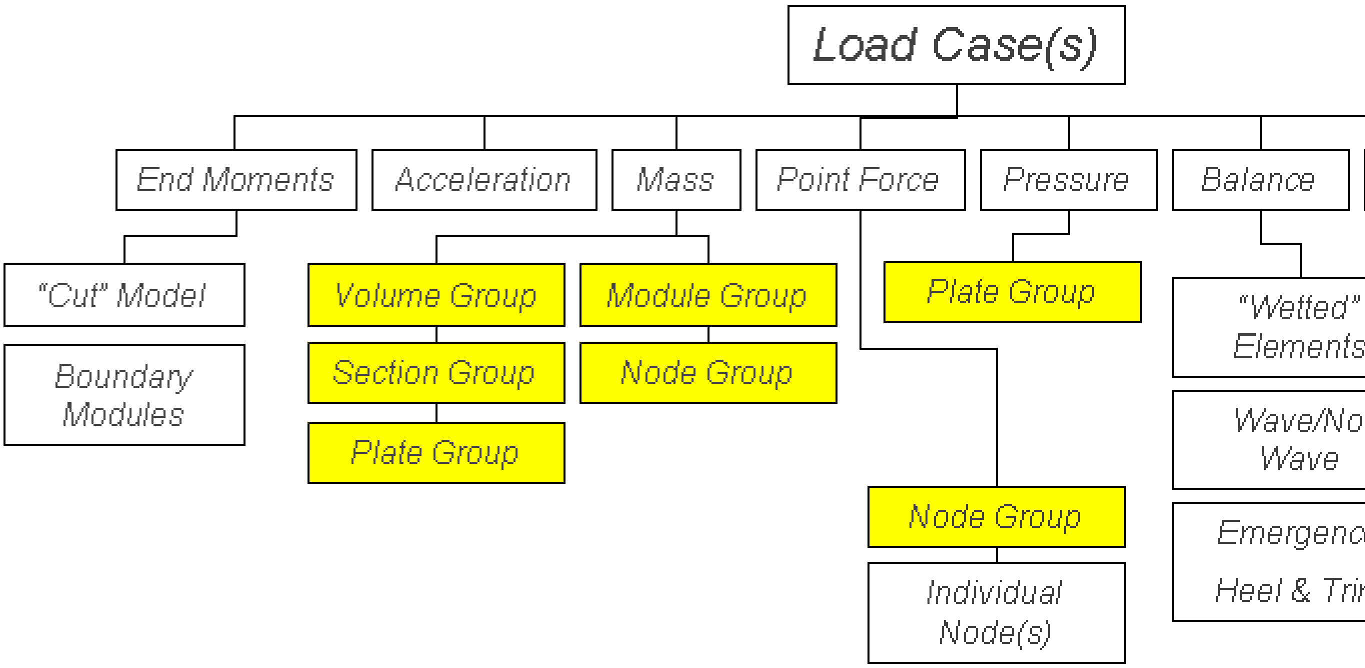
Toolbar	
Menu	**Loads > Create/Modify...**
Keyboard	**<Ctrl + I>**

A load case consists of all of the loads which act on the structure at the same time. Loads which do not act simultaneously should be placed in separate load cases (unless their interaction is negligible). Each load case produces a separate solution for the nodal displacements, and hence load effects, in the structure. In the evaluation portion of

© 2014 Optimum Structural Design, Inc.

MAESTRO, for each possible limit state, the solutions for all load cases are examined to find the worst case (lowest adequacy parameter) for that limit state. A dynamic load case requires masses and accelerations. The mass data is obtained by selecting any combination of masses and by adding properties that are specific to the intended load case.

The following figure presents the overview of MAESTRO's loading capabilities.



The following subsections give more details regarding the Loads dialog:

Creating A Load Case

General Tab

End Moments Tab

Acceleration Tab

Mass Tab

Point Force Tab

Pressure Tab

Balance Tab

Restraint Tab

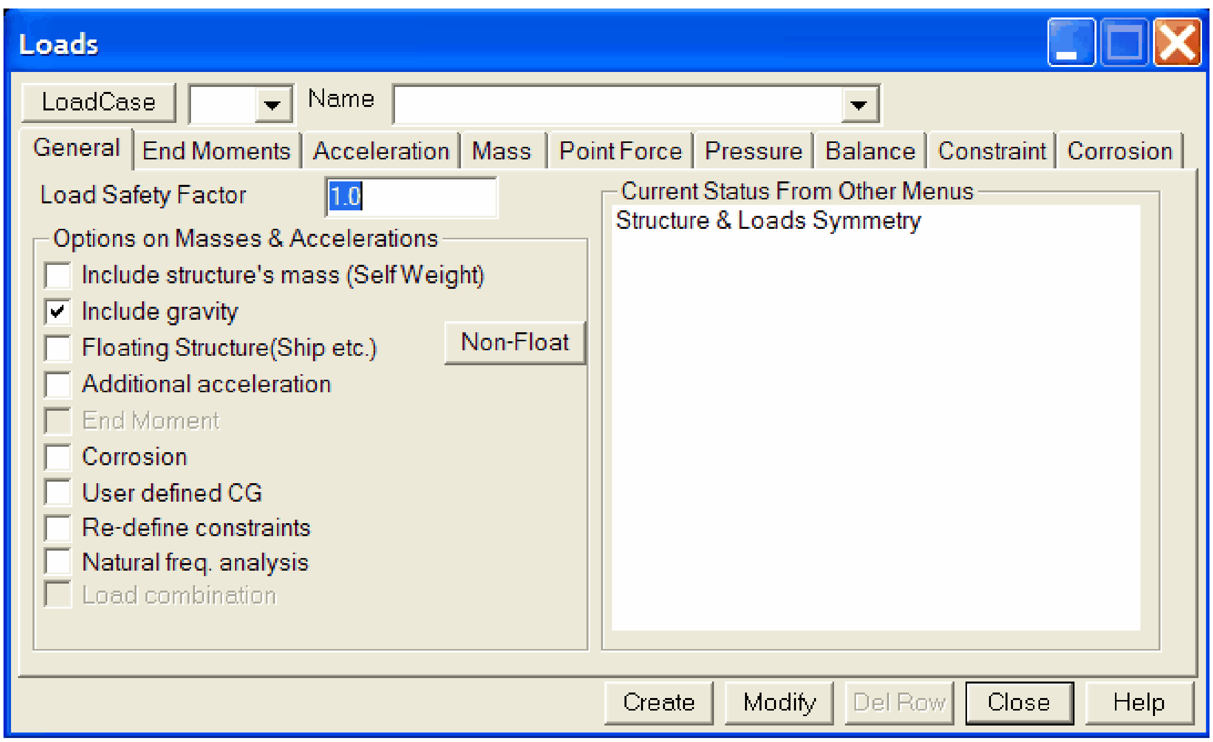
Corrosion Tab

© 2014 Optimum Structural Design, Inc.

9.3.1 Creating A Load Case

The following tutorial shows the procedure for defining a load case.

1. Begin by opening the loads dialog from the **Loads > Create/Modify...** menu option, or from the toolbar.



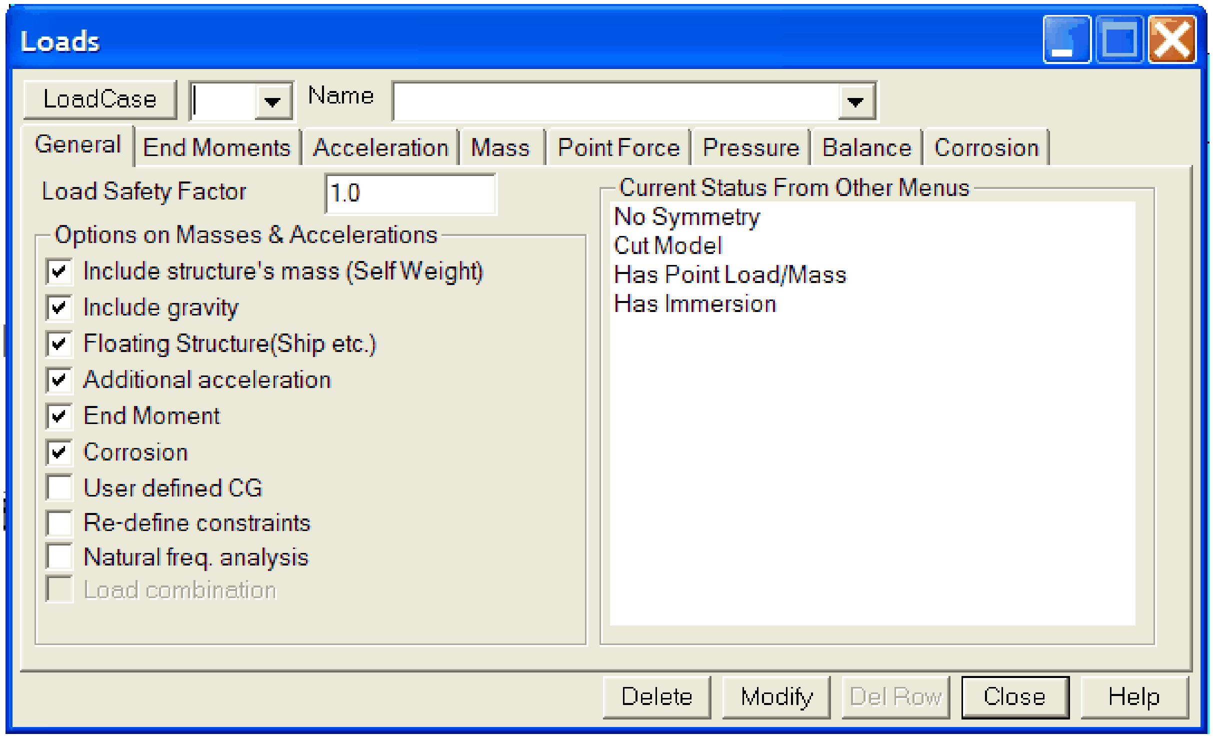
2. Click the LoadCase button to create a new load case.
3. Give the load case a descriptive title.
4. Check the desired options for the load case on the general tab and click Create.
5. Click on each of the relevant tabs to define each aspect of the load case.

Note: Fore more information on each tab option, see the specific tab topics of this section.

6. Repeat steps 2-5 to create additional load cases.

© 2014 Optimum Structural Design, Inc.

9.3.2 General Tab



The general tab allows the user to select the particular load options as well as view the status/summary of the currently selected load case.

Depending on which options are checked, the load tabs will activate/de-activate accordingly.

9.3.3 Mass Tab

The Mass tab allows the user to add previously defined groups to the load case definition as well as redefine the values of these groups. Properties of masses can be added using six options: as volumes, as scaled-up structural mass, as sections, as various groups of point masses, and as large solid masses whose centroid is at an appreciable distance from the supporting nodes, such as main engines and bays of containers. For each load case, any combination of the previously defined groups (Volume, Module, Section, Node, and Bay) can be chosen, and specific properties can be assigned if they differ from the default value. MAESTRO will use these values to calculate all of the inertia forces in all members throughout the structure, and apply these as loads.

NOTE:

In non-SI unit systems that specify densities in force units, the masses should be defined in terms of weight; for example, the fps or ips unit systems use pounds as a weight where mass is specified. Therefore, when using a unit systems that specifies density in terms of force (this can be found in the Units dialog), mass should be interpreted as weight.

© 2014 Optimum Structural Design, Inc.

MAESTRO will automatically convert these weights to mass internally by dividing by gravity.

NOTE:

In a half model all specified values of mass should be half values. For a half model, all masses (except for Bay Set) are assumed to be symmetric and there is no need to define the corresponding mass in the un-modeled half.   For example, a plate group defined in the modeled side will be assumed to be included in the un-modeled side as the same mass/weight.

Each method for defining a mass is described in more detail below:

Volume

Module

Section

Node

Plate

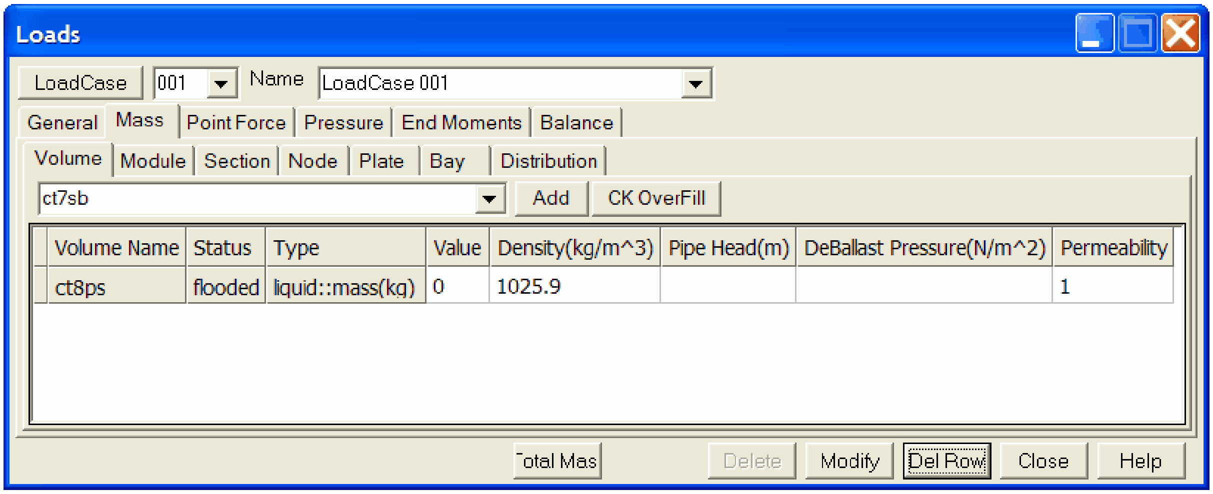
Bay

Distribution

Volume Mass

The Volume mass tab is for the user to add "tanks" or volume groups to the current load case definition. These previously defined volume groups are made available to the user via the drop-down menu. After a volume group is added the user can specify the parameters of the volume contents.

© 2014 Optimum Structural Design, Inc.



Once a group is selected from the drop-down menu, click *Add* to add the tank group to the dialog.

Volume Status
The volume status can be set to either *Intact* or *Flooded.* The *Flooded* status will assume the free surface of the fluid in the tank is at the equilibrium waterplane. If the flooded tank is fully submerged, the tank is fully filled. The fluid is assumed to have the same properties as the environment defined in the Job Information dialog if the original content's density is less than the environment density, otherwise the original content's density is used. When performing the hydrostatic balance of the model, MAESTRO will use the "added mass" method to calculate the model's new equilibrium.

Volume Type
The tank load "Type" can be defined three ways: mass, fraction of the volume filled, or the pressure "head" measured parallel to the total acceleration vector, from the point of lowest pressure to the point of highest pressure. The fraction and head option will automatically calculate the mass/weight of the tank using the given density. The volume can also be loaded with "dry::mass" as the type. This option distributes the pressure as does liquid mass, but the "free surface" remains fixed despite any trim or heel of the ship.

Volume Value
The value of the mass, fraction, or head is entered into the *Value* column.

Volume Density
Enter the density of the volume content in the units specified.

Volume Pipe Head
An optional value of the additional "head", measured as the height of fluid in a pipe for this volume, can be added by entering a value into the *Pipe Head* column. Using the fluid density of this volume, MAESTRO will calculate and add a constant pressure throughout the volume. The height is constant and assumed as defined. The additional "Pipe Head" option is to be used when the tank volume is filled to the tank top and a pressure head of a certain height is known to be applied above the tank top.

Volume DeBallast
Alternatively, if the pressure head is unknown, "DeBallast Pressure" can be applied to simulate deballasting a tank or to "press up" a tank. This pressure will be applied as a constant over all boundaries of the tank.

Volume Permeability
A permeability can be added by entering the decimal percent of space that can be occupied in the "Permeability" column. The permeability will multiply the density to adjust in the space occupied by the fluid volume.

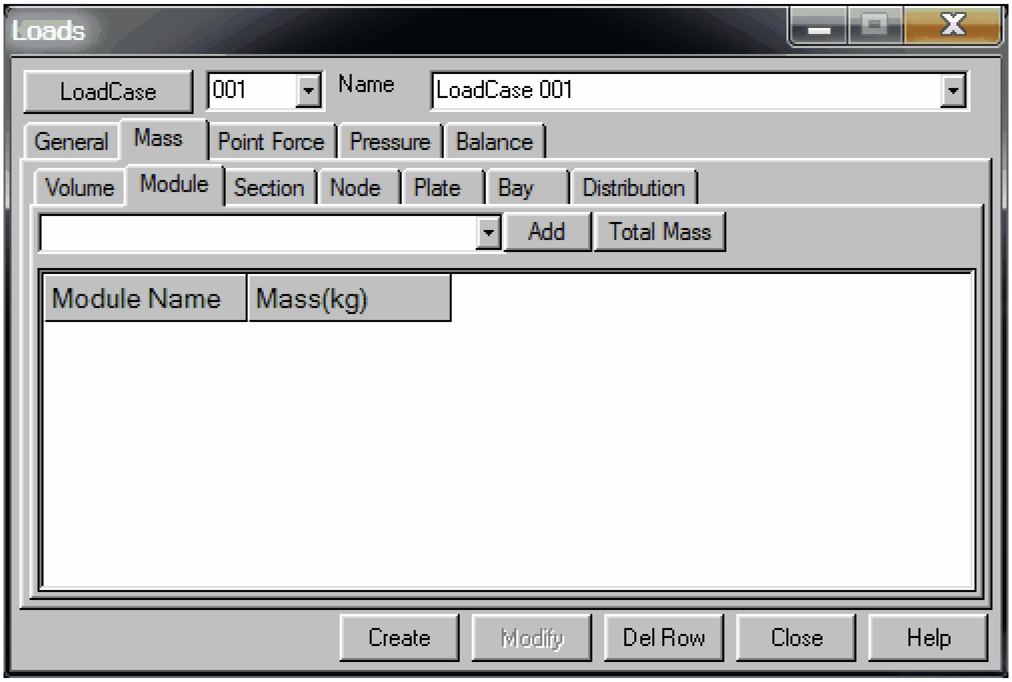
The *Total Mass* button will report the total mass of the defined volume groups based on their loading.

The *CK OverFill* will check that no tanks are overfilled based on the density and loading parameter.

Module Mass

Mass that is not of the load-carrying structure but whose spatial distribution approximates that of the structure is usually represented as a scaled version of the structural mass. This offers the user a method to generate a specified amount of additional (non structural) mass and allocate it to a group of specific modules in proportion to their structural mass. Here the user can add Scaled Mass groups, that were previously created using the Groups dialog, to the current load case definition. These previously defined scaled mass groups are made available to the user via the drop-down menu. In each load case a new value can be specified for the amount of mass to be added, and this overrides the value given in the scaled mass group dialog.

© 2014 Optimum Structural Design, Inc.



Once a module group is selected from the drop-down menu, click *Add.* The Mass column displays the previously defined Mass and can be overridden here, if desired.

The *Total Mass* button will report the the total mass of all the module groups in the current load case.

Section Mass

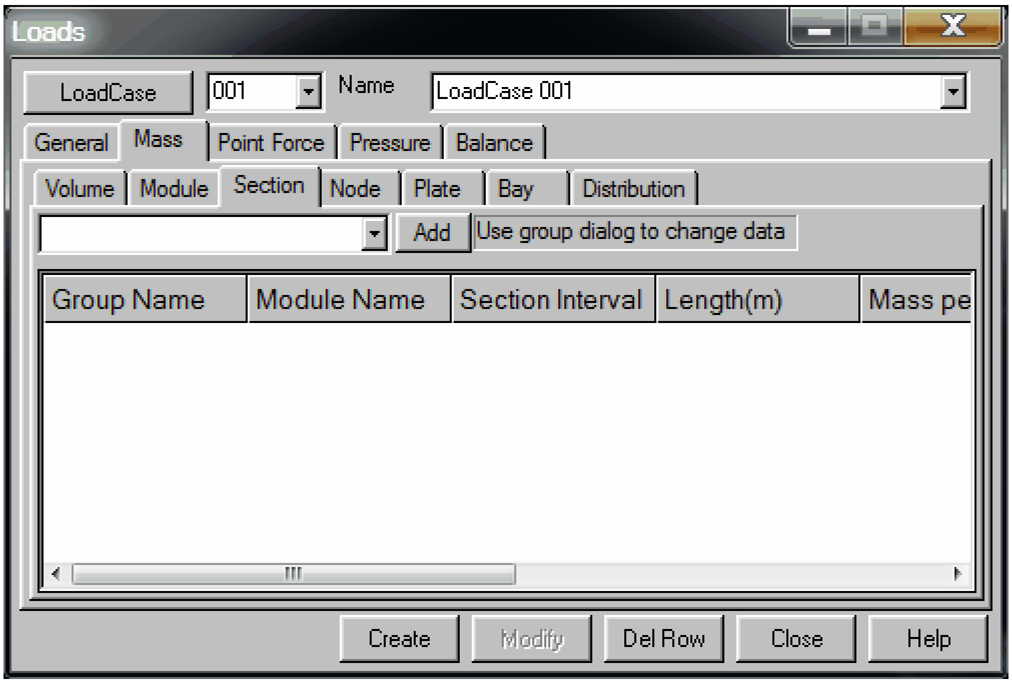
In the design of long structures such as ships, where overall bending is the dominant load effect, the lengthwise distribution of mass must be accounted for early in the design, and should be modeled as accurately as possible at all stages of the design. Ideally, in a three-dimensional model, the masses should be placed at their actual locations, using whichever of the methods presented here (volumes, bays/sets, scaled, nodal, plate) is most appropriate for each type of mass.

But at early stages of design some masses may be known (or estimated) only in the form of a one-dimensional distribution along the length of the structure, and the Section mass option is intended for this (and only this, because it is very approximate).

Here the user can add Section Interval groups that were previously created using the

Groups dialog to the current load case definition. These previously defined section groups are made available to the user via the drop-down menu.

Note: The values of the section groups can only be changed via the groups dialog.

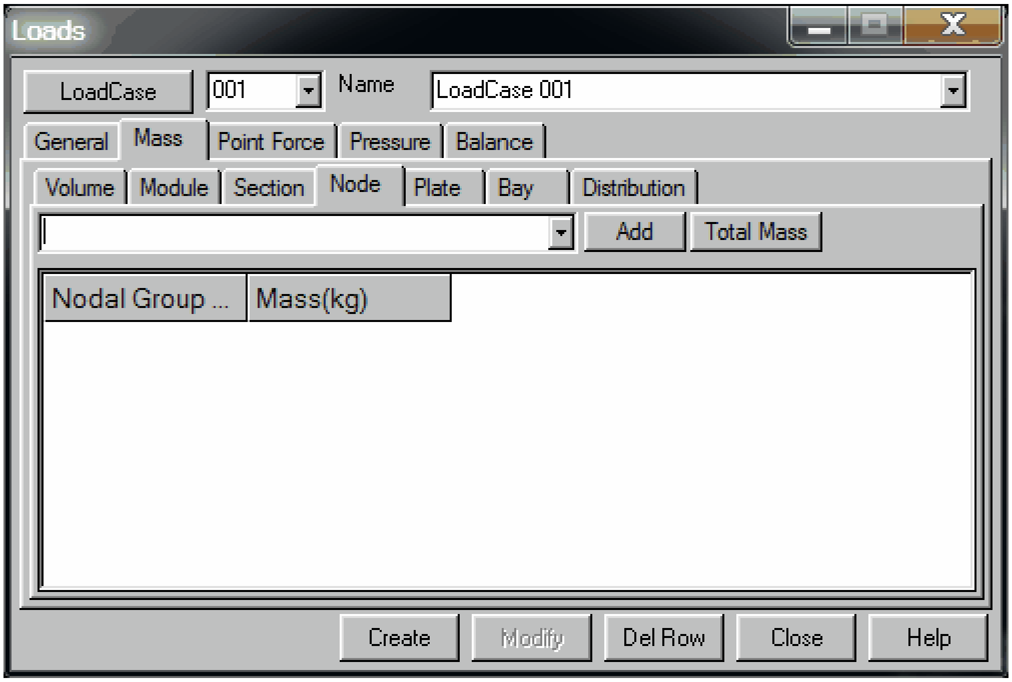


Once a section group is selected from the drop-down menu, click *Add.* The section interval, length, mass/length and mass values are updated from the section definition, but these values can only be changed using the groups dialog.

Node Mass

The Node mass option offers the user a method to generate a specified amount of additional (non structural) mass and allocate it equally among the nodes defined in the nodal group. The user can add Nodal groups, that were previously created using the Groups dialog, to the current load case definition. These previously defined nodal groups are made available to the user via the drop-down menu. The default value of total mass to be added can be overridden in this dialog.

© 2014 Optimum Structural Design, Inc.



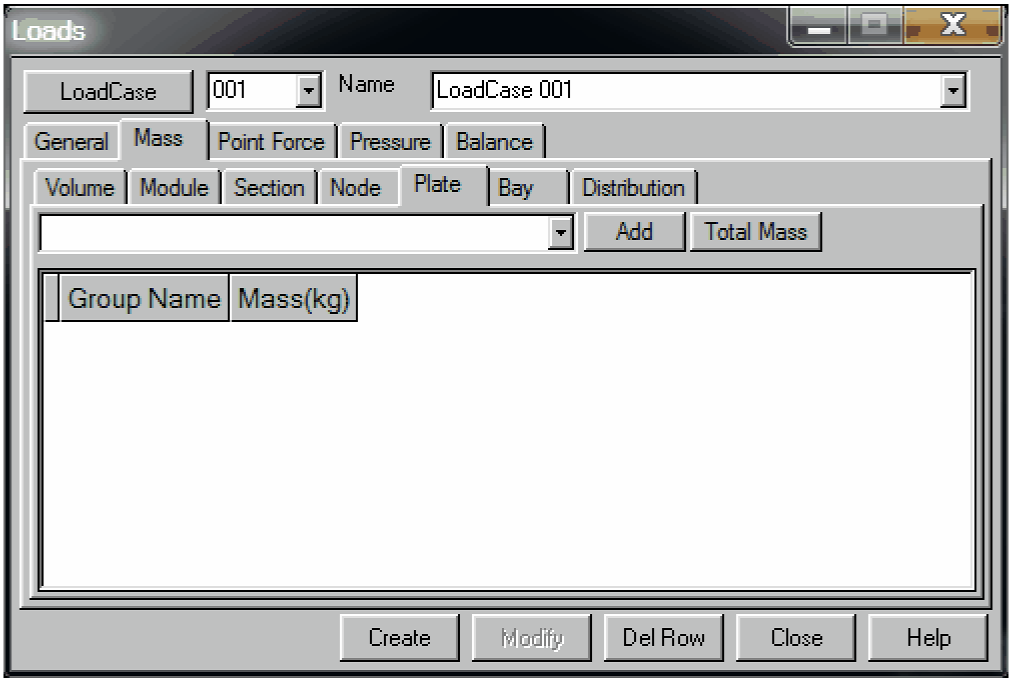
Once a node group is selected from the drop-down menu, click *Add.* The *Mass* column displays the previously defined mass and can be overridden here, if desired.

The *Total Mass* button will report the the total mass of all the node groups in the current load case.

Plate Mass

The Plate mass option offers the user a method to generate a specified amount of additional (non structural) mass and allocate it equally among each plate element that defines the plate group. Here the user can add Plate groups, that were previously created using the Groups dialog, to the current load case definition. These previously defined plate groups are made available to the user via the drop-down menu. The default value of total mass to be added can be overridden by another value.

© 2014 Optimum Structural Design, Inc.



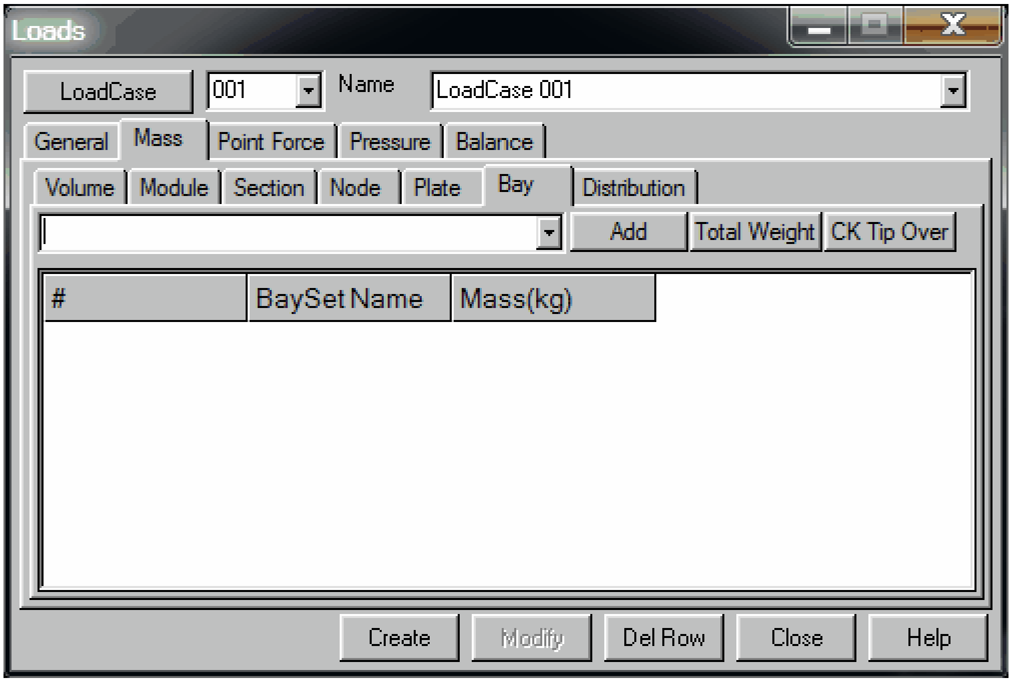
Once a plate group is selected from the drop-down menu, click *Add.* The *Mass* column displays the previously defined mass and can be overridden here, if desired.

The *Total Mass* button will report the the total mass of all the plate groups in the current load case.

Bay Mass

The bay mass option is intended for large solid masses that are supported at several nodes and whose center of mass is an appreciable distance from these nodes. Here the user can add Bay groups, that were previously created using the Groups dialog, to the current load case definition. These previously defined bay groups are made available to the user via the drop-down menu. The default value of mass to be added can be overridden by another value, if desired.

See the ...\MAESTRO\Models and Samples\Loads\Bayset directory for sample model data.



Once a bay group is selected from the drop-down menu, click *Add.* The *Mass* column displays the previously defined mass and can be overridden here, if desired.

The *Total Weight* button will report the the total weight of all the bay sets in the current load case.

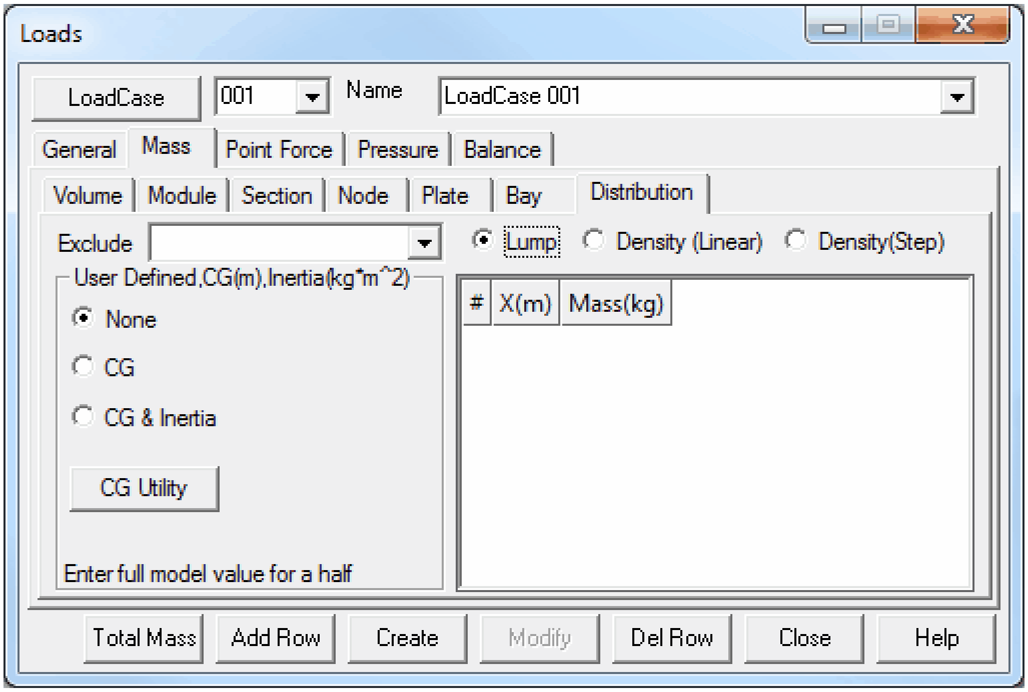
The *CK Tip Over* button will check to see if any of the bay sets are tipped over.

Mass Distribution

The mass distribution option is intended to allow the user to input a mass value or weight density at defined longitudinal coordinates. Users can choose to use linear interpolation or a step function for defined weight densities. It also allows the user to define the transverse and vertical center of gravity and inertias for the total mass distribution weight (the specified center of gravity and inertia is only for the weight represented by the distribution).

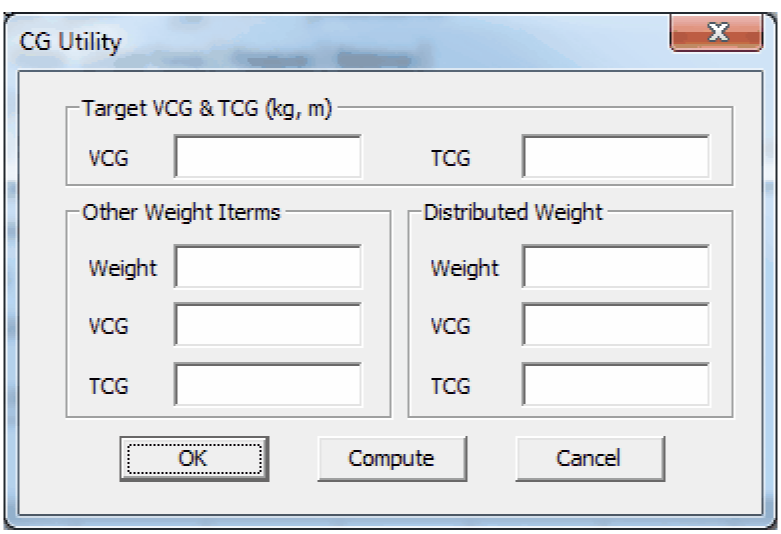
First, the user can *toggle* between the modes of Lump, Density (Linear), or Density (Step), by left-clicking on the Weight check box. Second, the user can *toggle* between Specifying CG, Specifying CG and Inertias, and No Specification of CG. Finally, the user can select particular nodal groups (via a drop-down) to Exclude from the mass distribution load.

© 2014 Optimum Structural Design, Inc.

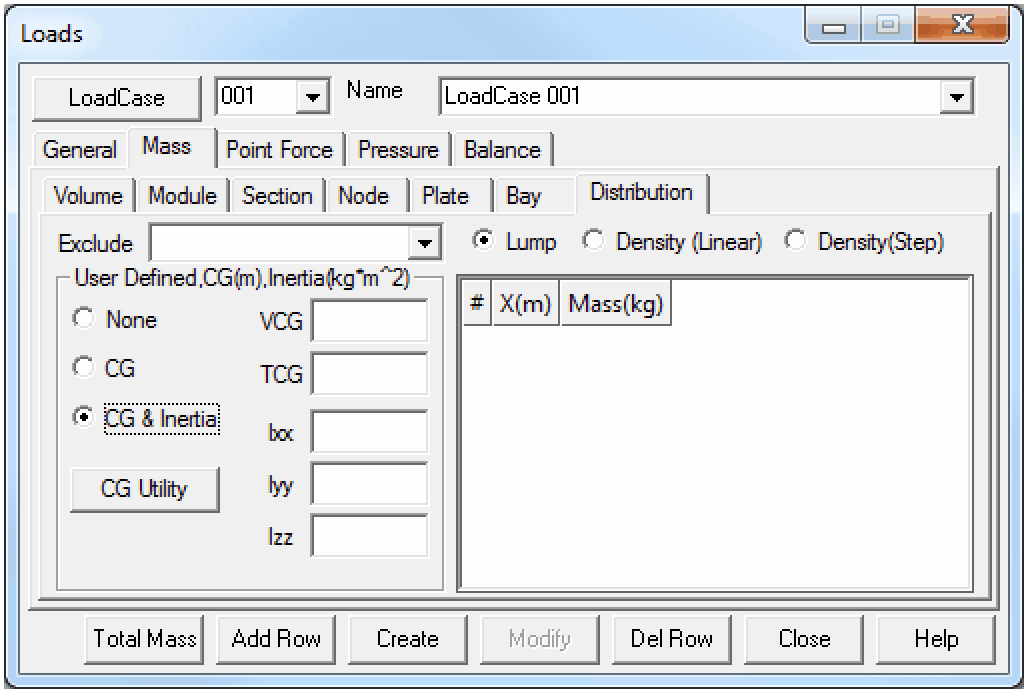


Click *Add Row* to add a new row of mass or mass density input. Enter the x-location of the mass in the "X(units)" column. Clicking the button next to "CG" or "CG & Inertia" will allow the user to enter the transverse and vertical CG or CG and Inertias for the total mass distribution weight. Note: if additional loads are imposed on the model, the total model CG will be different from the mass distribution CG.

The CG Utility allows the user to have MAESTRO automatically calculate the user-defined VCG and TCG for the weight distribution in order to reach a target VCG and TCG for the entire model. The user specifies the Target VCG and TCG, the weight and CG of any other loads in the load case, the weight distribution's weight, and clicks "Compute" to determine the user-defined VCG and TCG for the weight distribution.

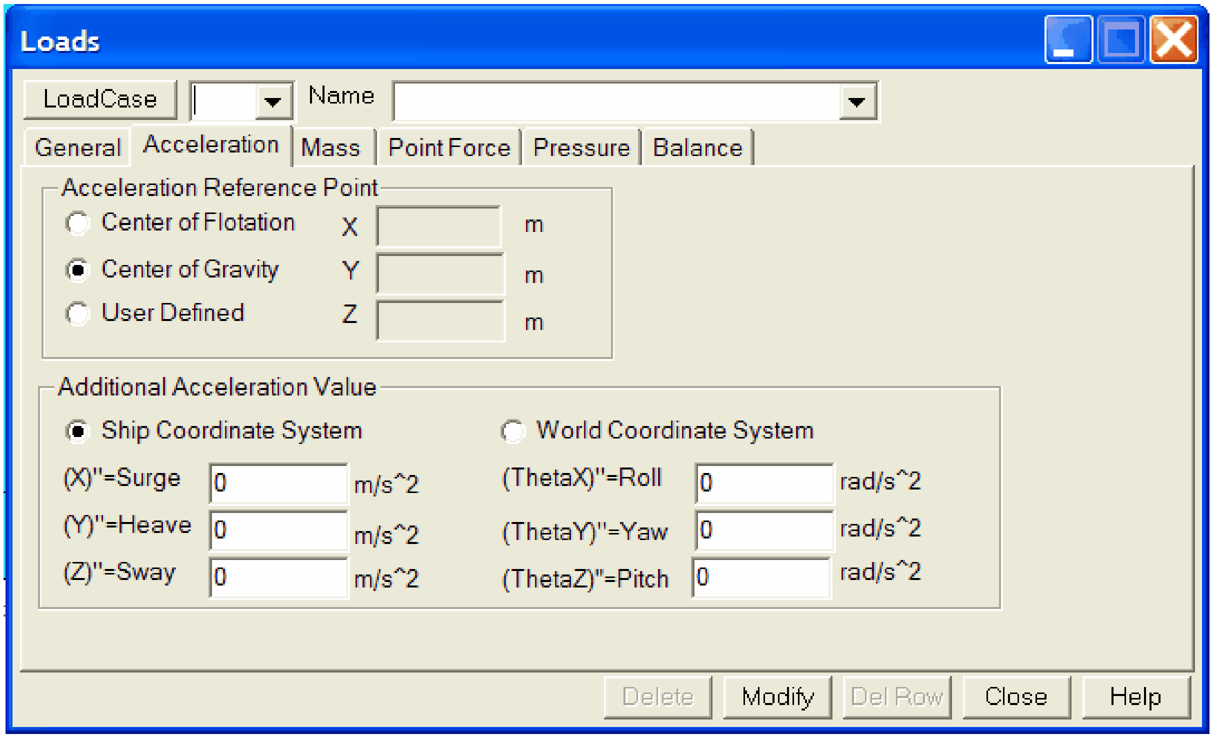


Mass distribution data can be copy and pasted from another program, such as Microsoft Excel. Before pasting the data, there needs to be enough blank rows to accommodate the data in the input window.



The *Total Weight* button will report the the total weight of all the mass distribution rows entered in the current load case.

9.3.4 Acceleration Tab

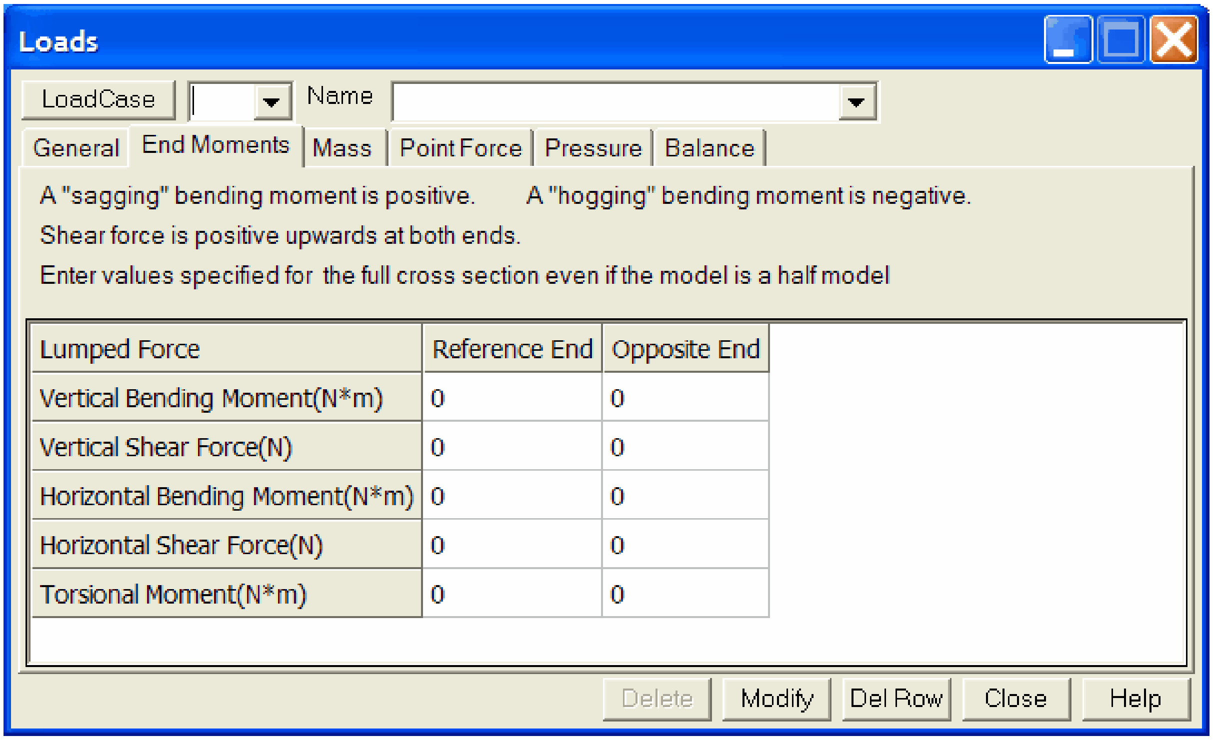


The Acceleration tab can be used to apply an additional acceleration to the model. The acceleration can be applied at the center of flotation, center of gravity, or at a user defined location (The center of flotation and center of gravity values are automatically calculated). The acceleration can be applied to all 6 degrees of freedom in the ship or world coordinate system.

The acceleration units and reference point length units are regarded as those defined in the Units dialog.

© 2014 Optimum Structural Design, Inc.

9.3.5 End Moments Tab



The End Moments tab can be used to apply a prescribed bending moment, shear force, or torsional moment to a "cut model" only. The "cut model" option can be selected in the Job Information dialog. The model boundaries are defined by the Start and End Modules in the Restraints dialog.

The force and length units are regarded as those defined in the Units dialog.

Even if only half of the model is defined, the values for the full cross section should be applied. If a bending moment is applied to the model, the MAESTRO balance command can be used to calculate the shear force. Please see the End Moments section of the Balancing the Model documentation.

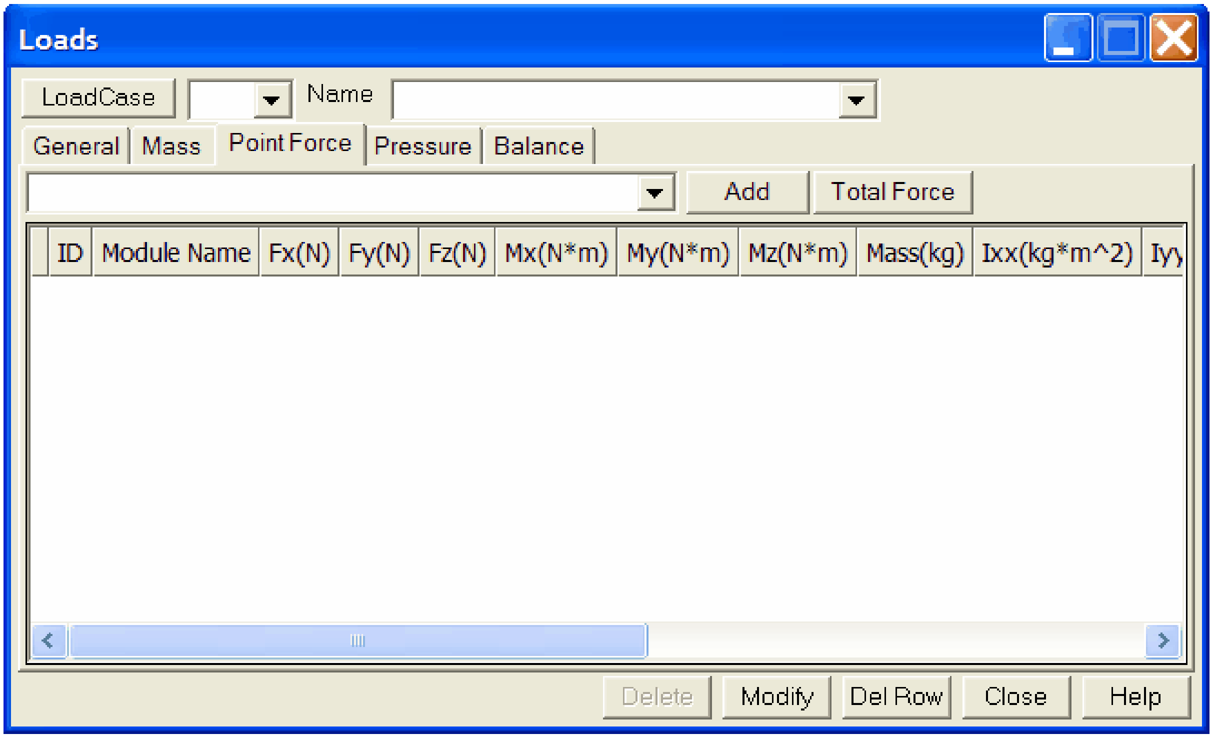
A "sagging" bending moment is positive and a "hogging" bending moment is negative. Shear force is positive upwards at the reference and opposite end.

A horizontal bending moment is positive if the resulting curvature in the model's XZ plane has its concave side in the +Z direction.

© 2014 Optimum Structural Design, Inc.

The sign of the torsional moment applied to the reference and opposite end is governed by the right-hand rule about the model's X axis.

9.3.6 Point Force Tab



The Point Force tab allows previously defined groups, or a user selected node to have a force, moment, or combination of forces and moments applied to it. If a group is given a point force or moment, the force or moment is evenly distributed to the nodes in the group.

If there is symmetry of structure (a half model) and if the current load set is also symmetric, then any point loads that lie in the center-plane of the structure should be full values.

The inertia components are defined for the group and are used for dynamic analyses.

9.3.7 Pressure Tab

This tab provides five methods of defining the location and magnitude of pressure loads which are to be applied to the panels (quadrilaterals and triangles) of this model in the current load case. Within each panel all pressure loads are cumulative. Thus if two pressure loads of opposite sign are specified for the same panel, the final total load is the net value of the two. For a panel in a strake, pressure is positive when it acts on the side of the plating opposite from the transverse frames. If it is desired that the pressure acts on the same side as the frames, then the pressure should be made negative. If a panel element has been

© 2014 Optimum Structural Design, Inc.

deleted by the Strake dialog deletion tab, Geometry and Load (rather than the Geometry Only) option, then the pressure load is not applied to that panel, even though it may lie within the range of sections specified in the Plate group.

Each method for applying a pressure load is described in more detail below.

Plate(LinPress)

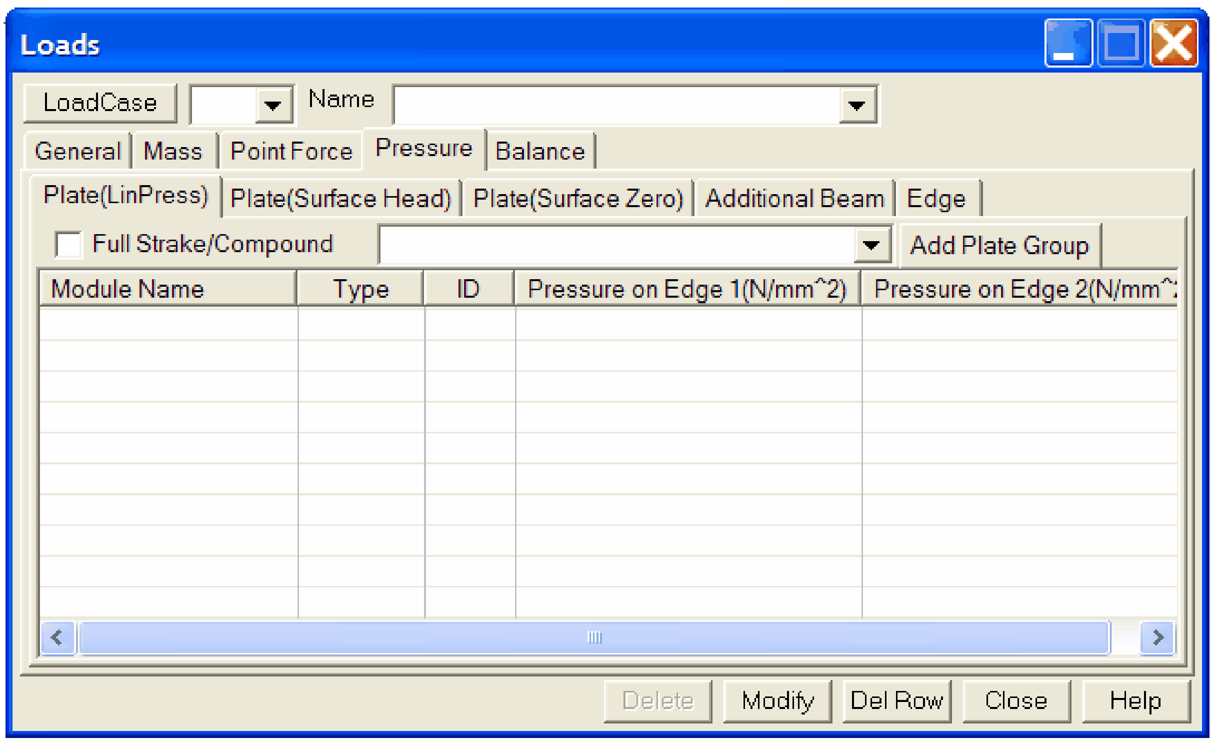
Plate(Surface Head)

Plate(Surface Zero)

Additional Beam

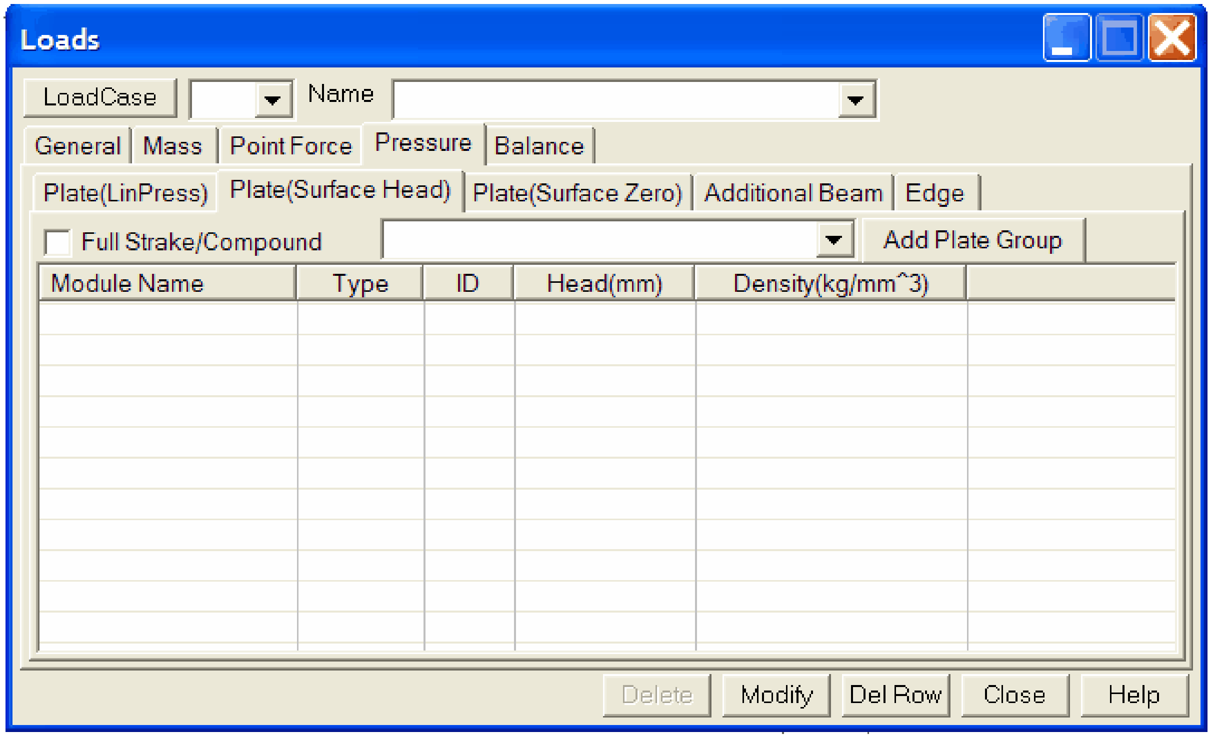
Edge

Plate(LinPress)



The Plate(LinPress) pressure loading option is that of an "actual" pressure. This pressure can be added to a predefined plate group (this will result in only being able to define a constant pressure) via the drop-down arrow just to the left of the Add Plate Group button or alternatively, can be added to a strake. If a pressure load is to be applied to panels of the strake, the user can define the pressure along the strake edge 1 and strake edge 2. Using this option, the pressure can vary linearly across the strake width.

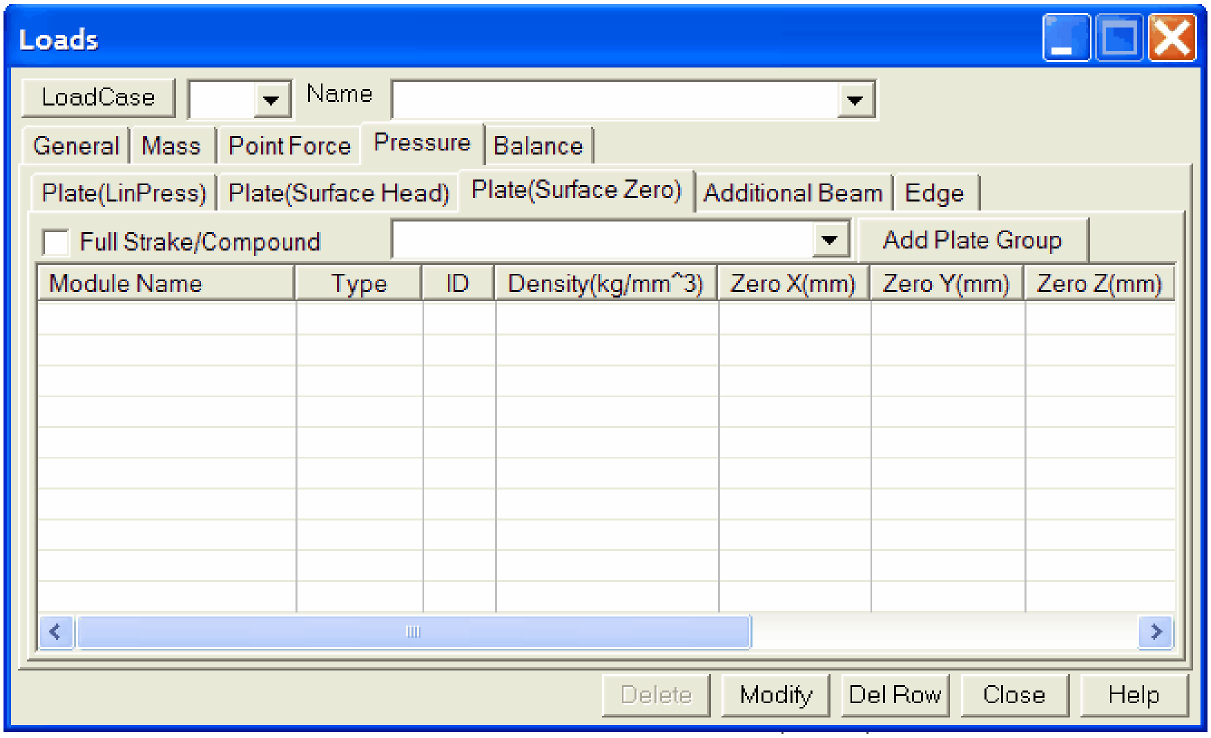
Plate(Surface Head)



The Surface Head option is intended for hydrostatic pressure, for which the value is proportional to the depth below the free surface of a fluid (or other zero pressure plane). In the Surface Head option the pressure is always an actual pressure. For strake panels, the pressure varies linearly across the strake width, in proportion to the local depth below the zero pressure surface, and in the lengthwise direction it is constant over each panel and is calculated separately for each panel, based on the depth of that panel below the zero pressure surface. For additional (non-strake) panels and triangles, the pressure is calculated at each corner of the element and then multiplied by either one fourth or one third of the element area.

NOTE:
The words depth, height, and head always mean normal to the zero pressure surface, which is also the direction of the total acceleration. If the current load case includes any heel or trim or other rotation of the structure, then the direction of the total acceleration is no longer parallel to the structure Y axis. If the current load case includes any dynamic accelerations (i.e. distinct from gravity) then the zero pressure surface will rotate so as to remain normal to the total acceleration vector, and the direction of the total acceleration will not be parallel to either the Y_{ref} axis or the structure Y axis.

Plate(Surface Zero)



The Surface Zero option is intended for hydrostatic pressure, for which the value is proportional to the depth below the free surface of a fluid (or other zero pressure plane). In the Surface Zero option the pressure is always an actual pressure, not a design pressure. For strake panels, the pressure varies linearly across the strake width, in proportion to the local depth below the zero pressure surface, and in the lengthwise direction it is constant over each panel and is calculated separately for each panel, based on the depth of that panel below the zero pressure surface. For additional (non-strake) panels and triangles, the pressure is calculated at each corner of the element and then multiplied by either one fourth or one third of the element area. Using the Surface Zero option, the user will be expected to define the zero pressure surface by specifying a point, in structure coordinates, through which the surface passes.

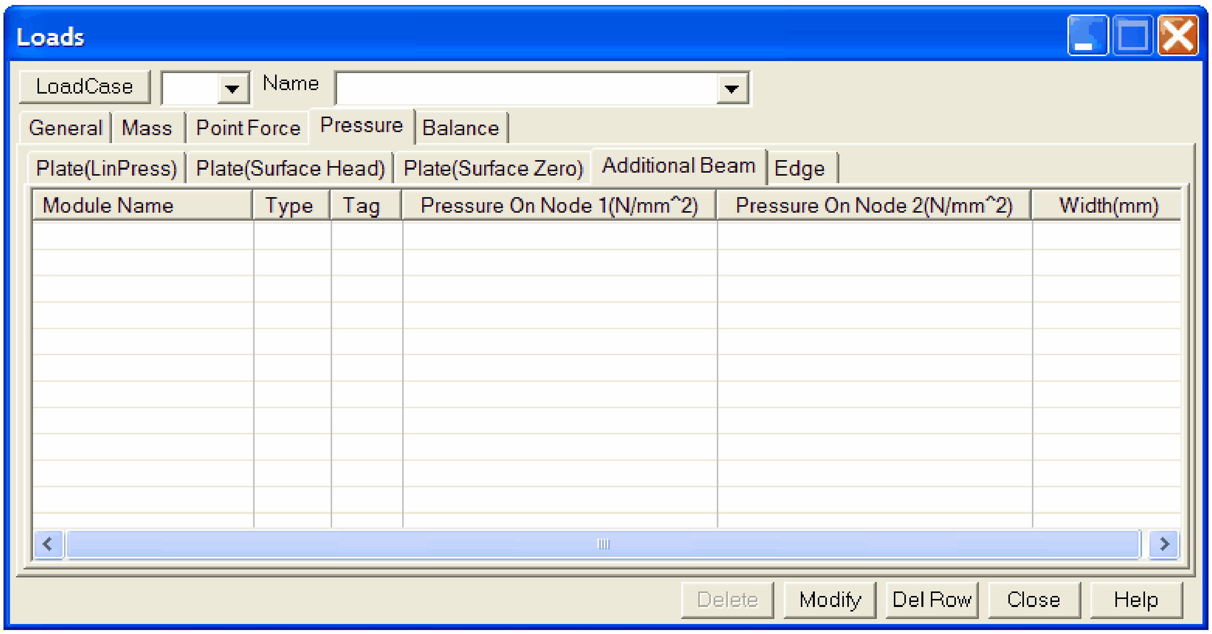
The Zero X, Zero Y, and Zero Z are the X, Y, and Z values, in structure coordinates, of a point through which the zero pressure surface passes. The surface will automatically be normal to the total acceleration vector.

NOTE:
The words depth, height, and head always mean normal to the zero pressure surface, which is also the direction of the total acceleration. If the current load case includes any heel or trim or other rotation of the structure, then the direction of the total acceleration is no longer parallel to the structure Y axis. If the current load case includes any dynamic accelerations

© 2014 Optimum Structural Design, Inc.

(i.e. distinct from gravity) then the zero pressure surface will rotate so as to remain normal to the total acceleration vector, and the direction of the total acceleration will not be parallel to either the Y_{ref} axis or the structure Y axis.

Additional Beam



The Additional Beam option allows the user to impose a pressure load to an additional beam. Positive pressure acts in the direction of the beam's Y axis, which is determined by the web orientation. This convention means that when the beam is attached to plating, as in a strake or on a compound, the pressure is positive when it acts on the "smooth" side of the plating; i.e. the side opposite the additional beam. For example, on a transverse bulkhead, if the additional beam faces forward, then a positive pressure would be applied to the aft side of the bulkhead, causing a forward force.

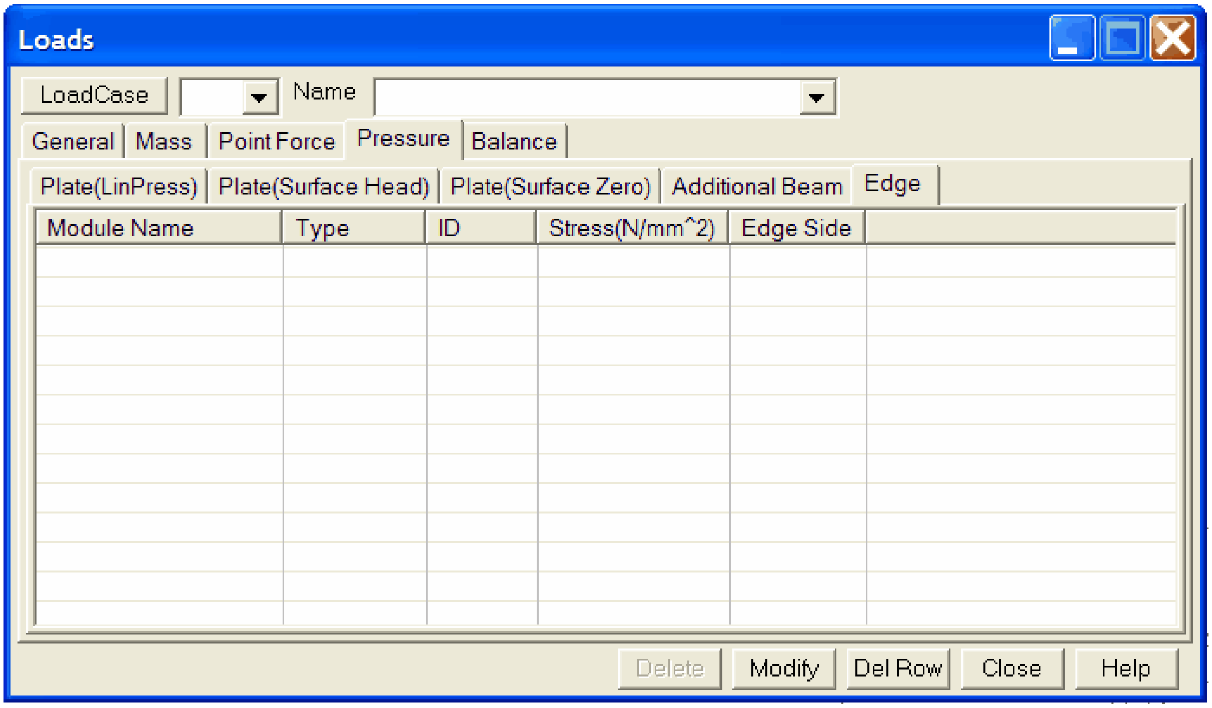
If an additional beam is a substitute for a strake based transverse frame element then it will automatically receive the strake related loads. If this is not desired then the frame element should be deleted (Delete Geom & Load) explicitly using the Deletion tab found in the Strake dialog. An additional beam will always receive whatever loads are defined for it here, in this data group.

The Pressure on Node 1 and Pressure on Node 2 columns define the pressure load value at the corresponding nodes.

© 2014 Optimum Structural Design, Inc.

The Width column defines the width of the pressure load. For vertical beams on the centerline of the overall structure, a half width should be used.

Edge

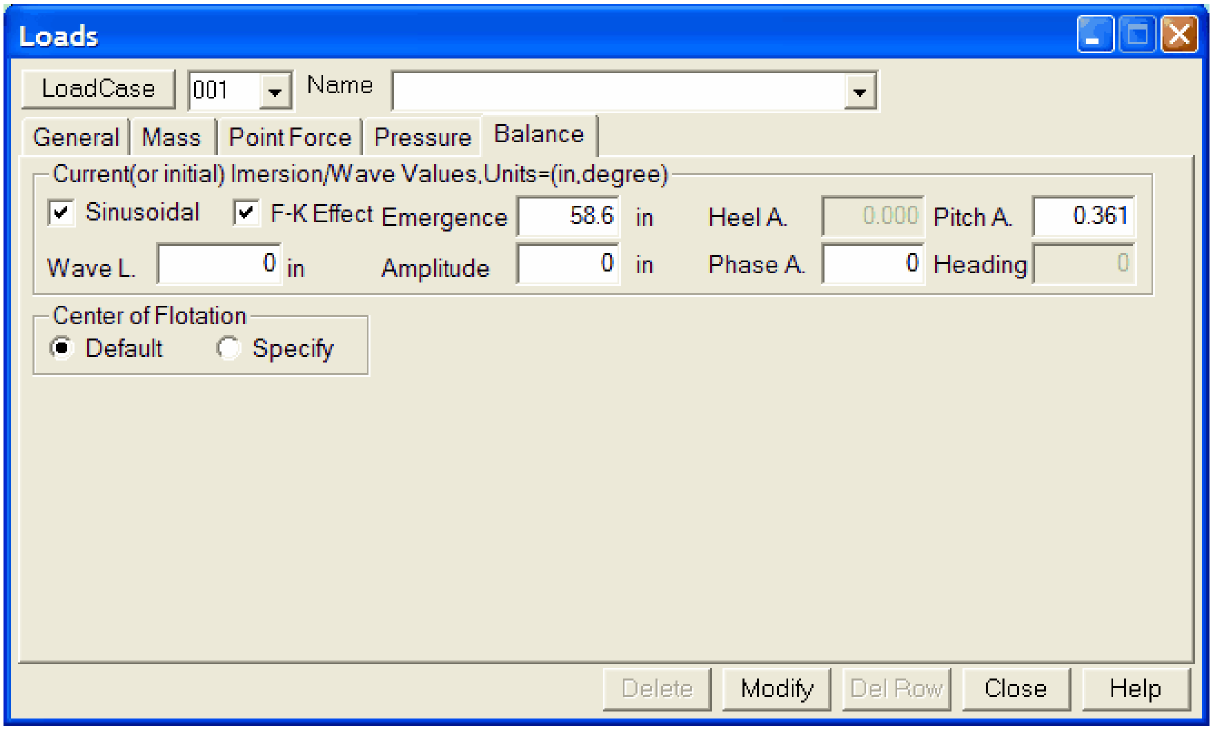


This pressure load option serves to impose a uniform in-plate stress at the lower or upper end of a strake panel, in the lengthwise direction; i.e. parallel to the X axis of the panel (or the mean plane panel if the strake is warped and/or skewed). At each end, the stress applied there is positive if it acts in the positive X direction, and negative if it acts in the negative X direction. Thus to impose a lengthwise compression on the panels of a strake, the stress applied at the lower end (smaller section number) would be positive, and the stress applied at the upper end would be negative. The intention is that the panel end coincides with a "cut" cross section of the model, and the stress comes from an adjacent portion of structure that is not being modeled. For greater flexibility in modeling, the stresses can be applied at any section along the strake; they are not limited to the reference and opposite ends of the strake. The program automatically converts the stress into equivalent nodal forces in the panel X direction and equivalent nodal moments about axis parallel to the panel Z axis. If the strake is warped and/or skewed, the axes are those of the mean plane panel. These forces and moments are then transformed to structure coordinates and applied to the model.

The Edge Side column defines the end (lengthwise) of the strake that will receive the specified stress. Lower represents the edge that corresponds to the smaller section number.

© 2014 Optimum Structural Design, Inc.

9.3.8 Balance Tab



The balance tab of the Loads dialog is used to define a stillwater immersion condition or wave condition if the load is defined as a floating structure. The first check box can be clicked to cycle through No Wave, Sinusoidal or Trochoidal. MAESTRO will automatically calculate and apply the Froude-Krylov forces, if desired, by clicking the check box F-K Effect. The Froude-Krylov effect is the fact that under a wave, due to the rotation of the water particles, the vertical pressure distribution is not linear – it approaches the hydrostatic pressure exponentially as you go deeper.

The trochoidal wave profile is calculated from the following equations.

$$x = L\frac{\phi}{2\pi} + h_w\frac{\sin\phi}{2}$$

$$y = h_w\frac{1-\cos\phi}{2}$$

The first line of inputs defines the models emergence, heel angle and pitch angle. The second line of inputs defines the wave length, amplitude, phase angle and heading. Note: all angles are defined in degrees.

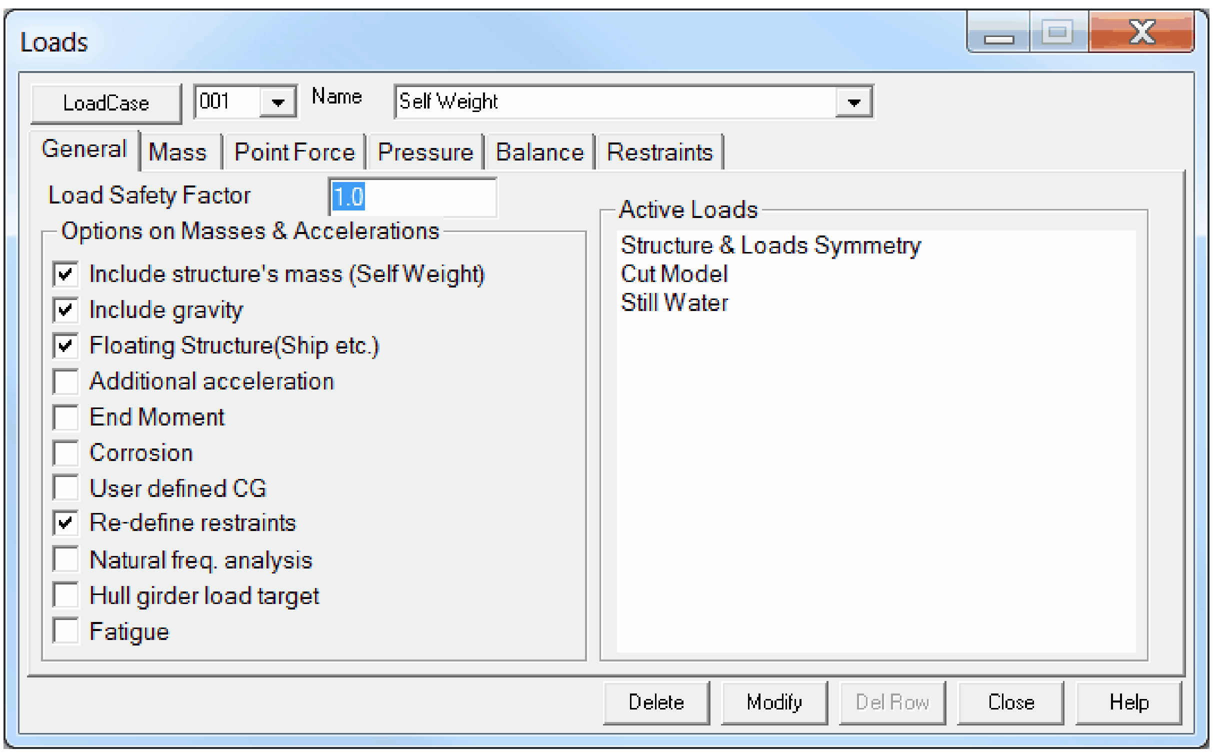
The phase angle is MAESTRO is defined as follows:

Assume the ship is stationary. For a positive phase angle shift, the wave moves forward compared to the original position.

The Center of Flotation will be automatically calculated by selecting Default, or it can be specified by the user. This functionality can be useful if the model will not balance and user wants to specify a center of flotation close to the expected in order to get the balance to solve.

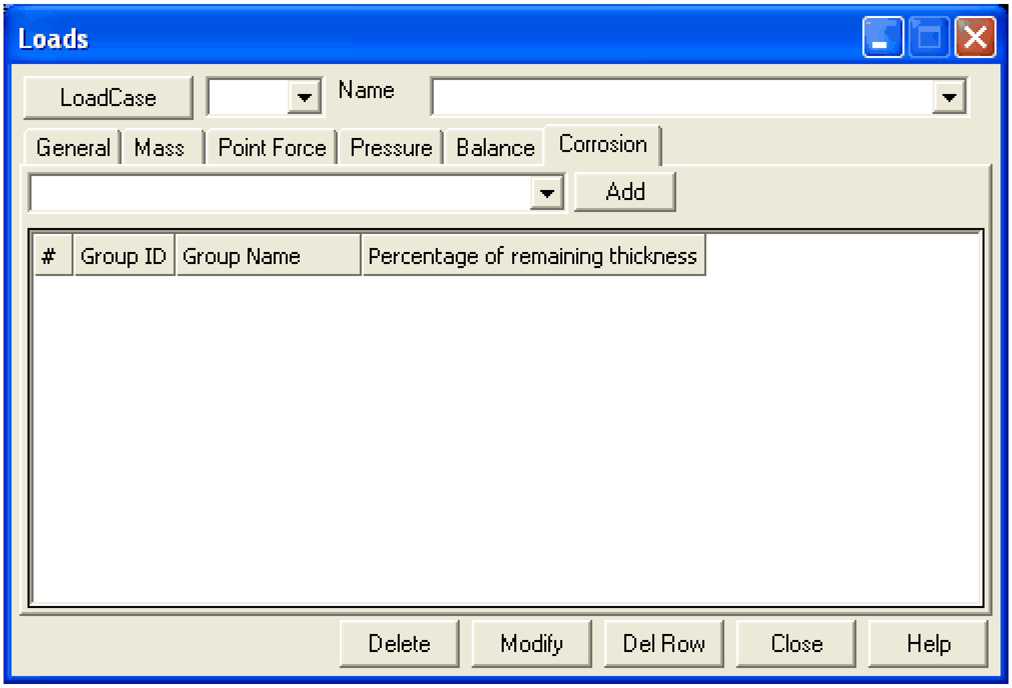
The Emergence (i.e., negative of immersion), Heel Angle, and Pitch Angle will automatically update if the MAESTRO balance function is used to place the model in an equilibrium position.

9.3.9 Restraint Tab



The Restraint tab is activated by selecting the Re-define Restraints box found under the General tab of the Loads dialog box. This allows the user to override restraints initially defined, if any, and have restraints specific to each load case.

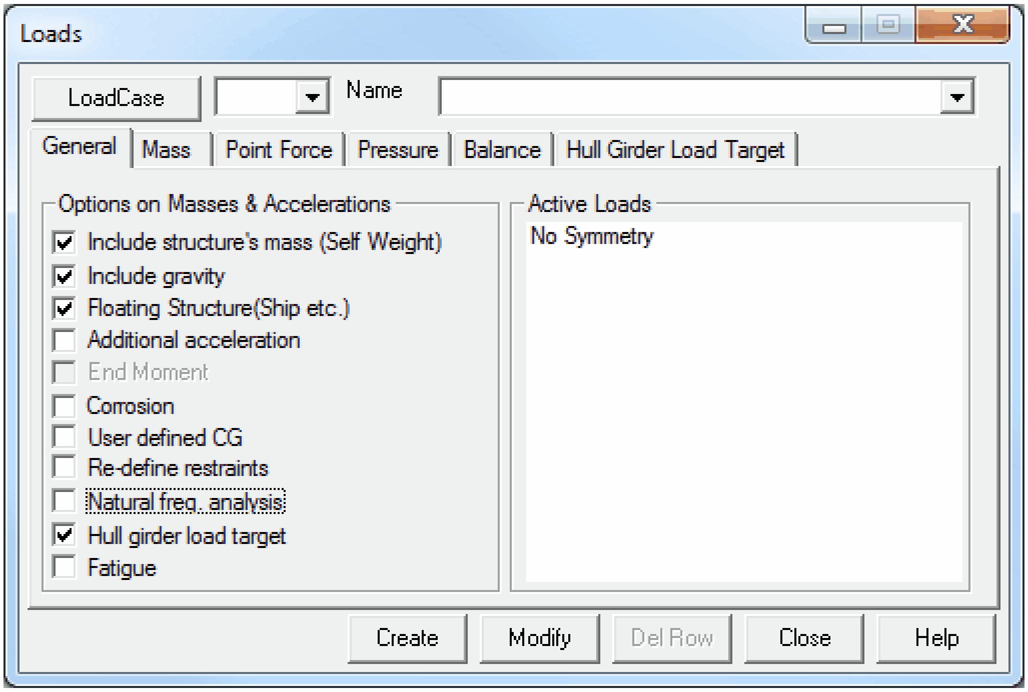
9.3.10 Corrosion Tab



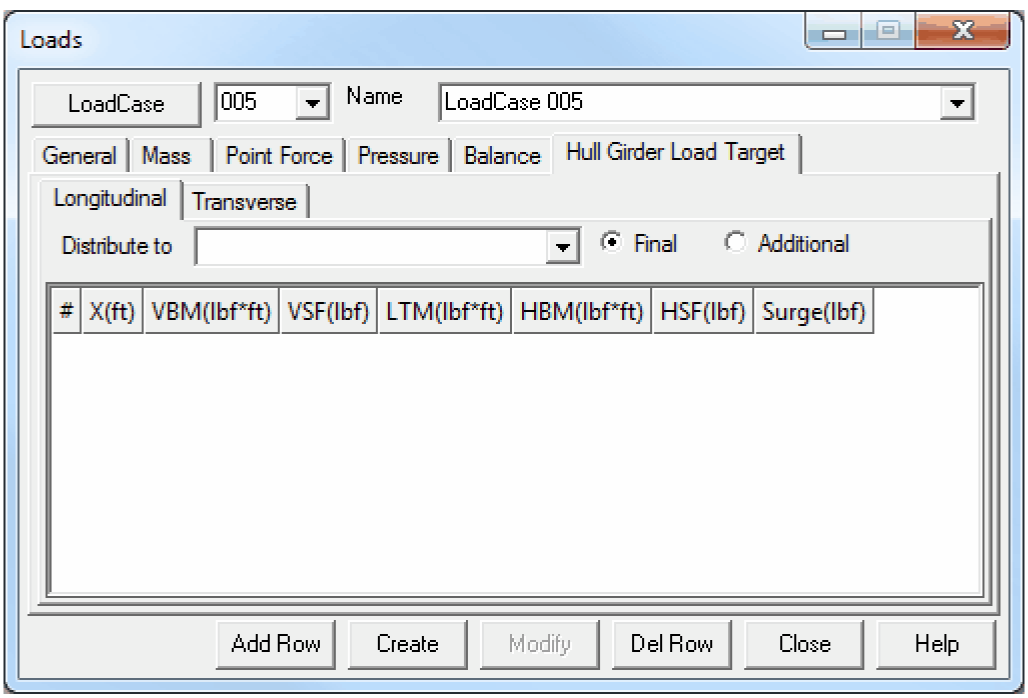
The Corrosion tab is activated by selecting the Corrosion box found under the General tab of the Loads dialog box. This allows the user incorporate the pre-defined corrosion groups into a particular load case. The "Percentage of remaining thickness" column allows the user to add or subtract thickness to the previously defined corrosion groups by multiplying the corrosion group's net thickness by a percentage. A percentage greater than 100% will increase the thickness of the elements in the corrosion group, whereas a percentage less than 100% will reduce the thickness. A percentage of 100% will keep the thickness as defined in the corrosion group.

9.3.11 Hull Girder Load Target

The hull girder load target can be used to ensure that either a minimum longitudinal or transverse loading envelope is met. The hull girder load target is used to supplement an existing load case. For example, if the intent is to impose an "Additional" bending moment in the form of a envelope, it is suggest the "base" loading condition (e.g., Full Load or Ballast Load) is established and balanced as a first step. The second step would then be to define the particular hull girder target "envelope." This second step would NOT be re-balanced as the intent is to provide "Additional" bending moment to the already existing (and balanced) load case. The hull girder load target dialog can also be used to add additional forces and moments on top of all other structural and hydro loads. The hull girder load target pattern can be applied by clicking the checkbox for "Hull girder load target" in the loads dialog:



This will enabled a new tab "Hull Girder Load Target".



The target load is to represent either an additional longitudinal or transverse loading envelope, but not both. Longitudinal or transverse coordinates may be assigned the target loads. The final or additional bending moment or shear force must be distributed to a general group of the user's choice. (See the Groups Menu section for details on creating Vertical Element Groups) Selecting "Final" will scale up all of the existing load patterns so that the total loading meets that defined in the Hull Girder Target Load. Selecting "Additional" will apply the specified loads on top of the existing loads from other load patters in the load case.

The available loads are:

VBM: Vertical Bending Moment

VSF: Vertical Shear Force

LTM: Longitudinal Torsional Moment

HBM: Horizontal Bending Moment

HSF: Horizontal Shear Force

Surge: Surge Force

9.4 Importing Hydrodynamic Loads

The following section is organized as follows:

Ship Motion File Definition

Importing the Ship Motion File

Reviewing the Ship Motion File

Ship Motion File Definition

Ultimately, the mapped pressures must be captured in MAESTRO's *Ship Motion* file (*.smn), which serves as the mechanism for incorporating the hydrodynamic loading component into the direct structural analysis. The *Ship Motion* file (*.smn) is a text file that can be generated by the analyst and must have the following items:

- Units [Keyword units with second-token key word /n-m/ips/fps/nmm/in lb/ft lb]: See Line 2 in the figure below.

```
1  % units: ips/fps/nmm/in lb/ft lb, if none unts=n-m
2  units n-m
```

© 2014 Optimum Structural Design, Inc.

- Loadcase Name
  - o [Keyword dynamicbaseload: The value represents the MAESTRO Base Load Case ID associated with the Hydrodynamic Analysis Output] See Line 3 in the figure below.
  - o [Keyword loadcase: The value represents the Seakeeping Analysis Load Case run] See Line 4 in the figure below.
- Keyword speed/heading/frequency: The value represents the speed, heading, and frequency for the particular loadcase.
- Displacement [Keyword displacement and displacementimag: Not currently required for response analysis] See Lines 9 & 10 in the figure below.
- Acceleration [Keyword acceleration and accelerationimag: Seakeeping Resulting Real and Imaginary Accelerations, respectively, at MAESTRO Loading Condition Center of Gravity. See Lines 12 & 13 in the figure below.

```
3  dynamicbaseload 1
4  loadcase 1
5  speed 2.57222217456722
6  heading 3.14159265358979
7  frequency 0.209439516523106
8  %              dx(surge)     dy(heave)    dz    rx(roll,degree)   ry(yaw, degree)    rz (pitch, degree)
9  displacement   -212.79533     -3.8847046E-07     100.43448     1.8489328E-07     0.038235227     5.738289E-07
10 displacementimag   -58.62785     -1.4451402E-05     0.042554483     6.7011694E-07     0.46015368     -1.400929E-07
11 %                 ax          ay               az             arx(rad/s^2)     ary(rad/s^2)     arz(rad/s^2)  coordinate-system(W==>world, S or none ==>Ship)
12 acceleration  -1.05887816931181     -1.93304472197072E-09     0.499766034894578     9.02555784361381E-09     0.00186645103030356     2.80114341055948E-08 W
13 accelerationimag  -0.291734553003054     -7.19107608881689E-08     0.000211752828668986     3.27117308100948E-08     0.0224623829259331     -6.83862914016997E-09 W
```

- Element Pressures (Keyword elmpressure): Element FETag (see Wetted Elements), and Pressures (Real and Imaginary). See figure below.

```
14 %            elm_ID     real_pressure    imag_pressure
15 elmpressure     10      -395.278107809915      -97.4389503834419
16 elmpressure     11      -392.433164173881      -88.7734367092311
17 elmpressure     12      -372.851282466803      -80.9700687195753
18 elmpressure     13      -347.249033696175      -73.4783741694714
```

Importing the Ship Motion File

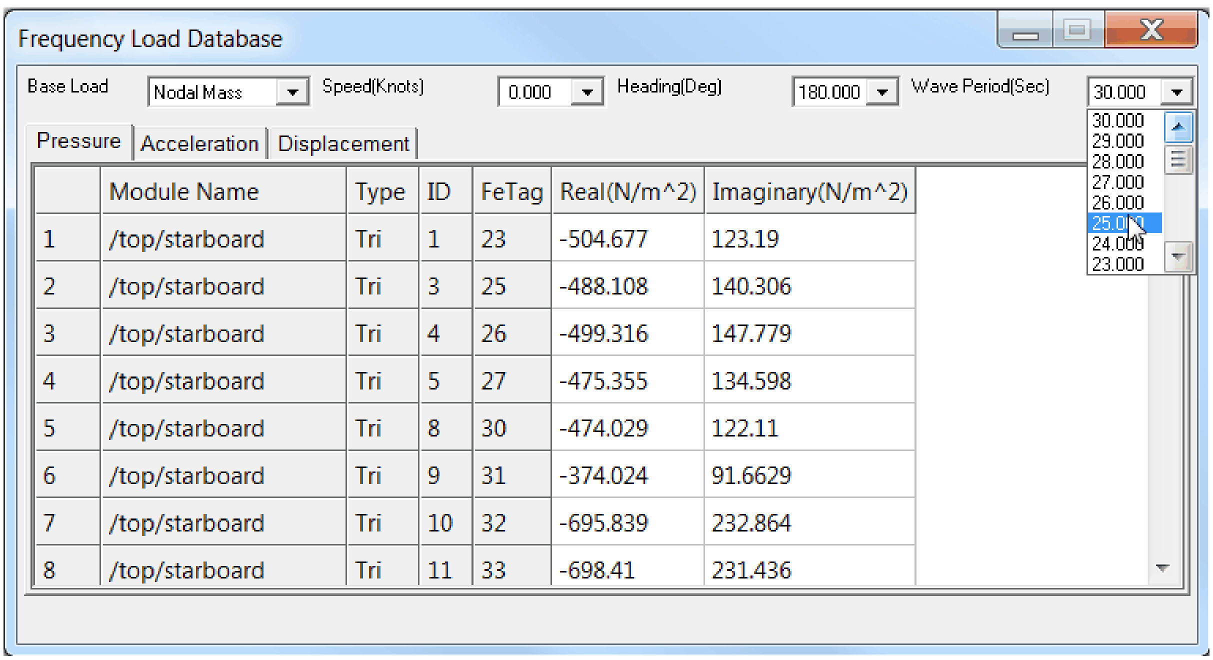
Before the Ship Motion file can be imported, the user must ensure the appropriate MAESTRO model (i.e., *.mdl file) is open. Note there is an example MAESTRO model located in the *Models and Samples/Extreme Load Analysis* directory. The file is named *S175-ITTC-2010-nd-mass-933.mdl.*

To import the Ship Motion file, click **File > Import > Ship Motion (*.smn)**. Choose the file of interest and click **OK**. Note there is an example MAESTRO Ship Motion file corresponding to the *S175-ITTC-2010-nd-mass-933.mdl* file and is located in the *Models and Samples/Extreme Load Analysis* directory.

Reviewing the Ship Motion File

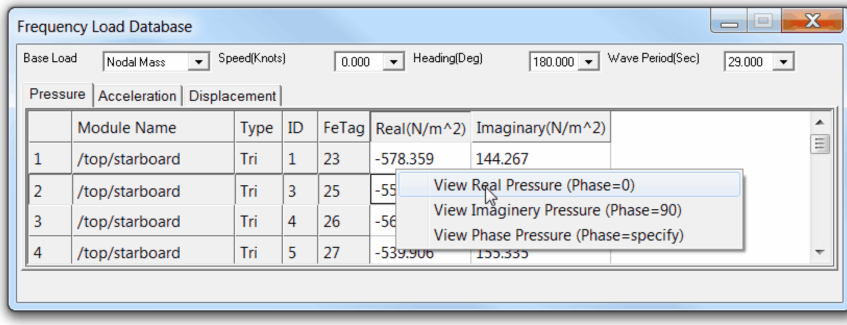
Once the Ship Motion file has been imported successfully, it is desirable to visually inspect the imported hydrodynamic loads to verify what ship motion data was imported. This can be done for each Base Load, Speed, Heading, and Wave Periods defined in the imported Ship Motion file.

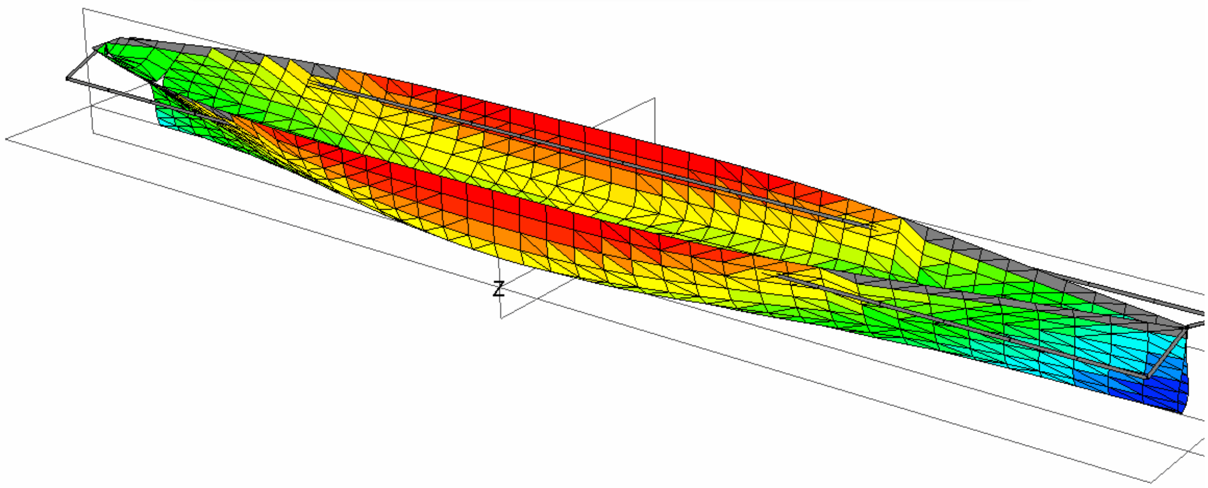
To visually inspect the imported Ship Motion file, click **ELA/SFA > Frequency Load Database**, which launches the dialog box shown below.



Base Load, Speed, Heading, and *Wave Period* cases can be chosen via their respective drop-downs. Once a particular case is chosen, the user can access the *Pressure, Acceleration,* or *Displacement* tabs to review the imported data.

Further, when reviewing the data presented in the *Pressure* tab, the user can right-click and chose to visually inspect the Phase 0, Phase 90, or a specific Phase angle complex pressures. This is shown in the figure below. Note that these pressures can be dynamically queried on the model by accessing the dynamic query icon, .





9.5 Balancing the Model

Toolbar	
Menu	**Model > Balance...**

How to Use the Balance Functionality

End Moments and/or Shear Force Balancing

Load Balancing Verification Samples

Balancing the model

MAESTRO provides two types of static equilibrium balance: hydrostatic balance and inertia relief balance. Hydrostatic balance occurs when the center of gravity and the center of buoyancy of a floating structure align vertically in earth coordinate system. As a result of hydrostatic balance, the floating structure has equilibriums in heave, pitch and roll, i.e.

$$\begin{cases} \sum F_{heave} = 0 \\ \sum M_{pitch} = 0 \\ \sum M_{roll} = 0 \end{cases}$$

To get equilibrium in surge, sway and yaw, inertia relief balance has to be used. Inertia relief balance automatically adjusts additional accelerations to get,

$$\begin{cases} \sum F_{surge} = 0 \\ \sum F_{sway} = 0 \\ \sum M_{yaw} = 0 \end{cases}$$

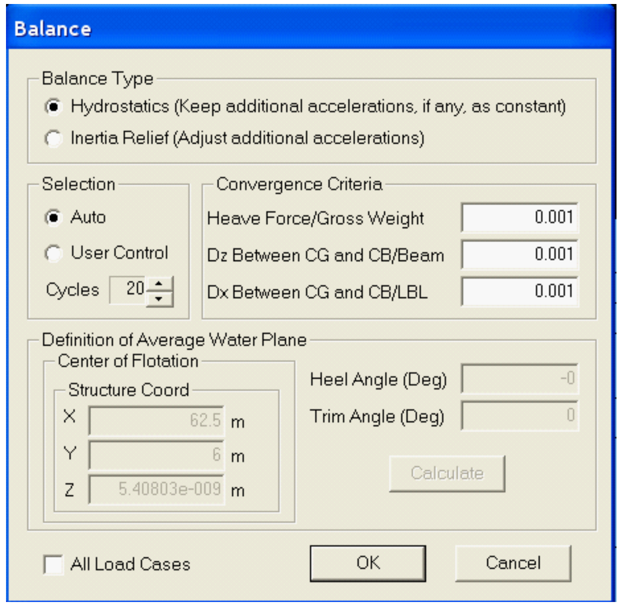
All reported resulting values (see image below) are for the *full ship* if the model is defined as having Transverse Symmetry (via the Job Information dialog).

```
***Information Only. No effect on FE-Analysis***
*Displacement= 1905.44 tonne, Volume=1857.34 m^3
*The following parameters are in the Ship Coordinate system:
*Center of Buoyancy: xCB= 32.6839 m, yCB=0.506099 m, zCB=0 m
*Center of Gravity:  xCG= 32.6839 m, yCG=3.86331 m, zCG=0 m
*Center of Flotation: xCF= 32.76 m, yCF=1.00716 m, zCF=0 m
*Trim Angle(Deg)=0.0121941
*BMT= 66.954 m
*BML= 357.718 m
*It=124356 m^4
*Il=664404 m^4
*GMT=63.5968 m
*GML=354.361 m

Balance Completed
```

How to use the balance tool?

1. Open the Hydrostatic Balance dialog from the **Model > Balance...** menu option, or the toolbar.



2. Run hydrostatic balance

   a. Select the balance type as "**Hydrostatics**"**.** This type is the default selection. For a floating structure model, user **should always run hydrostatic balance first**. In many cases, you only need to run hydrostatic balance. For a non-floating structure model, where a wave profile is not defined, this selection automatically switches to "Inertia relief".

   b. Check All Load Cases if you would like to balance all load cases, otherwise only the currently selected load case will be balanced.

   c. Select Auto balance or User Control to define the model center of floatation or heel and trim angles.

   d. Click OK to balance the model.

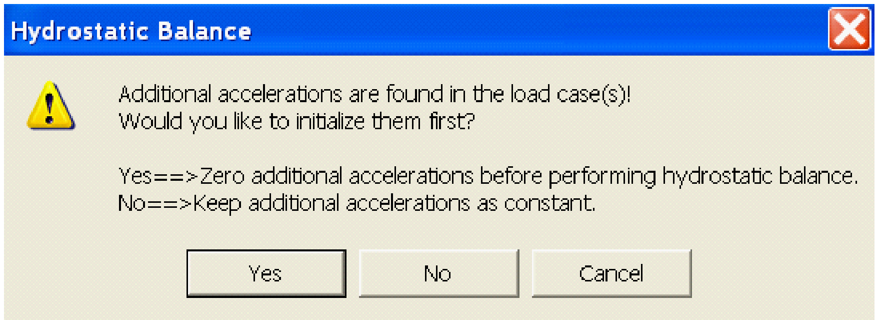
   e. Review the output tab to make sure the hydrostatic balance is successful.

3. Check bending moment and shear force distribution.

4. If either the bending moment or the shear force distribution has no closure, run inertia relief balance (**Optional**). Repeat step 2 with the balance type selection as "Inertia relief".

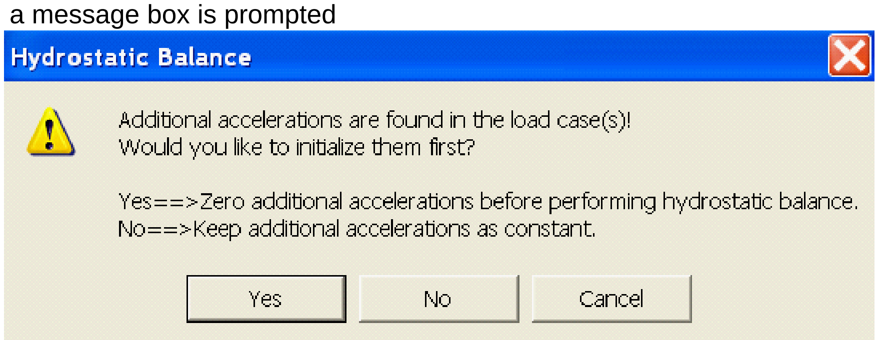
To correctly use the combination of hydrostatic balance and inertia relief balance, especially for the case of **re-balancing** the model, two common scenarios are given below,

Scenario #1: A floating structure without pre-defined (initial) additional accelerations
Step 1: Run hydrostatic balance for a selected load case.
Step 2: Run inertia relief balance. Additional accelerations are automatically defined for the selected load case.
Step 3: Check bending moment and shear force distribution to validate the equilibrium.
Step 4: If for any reason the load case is changed, the model has to be re-balanced. Again, we need to run hydrostatic balance first. Since additional accelerations are found in the load case, a message box is prompted

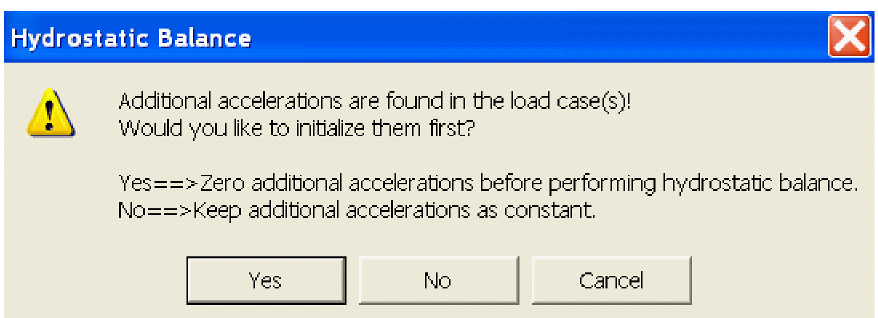


In this scenario, there are no initial additional accelerations, so we'll select "Yes" to initialize the additional accelerations due to previous inertia relief balance run.
Step 5: Repeat step 1~3.

Scenario #2: A floating structure with pre-defined (initial) additional accelerations. The initial accelerations may come from a third-party hydrodynamic analysis.
Step 1: Run hydrostatic balance for a selected load case. Since additional accelerations are found in the load case, a message box is prompted

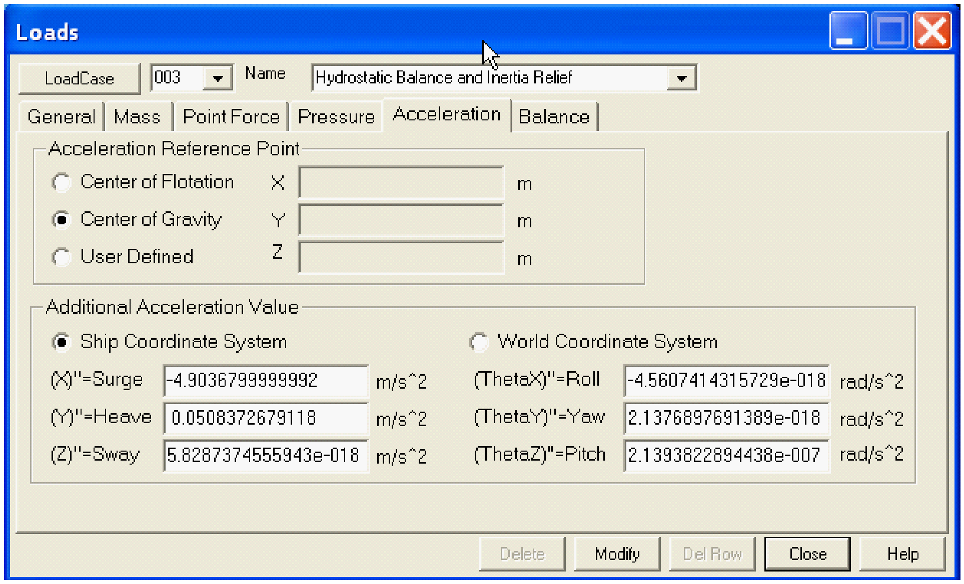


In this scenario, the initial additional accelerations are defined by the user, so we'll select "No" to keep the additional accelerations.
Step 2: Run inertia relief balance.
Step 3: Check bending moment and shear force distribution to validate the equilibrium.
Step 4: If for any reason the load case is changed, the model has to be re-balanced. Again, we need to run hydrostatic balance first. Since additional accelerations are found in the load case, a message box is prompted



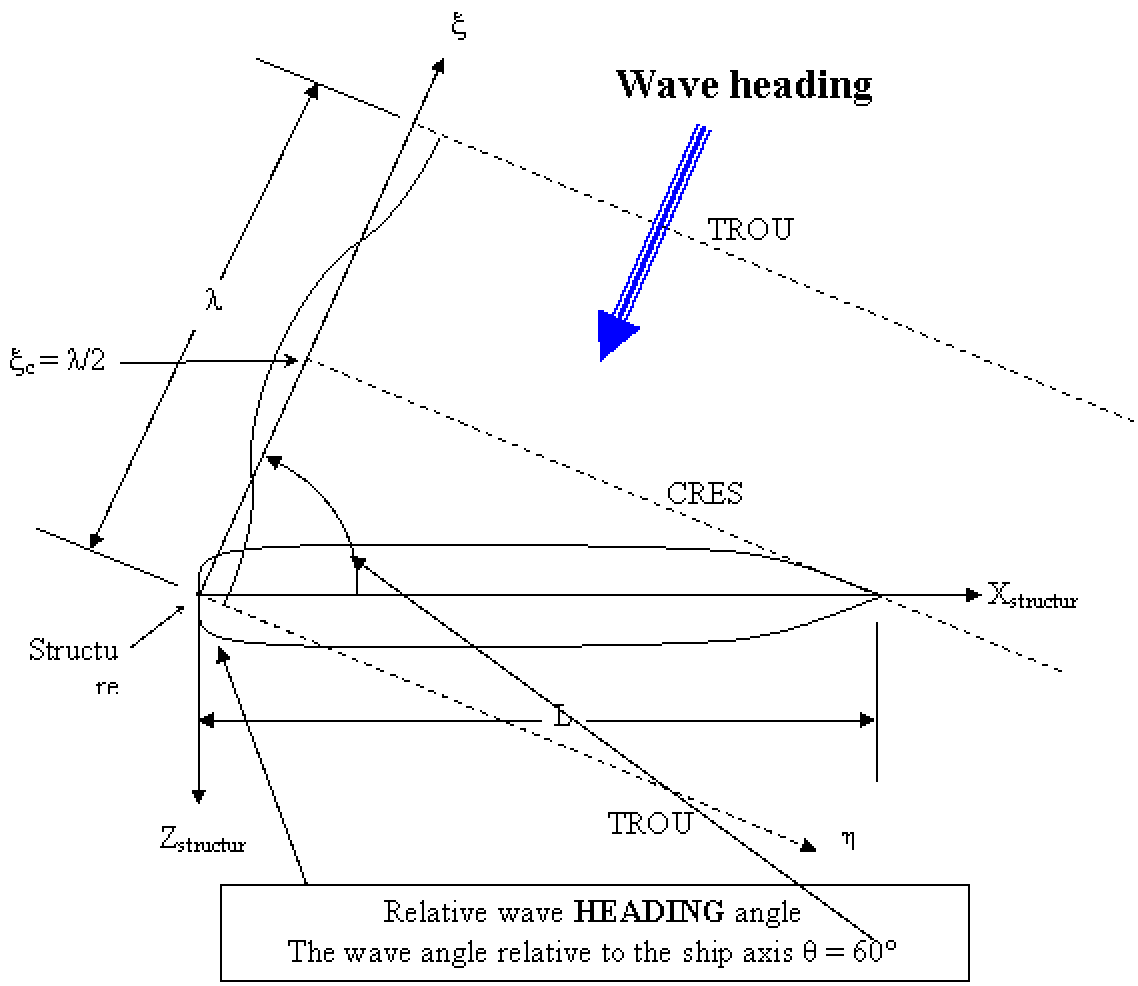
Because the additional accelerations have been modified by previous inertia relief balance, we'll select "Cancel" to stop the balance.
Step 5: Go to load dialog's additional acceleration page, and re-enter the user defined original accelerations.

© 2014 Optimum Structural Design, Inc.



Step 6: Repeat Step 1~3 to re-balance the model.

Balancing a Model in a Wave



L = Ship length.

l = Wave length. In the figure, l = L

a = Wave amplitude.

x = Wave axis, always measured from the Reference Origin

x_c = Location of a wave crest. In the figure xc = l/2

j = Phase angle. Moves a wave crest along the x axis away from the origin.

= (xc / l) ´ 360°. In the figure j = 180°, which places a trough at the origin.

q = The angle between structural coordinate and wave coordinate using the right hand rule.

To calculate the wave height in a ship structural location (x, z), the structural coordinates should be transformed to the wave coordinates. (Rotation about y axis)

© 2014 Optimum Structural Design, Inc.

x = x*cos(q)-z*sin(q)

h = x*sin(q)+z*cos(q)

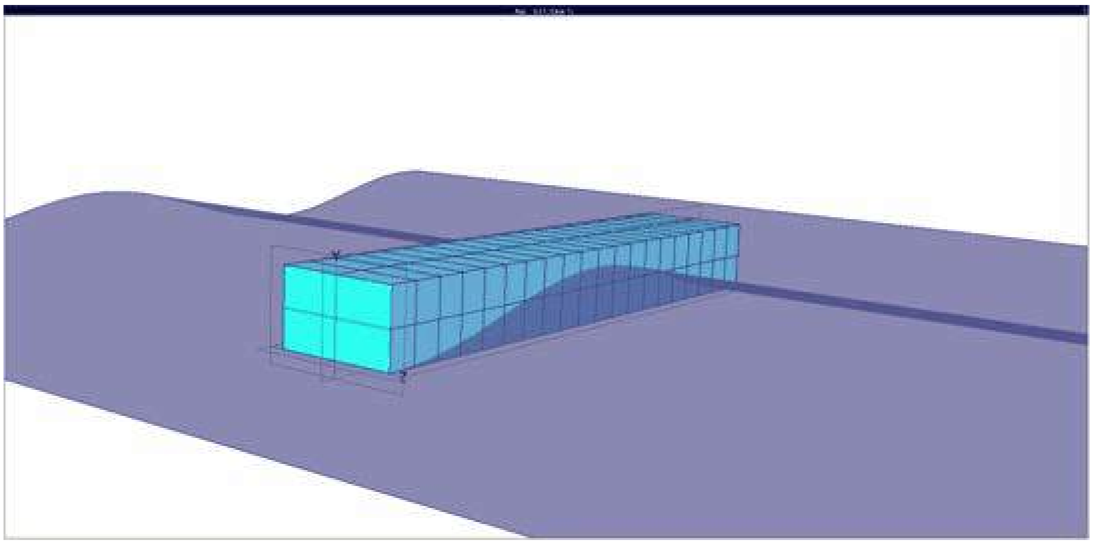


Figure 1, Balanced model, Heading=0

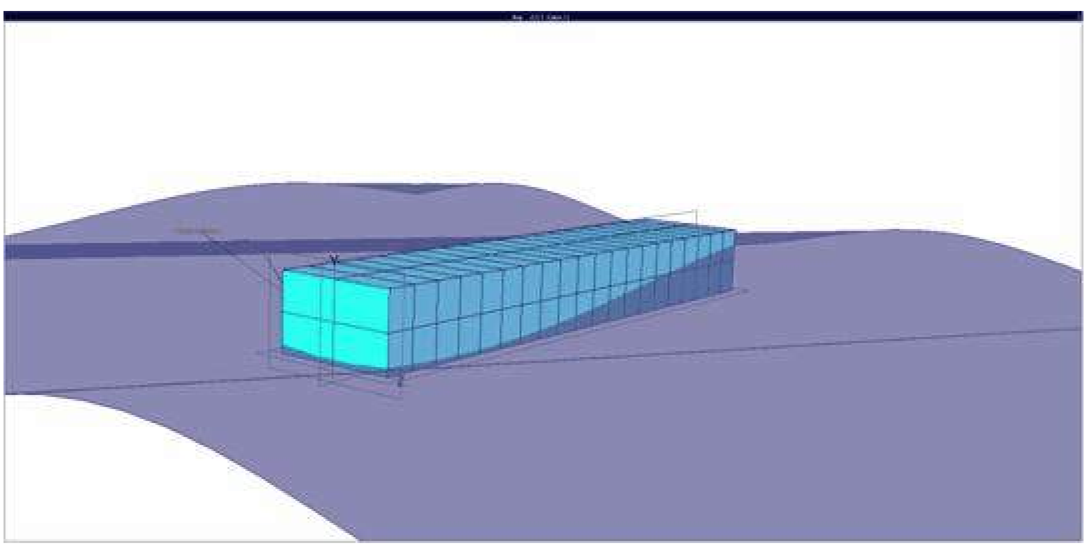


Figure 2, Un-balanced Model Heading = -60

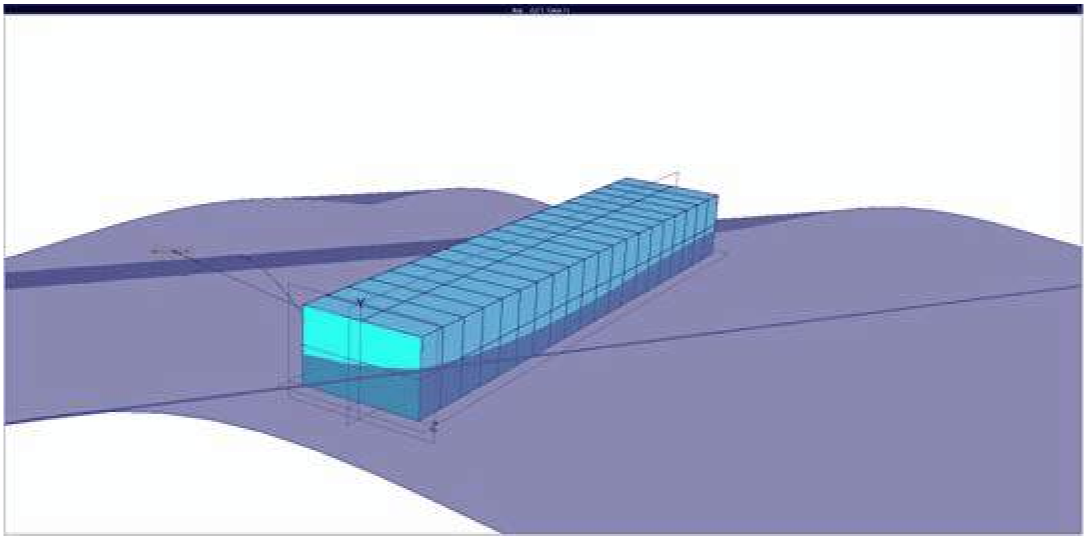


Figure 3, Balanced Model Heading = -60

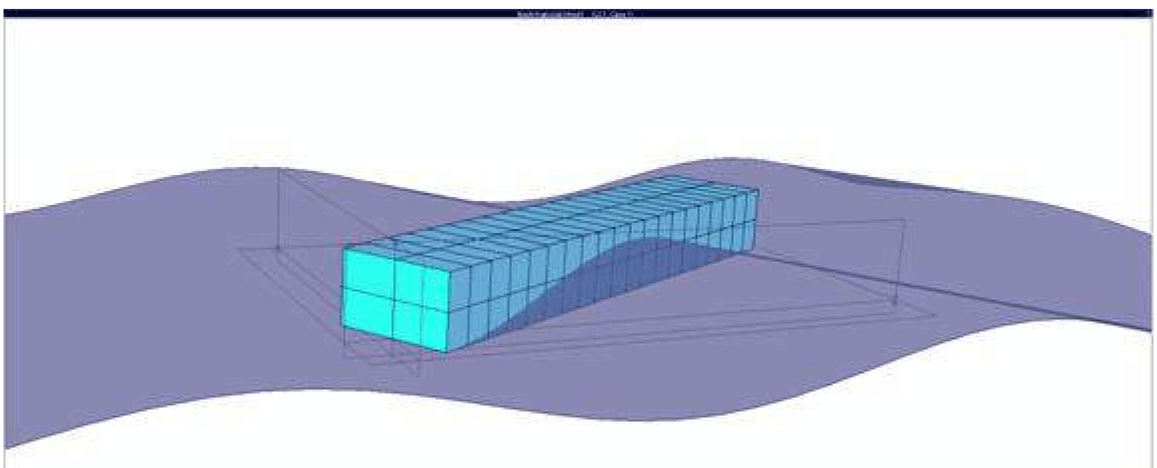


Figure 4, Balanced Model, Rotate Module -60 degrees, Heading = 60, Equivalent to Figure 1

© 2014 Optimum Structural Design, Inc.

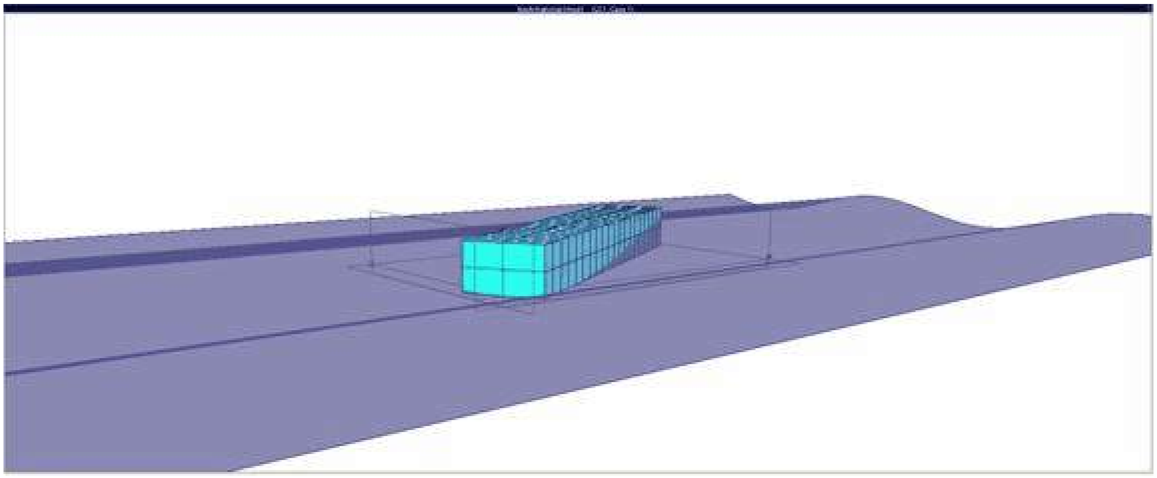


Figure 5, Un-Balanced Model, Rotate Module -60 degrees, Heading = 0, Equivalent to Figure 2

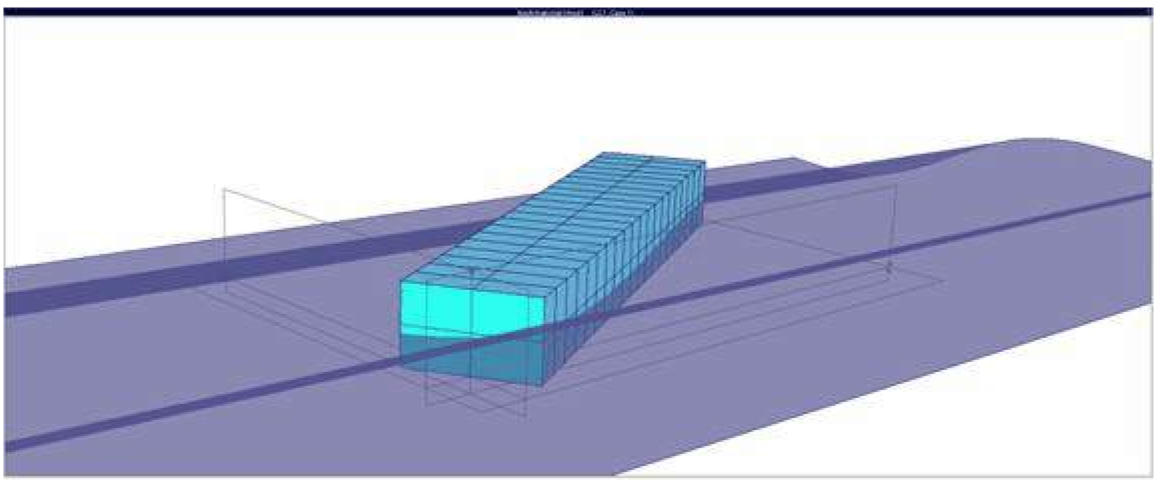


Figure 6, Balanced Model, Rotate Module -60 degrees, Heading = 0, Equivalent to Figure 3

These models (*balance_wave_60.mdl* and *balance_wave_rotate_60.mdl*) can be found in the *Models and Samples/Loads/Balance* directory of the MAESTRO installation directory.

End Moments and/or Shear Force Balancing

When you apply an End Moment to a cut model, you specify this moment by giving it a vertical bending moment at the reference and opposite end. When you balance the model, the inputs for the two moments are kept constant and the vertical shear force at each end is calculated so that there is a force and moment balance. The opposite end shear force in the loads dialog is calculated to be equal, but opposite to that of the actual shear force at that end, in order to balance the loads. The reference end shear force in the graphical representation is actually representing the shear force at a small distance in from the actual end. As a result, this shear force value also includes the distributed load from the end to that point, which is why the loads dialog and graphical representation do not match.

© 2014 Optimum Structural Design, Inc.

A sample model (*endshear.mdl*) can be found in the *Models and Samples/Loads/Balance* directory of the MAESTRO installation directory.

© 2014 Optimum Structural Design, Inc.

Analyzing and Post-Processing

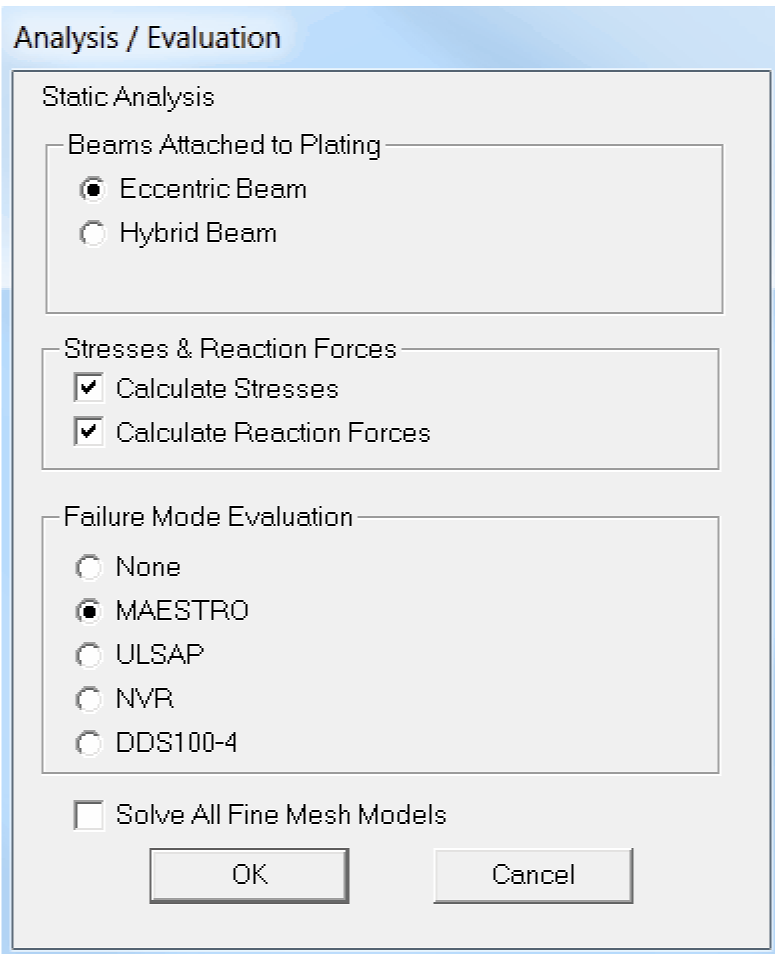
10

10 Analyzing and Post-Processing

The topics in this section provide detailed information on the MAESTRO functionality used for analyzing and post processing a model.

10.1 Solver Types

MAESTRO offers three solver methods for coarse mesh model analyses: Sparse, Iterative, and Skyline. Fine mesh model analyses use the Sparse solver method. The different solvers can be accessed from the Preferences dialog.



Sparse Solver

MAESTRO's Sparse solver method uses the Intel Pardiso sparse solver and is the default and most commonly used solver within MAESTRO. The package PARDISO is a high-performance, robust, memory efficient and easy to use software for solving large sparse symmetric and non-symmetric linear systems of equations on shared memory multiprocessors. The solver uses a combination of left- and right-looking Level-3 BLAS supernode techniques [11]. In order to improve sequential and parallel sparse numerical factorization performance, the algorithms are based on a Level-3 BLAS update and pipelining parallelism is exploited with a combination of left- and right-looking supernode techniques [6, 7, 8, 10]. The parallel pivoting methods allow complete supernode pivoting in order to compromise numerical stability and scalability during the factorization process. [13]

Iterative Solver

MAESTRO's iterative solver method uses Intel's iterative sparse solver which is based on a reverse communication interface. This scheme gives the solver great flexibility, as it is independent of the specific implementation of operations such as matrix-vector multiplication. [14] When this solver is selected, the user can choose a maximum number of iterations to perform and a tolerance. The analysis will complete whenever the number of iterations or tolerance is met. The iterative solver is mostly used for large models that are unable to be solved using the sparse solver.

Skyline Solver

MAESTRO's Skyline solver is an in-house sparse solver with the matrix stored using the skyline storage scheme. This solver is an alternative, or backup to MAESTRO's Sparse solver, and is less efficient. The benefit of the Skyline solver is that since it was developed in-house, it can be used to troubleshoot an error if the Sparse solver is not able to solve. Whereas an error from the Sparse solver is indescript, the Skyline solver will return an error message directing to a specific error in the matrix and an FE tag.

Beams Attached to Plating

Eccentric Beam Element

This element consists of only the beam itself. That is, both the area A (and the parameter a = AE/L) and the moment of intertia I (and the parameter b = EI/L^3) refer to the beam alone, and I is calculated about the beam's own centroid, the centroidal axes being denoted as $\underline{x},\underline{y}$. However, the element nodes are located not along the beam's centroidal axes, but rather at the toe of the web as shown in Fig. 8.16. That is, the origin of the element axes (x,y) is located at a distance g below the origin of the centroid axes, such that $y = \underline{y} + g$. Therefore in this type of modeling the plate elements are located at their true position, in the plane of the plating. However, the axial force in the beam continues to be proportional to the change in length of the neutral axis, and this is displaced from the element x-axis. [1]

© 2014 Optimum Structural Design, Inc.

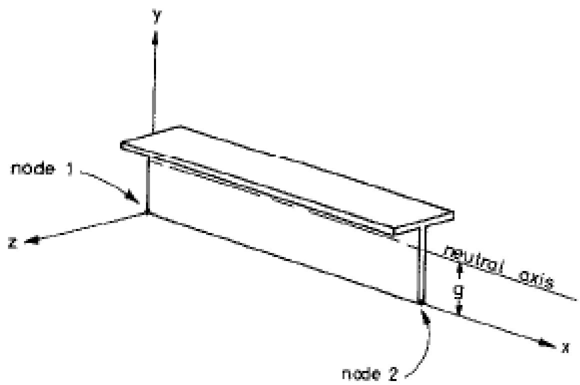


Figure 8.16 Eccentric beam element.

Hybrid Beam Element

The element is termed "hybrid" because the axial stiffness and the bending stiffness are based on different cross sections. The axial stiffness parameter is a = AE/L in which the area A is the area of only the beam itself (flange and web). The bending stiffness parameter is $b_e = EI_e/L^3$ in which the neutral axis position and the moment of inertia I_e are calculated for the combined section formed by the beam plus a plate flange of effective breadth b_e. The element nodes are located at the neutral axis of the combined section, as shown in figure 8.14. Thus, in the hybrid element the plate's contribution to the beam bending stiffness is accounted for in the calculation of I_e, and the expression for the beam stiffness matrix remains unchanged. [1]

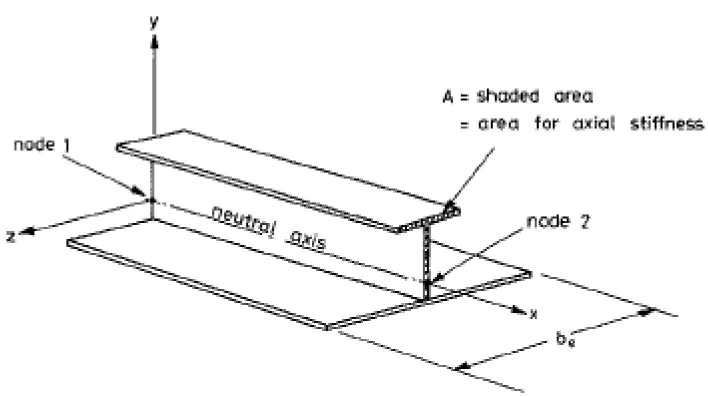


Figure 8.14 Hybrid beam element.

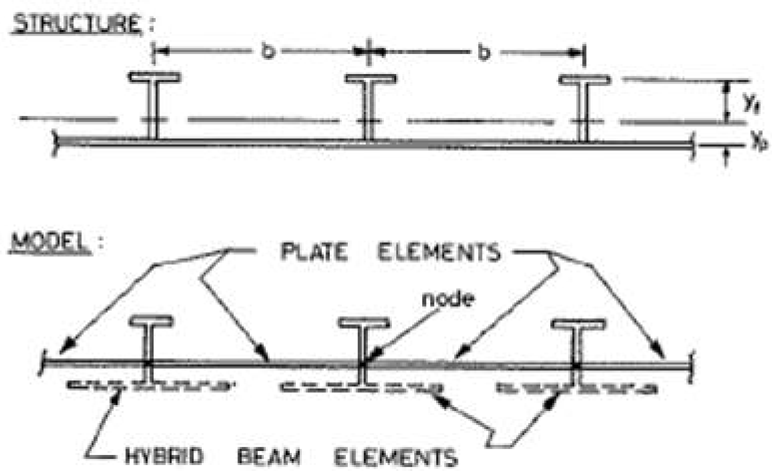


Figure 8.15 Use of the hybrid beam element.

Failure Mode Evaluation

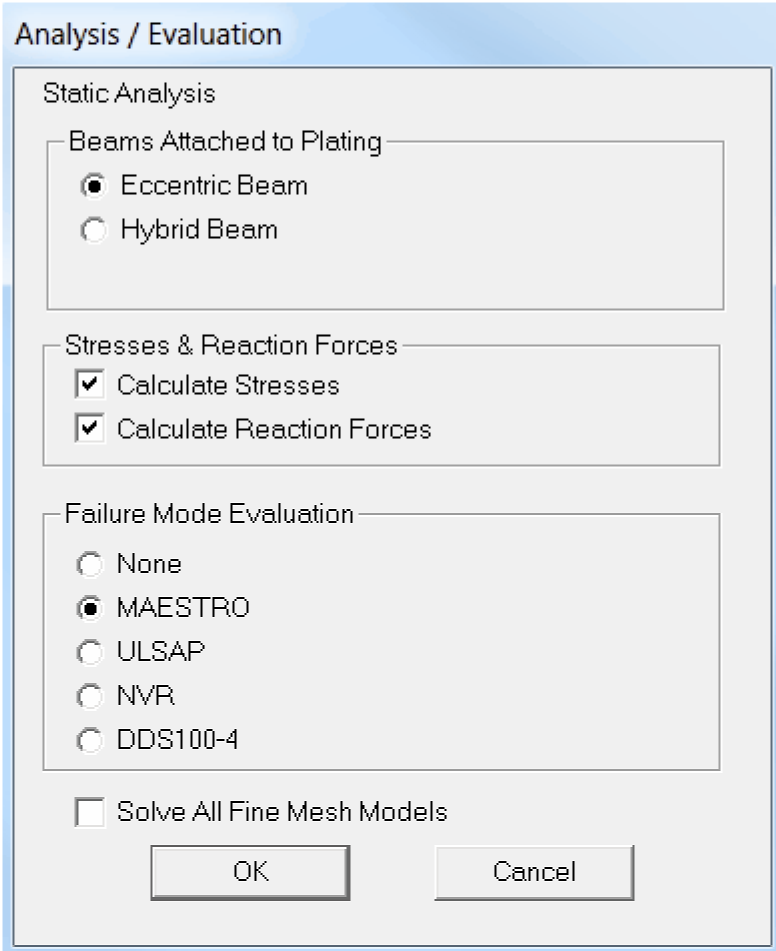
For detailed information on the two types of failure mode evaluations available, see the Failure Mode Evaluation section.

10.2 Analyzing the Model

Toolbar	
Menu	**File > Analysis/Evaluation > Global FEA**

The following tutorial shows the steps to analyze a coarse mesh model. A model can be analyzed with or without balancing the model first.

1. Open the Analyze dialog from the **File > Analysis/Evaluation > Global FEA** menu option, or from the toolbar.



2. Select the the beam elements should be treated, either as Eccentric Beams or Hybrid Beams.

3. Check whether to calculate stresses and reaction forces.

4. Click the radio button for the type of Failure Mode Evaluation to perform, if applicable. If evaluation patches have not been created, MAESTRO will automatically create the patches during the analysis process.

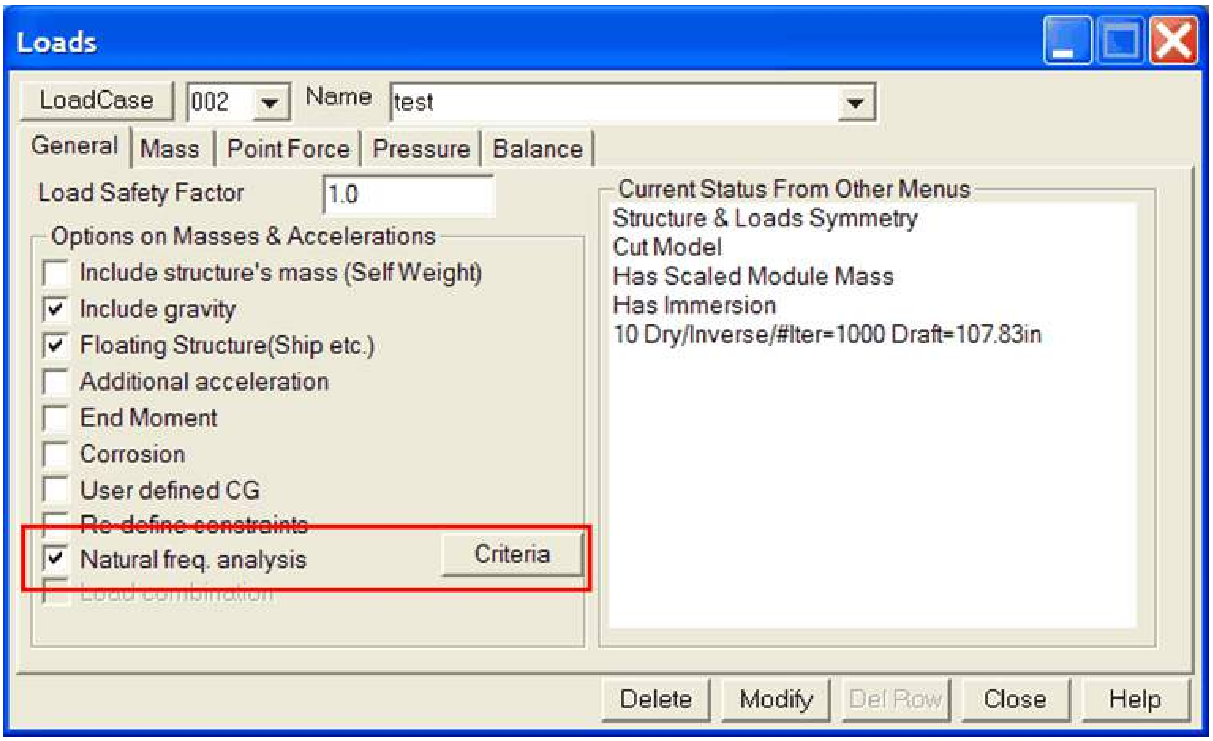
5. MAESTRO can also save the fine mesh models, if applicable, automatically after solving the global model by checking the box.

Any errors or issues with each load case will be shown in the output tab, otherwise a dialog box will appear stating the analysis is complete.

© 2014 Optimum Structural Design, Inc.

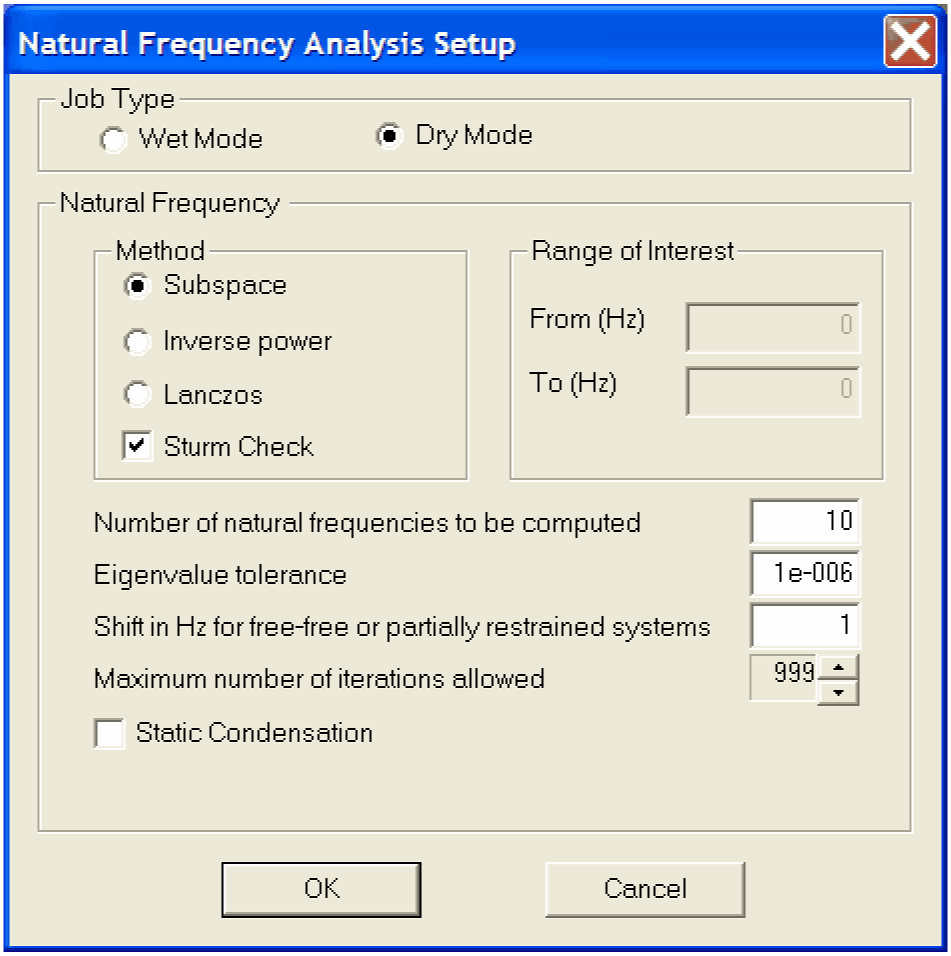
10.3 Natural Frequency Analysis

A natural frequency analysis can be performed by MAESTRO by checking the option in the Loads dialog. A separate load case should be created for the natural frequency analysis.



In the *Current Status from Other Menus* window on the General tab gives a summary of the natural frequency analysis criteria. To change the criteria, click the *Criteria* button next to the Natural freq. analysis line. Clicking this button will launch the Natural Frequency Analysis Setup dialog.

© 2014 Optimum Structural Design, Inc.



Under the Job Type section, MAESTRO can perform an analysis for the structure vibrating in vacuo or in fluid. When *Wet Mode* is selected, the added mass of the seawater is automatically applied to the "wetted" elements if an immersion condition is defined.

MAESTRO offers three iterative solver methods: Subspace, Inverse Power, and Lanczos.

The Lanczos and Subspace Iteration Methods are both computationally efficient with the Lanczos Method performing exceptionally well with medium to large size models. A Sturm Check may be used if the Skyline solver is the solver of choice. Of the two, the Subspace Method is the preferred algorithm in MAESTRO and is the most robust. However, the Lanczos Method is somewhat computationally quicker.

The Inverse Power algorithm (also known as simply Inverse Iteration or Inverse Iteration with Sweeping) is used to calculate the eigenvector for the lowest eigenvalue. At the end of each iteration, the new "guessed" eigenvalue is provided by the Rayleigh quotient, further improving upon the initial eigenvector. This method is computationally dense when retrieving many modes and has potential to miss modes of vibration. It is sufficient for finding the prominent global modes of vibration. A Sturm Check to prevent missing modes may be elected if the Skyline solver is the solver of choice.

The user can input the number of natural frequencies to compute, the eigenvalue tolerance for the iteration solver and the shift in hertz for free-free or partially restrained systems. The maximum number of iterations may need to be increased if the convergence is slow; the default value is 999.

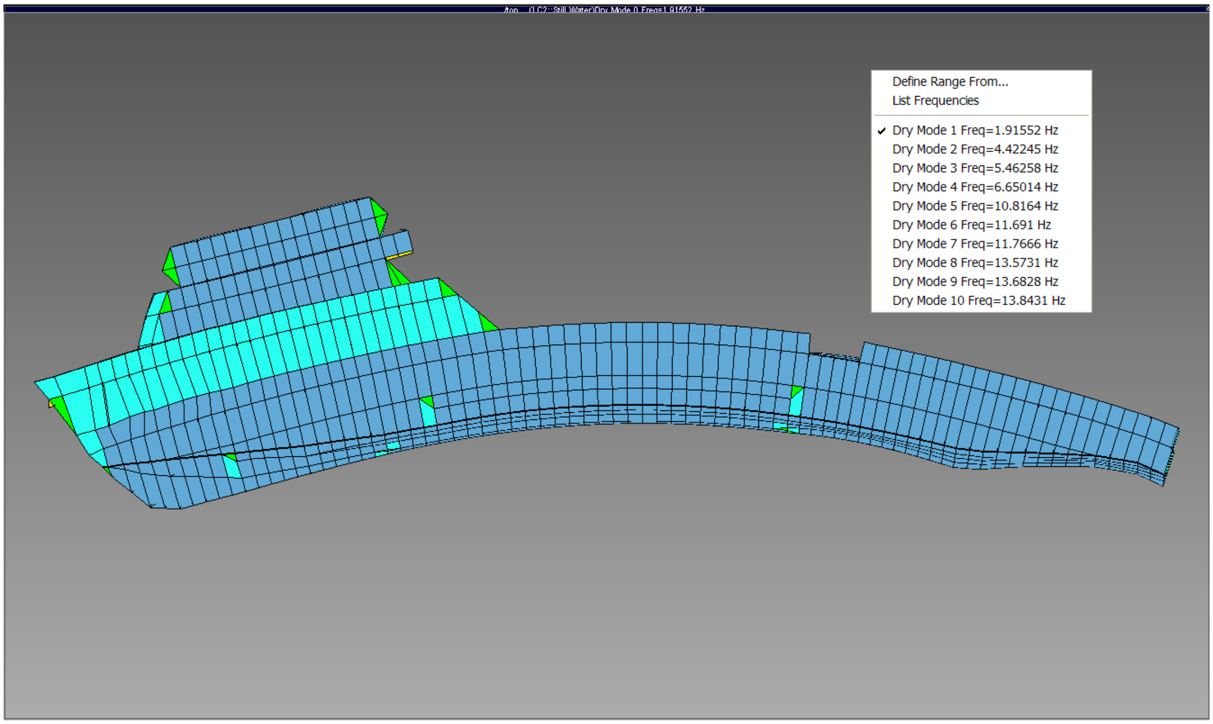
The natural frequency analysis load case will be solved during the coarse mesh analysis. The Output tab will display a summary of the natural frequencies calculated during the analysis:

```
Mode 1 is converged. Number of iterations=6, Freq=1.91552 (Hz)
Mode 2 is converged. Number of iterations=19, Freq=4.42245 (Hz)
Mode 3 is converged. Number of iterations=3, Freq=5.46258 (Hz)
Mode 4 is converged. Number of iterations=3, Freq=6.65014 (Hz)
Mode 5 is converged. Number of iterations=24, Freq=10.8164 (Hz)
Mode 6 is converged. Number of iterations=250, Freq=11.691 (Hz)
Mode 7 is converged. Number of iterations=3, Freq=11.7666 (Hz)
Mode 8 is converged. Number of iterations=130, Freq=13.5731 (Hz)
Mode 9 is converged. Number of iterations=128, Freq=13.6828 (Hz)
Mode 10 is converged. Number of iterations=3, Freq=13.8431 (Hz)
Model Participation Factor/Model Effective Mass
Mode     Frequency      Eff.M-xx      ACU-%         Eff.M-yy      ACU-%         Eff.M-zz      ACU-%
  1        1.9155       3601.849      0.73        416959.601      84.33          0.488        0.00
  2        4.4225       1519.570      1.04            16.492      84.34          0.280        0.00
  3        5.4626     441294.245     90.29          1230.083      84.59         17.364        0.00
  4        6.6501      40236.968     98.43            55.418      84.60          0.648        0.00
  5       10.8164       2308.512     98.90         41593.216      93.01         10.606        0.01
  6       11.6910        102.532     98.92          8367.377      94.70          0.220        0.01
  7       11.7666        287.932     98.98          5660.535      95.85          0.264        0.01
  8       13.5731        111.973     99.00           146.785      95.88         59.656        0.02
  9       13.6828         19.150     99.00            37.712      95.88          5.303        0.02
 10       13.8431        204.034     99.04           344.739      95.95          5.329        0.02
           TOTAL          489687                     474412                    100.159
```

"ACU-%" is the accumulated modal participation factor. Modal effective mass and participation factor are relative measurement to the total vibration. For example, if the "ACU-%" is 99% in the first five modes, it implies the first five modes are dominant modes, and the rest of the modes can often be ignored. Modal effective mass and participation factors are often used in constrained vibration such as Dynamic Design Analysis Method (DDAM).

To view the different mode shapes, select the natural frequency analysis load case from the load case drop-down and set the model to deformed view from **Results > Deformed Model.** The different mode shapes can be selected by right-clicking in the model space and selecting the frequency from the menu.

© 2014 Optimum Structural Design, Inc.



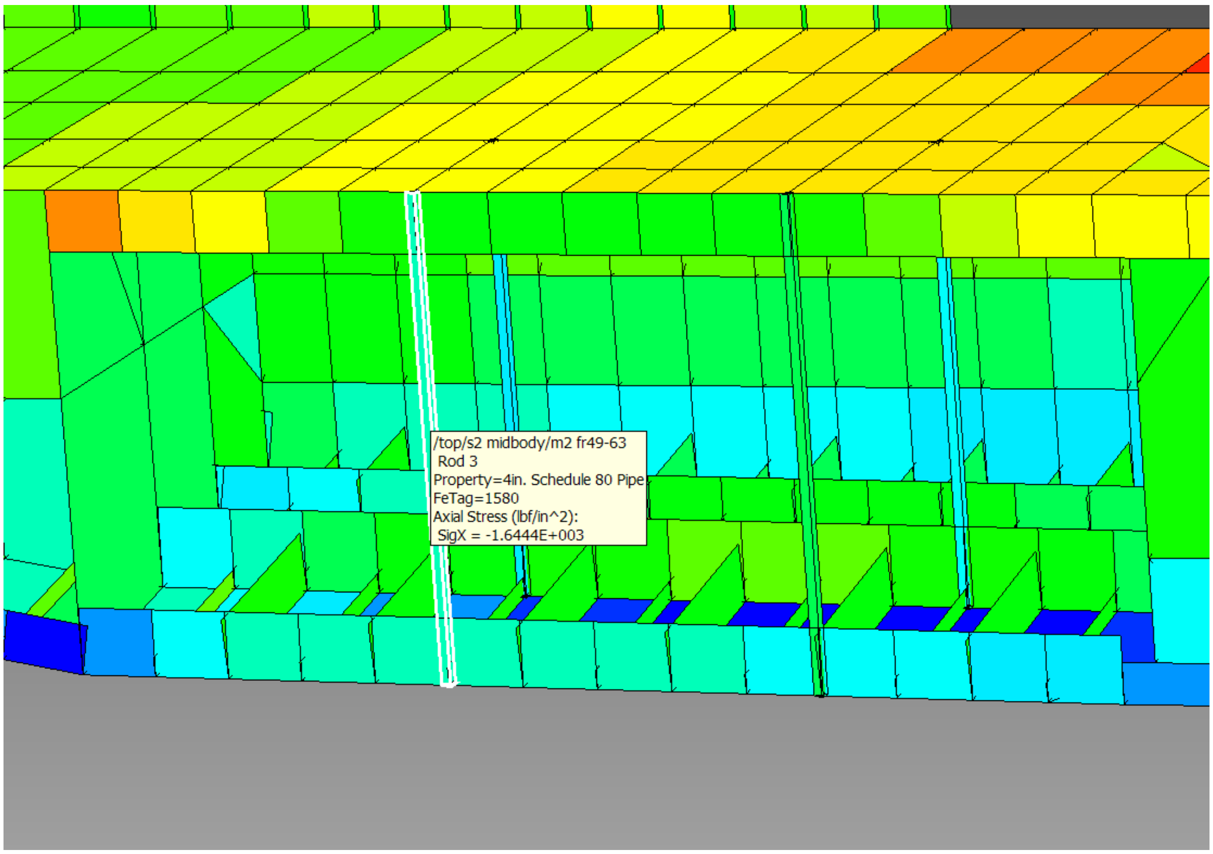
10.4 Stress Results

The following sections provide a description of MAESTRO's available elements and their associated stresses. MAESTRO uses an FEA standard method, called the *Gauss Quadrature* method, for element numerical integration for element stress computation. This method operates by evaluating (or sampling) elements at multiple points (Gauss points) within a given element. The stress values calculated at these Gauss points are subsequently extrapolated to *other* locations in the elements. These *other* locations (e.g., stresses at nodes, element centroids, element top surface, element mid-plane, etc.) are the more commonly exposed stresses for use by the engineering analyst. MAESTRO takes this industry standard approach.

MAESTRO's contour plot is either a nodal contour by checking the *Corner Stress* option in the View Options dialog, which averages all values at the nodes or an average elemental centroidal stress.

10.4.1 CROD (Rod)

The axial stress of rod elements will be displayed when *Mid Normal X* and *Mid Von Mises* are selected from the Results menu.



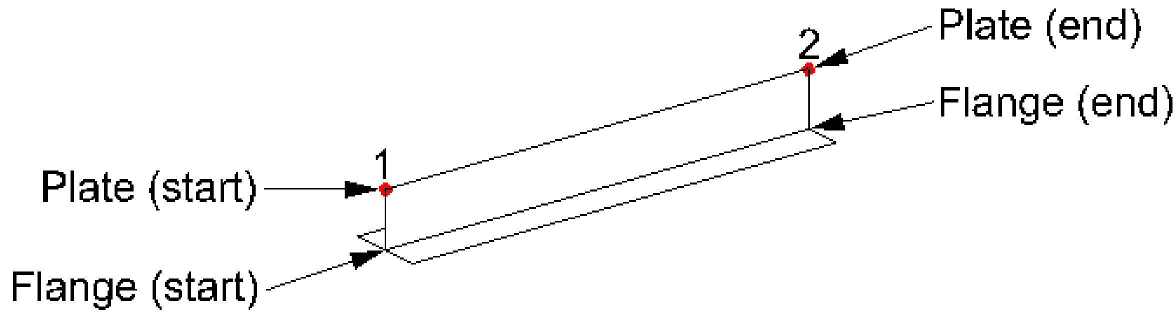
The dynamic query icon  can be used to highlight a rod element and recover the axial stress. Double-clicking the element will echo the results to the Output tab.

Rod elements will be grayed out when stress options other than the two listed above are chosen. However, the axial stress can still be recovered from a rod element using the dynamic query function.

10.4.2 CBAR (Beam)

The CBAR element is a uniaxial element with tension, compression, and bending capabilities. The CBAR element is used for modeling MAESTRO *frames*, *girders*, and *additional beams*. The CBAR element recovers the mid-plane axial stress (SigX), the axial stress at the beam *flange* and *plate* connection, as well as the beam stress. As shown below, these stresses are recovered at the *start*, mid-span, and *end* of the CBAR definition. The CBAR element also reports Moments and Shear Force at the *start* and *end* of the element.

© 2014 Optimum Structural Design, Inc.



In addition to the specific beam stress options, the *Mid X Normal* and *Mid Von Mises* are the only stress options that will graphically display the beam element's axial stress.

Hybrid Beam versus Eccentric Beam Method

The following discusses how MAESTRO *models* beams attached to plates. The engineering analyst can choose which method to use for a given analysis but should be aware of the impact on CBAR stress results. This option is set at the time of the analysis via the Analysis dialog.

Hybrid Beam Method

First, the Hybrid Beam method. MAESTRO has the ability to use the so-called Hybrid Beam element for the modeling of beams (frames, girders, and additional beams) attached to plating. The element is termed "hybrid" because the axial stiffness and the bending stiffness are based on different cross sections. The axial stiffness parameter is $\alpha = {AE}/{L}$, in which the area A, is the area of only the beam itself (flange and web). The bending stiffness parameter is $\beta_e = {EI_e}/{L^3}$, in which the neutral axis position and the moment of inertia (I_e) are calculated for the combined section formed by the beam plus a plate flange of effective breadth b_e. The element nodes are located at the neutral axis of the combined section, as shown in the figure below. Thus, in the hybrid element the plate's contribution to the beam bending stiffness is accounted for in the calculation of I_e and the expression for the beam stiffness matrix remains unchanged. This method requires the specification of the effective breadth of plating to be considered as part of the bending stiffness, which can be problematic for automatic data generation.

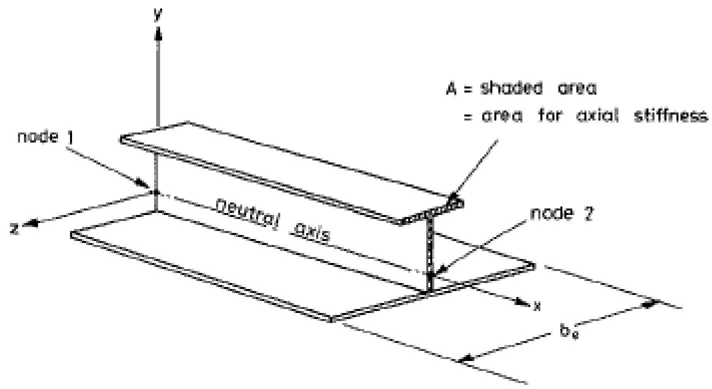


Figure 8.14 Hybrid beam element.

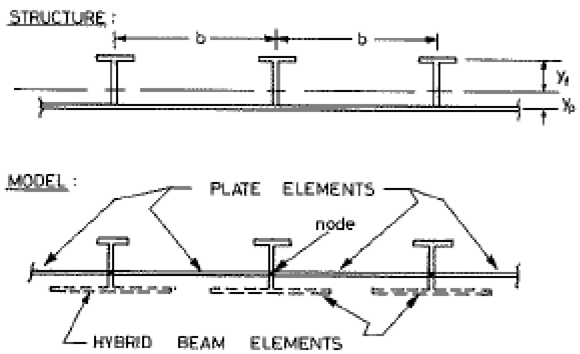


Figure 8.15 Use of the hybrid beam element.

Eccentric Beam Method

An alternative approach is the Eccentric method, or Offset Beam method, which consists of only the beam itself. That is, both the area A (and the axial stiffness parameter $\alpha = {AE}/{L}$) and the moment of inertia I (and the stiffness parameter $\beta = {EI}/{L^3}$) refer to the beam alone, and I is calculated about the beam's own centroid. However, the element nodes are located not along the beam's centroidal axes, but rather at the toe of the web as shown in the figure below. That is, the origin of the element axes (x,y) is located at a distance g below the origin of the centroid axes, such that $y = y + g$. Therefore in this type of modeling the plate elements are located at their true position, in the plane of the plating. Further, the locations between the node and the beam's neutral axis (i.e., *g)* can be regarded as joined by rigid link. Although this method is well suited for automatic data generation, issues arise with how this method handles particular loading scenarios where shear forces are more dominant because stiffener and plate are not jointly carrying the loads. However, this error decreases with increasing refinement of the FEM mesh.

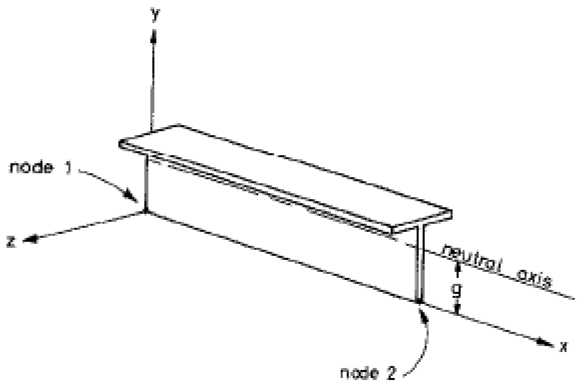


Figure 8.16 Eccentric beam element.

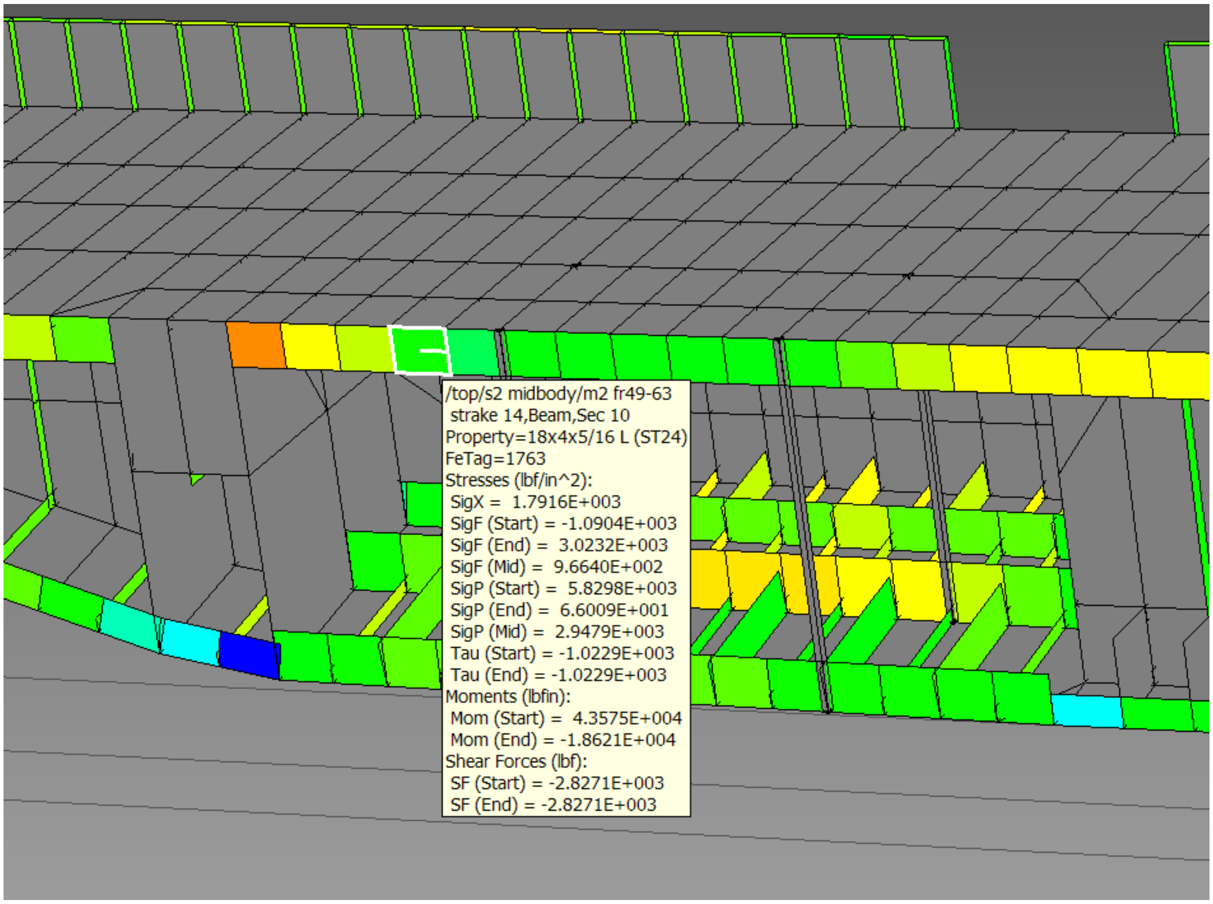
Beam Element Stress Recovery Locations

Flange (Start)
Flange (End)
Plate (Start)
Plate (End)
Flange Max Tension
Flange Max Compression

The beam stress menu from **Results > Stress > Beam >** allows the user to select the location of the recovered stress to plot graphically. The *Flange Max Tension* and *Flange Max Compression* options will plot the higher of the compressive or tensile stresses at the beam flange from either the start or end location. The beam stress results menu is only used to select which stress value to plot for each beam element.

The dynamic query icon  can be used to highlight a beam element and recover the stresses. Double-clicking the element will echo the results to the Output tab.

© 2014 Optimum Structural Design, Inc.



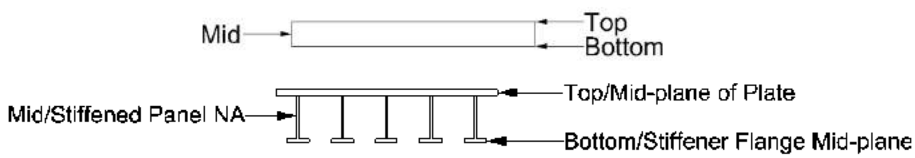
When querying a beam element, all stresses, including local axial stress and shear stress will be recovered as well as the bending moments and shear force at each end of the beam. The stresses at the mid location of the plate connection and flange are calculated by linearly interpolating the appropriate stresses at the start and end locations of the beam.

10.4.3 CQUAD4R

The CQUAD4R is the most commonly used element in MAESTRO. The CQUAD4R element is a 4 noded flat shell element with each node having 6 degrees of freedom (DoF) and has an additional *drilling* DoF, which is a rotational DoF whose vector is normal to the element's analysis plane. MAESTRO's CQUAD4R element has the ability to automatically include the appropriate bending stiffness if a *stiffener layout* definition is present. This is true in either the longitudinal direction or the transverse direction. This element is the same element used in NASTRAN, but in MAESTRO, the (quite large) task of calculating the orthotropic properties is automated.

The CQUAD4R element recovers bare pate and stiffened plate element stress at the top, middle and bottom of the element's centroid and corners. The different stresses at each location of the element can be plotted by selecting the option from the Results menu. The element X, Y, shear and Von Mises stresses can be plotted and recovered for each of the three locations on the plate element. When using the Results menu, the reported Top, Mid, and Bottom stresses are those found at the element centroid.

The figure below presents these three locations for a bare plate element (Figure a) and a stiffened plate (Figure b). Von Mises stresses are also reported at the Top, Mid, and Corners. The Von Mises corner stresses represent *effective* stress at the corners and is computed as $\sigma_{vm} = \sqrt{\sigma_x^2 + \sigma_y^2 - \sigma_x\sigma_y + 3\tau_{xy}^2}$. The Von Mises corner stresses are exposed to the user by dynamically querying individual elements. The Von Mises corner stress represent the stress at the mid-plane.

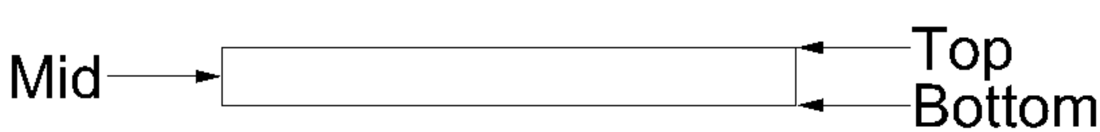


(a) Bare Plate **(b) Stiffened Plate**

Additionally, MAESTRO exposes the CQUAD4R X, Y, and shear mid-plane stress at the element corners using the *List* or *MS Excel* export option. Internally, MAESTRO is also computing additional stresses, such as: corner stresses at the Top and Bottom of the element plate field, but currently does not expose them to the user. Future development will expose these particular stresses.

10.4.3.1 CQUAD4R (Bare)

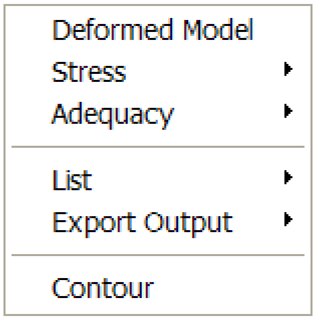
MAESTRO will recover stress on a bare plate element at the top, middle and bottom of the element. The different stresses at each location of the element can be plotted by selecting the option from the Results menu. The element local X, local Y, shear and Von Mises stresses can be plotted and recovered for each of the three locations on the plate element. The figure below presents these three locations for a bare plate element.

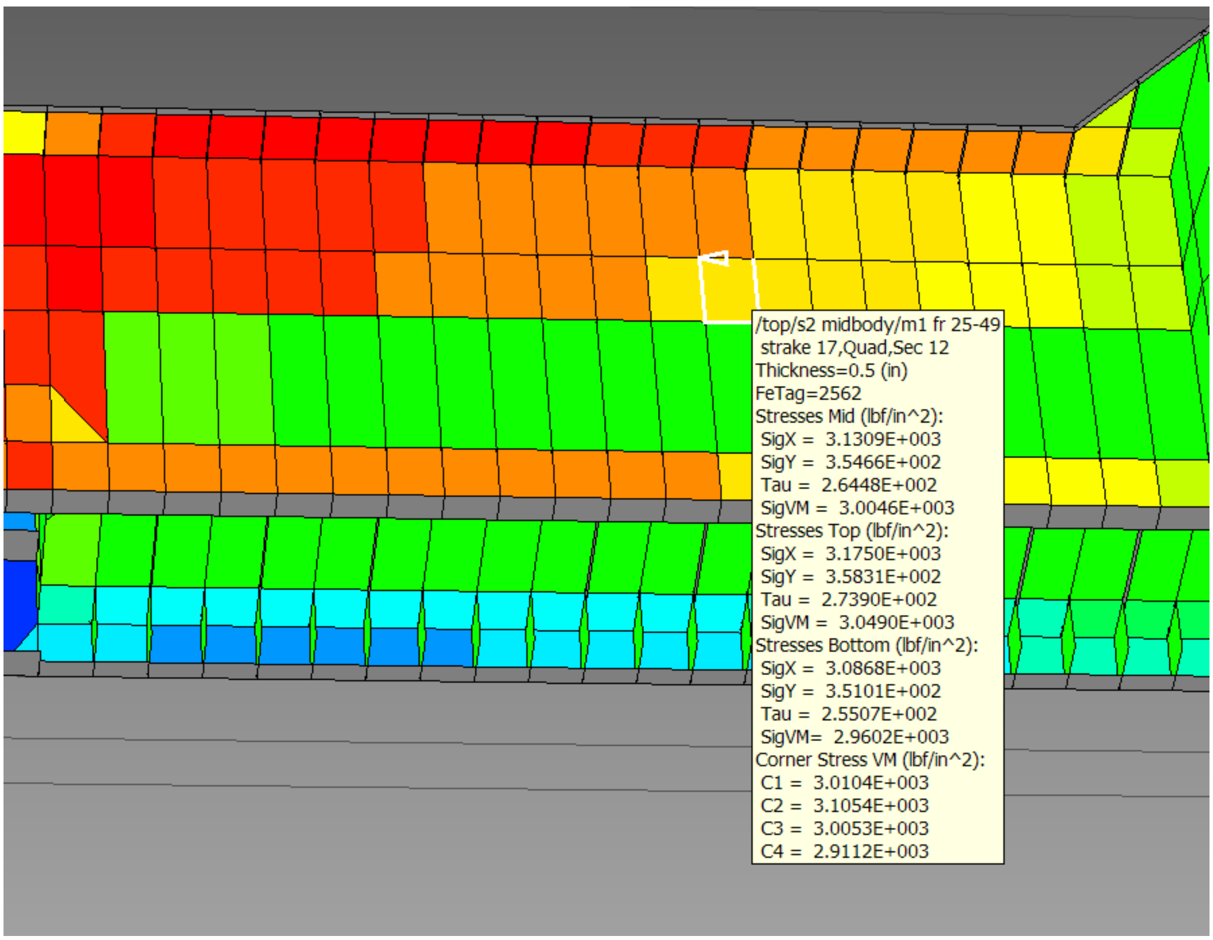


Bare Plate Stress Recovery Locations

The dynamic query icon  can be used to highlight a bare plate element and recover the stresses. The drop-down arrow next to the dynamic query icon can be used to select which stresses to display when highlighting an element.

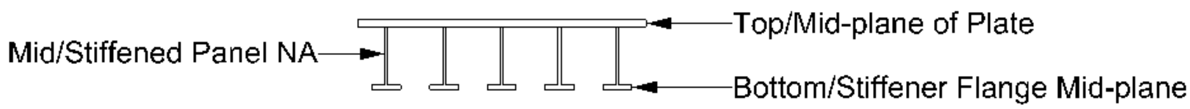
© 2014 Optimum Structural Design, Inc.





10.4.3.2 CQUAD4R (Stiffened)

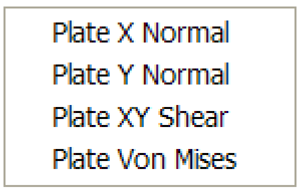
MAESTRO can recover stresses for a stiffened panel at the mid-plane of the plate (Top), the neutral axis of the plate and stiffener combination (Mid) and the axial stress at the mid-plane of the stiffener flange (Bottom/Stiffener Flange). The figure below presents these three locations.



Stiffened Panel Stress Recovery Locations

Top

The Plate X Normal, Plate Y Normal, Plate XY Shear, and Plate Von Mises stresses can be recovered at the mid-plane of the plate for a stiffened panel from the **Results > Stress > Top >** menu.



The dynamic query icon  can be used to highlight a stiffened panel and recover the stresses in the plate. Double-clicking the element will echo the results to the Output tab.

Mid

The X Normal, Y Normal, and Von Mises stresses are recovered at the neutral axis of the stiffener and plate structure. The shear stress reported for Mid is the Plate Shear stress from the mid-plane of the plate.

The dynamic query icon  can be used to highlight a stiffened panel and recover the stresses in the plate. Double-clicking the element will echo the results to the Output tab.

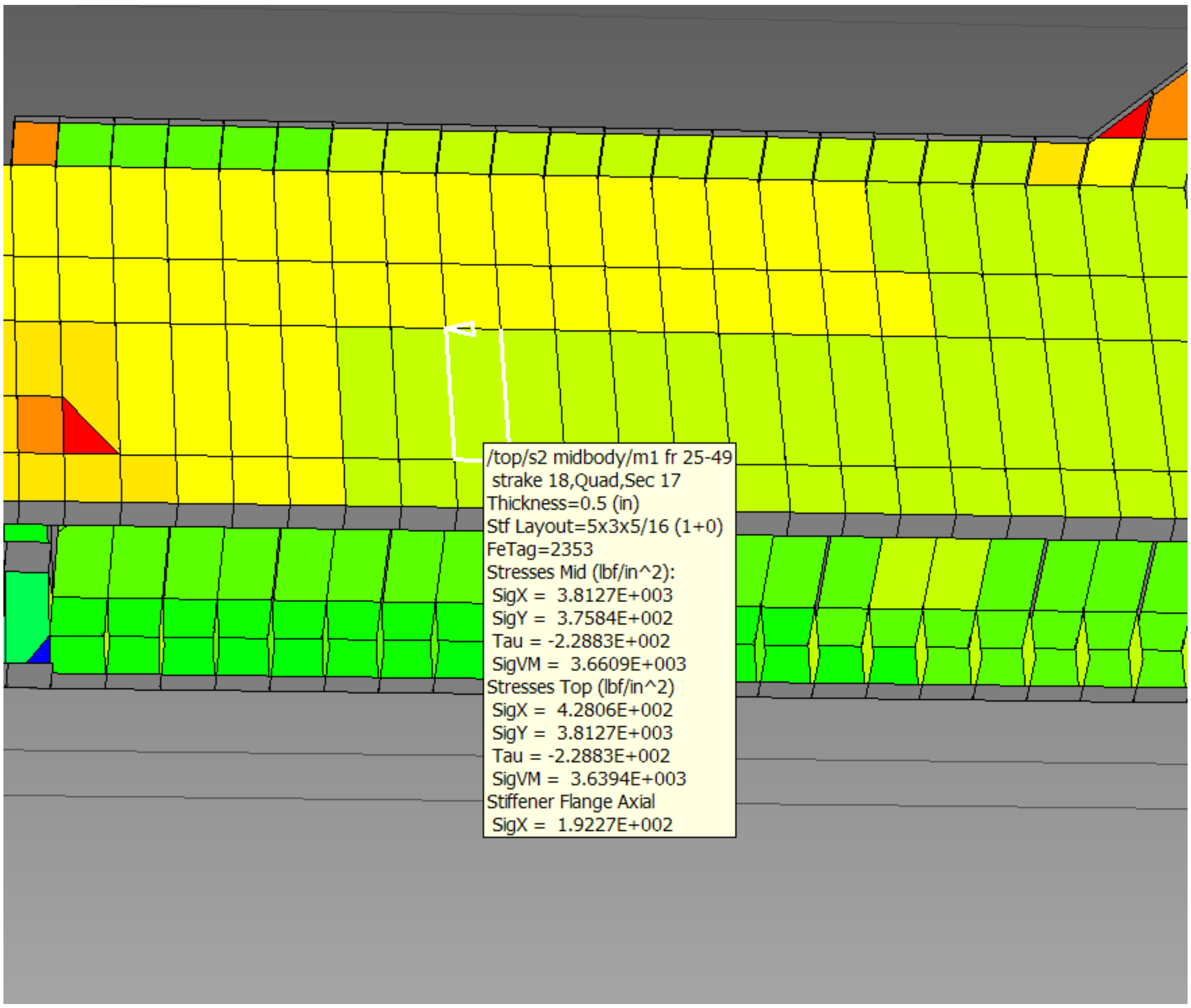
Bottom/Stiffener Flange

The only stress recovered for a stiffened panel at the bottom location is the axial stress at the mid-plane of the stiffener flange. This can be graphically displayed by selecting either **Results > Stress > Bottom > Stiffener Axial Flange** or selecting the appropriate stress direction (Normal X or Normal Y) corresponding to the orientation of the stiffener. If bottom Normal Y stress is chosen from the Results menu, any stiffened panels with longitudinal stiffeners will be grayed out because the axial stress of the stiffeners is orientated in the X

© 2014 Optimum Structural Design, Inc.

direction.

The dynamic query icon  can be used to highlight a stiffened panel and recover the stresses in the plate. Double-clicking the element will echo the results to the Output tab. The Stiffener Flange Axial stress will always be displayed as SigX, regardless of the stiffener orientation because it is reporting the stiffener local X, or axial, stress.



As with the bare plate elements, the dynamic query drop-down can be used to filter which location stresses should be displayed. Note there is no "bottom" stresses reported for a stiffened panel, only the Stiffener Flange Axial stress.

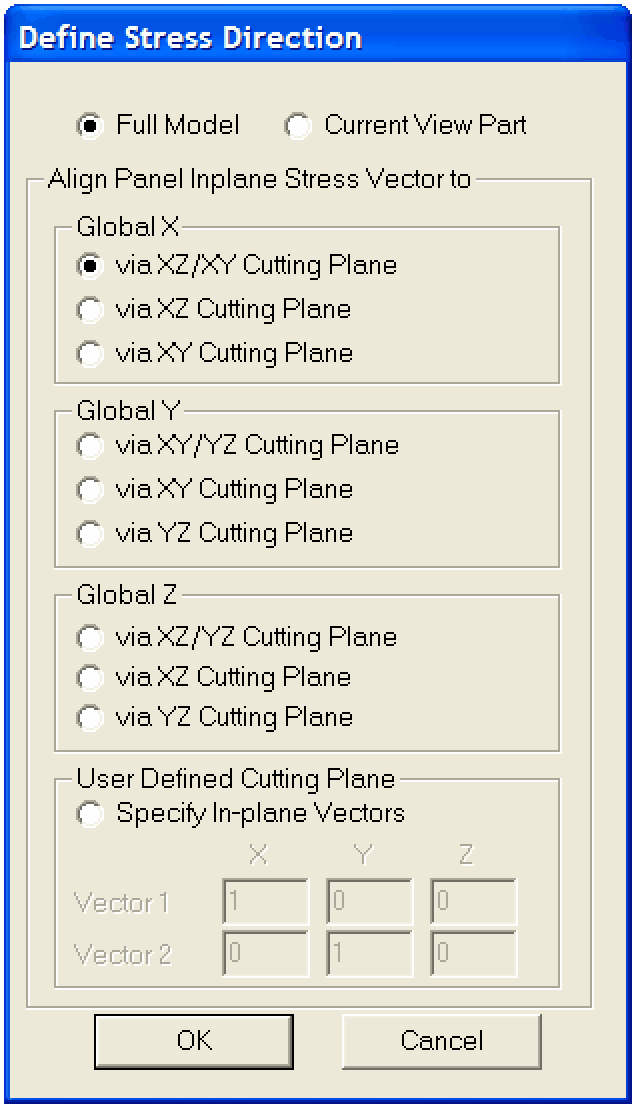
10.4.4 Directional Stress

Toolbar	***N/A***		
Menu	**Results > Stress > Directional**	**Results > Stress > Define Direction...**	**Results > Stress > Show Direction**

Introduction

When constructing a finite element model, it is ideal to generate elements such that their respective local X and Y axes are all aligned in a uniform direction, i.e. aligned to a global axis. The reason for this, is so the analyst can be assured the direction of the stress vectors are uniform when recovering stress in a given axis (e.g. post-processing Mid X Normal stress). The presentation of stress in a uniform manner such as this allows the analyst to better assess the stress path in a given direction. Because it is impossible to construct a complete full ship model in this manner, MAESTRO provides the option to display all panel elements' stress in a given global direction, regardless of the elements local orientation. When this option is checked on, all panel elements' in-plane stress vector will be aligned to the global X, global Y, global Z or a user-specified direction.

The dialog for defining directional stress is shown below. Before describing the options available for defining directional stress it is useful for the analyst to understand the internal logic used by MAESTRO to determine the re-aligned in-plane stress vector. The following example shows how in-plane stress vectors are determined and aligned to the Global X axis. The user has the ability to also align the in-plane stress vectors to the Global Y and Global Z axes in a similar manner.



Method for Determining In-plane Stress Vector

For panel elements, neither of whose local x and y axes are in the Global direction selected, the in-plane stress vector will be automatically determined by MAESTRO. To illustrate MAESTRO's logic for determining the in-plane stress vector let's take the following example of a given element in 3D space (Figure 1 below). MAESTRO creates two global cutting planes that intersect the centroid of the element. The example below illustrates the alignment of the in-plane stress vector with the Global X axis; therefore, the two cutting planes created are in the XY and XZ cutting planes. The XY and XZ cutting planes are shown below in Cyan and Green respectively (Figure 2 below).

© 2014 Optimum Structural Design, Inc.

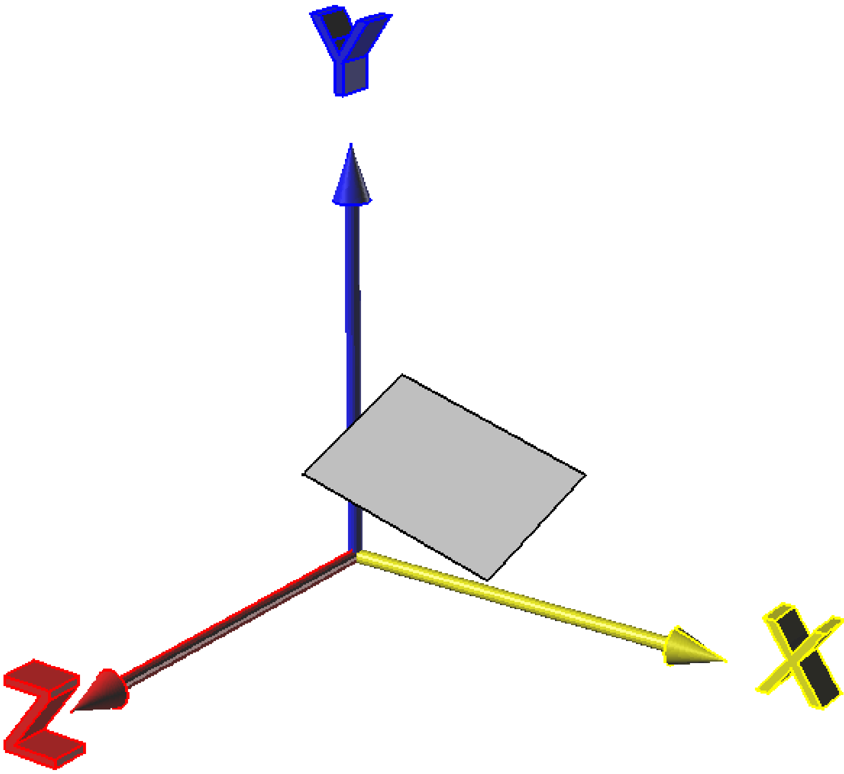


Figure 1. Element in 3D Space

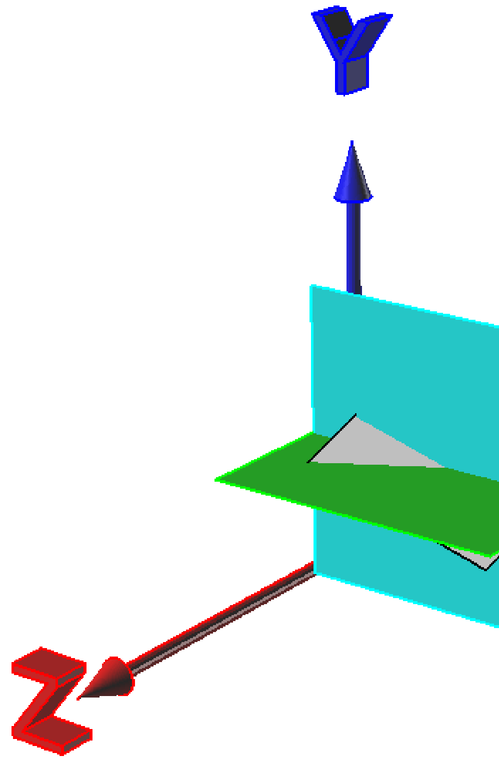


Figure 2. Global Cutting Planes: XY

Using the intersection of the element and the global cutting plane, an in-plane stress vector is created as shown for the global XY cutting plane in Figure 3 and Figure 4.

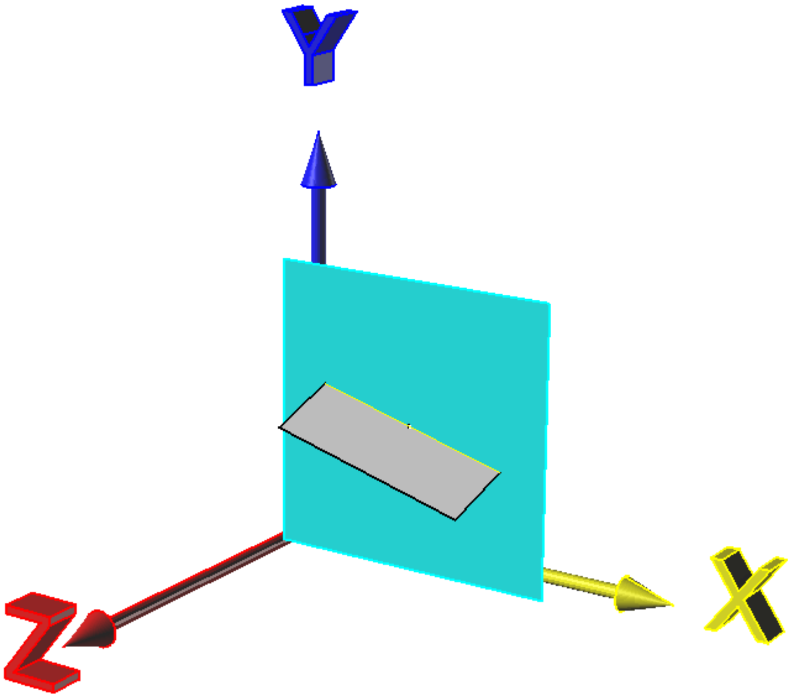


Figure 3. Element & XY Cutting Plane Intersection

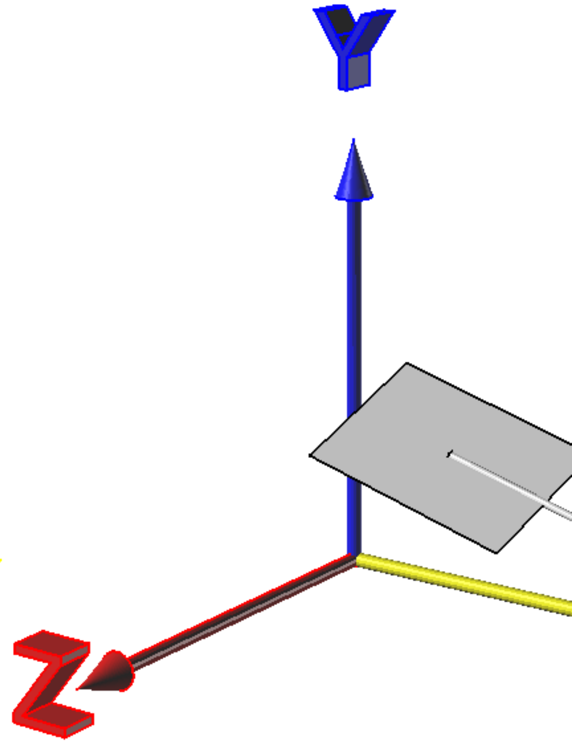


Figure 4. In-plane Stress Ve

A similar process for the global XZ cutting plane results in another in-plane stress vector as shown in Figure 5 and Figure 6.

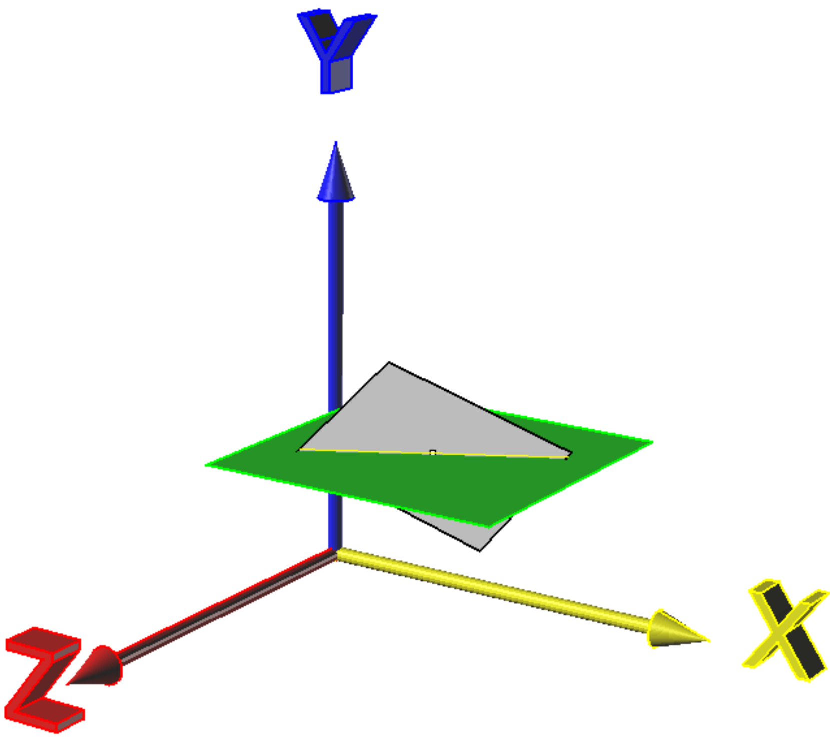


Figure 5. Element & XZ Cutting Plane Intersection

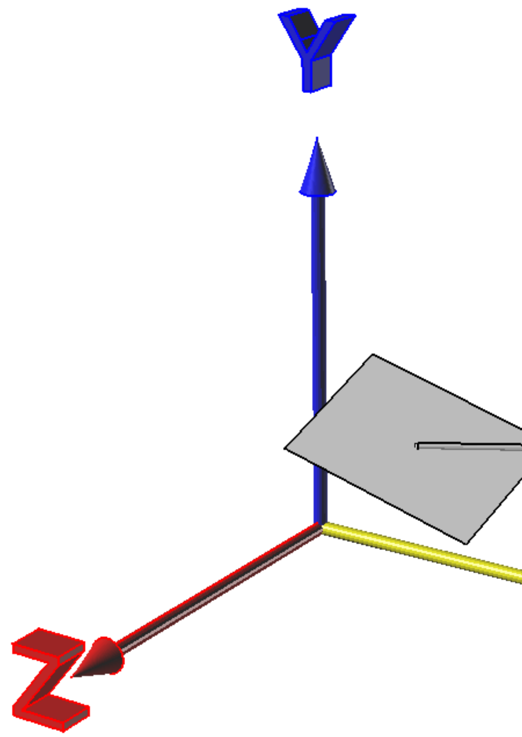


Figure 6. In-plane Stress

As described above, MAESTRO has located two in-plane stress vectors. The user now has the ability to use the in-plane stress vector generated from the XY or YZ cutting planes or alternatively, the user can ask MAESTRO to automatically choose between the two vectors based on the maximum projected area of the element onto the XY or YZ cutting plane. This projection of area is illustrated in Figure 7 and Figure 8 for the XY global cutting plane and Figure 9 and Figure 10 for the global XZ cutting plane.

© 2014 Optimum Structural Design, Inc.

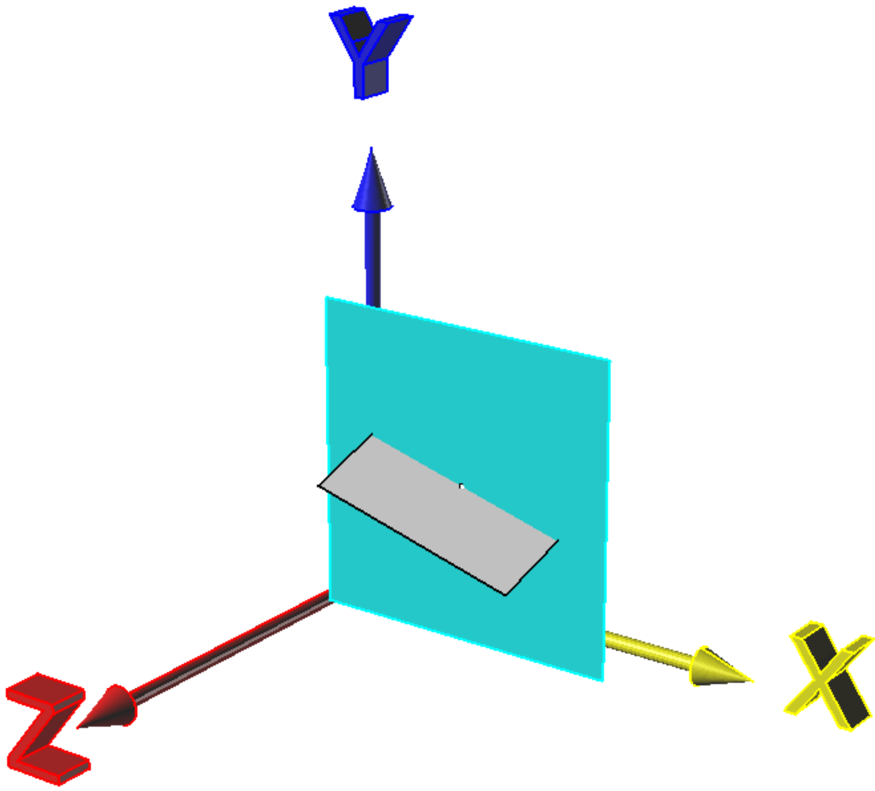


Figure 7. Element Projected Area on XY Cutting Plane

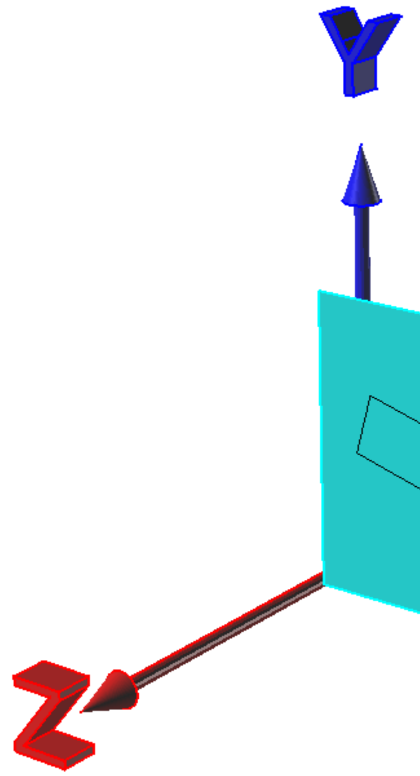


Figure 8. Projected A

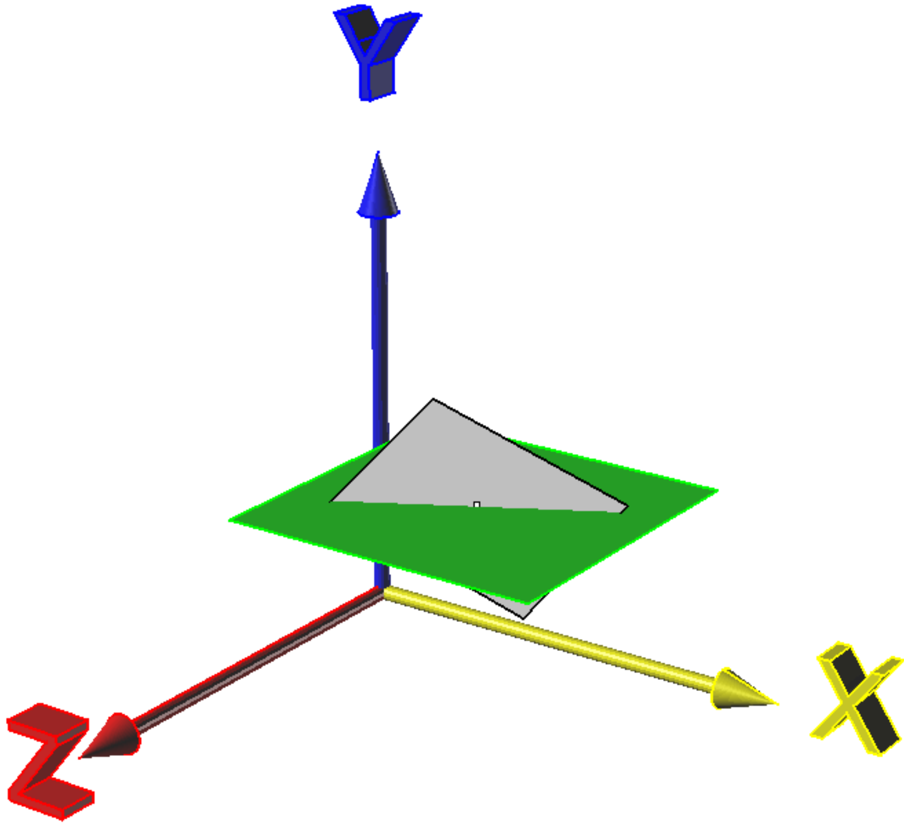


Figure 9. Element Projected Area on XZ Cutting Plane

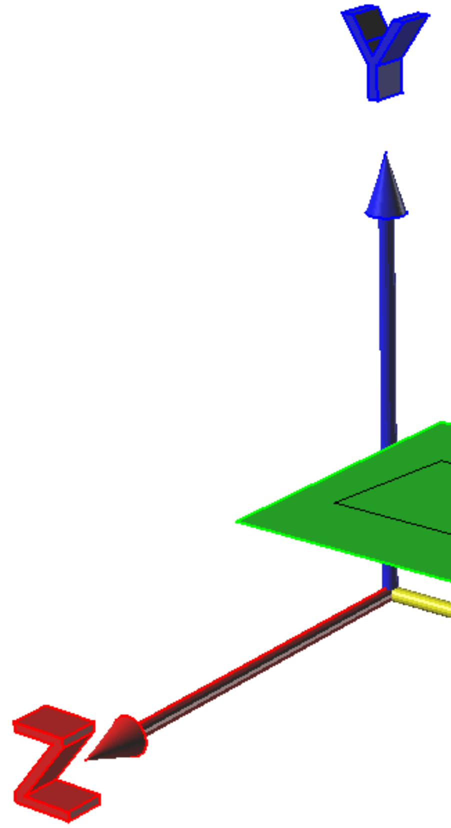


Figure 10. Projected A

Since in this example the projected area in the XZ cutting plane is the larger of the two, the chosen stress vector would lie in the XZ cutting plane as shown in Figure 10.

As noted in the beginning, the example described above shows how in-plane stress vectors are determined and aligned to the Global X axis. The user has the ability to also align the in-plane stress vectors to the Global Y and Global Z axes in a similar manner.

The sections below describe the options available for directional stress.

Global X

Global X via XZ/XY Cutting Plane

This will align the elements' in-plane stress vector to the global X direction based on largest projected area on the XZ or XY plane as described above.

Global X via XZ Cutting Plane

© 2014 Optimum Structural Design, Inc.

This will align the elements' in-plane stress vector to the global X direction based on the intersection of the XZ cutting plane and the element as described above.

Global X via XY Cutting Plane

This will align the elements in-plane stress vector to the global X direction based on the intersection of the YZ cutting plane and the element as described above.

Global Y

Global Y via XY/YZ Cutting Plane

This will align the elements' in-plane stress vector to the global Y direction based on largest projected area on the XY or YZ plane, in a similar manner to the above example, which was for the Global X direction.

Global Y via XY Cutting Plane

This will align the elements' in-plane stress vector to the global Y direction based on the intersection of the XY cutting plane, in a similar manner to the above example, which was for the Global X direction.

Global Y via YZ Cutting Plane

This will align the elements' in-plane stress vector to the global Y direction based on the intersection of the XZ cutting plane and the element, in a similar manner to the above example, which was for the Global X direction.

Global Z

Global Z via XZ/YZ Cutting Plane

This will align the elements' in-plane stress vector to the global Z direction based on largest projected area on the XZ or YZ plane, in a similar manner to the above example, which was for the Global X direction.

Global Z via XZ Cutting Plane

This will align the elements' in-plane stress vector to the global Z direction based on the intersection of the XZ cutting plane and the element, in a similar manner to the above example, which was for the Global X direction.

Global Z via YZ Cutting Plane

This will align the elements' in-plane stress vector to the global Z direction based on the intersection of the YZ cutting plane and the element, in a similar manner to the above

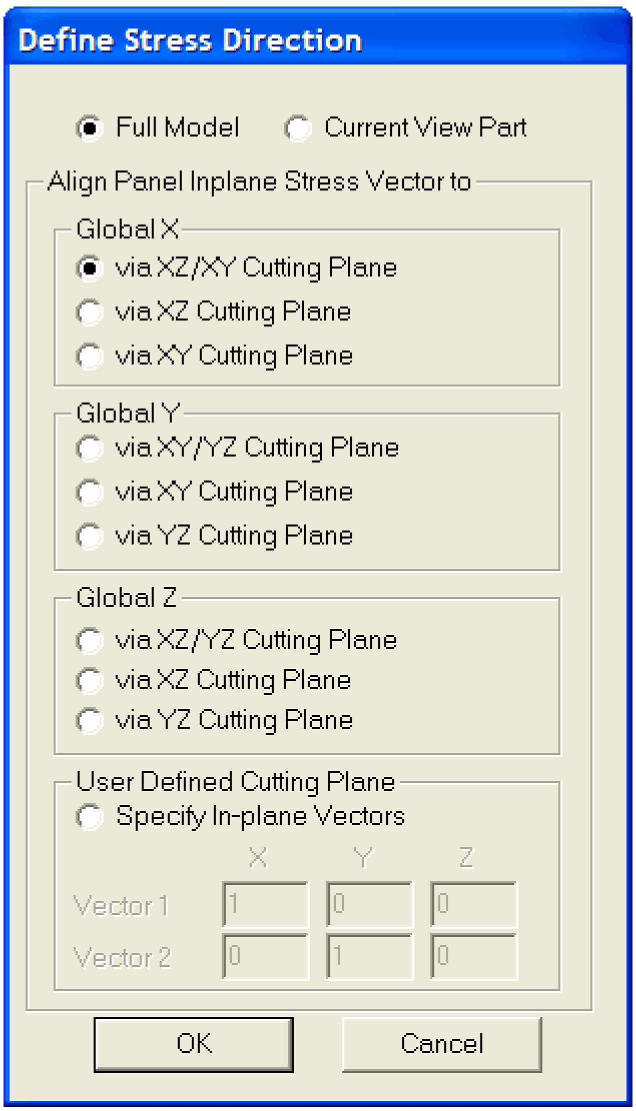
© 2014 Optimum Structural Design, Inc.

example, which was for the Global X direction.

User Defined Cutting Plane

This will allow the user to define the direction of two arbitrary vectors (Vector 1 and Vector 2), which determine a special cutting plane, to which the in-plane stress vector will be aligned.

The options for setting the stress vector are shown in the Define Stress Direction dialog, which can be opened from **Results > Stress > Define Direction...**



The stress direction can be either applied to the Full Model or the Current View Part by

© 2014 Optimum Structural Design, Inc.

clicking the appropriate radio button in the Define Stress Direction dialog. Stress direction settings can be set for multiple view parts and will be applied when viewing the entire model or multiple view parts.

As an example, the next two figures show a module with the elements stress vector first aligned in the global X direction (Figure 11) and the second aligned in the Global Z direction (Figure 12). Figure 11 shows the panel element's all have their local x axis orientated in the Global X direction, whereas Figure 12 shows the panel element's all have their local x axis orientated in the Global Z direction. In this example we would expect the sigma x stress of Figure 11 to be equal to sigma y (which is now in the direction of the Global X axis) of Figure 12 to be equal, and vice versa.

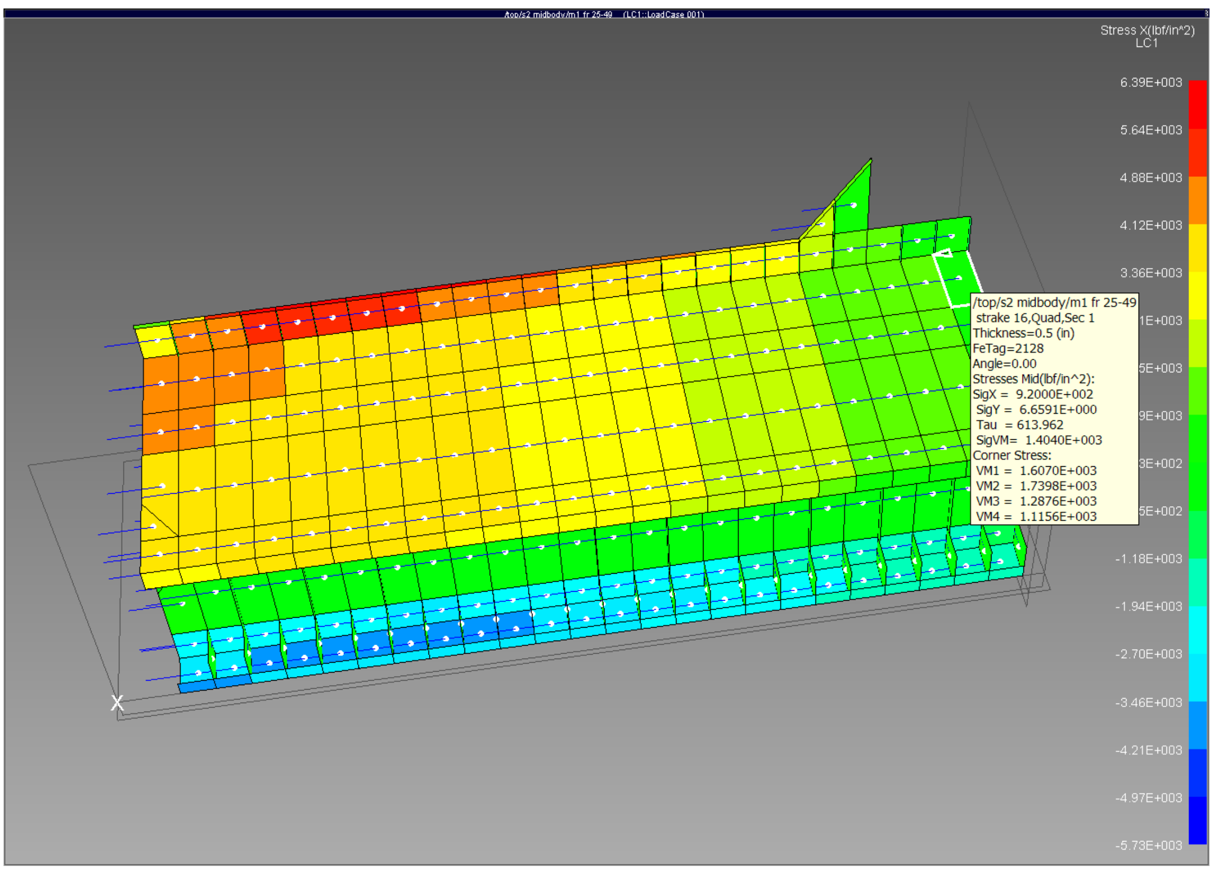


Figure 11: Align Stress to Global X, SigX = 9.2E02, SigY = 6.66, Tau = 613.96

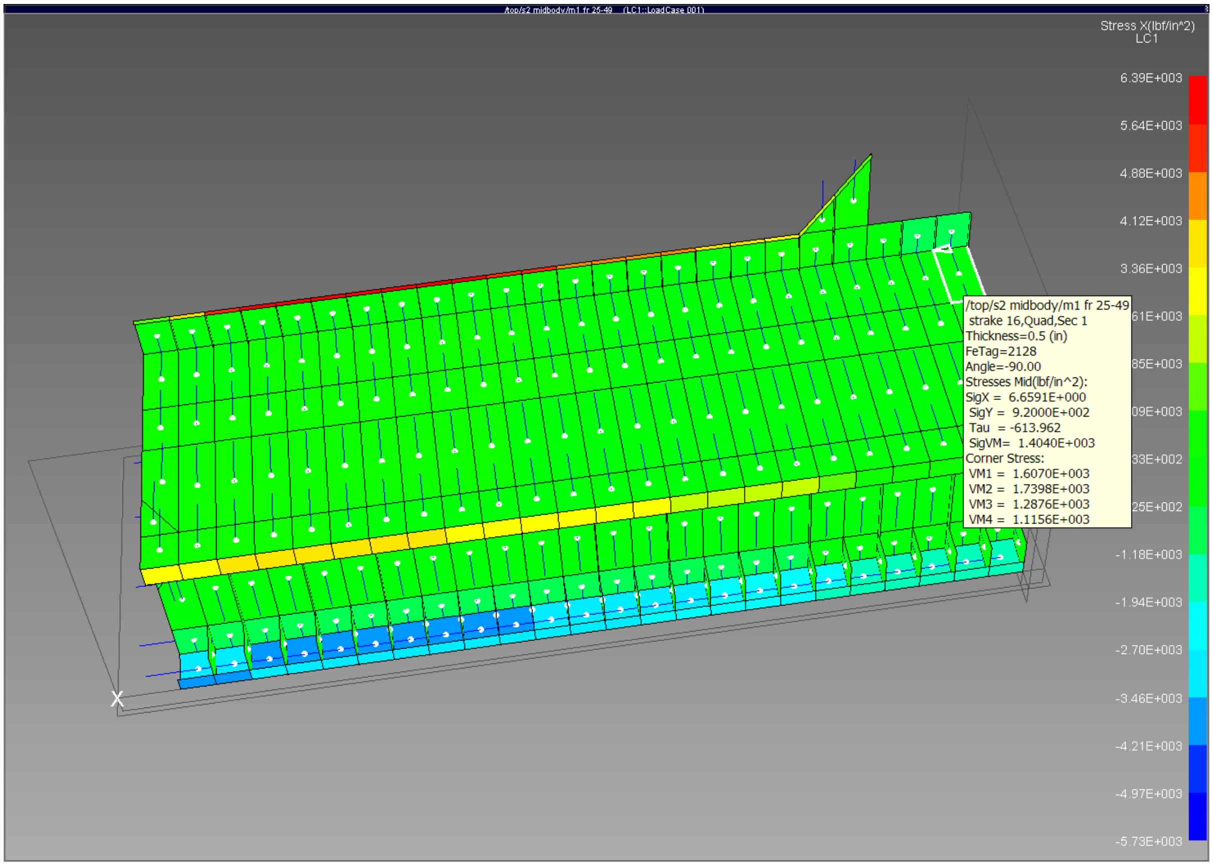


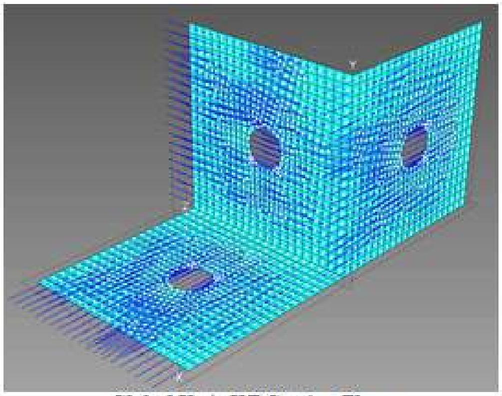
Figure 12: Align Stress to Global Z, SigX = 6.66, SigY = 9.2E02, Tau = -613.96

The directional stress option can be toggled on by selecting **Results > Stress > Directional** from the menu. A check mark will appear next to this option when it is turned on.

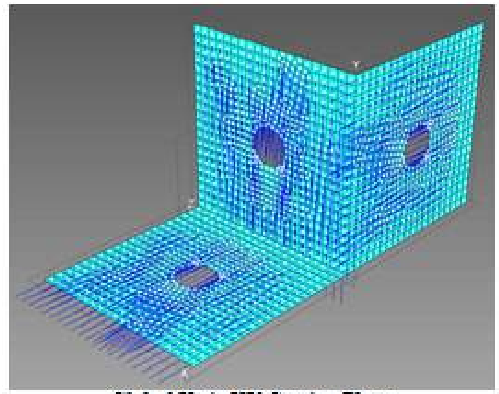
The direction for each element can be displayed with a quill by selecting **Results > Stress > Show Direction** from the menu. A check mark will appear next to this option when it is turned on. The length of the quill can be adjusted in the View Options dialog.

Figure 13 demonstrates the 6 user-defined stress directions on the directional stress.mdl model. The blue quills represent the new element local-x direction after the adjusted alignment. This model can be found in the MAESTRO installation directory under the *Models and Samples* folder.

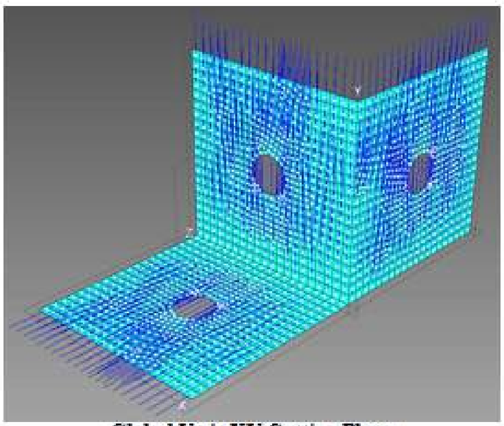
© 2014 Optimum Structural Design, Inc.



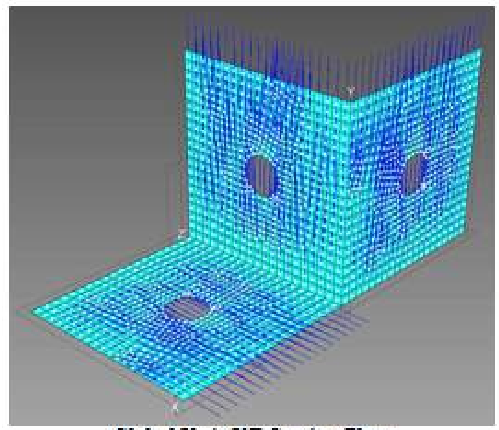
Global X via XZ Cutting Plane



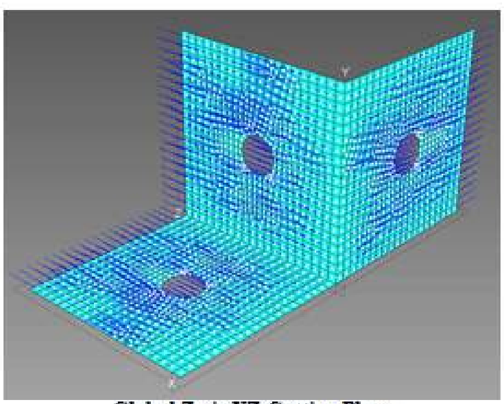
Global X via XY Cutting Plane



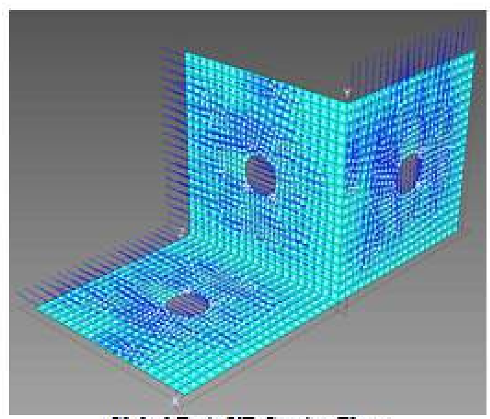
Global Y via XY Cutting Plane



Global Y via YZ Cutting Plane



Global Z via XZ Cutting Plane



Global Z via YZ Cutting Plane

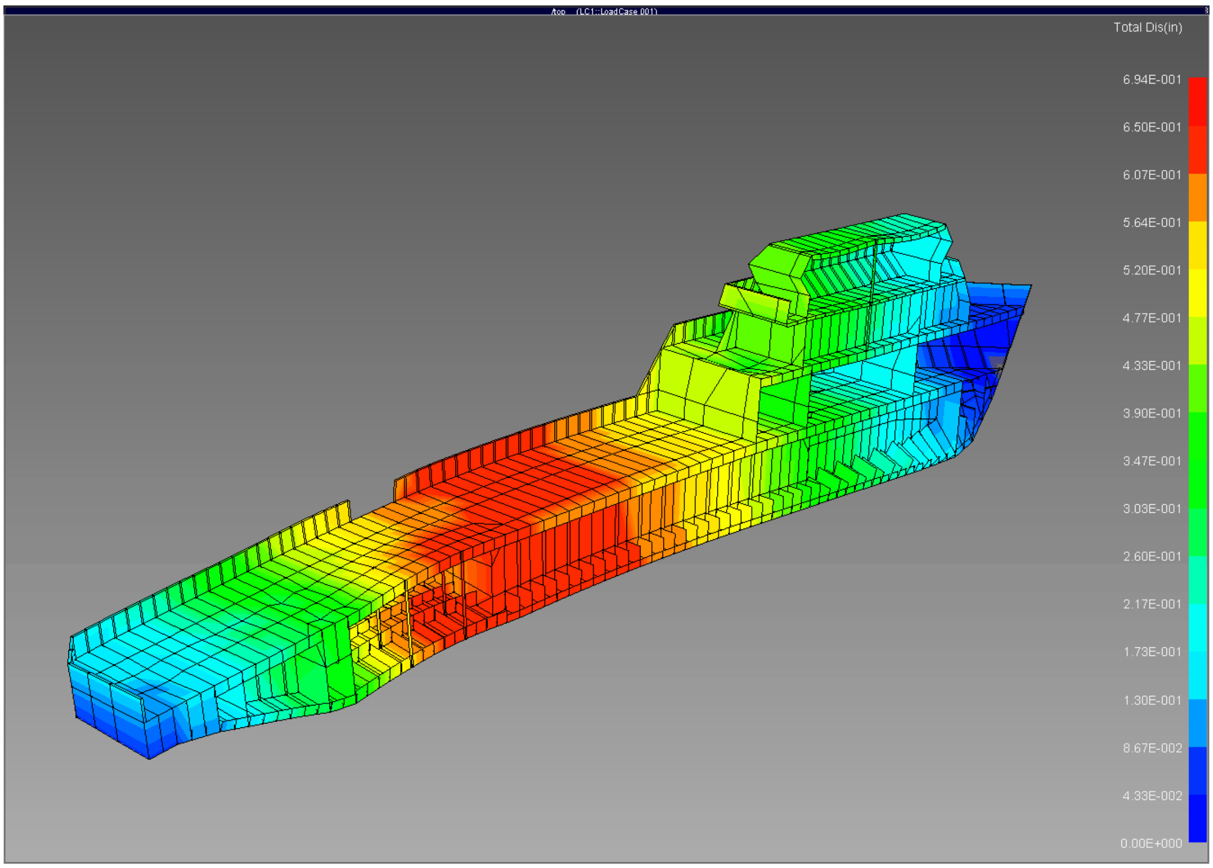
Figure 13: Demonstration of the 6 User-Defined In-Plane Stress Vector Orientations

10.4.5 Contour Plot

Toolbar	

Menu	Results > Contour

MAESTRO has the functionality to produce a contoured deformation or stress plot by toggling on the icon or selecting **Results > Contour** from the menu. A check mark will appear in the menu next to this option when contour view is toggled on.



Deformation Contour Plot

MAESTRO's contour plot is either a nodal contour, which averages all values at the nodes or an average elemental centroidal stress depending on whether the *Corner Stress* option is checked in the View Options dialog. Nodal contouring produces a smooth contour plot.

10.5 Analysis Results

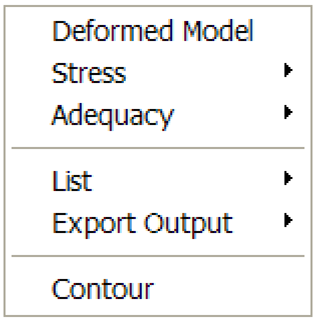
Toolbar	N/A

© 2014 Optimum Structural Design, Inc.

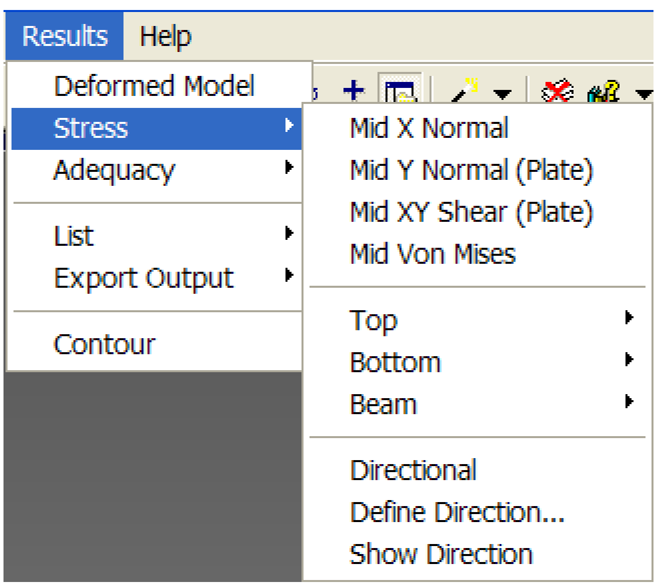
Menu	Results >

The following tutorials shows the basic functionality for viewing the stress and deformation results.

1. Click on the Result menu to open the drop down menu.



2. Click on Deform to show the model in a deformed or undeformed state.
3. Click on Stress and select which stress to display.

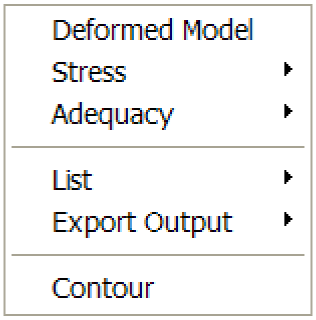


A detailed description of each type of stress that can be recovered can be found in the Stress Results sections.

4. A contoured stress plot can be shown by clicking the Contour Plot icon  or selecting **Results > Contour** from the menu.

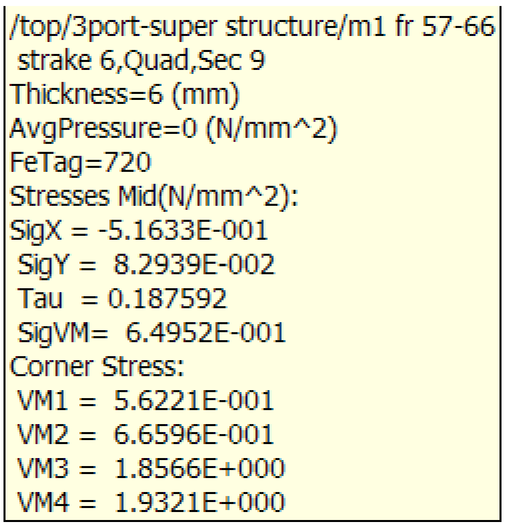
Individual element deformations and stresses and nodal deformations and forces can be identified using the dynamic query function of MAESTRO.

5. Click the dynamic query icon .



6. Select Node to query nodes, otherwise select the results you would like to query from the element. A check mark will appear next to the selected option.

7. You may now highlight an element or node to view the results.



Double-clicking on a node or element will output the results to the output window at the bottom of the screen.

8. All element stress results, nodal forces, reaction forces, or negative adequacy parameter report can be output to the grid or output tab by selecting from the **Results > List >** menu. This spreadsheet can be easily copy and pasted into another program if desired.

In order to output nodal forces, the option *Total Point Force* must be checked under the Loads menu before listing nodes.

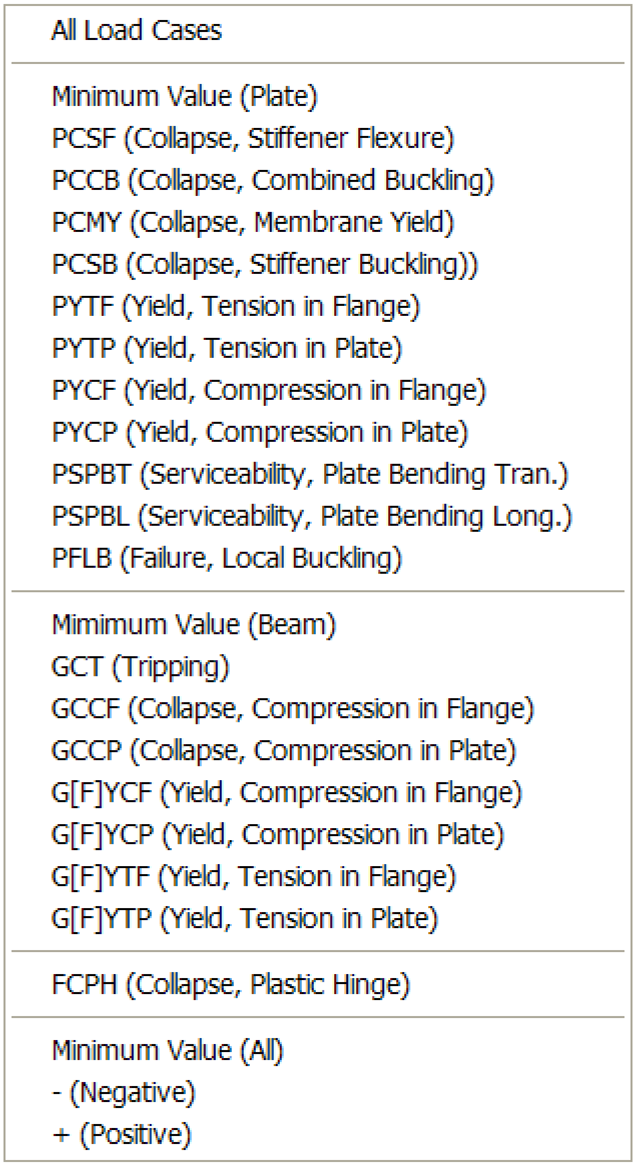
The negative adequacy option will list the number of elements with a negative adequacy parameter and those elements percentage of the total structural weight for the selected load case.

If a Failure Mode Evaluation method was selected for the analysis, the results can be

© 2014 Optimum Structural Design, Inc.

graphically displayed from the **Results > Adequacy** menu. The adequacy parameters menu will be different depending on whether MAESTRO or ULSAP evaluation is chosen.

MAESTRO



ULSAP

All Load Cases

Minimum Value (Plate)
PCPM (Plate induced failure at Midspan)
PCCB (Overall Grillage Collapse)
PCPE (Plate induced failure at Panel Edges)
PCSB (Stiffener induced failure-tripping)
PCWB (Panel Collapse Web Buckling)
PYM (Yield in Mid-Plane)
PYF (Yield in Flange)
PYP (Yield in Plate)

Mimimum Value (Beam)
BCT (Tripping)
BYC (Gross Yielding)
BCWB (Web Buckling)
BCC (Collapse, Beam-Column)

FCPH (Collapse, Plastic Hinge)

Minimum Value (All)
- (Negative)
+ (Positive)

Selecting one of the parameters will display the patches colored by their adequacy parameter value. MAESTRO can automatically display each elements smallest adequacy parameter by selecting Minimum Value (Plate), Minimum Value (Beam), or Minimum Value (All). You can also choose to display only the Positive or only the negative adequacy parameter results.

The dynamic query icon (with *Adequacy* checked from the drop-down) can be used to highlight an element and recover the adequacy parameters. Double-clicking the element will echo the results to the Output tab.

10.6 Viewing Stress Ranges

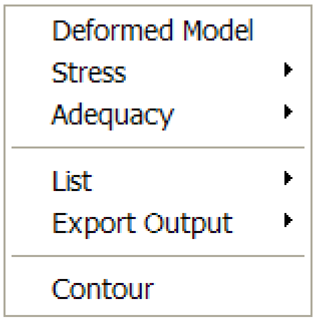
Toolbar	

© 2014 Optimum Structural Design, Inc.

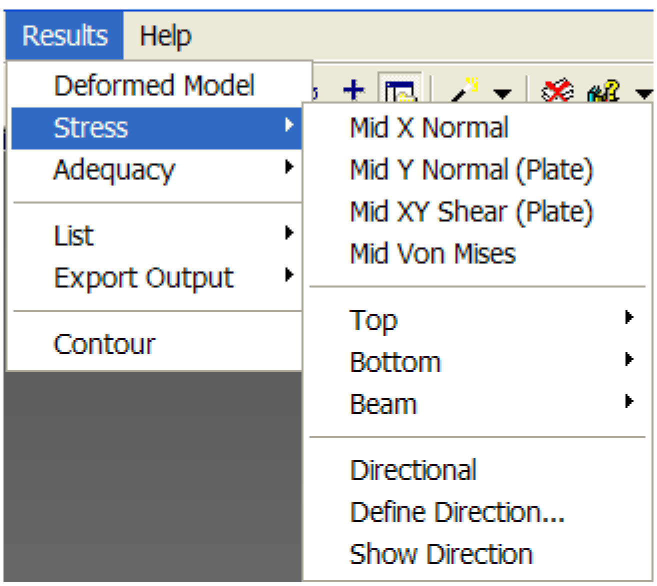
Menu	**View > Options**

The following tutorials shows the basic functionality for viewing stress ranges for a given load case. The user will first define and apply a range and then optionally hide any elements outside of this defined range.

1. After the results have been loaded, click on the Result menu to open the drop down menu.



2. Click on Stress and select which stress to display.

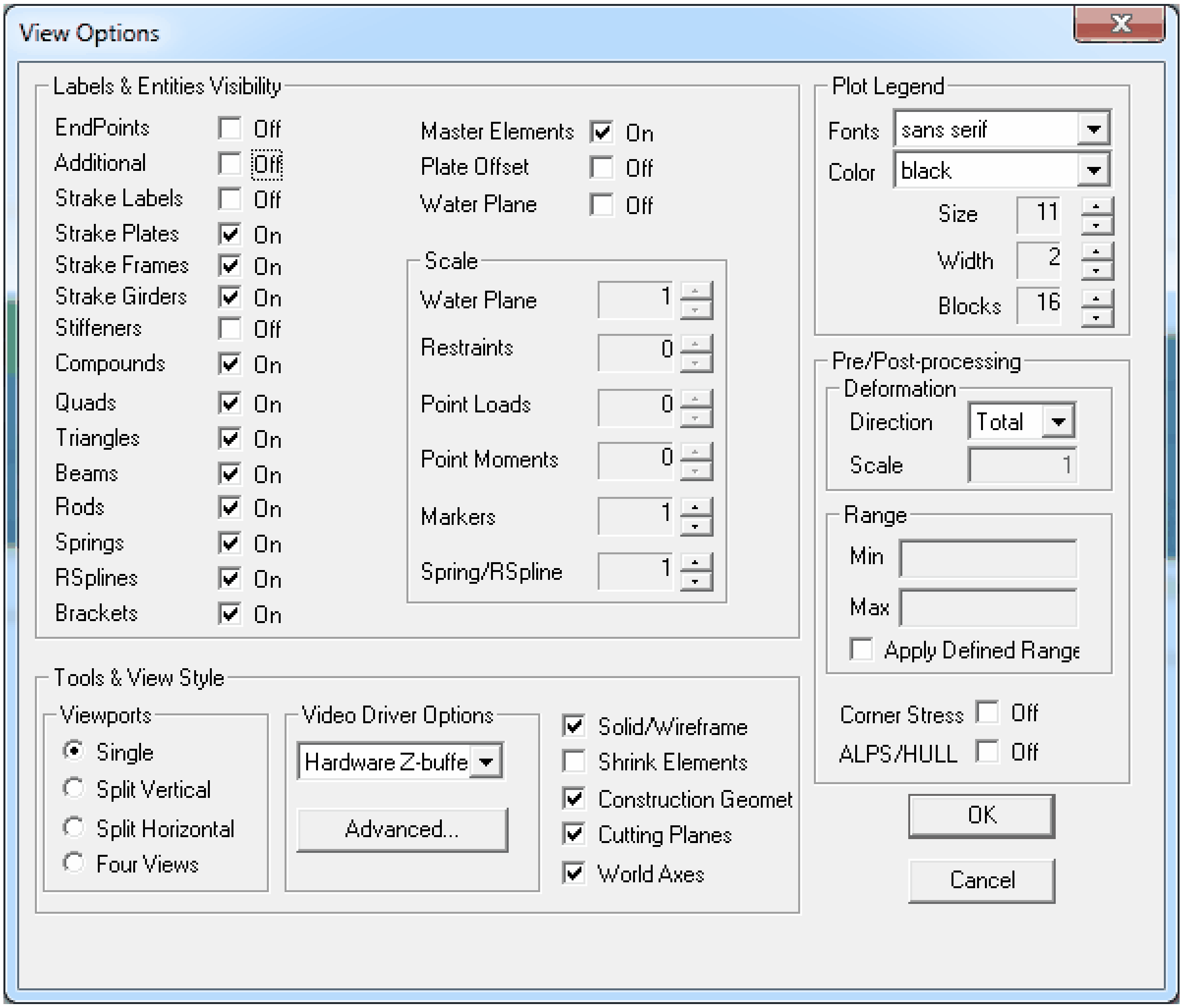


A detailed description of each type of stress that can be recovered can be found in the Stress Results sections.

3. Click on the View Options icon  or select **View > Options...** from the menu.

4. From the View Options dialog (see below), click the *Apply Defined Range* checkbox. This will enable the *Min* and *Max* Range fields for the user to define the stress range of interest.

© 2014 Optimum Structural Design, Inc.



5. Click **OK** when finished.

Individual element stresses for the defined range are now presented. Elements that are outside of the user-defined range are colored either blue (below range) or red (above range).

6. To hide the elements outside of the user-defined stress range, click the  icon.

10.7 Viewing Stress in a Given Direction

The following tutorials shows the process of viewing element stresses in a given direction once the model is solved and the results are loaded.

© 2014 Optimum Structural Design, Inc.

1. Create a general group for the plate elements of interest.
2. Define the top layer fiber orientation.

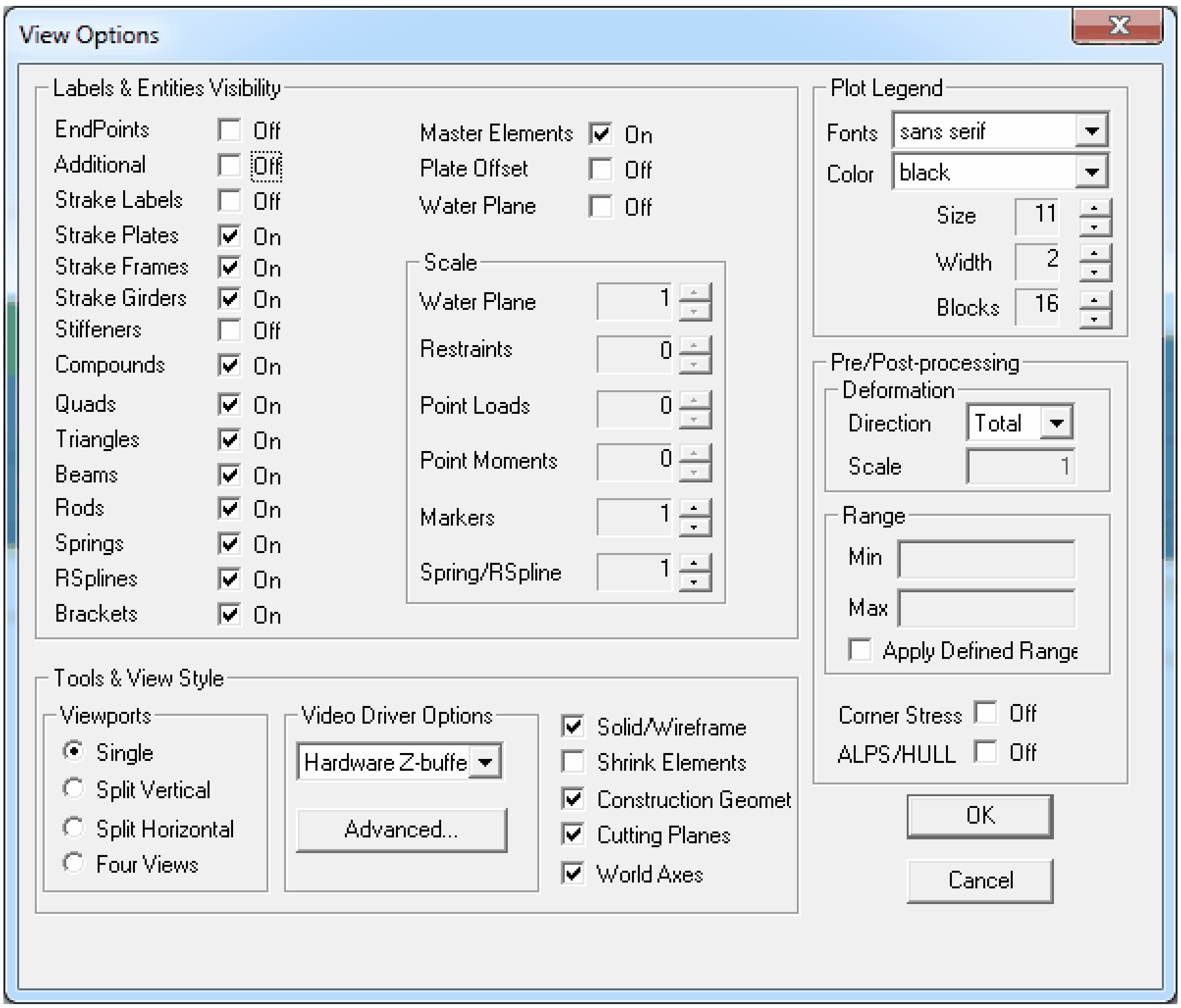
   a. Right-click on the group name in the group tab of the parts tree and select "Modify".

   b. Check the "Orientation" checkbox.

   c. Define a desired global orientation for the group.

   d. Click OK.

3. Launch the View Options dialog .



4. Click the Direction checkbox once. "Off" will change to "Fiber" next to the checkbox. (A small tick should be shown at each element centroid. You can scale the tick by increasing/decreasing the point load scale on the right side of the View Options dialog box.)

5. Once the orientation is verified, click the Direction checkbox once more to change it from "Fiber" to "Stress".

6. Click OK.

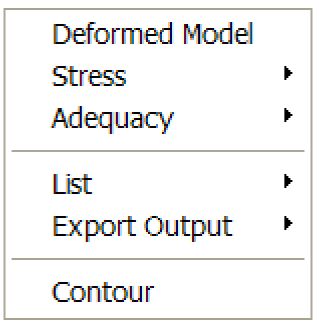
7. The displayed Stress X is now in the given direction.

10.8 Viewing Areas of Interest

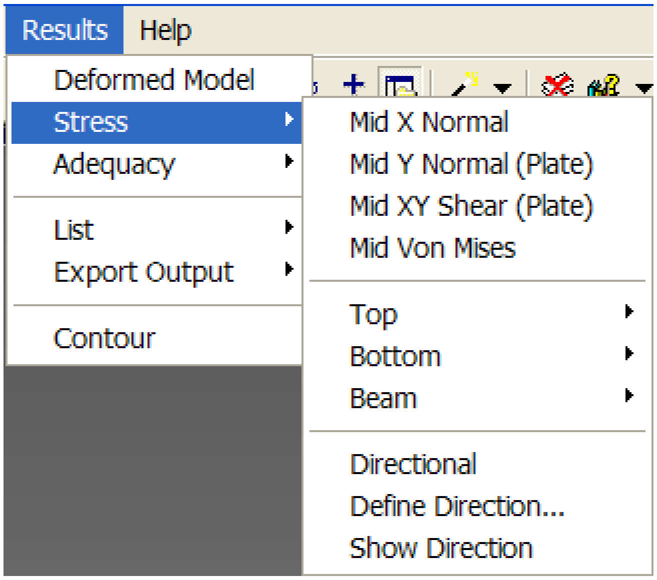
Toolbar	
Menu	View > Options

The following tutorials shows how a user can post-process a given area of interest. This is useful when the analyst would like to isolate a particular area to view deformation or stress.

1. After the results have been loaded, click on the Result menu to open the drop down menu.



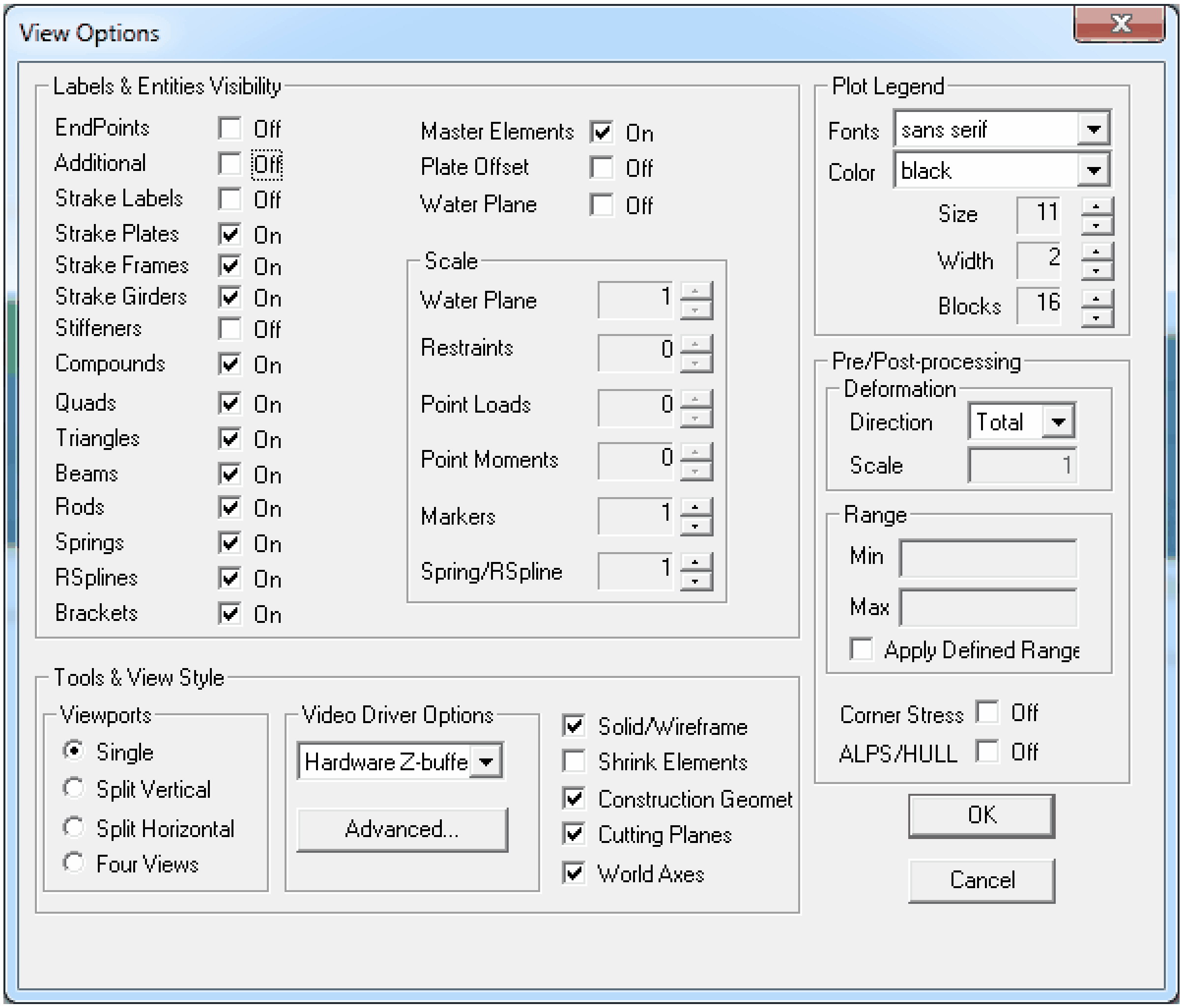
2. Click on Stress and select which stress to display.



A detailed description of each type of stress that can be recovered can be found in the Stress Results sections.

3. Click on the View Options icon  or select **View > Options...** from the menu.

4. From the View Options dialog (see below), click the *Apply Defined Range* checkbox. This will enable the *Min* and *Max* Range fields for the user to define the stress range of interest.



5. Click **OK** when finished.

Individual element stresses for the defined range are now presented. Elements that are outside of the user-defined range are colored grey.

6. To hide the elements outside of the user-defined stress range, click the  icon.

10.9 Recovering Composite Layer Stresses

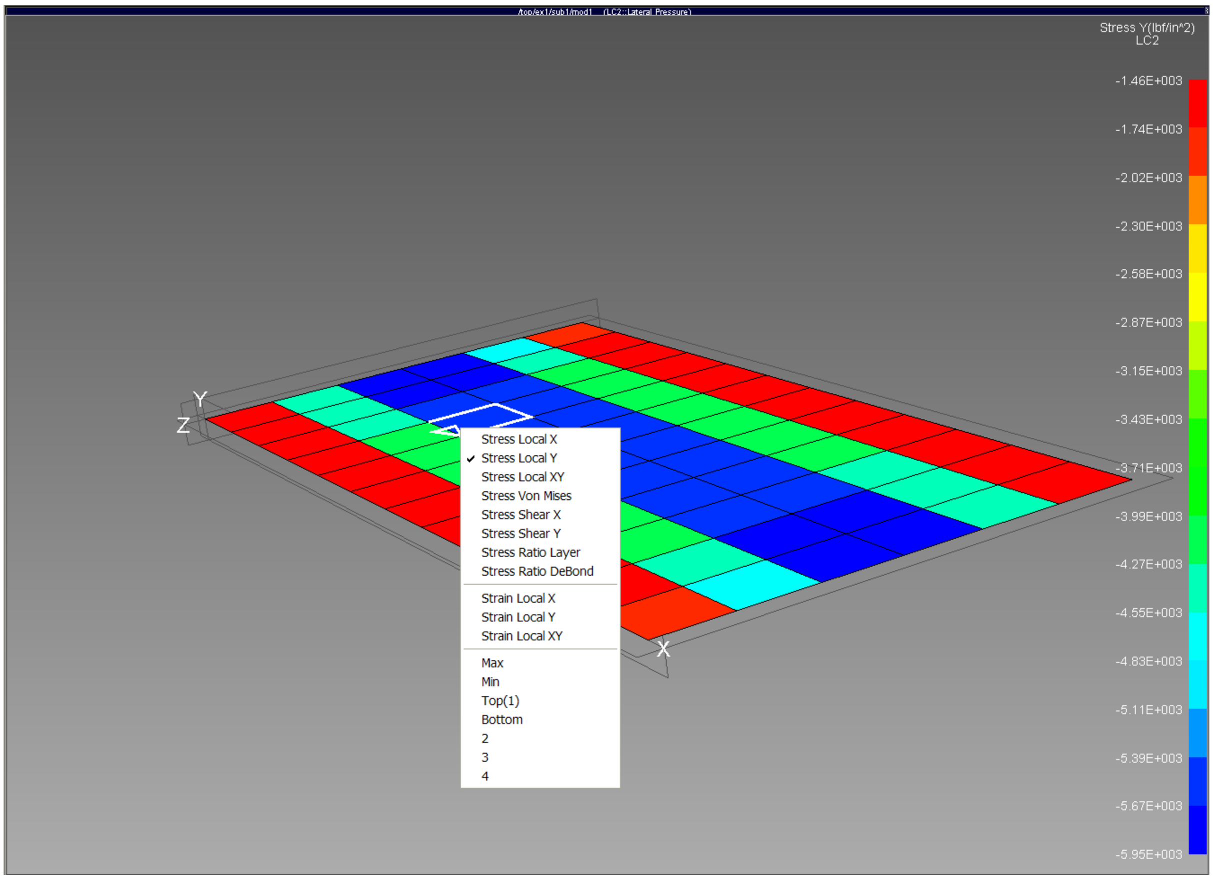
Stresses can be recovered in each layer of a composite shell element. The following steps will detail the procedure to recover stresses from each of the layers.

1. Solve the model.
2. Select one of the stress options from the **Results > Stress** menu.

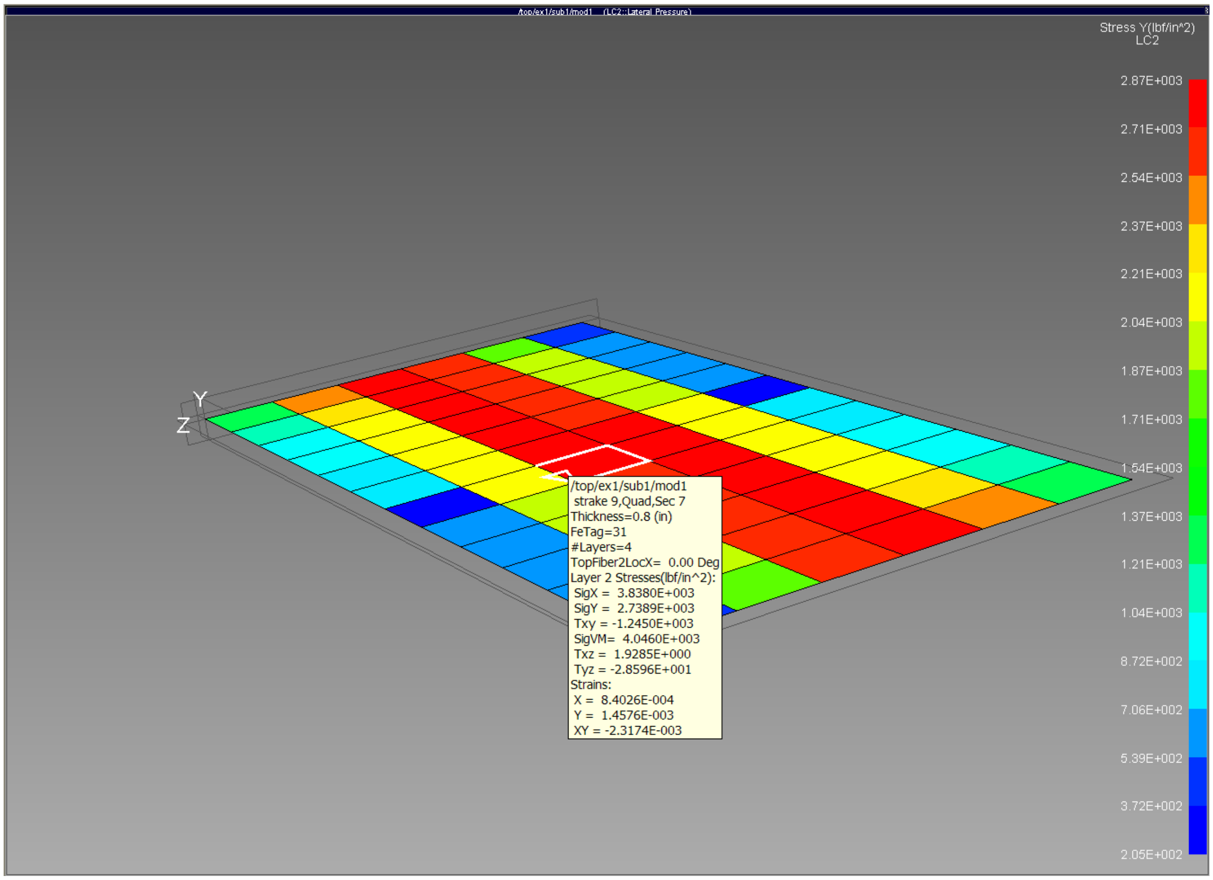
3. Click the Layers icon  to turn on layers.

4. Click the dynamic query icon .

5. Highlight an element and right-click to bring up the menu to choose which stress direction to display as well as to choose the composite layer at the bottom of the menu.



6. Once a stress option is chosen from the dynamic query fly-out menu, highlighting an element will now report the chosen layer stress.



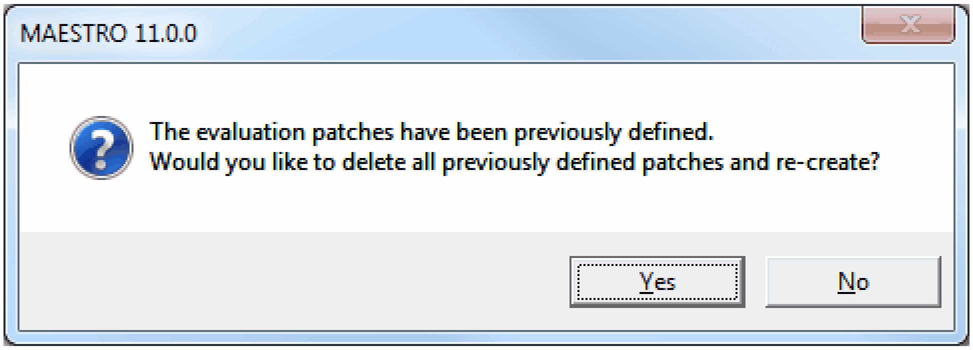
The model used in this sample (Composite_4Layer.mdl) can be found in the *Models and Samples* MAESTRO installation folder for reference.

10.10 Evaluation Patch (E-patch)

Central to the structural assessment paradigm in MAESTRO, is the concept of the Evaluation Patch (E-patch). The E-patch is a strength assessment entity that serves to represent the true stiffened panel, panel between stiffeners, stanchion span and/or beam column span that are located throughout a global the FEM. These Epatches are created by automatically searching the entire FEM and collecting (if necessary) the multiple finite elements (plates or beams) to represent the true panel geometric parameters. Thus, the true boundary conditions and true structural spans are calculated, stored, and subsequently used to calculate the limiting structural failure modes throughout the FEM.

10.10.1 Defining Evaluation Patches

When running an analysis with one of the failure mode options selected, MAESTRO will automatically check if evaluation patches are defined. If no evaluation patches are defined in the model, MAESTRO will automatically search and create patches using the Level 0 option. If patches were previously defined, the following will prompt the user:



Note: If "Yes" is clicked, all previous evaluation patch information will be erased and the Level 0 patches will automatically be found and created and the analysis will run.

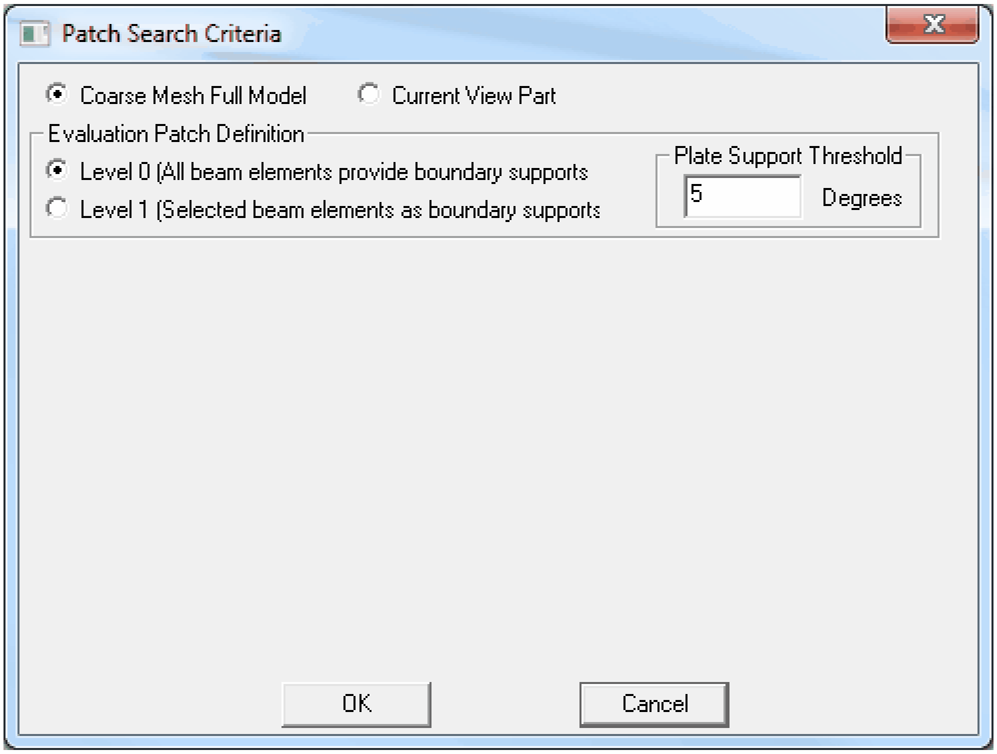
All evaluation patches in a model can also be deleted by selecting Model > Evaluation Patch > Delete All.

There are four ways to create evaluation patches in MAESTRO, which are described in more detail below:

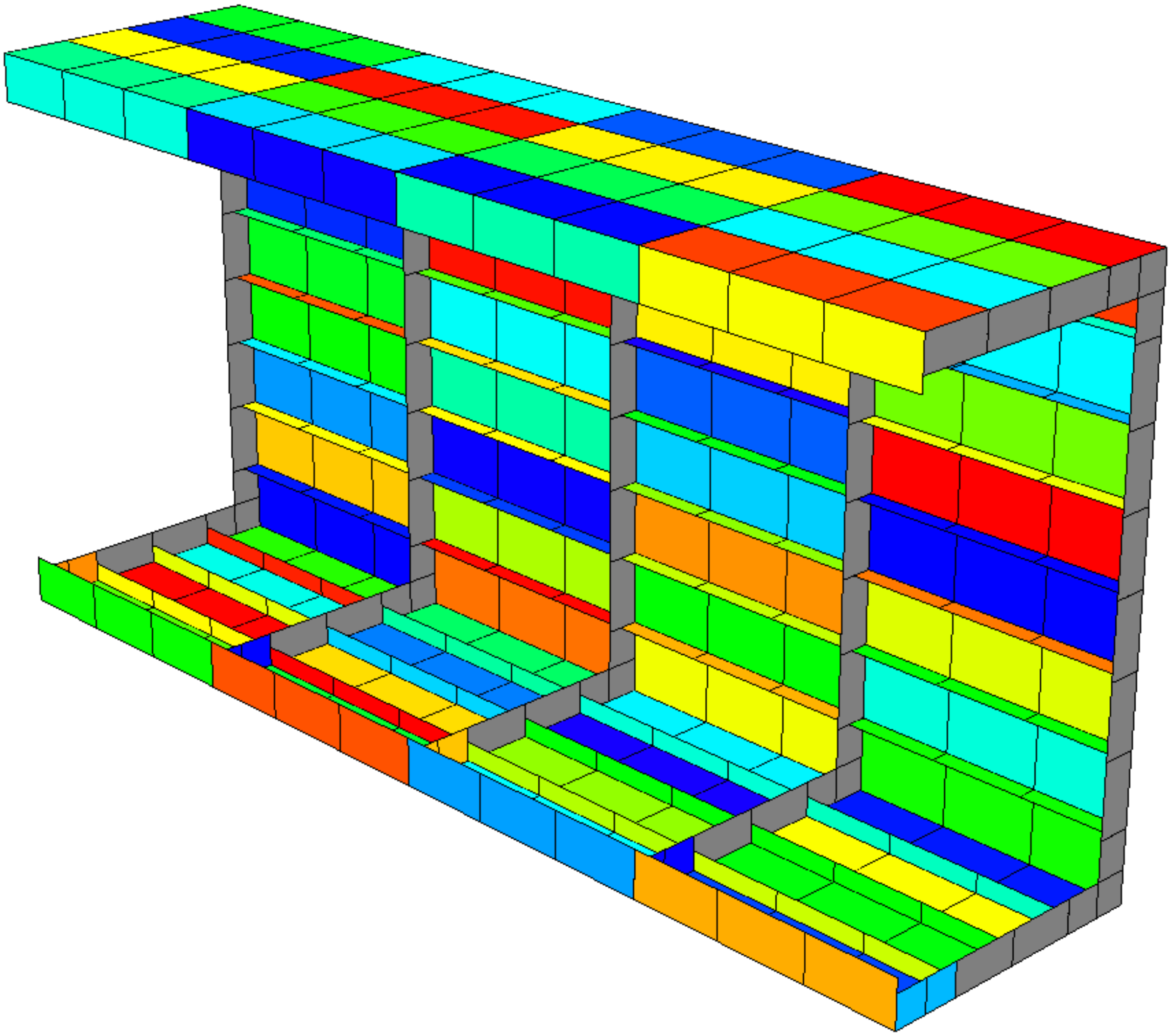
Level 0 Patch Automatic Search

The level 0 patch automatic search option can be accessed from the prompt when running an analysis with a failure mode evaluation option selected or from the Model > Evaluation Patch > Auto-Generate menu option. The menu option method will launch the Patch Search Criteria dialog:

© 2014 Optimum Structural Design, Inc.

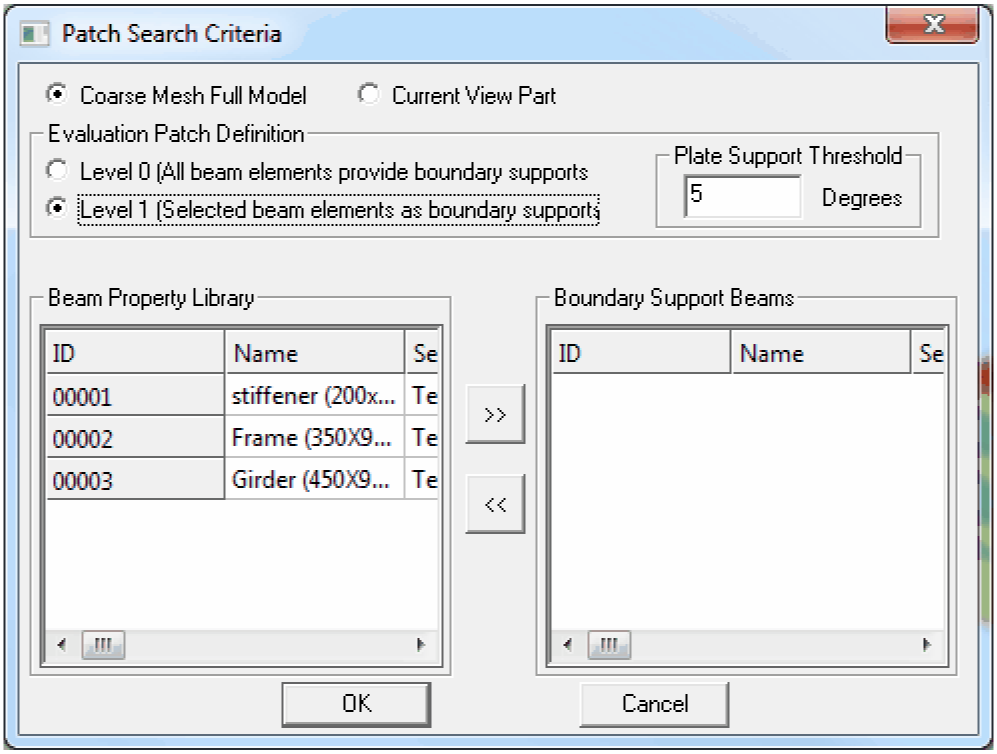


The level 0 search can be performed on the full coarse mesh model, or the current View Part only. This method will search the model and define evaluation patches based on the boundaries created by beam elements and changes in plate support direction (e.g., the intersection of the side shell and the main deck will be treated as a boundary). The threshold for the definition of a plate support boundary can be set by the user. This value is defined as any plate supports with a change in direction greater than the threshold value will be considered a boundary. An example of a level 0 patch search is shown below:



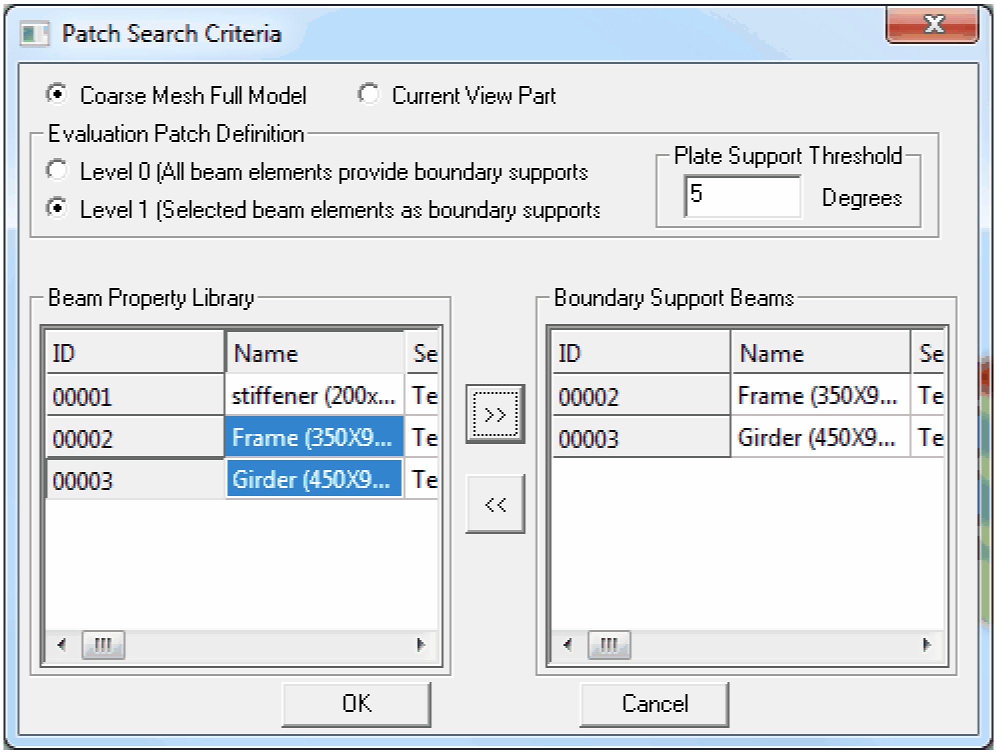
Level 1 Patch Automatic Search

When a model is more refined and stiffeners are now defined as beam elements, like the example module above, the level 1 patch search allows the user to quickly define patches representing the true span of a stiffened panel. Using the Patch Search Criteria Dialog, the radio button next to Level 1 will change the dialog when it is selected.

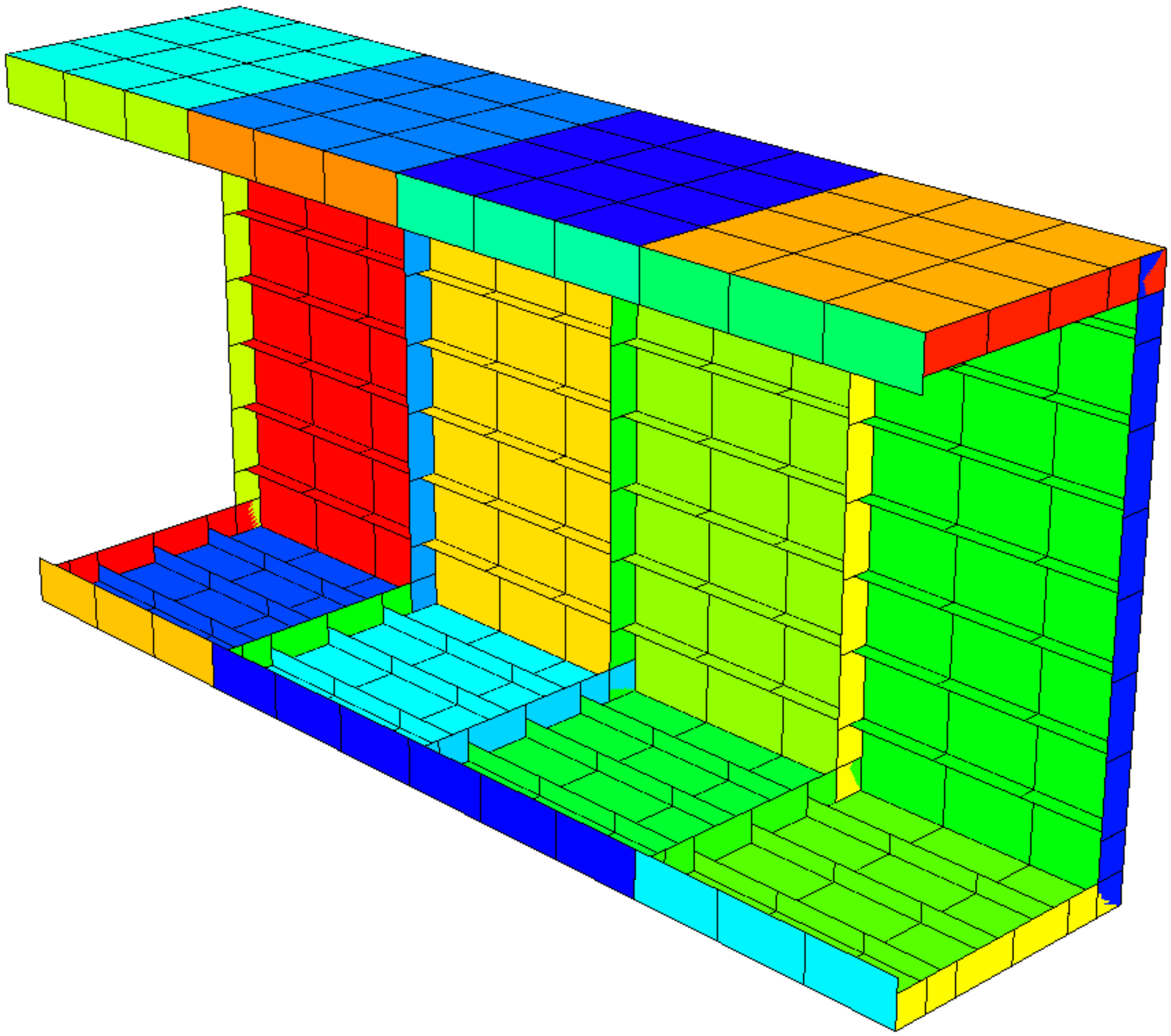


Now instead of using all beam elements as boundaries, the level 1 option allows the user to select the beam properties to be used in the search. Using the sample module again, the frame and girder beam properties can be selected and added to the "Boundary Support Beams" list by clicking on the item (multiple beam properties can be selected and moved at once) in the "Beam Property Library" list and clicking the ">>" button.

© 2014 Optimum Structural Design, Inc.



Beam properties can be removed from this list by using the "<<" button. Click OK to perform the search. The image below shows the evaluation patches created using the level 1 search option:



The evaluation patches now represent the true stiffened panel (including the beam elements representing stiffeners).

Manual Patch Creation

The manual patch creation method can be used in conjunction with the level 0 and level 1 automatic search algorithms. This operation will overwrite the applicable existing patches used to create the manual one.

A manual patch can be created by following these steps:

1. Create a general group of the elements making up the patch.

2. Expand the group tree and select the created group.

© 2014 Optimum Structural Design, Inc.

3. Right-click and select "Create Patch" from the menu.

4. This will open the Evaluation Patch dialog showing the new patch parameters.

Evaluation Patch without Geometry

The following steps walk through the process of creating an evaluation patch interactively through the Evaluation Patch Dialog:

1. Open the "Limit State Creation/Evaluation" dialog box.

2. Click the ID button.

3. Change the radio button from "Auto" to "User defined".

4. Enter the desired data.

5. Click Create.

10.10.2 Element Query & Collection

Fundamentally, the E-patch is a collection of elements with its boundary supported by bulkheads or beams. It should be noted that an E-patch can also be a single element if that element meets the boundary support criteria. In traditional MAESTRO, an E-patch was a lengthwise *strake* panel or a few strake panels, if the section per bay was defined as greater than one (this will make sense to the seasoned MAESTRO user). The E-patch searching algorithm queries each element within the FEM to:

- Identify possible element free edge(s),
- Identify element supports (e.g., is the element edge bound by beam elements, is the element edge bound by a BHD, etc.),
- Identify element material, scantlings,
- Identify stiffener definition (e.g., stiffener layout),
- Identify element length/width,
- Check element collection flatness,
- Identify if an element is evaluable,
- Determine beam-column effective width,
- Determine the directional stress and average panel pressure.

Based on the query findings, the E-patch is created and parameters stored. It should be noted that the E-patch generation process can be used with imported NASTRAN FEMs. Because most NASTRAN explicitly model beams to represent secondary members, the E-patch algorithm will only collect elements between stiffeners (i.e., local plate between stiffeners).

Effective Plating
The collection of beam elements into E-patches has one additional component that addresses the need to calculate *effective plating* attached to a beam (i.e., plate-stiffener column). This provides a thorough method to compute the effectiveness of plating that acts in conjunction with a stiffening member. To do this, the E-patch searching algorithm finds each beam and the associated plate elements. The associated plate elements are the

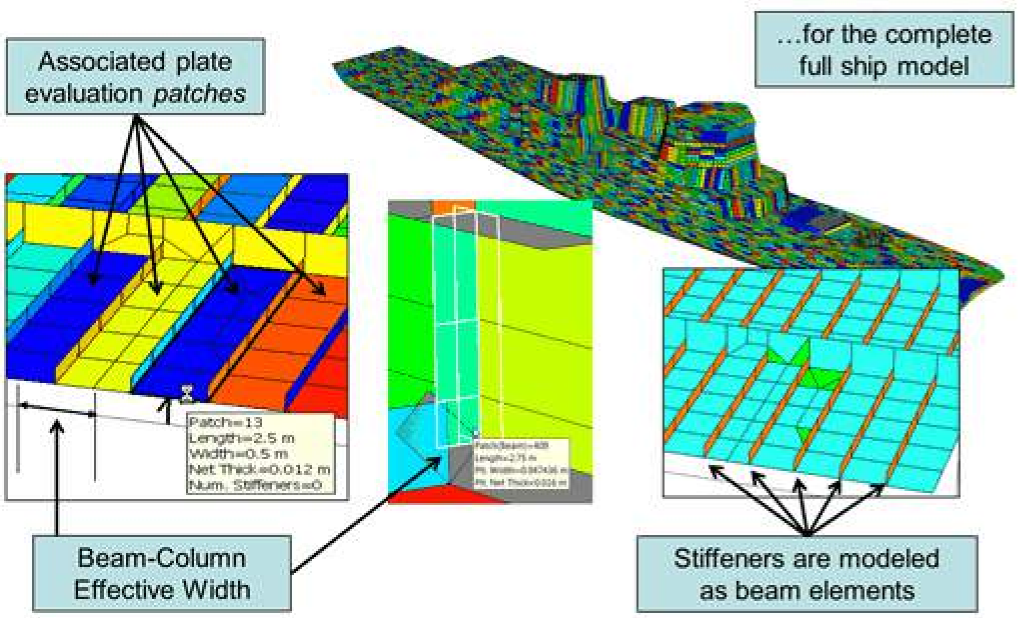
© 2014 Optimum Structural Design, Inc.

collection of plate elements (i.e. panel E-patch) on either side of the beam element. This information (i.e., breadth between stiffener), in addition to other recovered parameters (e.g., beam span (L), E (modulus of elasticity), and F_Y (material yield)) are used to find the lesser value of the following:

- Post-buckling, $b_e = 2t\sqrt{F_Y/E}$
- Average breadth between neighboring spacing, $b_e = 1/2(b_1 + b_2)$
- 1/3 of beam span, $b_e = L/3$

As stated previously, E-patches are defined automatically and can graphically be presented by a quilt of colors (see the figure below). This figure demonstrates, there can be hundreds, if not thousands, of generated E-patches throughout the FEM. The E-patch has been exercises with FEM that had over 1 million elements.

Finally, regarding Element Collection for E-patches, it should be noted that the user has the ability to overwrite the E-patch definition if so desired. The steps to perform manual E-patch creation are documented in the following sections of the Help manual.



10.10.3 Element Scantlings & Stress Recovery

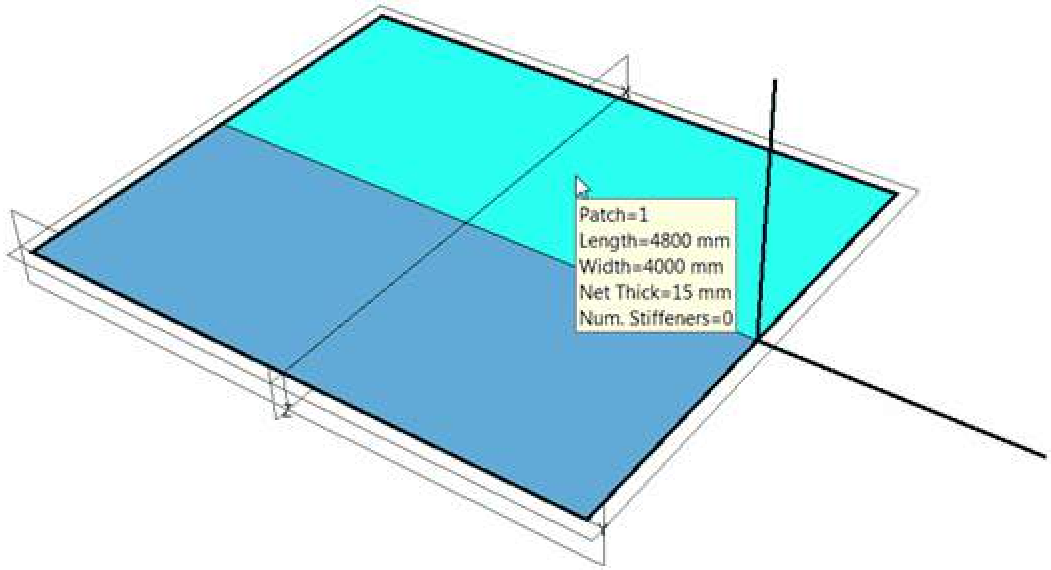
After elements are queried and collected, it is the job of the E-patch algorithm to determine the representative scantlings and stresses amongst the E-patch. The sections below discuss the various scenarios an FEM may present in this regard and is important for the analyst to understand.

Scantling Determination

If scantlings of individual elements within an E-patch definition are different, MAESTRO

© 2014 Optimum Structural Design, Inc.

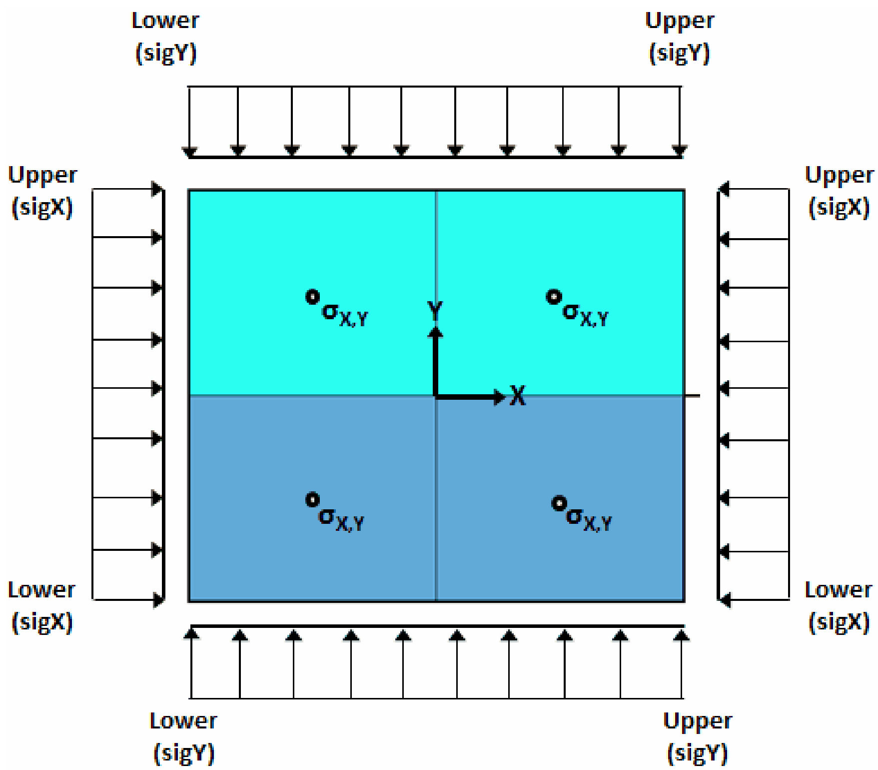
must determine what scantlings to use for the strength assessment of the E-patch. For MAESTRO's native strength assessment limit states as well as the ALPS/ULSAP method of failure mode evaluation implemented in MAESTRO, the approach used for determining the E-patch scantling (i.e., in this scenario) is a weighted average (e.g., $\sum(Plate\ Thickness * Plate\ Area)/(Total\ EPatch\ Area)$).



Stresses of individual elements within an E-patch definition are also likely to be different; therefore, MAESTRO must determine what stresses to use for performing the strength assessment within the Epatch.

MAESTRO Native Limit State Evaluation

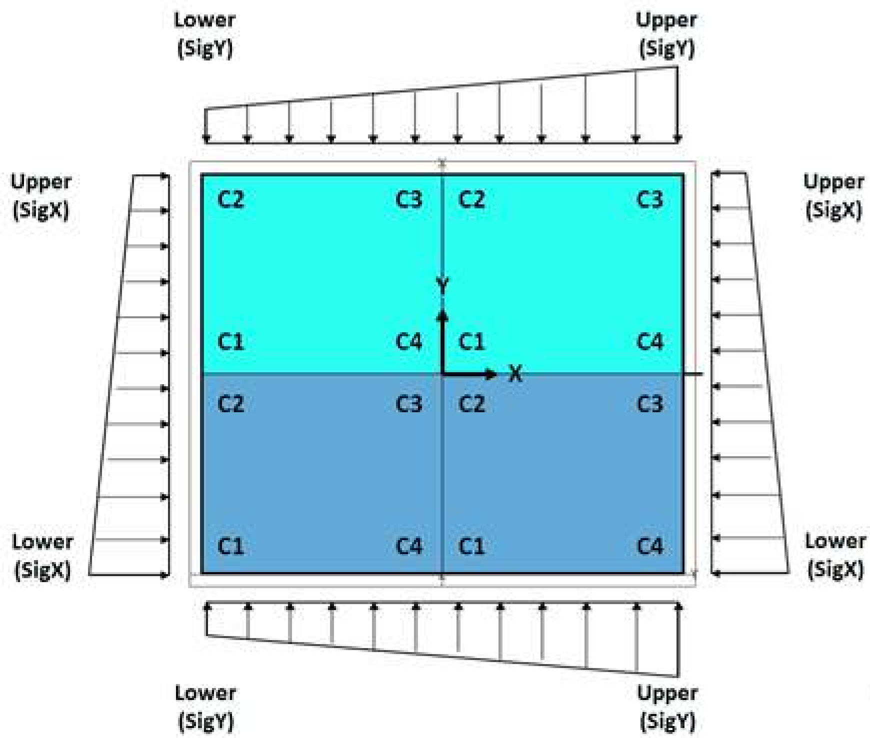
For MAESTRO's native strength assessment limit states (see image below), the approach used for determining the E-patch stress is a weighted average of the directional stress (i.e., stress vectors aligned with E-patch orientation). The approach for the ALPS/ULSAP method is slightly different. For this strength assessment method, the E-patch stress collection finds the nodal stress along the E-patch edge. The edge nodal stresses are then averaged and the SigX Upper, SigX Lower, SigY Upper, and SigY Lower are thus defined (see the figures below).



MAESTRO E-patch Stress

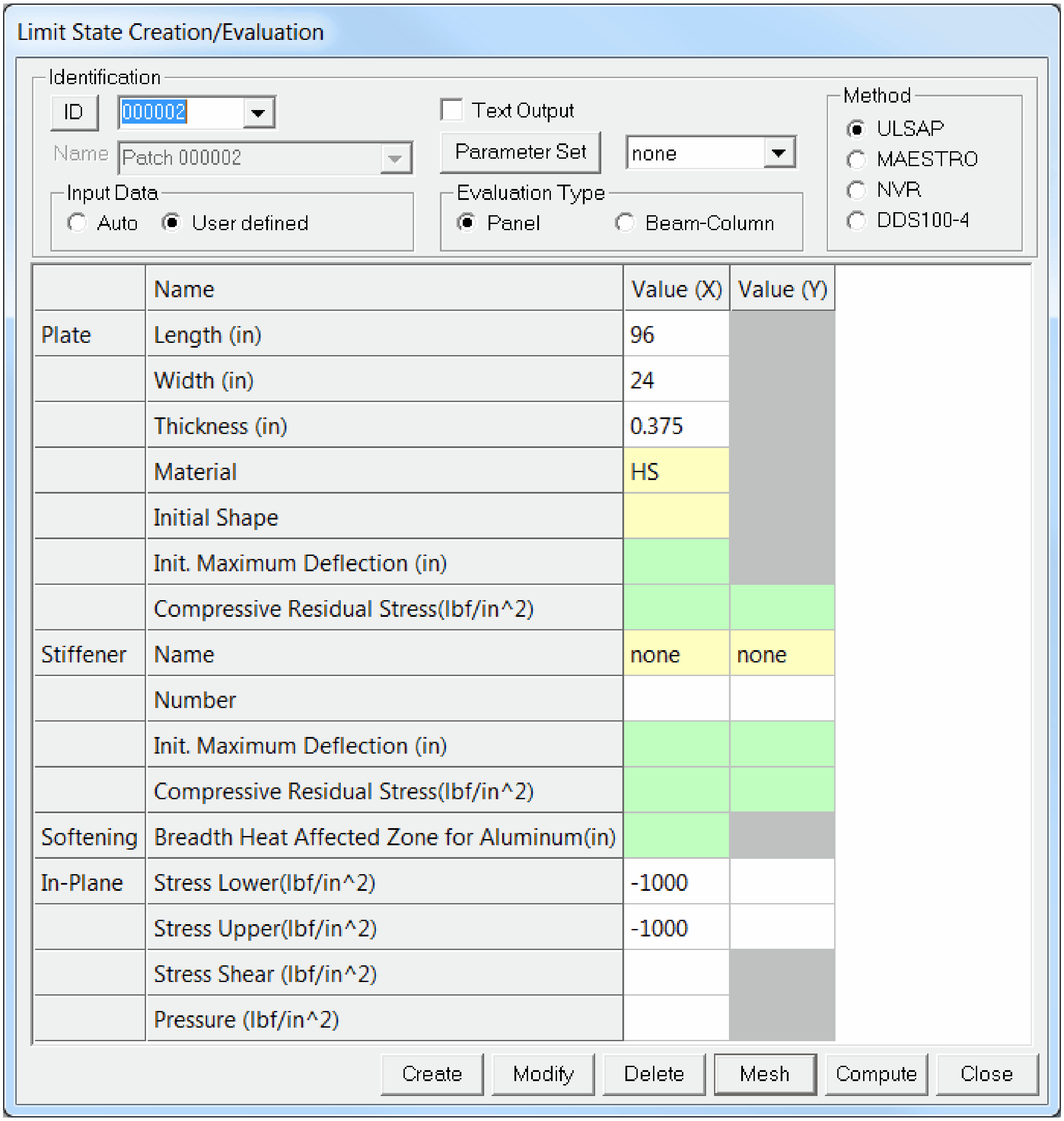
ALPS/ULSAP Limit State Evaluation

For the ALPS/ULSAP strength assessment, the figure below shows an E-patch that has four CQUAD4R (Quad) elements. Two of these elements were defined via a Strake and two were defined via an Additional Quad. Each of these elements has their own stresses (e.g., SigX, SigY, SigXY) and each node (or corner) that defines the element also has stress values. Further, for an individual Quad element there are four nodes representing the corners (C1, C2, C3, and C4). These corners are shown in the figure below.



ALPS/ULSAP E-Patch Stress

The E-patches can be further post-processed by accessing the individual E-patch definition, which is exposed in MAESTRO's Limit State Creation/Evaluation dialog (see figure below). Here the user can review the E-patch definition parameters such as Length, Width, and Stresses. The design pressure input value is only used in the calculation of the panel adequacy parameters and does not affect the general finite element analysis. The user has the ability to redefine manually the E-patch parameters by using the User Defined button and directly inputting the appropriate values. Once these values are defined, the user can click the Compute button, which accesses a summary of the strength assessment computations. Note that these values can be presented, via the Compute button, when the Auto button is selected in the Limit State Creation/ Evaluation dialog as well.



10.10.4 Working with Evaluation Patches

Toolbar	
Menu	**Model > Evaluation Patch > Create/Evaluate**

© 2014 Optimum Structural Design, Inc.

The following tutorial describes what an evaluation patch is and how it is used in MAESTRO.

How do you automatically define patches?

From the main menu, select Model>Evaluation Patch>Auto-Generate. Patches can also be automatically generated when evaluation is selected in an analysis.

How do you delete all patches?

From the main menu, select Model>Evaluation Patch>Delete All.

How do you manually create a patch?

1. Create a general group. It is suggested the group to be named as “Eval/abcdefg1”.
2. Expand the group tree and select the created group.
3. Right-click the mouse and select "Create Patch” from the menu.

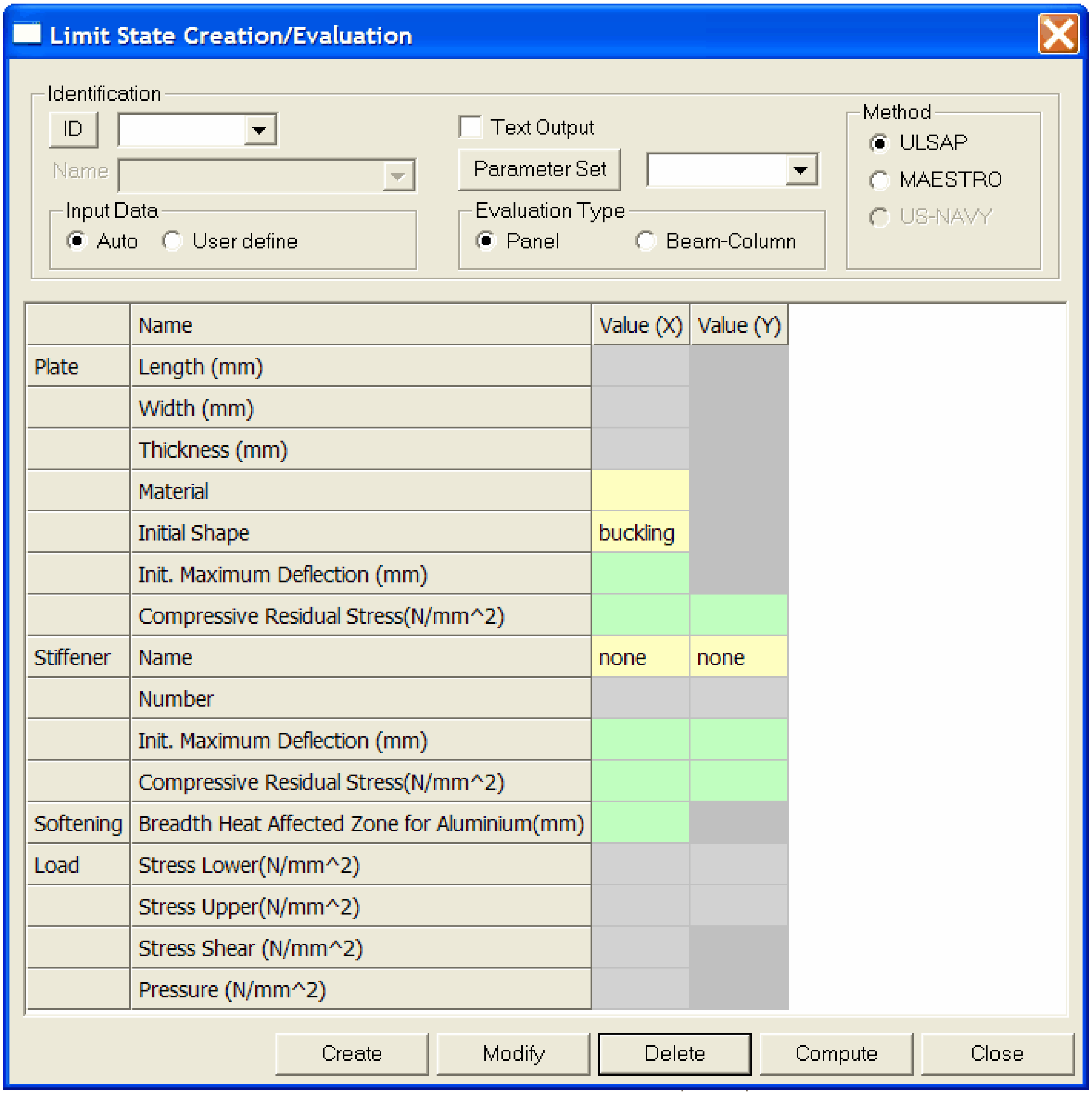
How do you query a patch?

1. From the Dynamic Query icon (), select patch.
2. Use the mouse to highlight the desired structure.

How do you evaluate a patch interactively?

1. From the main menu, Select Model>Evaluation Patch>Create/Evaluate, or click the patch icon  to open the “Limit State Creation/Evaluation” dialog box.
2. Select a patch from the ID drop down box and click compute.

© 2014 Optimum Structural Design, Inc.

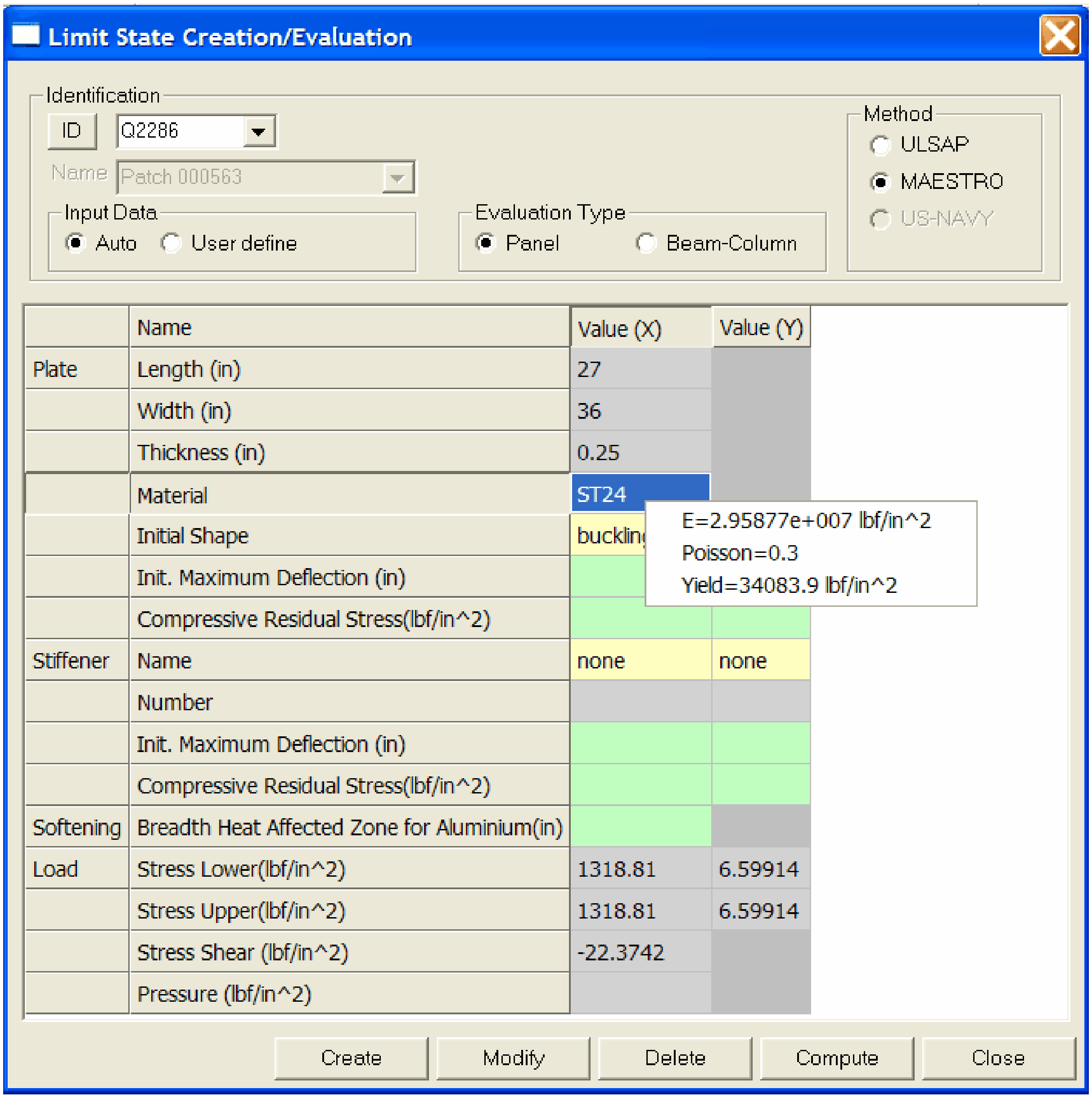


How do you view the properties of a patch?

1. Open the "Limit State Creation/Evaluation" dialog box.

2. Expand the column until the name of the plate property is seen.

3. Move the cursor to the plate property, and right-click it.



How do you modify the scantlings and stress parameters of a patch?

1. Open the “Limit State Creation/Evaluation” dialog box.

2. Change the radio button from “Auto” to “User defined”.

3. Enter the desired data

4. Click Modify.

How do you change an evaluation method?

© 2014 Optimum Structural Design, Inc.

1. Open the “Limit State Creation/Evaluation” dialog box.
2. Change the radio button in the method frame to the desired evaluation method.

How do you evaluate an individual element?

1. Open the “Limit State Creation/Evaluation” dialog box.
2. Place the cursor in the patch ID drop down box.
3. Use the mouse to select an element.
4. Modify the geometry and load parameters if necessary.
5. Click Compute button.

How do you define a patch without building a model?

1. Open the “Limit State Creation/Evaluation” dialog box.
2. Click the ID button.
3. Change the radio button from “Auto” to “User defined”.
4. Enter the desired data.
5. Click Create.

How do you view the element evaluation flag?

From the main menu, click View>Patches>Element Evaluation.

How do you turn an element evaluation flag on or off?

1. Query a Patch
2. Right-click and select "Evaluation Off”.

How do you remove a patch?

1. Query a patch.
2. Right-click and select "Remove”.

How do you define the Ultimate Strength Parameters? (Applies to ULSAP evaluation method)

1. From the main menu, select Model>ULSAP Parameters.
2. Click the ID button. Enter proper values for the ULSAP ultimate limit state parameters and

© 2014 Optimum Structural Design, Inc.

click “Create”.

Note, the default ultimate limit state parameters will be automatically applied to any patch which is not specifically associated to a set of ultimate limit state parameter.

If a patch is in a perfect initial condition, the user has to define a set of ultimate limit state parameters with zero imperfection. If all patches are in a perfect condition, then the user does not define any set of ultimate limit state parameters.

How do you get the ULSAP text output file for an individual patch?

1. Open the “Limit State Creation/Evaluation” dialog box.
2. Select the radio button “ULSAP”.
3. Click the ULSAP text output checkbox.
4. Click Compute and then follow the result processing instructions.

How do you get the ULSAP text output results for all panels?

1. Open the "Limit State Creation/Evaluation" dialog box.
2. Select the radio button "ULSAP".
3. Double-click the ULSAP text output checkbox.
4. Click Compute and then open the Output tab to review the results.

See ALPS/ULSAP for more details.

How do you associate a set of ultimate limit state parameters to a patch?

1. Define a set of ultimate limit state parameters.
2. Open the “Limit State Creation/Evaluation” dialog box.
3. Select the radio button “ULSAP”.
4. Select the parameter set.
5. Click Modify.

How do you create a MAESTRO Scalable Solver Limit State Data?

1. Manually define a patch from a general group.
2. Associate the patch to a set of limit state parameters.
3. Click Modify.

4. Export the MAESTRO Data

10.11 Failure Mode Evaluation

Introduction

A large and complex thin-wall structure can fail in many different ways, which are called the failure modes. The various factors that determine the failure modes include the geometry of the structure, the boundary conditions, and the loading on the structure.

MAESTRO provides two different types of failure mode evaluations for stiffened panels and their components: MAESTRO and ALPS/ULSAP. MAESTRO also provides ALPS/HULL, which addresses progressive collapse of the hull girder. Click on the link for either method for more detailed information on how to perform and interpret each type of evaluation. The evaluation includes yielding and buckling failure modes. For a given ship structural system and pertinent loading conditions, the calculated stresses must not be greater than the limits prescribed and/or computed for these failure modes.

Adequacy Parameters as a Measure of Safety

For a structure to be safe under a given load condition, the load effect Q must remain below the limit value Q_L by a certain factor, called the safety factor, g.

Expressed mathematically, the safety requirement is,

$$\gamma Q \le Q_L$$

If we define a “strength ratio”

$$R = \frac{\gamma Q}{Q_L}$$

then the safety requirement becomes

$$R \le 1$$

Each of these requirements constitutes a constraint on the design. In MAESTRO each constraint is expressed in the form

© 2014 Optimum Structural Design, Inc.

$$g(R) > 0$$

Safety against structural failure is measured by a parameter called the Adequacy parameter, denoted by g(R), which is defined as

$$g(R) = \frac{1 - \gamma R}{1 + \gamma R}$$

The advantage of using an adequacy parameter of the strength ratio is that g always lies within the normalized limits of -1 and +1, whereas R ranges from 0 to infinity. Specifically, $g(R) \rightarrow 1$ as $R \rightarrow 0$ either as a result of very small load or very large limit value and at other extreme, $g(R) \rightarrow -1$ as $R \rightarrow \infty$, either as a result of very large load or very small limit value. Since all of the constraints in MAESTRO are of the above form, it is sufficient, in explaining each of the failure types to simply explain what is Q and Q_L for that failure type.

For convenience of reference in post-processing, each of the adequacy parameters (for both MAESTRO and ALPS/ULSAP) is identified by an acronym, usually consisting of four letters. First, it should be states that there are two broad categories of limit states, i.e. adequacy parameters, the *ultimate* or *collapse* limit states, in which the structure or member has failed in its primary, load-carrying role; and the *serviceability* limit states, which involve the deterioration or loss of other, less vital functions. Typically the first letter is P, G, F or B referring to panel, girder, frame, or beam. The second letter indicates the type of failure; it may be either C (collapse) or S (serviceability) or it may indicate a particular type of unserviceability: Y (yield) or B (buckling).

MAESTRO Adequacy Parameters

All Load Cases

Minimum Value (Plate)
PCSF (Collapse, Stiffener Flexure)
PCCB (Collapse, Combined Buckling)
PCMY (Collapse, Membrane Yield)
PCSB (Collapse, Stiffener Buckling))
PYTF (Yield, Tension in Flange)
PYTP (Yield, Tension in Plate)
PYCF (Yield, Compression in Flange)
PYCP (Yield, Compression in Plate)
PSPBT (Serviceability, Plate Bending Tran.)
PSPBL (Serviceability, Plate Bending Long.)
PFLB (Failure, Local Buckling)

Mimimum Value (Beam)
GCT (Tripping)
GCCF (Collapse, Compression in Flange)
GCCP (Collapse, Compression in Plate)
G[F]YCF (Yield, Compression in Flange)
G[F]YCP (Yield, Compression in Plate)
G[F]YTF (Yield, Tension in Flange)
G[F]YTP (Yield, Tension in Plate)

FCPH (Collapse, Plastic Hinge)

Minimum Value (All)
- (Negative)
+ (Positive)

ALPS/ULSAP Adequacy Parameters

All Load Cases

Minimum Value (Plate)
PCPM (Plate induced failure at Midspan)
PCCB (Overall Grillage Collapse)
PCPE (Plate induced failure at Panel Edges)
PCSB (Stiffener induced failure-tripping)
PCWB (Panel Collapse Web Buckling)
PYM (Yield in Mid-Plane)
PYF (Yield in Flange)
PYP (Yield in Plate)

Mimimum Value (Beam)
BCT (Tripping)
BYC (Gross Yielding)
BCWB (Web Buckling)
BCC (Collapse, Beam-Column)

FCPH (Collapse, Plastic Hinge)

Minimum Value (All)
- (Negative)
+ (Positive)

10.11.1 Panel Collapse and Serviceability

The following sections discuss the available panel collapse and serviceability analysis methods implemented in MAESTRO.

10.11.1.1 MAESTRO

MAESTRO's limit states cover failure modes associated with Panels and Beams (Girders/Frames). The table below provides a summary of the 14 failure modes automatically evaluated using the MAESTRO native criteria.

© 2014 Optimum Structural Design, Inc.

Panel Failure Modes

PCSF	Panel Collapse, Stiffener Flexure
PCCB	Panel Collapse, Combined Buckling
PCMY	Panel Collapse, Membrane Yield
PCSB	Panel Collapse, Stiffener Buckling
PYF	Panel Yield, Flange
PYP	Panel Yield, Plate
PSPB	Panel Serviceability, Plate Bending
PFLB	Panel Failure, Local Buckling

Beam Failure Modes

BCT	Beam Collapse, Tripping
BCCF	Beam Collapse, Compression in Flange
BCCP	Beam Collapse, Compression in Plate
BYF	Beam Yield, Flange
BYP	Beam Yield, Plate
BCPH	Beam Collapse, Plastic Hinge

10.11.1.1.1 Panel Failure Modes

MAESTRO's limit states for panels cover 8 different modes of failure. The table below provides a summary of these failure modes.

Panel Failure Modes

PCSF	Panel Collapse, Stiffener Flexure
PCCB	Panel Collapse, Combined Buckling
PCMY	Panel Collapse, Membrane Yield
PCSB	Panel Collapse, Stiffener Buckling
PYF	Panel Yield, Flange
PYP	Panel Yield, Plate
PSPB	Panel Serviceability, Plate Bending
PFLB	Panel Failure, Local Buckling

Panel Failure Modes: Collapse

Collapse limit states are defined when the structure or member has failed in its primary, load-carrying role.

Panel Collapse, Stiffener Flexure (PCSF)

The PCSF failure mode examines either the (a) Ultimate Strength of Plates, Chapter 12.6 of Reference [1], or (b) Ultimate Strength of Stiffened Panels, Chapter 14 of Reference [1]. The Ultimate Strength of Plates is examined when an element is a bare plate (i.e., unstiffened), while the Ultimate Strength of the Stiffened Panels is examined when the element has stiffeners associated with it (i.e., a stiffener layout is defined).

For stiffened plates, this collapse occurs due to the axial compression and flexure of the plate-stiffener combination or by the axial compression . Each stiffener is regarded as an isolated beam column, with the plating acting as one of the two flanges. For this scenario, there are three different types of failure modes depending on the sign of the bending moment, and the deflection of the plating/flange: Mode I, Mode II and Mode III. The underlying theory for this failure mode is given in Chapter 14 of Reference [1].

The following sign convention (for theoretical computation) is used:

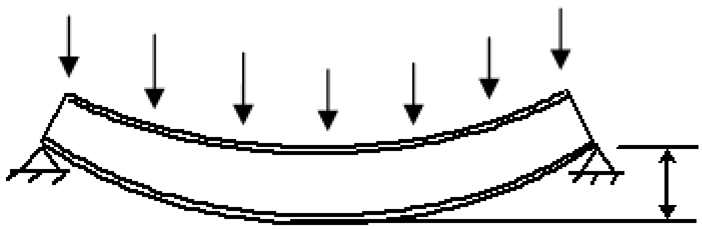
- Stress : Positive if compressive.
- Bending moment : Positive when the plating is in compression
- Lateral Deflection : Positive towards the stiffeners.
- Eccentricity : Positive towards the stiffeners.

© 2014 Optimum Structural Design, Inc.

The three modes of failure are :

Mode I : "Stiffener-induced" collapse due to compression failure of the stiffener.

Collapse occurs due to compression failure of the stiffener flange. The combination of in-plane compression and negative bending is illustrated in the figure below.



$$\sigma_f = \sigma_a + \frac{M_0 y_f}{I} + \frac{\sigma_a A(\delta_0 + \Delta) y_f}{I} \Phi$$

Collapse occurs when the stress in the mid-thickness of the flange, σ_f equals the failure value, which is the minimum of yield stress σ_{Ys} or the elastic tripping stress $\sigma_{a,T}$.

The terms in the above equation are:

A and I : Cross sectional area and moment of inertia of the beam column.

M_0 and δ_0 : Bending moment and deflection due to lateral load ("dead load").

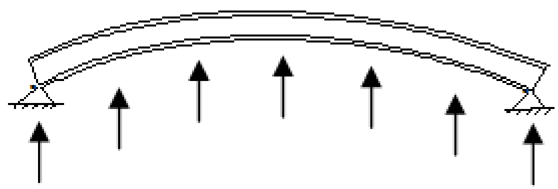
Δ : Eccentricity

y_f : distance from the centroidal axis of the c.s. to the mid-thickness of the stiffener flange.

Mode II : "Plate induced" collapse due to compression failure of the plating.

This failure mode occurs due to a combination of in-plane compression and a positive bending moment as shown in the figure below. The total stress acting through the plating as obtained from the beam-column formula would be

$$\sigma_F = \sigma_{a,tr} + \frac{M_0 y_{p,tr}}{I_{tr}} + \frac{\sigma_{a,tr} A_{tr} (\delta_0 + \Delta) y_{p,tr}}{I_{tr}} \Phi$$



where σ_F is the failure stress and the subscript 'tr' stands for the property of the transformed section. Collapse occurs when σ_f equals the failure value, which is the minimum of yield stress σ_{Ys} or the elastic tripping stress $\sigma_{a,tr}$

Mode III : Combined failure of stiffener and plating.

This failure occurs due to a large positive bending moment M_0, which would cause a large tensile stress in the stiffener. The failure is due to a simultaneous stiffener tensile yielding and compressive yielding of the plate. It can be shown from the interaction curve of Figure 14.2 of [1] that the load combination required for mode III failure is given by the expression

$$\sigma_{Ys} = (\sigma_{a,tr})_{ult} + \frac{M_0 y_{f,tr}}{I_{tr}} + \frac{(\sigma_{a,tr})_{ult} A_{tr} (\delta_0 + \Delta) y_{f,tr}}{I_{tr}} \Phi$$

$$+ \frac{(\sigma_{a,tr})_{ult} A_{tr} \Delta_p y_{f,tr}}{I_{tr}}$$

While evaluating PCSF, the effects of transverse compression and in plane shear are also taken into account. The respective reduction factors r_{ay} and r_τ are given by

$$r_{ay} = 1 - \frac{\sigma_{ay}}{\sigma_{ay,u}}$$

© 2014 Optimum Structural Design, Inc.

$$r_\tau = \sqrt{1 - 3\left(\frac{\tau}{\sigma_{Yp}}\right)^2}$$

Panel Collapse, Combined Buckling (PCCB)

In general, a stiffened panel is subjected to longitudinal, transverse, and shear loads, and the different buckling modes corresponding to these loads will interact to a considerable degree. This failure algorithm examines the interaction between these loads and the elastic overall panel buckling due to any combination of longitudinal stress σ_x, transverse stress σ_y and shear stress τ. The strength ratio, R_{PCCB}, for this combined buckling mode is calculated by using an interaction formula to combine the strength ratios for longitudinal buckling, transverse buckling and shear buckling for the panel.

The algorithm first calculates the uniaxial elastic critical stresses $\sigma_{x,cr}$, $\sigma_{y,cr}$ and τ_{cr}, based on the theory in Chap 13 of [1]. It then formulates the uniaxial strength ratios Rx = $\sigma_x/\sigma_{x,cr}$,

$R_y = \sigma_y/\sigma_{y,cr}$ and $R_s = \tau_y/\tau_{cr}$ and inserts these into the following formula for the total (interactive) strength ratio:

$$R_x + \frac{0.625\left(1 + \frac{0.6}{\alpha}\right)R_y}{1 + R_x} + R_s^2 = 1 \quad (\alpha \geq 1)$$

The elastic buckling modes for a stiffened panel are presented in chapter 13 of Ref. [1].

Panel Collapse, Membrane Yield (PCMY)

This failure mode checks for any yielding that occurs through the thickness of the plating (membrane yield). When yielding occurs, it can easily extend into a significant portion of the structure and could lead to overall collapse. MAESTRO evaluates PCMY using the Von Mises equivalent stress, which is obtained by using the mid-plane mean longitudinal stress acting on the panel, the mean transverse stress and the shear stress, as

$$\sigma_{vm} = \sqrt{\sigma_x^2 + \sigma_y^2 - \sigma_x\sigma_y + 3\tau_{xy}^2}$$

$$R_{PCMY} = \frac{\gamma_c \sigma_{vm}}{\sigma_Y}$$

where

$$\sigma_{vm} = \textit{Von Mises stress}$$

$$\gamma_c = \textit{combined safety factor for collapse}$$

Panel Collapse, Stiffener Buckling (PCSB)

This failure mode checks the flexural-torsional and lateral-torsional buckling ("tripping") behavior of flanged stiffener subjected to axial compression, end moment, lateral pressure and any combination of these. The effects of cross-sectional distortion, postbuckling behavior of the plate (incorporated by considering the plate effective width) and plasticity are included. A detailed description of this mode of failure can be found in Reference [17] and [18].

For the evaluation of PCSB (Panel Collapse Stiffener Buckling), the overall evaluation patch parameters of length, width, etc., are used. However, the stress and pressure loads used for the evaluation are evaluated at each individual element.

For transversely stiffened panels, the stiffeners are at right angles to the principal compressive stress and so they can do little to prevent or delay plate buckling. However, if they buckle before the plating then overall panel buckling follows immediately. This is a very undesirable mode of collapse since it occurs rapidly and without warning. Therefore the PCSB limit state becomes the requirement that stiffener buckling must not precede plate buckling.

Panel Failure Modes: Unservicability

Serviceability limit states are defined when the deterioration or loss of other, less vital functions have occurred.

Panel Yield Flange (PYF)

For plates that have stiffeners defined, this limit state examines the combined axial stress and stiffener bending stress in the stiffener flange. The bending stress in the stiffener flange is simply the (M*y)/I and is unidirectional.

For unstiffened plates, the *flange* is taken as the bottom fiber (i.e., extreme fiber) of the plate (see Bare plate).

© 2014 Optimum Structural Design, Inc.

Panel Yield Plate (PYP)

For plates that have stiffeners defined, this limit state examines the von Mises stress due to the combination of axial stress, shear stress, and bending stress calculated at the root of the stiffener flange and compared tot he stiffener yield stress.

For unstiffened plates, the *plate* is taken as the top fiber (i.e., extreme fiber) of the plate (see Bare plate).

Panel Serviceability, Plate Bending (PSPB)

The limit state PSPB (Panel Serviceability, Plate Bending) deal with plate bending due to pressure and do not consider or allow for any permanent set; they require that the total stress at the plate surface (membrane stress + bending stress) remains below yield. Since pressure can vary within an evaluation patch these limit states are evaluated for every element within the evaluation patch. The bending stress due to lateral pressure is given by the expression

$$\sigma_b = kp\left(\frac{b}{t}\right)^2$$

where k is a constant that depends on the boundary conditions and the aspect ratio of the plate. Values of k are given in Figure 9.6 of Ref. [1]. This bending stress is automatically calculated by MAESTRO based on the evaluation patch parameters and incorporated into the Von Mises stress, which is then compared to yield.

Panel Failure, Local Buckling (PFLB)

PFLB refers to the failure related to the buckling, elastic or inelastic, of plating between stiffeners. Buckling may be caused by σ_x, σ_y, or τ, or by various combinations of these.

“Local” buckling implies buckling of plating between stiffeners. When failure occurs due to buckling of plating crosswise to the stiffeners (e.g. buckling due to lengthwise compression in a transversely stiffened panel) then such buckling is collapse of the panel rather than unserviceability. Therefore this limit state uses the word “failure”, represented by the letter F in the acronym PFLB, rather than the letter C (for collapse) or S (for serviceability).

σ_L denotes a stress that acts lengthwise (parallel to the stiffeners) and σ_T will denote a stress that acts transversely (across the stiffeners). The corresponding failure stresses (elastic or inelastic buckling stress or, for thick plating, yield stress) when the applied stresses act in isolation are denoted $\sigma_{L,F}$ and $\sigma_{T,F}$. Similarly the value of shear stress that would cause

© 2014 Optimum Structural Design, Inc.

failure (again elastic or inelastic buckling, or yield) when it alone acts on the plating is denoted τ_F. The values of $\sigma_{L,F}$, $\sigma_{T,F}$ and τ_F are calculated from the equations of chapter 12 of [1]. The corresponding strength ratios are $R_L = \sigma_L/\sigma_{L,F}$, $R_T = \sigma_T/\sigma_{T,F}$, and $R_S = \tau/\tau_F$. These "act alone" strength ratios are then converted to interactive values $R_{L,F}$ and $R_{T,F}$ by an extension of the procedure given on page 435 of Reference [1]. To give a brief picture of this, for a slender plate, the shape of the curve is almost linear, whereas for a thick plate it approaches the Von Mises ellipsoid.

The partial safety factor varies between γ_S and γ_C, depending on the stiffening and on the relative values of $R_{L,F}$ and $R_{T,F}$. For unstiffened panels γc is used. For a stiffened panel: if the transverse stress is zero or tensile use γs; otherwise use a value between γs and γc according to the value of $R_{T,F}$ relative to $R_{L,F}$

10.11.1.1.2 Beam Failure Modes

MAESTRO's limit states for girders cover 6 different modes of failure. The table below provides a summary of these failure modes.

Girder Failure Modes	
BCT	Beam Collapse, Tripping
BCCF	Beam Collapse, Compression in Flange
BCCP	Beam Collapse, Compression in Plate
BYF	Beam Yield, Flange
BYP	Beam Yield, Plate
BCPH	Beam Collapse, Plastic Hinge

Beam Failure Modes : Collapse

Collapse limit states are defined when the structure or member has failed in its primary, load-carrying role.

Beam Collapse, Tripping (BCT)

This failure mode checks the flexural-torsional and lateral-torsional buckling ("tripping") behavior of flanged stiffener subjected to axial compression, end moment, lateral pressure and any combination of these. The effects of cross-sectional distortion, postbuckling behavior of the plate (incorporated by considering the plate effective width) and plasticity are included. A detailed description of this mode of failure can be found in Reference [17] and [18]. In particular, Reference [17] deals with girders.

© 2014 Optimum Structural Design, Inc.

Beam Collapse, Compression, Flange/Plate (BCCF, BCCP)

Beams are loaded as a “beam column” and collapse may occur due to combined bending and buckling. Collapse is assumed to occur when there is compressive yielding in either the beam flange or plate flange, as there is very little reserve strength beyond this point for beam columns. The stresses in the beam flange is the beam axial stress plus the bending stress at the flange (M*y)/I. Detailed theory can be obtained from section 11.3 of Ref. [1].

Beam Failure Modes : Serviceability

Serviceability limit states are defined when the deterioration or loss of other, less vital functions have occurred.

Beam Yield, Flange/Plate (BYF, BYP)

Compressive failure of a beam is, in general, a complex buckling phenomenon, and to serve as an approximate preliminary check for failure, the Von Mises stress at the beam flange and the plate flange are calculated and are checked against the yield stress. This occurs for both tension and compression stress.

Beam Failure Modes : Collapse

Collapse limit states are defined when the structure or member has failed in its primary, load-carrying role.

Beam Collapse, Plastic Hinge (BCPH)

Collapse of a beam will occur if sufficient plastic hinges are formed to allow it to undergo large deformations as a mechanism. If the bending moment at one of the two ends in the beam segment exceeds the plastic moment for that beam's cross section, collapse is assumed to have occurred for that beam. Chapter 16 of Reference [1] provides detailed theory of plastic hinge.

10.11.1.2 ALPS/ULSAP

The following capability is an optional MAESTRO module. Run Fast Lock if you are not certain have purchased this optional module.

A complete description of the theoretical basis for the ALPS/ULSAP module can be found in:

1. MARSTRUCT 2011 Paper (ULSAP)

2. Benchmark Study (ULSAP)

© 2014 Optimum Structural Design, Inc.

Supporting verification tables presented in the above documents can be found in the *...\MAESTRO\Models and Samples\Verification Models\ULSAP* directory.

For a ALPS/ULSAP tutorial, go here: ALPS/ULSAP

ALPS/ULSAP's limit states cover failure modes associated with Panels, Girders and Frames. The table below provides a summary of the 13 failure modes automatically evaluated using the ALPS/ULSAP criteria.

Panel Failure Modes	
PCCB	Panel Collapse, Overall Grillage Collapse - Mode I
PCPE	Panel Collapse, Plate Induced Failure, Yield at Panel Edges - Mode II
PCPM	Panel Collapse, Stiffener Induced Failure, Yield at Midspan - Mode III
PCWB	Panel Collapse, Stiffener Induced Failure, Web Buckling - Mode IV
PCSB	Panel Collapse, Stiffener Induced Failure, Tripping - Mode V
PYM	Panel Yield, Mid-plane
PYF	Panel Yield, Stiffener Flange
PYP	Panel Yield, Plate

Beam Failure Modes	
BCT	Beam Collapse, Tripping
BYC	Beam Gross Yielding
BCWB	Beam Collapse, Web Buckling
BCC	Beam Collapse, Beam-column
FCPH	Frame Collapse, Plastic Hinge

10.11.1.3 ABS HSNC

The ABS High Speed Naval Craft rules have been implemented into MAESTRO's limit state framework. When exercising these limit states, level 0 evaluation patches should be used and the HSNC Allowable Stress value in the material properties should be entered according to the rules. For more details on the different limit states and equations, please reference the following ABS Guides:

© 2014 Optimum Structural Design, Inc.

ABS Rules for Building High Speed Naval Craft

ABS Guide for Buckling and Ultimate Strength For Offshore Structures

10.11.1.4 Running Standalone Evaluation

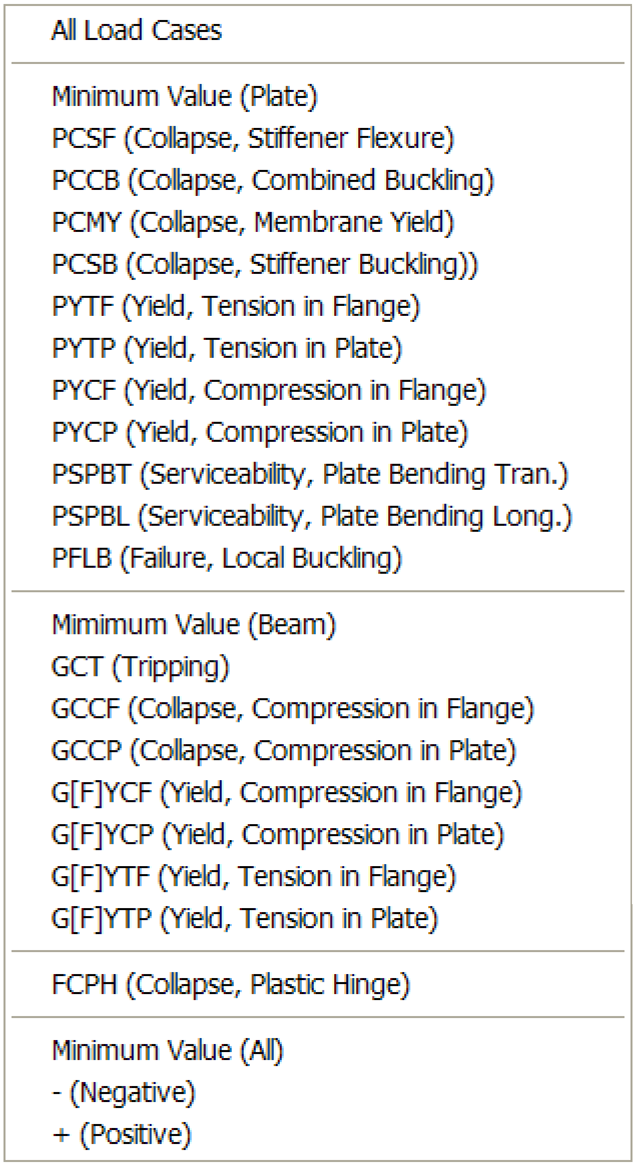
Standalone evaluation can be run be accessing the Limit State Creation/Evaluation dialog and following the procedure outlined in Using Evaluation Patches.

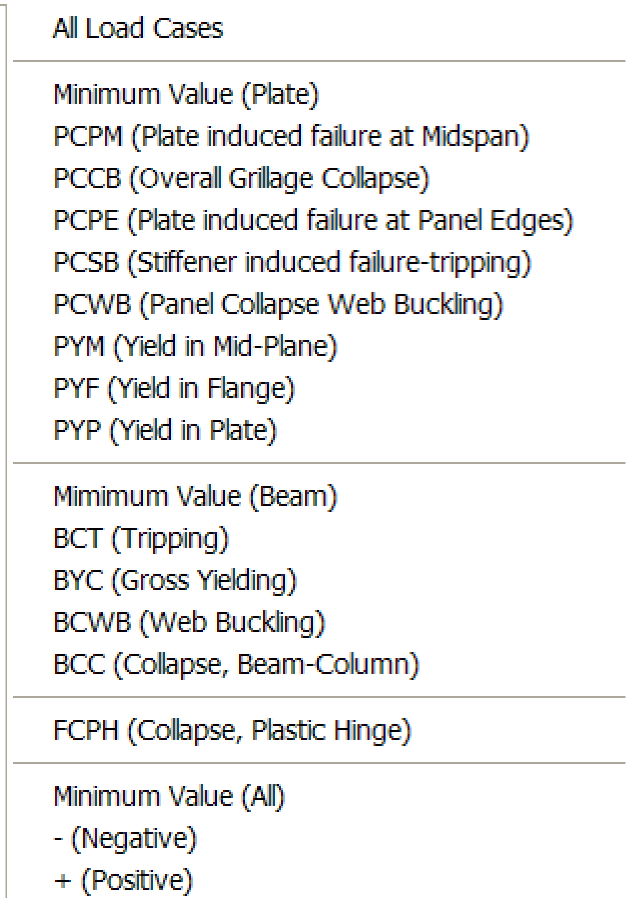
10.11.1.5 Post-processing Failure Modes

Toolbar	**N/A**
Menu	**Results >**

Post-processing of the various failure modes can be accomplished by accessing the **Results > Adequacy** menu. The adequacy parameters menu will be different depending on whether MAESTRO or ULSAP evaluation is chosen.

MAESTRO *ULSAP*





Selecting one of the parameters will display the patches colored by their adequacy parameter value. MAESTRO can automatically display each elements smallest adequacy parameter by selecting Minimum Value (Plate), Minimum Value (Beam), or Minimum Value (All). You can also choose to display only the Positive or only the negative adequacy parameter results.

Similar to Viewing Stress Ranges, Adequacy values can be defined by the user and viewed, e.g., to determine all negative adequacy values.

© 2014 Optimum Structural Design, Inc.

The dynamic query icon  (with *Adequacy* checked from the drop-down) can be used to highlight an element and recover the adequacy parameters. Double-clicking the element will echo the results to the Output tab.

10.11.2 Hull Girder Progressive Collapse

The following sections discuss the available hull girder progressive collapse methods implemented in MAESTRO.

10.11.2.1 ALPS/HULL

The following capability is an optional MAESTRO module. Run Fast Lock if you are not certain have purchased this optional module.

A complete description of the theoretical basis for the ALPS/HULL module can be found in:

1. MARSTRUCT 2011 Paper (HULL)
2. Benchmark Study (HULL)

For a ALPS/Hull tutorial, please go here: ALPS/HULL

Ship hulls are subjected to a variety of hull girder or local load components. Of these, vertical bending is a primary hull girder load component. It is known that the horizontal bending may sometimes be large in the magnitude, approaching the magnitude of vertical bending moment when the ship runs at an oblique heading in waves. Also, in some vessels such as bulk carriers carrying dense cargo such as iron ore, an uneven alternate hold loading condition is normally applied, and, as a result, large shearing forces will be imposed. Moreover, torsion is normally considered to be important for vessels with low torsional rigidity due to large deck opening such as for instance in container vessels and some large bulk carriers.

© 2014 Optimum Structural Design, Inc.

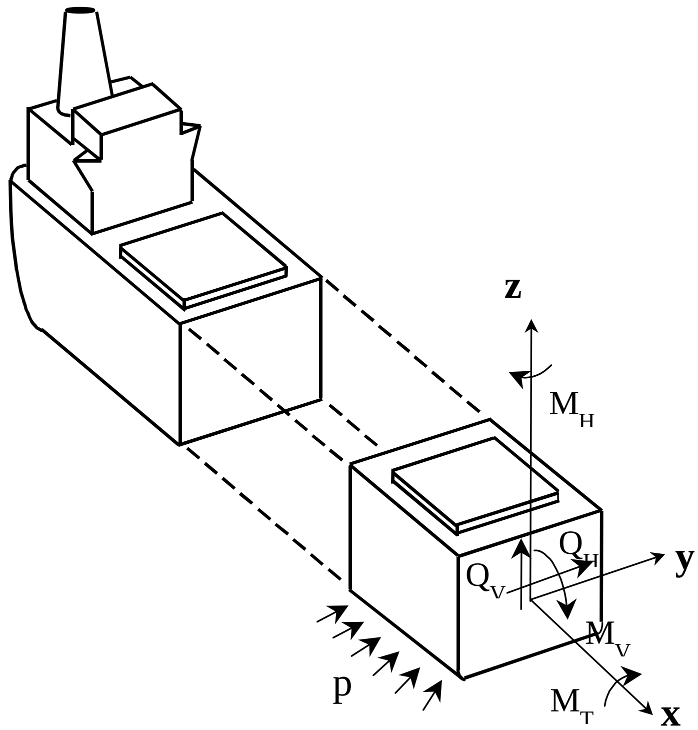


Figure 1 Hull girder sectional load components

Local plate elements of ship structures are subjected to lateral pressure loads due to cargo or water. To view a complete picture of hull girder ultimate strength behavior, therefore, all hull girder sectional load components mentioned above, i.e., vertical bending, horizontal bending, vertical sectional shear, horizontal sectional shear and torsion in addition to local lateral pressure loads, see Fig.1, should be considered simultaneously in a consistent procedure for hull girder ultimate strength analysis.

By application of the finite element method, quite accurate ultimate strength solutions have been obtained in several specific cases by a number of investigators. However, a weak feature of the conventional finite element method is that it requires enormous modeling effort and computing time for large sized structures. Therefore, most efforts in the development of new calculation methods have focused on reducing modeling and computing times.

The obvious way to reduce modeling effort and computing time is to reduce the number of degrees of freedom. Modeling the object structure with very large sized structural units is perhaps the best way to do that. Properly formulated structural units in such an approach can then be used to efficiently model the actual nonlinear behavior of large structural units.

© 2014 Optimum Structural Design, Inc.

Ueda and Rashed (1974, 1984), who suggested this idea, called it the idealized structural unit method (ISUM).

For applying ISUM, various structural units making up the object structure should be developed in advance. Until now, several ISUM units have been developed, and based on these units a family of the computer program ALPS has been written by Paik (1995b). ALPS is an acronym for the nonlinear Analysis of Large Plated Structures.

The ALPS/HULL module calculates efficiently the progressive collapse behavior of ship hulls. The benefit of the ALPS/HULL module is that it can accommodate the effects of all possible hull girder sectional load components, i.e., vertical bending, horizontal bending, vertical shearing force, horizontal shearing force and torsion in addition to local pressure loads, in the ultimate strength calculations.

The ALPS/Hull failure modes are summarized in the table below:

Failure Mode Acronym	*Failure Mode Description*
OC	Overall Collapse
PB	Collapse of plating between support members (stiffeners)
BCC	Beam-column Type Collapse
SWB	Local Buckling of Stiffener Web
TR	Flexural-torsional Buckling of Stiffener
GY	Gross Yielding
RT	Rupture due to Tension
CC	Crushing due to Compression

.

10.11.2.2 ProColl

Enter topic text here.

10.12 Creating and Analyzing a Fine Mesh Model

The following tutorial shows the process for creating and analyzing a fine mesh model.

There are four different types of fine mesh analysis models that can be automatically created in MAESTRO: a top down model, an embedded model, a ALPS/HULL model, and a Nastran Map model.

When automatically creating a top down or embedded fine mesh model within MAESTRO, the following loads are automatically prescribed to the fine mesh model from the coarse mesh model:

- Immersion
- LinPress
- Surface Head
- Surface Zero
- Point load
- Convert mass nodal group into point mass
- Convert mass bay group into point mass
- Convert mass plate group into point mass

Top Down Analysis

The top-down analysis is a two steps analysis. The coarse mesh model has to be solved first. The boundary displacements of the fine mesh model are calculated using coarse mesh results via linear interpolation. The displacements of the coarse mesh nodes are then applied to the fine mesh nodes through RSpline elements. The fine mesh model is a different finite element model with imposed displacement boundary conditions. This method uses the same logic as MG/DSA.

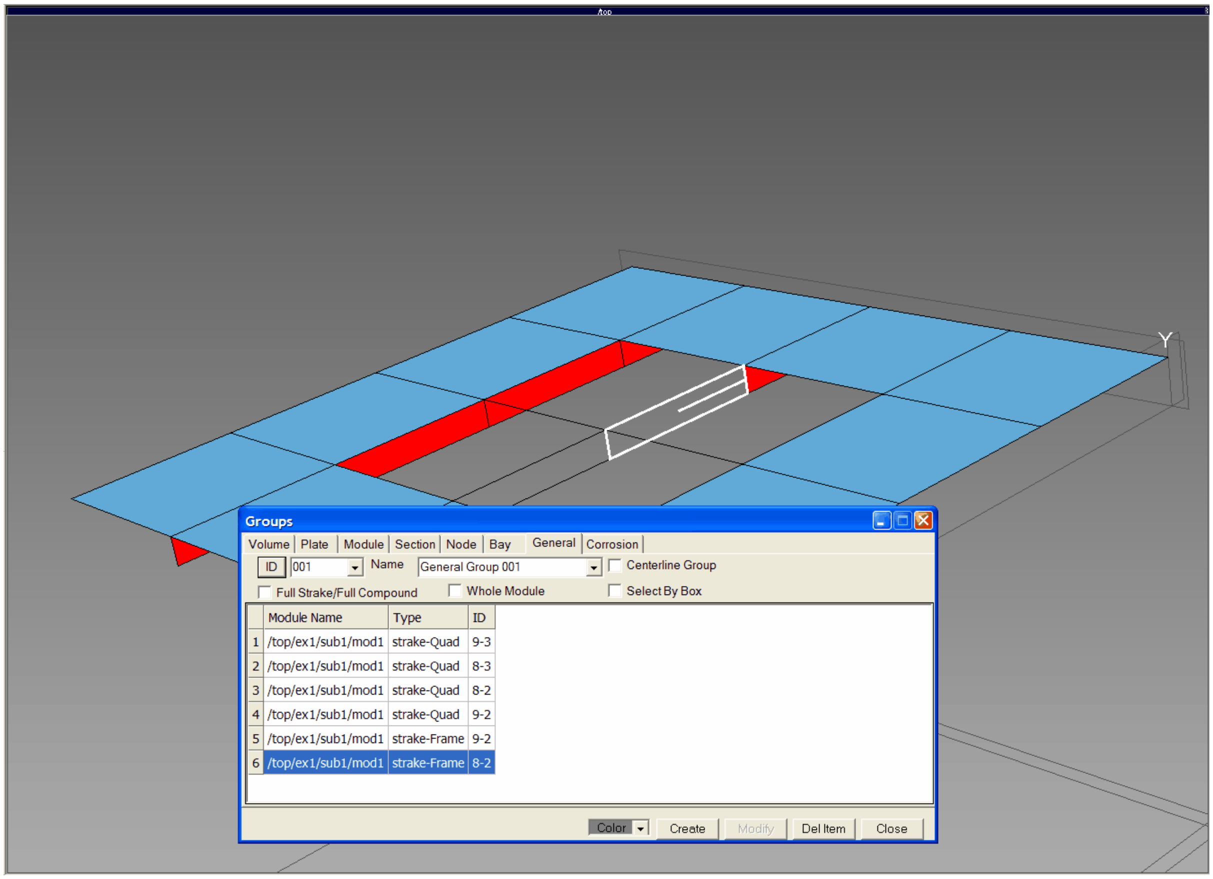
Groups can be created within the fine mesh model and loaded in the load case dialog as with the coarse mesh model. MAESTRO will automatically flag these groups as fine mesh and ignore them while solving the coarse mesh model.

The following example will walk through creating a simple top down fine mesh model:

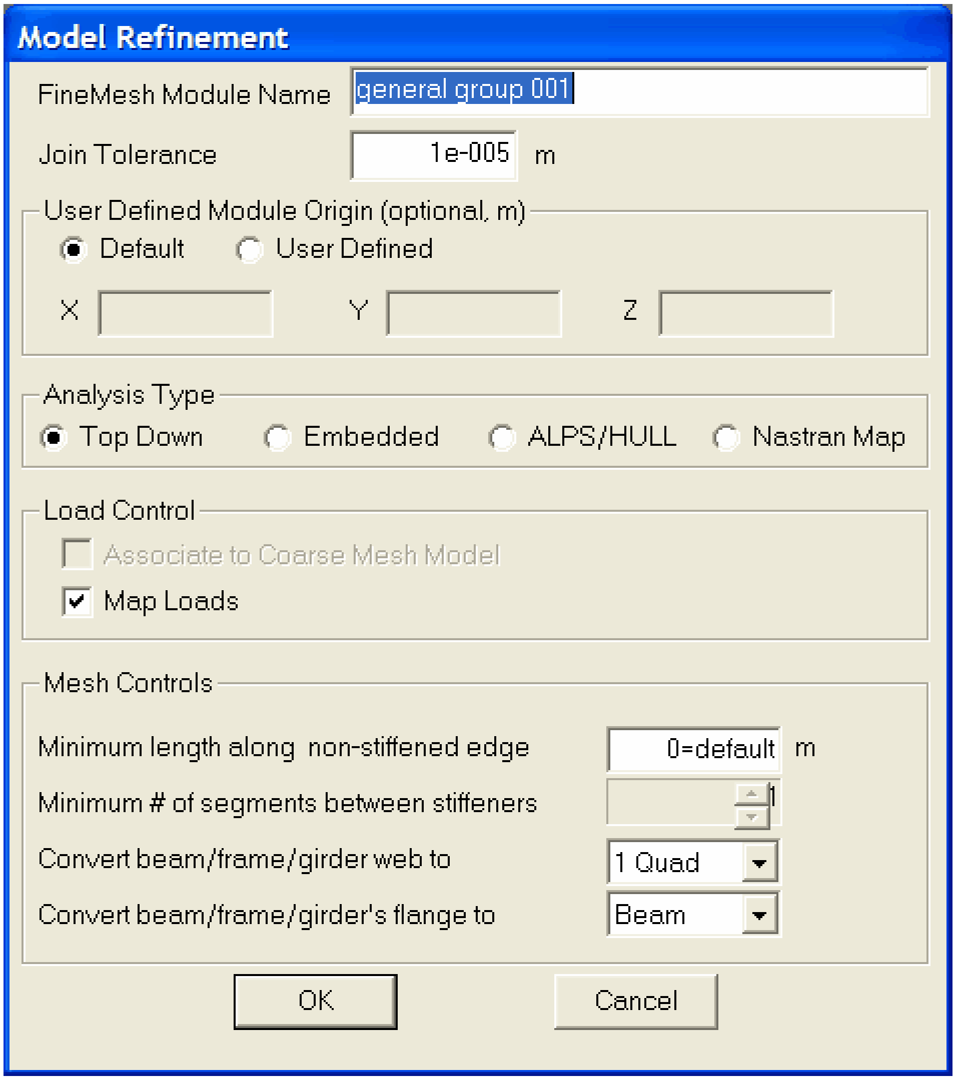
© 2014 Optimum Structural Design, Inc.

1. Begin by opening FineMeshStep0.mdl from the *Models and Samples* MAESTRO Installation directory.

2. Create a general group of the middle 4 strake panels, including the strake frames.



3. Right-click on the group in the parts tree and select Refine.

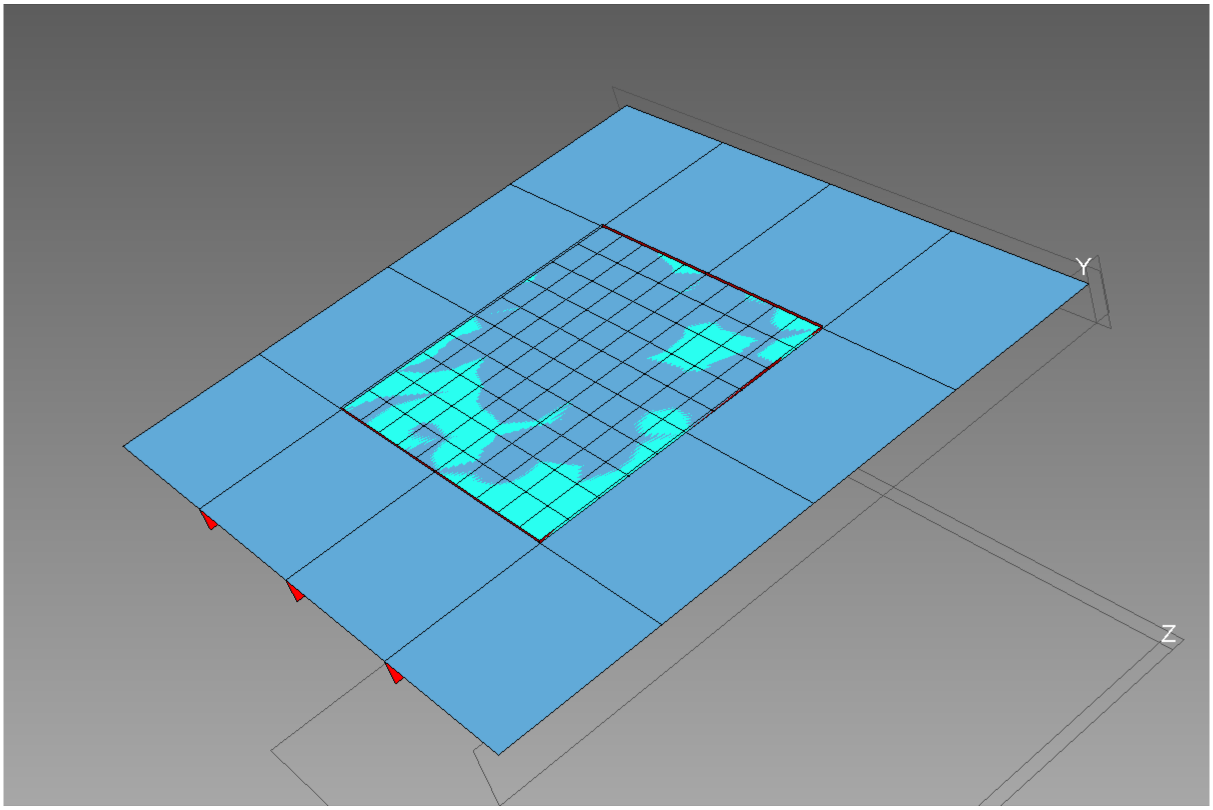


4. Use the default settings and click OK.

You should now be to the same point as FineMeshStep2.mdl.

In a top down analysis, the original coarse mesh elements are not deleted from the model. To view the coarse and fine mesh elements together, select **View > All Modules** from the menu.

© 2014 Optimum Structural Design, Inc.



In order to solve the fine mesh model, you must first solve the coarse mesh model and then solve the fine mesh model separately. This can be done by solving the coarse mesh model as usual and then right-clicking on the fine mesh model in the parts tree and selecting solve. A box can also be checked to "Solve All Fine Mesh Models" in the Analysis/Evaluation dialog box. This option will first solve the coarse mesh model and then the fine mesh model with one command. RSplines are used to transmit the displacements of the coarse mesh nodes to the fine mesh nodes along the RSpline element.

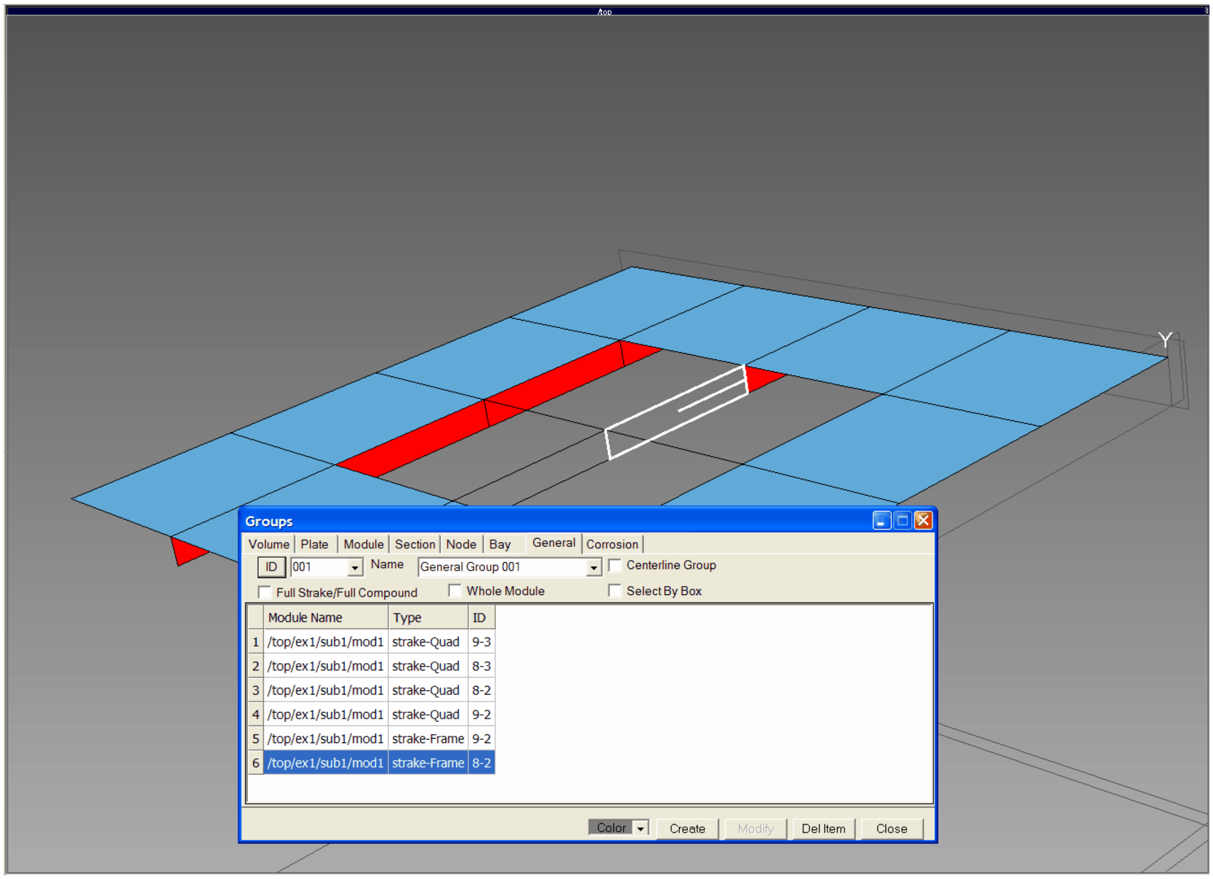
Embedded Analysis

The embedded fine mesh analysis is a one step analysis. The coarse mesh model and the fine mesh model are tightly connected by the Rspline elements. When the embedded model is created by the refine method, the "Master element" has to be deleted. Otherwise the structure is over stiffened by the fine mesh model. An embedded fine mesh model has no "Master element". The RSpline element provides a "restraint" stiffness matrix, and the stiffness matrix is added into the global stiffness matrix, so the coarse mesh model and the fine mesh model can be solved together at the cost of larger band width and much more computer time. For "top-down" analysis, the global result never changes no matter how the fine mesh model is modified; but for "Embedded" fine mesh analysis, the global results are dependent on the fine mesh model and they are closely coupled.

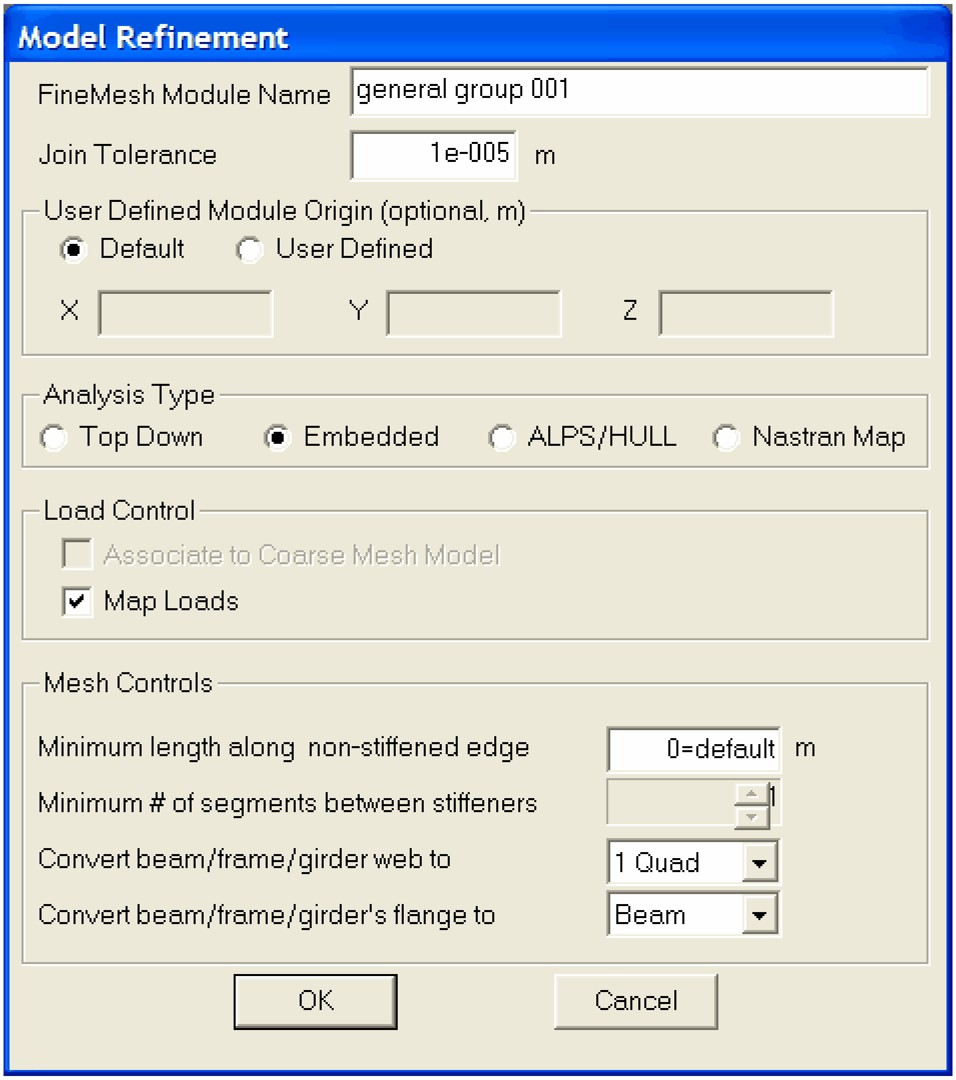
The following example will walk through creating a simple embedded fine mesh model:

1. Begin by opening FineMeshStep0.mdl from the *Models and Samples* MAESTRO Installation directory.

2. Create a general group of the middle 4 strake panels, including the strake frames.



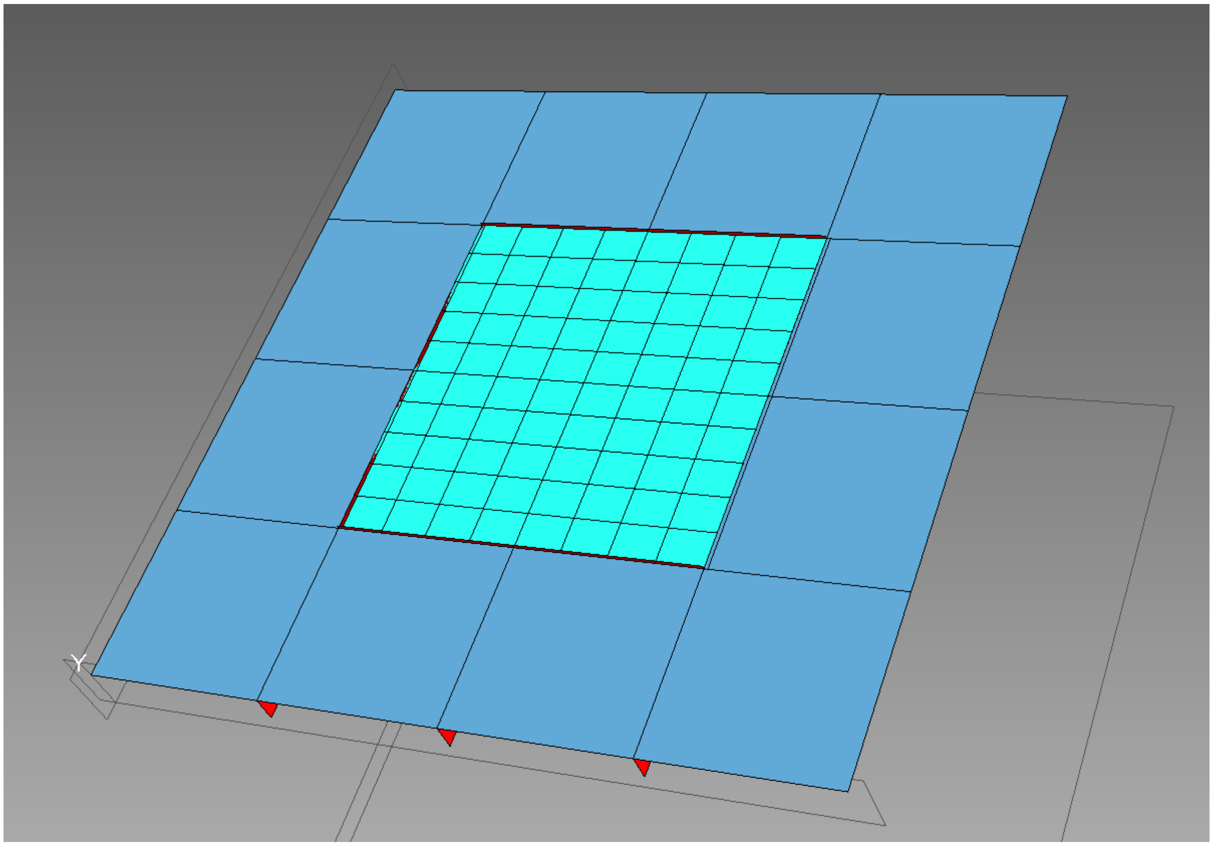
3. Right-click on the group in the parts tree and select Refine.



4. Select Embedded for Analysis Type and use the default values for the rest of the input and click OK.

You should now be to the same point as FineMeshStep2Embed.mdl.

In an embedded analysis, the coarse mesh elements from the general group are now replaced by the fine mesh elements.



The fine mesh and coarse mesh models will now solve simultaneously in an embedded analysis. Again, RSplines are used to transmit the displacements of the coarse mesh nodes to the fine mesh nodes along the RSpline element.

ALPS/HULL Analysis

Due to the complexity of this subject, please see the ALPS/HULL section.

Nastran Map Analysis

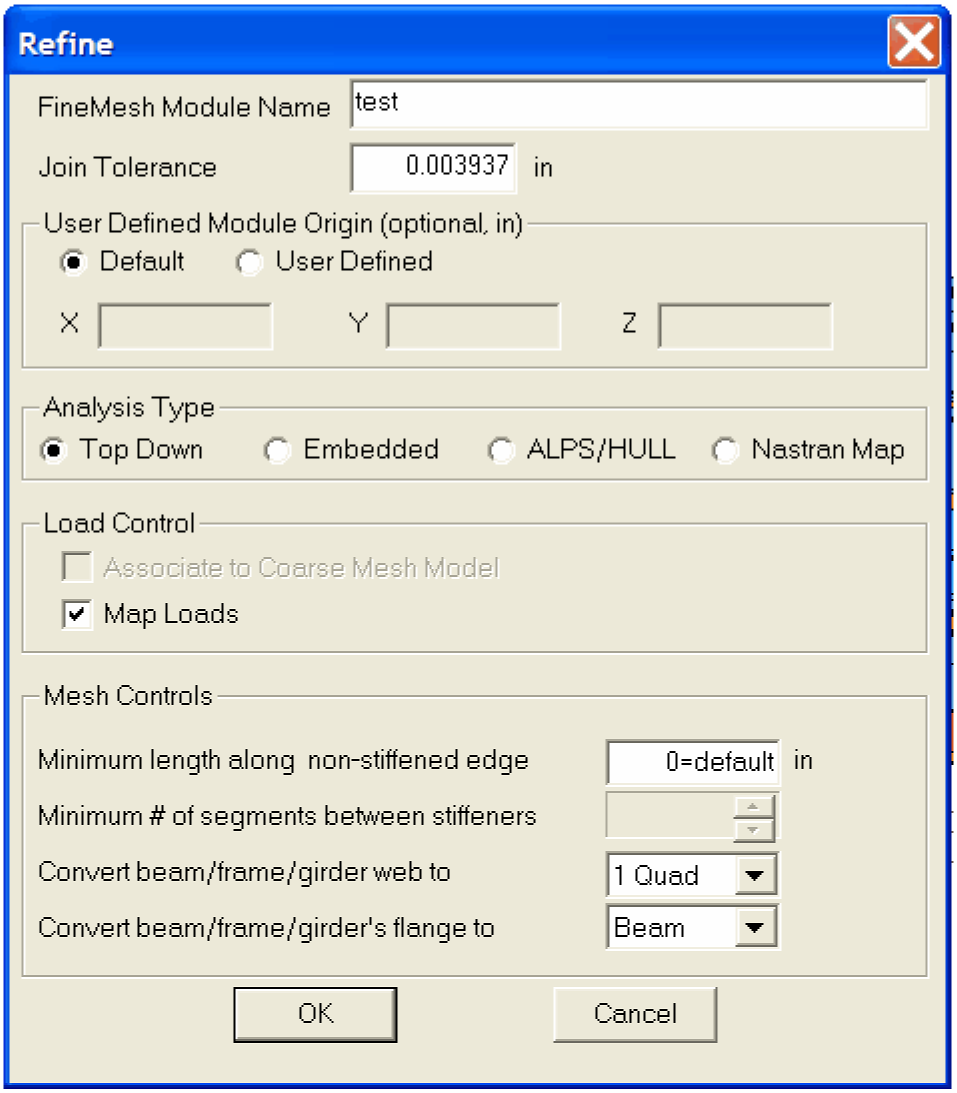
The Nastran Map option will create a separate fine mesh analysis model of the general group with a 1 to 1 ratio. All strake panels, frames and girders will be converted to quad and beam elements. This model can then be exported to Nastran to analyze with the global response imposed through the use of RSpline elements. Internal stiffener layout properties, if existing, are automatically lumped to the edges of their associated element.

© 2014 Optimum Structural Design, Inc.

Either method follows the same general procedure.

1. Create a general group of the elements that will be refined into the fine mesh model.

2. Right-click on the group name in the parts tree and select refine. This will open the Refine dialog box.



The fine mesh model will be created as a module with the given name either under top if you are using the embedded method or separately for the top down method. The join tolerance and user defined module origin can be overwritten if desired.

3. Select the Analysis Type.

4. Select whether the element loads from the coarse mesh model should be mapped onto the fine mesh model. This would include tank, pressure, mass, etc. loads.

5. The minimum length along non-stiffened edge can be set to control the mesh size.

© 2014 Optimum Structural Design, Inc.

6. Select how the model's web and flange elements should be treated.
7. Click OK.

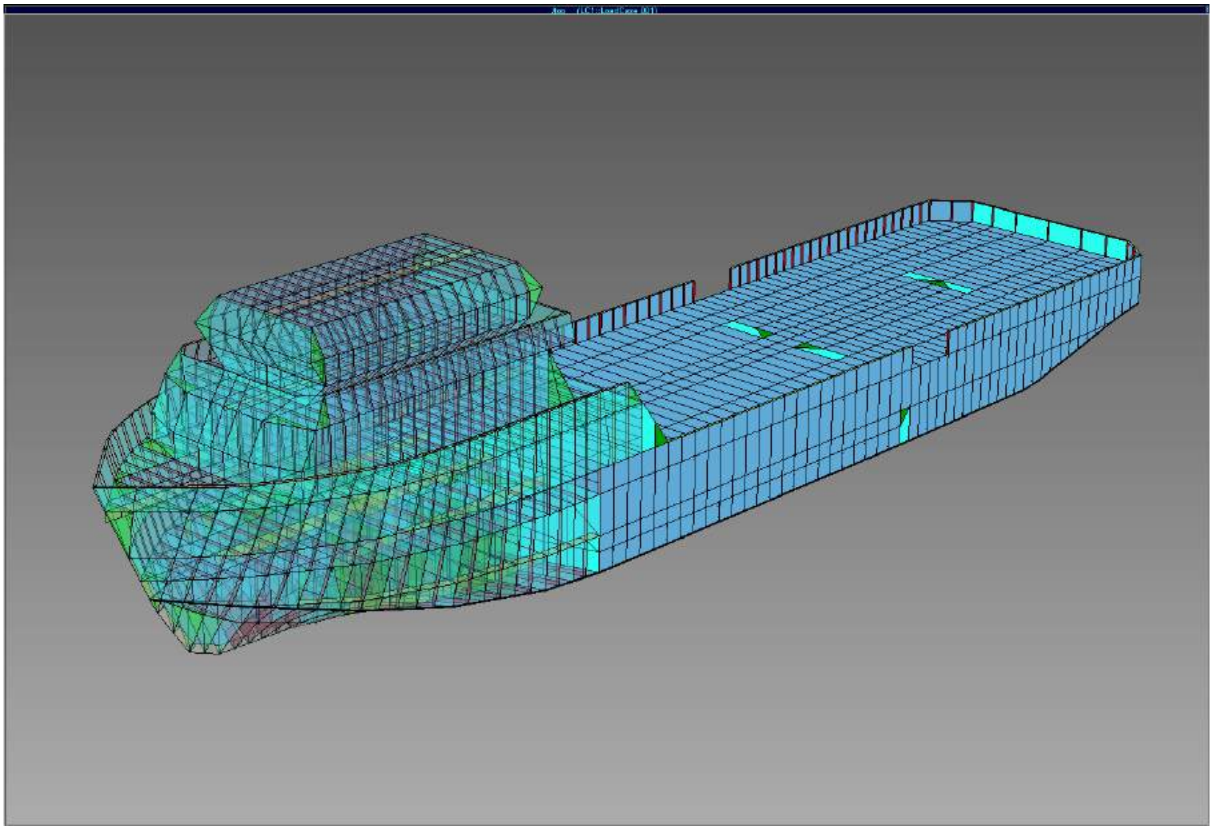
10.13 Transparency View

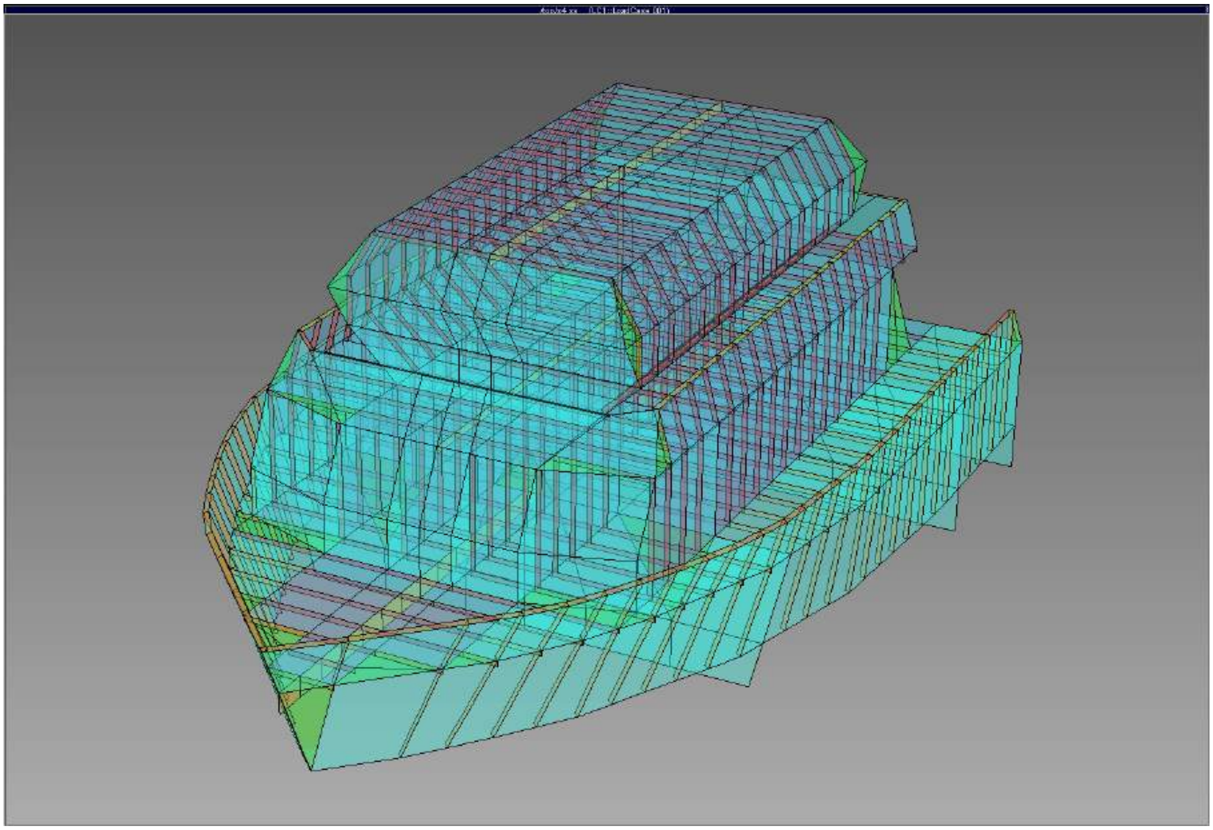
Toolbar		
Menu	**View > Set Visibility > Visibility On**	**View > Set Visibility > Visibility Off**
Parts Tree	**Right-click > Set Transparency On**	**Right-click > Set Transparency Off**

MAESTRO has the ability to toggle a part in a regular or transparent view. This view can be applied at the module level, or the full model. The transparent view will allow all internal structure and external structure to be seen at once.

The transparency view can be set using the toolbar, by clicking either the *Set Part Transparent* icon  or the *Unset Part Transparent* icon  and then clicking the part desired, using the main menu, or by right-clicking on a part within the Parts Tree and selecting *Set Transparency On* or *Off*.

© 2014 Optimum Structural Design, Inc.





ULS-Based Optimization

11

11 ULS-Based Optimization

MAESTRO allows a user to optimize a given structure ranging from a single module to a full-ship model. The optimization uses the MAESTRO loading patterns and failure mode evaluations (and safety factors) to minimize the structural weight and cost, while maximizing safety, given a set of user-defined constraints. At each scantling iteration, a new finite element analysis is run and adequacy parameters are evaluated for all patches in the model. The worst case evaluation patch for each design cluster (a group of evaluation patches with the same scantling properties) is then optimized and the FE analysis is re-run in an iterative manner. The work-flow for performing a structural optimization is presented below. Each of these steps is discussed in more detail in the Optimization sub-sections.

Optimization examples can be found in the MAESTRO Models & Samples directory.

Optimization Work-Flow

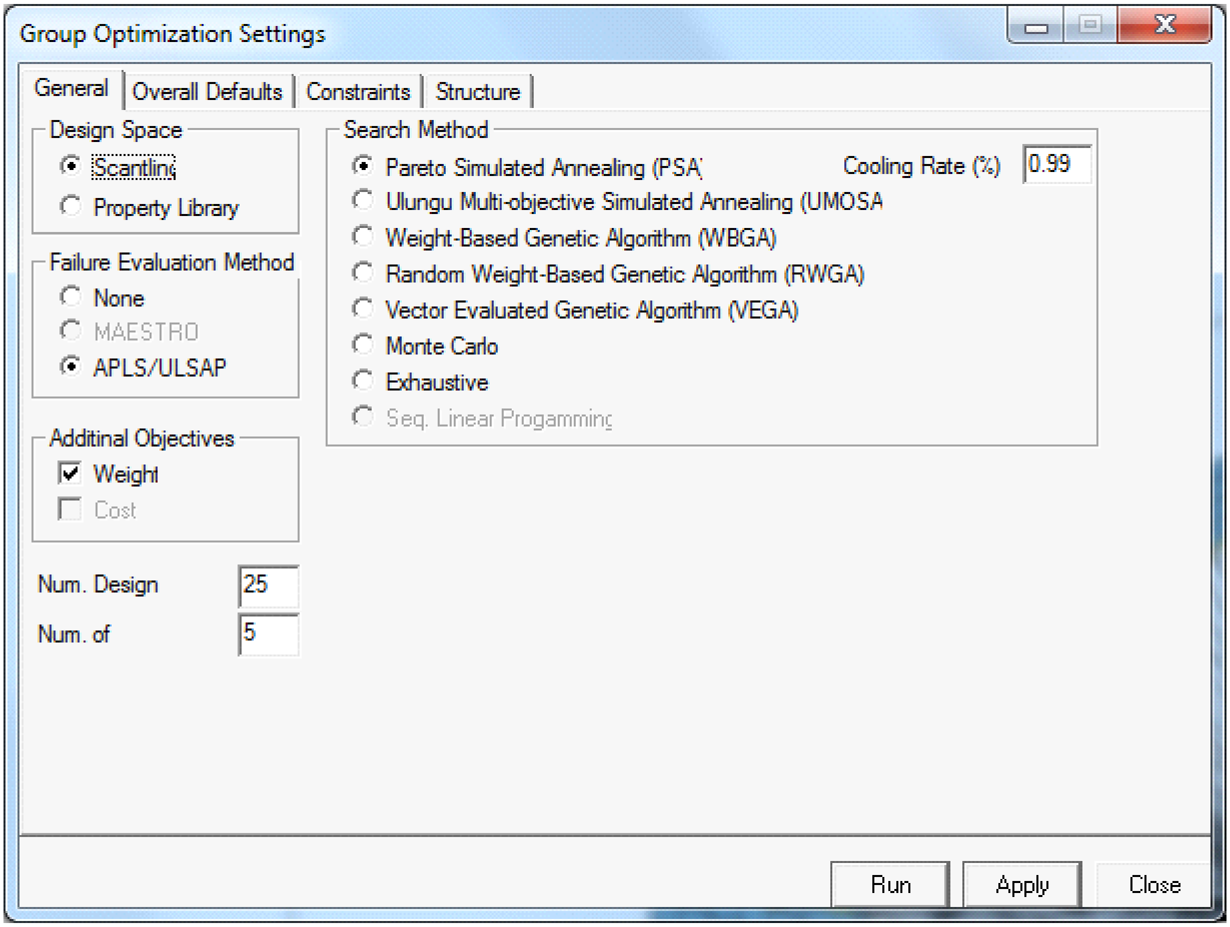
The workflow and subsections assume that the finite element model has been created and the appropriate load cases to use in the optimization process have been defined.

1. Create evaluation patches representing the true stiffened panels of the structure to be optimized. Information on the different methods for creating evaluation patches can be found here.

2. Define general groups of the structure to be optimized which represent areas of common plate and beam properties. For example, the user may desire one or more strakes (or other area) to all have the same plate thickness, frames, and girders for manufacturing purposes.

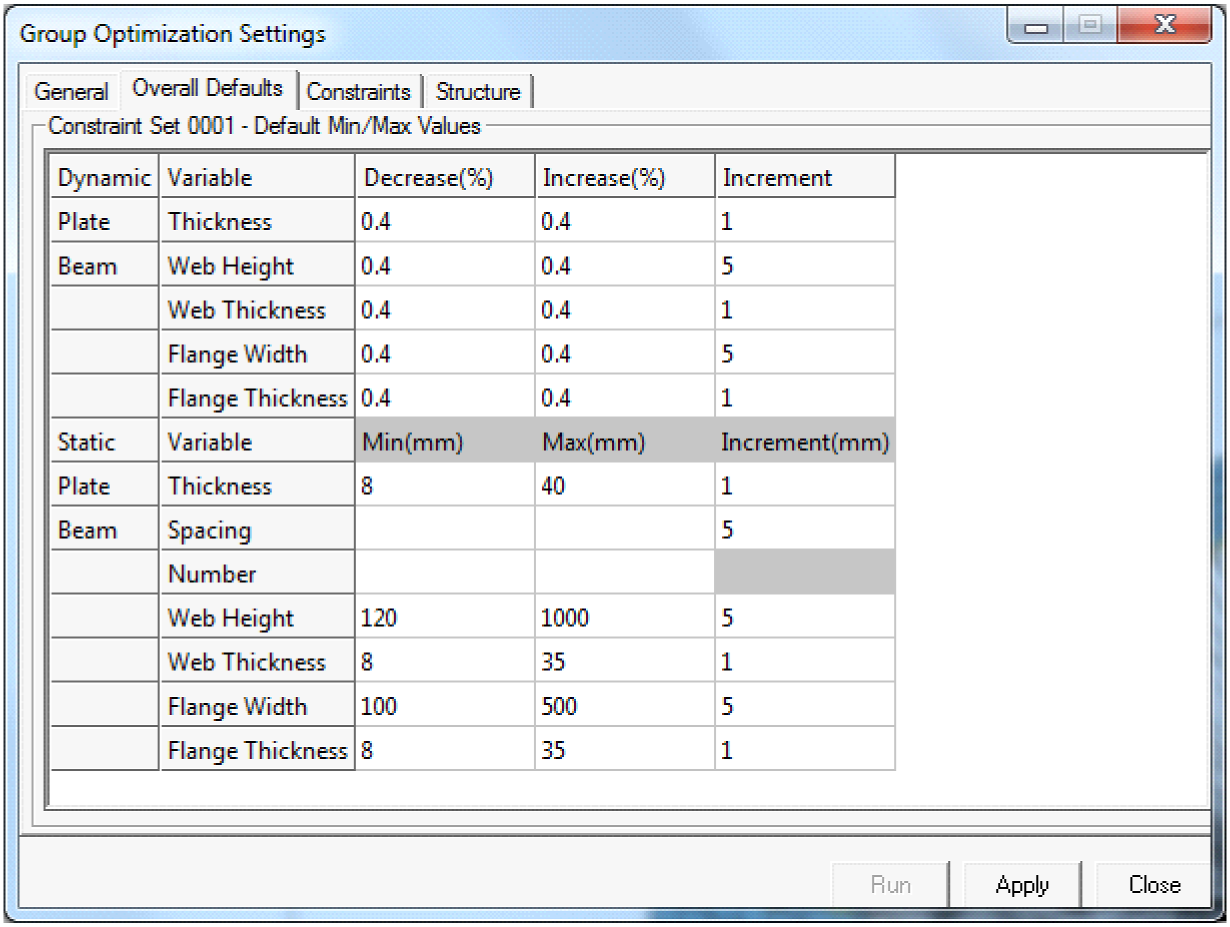
3. Launch the Optimization dialog from the Model > Optimization... menu option and select the general settings for the design, including the search method.

© 2014 Optimum Structural Design, Inc.



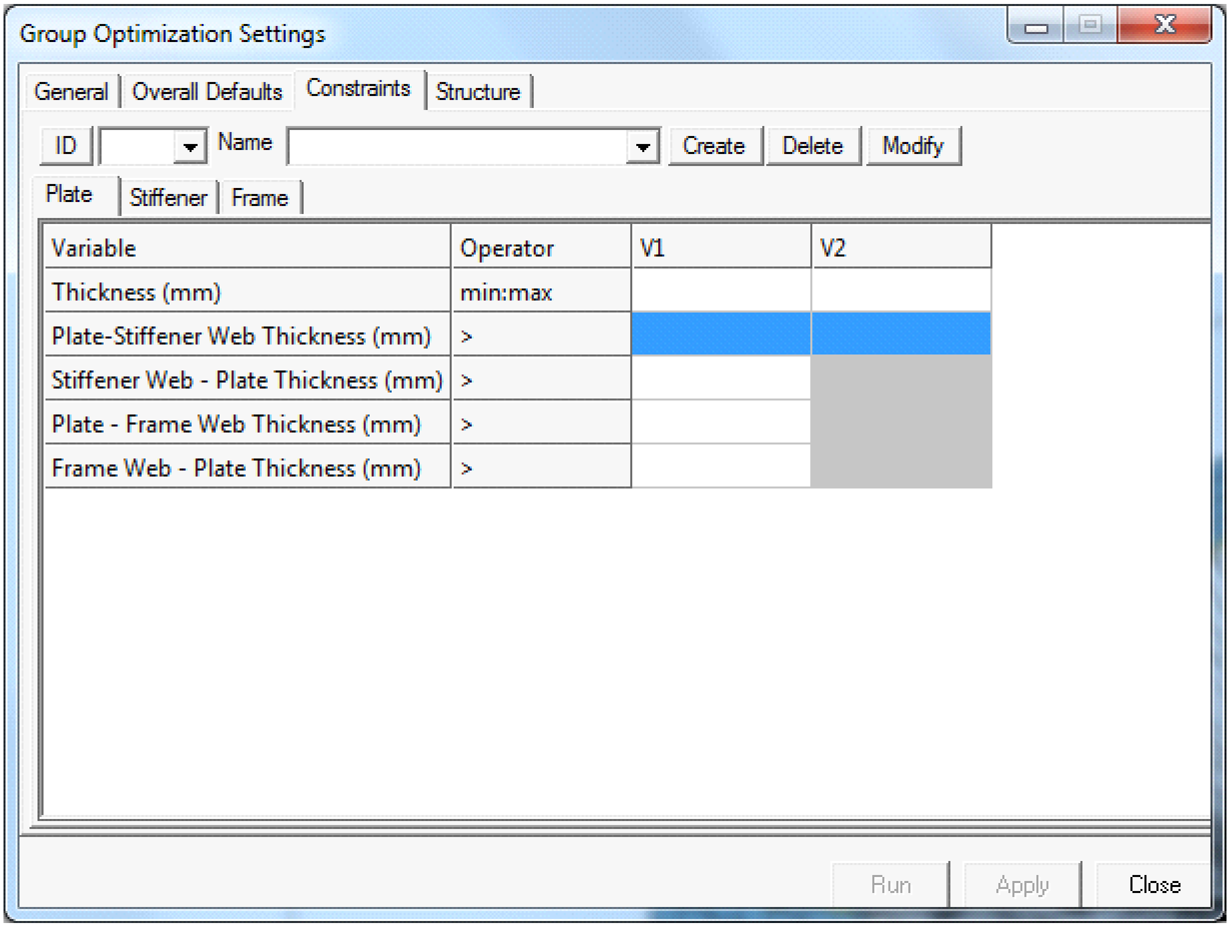
The design space can be defined by either scantling constraints or by limiting properties to those available in the MAESTRO property library.

4. Set the default minimum and maximum constraint values for plates and beams.

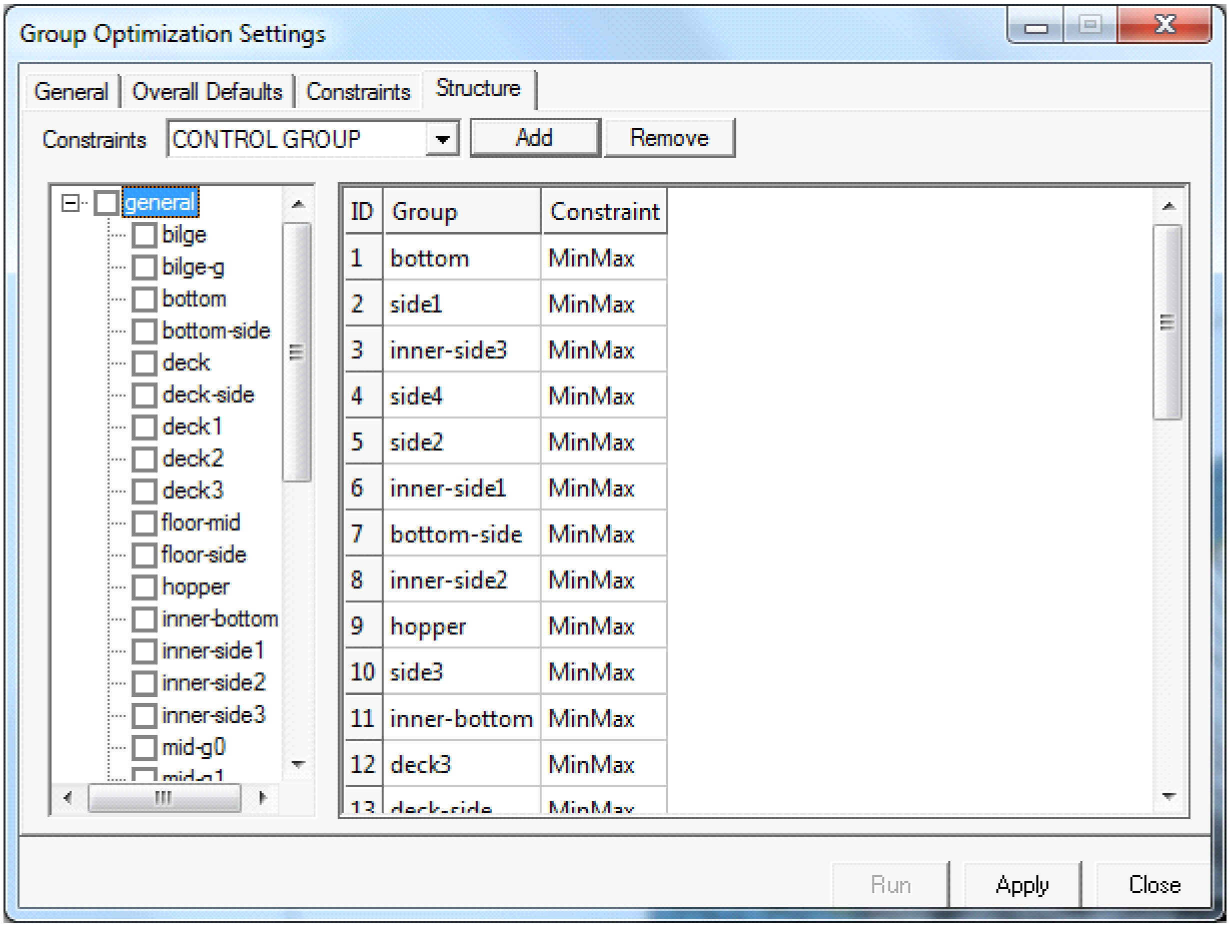


5. Define any custom plate, stiffener, or frame constraints to apply to specific design clusters.

© 2014 Optimum Structural Design, Inc.



6. Add the design clusters to be optimized and assign the constraints to each group.



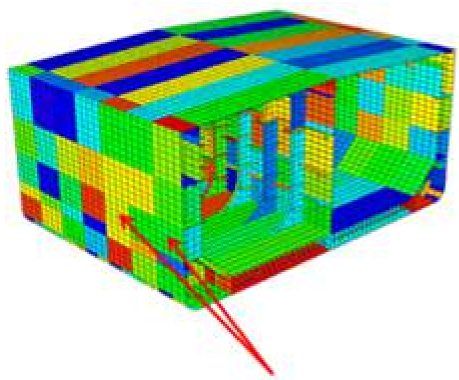
7. Click Run to start the Optimization process. The results will be displayed in the Grid tab of the Output window.

11.1 Setting up the Optimization Model

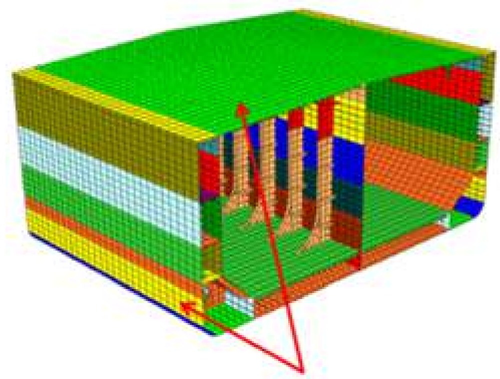
Setting up the Model

During the optimization process, MAESTRO will calculate the adequacy parameters for each evaluation patch in the model, but then use the worst case patch from each design cluster to optimize the structure. The design cluster is intended to represent the typical structural layout of a ship where there are areas of common plating, frames, and stiffeners larger than just a single stiffened panel. The image below shows a midship section with the evaluation patches defined on the left and the design clusters defined on the right.

© 2014 Optimum Structural Design, Inc.



Evaluation Patches

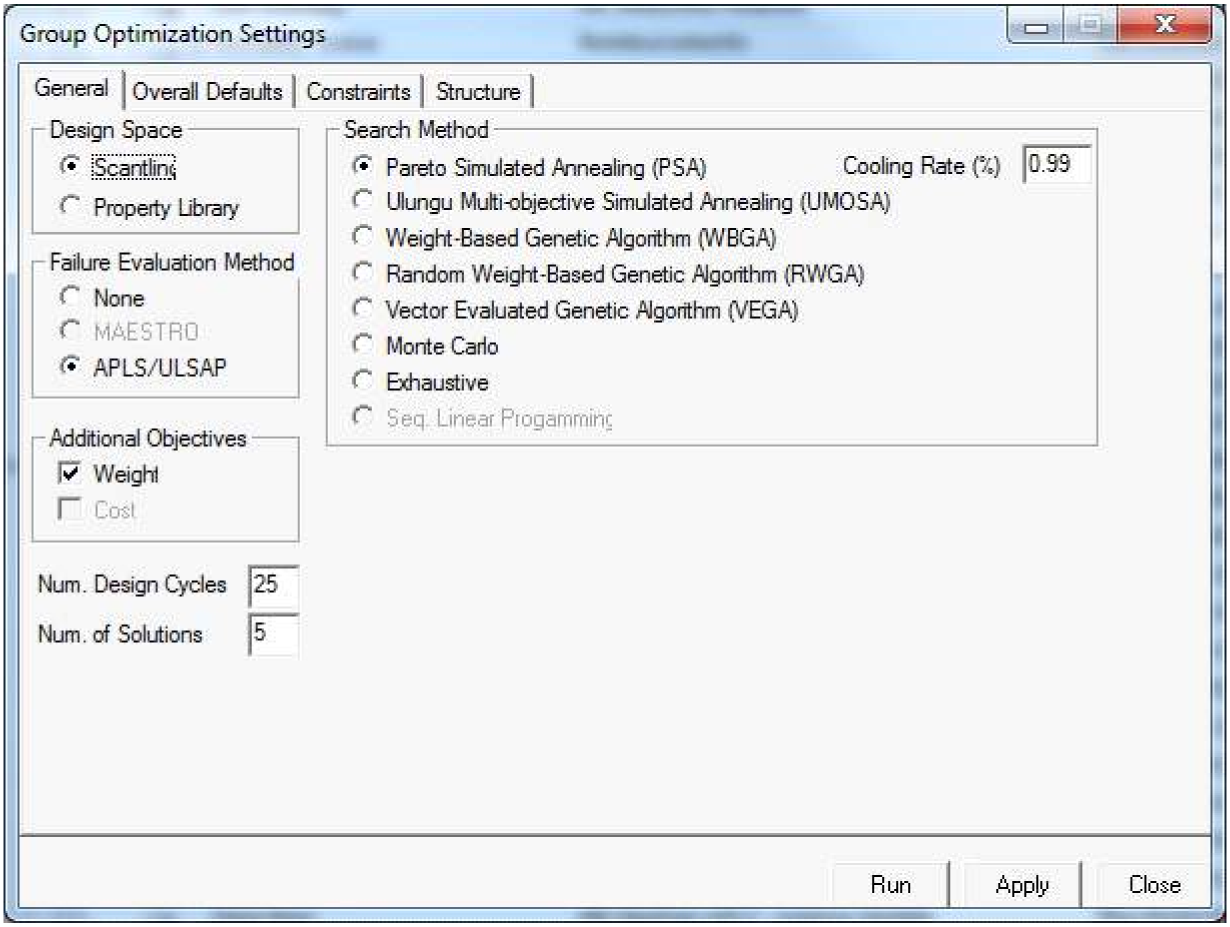


Design Clusters

The design clusters are created as general groups and can be selected using the evaluation patch selection method from the groups dialog. A design cluster must be created for each piece of structure to be included in the optimization. For example, a single module of a ship can be optimized using the full ship FEA results by only creating and adding design clusters for that particular structure representing the module.

Once the design clusters have been defined, the optimization settings can be defined by clicking on the *Model > Optimization* menu item.

The first tab allows the user to define the general settings for the optimization:



The Design Space can be defined by plate, stiffener, and frame properties or by selecting available properties from the existing ones in the MAESTRO properties library. The next section will discuss the implications of these two choices.

The failure mode evaluation defines which failure mode calculations to use during the optimization. Currently, ALPS/ULSAP is the only supported method; however additional criteria will be added in the future.

The Num. Design Cycles field allows the user to specify the number of iterations to perform without finding a more optimal solution before ending the optimization. The Num. of Solutions field allows the user to specify the number of solutions to find before ending the analysis.

The search method field allows the user to select between a variety of search methods including Simulated Annealing, Genetic Algorithms, Monte Carlo, and exhaustive. It is not recommended to use the exhaustive search option with the scantling Design Space option unless the model is very small or the constraints are very strict (i.e., small design space) as the computation time can be very long.

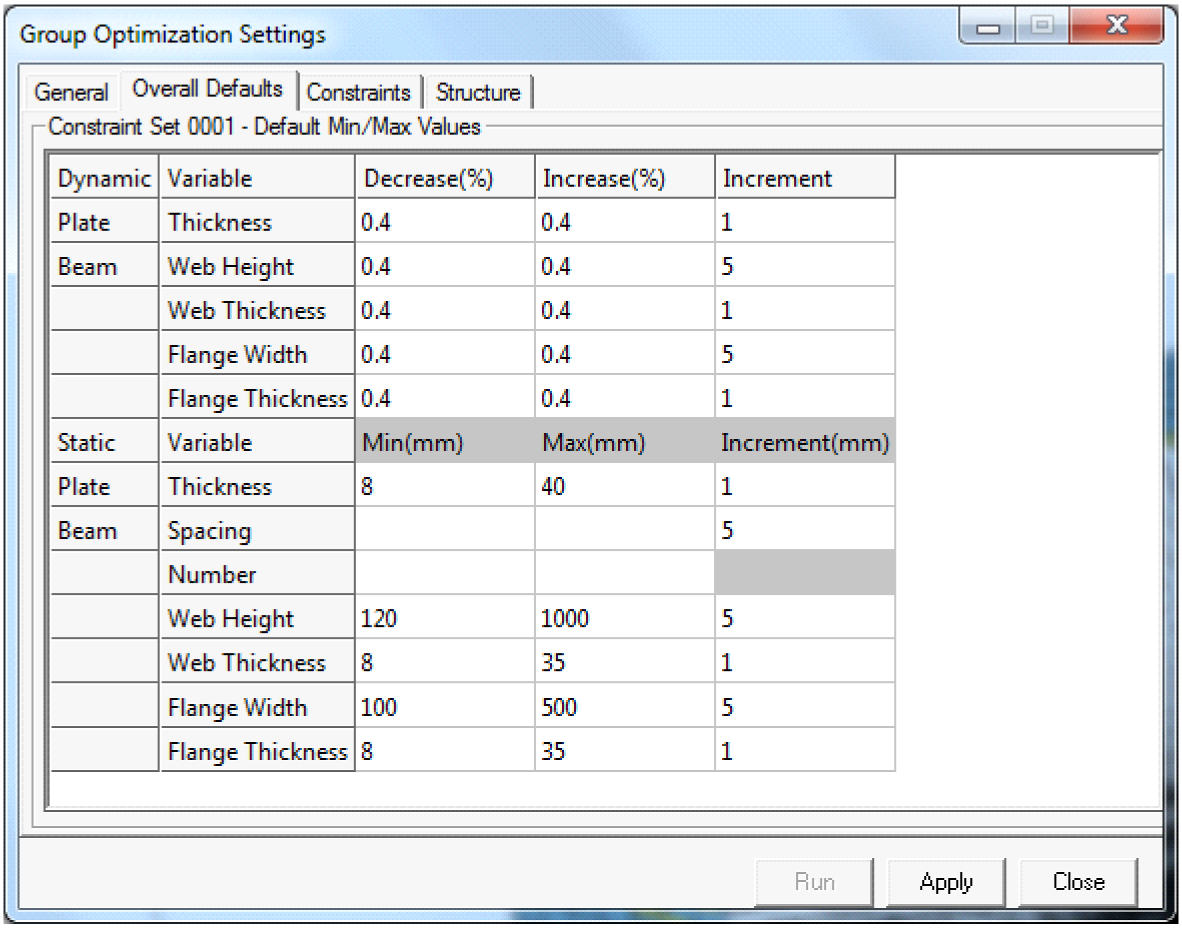
© 2014 Optimum Structural Design, Inc.

11.2 Optimization Parameters

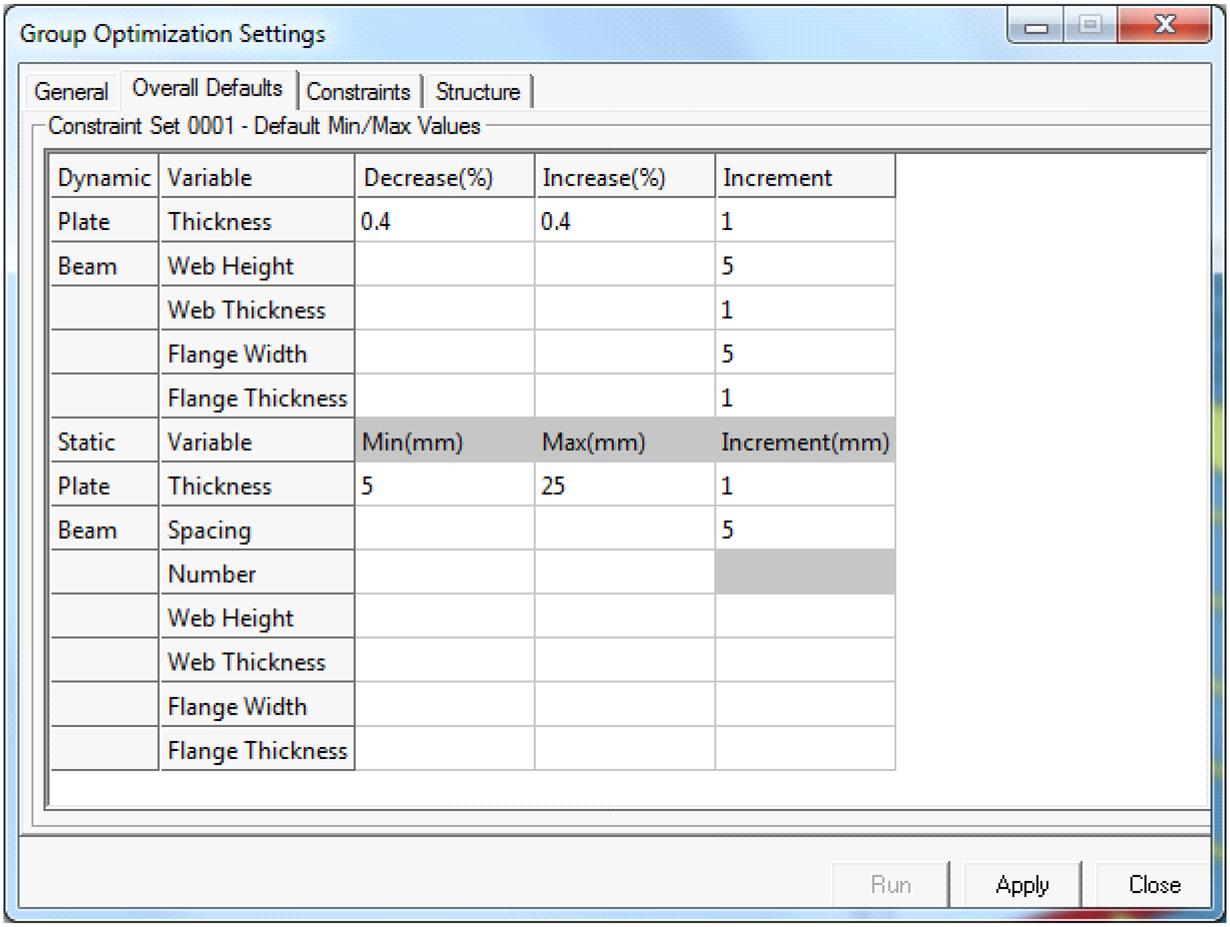
The next three tabs of the Optimization dialog allow the user to specify the constraints for the overall design as well as specific constraints for each design cluster depending on how the Design Space is defined.

Scantling Constraints

If the Scantling Design Space option is selected, Overall and “local” constraints can be defined. On the Overall Defaults tab, the plating and beam properties for the overall design can be set.



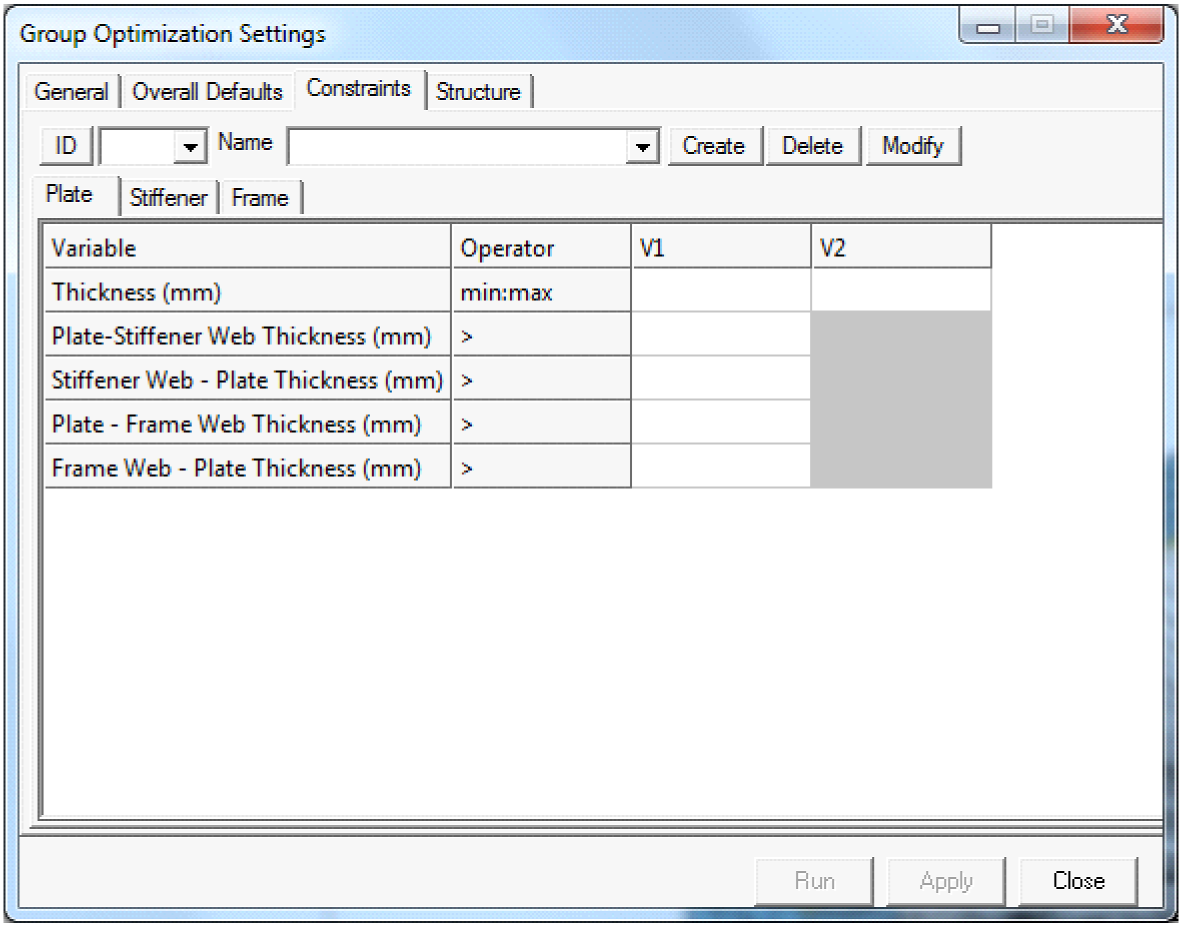
The plate and beam constraints can be set as dynamic which vary by a percent increase and decrease or a minimum and maximum value for each parameter. Leaving a static constraint parameter blank will create a constraint in which that parameter remains fixed, unless overridden by a local constraint. For example, to only allow only the plating thickness to change, the following settings could be used:



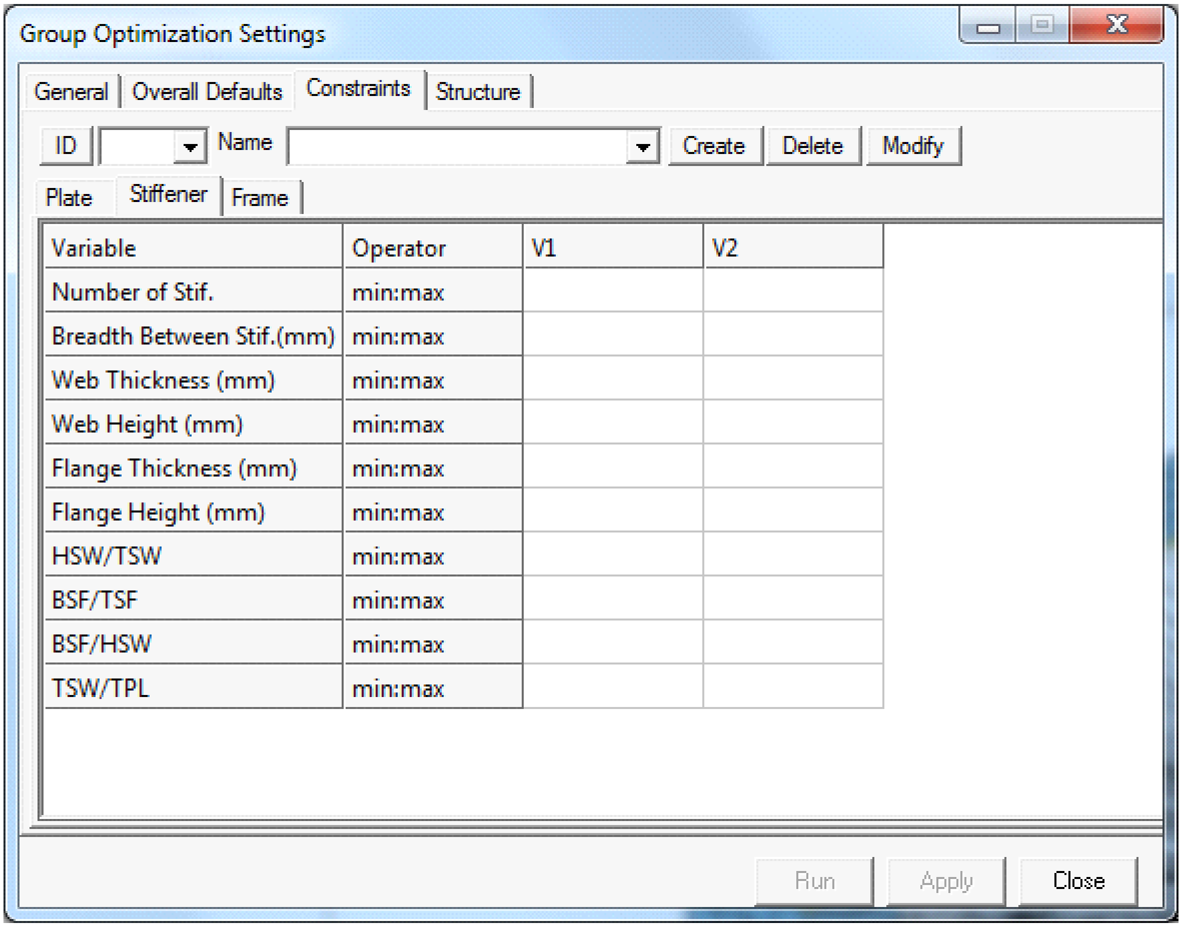
Note: Any static constraints defined will override dynamic constraints for the same parameter.

The constraints tab allows the user to specify local constraints that can be applied to one or more design clusters. To create a new constraint, click the ID, type a name for the constraint (this will be used to identify the constraint when applying to manufacturing groups), enter the plate constraint parameters and click create. Click modify after making changes to the constraint.

© 2014 Optimum Structural Design, Inc.



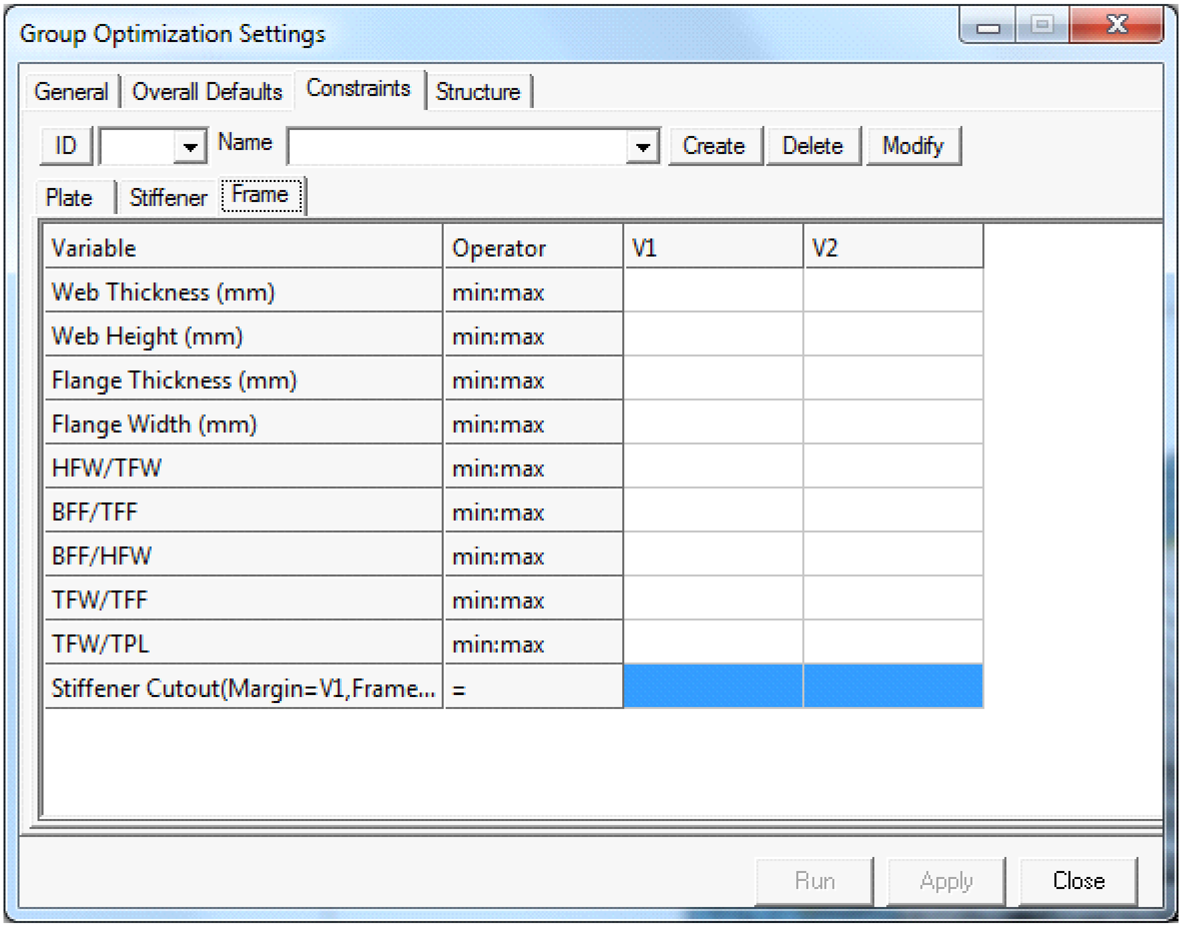
The plate property constraints can be defined in multiple ways by specifying operators and values for one or more inputs. Double-clicking in the operator field changes the way the constraint variable is defined. In addition to specifying possible plate thickness, the differences in plate thickness and stiffener web and frame web can be specified.



The stiffener constraints can be defined by the limits of the explicit stiffener properties as well as the following ratios:

HSW/TSW: Height of stiffener web to thickness of stiffener web.
BSF/TSF: Breadth of stiffener flange to thickness of stiffener flange.
BSF/HSW: Breadth of stiffener flange to height of stiffener web.
TSW/TPL: Thickness of stiffener web to thickness of plate.

© 2014 Optimum Structural Design, Inc.

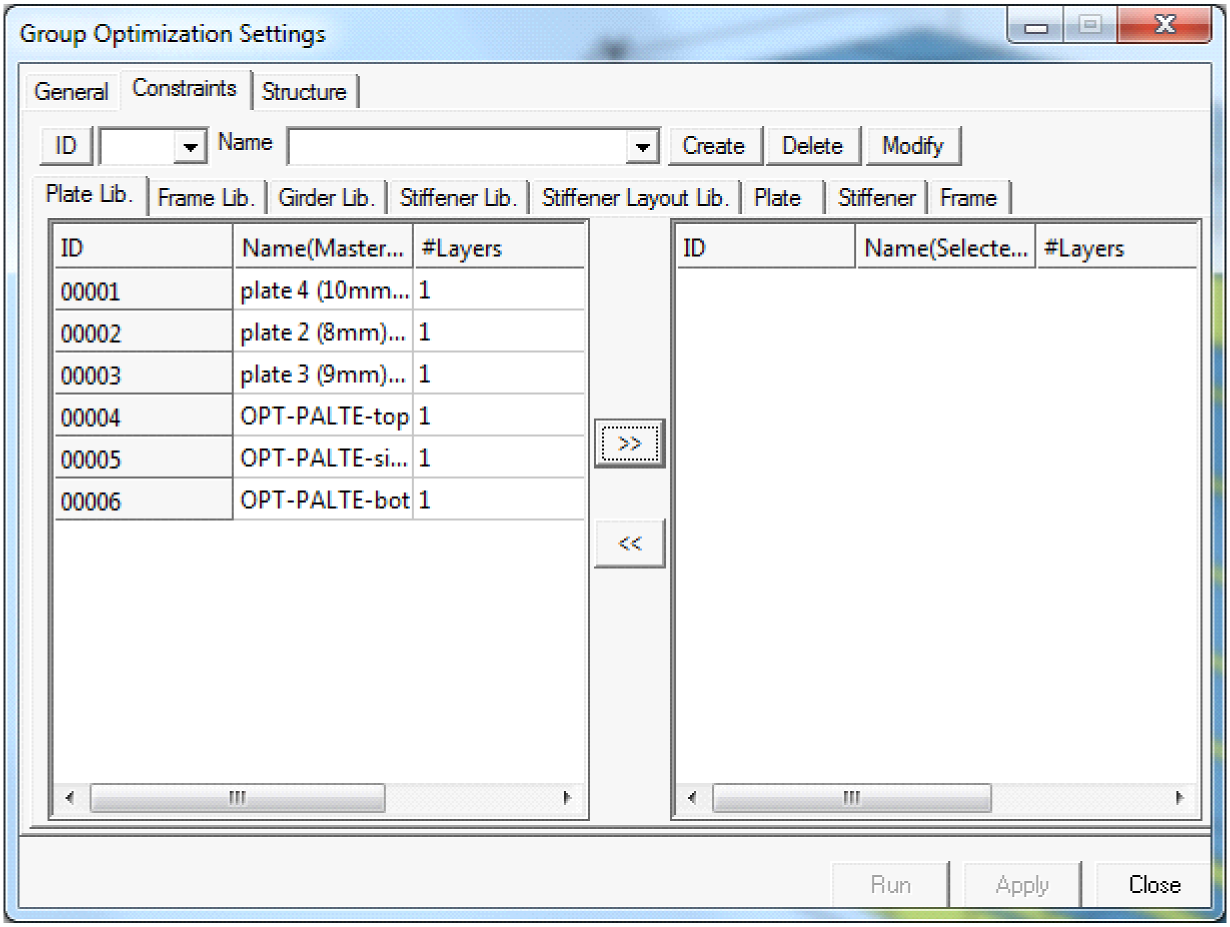


The frame constraints tab is similar to the stiffener tab, but it also allows the user to specify a stiffener cutout parameter where V1 is the margin and V2 is the ratio of frame height to cutout height.

Property Library Constraints

If the Property Library design space definition option is selected, only a single constraints tab is shown in the Optimization dialog.

© 2014 Optimum Structural Design, Inc.

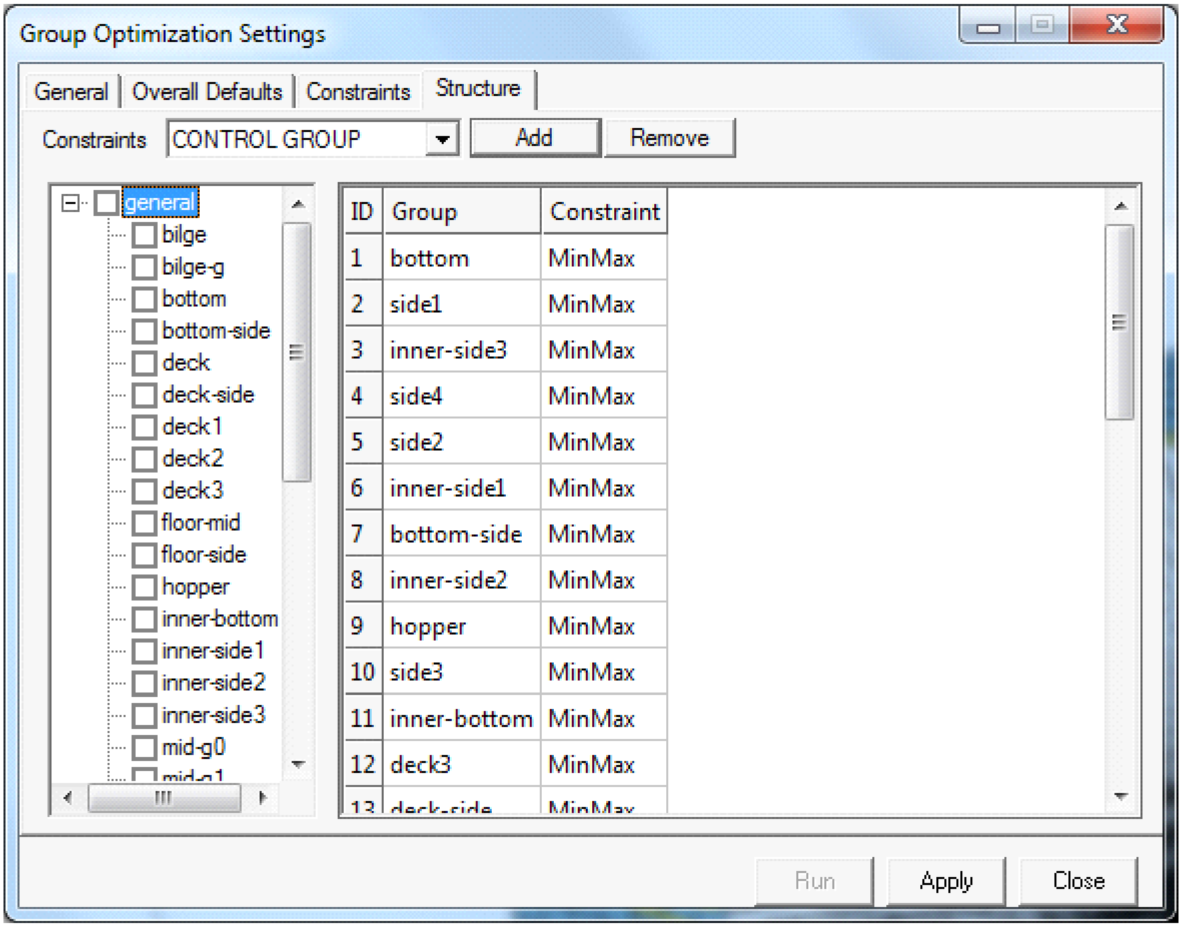


The tabs ending in "Lib." List the available plate, frame, girder, stiffener, and stiffener layouts based on what is defined in the model. In these tabs, constraints are now defined by selecting available properties from the left-hand side of the dialog and moving them to the right-hand side. This constraint will then limit the optimization to only using these selected properties in the constraint definition when applied to a design cluster.

More general, local constraints can also be set and used as they were in the Scantling design space definition.

Applying Constraints

Once the constraints are defined for either method of defining the Design Space, the individual design clusters constraints can be set on the Structure Tab.



The left hand side of the dialog shows the list of available general groups (design clusters) defined for the model. To add a constraint to a design cluster, click the check box next to one or more of the groups in the tree, select the constraint from the drop-down menu, and click Add. Constraints can be removed by selecting the group from the list on the right and clicking Remove.

The MinMax constraint is the overall constraints defined on the Overall Defaults tab. Any manufacturing group can have multiple constraints applied to the group. In this case, the plate, frame, or stiffener constraints would override the overall default constraints. For example, a design cluster may have the overall defaults (MinMax) constraint applied and a frame constraint applied. In this case, the plating and stiffener constraints from the MinMax set would remain and the frame constraint would now use the parameters set in the local constraint set.

A control group defines active evaluation patches. A manufacture group usually has many evaluation patches. However, if one of the patches in the design cluster is defined in the control group, MAESTRO will optimize this patch instead of optimizing the worst patch in a manufacture strake.

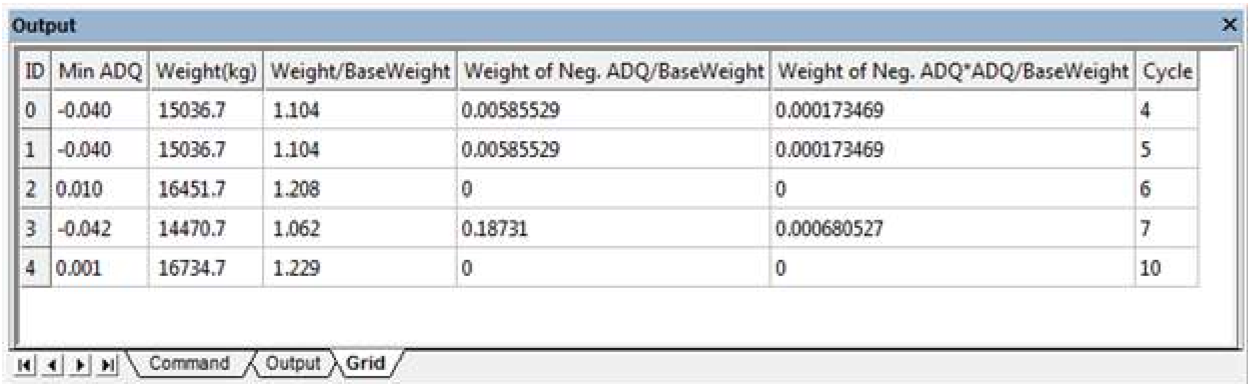
11.3 Optimization Results

To run the optimization after the settings are defined, click Run at the bottom of the Optimization dialog. The progress of the optimization can be followed in the Output tab at the bottom of the

© 2014 Optimum Structural Design, Inc.

screen.

Once an optimization run is complete, the user will be prompted with a message asking whether to display the results. The results of the optimization will be shown in the grid tab at the bottom of the window.



ID	Min ADQ	Weight(kg)	Weight/BaseWeight	Weight of Neg. ADQ/BaseWeight	Weight of Neg. ADQ*ADQ/BaseWeight	Cycle
0	-0.040	15036.7	1.104	0.00585529	0.000173469	4
1	-0.040	15036.7	1.104	0.00585529	0.000173469	5
2	0.010	16451.7	1.208	0	0	6
3	-0.042	14470.7	1.062	0.18731	0.000680527	7
4	0.001	16734.7	1.229	0	0	10

The output displays the results for the X number of designs requested from the general tab of the Optimization dialog. For each design found, the minimum adequacy parameter, new structural weight, ratio of the new structural weight to the original structural weight, ratio of the weight of the structural members with negative adequacy parameters to the original structural weight, ratio of the weight of the structural members with negative adequacy parameters multiplied by the structures adequacy parameter to the original structural weight, and the number of cycles ran before finding the design will be shown. Right-clicking on one of the designs brings up a menu allowing the user to select this design and update the structure. After a new design is selected, a finite element analysis run is required to see stress, displacement, and adequacy parameter results.

© 2014 Optimum Structural Design, Inc.

MAESTRO-Wave (Hydrodynamic Loads)

12

12 MAESTRO-Wave (Hydrodynamic Loads)

The MAESTRO-Wave module provides the ship designer with an integrated frequency-domain and time-domain computational tool to predict the motions and wave loads of any vessel. To compute these hydrodynamic motions and loads, MAESTRO-Wave first calculates the velocity potentials, source strengths, and flow velocities at the centroids of the hydrodynamic panels for each speed, heading, and frequency requested, and then maps the source strength to the structural panels. The equations of motion are formulated using the structural mesh rather than the hydrodynamic mesh. This approach results in a perfect equilibrium for the structural model. Bending moments, shear forces and torsional moments are automatically in closure. No inertia relief and artificial loads are needed to balance the model. The computation of these hydrodynamic forces is based on a user-selected wet panel discretization method:

- 3D potential theory using the zero speed Green function with a speed correction parameter.
- 2D Strip Theory using either the Free Surface Green Function or the Rankine Source Method
- 2.5D High Speed Strip Theory using the Rankine Source Method
- Universal RAO

A variety of visualizations and output data are available for purposes of post-processing.

A typical workflow for using the frequency-domain and time-domain MAESTRO-Wave capabilities is:

1. Run the frequency-domain MAESTRO-Wave for a full spectrum of ship speeds, headings, and waves.
2. Using the frequency-domain results, identify the parameters (i.e., speed, heading, wave frequency, and wave amplitude) of the extreme load cases.
3. Run the time-domain MAESTRO-Wave with the specified speed, heading, wave frequency, and wave amplitude combination.
4. Use the time-domain results to determine the worst-case wave phase.
5. Create a design-wave at this worst-case phase, which includes the non-linear

© 2014 Optimum Structural Design, Inc.

Froude-Krylov affects.

The sections below describe the details of the MAESTRO-Wave computation technology being developed by Dr. Chengbi Zhao (South China University of Technology) and DRS Training & Control Systems, LLC, Advanced Marine Technology Center (DRS AMTC). These sections include theoretical background to MAESTRO-Wave, as well as its application and use within the MAESTRO system. The last section provides validation material, which compares with results from industry recognized numerical tools and experimental results.

Table of Contents

12.1 Theory Manual

MAESTRO-Wave Theoretical Background

Introduction

This section describes the theoretical background of the computer program MAESTRO-WAVE. It provides an overview of the theories and the computational methodologies in a concise manner. The use of MAESTRO-Wave and its capabilities are described in the MAESTRO-Wave User Manual section.

MAESTRO-Wave is a panel program designed to solve the boundary-value problem for the interaction of water-waves with bodies in finite- and infinite-water depth. Viscous effects of the fluid are ignored and thus the flow field is potential without circulation. A perturbation series solution of the nonlinear boundary value problem is postulated with the assumption that the wave amplitude is small compared to the wave length. It is also assumed that the body stays at its mean position and any oscillatory amplitude of the body motion is of the same order as the wave amplitude. The time harmonic solutions corresponding to the first-terms of the series expansion are solved for a given steady-state incident wave field. The incident wave field is assumed to be represented by a superposition of the fundamental first-order solutions of particular frequency components in the absence of the body. The boundary value problem is recast into integral equations using the wave source potential as a Green function. The integral equation is then solved by a panel method for the unknown velocity potential and/or the source strength on the body surface. Using the latter, the fluid velocity on the body surface is evaluated.

For incident waves of specific frequencies and wave headings, the linear problem is solved first. The following sections describe the problem setup and the procedure for finding a solution, followed by a discussion on the rigid body motions. The expressions for hydrodynamic forces are also derived, including hydrodynamic coefficient definitions.

Since the focus is primarily on hydrodynamic interactions, it is appropriate to mention any restrictions on or assumptions about the bodies which may be analyzed using MAESTRO-Wave. The bodies are assumed to be neutrally buoyant, and they may be submerged or surface piercing. An extremely thin submerged body may be difficult to analyze when the thickness between the two facing surfaces is small compared to the other dimensions of the body may render the integral equation ill-conditioned. However, a thin body floating on the free surface (in other words the draft is very small) is not included in this category, since only one side of the body would be adjacent to the fluid domain. Other than these panel considerations, the standard constraints for potential flow theory apply.

© 2014 Optimum Structural Design, Inc.

Linear Boundary Value Problem

A ship with forward speed U in deep water is considered. The right-handed Cartesian coordinate system O-xyz is adopted. The origin of the system is located at the intersection of the center-plane and the mid-ship section of the undisturbed free surface, with x-axis pointing forward, z-axis vertically upward and y-axis to port side, see Figure 1. This is a ship-fixed system and the ship is free to sink and trim.

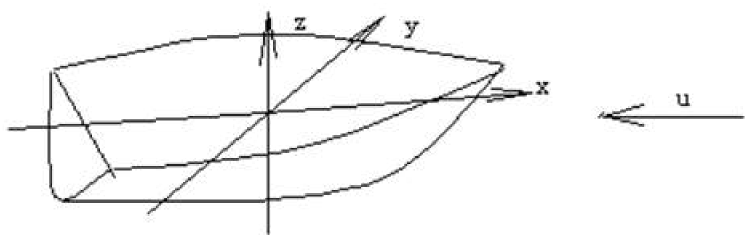


Figure 1. MAESTRO Wave Coordinate System

The unsteady six degree-of-freedom ship motions are described using the standard seakeeping notation, including three translations (surge, sway, and heave) and three rotations (roll, pitch, and yaw) about the x-, y-, and z-axes, respectively. The rotations are given by Euler angles. The incoming wave system is defined later.

The total velocity potential in the fluid domain can be expressed as

$$\psi = \Phi + \phi \qquad (3.1)$$

where Ψ is the total potential, Φ is the double body potential with advanced speed U, and ϕ is the unsteady potential.

Double Body Potential

The double body potential Φ can be expressed as

$$\Phi = -Ux + \varphi \qquad (3.2)$$

where the advance speed U introduces a steady basis flow, and φ is the disturbance potential due to body in the steady flow. The double body potential is subject to the following:

$$[L] \qquad \nabla^2\Phi = 0 \qquad (3.3)$$

$$[F] \qquad \frac{\partial \Phi}{\partial z} = 0 \tag{3.4}$$

$$[B] \qquad \frac{\partial \Phi}{\partial n} = 0 \tag{3.5}$$

Where $\vec{n} = (n_x, n_y, n_z)$ denotes the unit normal vector of body surface, pointing into the body.

The expressions above can be rewritten in terms of the disturbance potential φ, which is represented as the potential of a source density distribution over the body surface S. The potential at a point $p(x, y, z)$ in space due to a unit point source located at a point $q(\xi, \eta, \delta)$ on the body surface is $1/r(p,q)$,where $r(p,q)$ is the distance between points p and q. The point $q_1(\xi, \eta, -\delta)$ is the symmetry point of point q for waterplane, and $r_1(p, q_1)$ is the distance between points p and q_1. Accordingly, the potential at p due to a source density distribution $\sigma(q)$ on the body surface is

$$\varphi(x, y, z) = \oiint_S G(p,q)\sigma(q)ds \tag{3.6}$$

$G(p,q)$ is the double body Green function

$$G(p,q) = 1/r(p,q) + 1/r_1(p, q_1) \tag{3.7}$$

According to the boundary of body surface, the integral equation for source density σ can be obtained by

$$2\pi\sigma(p) + \oiint_S \frac{\partial G(p,q)}{\partial n}\sigma(q)ds = n_x U \tag{3.8}$$

Once this equation is solved for σ, the potential at any point can be obtained by equation (3.4). The velocity components at any point of the flow are obtained by equation (3.9).

$$\nabla\varphi(x, y, z) = \oiint_S \nabla G(p,q)\sigma(q)ds \tag{3.9}$$

Unsteady Potential

According to linear potential theory, and using the classical convention of notation, the unsteady potential ϕ of a floating body is a superposition of the potentials due to the undisturbed incoming wave ϕ_0, the potential due to the diffraction of the undisturbed incoming wave on the fixed body ϕ_7 and the radiation potentials due to the six body motions $\phi_i, i = 1,2,3,4,5,6$ (or surge, sway, heave, roll, pitch, yaw),

$$\phi = \phi_0 + \phi_7 + \sum_{i=1}^{6} \phi_i \qquad (3.10)$$

In regular waves a linear potential ϕ, which is a function of the earth-fixed coordinates and of time *t*, can be written as a product of a space-dependent term and a harmonic time-dependent terms, as follows:

$$\phi = \left\{ [A(\varphi_0 + \varphi_7) + \sum_{i=1}^{6} \xi_i \varphi_i] e^{i \cdot \omega \cdot t} \right\} \qquad (3.11)$$

Where φ_0 is the unit incident wave potential, φ_7 is the unit diffraction potential, $\varphi_i, i = 1,2,3,4,5,6$ are the unit radiation potentials, and A is the amplitude of the incident wave. The incident wave frequency is given by ω_0, and ω is the frequency of encounter, which are related by the ship's speed and relative heading to the incident wave direction β, and wave number k, defined as follows

$$\omega = \omega_0 - Uk \cos \beta \qquad (3.12)$$

$$k = \frac{\omega_0^{\ 2}}{g} \qquad (3.13)$$

The wave direction angle β is $\beta = 180$ for head seas, $\beta = 135$ for bow quartering seas, $\beta = 90$ for beam seas, $\beta = 45$ for stern quartering seas, and $\beta = 0$ for the following seas.

Incident Wave Potential

The incident wave, known *a priori*, is defined using the expression for infinite deep water

$$\varphi_0 = i\frac{g}{\omega_0} e^{k(z - ix\cos\beta - iy\sin\beta)} \tag{3.14}$$

The velocity of the incident wave at any point is

$$\frac{\partial\varphi_0}{\partial x} = \frac{kg\cos\beta}{\omega_0} \cdot \frac{\cosh k(z+h)}{\cosh kh} . e^{k(-ix\cos\beta - iy\sin\beta)} \tag{3.15}$$

$$\frac{\partial\varphi_0}{\partial y} = \frac{kg\sin\beta}{\omega_0} \cdot \frac{\cosh k(z+h)}{\cosh kh} . e^{k(-ix\cos\beta - iy\sin\beta)} \tag{3.16}$$

$$\frac{\partial\varphi_0}{\partial z} = i\frac{kg}{\omega_0} \cdot \frac{\sinh k(z+h)}{\cosh kh} . e^{k(-ix\cos\beta - iy\sin\beta)} \tag{3.17}$$

Radiation Potential

Each radiation component $\varphi_i, (i = 1,2,3,4,5,6)$ is subject to the following boundary conditions

$$[L] \qquad \nabla^2\varphi_i = 0 \tag{3.18}$$

$$[F] \qquad \frac{\partial\varphi_i}{\partial z} - k\varphi_i = 0 \tag{3.19}$$

$$[B] \qquad \frac{\partial\varphi_i}{\partial n} = i\omega n_i + m_i \tag{3.20}$$

$$[Bot] \qquad \frac{\partial\varphi_i}{\partial n} = 0 \tag{3.21}$$

$\vec{m}$ is the m -item.

$$\vec{r} = (x, y, z) \tag{3.22}$$

$$\vec{n} = (n_1, n_2, n_3) \tag{3.23}$$

© 2014 Optimum Structural Design, Inc.

$$(n_4, n_5, n_6) = \vec{r} \times \vec{n} \tag{3.24}$$

$$(m_1, m_2, m_3) = -(\vec{n} \cdot \nabla)\nabla\Phi \tag{3.25}$$

$$(m_4, m_5, m_6) = -(\vec{n} \cdot \nabla)(\vec{r} \times \nabla\Phi) \tag{3.26}$$

The source density on body surface for radiation potential can be obtained by

$$2\pi\sigma_i(p) + \oiint_S \frac{\partial G(p,q)}{\partial n}\sigma_i(q)ds = i\omega n_i + m_i \tag{3.27}$$

for $i = 1,2,3,4,5,6$

The Green function is

$$G(p,q) = \frac{1}{r(p,q)} + \frac{1}{r_1(p,q_1)} + kF(X,Y) - 2\pi ike^{-Y}J_0(X) \tag{3.28}$$

Where

$$X = kR$$

$$Y = k(z+\delta)$$

$$R = [(x-\xi)^2 + (y-\eta)^2]^{\frac{1}{2}}$$

$$F(X,Y) = -\pi e^{-Y}[H_0(X) + Y_0(X)] - 2e^{-Y}\int_0^Y \frac{e^t}{(X^2+t^2)^{\frac{1}{2}}}dt \tag{3.29}$$

And J_0 is the Bessel function of the first kind, Y_0 is the Bessel function of the second kind, H_0 is the Struve function. The integral term in (3.29) can be calculated by a self-adaptive trapezoidal integral.

Once equation (3.27) is solved for σ_i $i = 1,2,3,4,5,6$, the radiation potential and velocity components at any point are obtained by

© 2014 Optimum Structural Design, Inc.

$$\varphi_i(x,y,z) = \oiint_S G(p,q)\sigma_i(q)ds \qquad i = 1,2,3,4,5,6 \tag{3.30}$$

$$\nabla\varphi_i(x,y,z) = \oiint_S \nabla G(p,q)\sigma_i(q)ds \qquad i = 1,2,3,4,5,6 \tag{3.31}$$

Diffraction Potential

The diffraction potential φ_7 is subject to the boundary conditions

$$[L] \qquad \nabla^2\varphi_7 = 0 \tag{3.32}$$

$$[B] \qquad \frac{\partial(\varphi_7 + \varphi_0)}{\partial n} = 0 \tag{3.33}$$

$$[Bot] \qquad \frac{\partial\varphi_7}{\partial n} = 0 \tag{3.34}$$

And the source density on body surface for diffraction potential can be obtained by

$$2\pi\sigma_7(p) + \oiint_S \frac{\partial G(p,q)}{\partial n}\sigma_7(q)ds = -\frac{\partial\varphi_0}{\partial n} \tag{3.35}$$

The Green function of (3.28) is adopted in here to solve the boundary value problem.

Similarly as was done for radiation, once equation (3.35) is solved for σ_7, the diffraction potential and velocity components at any point are obtained by

$$\varphi_7(x,y,z) = \oiint_S G(p,q)\sigma_7(q)ds \tag{3.36}$$

$$\nabla\varphi_7(x,y,z) = \oiint_S \nabla G(p,q)\sigma_7(q)ds \tag{3.37}$$

© 2014 Optimum Structural Design, Inc.

Unsteady Pressure, Force, and Moment

Unsteady Pressure

The pressure of body surface on wave can be calculated by Bernoulli's equation.

$$p = -\rho(\psi_t + \frac{1}{2}\nabla\psi \cdot \nabla\psi - \frac{1}{2}U^2 + gz) \tag{4.1}$$

Where $\psi = \Phi + \phi$

A Taylor series expansion of about the mean position of the body, up to terms linear in ϕ, gives

$$p = -\rho\left[\frac{1}{2}\nabla\Phi \cdot \nabla\Phi - \frac{1}{2}U^2 + gz\right] - \rho(\phi_t + \nabla\Phi \cdot \nabla\phi) - \rho\left[(\vec{a} \cdot \nabla)(\frac{\nabla\Phi \cdot \nabla\Phi}{2} + gz)\right] \tag{4.2}$$

The steady part of the pressure field can be extracted from above formula.

$$p_s = -\rho\left[\frac{1}{2}\nabla\Phi \cdot \nabla\Phi - \frac{1}{2}U^2 + gz\right] \tag{4.3}$$

The unsteady part is

$$p_u = -\rho(\phi_t + \nabla\Phi \cdot \nabla\phi) - \rho\left[(\vec{a} \cdot \nabla)(\frac{\nabla\Phi \cdot \nabla\Phi}{2} + gz)\right] \tag{4.4}$$

In here

$$\phi = \left\{[A(\varphi_0 + \varphi_7) + \sum_{i=1}^{6}\xi_i\varphi_i]e^{i\cdot\omega\cdot t}\right\} \tag{4.5}$$

$$\phi_t = i\varpi\left\{[A(\varphi_0 + \varphi_7) + \sum_{i=1}^{6}\xi_i\varphi_i]e^{i\cdot\omega\cdot t}\right\} \tag{4.6}$$

where

$$\vec{a} = \vec{\xi} + \vec{\Omega} \times \vec{r} \tag{4.7}$$

© 2014 Optimum Structural Design, Inc.

and $\vec{\xi} = \{\xi_1, \xi_2, \xi_3\}$ is the response amplitude operator (RAO) of the surge, sway and heave.

$\bar{\Omega} = \{\xi_4, \xi_5, \xi_6\}$ is the response amplitude operator (RAO) of the roll, pitch and yaw.

Added Mass and Damping Coefficients

The added mass and damping coefficient are associated with the first term in the right hand side of (4.4) and depend on both the forward speed and the frequency of oscillation.

The add mass coefficient is

$$a_{ij} = -\frac{\rho}{\omega^2} \mathrm{Re}\left\{\iint_B \left[i\omega\varphi_j + \nabla\Phi \cdot \nabla\varphi_j\right] n_i ds\right\} \begin{matrix} i = 1,2,3,4,5,6 \\ j = 1,2,3,4,5,6 \end{matrix} \quad (4.8)$$

The damping coefficient is

$$b_{ij} = \frac{\rho}{\omega} \mathrm{Im}\left\{\iint_B \left[i\omega\varphi_j + \nabla\Phi \cdot \nabla\varphi_j\right] n_i ds\right\} \begin{matrix} i = 1,2,3,4,5,6 \\ j = 1,2,3,4,5,6 \end{matrix} \quad (4.9)$$

Restoring Force and Moment Coefficient

The restoring coefficient is associated with the second term in the right hand side of (4.4) and depends on both the forward speed and they are associated with gradient of the hydrostatic pressure and of the steady dynamic pressure field. The restoring coefficient is

$$c_{ij} = \rho \iint_{(\bar{B})} ((\vec{a} \cdot \nabla)(\frac{\nabla\Phi \cdot \nabla\Phi}{2} + gz)) n_i ds \quad \begin{matrix} i = 1,2,3,4,5,6 \\ j = 1,2,3,4,5,6 \end{matrix} \quad (4.10)$$

Exciting Force and Moment

The Exciting force is due to the incident and diffracted wave potential. In view of the first term in the right hand side of (4.4), the exciting force can be expressed as follows

$$X_i = -\rho\left\{\iint_B \left[i\omega(\varphi_0 + \varphi_7) + \nabla\Phi \cdot \nabla(\varphi_0 + \varphi_7)\right] n_i ds\right\} \quad i = 1,2,3,4,5,6 \quad (4.11)$$

© 2014 Optimum Structural Design, Inc.

Motion in Regular Waves

The unsteady component of the pressure is examined under the assumptions of small monochromatic motions at a single frequency. The unsteady force and moment on the body due to the oscillatory motion are expressed as follows

$$F_i(t) = \left\{ e^{i\omega t} \left[X_i + \sum_{j=1}^{6} \xi_j (\omega^2 a_{ij} - i\omega b_{ij} - c_{ij}) \right] \right\} \qquad i = 1,2,3,4,5,6 \tag{5.1}$$

And the components of unsteady force is $F = (F_1, F_2, F_3)$ while the components of unsteady moment is $M = (F_4, F_5, F_6)$.

The equations of motion that govern the steady-state time-harmonic response of the body follow from the application of Newton's second law. Using the definitions (5.1) of the resultant force, the response amplitude operator RAO can be determined by the linear equation system.

$$\sum_{j=1}^{6} [-\omega^2 (M_{ij} + a_{ij}) + i\omega b_{ij} + c_{ij}]\xi_j = X_i \qquad i = 1, \cdots, 6 \tag{5.2}$$

Where ξ_j is RAO, $\xi_j = \xi_R + \xi_I \cdot i$. M_{ij} is the mass matrix. Newman (1977) gives a general expression for the mass matrix, as follows

$$M_{ij} = \begin{bmatrix} M & & & & & \\ & M & & & & \\ & & M & & & \\ & & & I_{11} & I_{12} & I_{13} \\ & & & I_{21} & I_{22} & I_{23} \\ & & & I_{31} & I_{32} & I_{33} \end{bmatrix}$$

and

$$I_{ij} = \iint_{V_B} \rho_B [(x^2 + y^2 + z^2)\delta_{ij} - x_i x_j] dv$$

$$M = \iint_{V_B} \rho_B dv$$

where

$$x_1 = x, x_2 = y, x_3 = z$$

© 2014 Optimum Structural Design, Inc.

$$\delta_{ij} = \begin{cases} 1 & i = j \\ 0 & i \neq j \end{cases}$$

The RAO $\xi_j \quad j = 1,2,3,4,5,6$ can be obtained by solved the complex coefficient linear equation group (5.2).

Wave Induced Shear Force and Bending Moment

The wave induced shear force and bending moment are defined as Figure 2:

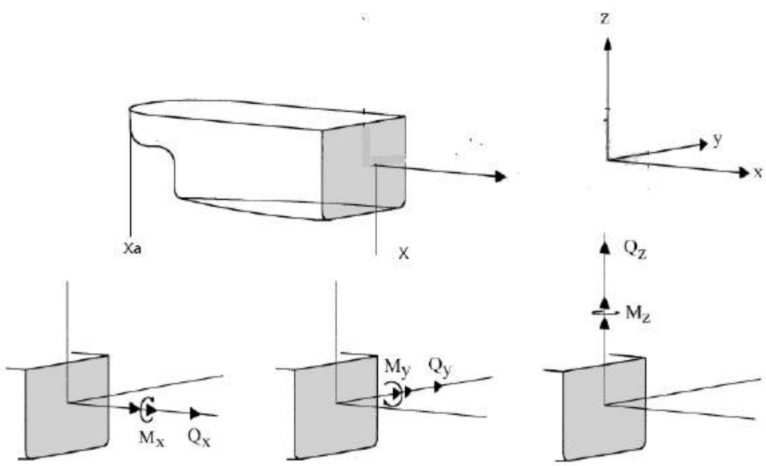


Figure 2. Definition of Shear Force and Bending Moment

The Wave Induced SF and BM Shear Force in Wave is:

$$\begin{bmatrix} Q_x \\ Q_y \\ Q_z \\ M_x \\ M_y \\ M_z \end{bmatrix} = \iint_B p \begin{bmatrix} n_1 \\ n_2 \\ n_3 \\ n_4 \\ n_5 \\ n_6 \end{bmatrix} ds - \omega^2 [M] \begin{bmatrix} \xi_1 \\ \xi_2 \\ \xi_3 \\ \xi_4 \\ \xi_5 \\ \xi_6 \end{bmatrix}$$

Roll Damping

MAESTRO-Wave can account for the effects of viscous roll damping in addition to the potential flow damping during the analysis. The damping is defined as:

$$B_{44}(w) + B_{44,v} \qquad (6.1)$$

where $B_{44,v}$ is the viscous damping calculated using the user-defined critical damping ratio, k:

$$k=(B_{44}(w_n)+B_{44,v})/(B_{44,critical}) \qquad (6.2)$$

The critical damping is defined as:

$$B_{44,critical} = 2.0*sqrt(C_{44}*(A_{44}(w_n)+I_{44})) \qquad (6.3)$$

Assuming that $B_{44}(w_n)$ is small compared to $B_{44,v}$ the resulting damping force can be found by:

$$B_{44}(w) + k*B_{44,critical}(w) \qquad (6.4)$$

12.2 User Manual

This section is intended to provide a general introduction and how-to guide for the MAESTRO-Wave inte

Model Requirements

General Tab

Model Requirements

The model requirements in order to run the MAESTRO-Wave analysis are:

- The full model must be modeled (half models must be mirrored)

- The loading condition (i.e., mass distribution) must be defined in a MAESTRO load case

- The forward direction must be in the +X direction for non-zero speed runs

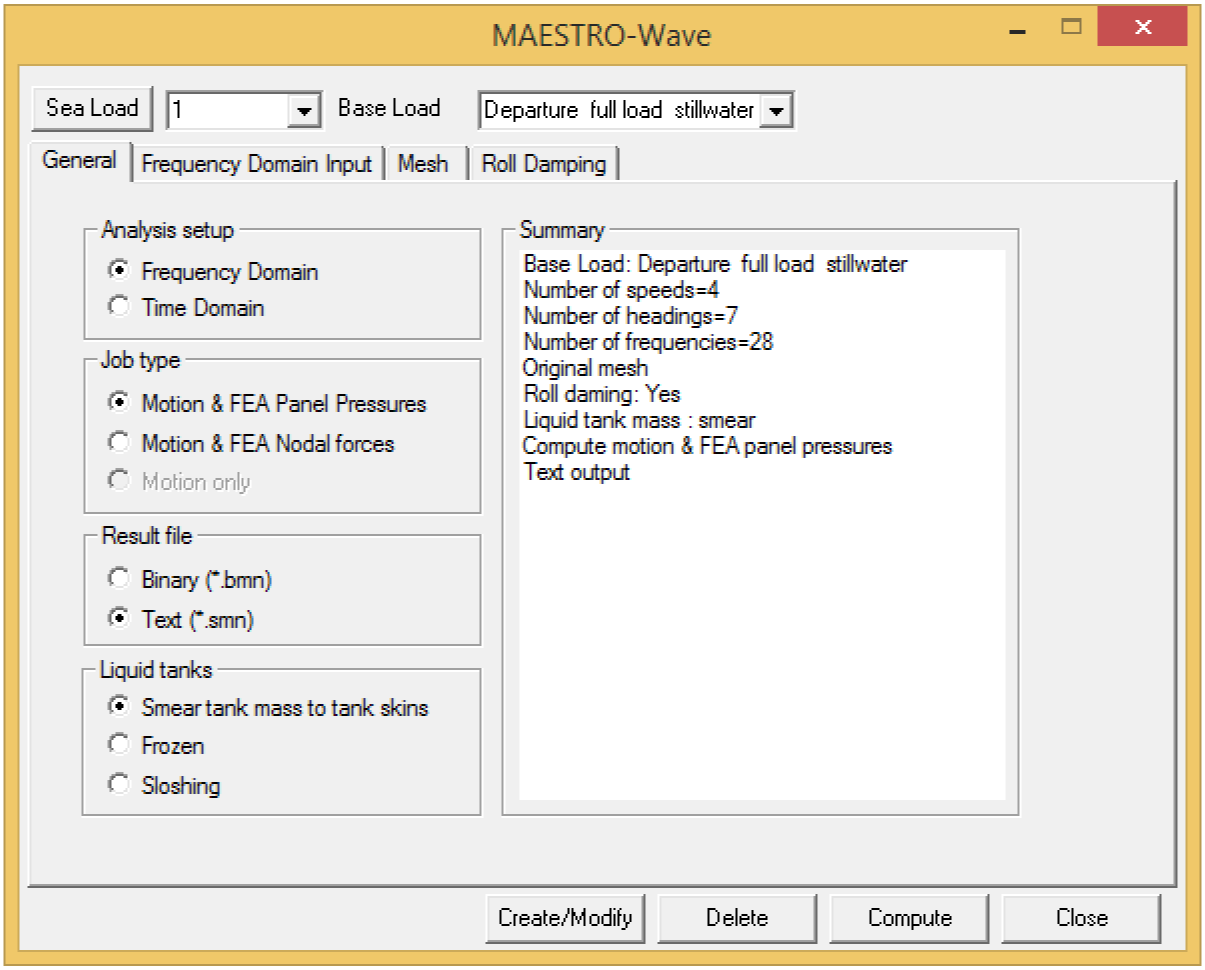
- "Wettable" elements must be defined in the model and an immersion definition must exist in the MAESTRO load case

MAESTRO-Wave can be launched from the Wave menu.

The MAESTRO-Wave dialog is split into 4 tabs, each of which will be discussed below. The first step is to create a new "Sea Load". The "Sea Load" is a unique load case used by MAESTRO-Wave, which is derived from a "Base Load" (i.e. MAESTRO load case). Similar to the loads dialog, the "Sea Load" can be created or modified using the "Create/Modify" button or deleted using the "Delete" button. Once the "Sea Load" case(s) is created, the "Compute" button will run the MAESTRO-Wave calculations. It is recommended to click the "Create/Modify" button after making changes on each tab to ensure the chosen options are saved. Once a "Sea Load" is created, it is saved with the model, similar to a MAESTRO load case.

General Tab

© 2014 Optimum Structural Design, Inc.



The "Analysis Setup" section allows the user to select whether to run the analysis as a frequency domain or time domain analysis.

The "Job Type" section provides the user with three options of what output to produce:

Motion & FEA Panel Pressures: Computes the motion and loads RAOs while saving FEA panel pressure data with minor point force corrections (e.g., for later use in buckling checks)

Motion & FEA Nodal Forces: Generates the motion and loads RAOs while saving FEA nodal force data (e.g., for later use in fatigue analysis)

Motion only: Not yet implemented.

The "Results File" section allows the user to save the MAESTRO-Wave data in either binary or ASCII format. The binary format results in a smaller, faster loading file that is accessed as needed when post-processing MAESTRO-Wave data. The ASCII format results will be entirely loaded into memory when imported, but the *.smn files can be edited by the user. It is recommended to use the binary format unless there is a specific need to access the ASCII version of the results.

The "Liquid Tanks" section provides the user with three options of how to handle the liquid tank loads in the model:

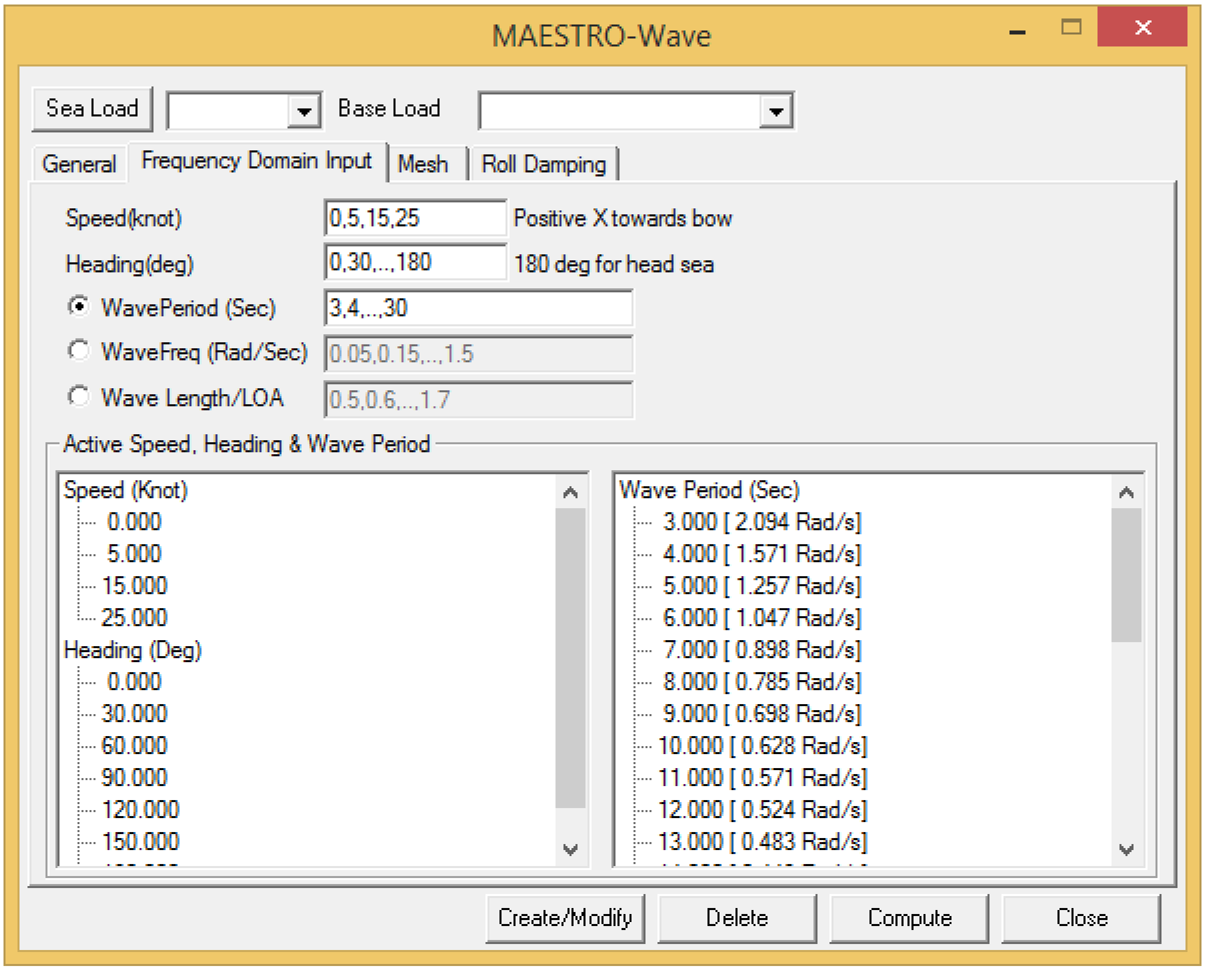
Smear tank mass to tank skins: This option will calculate the mass of the tanks for the base loading condition and smear it to the tank boundary shell elements.

Frozen: This option will use the same method as the *Sloshing* option but assume that the tanks are full so no free surface effects are included.

Sloshing: This option will solve the tank boundary value problem separately and add the loads to the equation of motion.

The "Summary" section lists the current settings from all tabs for the current Sea Load.

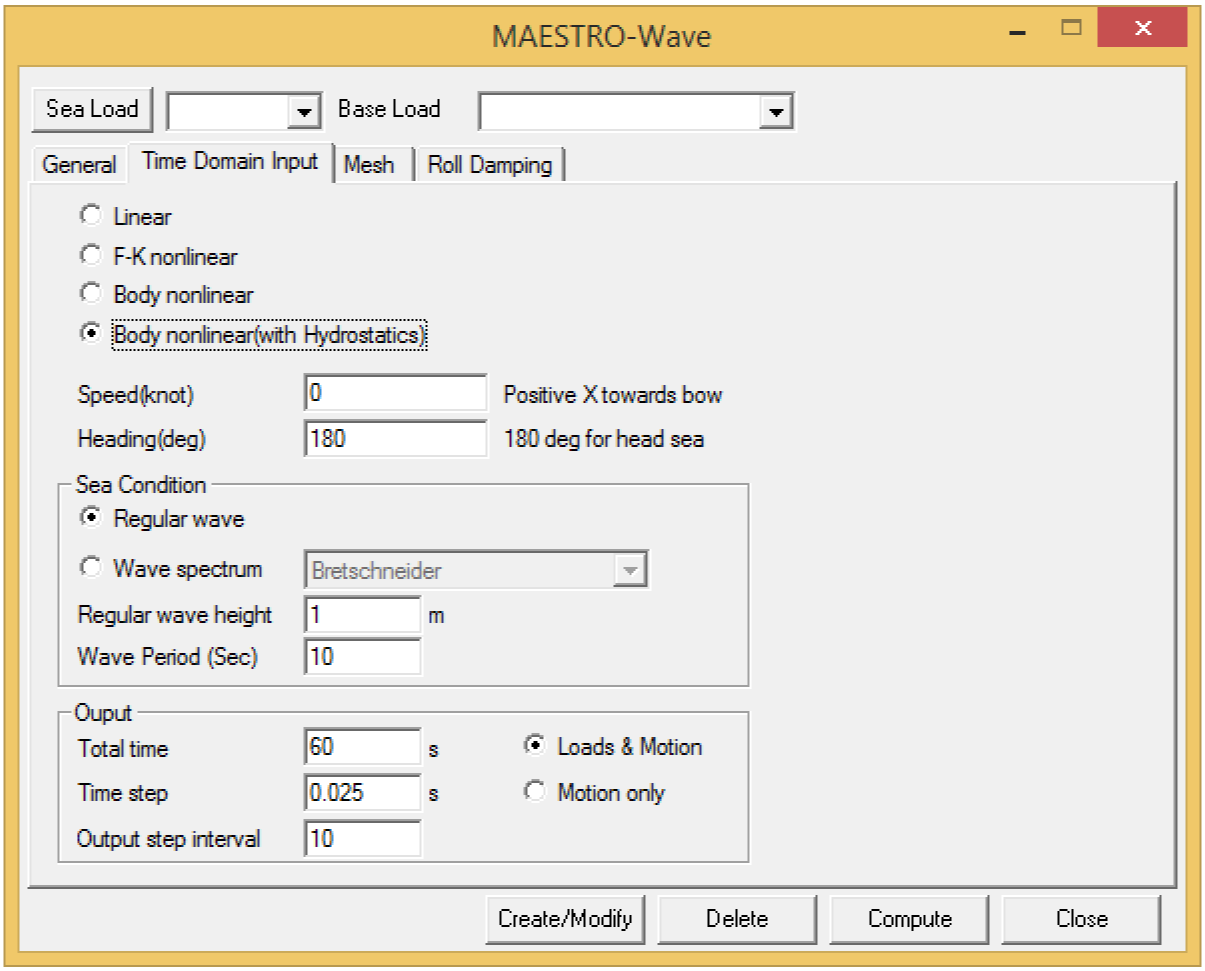
Frequency-Domain Input



The frequency domain input tab defines the conditions to be run during the analysis for the specific "Sea Load" case. The values for speed, heading and wave period can be entered as a comma separated list or an an ellipse. For example, "0,5,...,25" in the speed entry would be interpreted as 0, 5, 10, 15, 20, 25. The units for speed and heading are in knots and degrees, respectively and the wave period can be entered in one of three ways: wave period in seconds, wave frequency in radians per second or the ratio of wave length over length overall of the vessel. Clicking the radio button next to the input option desired will allow the input to be changed. A summary of the conditions for the selected "Sea Load" case are displayed at the bottom of the page. The total number of conditions that will be run is equal to:

conditions = # speeds x # headings x # wave periods

Time-Domain Input



Linear: The velocity potential and Froude-Krylov forces are evaluated on the hull mean wetted surface.

F-K nonlinear: The velocity potential is evaluated on the hull mean wetted surface and the Froude-Krylov forces are evaluated on the instantaneous hull wetted surface up to the incident wave.

Body nonlinear: The velocity potential and the Froude-Krylov forces are evaluated on the instantaneous hull wetted surface up to the incident wave.

F-K nonlinear (with Hydrostatics): The velocity potential is evaluated on the hull mean wetted surface and the Froude-Krylov forces are evaluated on the instantaneous hull wetted surface up to the incident wave and combined with linear hydrostatic forces.

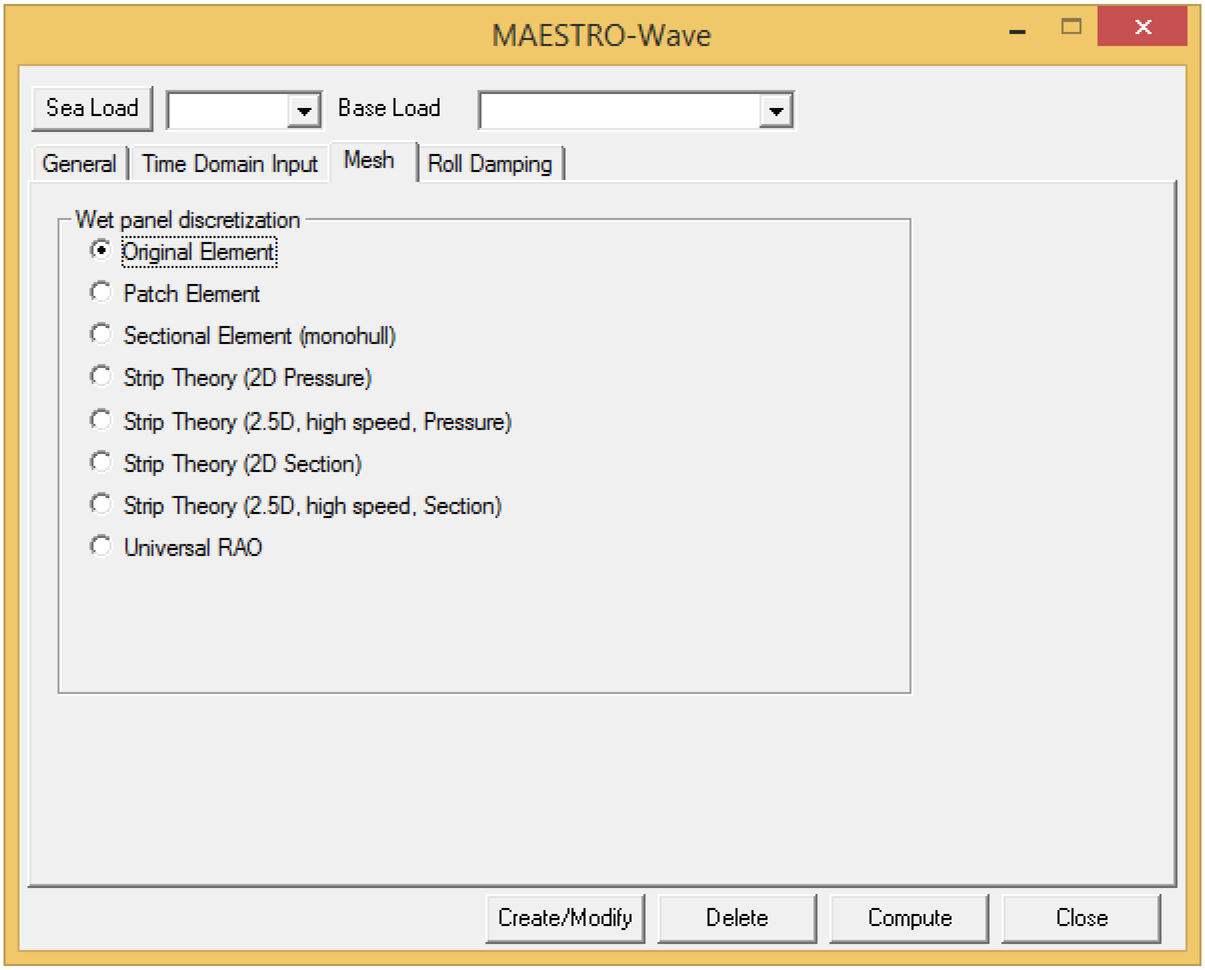
The time domain input tab allows the user to specify one condition to run.

The user specifies the speed, heading, and sea condition for the run. The sea condition is defined by a regular wave with a wave height and wave period, or by a wave spectrum defined by a significant wave height and peak period.

The output section allows the user to specify the length, time step, and output interval for the simulation run as well as whether to calculate the motion only or the motion and the loads.

Mesh Tab

© 2014 Optimum Structural Design, Inc.



The Mesh tab allows the user to select how the wetted surface is discretized for the analysis.

Original Element: This option will use the MAESTRO element mesh in the analysis. This is a higher fidelity analysis, but can be slow. It is recommended to use this option when the number of wetted panels is less than 3,000.

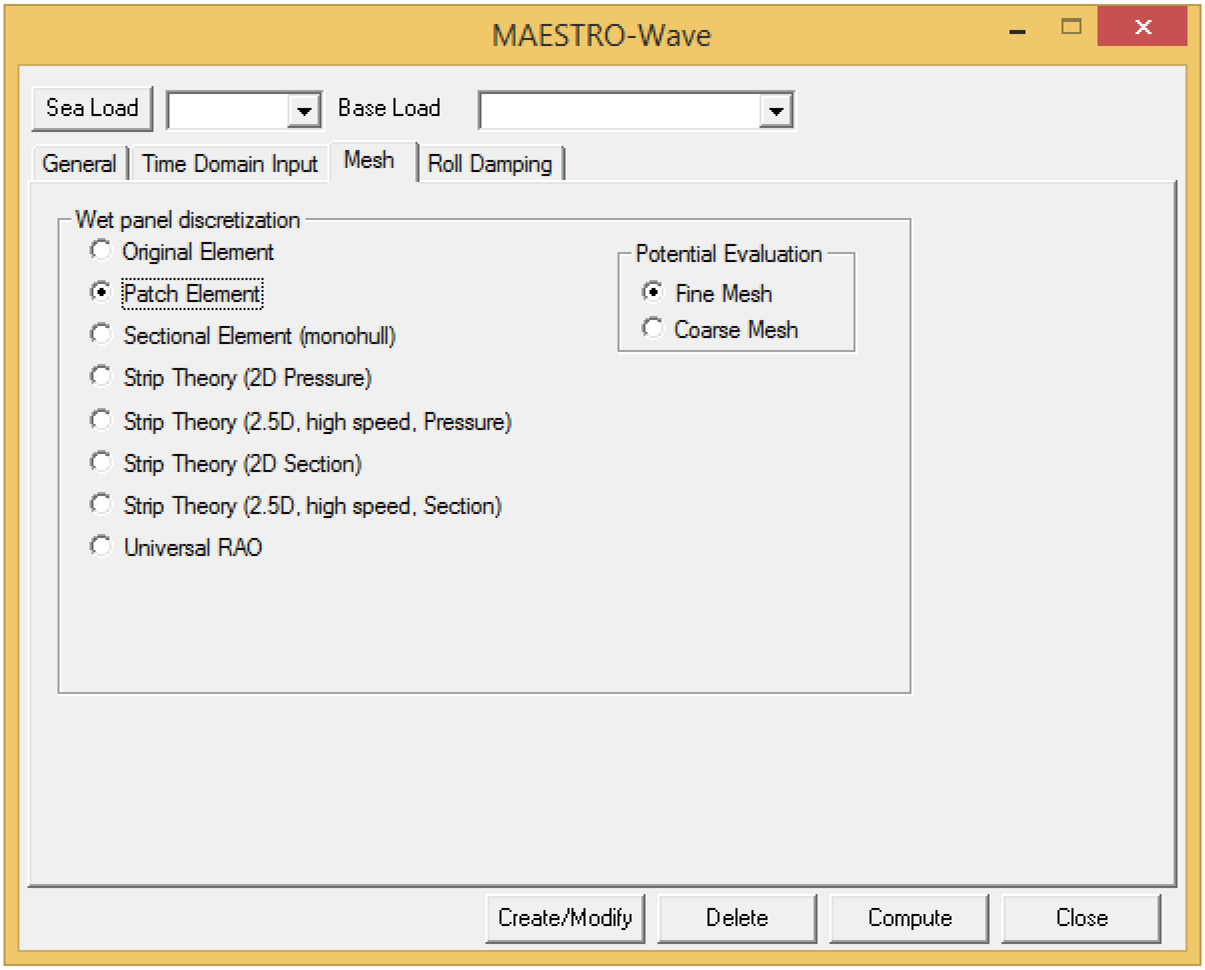
Patch Element: This option will use the MAESTRO evaluation patch elements in the analysis and will be faster than the "Original Element" option. Source strengths will be computed based on the evaluation patches and mapped to the FEA elements. An additional input section will appear where the user can select how the velocity potentials (and associated pressures due to radiation and diffraction) will be evaluated:

Fine Mesh: Each FEA element will have its own velocity potential computed.

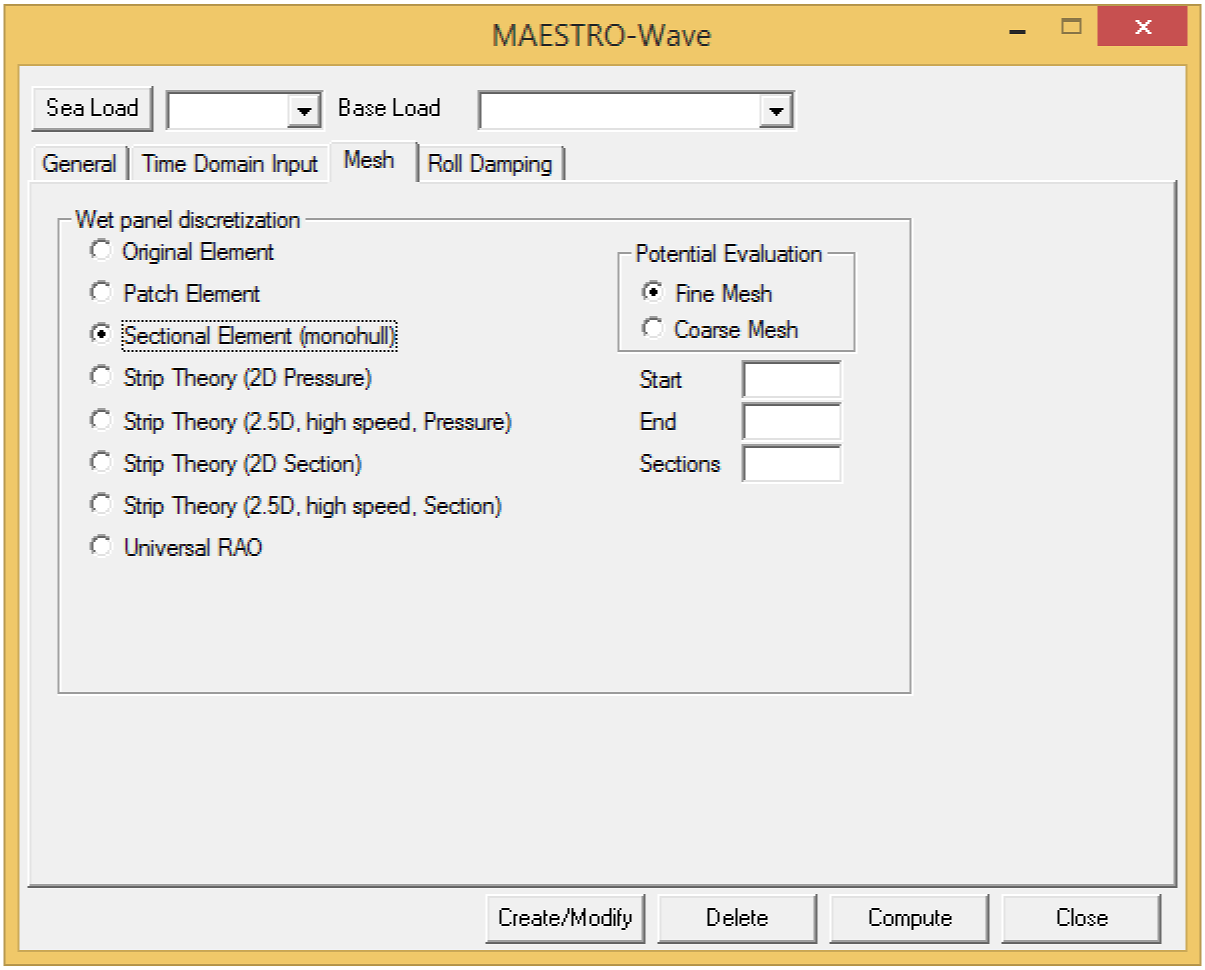
Coarse Mesh: All FEA elements within the patch will use the same value for velocity potential.

© 2014 Optimum Structural Design, Inc.

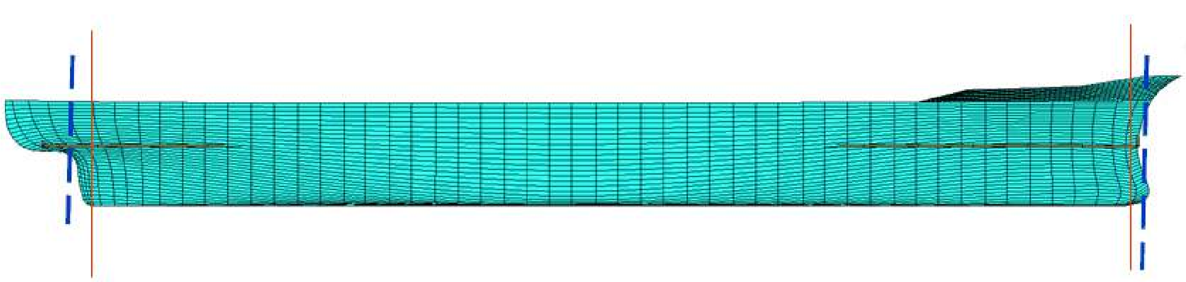
Regardless of which Potential Evaluation method is chosen, the hydrostatic and Froude-Krylov pressures will be computed for each FEA element individually.

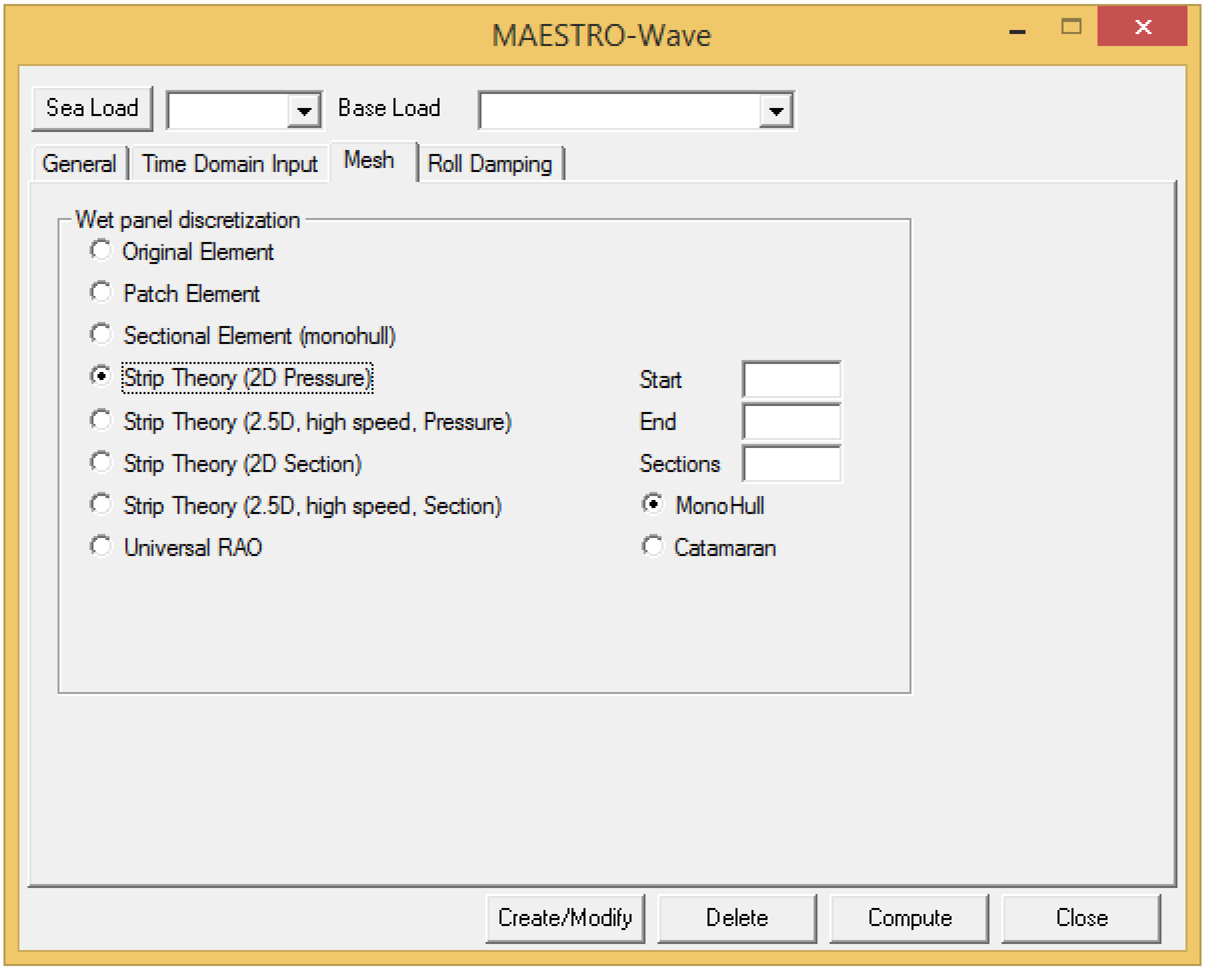


Sectional Element: This option will open additional input sections in the dialog allowing the user to select how the velocity potentials will be evaluated and to specify the section locations to be used in the calculation. The velocity potential evaluation options are the same as described in the Patch Element section above, but in this case, the MAESTRO evaluation patch elements are replaced by internally created patches that span between each defined section. The width of the patches is dependant on the number of points used internally to define each section. The section locations are defined by the user input Start and End locations and the number of sections to be cut in between them.



The Start and End locations should be located so as to cut sections so that the 2D section cut is a single curve (e.g., do not make a section cut in a bulbous bow where there would be two curves defining the section). The example below shows where to cut the bow of the model (red line) and where not to cut the model (blue line). For the stern cut, this example shows a recommended location to start the sections because if the spacing is wide, the interpolation of the section area between cuts could result in unwanted area. The start location should be at the aft end of the ship.





The four strip theory methods also require the same section input described above and also allow the user to specify whether the vessel is a monohull or catamaran.

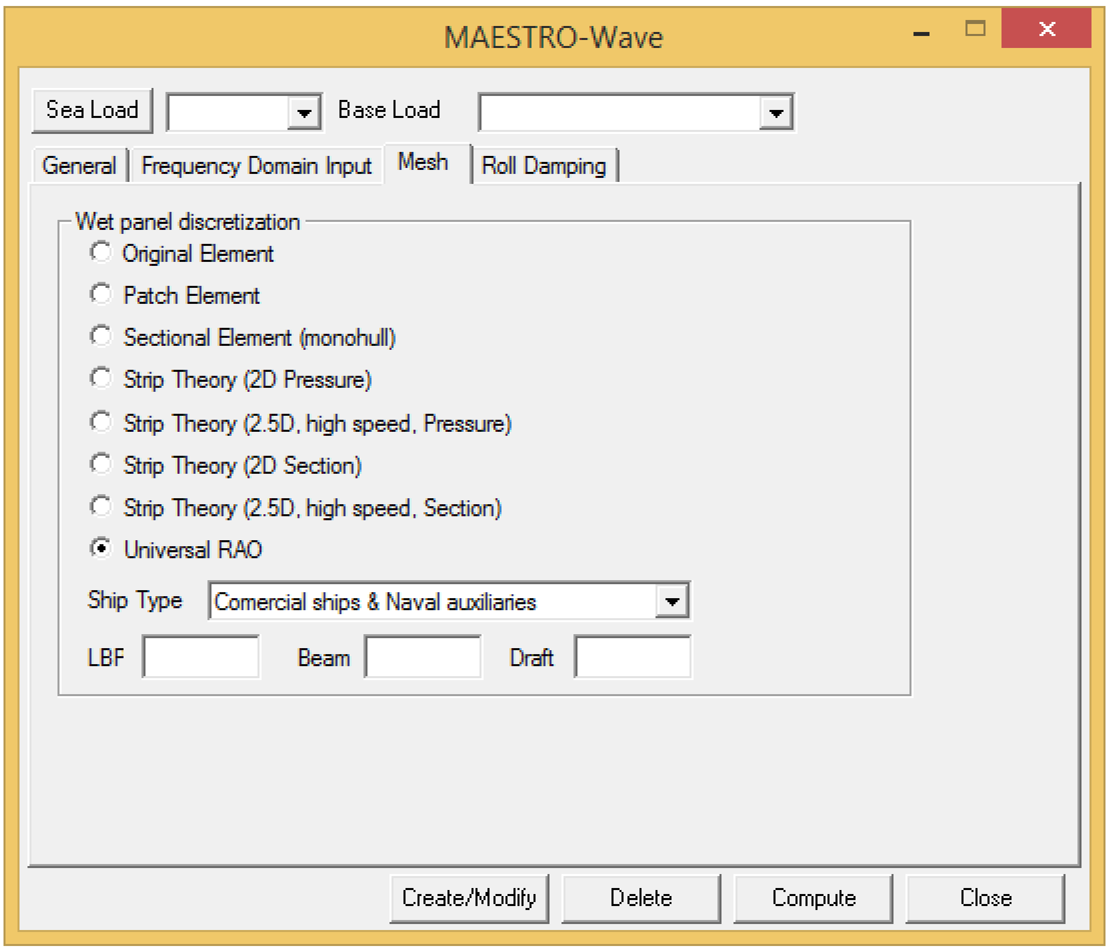
Strip Theory (2D Pressure): This option uses a zero-speed Green's Function strip theory approach and will compute the resulting pressures on the elements for use in post-processing evaluations.

Strip Theory (2.5D, high speed, Pressure): This option uses a Rankine Source method with a forward speed correction term in the free surface computation. This method is recommended when running cases with a Froude number greater than 0.4. This option includes the panel pressure calculations.

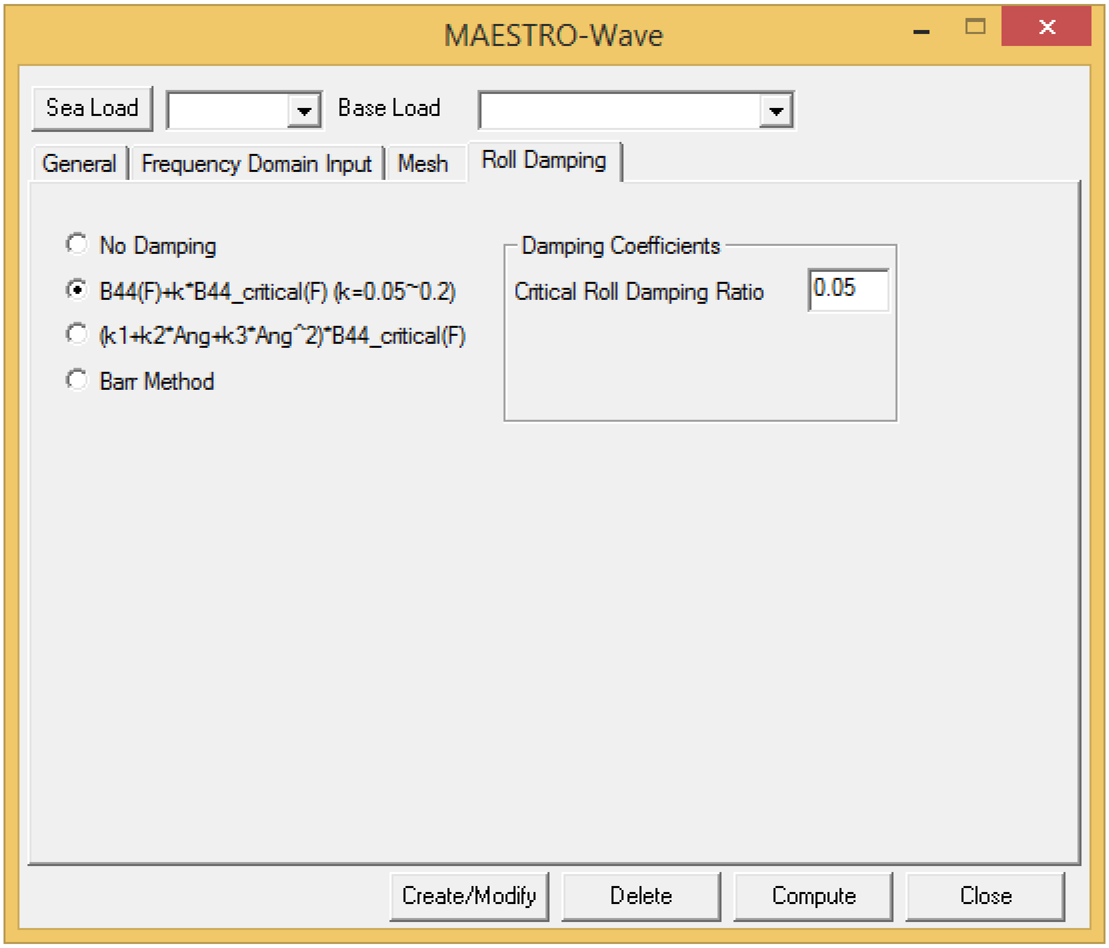
Strip Theory (2D Section): This option uses a zero-speed Green's Function strip theory approach, but does not compute the element pressure information. It is faster than the 2D Pressure method, but only the hull girder and motion RAOs will be available for review and post-processing. (i.e., the user can post process the hull girder RAOs to select a design waves, but would not be able to create the resulting load cases without the pressure information.)

Strip Theory (2.5D, high speed, Section): This option uses a Rankine Source method with a forward speed correction term in the free surface computation. This method is recommended when running cases with a Froude number greater than 0.4. It is faster than the 2.5D Pressure method, but only the hull girder and motion RAOs will be available for review and post-processing. (i.e., the user can post process the hull girder RAOs to select a design waves, but would not be able to create the resulting load cases without the pressure information.)

Universal RAO: This method uses algorithms to generate RAOs for various vessel types based on the length, beam and draft entered by the user.



Roll Damping Tab



The Roll Damping tab allows for roll damping effects to be accounted for in the analysis. Depending on the method that is chosen, additional inputs are required to define the method.

$B_{44}(W) + k*B_{44,critical}(W)$: This method accounts for the potential flow and viscous damping effects. The critical roll damping ratio, k, is the ratio of the actual (potential plus viscous) damping to the critical damping and the critical damping is defined as:

$$\mathbf{B_{44,critical}= 2.0*sqrt(C_{44}*(A_{44}(W_n)+I_{44}))}$$

$(k_1+k_2*Ang+k_3*Ang)*(B_{44,critical}(W))$: This method allows the user to specify constants k1, k2, and k3 to apply a second order, roll amplitude dependent scale factor on the critical damping term.

Barr Method: This method calculates the roll damping based on the bilge keel definition, ship characteristics, and forward speed:

$$B_{CR} = 2\sqrt{(M_{44} + m_{44})C_{44}}$$

© 2014 Optimum Structural Design, Inc.

$$B_{44} = \frac{k \cdot B_{CR}}{2}$$

Where:

$$k = \begin{cases} k = k_1 \cdot \sqrt{\theta} + k_2 \cdot f_1 & \frac{F_r}{C_b} \le 0.4 \\ k = k_1 \sqrt{\theta} \cdot f_2 & \frac{F_r}{C_b} > 0.4 \end{cases}$$

$$k_1 = 19.432 \cdot (A_{BK} \cdot \sqrt{B_{BK}} + 0.0024 \cdot L_{pp} \cdot B \cdot \sqrt{Dis_{BK}}) \cdot (Dis_{BK})^{2.5} / VOL / B$$

$$k_2 = 0.00085 \cdot \frac{L_{pp}}{B} \sqrt{\frac{L_{pp}}{GM^{0.5}}}$$

$$f_1 = \frac{Fr}{C_b} \cdot [1 + \frac{Fr}{C_b} + 2 \cdot (\frac{Fr}{C_b})^2]$$

$$f_2 = 1 + 0.112 \cdot F_r + 0.409 \cdot F_r^{\ 2} + 1.42 \cdot F_r^{\ 3} - 0.754 \cdot Fr^4$$

$$F_r = \frac{U}{\sqrt{gL_{pp}}}$$

U -- Ship Speed

A_{BK} -- Area of bilge keel

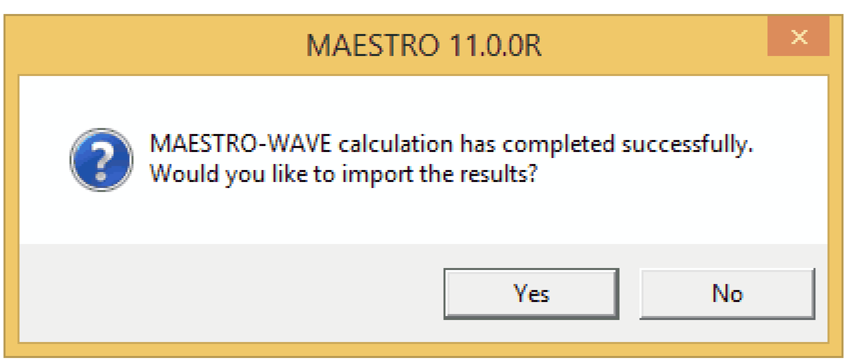
B_{BK} -- Breadth of bilge keel

Dis_{BK} -- Distance from bilge keel to centerline of waterplane

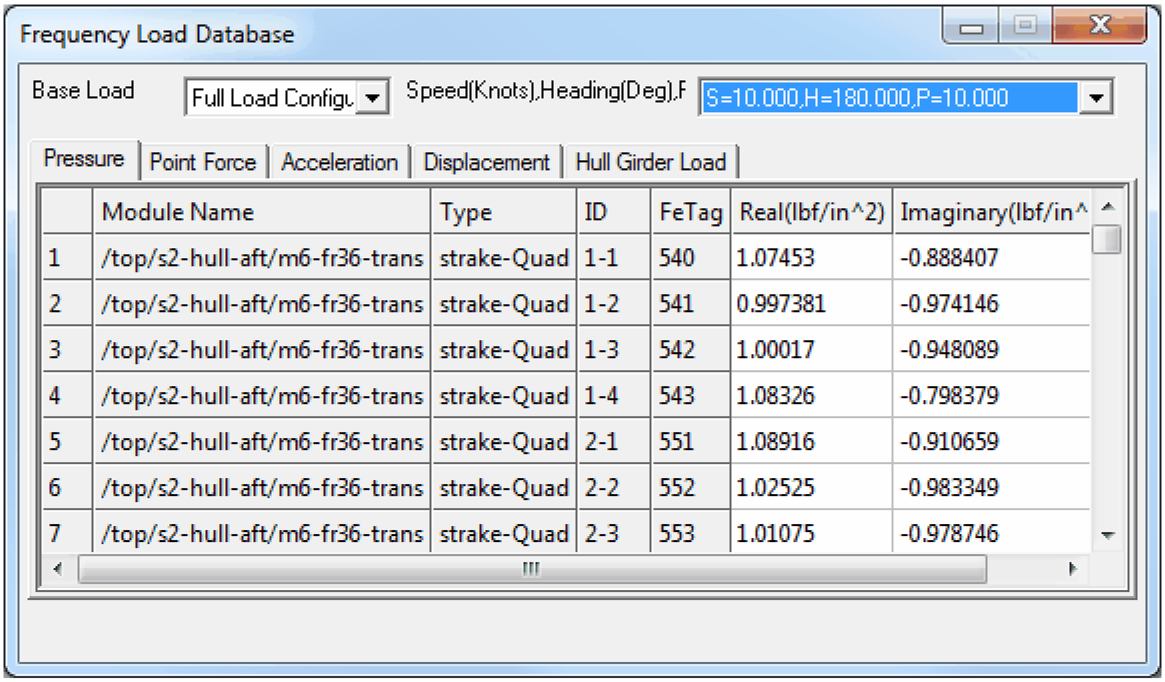
VOL -- Ship Volume $= L_{pp} B d C_b$

Results

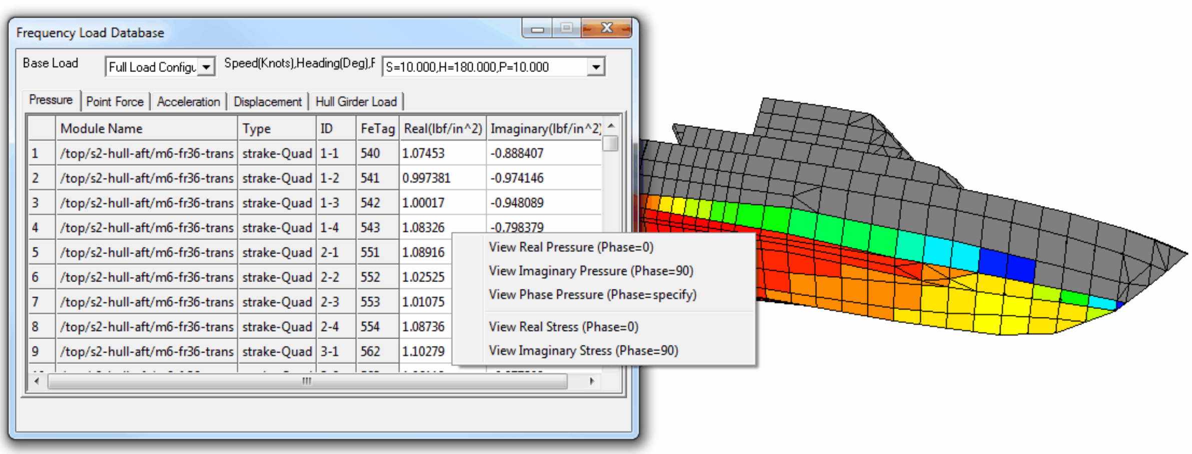
Once the MAESTRO-Wave analysis has been run, a dialog will open asking to import the results. Click "Yes" to have the Frequency Load Database automatically populated with the results.



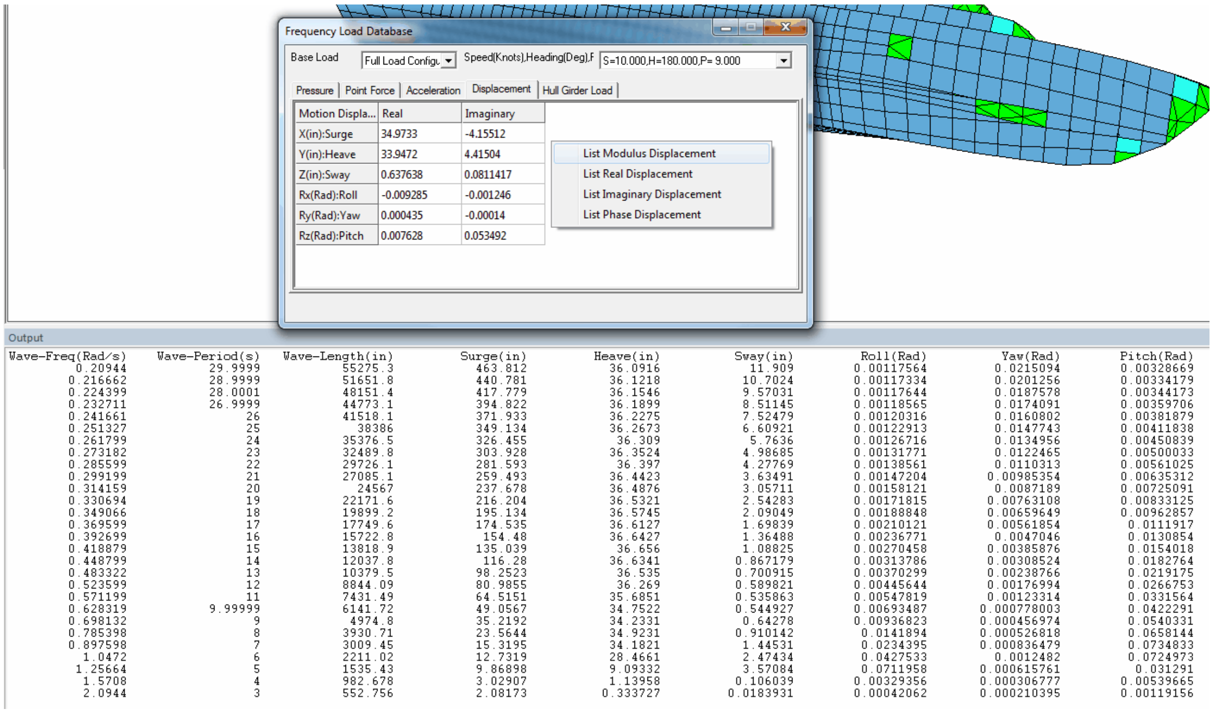
Once the results are imported, the hydrodynamic pressures can be viewed graphically by opening the Frequency Load Database dialog from the Wave > Regular Wave Database menu option.



This dialog allows you to select the base load that was run and the speed, heading and wave period condition of interest. The first tab provides the list of elements and their calculated pressures. Right-clicking in this dialog allows the calculated pressures or stresses to be plotted on the model elements.



The Point Force tab provides the resultant nodal forces at each of the FE nodes in the model. The Acceleration and Displacement tabs provide the displacement and acceleration RAOs for the selected load case and condition. Right clicking in the dialog will output the accelerations, velocities, or displacements for all frequencies run at the selected speed and heading to the Output tab. These results can be viewed in MAESTRO (Viewing RAOs and Statistics) or easily exported to a spreadsheet tool such as Microsoft Excel to plot the RAOs.



12.3 Validation

The S-175 container ship is well known because it was used by the ITTC (2010) to carry out a comparative numerical study of linear wave-induced motions and structural loads. The database that resulted from that study includes numerical results from many institutions, and also some experimental data. Table 1 lists the main particulars of this ship.

Length between perpendiculars	175 m
Breadth	25.4 m
Depth	15.4 m
Draught	9.5 m
Displacement	24742 ton
LCG aft of midship	2.5 m
Block coefficient	0.572
Midship section coefficient	0.98
Total mass	23972251 kg

XG(from AP)	84.97 m
YG(from centerline)	0 m
R_{xx}	9.652
R_{yy}	42.07
R_{zz}	43.17

Table 1: Main particulars of the S175 container ship

Three hydrodynamic analyses were conducted for this comparison. For the first analysis, which is labeled as "MAESTRO-Wave" in the results figures, the hydrodynamic mesh is the same as the finite element mesh, as shown in Figure 1(a). For the second analysis, labeled as "MAESTRO-Wave (Coarse Mesh)," the hydrodynamic mesh is coarser than the finite element mesh, as shown in Figure 1(b). The first two models were analyzed using MAESTRO-Wave. The third model, as shown in Figure 1(c), was analyzed using PRECAL6.6 (2010).

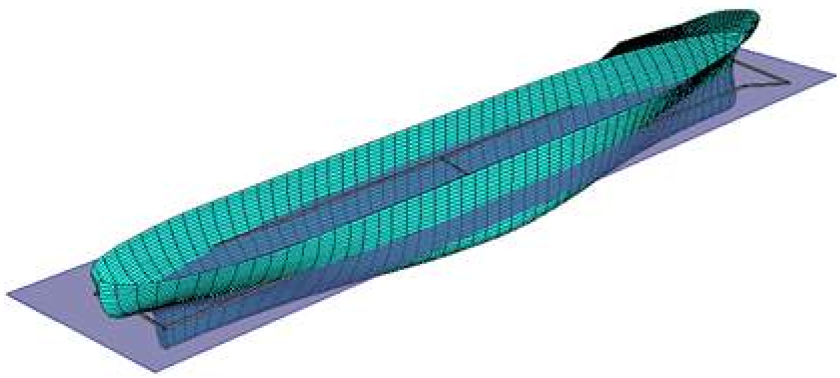


Fig. 1(a) MAESTRO-Wave Seakeeping model

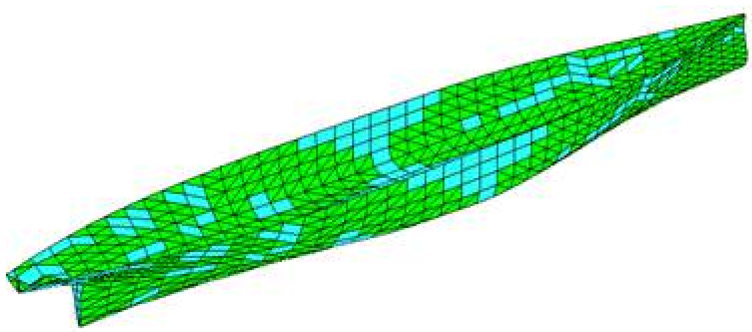


Fig. 1(b) MAESTRO-Wave (Coarse Mesh) Seakeeping model

© 2014 Optimum Structural Design, Inc.

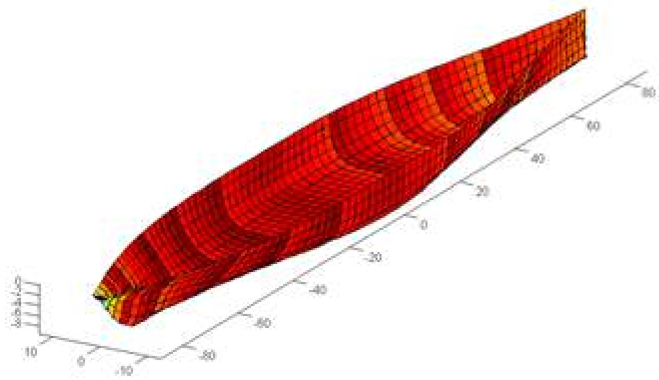


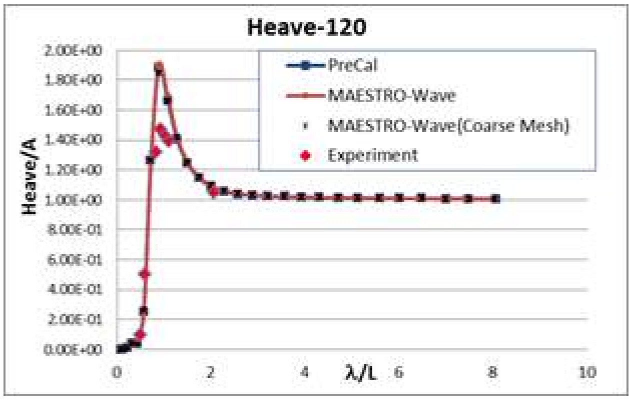
Fig. 1(c) PRECAL Seakeeping model

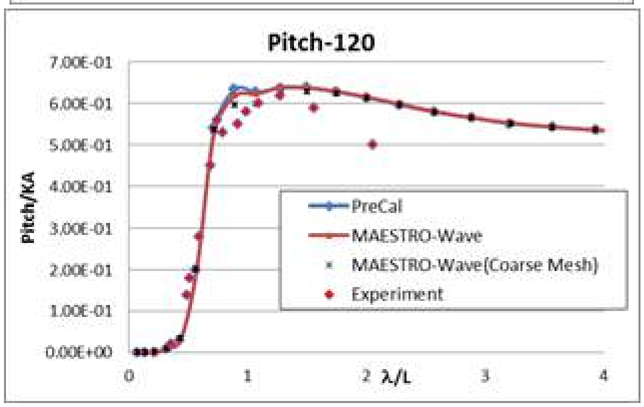
The ship is advancing with constant speed *v*=22.145 knots with 4 different headings, β=0°, β=90°, β =120° and β=180°. All results are presented in a non-dimensional way using wave amplitude (A), wave number (k), encounter frequency (W), water density (r), gravitational acceleration (g), ship beam(B) and ship length between perpendiculars (L_{pp}) as given in Table 2:

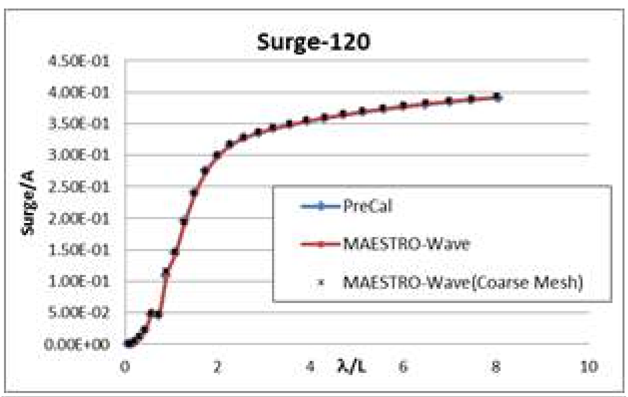
Translational motions (heave, sway and surge)	$x' = x\frac{1}{A}$
Rotational motions (roll, pitch and yaw)	$\theta' = \theta\frac{1}{kA}$
Bending moments	$M' = M\frac{1}{\rho g B L_{pp}^2 A}$
Shear forces	$F' = F\frac{1}{\rho g B L_{pp} A}$

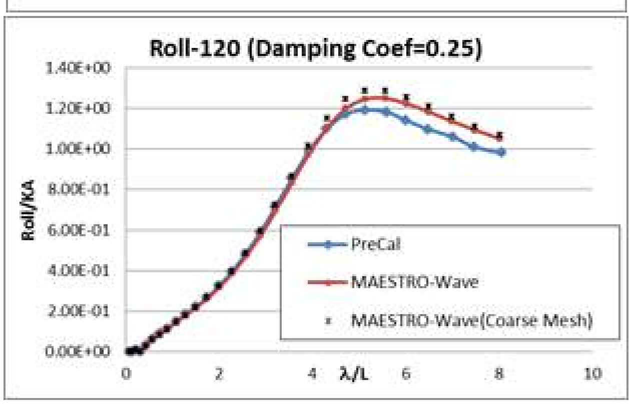
Table 2: Non-dimensional parameter definition

The motion and load RAO results, along with the available experimental data from ITTC, are shown below in Figures 2 to 6.


Heave-120
Heave/A
λ/L
PreCal
MAESTRO-Wave
MAESTRO-Wave(Coarse Mesh)
Experiment


Pitch-120
Pitch/KA
λ/L
PreCal
MAESTRO-Wave
MAESTRO-Wave(Coarse Mesh)
Experiment


Surge-120
Surge/A
λ/L
PreCal
MAESTRO-Wave
MAESTRO-Wave(Coarse Mesh)


Roll-120 (Damping Coef=0.25)
Roll/KA
λ/L
PreCal
MAESTRO-Wave
MAESTRO-Wave(Coarse Mesh)

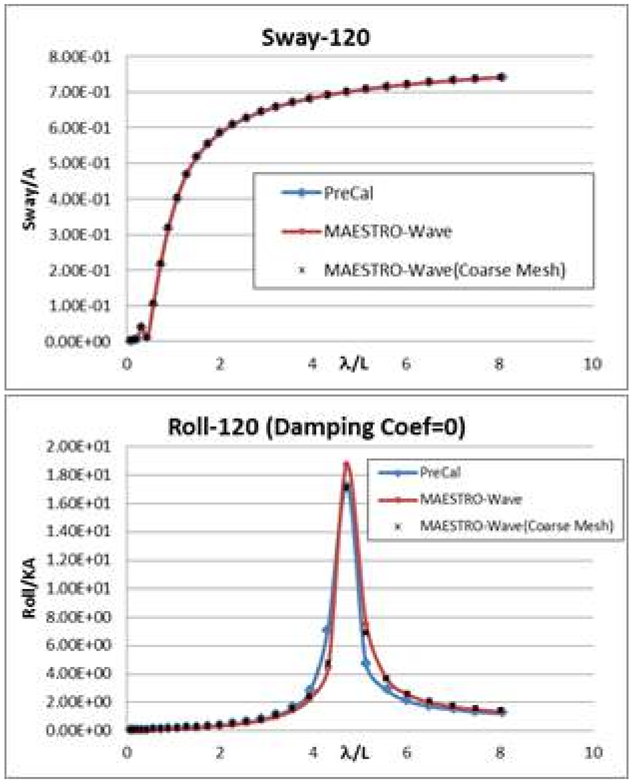
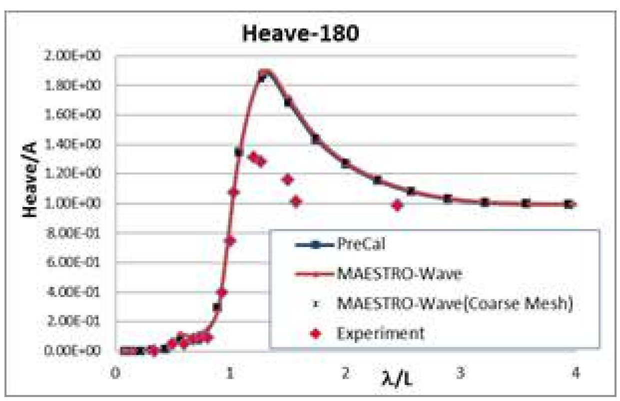
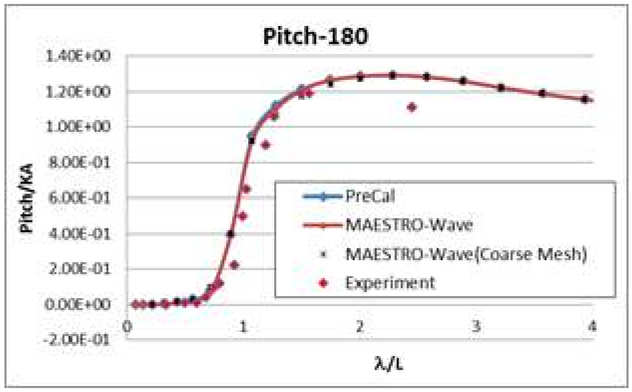


Fig. 2 Motion RAO (β=120° , Fr=0.275)





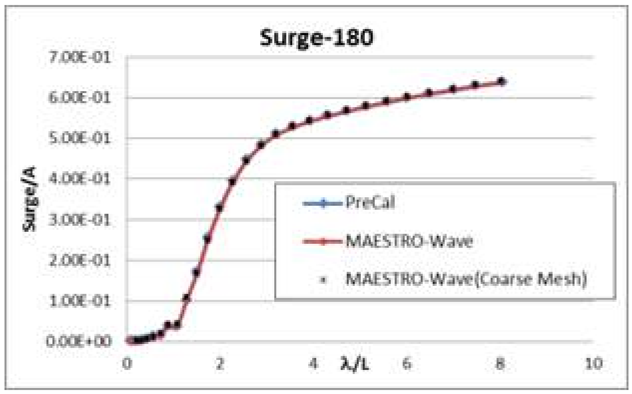
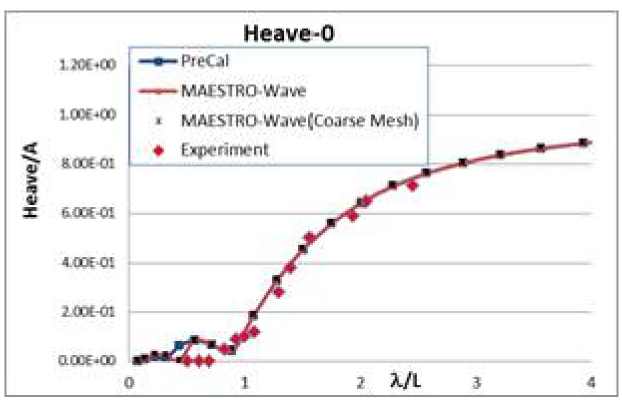


Fig.3 Motion RAO (β=180° , Fr=0.275)



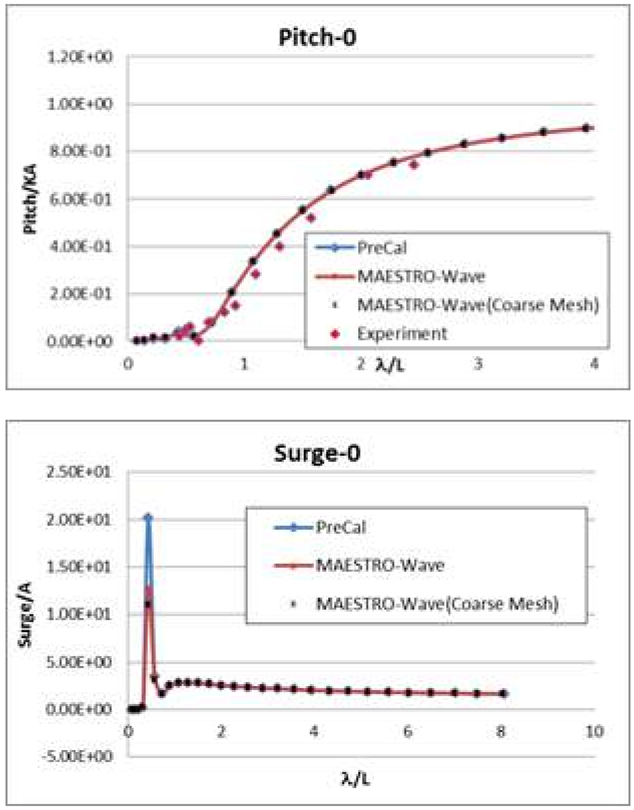
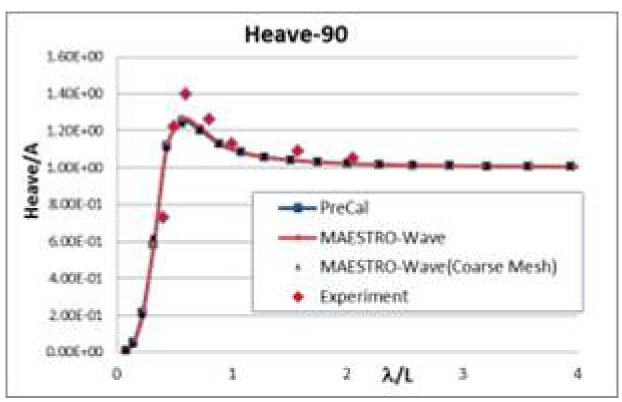


Fig. 4 Motion RAO (β=0° , Fr=0.275)



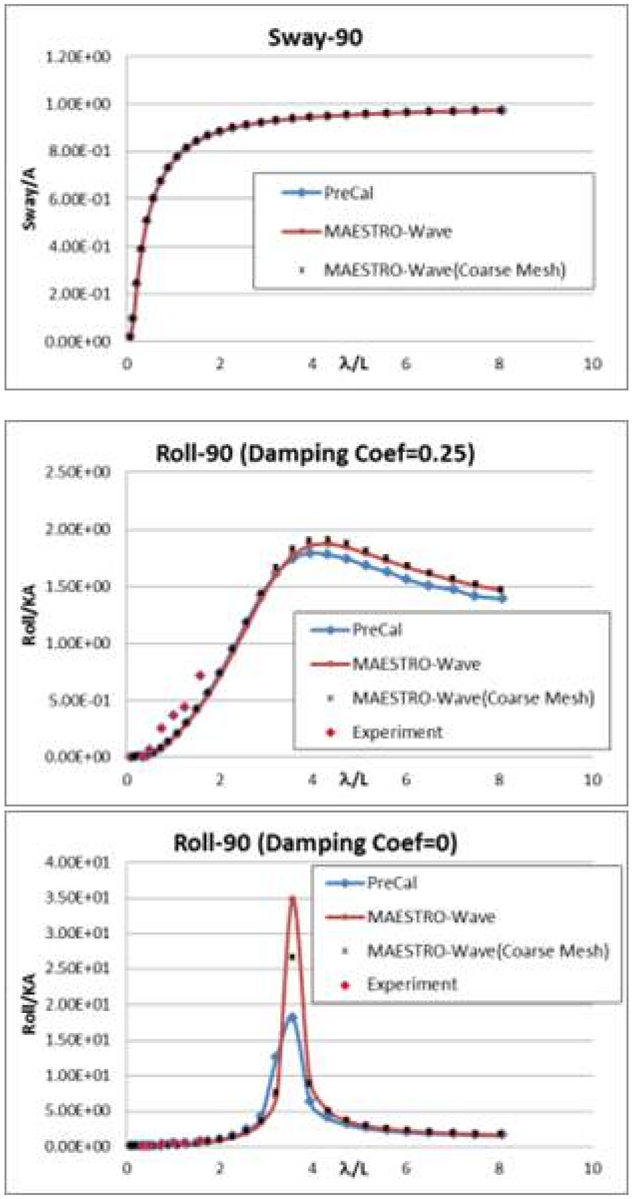
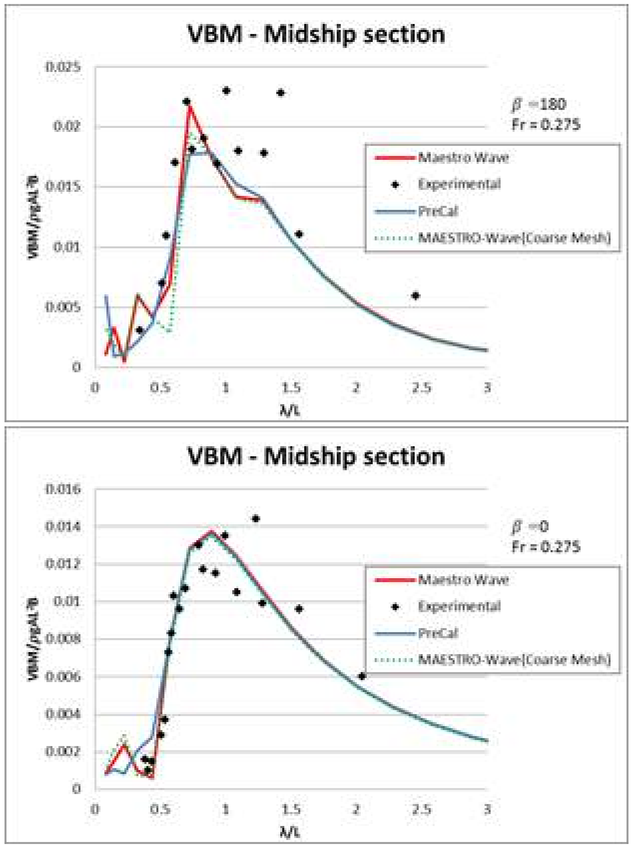


Fig. 5 Motion RAO (β=90° , Fr=0.275)



β=*180*° β=*0*°

Fig. 6 Vertical bending moment RAO (Fr=0.275)

Extreme Load Analysis

13

13 Extreme Load Analysis

The Extreme Load Analysis (ELA) module imports unit wave load data from a hydrodynamic analysis (MAESTRO-Wave or other) via *.smn file format. It then allows the user to calculate hull girder load response RAOs, and provides the necessary short-term and long-term statistical computations to predict extreme values of the maximum loads for a given vessel. This includes the ability to define or import wave scatter diagrams, operational profiles, and wave spectra as well as compute hull girder RAOs for user-defined dominant load parameters (e.g., vertical bending moment). Finally, extreme equivalent regular waves (equivalent design waves) are internally computed and selected for assessment of extreme global loads. The user has a variety of options to add still water loads to the wave-induced wave loads and re-balance these components.

Table of Contents

13.1 Introduction

The structural design portion of many classification society's *guides* and *rules* are intended to provide the basis for a preliminary step-by-step design of a hull structure. Often however, due to the novel nature of a design or additional *class notation* requirements set by the owner, it is the task of the designer to conduct a direct structural analysis. Central to this direct analysis, is the use of a computational tool based on linear seakeeping theory for calculating the load and motion parameters that best characterize the extreme ship response behavior for critical loading conditions (e.g., Full Load). Once these extreme ship responses are identified, the structural integrity (i.e., global and local structure) of the vessel can be assessed.

© 2014 Optimum Structural Design, Inc.

13.2 Procedure Using MAESTRO

As shown in the figure below, MAESTRO can be leveraged to execute a direct analysis process where both the global and local design is assessed for structural adequacy. The following steps outline the primary steps in the process:

1. Generate MAESTRO Global Model (Structure and Base Loading Conditions)

Option 1:

2. Run MAESTRO-Wave

Option 2:

2a. Export MAESTRO Mass Distribution and *Wetted* Elements Data (See the Exporting Models and Data section)

2b. Generate Hydrodynamic Model and Perform Hydrodynamic Analysis in 3rd party tool

2c. Apply Hydrodynamic Loads to MAESTRO Model (Ship Motions File *.smn)

2d. Compute RAOs in MAESTRO (inertia relief)

3. View RAOs

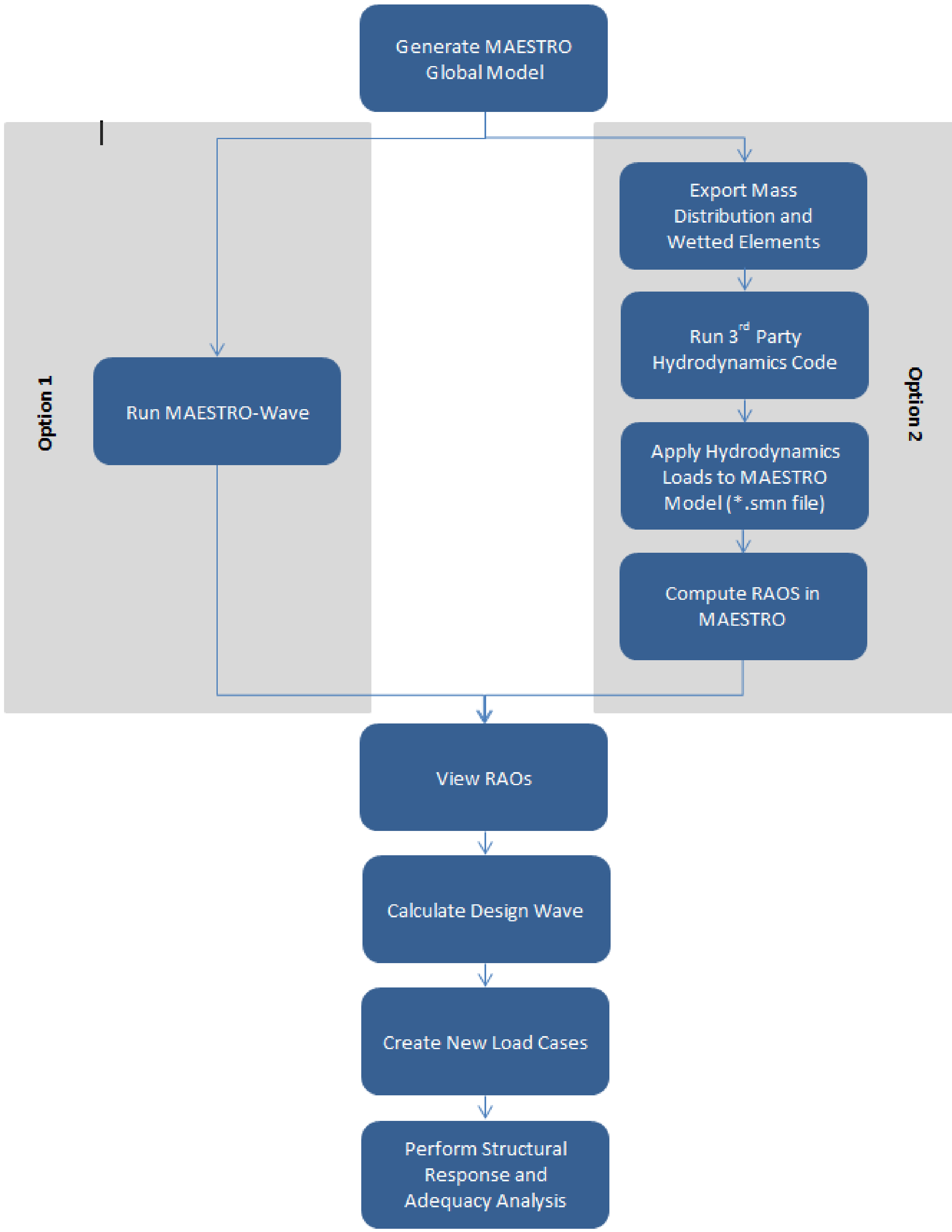
4. Calculate Equivalent Extreme Design Wave (Wave Spectra, Wave Scatter Diagram, Operating Profiles)

5. Combine Hydrostatic and Hydrodynamic Loads into New Load Cases

6. Perform Structural Response Analysis (Global and Local)

7. Assess Structural Adequacy (Global and Local)

© 2014 Optimum Structural Design, Inc.

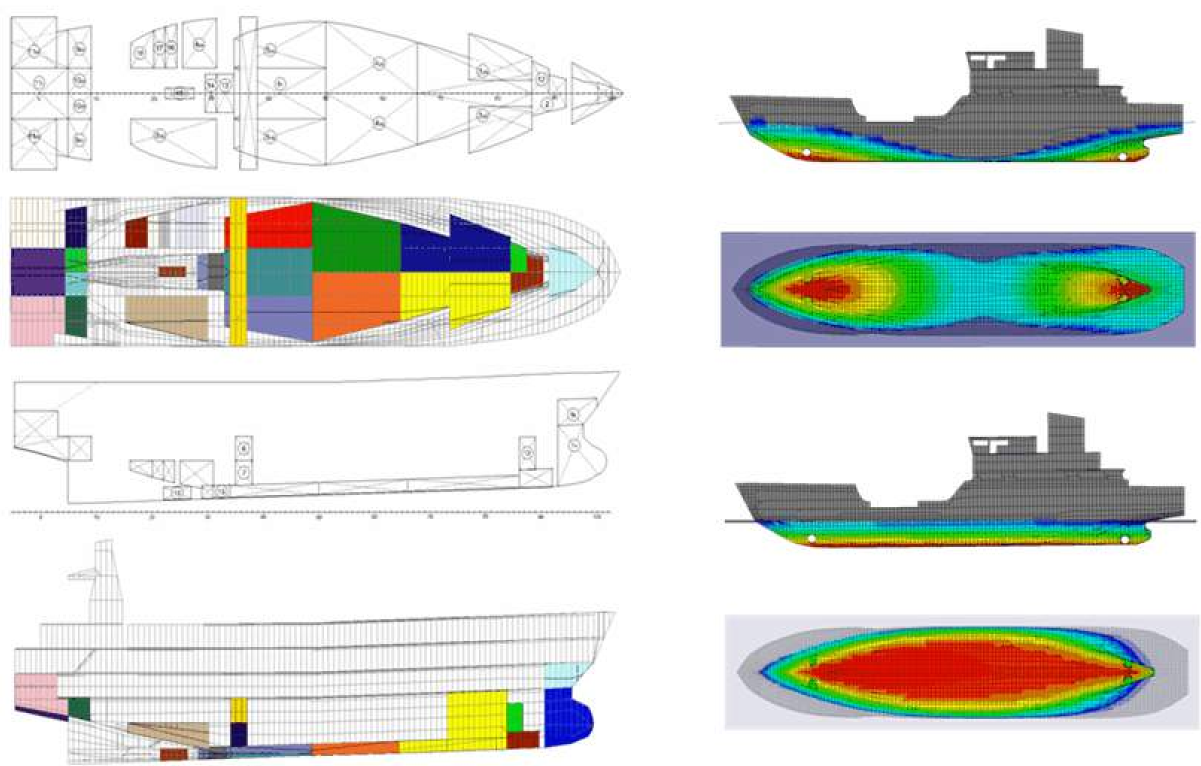

Generate MAESTRO Global Model
Option 1
Run MAESTRO-Wave
Export Mass Distribution and Wetted Elements
Run 3rd Party Hydrodynamics Code
Apply Hydrodynamics Loads to MAESTRO Model (*.smn file)
Compute RAOS in MAESTRO
Option 2
View RAOs
Calculate Design Wave
Create New Load Cases
Perform Structural Response and Adequacy Analysis

13.3 Structural Model and Loads

To meet the analysis needs and avoid inaccuracies and uncertainties, it is important that the MAESTRO model is sufficiently comprehensive. This means that the overall model should encompass all hull structure and ideally superstructure (i.e. when the superstructure participates in the global response). Further, there must be sufficient compatibility (e.g. mesh size, mass distribution, CGs, etc.) between the hydrodynamic and structural model so that the mapping of computed hydrodynamic pressures onto the MAESTRO model can be done appropriately.

The results of the Global Design Analysis are to be used to identify *hot spots* so that Local Design Analysis (i.e. mesh refinement) can be undertaken.

All of the static loading components necessary to accurately develop vessel loading conditions (i.e. Base Loading Conditions) can be created using MAESTRO. These loading components include, but are not limited to: lightship distribution (*Scaled Modules, Self-Weight, Mass Distribution*), tank loads, hydrostatic loads, and concentrated mass. See Loading the Model for complete details. These load components are typically combined in a MAESTRO load case and represent a base load case (e.g. Full Load, Minimum Operations, and Arrival). The figure below shows an example of MAESTRO's tank and hydrostatic loading.



© 2014 Optimum Structural Design, Inc.

13.4 Export Mass Distribution and Wetted Elements

The next major step in this type of direct analysis is to conduct a seakeeping analysis to determine the motions and loads resulting from operations of the vessel.

If using MAESTRO-Wave to perform the seakeeping analysis, go here first and then jump to the Viewing RAOs section, otherwise follow the steps below:

After computing the motions and loads, it will be the task of the analyst to appropriately map these loads to the MAESTRO model. To facilitate both of these steps, MAESTRO provides the ability to export key data: overall mass distribution and *Wetted* Element definition or hull geometry.

The following section is organized as follows:

Export MAESTRO Mass Distribution for VERES or PRECAL

Export MAESTRO Wetted Elements and Mass Distribution

[****]

Export MAESTRO Mass Distribution and Geometry for VERES and PRECAL

For a particular base load case, MAESTRO provides the ability to export geometry and mass distribution files specifically intended to be consumed by the seakeeping analysis tool VERES (VEssel RESponse program), which is developed by MARINTEK, and the 3D Hydrodynamic PREssure CALculation Program PRECAL, which is developed by MARIN.

In addition to the mass distribution and geometry, MAESTRO also has the ability to compute the radii of gyration about a location, which can be used as additional input and/or comparison to the seakeeping tool of choice.

Export MAESTRO Wetted Elements and Mass Distribution

MAESTRO offers the ability to export particular information of all wetted elements, tank elements, and nodal mass in the FE model. This is accomplished by exporting a *Wetted Surface* file (*.wet).

To export the *Wetted Surface* file, click **File > Export > Wetted Surface**, which will allow the user to name and specify the location of the file. A sample *Wetted Surface* file is located in the Models and Samples/Extreme Load Analysis directory and is called *S175-ITTC-2010-nd-mass-933.wet*.

© 2014 Optimum Structural Design, Inc.

The exported file includes the following information associated with the model and elements:

- Model units (See Line 2 in the figure below)
- Nodes of Wetted Elements: Number of Nodes, FeTag, Node XYZ location, Location of Part in MAESTRO model (See Lines 1-10 in the figure below)

```
1  $Nodes of Wetted Elements
2  Units in lbm
3  NumNodes 677
4  $FeTag X Y Z LongName
5  nodewet      1  288 0    0    "/top/s1-hull/m5-frame 6-9 EndPt=1, Sec=0"
6  nodewet      2  288 0    5.74591 "/top/s1-hull/m5-frame 6-9 EndPt=2, Sec=0"
7  nodewet      3  288 27.3327 35  "/top/s1-hull/m5-frame 6-9 EndPt=12, Sec=0"
8  nodewet      5  288 54  63.5358 "/top/s1-hull/m5-frame 6-9 EndPt=7, Sec=0"
9  nodewet      6  288 40.8803 49.5    "/top/s1-hull/m5-frame 6-9 EndPt=9, Sec=0"
10 nodewet      7  288 63.1087 73.2807 "/top/s1-hull/m5-frame 6-9 EndPt=3, Sec=0"
```

- Wetted Elements: Number of Elements, FeTag ("-" sign indicates pressure side opposite of normal), Element Node IDs, Center of Gravity Location, Normal Vector (normalized), Location of *Part* in MAESTRO model (See Lines 682-687 in the figure below)

```
682 $Elements
683 $ID Node1 Node2 Node3 Node4 CGX CGY CGZ NormalX NormalY NormalZ LongName
684 NumElements 644
685 element -1      1     24     23      2  312.054 0   2.89267 0   -1  0   "/top/s1-hull/m5-frame 6-9 Strake 1() Sec 1"
686 element -2     24     41     42     23  360.054 0   2.93202 0   -1  0   "/top/s1-hull/m5-frame 6-9 Strake 1() Sec 2"
687 element -3     41    117    119     42  408.053 0   2.97137 0   -1  0   "/top/s1-hull/m5-frame 6-9 Strake 1() Sec 3"
```

- Tank Elements:
  - o Tank (Volume Group) ID and Name (See Line 1330 in the figure below)
  - o Element FeTag, Element Node IDs, Center of Gravity Location, Normal Vector (normalized), Location of Part in MAESTRO model (See Lines 1331-1333 in the figure below)

```
1329 $Tank Elements
1330 tank 1  "volume/ballast tank 1.s fr 6-9"
1331 tankelm 5169      2714    2713    2715  0    432 51.9454 -90.4802    1    0    -0  "/top/s1-hull/m6-frame 9-12 Tri 4"
1332 tankelm 5170      2704    2712    2699  0    432 11.3785 -15.6607    1    0    0   "/top/s1-hull/m6-frame 9-12 Tri 5"
1333 tankelm 5171      2701    2700    2704  0    432 24.2778 -39.8333    1    0    -0  "/top/s1-hull/m6-frame 9-12 Tri 6"
```

- All Nodes: Number of Nodes, FeTag Node XYZ location, Location of Part in MAESTRO model (See Lines 2137-2142 in the figure below)

© 2014 Optimum Structural Design, Inc.

```
$All Nodes including Wetted Nodes and top-down finemesh module nodes
NumNodes 12647
node      1 288 0    0    "/top/s1-hull/m5-frame 6-9 EndPt=1, Sec=0"
node      2 288 0    5.74591 "/top/s1-hull/m5-frame 6-9 EndPt=2, Sec=0"
node      3 288 27.3327 35  "/top/s1-hull/m5-frame 6-9 EndPt=12, Sec=0"
node      4 288 63.1098 49.5    "/top/s1-hull/m5-frame 6-9 AddPt=1"
```

- Equivalent Nodal Mass (each load case): Load Case ID and Name, FeTag, Mass Value (See Lines 14786-14793 in the figure below)

```
14786  $Equivalent Nodal Mass for each load case
14787  loadcase 1  "Departure without Torpedoes"
14788  nodalmass 1 77.1511
14789  nodalmass 2 175.908
14790  nodalmass 3 259.612
14791  nodalmass 4 61.9805
14792  nodalmass 5 158.34
14793  nodalmass 6 155.077
```

13.5 3rd Party Hydrodynamic Analysis

At this point in the direct analysis process, seakeeping analysis tools can be utilized to predict ship motions and wave induced loads.

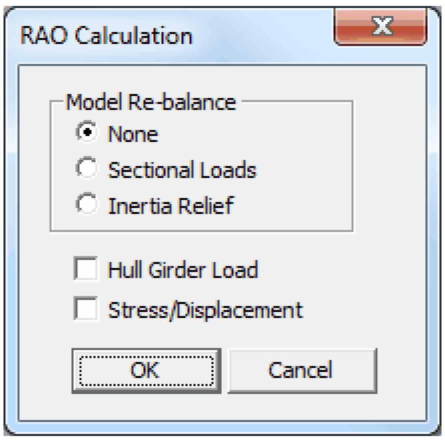
This analysis will incorporate the vessels: Base Loading Conditions, Operational Profile, Environmental Conditions, and Hull Geometry (including appendages), some of which can be extracted from the MAESTRO Global model, as described in previous sections.

Typically this computation is executed using a spectral-based approach and relies on Response Amplitude Operators (RAOs). RAOs are calculated for regular waves of unit amplitude for a range of wave frequencies and wave headings, typically set forth by *Class* guidance.

The seakeeping analysis tool can also be used to identify Most Probable Extreme Values (MPEVs) for global load parameters of interest (i.e. Vertical Bending Moment, Shear Force, etc) as well an equivalent design wave (EDW). The EDW is a sinusoidal wave characterized by its: amplitude, length (or frequency), heading, and phase angle. This EDW simulates the magnitude and location of the extreme value of the global load parameter of interest. Ultimately, the external hydrodynamic pressures, computed by the seakeeping analysis tool, should be extracted.

13.6 Compute RAOs in MAESTRO

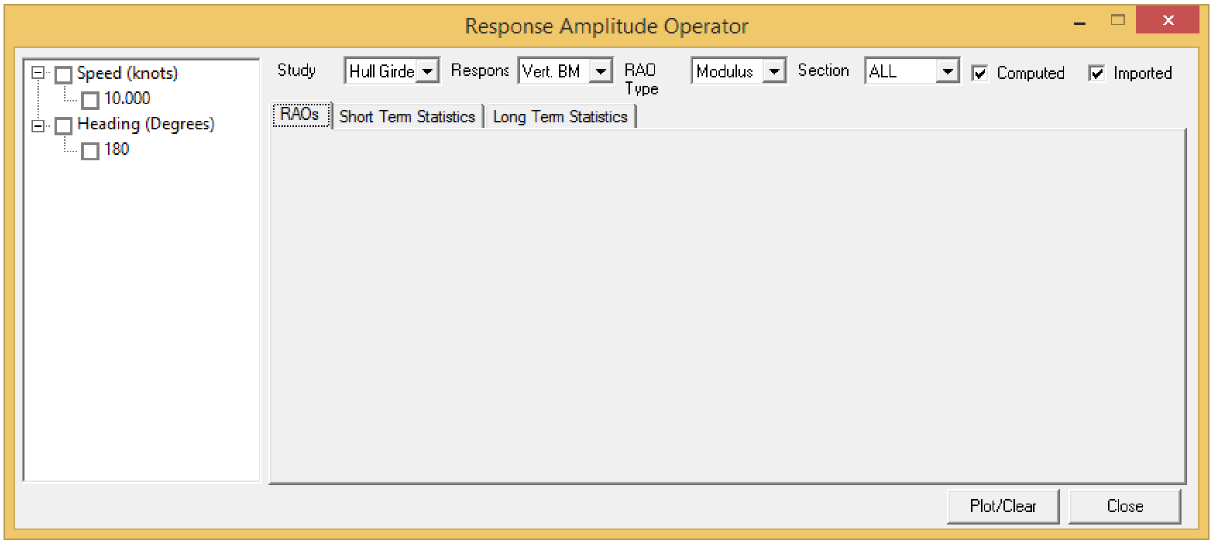
Once a Ship Motions File (*.smn or *.bmn) is loaded into MAESTRO, the hull girder load and stress/displacement RAOs can be calculated and the model can be rebalanced, if desired. From the Wave menu, select the "Compute RAOs" option:



If MAESTRO-Wave was used to create the Ship Motions File (*.smn or *.bmn), the RAOs have already been calculated, with the exception of the Stress RAOs. It is recommended that the model be re-balanced using the "Sectional Loads" option when importing loads from a 3rd party hydrodynamics code. The stress RAO is only needed for the Spectral Fatigue Analysis. Click OK to run the RAO calculation. A dialog will display when the calculations are complete.

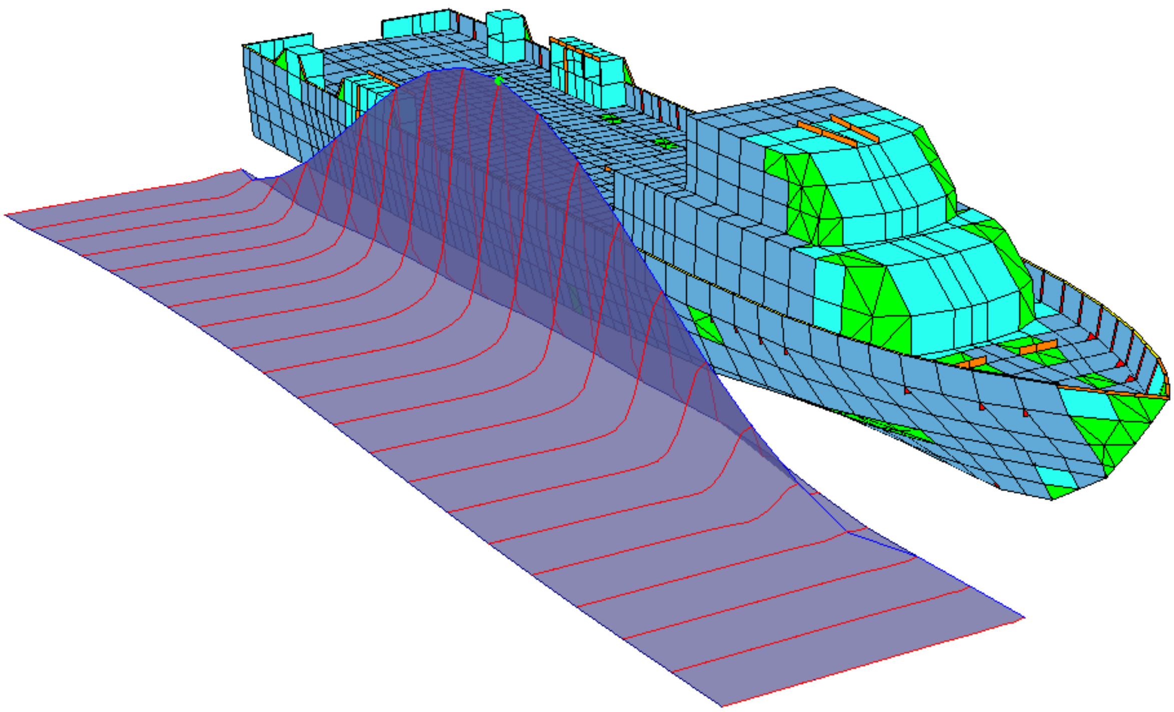
13.7 Viewing RAOs and Statistics

Once a Ship Motions File is imported from a 3rd party analysis or from MAESTRO-Wave and RAOs have been calculated, they can be viewed graphically in the MAESTRO GUI. Select Wave > View RAOs/Statistics/Time series from the menu:

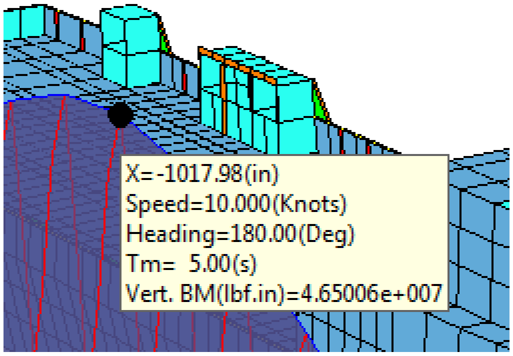


Listed on the left-hand side of the dialog, are the speeds and headings available for display. One speed and one heading should be selected at a time. The options along the top allow the selection of the type of study, response, RAO type, and which section or sections to plot. The available Study types are: Hull Girder Load, Motion, Stress, Deflection, Stress Spectrum and MSI. The default Section is "ALL" which will plot all sections, or a specific section can be selected from the drop-down. Choosing a specific section allows the user to export the RAO from that specific section versus the maximum value. The check boxes for "Computed" and "Imported" allow the user to plot either the computed (i.e. Wave > Compute RAOs) RAO or the imported (MAESTRO-Wave or 3rd party results) RAO, or both.

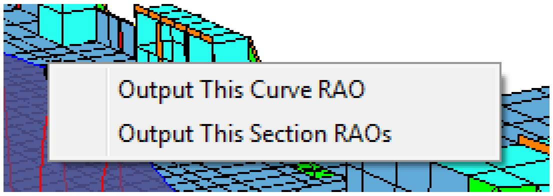
Clicking "Plot/Clear" will plot the RAO next to the MAESTRO model, populate the Output window with a list of computed values or some combination of both depending on what is being queried.

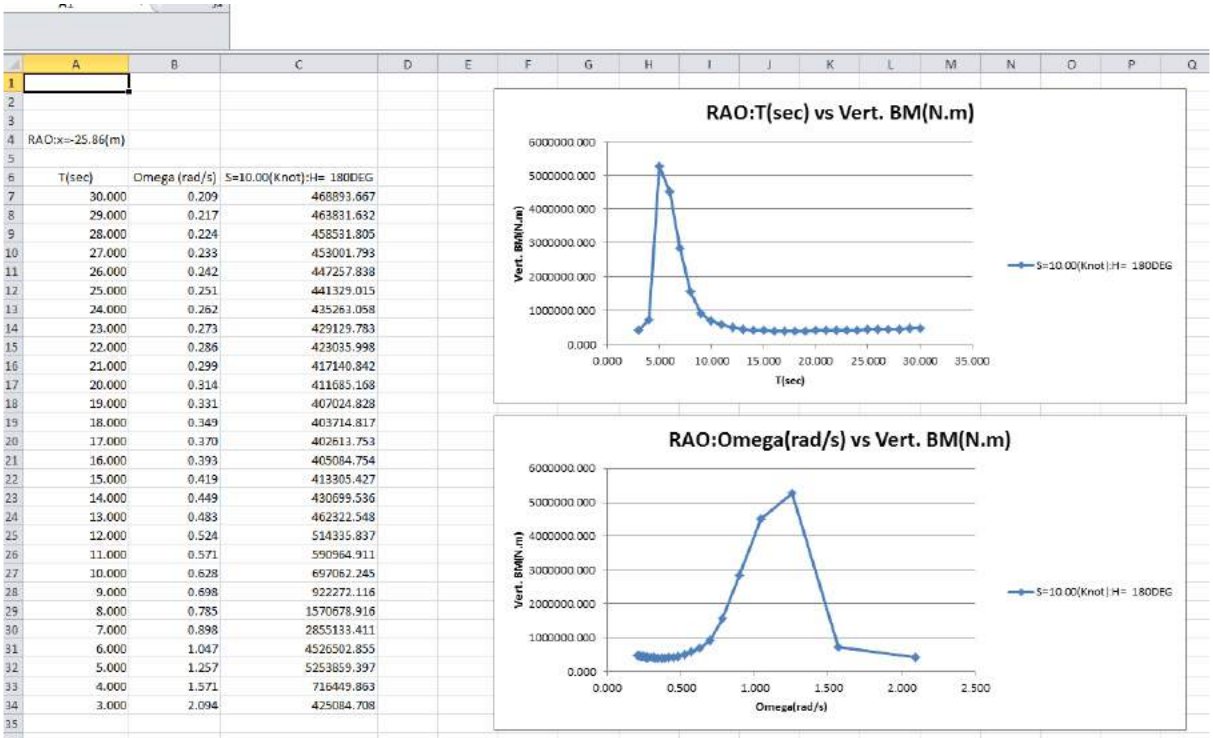


Using the dynamic query, you can select the node at the maximum value and recover the location, condition and maximum value.



Right-clicking the queried node allows the associated RAO curve or all the RAOs currently displayed at the selected section to be output and plotted in Microsoft Excel.

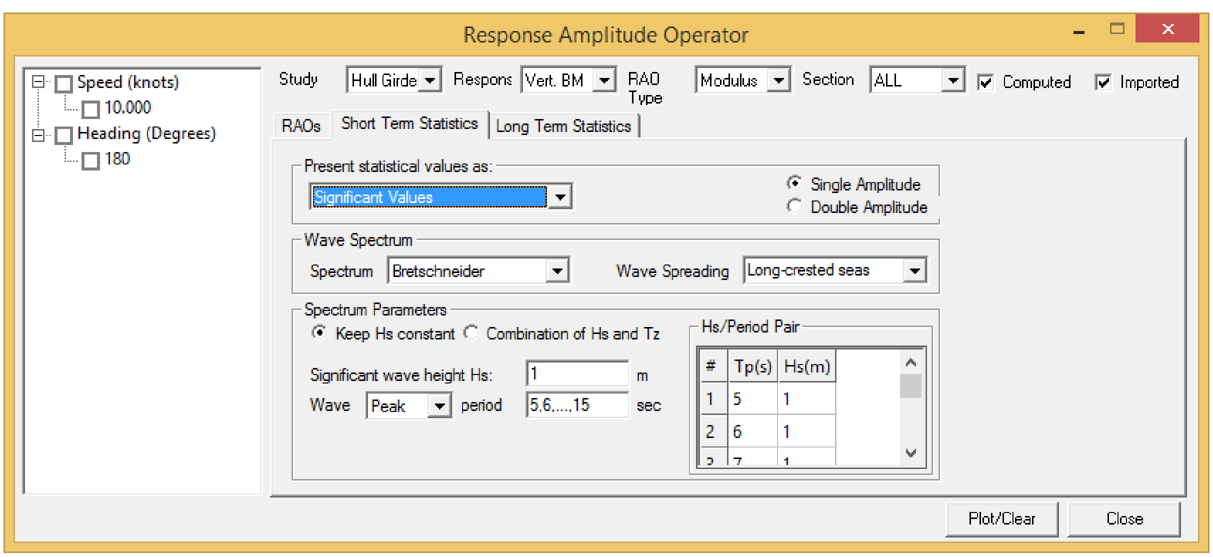




T(sec)	Omega (rad/s)	S=10.00(Knot):H= 180DEG
30.000	0.209	468893.667
29.000	0.217	463831.632
28.000	0.224	458531.805
27.000	0.233	453001.793
26.000	0.242	447257.838
25.000	0.251	441329.015
24.000	0.262	435263.058
23.000	0.273	429129.783
22.000	0.286	423035.998
21.000	0.299	417140.842
20.000	0.314	411685.168
19.000	0.331	407024.828
18.000	0.349	403714.817
17.000	0.370	402613.753
16.000	0.393	405084.754
15.000	0.419	413305.427
14.000	0.449	430699.536
13.000	0.483	462322.548
12.000	0.524	514335.837
11.000	0.571	590964.911
10.000	0.628	697062.245
9.000	0.698	922272.116
8.000	0.785	1570678.916
7.000	0.898	2855133.411
6.000	1.047	4526502.855
5.000	1.257	5253859.397
4.000	1.571	716449.863
3.000	2.094	425084.708

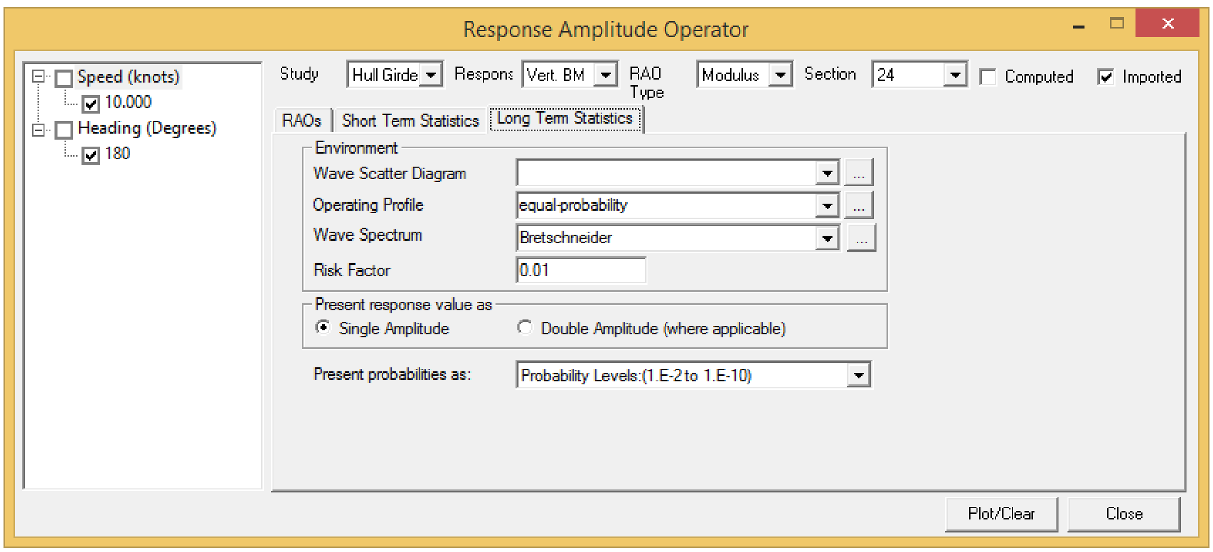
To recover the RAO at a specific section, use the section drop-down in the RAO dialog to select the desired location.

The Short Term Statistics tab allows the user to compute the extreme values using short term statistics and display the results in the Output tab. The short term statistics calculations default to a 3 hour exposure.



The statistical values that can be presented are: Significant Values, Standard Deviation (RMS), Expected Maximum, or Average 1/n'th Largest Value (most of which can be set to display single or double amplitude). Where applicable, additional inputs will be displayed (i.e., exposure duration for Expected Maximum and 'n' for the 1/n'th Largest). The Wave Spectrum can be selected as Bretschneider, JONSWAP, NorthAtlantic2, OCHI, PiersonMoskowitz or any other spectrum that has been added by the user through the Wave Spectrum dialog. The Wave Spreading is currently limited to Long-crested seas. If using an analytical spectrum, the spectrum parameters are user-defined by a significant wave height (s) and period range (defined in seconds). Selecting the "Combination of Hs and Tz" allows for multiple combinations of significant wave heights and periods. A summary of the current height/period pairs is shown in the table in the lower right. Pairs can also be added by right-clicking and selecting "Add" from within the table. If a user defined spectrum is chosen, spectrum parameters do not apply and are hidden.

The Long Term Statistics tab allows the user to compute the extreme values using long term statistics.



The Wave Scatter Diagram and Operating Profile will be discussed in the next section. Similar to the short term statistics output, the long term hull girder responses are listed in the Output tab.

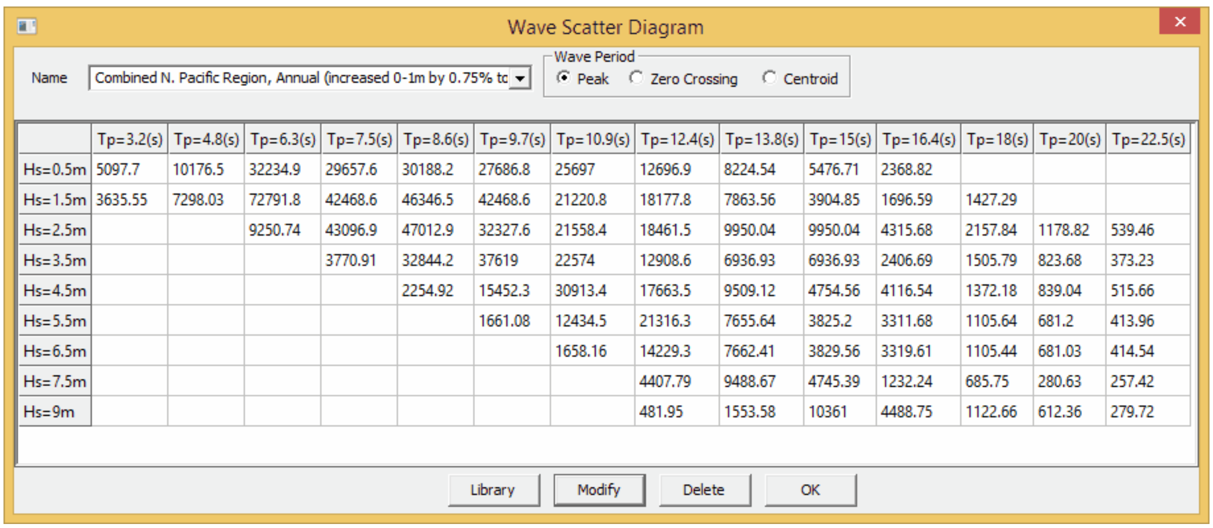
13.8 Calculate Design Wave

It should be noted that the imported hydrodynamic loads are *unit wave* generated responses, therefore, an equivalent design wave must be defined to simulate the magnitude and location of the extreme loading condition. In MAESTRO, the equivalent design waves

© 2014 Optimum Structural Design, Inc.

are automatically calculated for the long and short term analysis.

Before we can calculate the design waves using long term statistics, we first need to define a wave scatter diagram and operating profile. MAESTRO is able to import Wave Scatter (*.sea) and Operating Profile (*.opf) files. User-defined Wave Scatter diagrams can be imported from the File menu. To view an imported Wave Scatter diagram, select Wave > Wave Scatter Diagrams from the menu. There is also a Library button that allows the user to select one of four pre-defined wave probabilities. The Library wave probabilities only expose the wave height probability to the user; associated wave period probabilities and/or spectral shapes are derived from the wave height and are computed internally in a similar fashion to how they are implemented in the Navy's SPECTRA program.

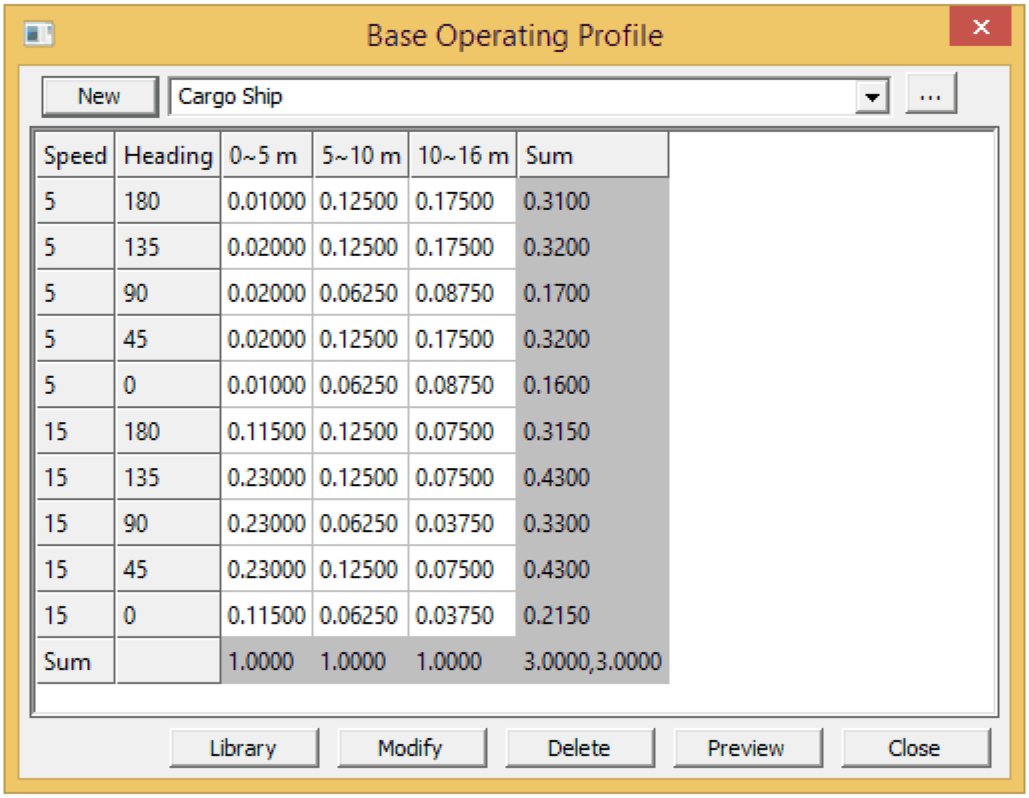


	Tp=3.2(s)	Tp=4.8(s)	Tp=6.3(s)	Tp=7.5(s)	Tp=8.6(s)	Tp=9.7(s)	Tp=10.9(s)	Tp=12.4(s)	Tp=13.8(s)	Tp=15(s)	Tp=16.4(s)	Tp=18(s)	Tp=20(s)	Tp=22.5(s)
Hs=0.5m	5097.7	10176.5	32234.9	29657.6	30188.2	27686.8	25697	12696.9	8224.54	5476.71	2368.82			
Hs=1.5m	3635.55	7298.03	72791.8	42468.6	46346.5	42468.6	21220.8	18177.8	7863.56	3904.85	1696.59	1427.29		
Hs=2.5m			9250.74	43096.9	47012.9	32327.6	21558.4	18461.5	9950.04	9950.04	4315.68	2157.84	1178.82	539.46
Hs=3.5m				3770.91	32844.2	37619	22574	12908.6	6936.93	6936.93	2406.69	1505.79	823.68	373.23
Hs=4.5m					2254.92	15452.3	30913.4	17663.5	9509.12	4754.56	4116.54	1372.18	839.04	515.66
Hs=5.5m						1661.08	12434.5	21316.3	7655.64	3825.2	3311.68	1105.64	681.2	413.96
Hs=6.5m							1658.16	14229.3	7662.41	3829.56	3319.61	1105.44	681.03	414.54
Hs=7.5m								4407.79	9488.67	4745.39	1232.24	685.75	280.63	257.42
Hs=9m								481.95	1553.58	10361	4488.75	1122.66	612.36	279.72

A list of the available wave scatter diagrams are shown in the drop-down at the top of the dialog. The values within the diagram can be edited as well as how the wave period is defined. If changes are made, the user must click "Modify" to save the changes.

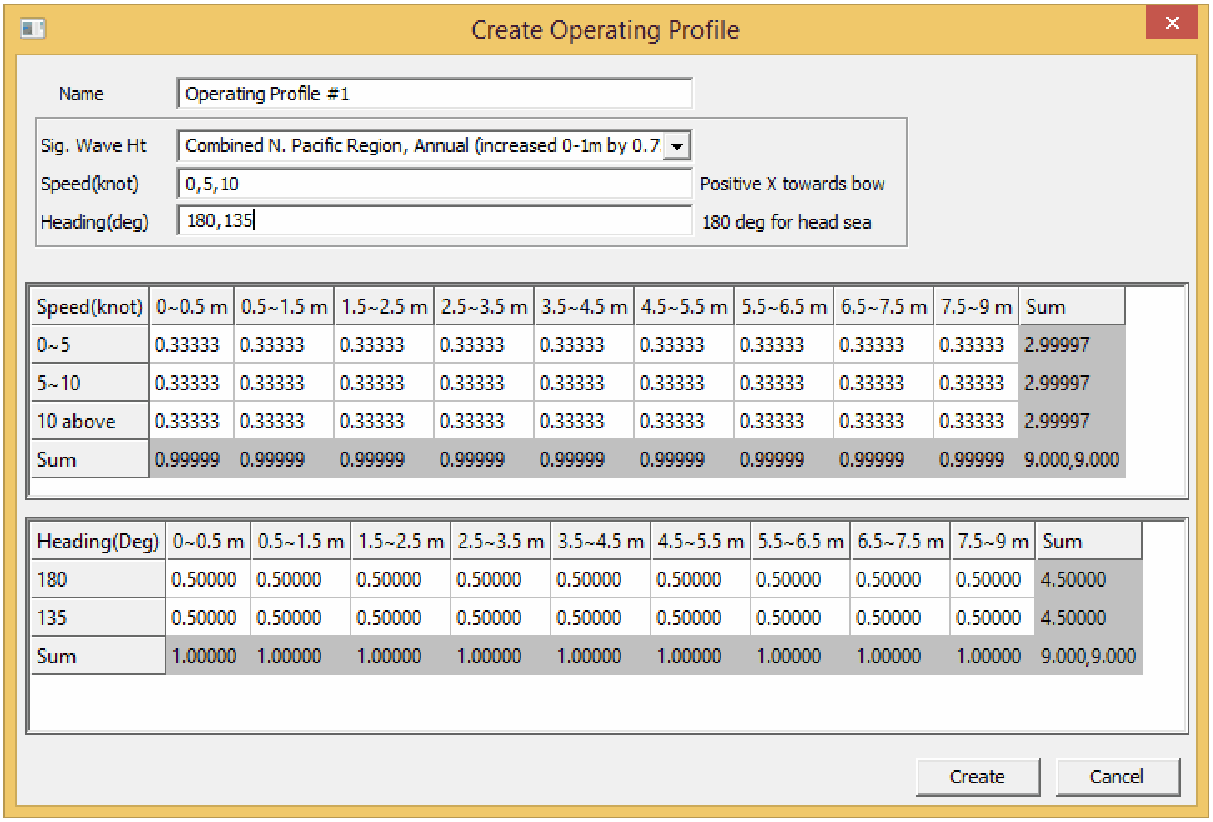
The Operating Profile can be imported from the File menu, or created by the user. To create or view an existing profile, select Wave > Base Operational Profiles from the menu.

© 2014 Optimum Structural Design, Inc.



Speed	Heading	0~5 m	5~10 m	10~16 m	Sum
5	180	0.01000	0.12500	0.17500	0.3100
5	135	0.02000	0.12500	0.17500	0.3200
5	90	0.02000	0.06250	0.08750	0.1700
5	45	0.02000	0.12500	0.17500	0.3200
5	0	0.01000	0.06250	0.08750	0.1600
15	180	0.11500	0.12500	0.07500	0.3150
15	135	0.23000	0.12500	0.07500	0.4300
15	90	0.23000	0.06250	0.03750	0.3300
15	45	0.23000	0.12500	0.07500	0.4300
15	0	0.11500	0.06250	0.03750	0.2150
Sum		1.0000	1.0000	1.0000	3.0000,3.0000

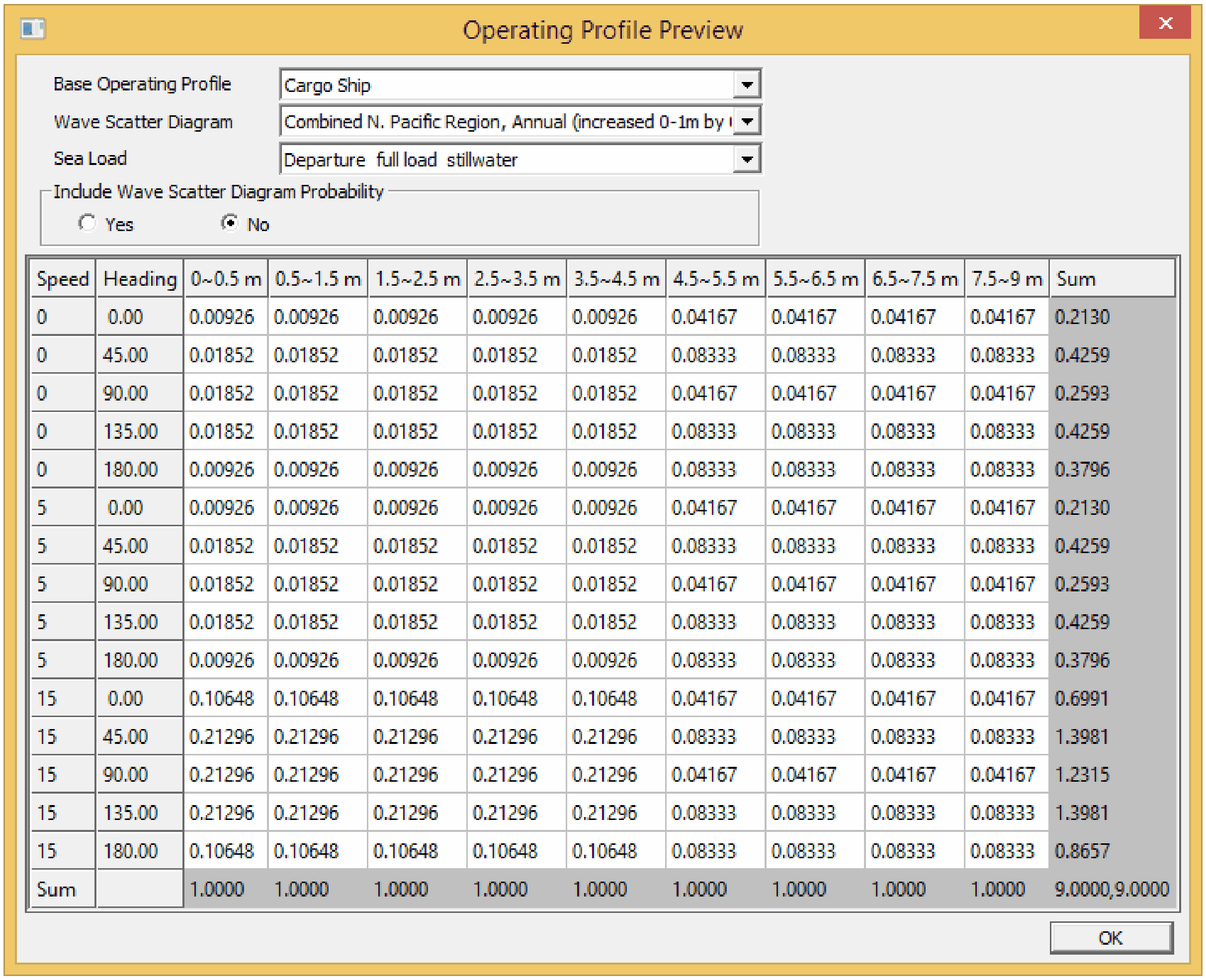
To create a new profile, click the "New" button to open the Create Operating Profile dialog. Here, the user can enter a name for the profile at the top of the form. The significant wave height probabilities are defined by the selected wave scatter diagram. The probabilities of each speed/wave or heading/wave cells can be input by the user. The sum of the speed/wave and heading/wave probabilities must each equal 1.0. To save the operating profile, the user must click the "Create" button to update the values, save the profile and close the dialog.



Speed(knot)	0~0.5 m	0.5~1.5 m	1.5~2.5 m	2.5~3.5 m	3.5~4.5 m	4.5~5.5 m	5.5~6.5 m	6.5~7.5 m	7.5~9 m	Sum
0~5	0.33333	0.33333	0.33333	0.33333	0.33333	0.33333	0.33333	0.33333	0.33333	2.99997
5~10	0.33333	0.33333	0.33333	0.33333	0.33333	0.33333	0.33333	0.33333	0.33333	2.99997
10 above	0.33333	0.33333	0.33333	0.33333	0.33333	0.33333	0.33333	0.33333	0.33333	2.99997
Sum	0.99999	0.99999	0.99999	0.99999	0.99999	0.99999	0.99999	0.99999	0.99999	9.000,9.000

Heading(Deg)	0~0.5 m	0.5~1.5 m	1.5~2.5 m	2.5~3.5 m	3.5~4.5 m	4.5~5.5 m	5.5~6.5 m	6.5~7.5 m	7.5~9 m	Sum
180	0.50000	0.50000	0.50000	0.50000	0.50000	0.50000	0.50000	0.50000	0.50000	4.50000
135	0.50000	0.50000	0.50000	0.50000	0.50000	0.50000	0.50000	0.50000	0.50000	4.50000
Sum	1.00000	1.00000	1.00000	1.00000	1.00000	1.00000	1.00000	1.00000	1.00000	9.000,9.000

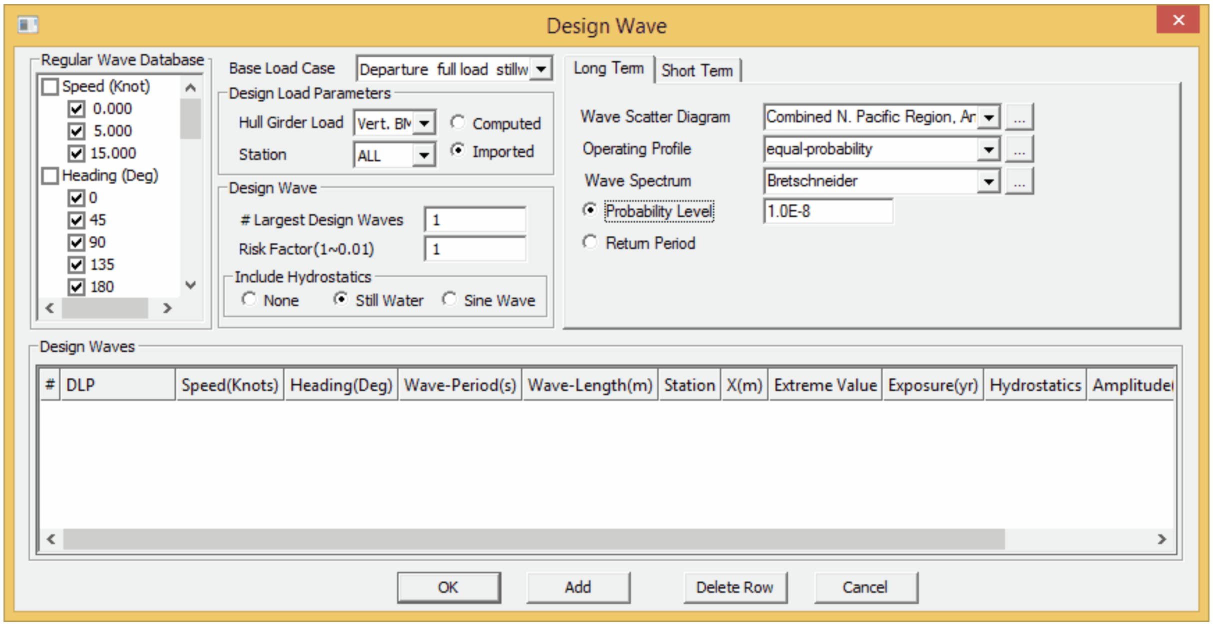
Once back in the Base Operating Profile dialog, the profile probabilities can be modified by making changes in the table and clicking the "Modify" button. Unneeded profiles can be deleted using the "Delete" button and the user can click the "Preview" button to view the joint probabilities defined for the operating profile. The Operating Profile Preview dialog allows the user to select any available Operating Profile, Wave Scatter Diagram and Sea Load. The joint probabilities can be displayed with or without the probabilities of the seaway included so the user can evaluate how the operational profile is combined with the speeds and headings used to define the Sea Load.



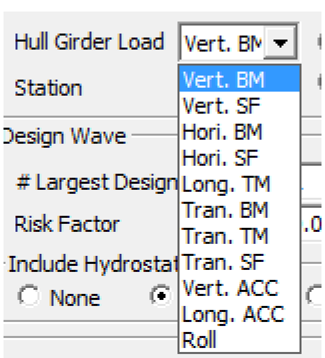
Speed	Heading	0~0.5 m	0.5~1.5 m	1.5~2.5 m	2.5~3.5 m	3.5~4.5 m	4.5~5.5 m	5.5~6.5 m	6.5~7.5 m	7.5~9 m	Sum
0	0.00	0.00926	0.00926	0.00926	0.00926	0.00926	0.04167	0.04167	0.04167	0.04167	0.2130
0	45.00	0.01852	0.01852	0.01852	0.01852	0.01852	0.08333	0.08333	0.08333	0.08333	0.4259
0	90.00	0.01852	0.01852	0.01852	0.01852	0.01852	0.04167	0.04167	0.04167	0.04167	0.2593
0	135.00	0.01852	0.01852	0.01852	0.01852	0.01852	0.08333	0.08333	0.08333	0.08333	0.4259
0	180.00	0.00926	0.00926	0.00926	0.00926	0.00926	0.08333	0.08333	0.08333	0.08333	0.3796
5	0.00	0.00926	0.00926	0.00926	0.00926	0.00926	0.04167	0.04167	0.04167	0.04167	0.2130
5	45.00	0.01852	0.01852	0.01852	0.01852	0.01852	0.08333	0.08333	0.08333	0.08333	0.4259
5	90.00	0.01852	0.01852	0.01852	0.01852	0.01852	0.04167	0.04167	0.04167	0.04167	0.2593
5	135.00	0.01852	0.01852	0.01852	0.01852	0.01852	0.08333	0.08333	0.08333	0.08333	0.4259
5	180.00	0.00926	0.00926	0.00926	0.00926	0.00926	0.08333	0.08333	0.08333	0.08333	0.3796
15	0.00	0.10648	0.10648	0.10648	0.10648	0.10648	0.04167	0.04167	0.04167	0.04167	0.6991
15	45.00	0.21296	0.21296	0.21296	0.21296	0.21296	0.08333	0.08333	0.08333	0.08333	1.3981
15	90.00	0.21296	0.21296	0.21296	0.21296	0.21296	0.04167	0.04167	0.04167	0.04167	1.2315
15	135.00	0.21296	0.21296	0.21296	0.21296	0.21296	0.08333	0.08333	0.08333	0.08333	1.3981
15	180.00	0.10648	0.10648	0.10648	0.10648	0.10648	0.08333	0.08333	0.08333	0.08333	0.8657
Sum		1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	9.0000,9.0000

To calculate the design waves, select Wave > Compute Design Wave from the menu.

© 2014 Optimum Structural Design, Inc.



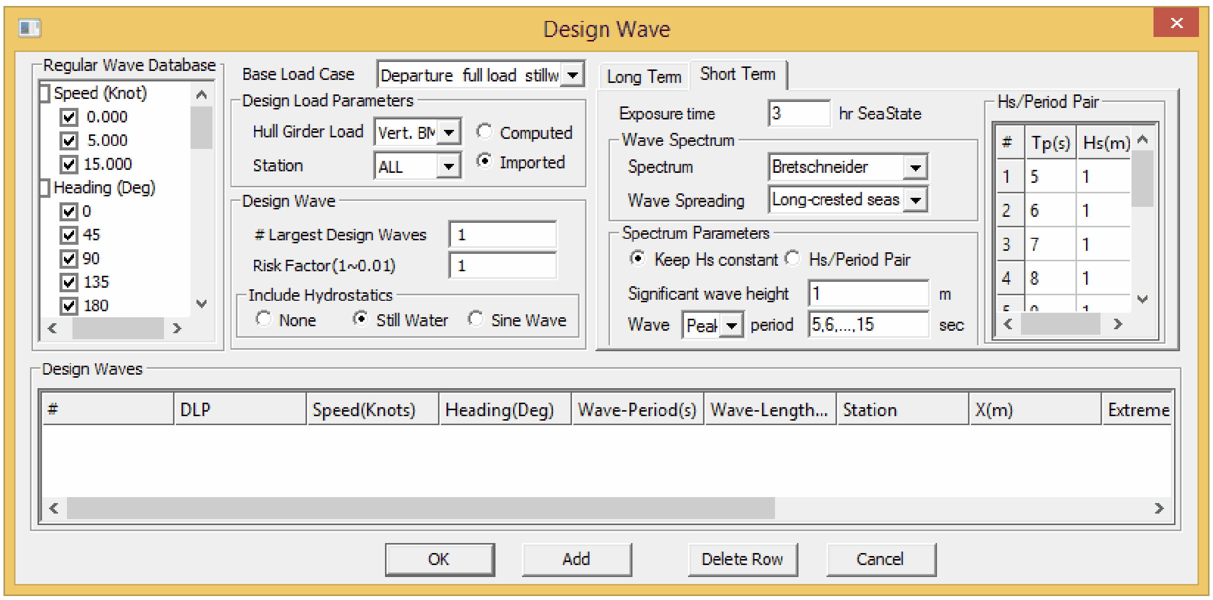
Similar to the View RAO dialog, the available speeds and headings from the database are shown on the left which can be used to control which speeds and headings are included in the evaluation. Unlike the View RAO dialog, however, this tree also includes a list of wave periods below the headings which can be used to control which wave periods are allowed to be used when creating the equivalent design wave. The base MAESTRO load case can be selected from the load case drop-down. The design load parameter (DLP) can be selected from the drop-down list and includes:



This DLP value can be computed using the imported hull girder loads from the *.smn file or the ones computed within MAESTRO and can be computed for all stations, or a user-specified one. The user specifies how many of the largest design waves to compute for the selected DLP and whether or not to include hydrostatics, either from still water or a sine wave. The risk factor is is effectively a scaling factor for the selected exposure time (Exposure time / Risk Factor = extreme value exposure time). The Risk Factor is implemented as described by OCHI's book, *Ocean Waves*, and can be left at 1 for normal use.

The wave scatter diagram, operating profile and wave spectrum can be selected from those available in the drop-down lists. The extreme value(s), using long term statistics can be calculated either by finding a value with a given probability of exceedance or defining the return period (in years), which finds the most probable extreme value expected to occur in the given time period.

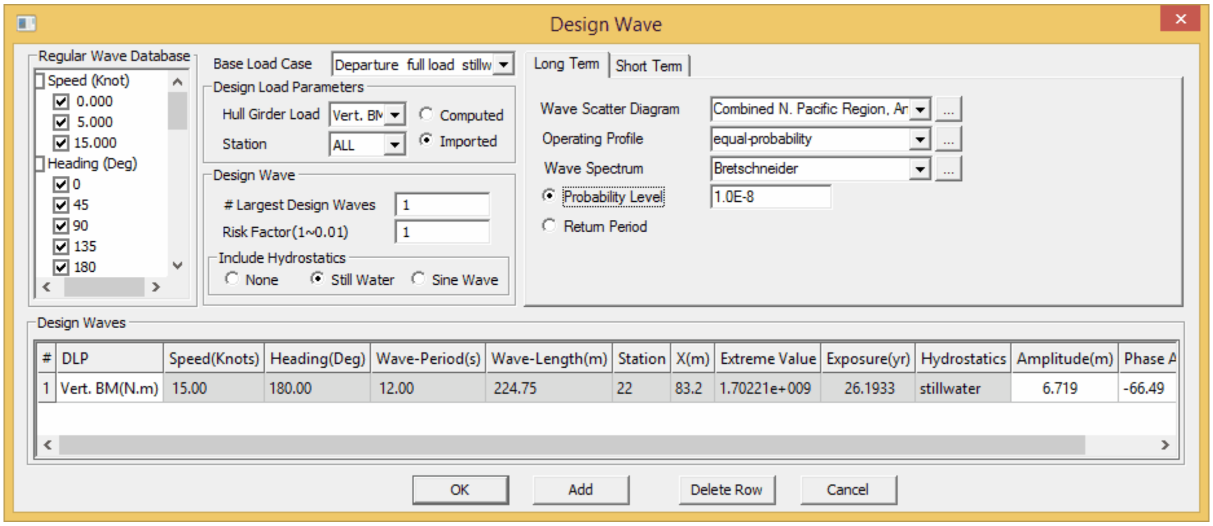
The short term statistics design waves are calculated in a similar fashion to the long term, but using a user-defined exposure time and significant wave height/wave period pairs describing the short term design sea state(s). (If a user defined spectrum is chosen, spectrum parameters do not apply and are hidden.) The full exposure time for each defined seaway is given to each combination from the selected speeds and headings.



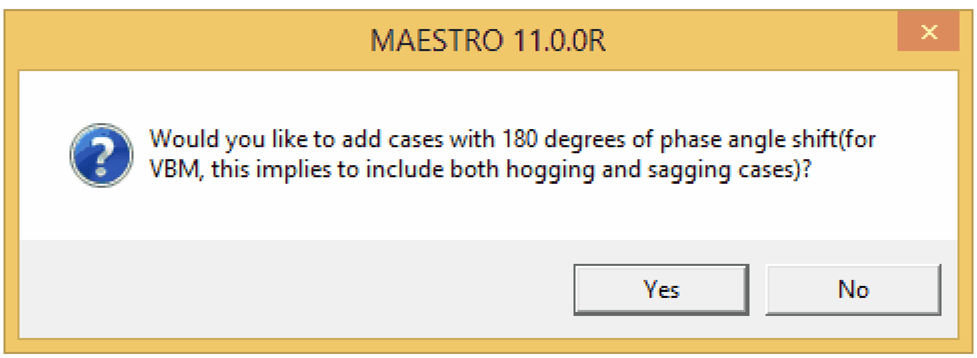
To calculate a design wave or waves, click "Add".

Once calculated, the design waves table at the bottom of the dialog will display the list of design waves created.

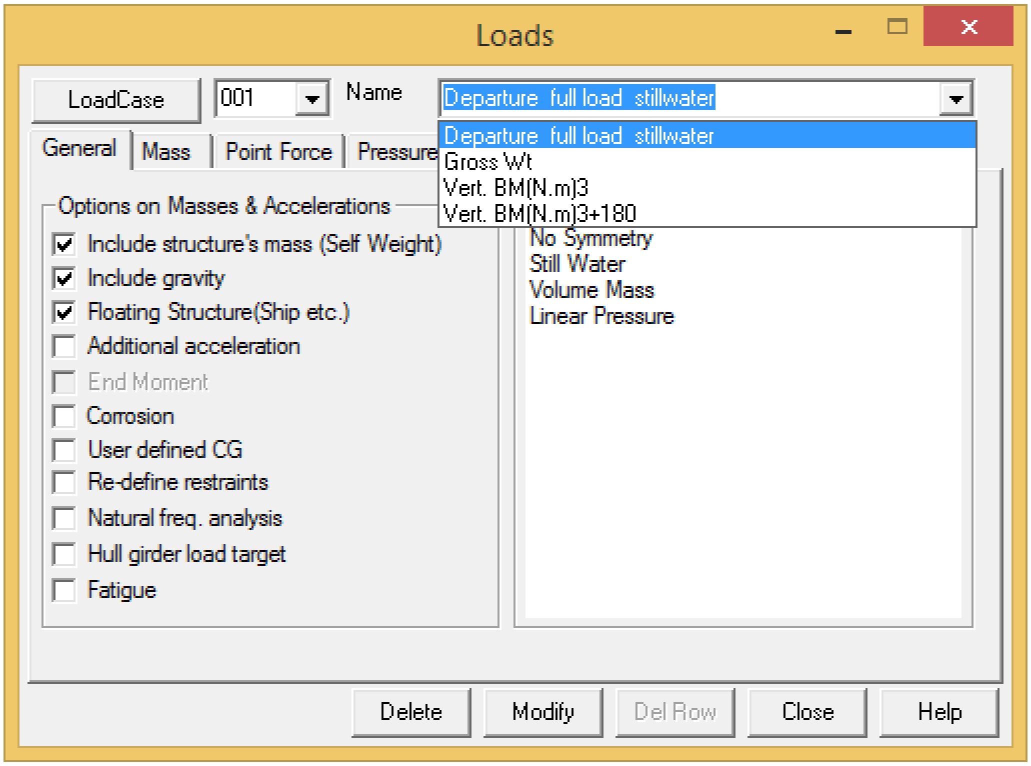
© 2014 Optimum Structural Design, Inc.



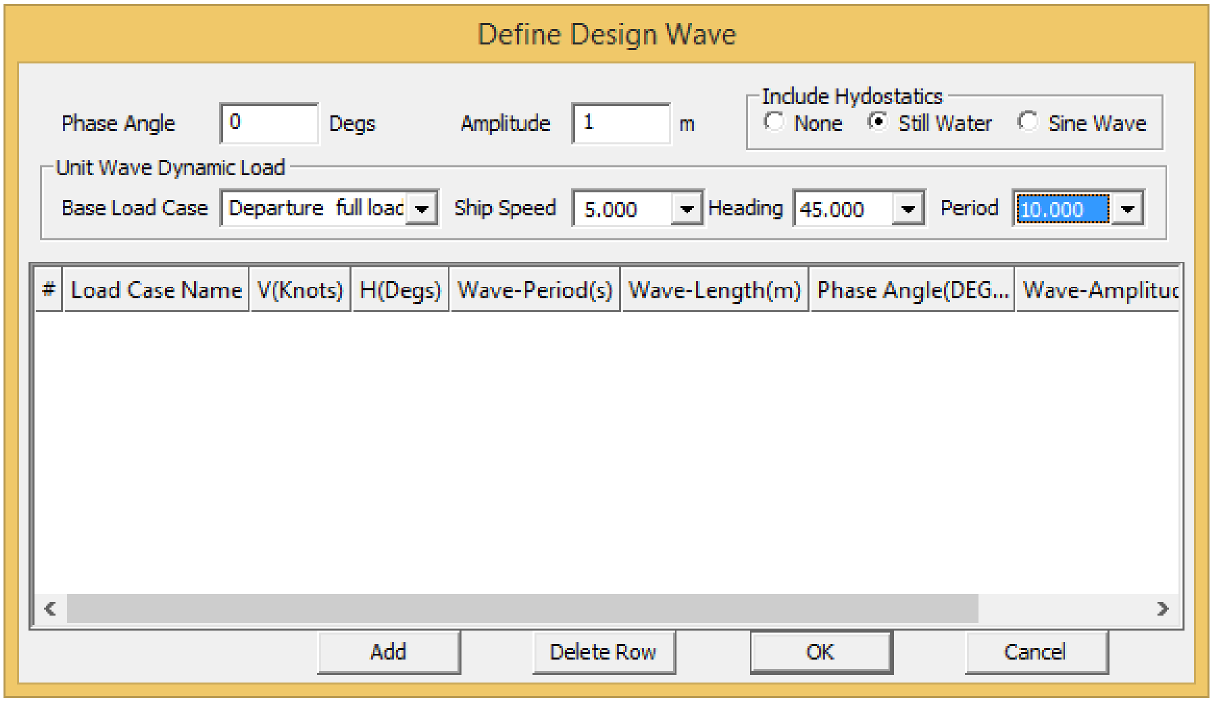
These can be added as new MAESTRO load cases by clicking the "OK" button. When applicable, a dialog will display allowing the user to create a second load case for each DLP with a 180 degree phase shift for the wave. For example, the vertical bending moment DLP would include a hogging and sagging case.



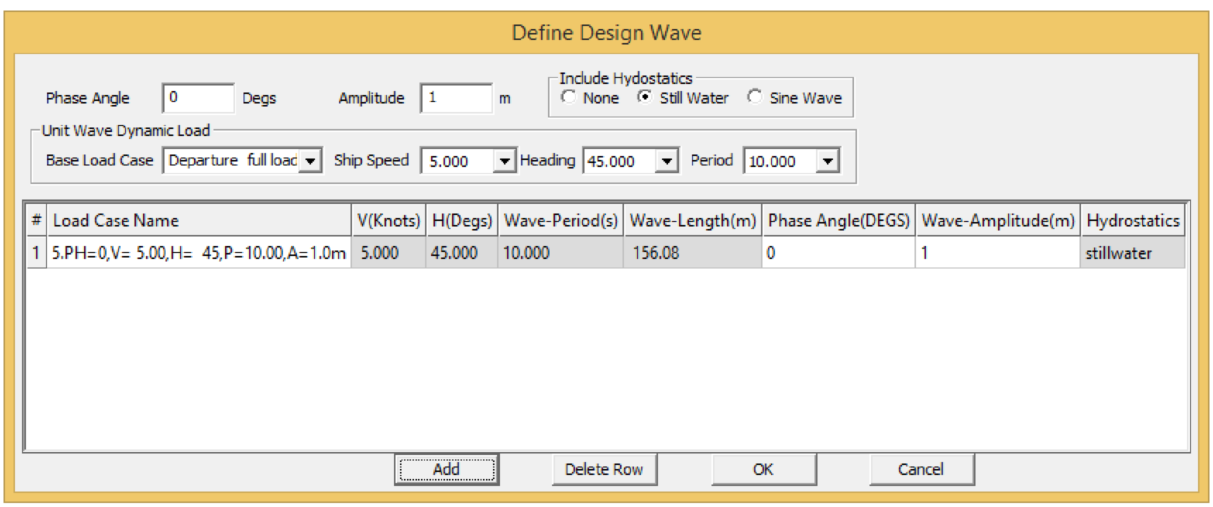
In the end, MAESTRO has added the loading components from the Base Load Case together with the hydrodynamic loading components defined in the Extreme Load dialog. These new load cases can be seen in the MAESTRO load case dialog.



In addition to having MAESTRO automatically find the largest design waves for each DLP, the user can specify their own design wave condition by selecting Wave > Define Design Wave from the menu.



This dialog allows the user to select the specific case (base load, speed, heading and wave period) and recover the design wave. The user can also specify a wave phase angle, wave scale factor and whether to include hydrostatics. Clicking "Add" will create this new design wave and clicking "OK" will create a new MAESTRO load case.



13.9 Structural Response Analysis and Assessment

After equilibrium has been achieved, the analyst can perform structural response analysis and assessment.

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.

Review the following sections for details of MAESTRO structural assessment capabilities:

Finite Element Analysis

Limit State Buckling Analysis

Fine Mesh Analysis

13.10 Validation

Short Term Statistics Validation

To validate the implementation of MAESTRO's short term statistics, the S175 model from the MAESTRO-Wave validation was used. The hull girder load RAOs computed by VERES were first imported into MAESTRO, and then the short term statistics of vertical bending moment and shear force at x=-0.831m were calculated using MAESTRO. The comparison between MAESTRO and VERES are shown in the figures below. The procedure for obtaining the result and the raw data is provided in this section below.

© 2014 Optimum Structural Design, Inc.

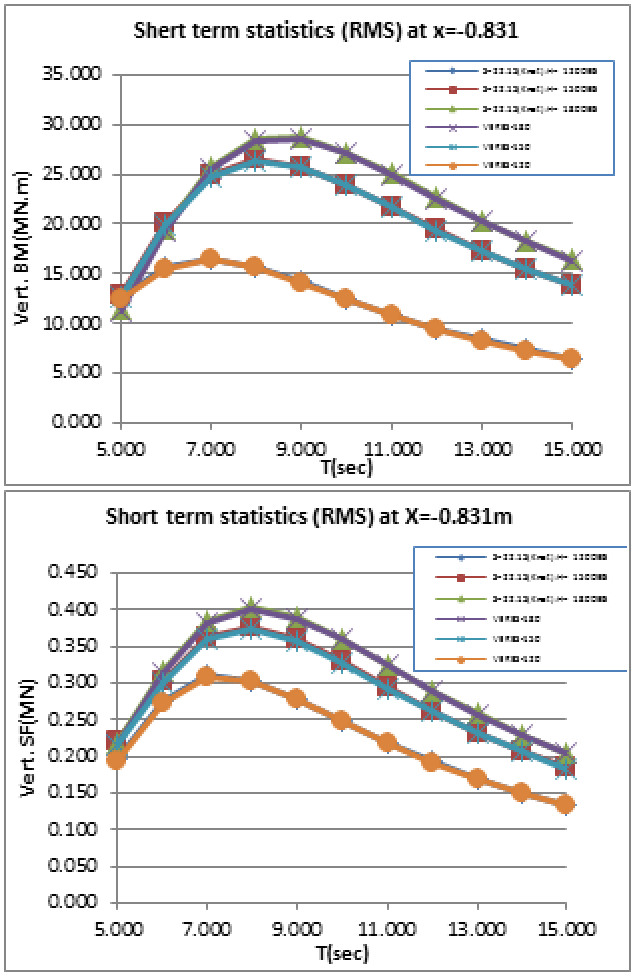
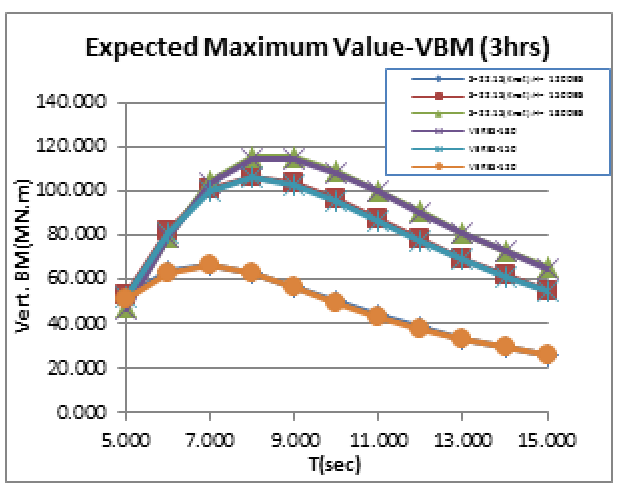


Figure 1. RMS comparison (VBM and VSF)



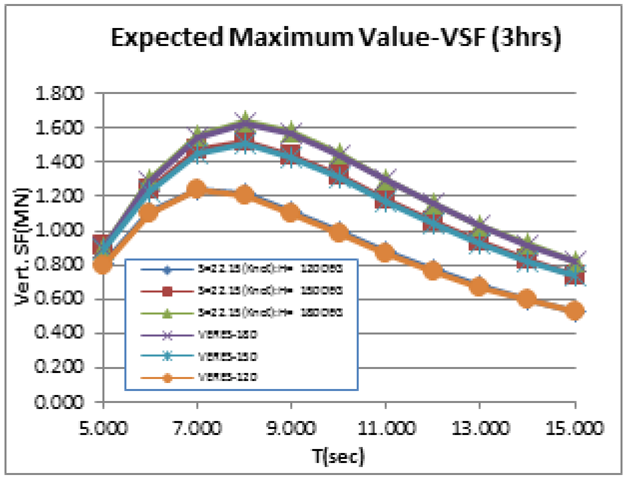


Figure 2. Expected maximum value (3 hours exposure)

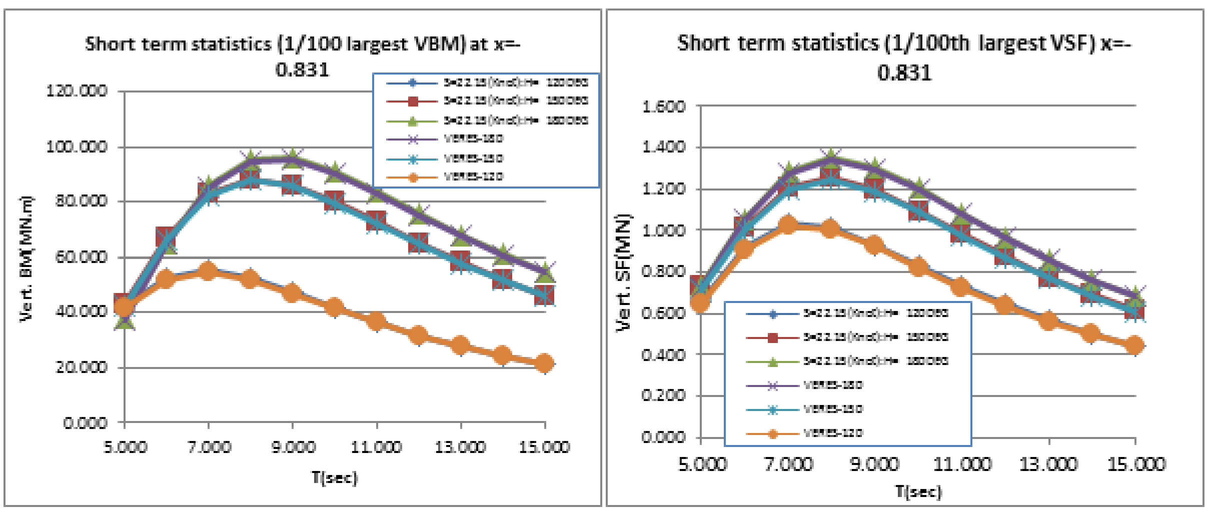


Figure 3. 1/100th largest value

Long Term Statistics and Design Wave Validation

S175 Example

MAESTRO's long term statistics and design wave calculations follow the guideline given by ABS's "Guide for 'SafeHull-Dynamic Loading Approach' for Vessels", where the long term probability of the response is expressed as a summation of joint probability over the short-term sea states. This approach is a little different from the Weibull fit approach adopted by VERES. A quantitative comparison of long term statistics and design waves is given in this section. Supporting data and procedure are provided in this section below.

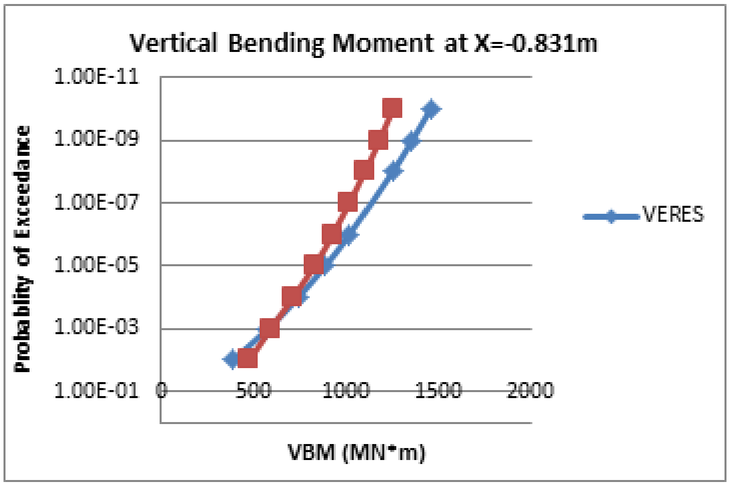


Figure 4. Comparison of Probability of Exceedance

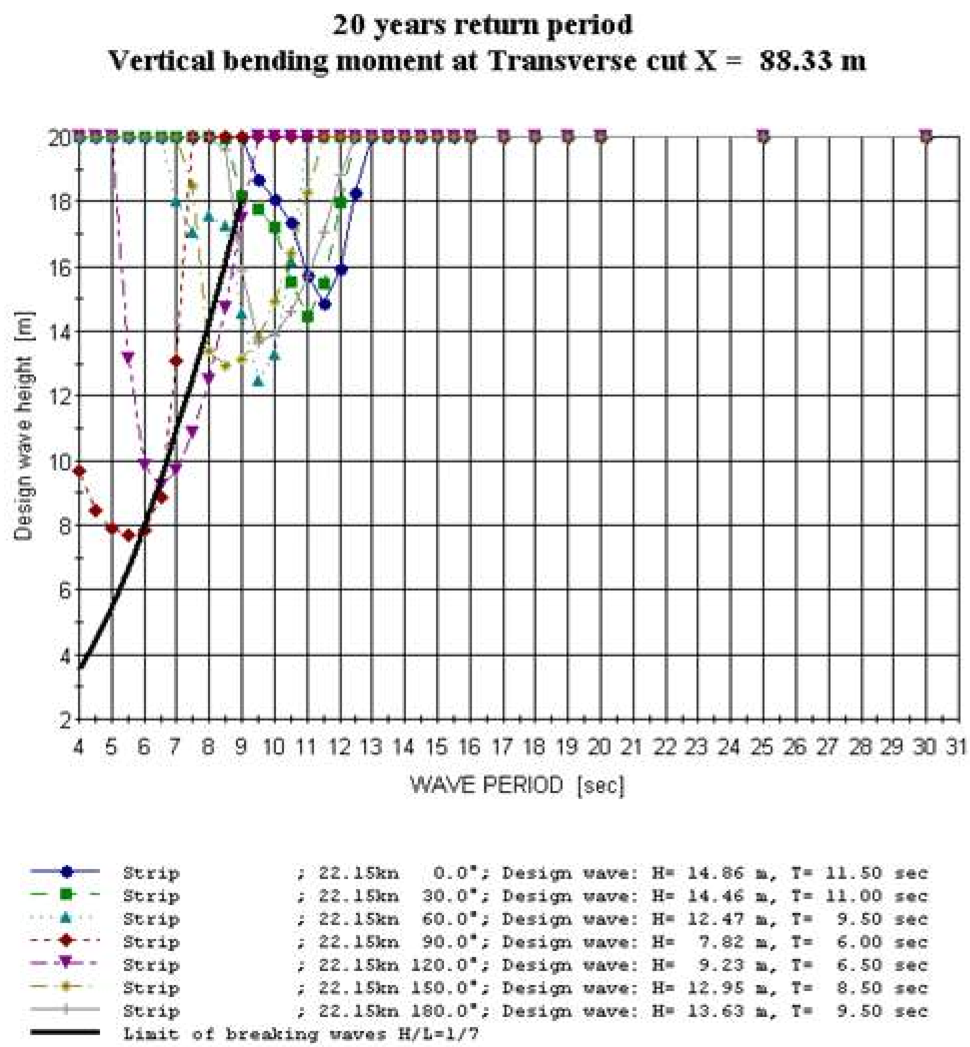


Figure 5. VERES Design wave

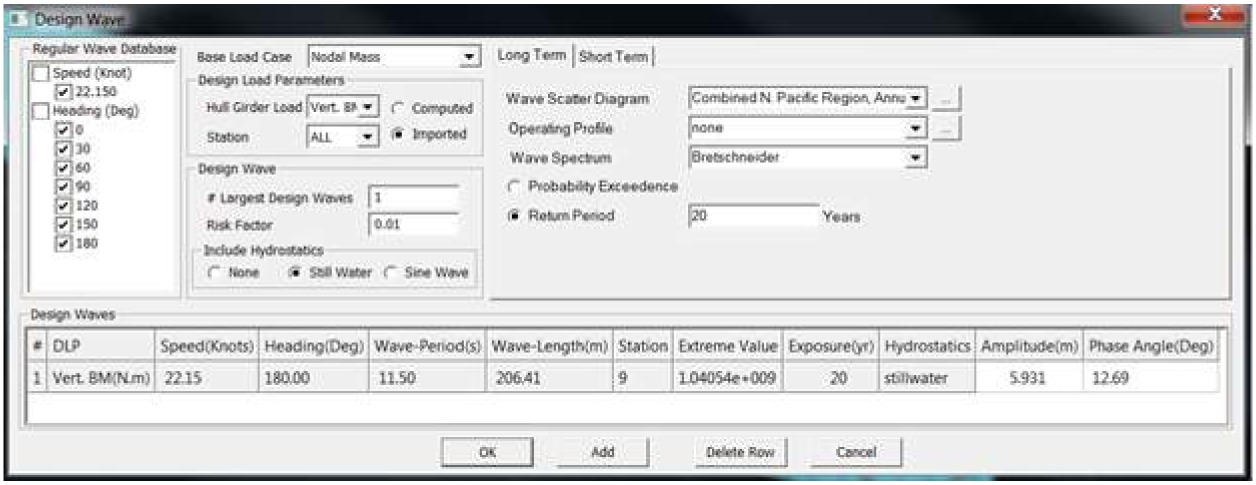


Figure 6. MAESTRO Design Wave

	VERES	MAESTRO
Wave period	11.5 s	11.5 s
Heading	Head sea	Head sea
Wave amplitude	7.43 m	5.93 m

Figure 7. Design Wave Comparison

44,000 Ton Tanker Example

The second example of the design wave calculations uses a 44,000 ton tanker. Table 1 below lists the main particulars.

Length between perpendiculars	180 m
Breadth	32.2 m
Draft	12.1 m
Displacement	44000 ton
Block coefficient	0.8

Table 1. Main particulars of the 44,000 ton tanker

Wave induced extreme vertical bending moment can be obtained using CSR's empirical formula:

$$M_{wv,hog} = f_{prob} 0.19 f_{wv-v} C_{wv} L^2 B C_b$$

© 2014 Optimum Structural Design, Inc.

$$M_{wv,sag} = -f_{prob} 0.11 f_{wv-v} C_{wv} L^2 B (C_b + 0.7)$$

For 150<L<300,

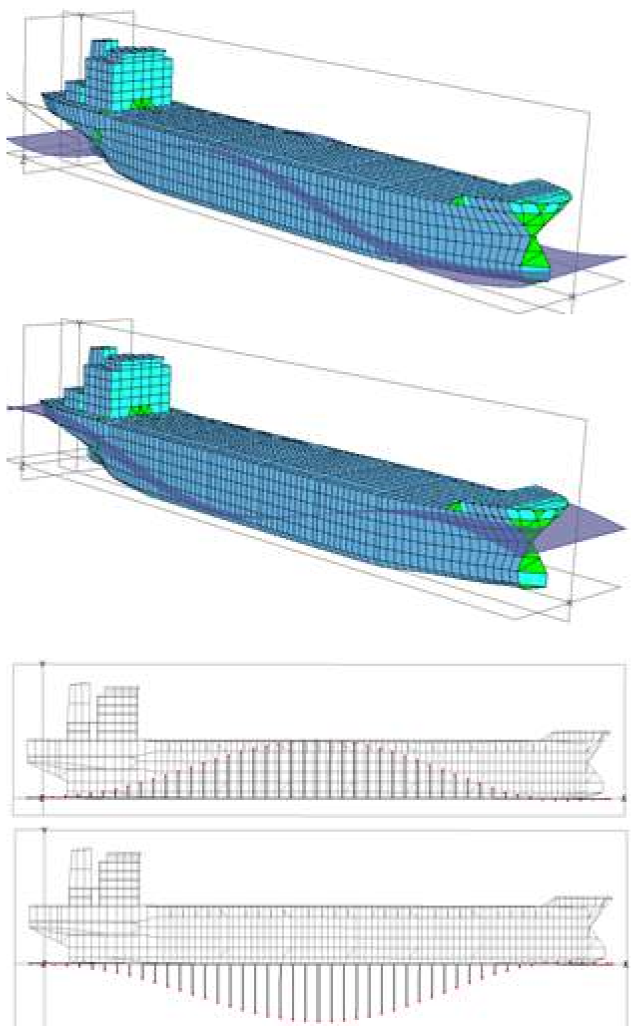
$$C_{wv} = 10.75 - \left(\frac{300 - L}{100}\right)^{\frac{3}{2}}$$

So:

$M_{wv,hog}$=1.496263E6 kNm

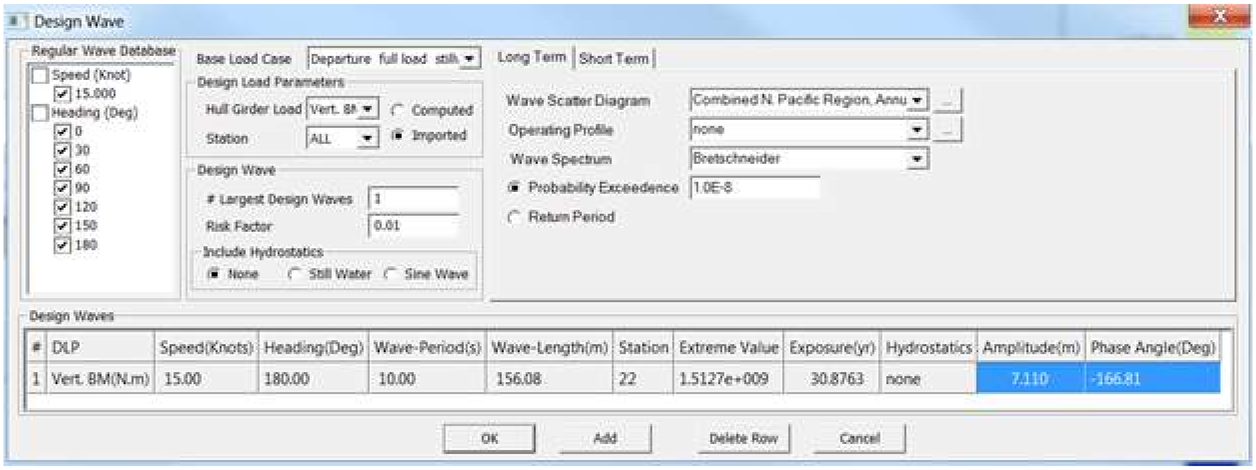
$M_{wv,sag}$=1.624232E6 kNm

For the direct calculation, the ship is assumed to travel at a speed of 15 knots. MAESTRO-Wave was used to generate a unit wave hull girder load database. The design wave for the extreme bending moment with the probability of exceedance of 10E-8 was obtained using the MAESTRO-ELA procedure. The wave scatter diagram used in these calculations is provided here.



© 2014 Optimum Structural Design, Inc.

Hogging **Sagging**



Extreme Vertical BM	CSR (kN*m)	MAESTRO-Wave-ELA(kN*m)
Hogging	1.496263E6	1.5127E6
Sagging	1.624232E6	1.5127E6

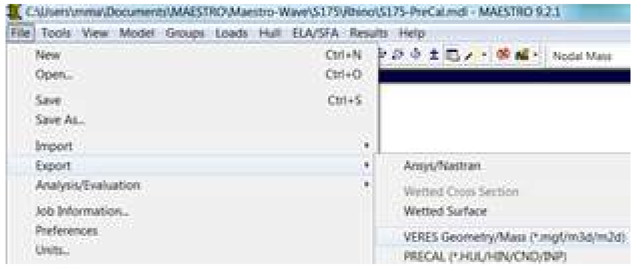
Table 2. Design wave comparison between CSR and MAESTRO-Wave-ELA

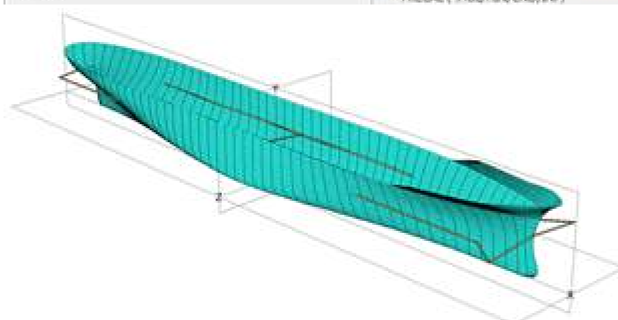
Comparison Procedures of Short Term Statistics and Data

To validate the implementation of the short term statistics, the following procedure was used to compare the numerical calculations between MAESTRO and VERES:

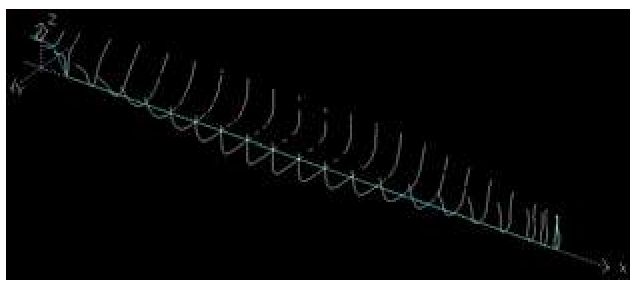
1. From the MAESTRO model, export input data (*.mgf and *.m3d) to VERES
2. Use VERES to calculate the hull girder load RAOs and short term statistics, then import the hull girder load RAOs (*.re3) obtained from VERES into MAESTRO
3. Evaluate the short term statics using MAESTRO
4. Compare the short term statistics between MAESTRO and VERES

Step 1: From a MAESTRO model (S175), export *.mgf and *m3d



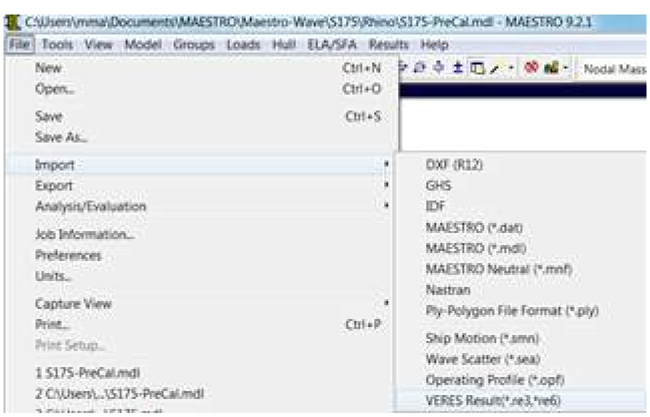


MAESTRO

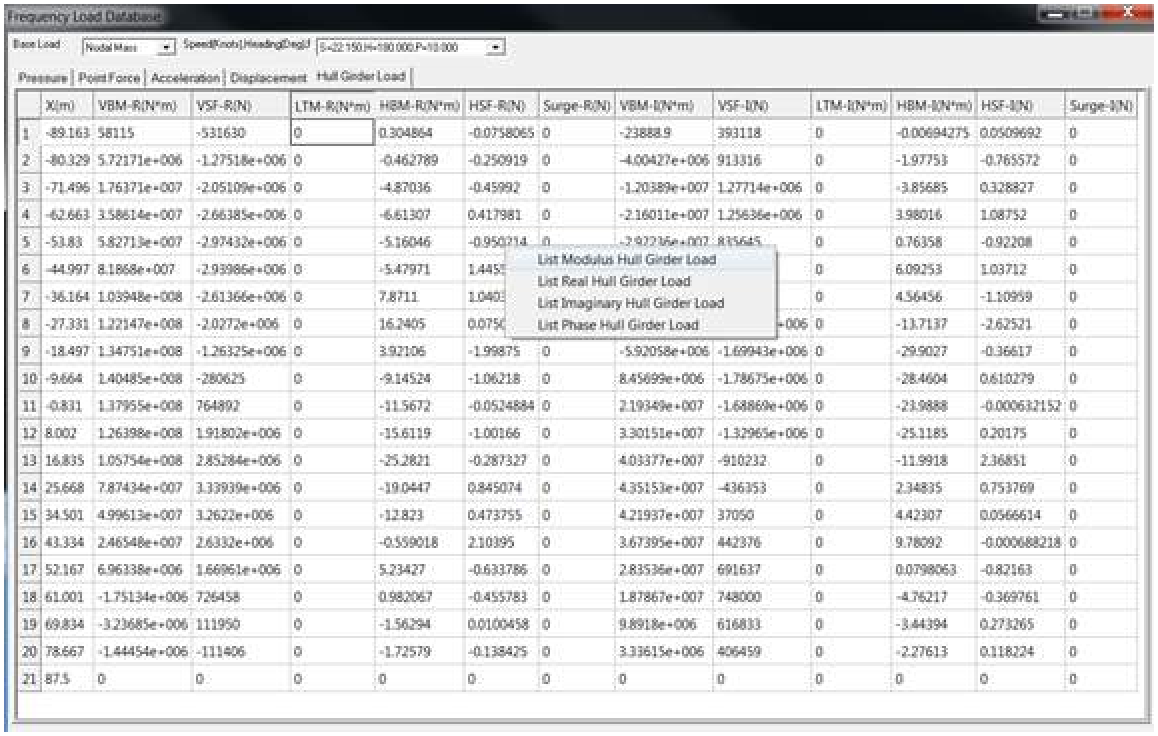


VERES

Step 2: Run VERES, then import VERES's hull girder load RAOs (*.re3) into MAESTRO



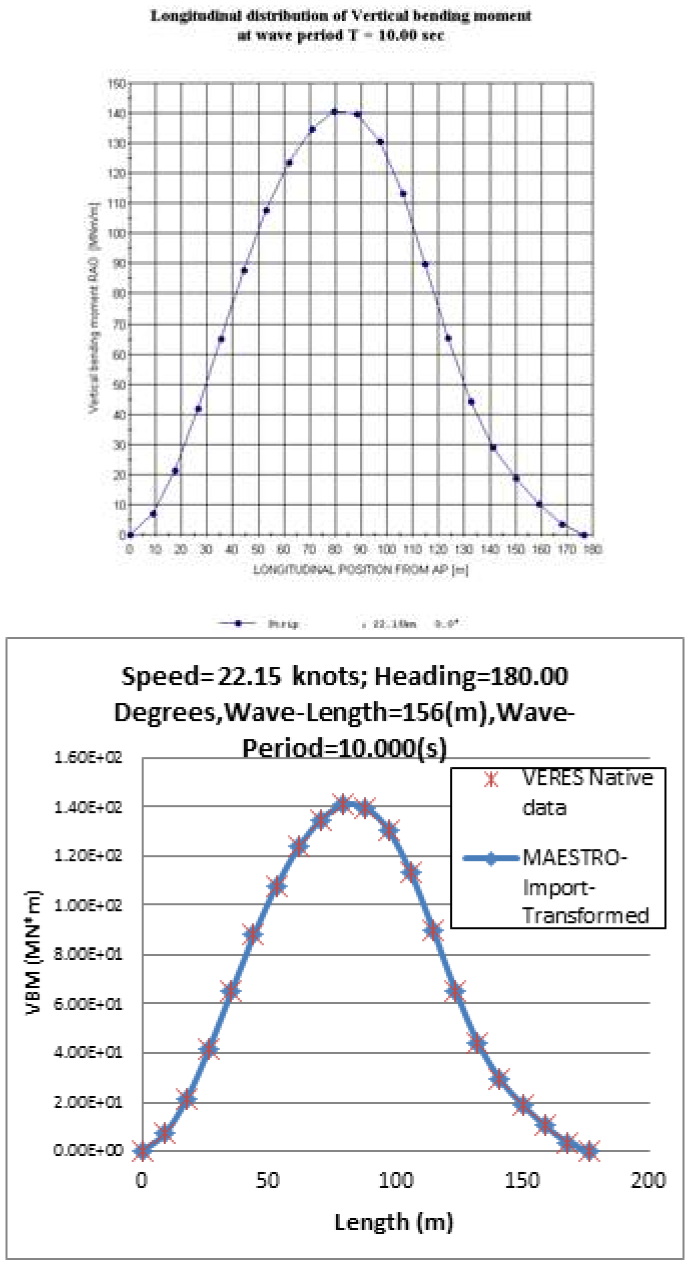
Because of the different definition of the standard coordinate system and forward speed, we need to first validate the hull girder load transformation:



	X(m)	VBM-R(N*m)	VSF-R(N)	LTM-R(N*m)	HBM-R(N*m)	HSF-R(N)	Surge-R(N)	VBM-I(N*m)	VSF-I(N)	LTM-I(N*m)	HBM-I(N*m)	HSF-I(N)	Surge-I(N)
1	-89.163	58115	-531630	0	0.304864	-0.0758065	0	-23888.9	393118	0	-0.00694275	0.0509692	0
2	-80.329	5.72171e+006	-1.27518e+006	0	-0.462789	-0.250919	0	-4.00427e+006	913316	0	-1.97753	-0.765572	0
3	-71.496	1.76371e+007	-2.05109e+006	0	-4.87036	-0.45992	0	-1.20389e+007	1.27714e+006	0	-3.85685	0.328827	0
4	-62.663	3.58614e+007	-2.66385e+006	0	-6.61307	0.417981	0	-2.16011e+007	1.25636e+006	0	3.98016	1.08752	0
5	-53.83	5.82713e+007	-2.97432e+006	0	-5.16046	-0.950214	0	-2.92236e+007	835645	0	0.76358	-0.92208	0
6	-44.997	8.1868e+007	-2.93986e+006	0	-5.47971	[illegible]	[illegible]	[illegible]	[illegible]	0	6.09253	1.03712	0
7	-36.164	1.03948e+008	-2.61366e+006	0	7.8711	[illegible]	[illegible]	[illegible]	[illegible]	0	4.56456	-1.10959	0
8	-27.331	1.22147e+008	-2.0272e+006	0	16.2405	[illegible]	[illegible]	[illegible]	[illegible]	0	-13.7137	-2.62521	0
9	-18.497	1.34751e+008	-1.26325e+006	0	3.92106	-1.99875	0	-5.92058e+006	-1.69943e+006	0	-29.9027	-0.36617	0
10	-9.664	1.40485e+008	-280625	0	-9.14524	-1.06218	0	8.45699e+006	-1.78675e+006	0	-28.4604	0.610279	0
11	-0.831	1.37955e+008	764892	0	-11.5672	-0.0524884	0	2.19349e+007	-1.68869e+006	0	-23.9888	-0.000632152	0
12	8.002	1.26398e+008	1.91802e+006	0	-15.6119	-1.00166	0	3.30151e+007	-1.32965e+006	0	-25.1185	0.20175	0
13	16.835	1.05754e+008	2.85284e+006	0	-25.2821	-0.287327	0	4.03377e+007	-910232	0	-11.9918	2.36851	0
14	25.668	7.87434e+007	3.33939e+006	0	-19.0447	0.845074	0	4.35153e+007	-436353	0	2.34835	0.753769	0
15	34.501	4.99613e+007	3.2622e+006	0	-12.823	0.473755	0	4.21937e+007	37050	0	4.42307	0.0566614	0
16	43.334	2.46548e+007	2.6332e+006	0	-0.559018	2.10395	0	3.67395e+007	442376	0	9.78092	-0.000688218	0
17	52.167	6.96338e+006	1.66961e+006	0	5.23427	-0.633786	0	2.83536e+007	691637	0	0.0798063	-0.82163	0
18	61.001	-1.75134e+006	726458	0	0.982067	-0.455783	0	1.87867e+007	748000	0	-4.76217	-0.369761	0
19	69.834	-3.23685e+006	111950	0	-1.56294	0.0100458	0	9.8918e+006	616833	0	-3.44394	0.273265	0
20	78.667	-1.44454e+006	-111406	0	-1.72579	-0.138425	0	3.33615e+006	406459	0	-2.27613	0.118224	0
21	87.5	0	0	0	0	0	0	0	0	0	0	0	0

VERES's result: Speed= 22.15 knots; Heading=180.00 Degrees, Wave-Period=10.000(s)

Speed= 22.15 knots; Heading=180.00 Degrees, Wave-Length=156(m),Wave-Period=10.000(s)							VERES(*.mpl)
X(m)	FX(N)	VSF(N)	HSF(N)	TM(Nm)	HBM(Nm)	VBM(Nm)	GLOBAL WAVE INDUCED LOADS Longitudinal distribution of Vertical bending moment\nat wave period T = 10.00 sec Project: Strip LONGITUDINAL POSITION FROM AP [m] Vertical bending moment RAO [MNm/m] Strip ; 22.15kn 0.0° 21
-89.163	0	661190	0.091348	0	0.304943	62833.4	176.662003 0.000000e+000
-80.329	0	1.57E+06	0.805643	0	2.03E+00	6.98E+06	167.829010 3.635467e+000
-71.496	0	2.42E+06	0.565379	0	6.21E+00	2.14E+07	158.996002 1.040793e+001
-62.663	0.00E+00	2.95E+06	1.16508	0	7.72E+00	4.19E+07	150.162994 1.886813e+001
-53.83	0.00E+00	3.09E+06	1.32406	0	5.22E+00	6.52E+07	141.330002 2.919614e+001
-44.997	0.00E+00	2.94E+06	1.77911	0	8.19E+00	8.79E+07	132.497009 4.424533e+001
-36.164	0.00E+00	2.69E+06	1.52103	0	9.10E+00	1.08E+08	123.662994 6.539709e+001
-27.331	0.00E+00	2.41E+06	2.62628	0	2.13E+01	1.24E+08	114.830002 8.996996e+001
-18.497	0.00E+00	2.12E+06	2.03201	0	3.02E+01	1.35E+08	105.997002 1.131856e+002
-9.664	0.00E+00	1.81E+06	1.22501	0	2.99E+01	1.41E+08	97.164001 1.306382e+002
-0.831	0.00E+00	1.85E+06	0.052492	0	2.66E+01	1.40E+08	88.331001 1.396876e+002
8.002	0.00E+00	2.33E+06	1.02178	0	2.96E+01	1.31E+08	79.498001 1.407389e+002
16.835	0.00E+00	2.99E+06	2.38588	0	2.80E+01	1.13E+08	70.665001 1.348807e+002
25.668	0.00E+00	3.37E+06	1.13239	0	1.92E+01	9.00E+07	61.832001 1.236251e+002
34.501	0.00E+00	3.26E+06	0.477131	0	1.36E+01	6.54E+07	52.999001 1.077414e+002
43.334	0.00E+00	2.67E+06	2.10395	0	9.80E+00	4.42E+07	44.166000 8.786892e+001
52.167	0.00E+00	1.81E+06	1.03767	0	5.23E+00	2.92E+07	35.332001 6.518555e+001
61.001	0.00E+00	1.04E+06	0.586908	0	4.86E+00	1.89E+07	26.499001 4.186457e+001
69.834	0.00E+00	626910	0.273449	0	3.78E+00	1.04E+07	17.666000 2.135419e+001
78.667	0.00E+00	421450	0.18204	0	2.86E+00	3.64E+06	8.833000 6.983705e+000
87.5	0	0	0	0	0	0	0.000000 6.283337e-002



Validation of hull girder load transformation

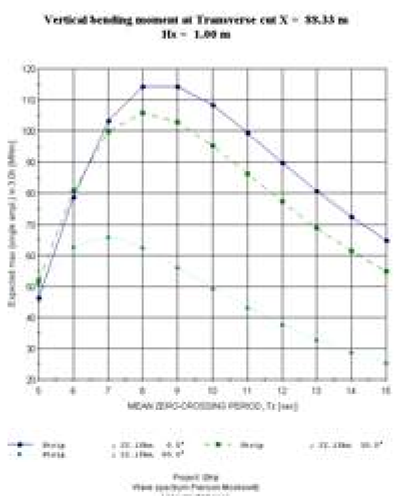
VERES hull girder load output (*.re3)

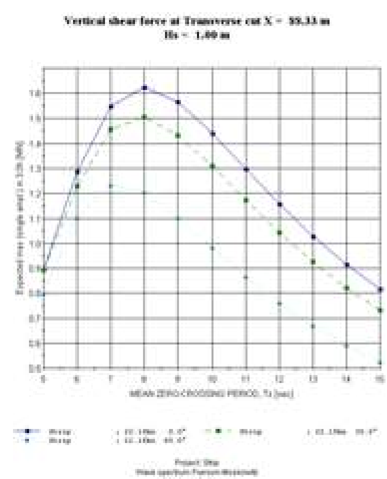
```
Loading condition description: Design waterline

ShipX exported data
0.10250000E+04 0.98100004E+01
0.17500000E+03 0.25400000E+02 0.95000000E+01
0.25521638E+01 0.95000114E+01
0.24589460E+08 10.7471 36.0770 37.7174 1.5489
1 2 2
21 0
0 1 1 1 1 1
0.0000 0.0000
-89.1630
-80.3290
-71.4960
-62.6630
-53.8300
-44.9970
-36.1640
-27.3310
-18.4970
-9.6640
-0.8310
8.0020
16.8350
25.6680
34.5010
43.3340
52.1670
61.0010
69.8340
78.6670
87.5000
1 1 1
6.173333 0.0000000E+00 0.0000000E+00 2.552164 9.500000
0.00
0.5235988
(0.0000000E+00,0.0000000E+00) (0.0000000E+00,0.0000000E+00)
(0.0000000E+00,0.0000000E+00)
(2.9278616E-04,-1.0993685E-04) (-1.8454379E-09,-5.2993437E-10)
(6.6216243E-04,3.6967665E-04)
(5.1601825E-04,-2.9952833E-04) (-6.6783681E-09,1.6185717E-09)
(2.6169475E-03,1.3457955E-03)
(5.4996356E-04,-4.8918091E-04) (2.0920536E-09,-3.7759862E-09)
(4.8721568E-03,2.2767528E-03)
(2.7487063E-04,-6.9205387E-04) (4.4208548E-09,-2.5479796E-09)
(6.9558271E-03,3.0020671E-03)
(-3.1623701E-04,-8.5914182E-04) (2.9644256E-09,1.0749742E-08)
(8.3225975E-03,3.2373371E-03)
(-1.1267348E-03,-8.9411967E-04) (-2.1466644E-09,9.3044701E-09)
(8.6264079E-03,2.7909027E-03)
(-2.0068453E-03,-7.0888991E-04) (-5.6307727E-09,9.3655839E-10)
(7.7891247E-03,1.6865621E-03)
(-2.8119758E-03,-2.5684034E-04) (7.6665729E-10,-1.1113070E-09)
(5.9814430E-03,1.6664337E-04)
(-3.3574887E-03,4.6942133E-04) (2.0180618E-10,-3.1927692E-09)
(3.5340027E-03,-1.4063178E-03)
(-3.6380005E-03,1.3604547E-03) (6.2689764E-09,-1.1599233E-09)
(7.9582509E-04,-2.6489859E-03)
(-3.7343407E-03,2.2991067E-03) (2.2751414E-09,1.1482131E-09)
(-1.9480305E-03,-3.2877352E-03)
(-3.6831100E-03,3.1887067E-03) (-1.4027158E-09,-1.0879617E-09)
(-4.4871401E-03,-3.2296702E-03)
(-3.4928671E-03,3.9092861E-03) (1.3068827E-09,-3.2540275E-09)
(-6.5681450E-03,-2.5775346E-03)
(-3.2476222E-03,4.3408046E-03) (2.8426259E-09,-1.6119713E-09)
(-7.8658685E-03,-1.5777572E-03)
(-3.0614564E-03,4.3869969E-03) (-3.4056080E-09,-1.7949733E-09)
(-8.1349127E-03,-5.1186705E-04)
(-3.0641919E-03,4.0294547E-03) (8.1236601E-10,-6.4131922E-09)
(-7.3043732E-03,3.1714491E-04)
(-3.3327376E-03,3.3218495E-03) (5.3662772E-09,7.5477519E-10)
(-5.5535613E-03,6.6439860E-04)
(-3.8599116E-03,2.3976176E-03) (1.1874810E-09,-9.4807184E-10)
(-3.3636054E-03,4.3253604E-04)
(-4.6241558E-03,1.2959157E-03) (-9.2287333E-10,-5.8274041E-10)
(-1.3772559E-03,-1.2683733E-04)
(-5.4090121E-03,3.6898727E-04) (8.1908363E-10,5.8054200E-10)
(-7.4647163E-05,-2.7826574E-04)
(0.0000000E+00,0.0000000E+00) (0.0000000E+00,0.0000000E+00)
(0.0000000E+00,0.0000000E+00)
(6.2992965E-11,1.7610052E-11) (1.7806091E-05,1.6310177E-06)
(2.1644369E-11,5.9500710E-11)
(2.3037278E-10,-3.9732627E-11) (9.2772163E-05,3.0600451E-05)
(2.0652112E-10,1.7508304E-11)
(-8.2977222E-11,1.3332574E-10) (2.6316993E-04,1.0279805E-04)
(4.7777038E-10,2.4102845E-11)
(-1.6335887E-10,3.8373058E-11) (5.3349003E-04,2.1750382E-04)
(3.0113223E-10,1.6646673E-10)
(-5.9925079E-11,-6.0073463E-10) (8.8210101E-04,3.5988213E-04)
(7.9465393E-11,1.8254745E-10)
(1.4964649E-10,-5.1905896E-10) (1.2682551E-03,5.0479476E-04)
(-5.7614604E-11,-4.3306056E-10)
(4.6976317E-10,3.6861528E-11) (1.6401167E-03,6.1986136E-04)
(2.1607859E-10,-9.6233022E-10)
(2.0286178E-10,1.0826677E-10) (1.9472582E-03,6.7767146E-04)
(5.6839455E-10,-1.0647837E-09)
```

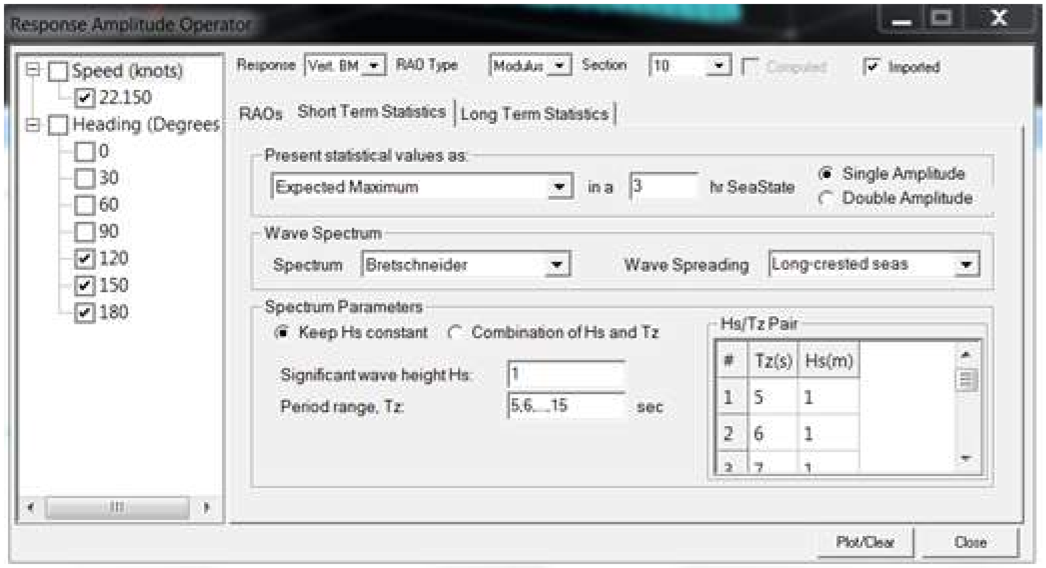
© 2014 Optimum Structural Design, Inc.

VERES's short term statistics output	
1	1
SHORT TERM STATISTICS	SHORT TERM STATISTICS
Vertical bending moment at Transverse cut X = 88.33 m\nHs = 1.00 m	Vertical shear force at Transverse cut X = 88.33 m\nHs = 1.00 m
Project: Strip\nWave spectrum Pierson-Moskowitz \nLong-crested seas	Project: Strip\nWave spectrum Pierson-Moskowitz \nLong-crested seas
MEAN ZERO-CROSSING PERIOD, Tz [sec]	MEAN ZERO-CROSSING PERIOD, Tz [sec]
Expected max (single ampl.) in 3.0h [MNm]	Expected max (single ampl.) in 3.0h [MN]
3	3
Strip ; 22.15kn 0.0°	Strip ; 22.15kn 0.0°
Strip ; 22.15kn 30.0°	Strip ; 22.15kn 30.0°
Strip ; 22.15kn 60.0°	Strip ; 22.15kn 60.0°
11	11
5.000000 4.628933e+001	5.000000 8.960674e-001
6.000000 7.873611e+001	6.000000 1.287121e+000
7.000000 1.032626e+002	7.000000 1.548043e+000
8.000000 1.142099e+002	8.000000 1.623498e+000
9.000000 1.143639e+002	9.000000 1.564158e+000
10.000000 1.082660e+002	10.000000 1.440573e+000
11.000000 9.944653e+001	11.000000 1.296750e+000
12.000000 8.995063e+001	12.000000 1.156063e+000
13.000000 8.078248e+001	13.000000 1.027967e+000
14.000000 7.238907e+001	14.000000 9.147770e-001
15.000000 6.488220e+001	15.000000 8.161898e-001
11	11
5.000000 5.180000e+001	5.000000 8.918413e-001
6.000000 8.098574e+001	6.000000 1.229682e+000
7.000000 9.982944e+001	7.000000 1.455018e+000
8.000000 1.057945e+002	8.000000 1.503943e+000
9.000000 1.028170e+002	9.000000 1.432619e+000
10.000000 9.537379e+001	10.000000 1.309156e+000
11.000000 8.634589e+001	11.000000 1.173150e+000
12.000000 7.727655e+001	12.000000 1.042496e+000
13.000000 6.886210e+001	13.000000 9.251044e-001
14.000000 6.133758e+001	14.000000 8.220065e-001
15.000000 5.472401e+001	15.000000 7.325537e-001
11	11
5.000000 5.110350e+001	5.000000 7.975678e-001
6.000000 6.292495e+001	6.000000 1.103233e+000
7.000000 6.598369e+001	7.000000 1.234206e+000
8.000000 6.257636e+001	8.000000 1.205639e+000
9.000000 5.637862e+001	9.000000 1.104630e+000
10.000000 4.961499e+001	10.000000 9.836690e-001
11.000000 4.329860e+001	11.000000 8.664744e-001
12.000000 3.775369e+001	12.000000 7.616724e-001
13.000000 3.301147e+001	13.000000 6.706778e-001
14.000000 2.899287e+001	14.000000 5.924718e-001
15.000000 2.560390e+001	15.000000 5.252566e-001





Step 3: Evaluate the short term statics using MAESTRO



Stationat -0.831 (m)
Tp(s) Tz(s) Tm(s) Vert. BM(MN.m)(Expected Maximum Value,3 hours exposure,Single Amplitude)
==>[Speed=11.3949(m/s),Heading=120.00]

Tp(s)	Tz(s)	Tm(s)	Vert. BM(MN.m)
7.035	5.000	5.430	51.6663
8.442	6.000	6.516	63.418
9.849	7.000	7.602	66.4764
11.256	8.000	8.688	63.0189
12.663	9.000	9.774	56.7502
14.070	10.000	10.860	49.9362
15.477	11.000	11.946	43.5731
16.884	12.000	13.032	37.9874
18.291	13.000	14.118	33.2133
19.698	14.000	15.204	29.1767
21.105	15.000	16.290	25.7743

Stationat -0.831 (m)
Tp(s) Tz(s) Tm(s) Vert. BM(MN.m)(Expected Maximum Value,3 hours exposure,Single Amplitude)
==>[Speed=11.3949(m/s),Heading=150.00]

Tp(s)	Tz(s)	Tm(s)	Vert. BM(MN.m)
7.035	5.000	5.430	52.8213
8.442	6.000	6.516	81.7938
9.849	7.000	7.602	100.567
11.256	8.000	8.688	106.487
12.663	9.000	9.774	103.434
14.070	10.000	10.860	95.8955
15.477	11.000	11.946	86.8029
16.884	12.000	13.032	77.6837
18.291	13.000	14.118	69.2166
19.698	14.000	15.204	61.6419
21.105	15.000	16.290	54.9907

Stationat -0.831 (m)
Tp(s) Tz(s) Tm(s) Vert. BM(MN.m)(Expected Maximum Value,3 hours exposure,Single Amplitude)
==>[Speed=11.3949(m/s),Heading=180.00]

Tp(s)	Tz(s)	Tm(s)	Vert. BM(MN.m)
7.035	5.000	5.430	47.0364
8.442	6.000	6.516	79.4232
9.849	7.000	7.602	103.92
11.256	8.000	8.688	114.886
12.663	9.000	9.774	114.954
14.070	10.000	10.860	108.799
15.477	11.000	11.946	99.9402
16.884	12.000	13.032	90.4011
18.291	13.000	14.118	81.1864
19.698	14.000	15.204	72.7309
21.105	15.000	16.290	65.1753

Stationat -0.831 (m)
Tp(s) Tz(s) Tm(s) Vert. SF(MN)(Expected Maximum Value,3 hours exposure,Single Amplitude)
==>[Speed=11.3949(m/s),Heading=120.00]

Tp(s)	Tz(s)	Tm(s)	Vert. SF(MN)
7.035	5.000	5.430	0.809491
8.442	6.000	6.516	1.11451
9.849	7.000	7.602	1.24572
11.256	8.000	8.688	1.2163
12.663	9.000	9.774	1.11368
14.070	10.000	10.860	0.991438
15.477	11.000	11.946	0.873247
16.884	12.000	13.032	0.767418
18.291	13.000	14.118	0.675627
19.698	14.000	15.204	0.597073
21.105	15.000	16.290	0.530185

Stationat -0.831 (m)
Tp(s) Tz(s) Tm(s) Vert. SF(MN)(Expected Maximum Value,3 hours exposure,Single Amplitude)
==>[Speed=11.3949(m/s),Heading=150.00]

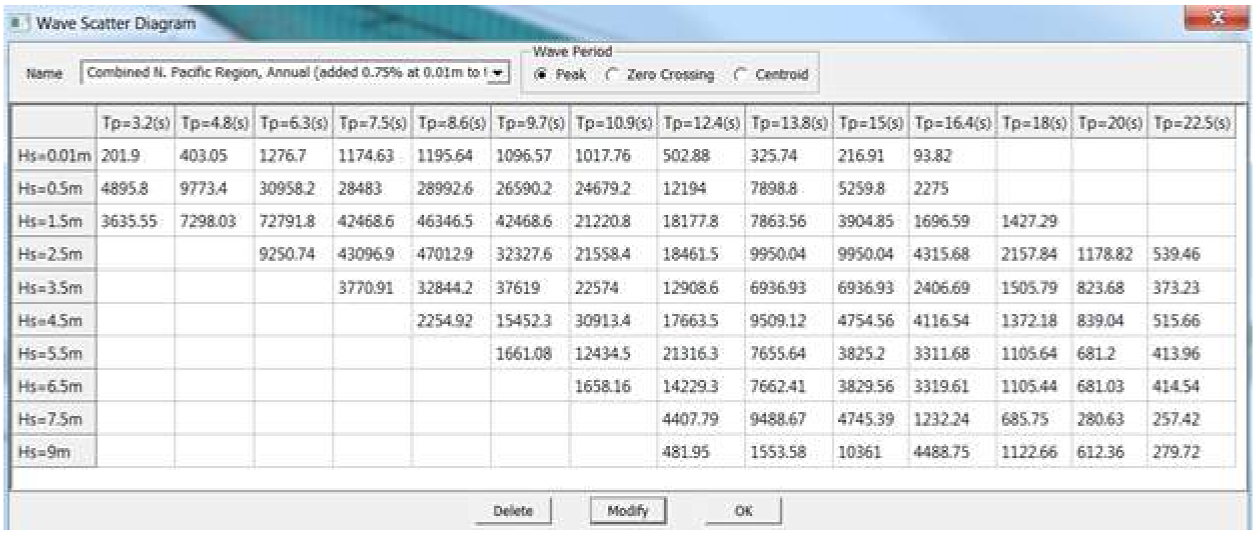
Tp(s)	Tz(s)	Tm(s)	Vert. SF(MN)
7.035	5.000	5.430	0.911049
8.442	6.000	6.516	1.24261
9.849	7.000	7.602	1.46794
11.256	8.000	8.688	1.51567
12.663	9.000	9.774	1.44321
14.070	10.000	10.860	1.31818
15.477	11.000	11.946	1.18071
16.884	12.000	13.032	1.0493
18.291	13.000	14.118	0.930848
19.698	14.000	15.204	0.826972
21.105	15.000	16.290	0.737064

Stationat -0.831 (m)
Tp(s) Tz(s) Tm(s) Vert. SF(MN)(Expected Maximum Value,3 hours exposure,Single Amplitude)
==>[Speed=11.3949(m/s),Heading=180.00]

Tp(s)	Tz(s)	Tm(s)	Vert. SF(MN)
7.035	5.000	5.430	0.906277
8.442	6.000	6.516	1.29622
9.849	7.000	7.602	1.5582
11.256	8.000	8.688	1.63398
12.663	9.000	9.774	1.57389
14.070	10.000	10.860	1.44866
15.477	11.000	11.946	1.3041
16.884	12.000	13.032	1.16275
18.291	13.000	14.118	1.03371
19.698	14.000	15.204	0.91967
21.105	15.000	16.290	0.820471

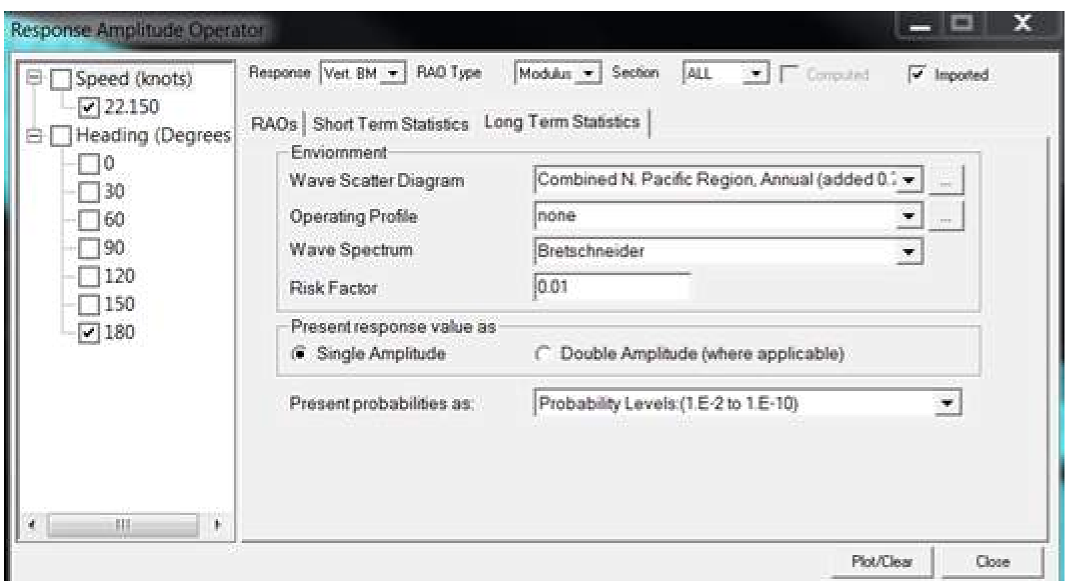
MAESTRO output: Expected maximum value (3 hrs)

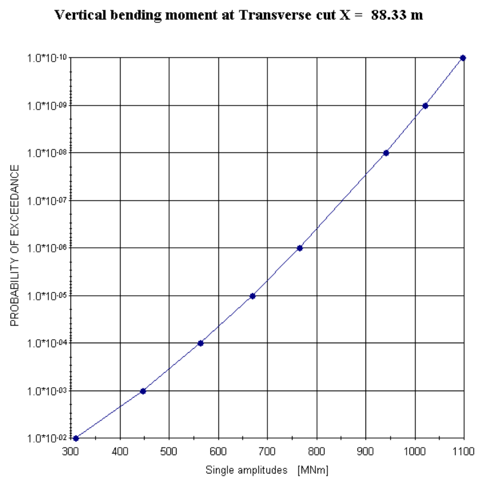
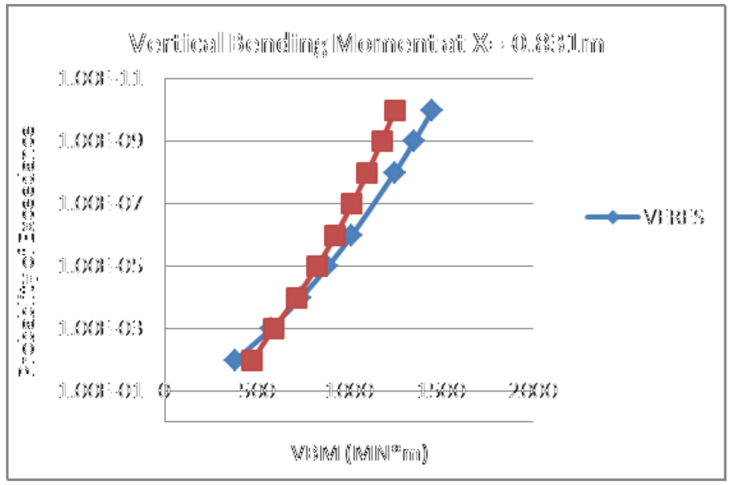
Comparison Procedures of Long Term Statistics and Design Waves



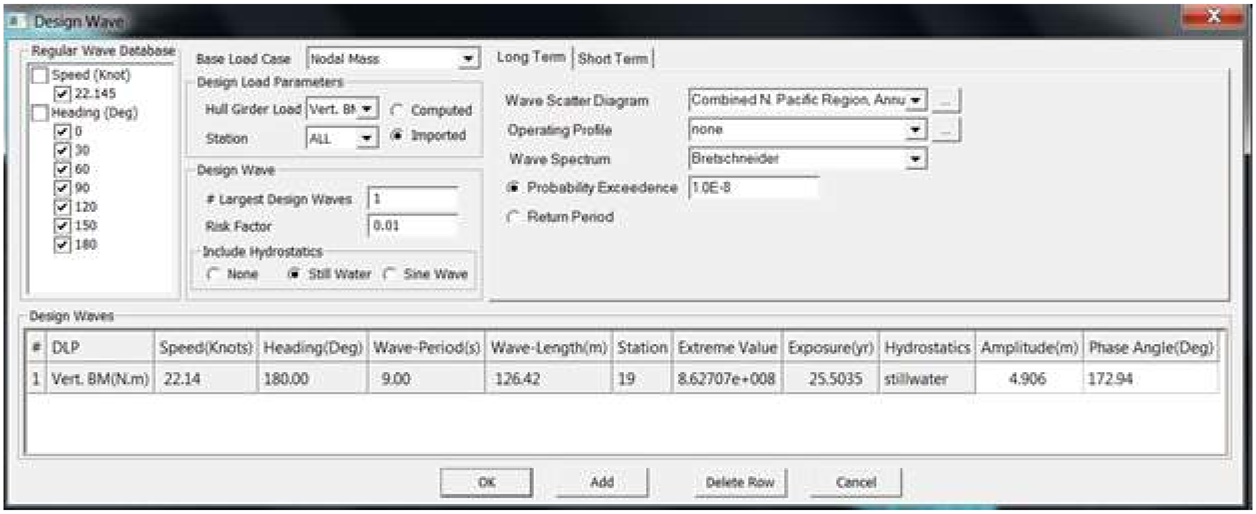
	Tp=3.2(s)	Tp=4.8(s)	Tp=6.3(s)	Tp=7.5(s)	Tp=8.6(s)	Tp=9.7(s)	Tp=10.9(s)	Tp=12.4(s)	Tp=13.8(s)	Tp=15(s)	Tp=16.4(s)	Tp=18(s)	Tp=20(s)	Tp=22.5(s)
Hs=0.01m	201.9	403.05	1276.7	1174.63	1195.64	1096.57	1017.76	502.88	325.74	216.91	93.82			
Hs=0.5m	4895.8	9773.4	30958.2	28483	28992.6	26590.2	24679.2	12194	7898.8	5259.8	2275			
Hs=1.5m	3635.55	7298.03	72791.8	42468.6	46346.5	42468.6	21220.8	18177.8	7863.56	3904.85	1696.59	1427.29		
Hs=2.5m			9250.74	43096.9	47012.9	32327.6	21558.4	18461.5	9950.04	9950.04	4315.68	2157.84	1178.82	539.46
Hs=3.5m				3770.91	32844.2	37619	22574	12908.6	6936.93	6936.93	2406.69	1505.79	823.68	373.23
Hs=4.5m					2254.92	15452.3	30913.4	17663.5	9509.12	4754.56	4116.54	1372.18	839.04	515.66
Hs=5.5m						1661.08	12434.5	21316.3	7655.64	3825.2	3311.68	1105.64	681.2	413.96
Hs=6.5m							1658.16	14229.3	7662.41	3829.56	3319.61	1105.44	681.03	414.54
Hs=7.5m								4407.79	9488.67	4745.39	1232.24	685.75	280.63	257.42
Hs=9m								481.95	1553.58	10361	4488.75	1122.66	612.36	279.72

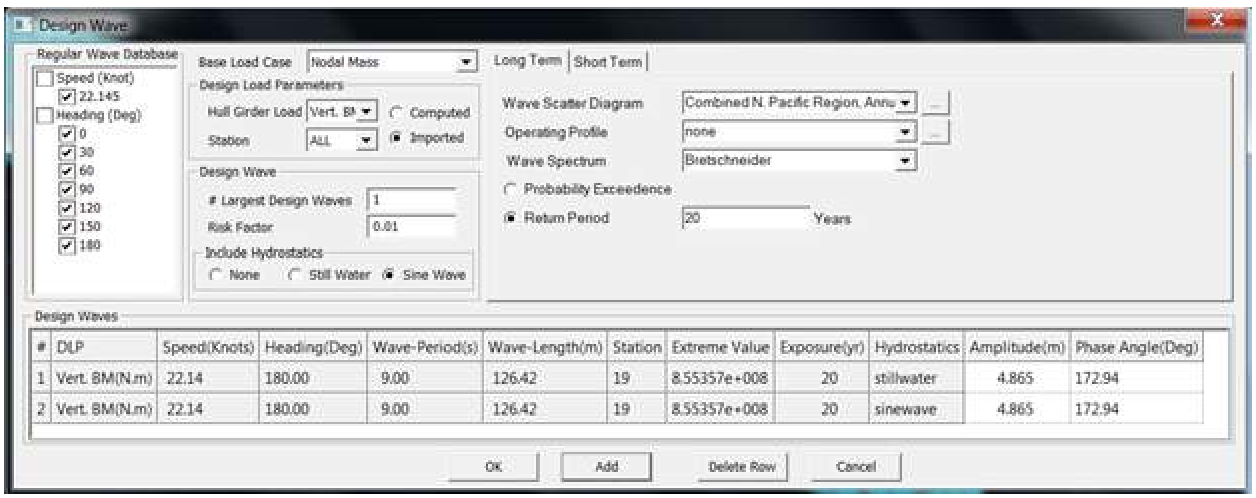
Scatter Diagram



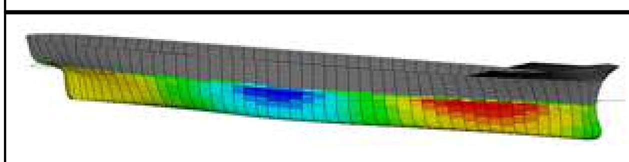
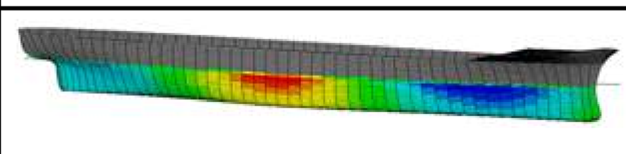
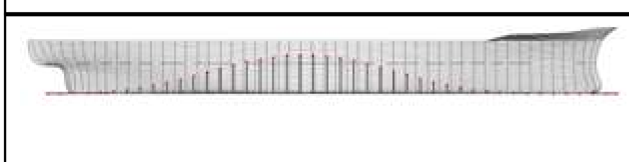
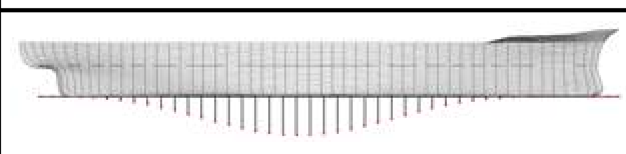
VERES	MAESTRO
	
LONG TERM STATISTICS Vertical bending moment at Transverse cut X = 88.33 m Project: Strip\nCombined N. Pacific Region, Annual (added 0.75% at 0.01m to total 100%)\nWave spectrum Pierson-Moskowitz \nLong-crested seas\nIndependent headings. Single amplitudes [MNm] PROBABILITY OF EXCEEDANCE 1 Strip ; 22.15kn 180.0°; T_R = 36.13 [s], Weibull slope, h = 1.2842 [-] 8 309.759369 1.000000e-002 445.872803 1.000000e-003 564.172180 1.000000e-004 669.540161 1.000000e-005 765.747314 1.000000e-006 939.869995 1.000000e-008 1020.297852 1.000000e-009 1097.056519 1.000000e-010	Station=-0.831(m), Speed=22.15(Knots),Heading= 180.00 DEG Exceedence Extreme-Value 0.01 4.77771e+008 0.001 5.92224e+008 0.0001 7.15364e+008 1e-005 8.293e+008 1e-006 9.29434e+008 1e-007 1.01981e+009 1e-008 1.10282e+009 1e-009 1.18001e+009 1e-010 1.25246e+009

Long term design wave validation





S175: Dynamic Only

Sagging	Hogging
	
	

Load Case =6, "Extreme6"

Station	X(m)	Bending Moment (N*m)
0	-87.5	205159
1	-83.25	327658
2	-79	2.53624e+006
3	-74.75	8.24339e+006
4	-70.5	2.21185e+007
5	-66.25	4.45718e+007
6	-62	7.77594e+007
7	-57.75	1.18727e+008
8	-53.5	1.74835e+008
9	-49.25	2.39426e+008
10	-45	3.13038e+008
11	-40.75	3.91612e+008
12	-36.5	4.73895e+008
13	-32.25	5.55303e+008
14	-28	6.34343e+008
15	-23.75	7.04733e+008
16	-19.5	7.64713e+008
17	-15.25	8.10706e+008
18	-11	8.41269e+008
19	-6.75	8.54534e+008
20	-2.5	8.48414e+008
21	1.75	8.22708e+008
22	6	7.79692e+008
23	10.25	7.23153e+008
24	14.5	6.53821e+008
25	18.75	5.76905e+008
26	23	4.93515e+008
27	27.25	4.11058e+008
28	31.5	3.30191e+008
29	35.75	2.55918e+008
30	40	1.8852e+008
31	44.25	1.31872e+008
32	48.5	8.46629e+007
33	52.75	5.27907e+007
34	57	2.69063e+007
35	61.25	1.08585e+007
36	65.5	219655
37	69.75	-4.15442e+006
38	74	-5.15871e+006
39	78.25	-4.36895e+006
40	82.5	-3.48765e+006
41	86.75	-2.21861e+006
42	91	-1.75749e+006
43	94.423	-1.8179e+006

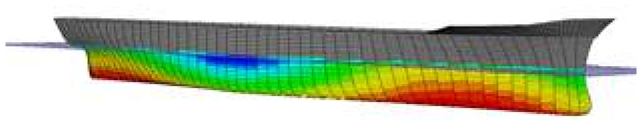
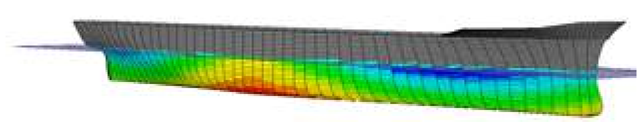
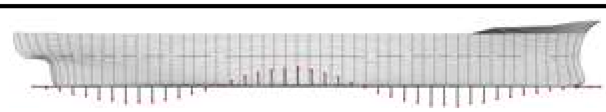
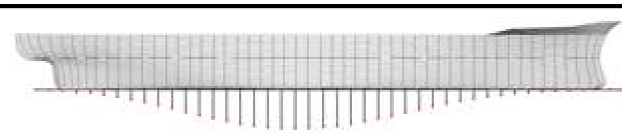
Maximum Bending Moment (Full Ship) in Structural Coordinates:
M=8.54534e+008(Nm)

Load Case =7, "Extreme6+180"

Station	X(m)	Bending Moment (N*m)
0	-87.5	-205159
1	-83.25	-327659
2	-79	-2.53624e+006
3	-74.75	-8.24339e+006
4	-70.5	-2.21185e+007
5	-66.25	-4.45718e+007
6	-62	-7.77594e+007
7	-57.75	-1.18727e+008
8	-53.5	-1.74835e+008
9	-49.25	-2.39426e+008
10	-45	-3.13038e+008
11	-40.75	-3.91612e+008
12	-36.5	-4.73895e+008
13	-32.25	-5.55303e+008
14	-28	-6.34343e+008
15	-23.75	-7.04733e+008
16	-19.5	-7.64713e+008
17	-15.25	-8.10706e+008
18	-11	-8.41269e+008
19	-6.75	-8.54534e+008
20	-2.5	-8.48414e+008
21	1.75	-8.22708e+008
22	6	-7.79692e+008
23	10.25	-7.23153e+008
24	14.5	-6.53821e+008
25	18.75	-5.76905e+008
26	23	-4.93515e+008
27	27.25	-4.11058e+008
28	31.5	-3.30191e+008
29	35.75	-2.55918e+008
30	40	-1.8852e+008
31	44.25	-1.31872e+008
32	48.5	-8.46629e+007
33	52.75	-5.27907e+007
34	57	-2.69063e+007
35	61.25	-1.08585e+007
36	65.5	-219662
37	69.75	4.15441e+006
38	74	5.1587e+006
39	78.25	4.36894e+006
40	82.5	3.48764e+006
41	86.75	2.2186e+006
42	91	1.75748e+006
43	94.423	1.81789e+006

Maximum Bending Moment (Full Ship) in Structural Coordinates:
M=-8.54534e+008(Nm)

S175: Dynamic + Stillwater

Sagging	Hogging
	
	

Load Case =2, "Extreme2"

Station	X(m)	Bending Moment(N*m)
0	-87.5	228765
1	-83.25	-1.74478e+007
2	-79	-3.45041e+007
3	-74.75	-6.7919e+007
4	-70.5	-8.81071e+007
5	-66.25	-1.19532e+008
6	-62	-1.32021e+008
7	-57.75	-1.58248e+008
8	-53.5	-1.51691e+008
9	-49.25	-1.50235e+008
10	-45	-1.25755e+008
11	-40.75	-1.04325e+008
12	-36.5	-6.54408e+007
13	-32.25	-3.07289e+007
14	-28	1.58479e+007
15	-23.75	5.69865e+007
16	-19.5	1.00997e+008
17	-15.25	1.34296e+008
18	-11	1.58932e+008
19	-6.75	1.75542e+008
20	-2.5	1.8102e+008
21	1.75	1.59776e+008
22	6	1.22504e+008
23	10.25	8.80165e+007
24	14.5	3.64368e+007
25	18.75	-4.13126e+006
26	23	-5.7696e+007
27	27.25	-8.95541e+007
28	31.5	-1.34053e+008
29	35.75	-1.52314e+008
30	40	-1.80504e+008
31	44.25	-1.78415e+008
32	48.5	-1.8701e+008
33	52.75	-1.57909e+008
34	57	-1.49241e+008
35	61.25	-1.16222e+008
36	65.5	-1.01747e+008
37	69.75	-7.09816e+007
38	74	-5.45563e+007
39	78.25	-3.21274e+007
40	82.5	-2.11692e+007
41	86.75	-7.04854e+006
42	91	635005
43	94.423	574342

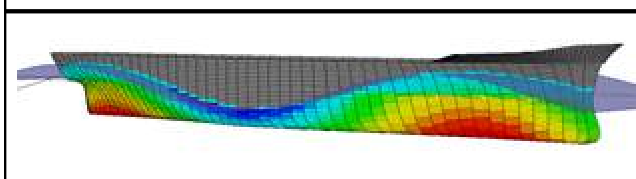
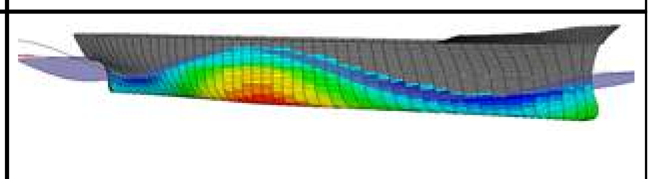
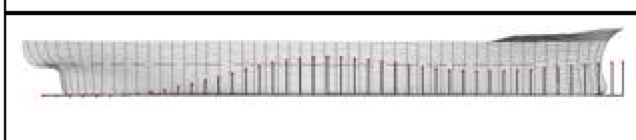
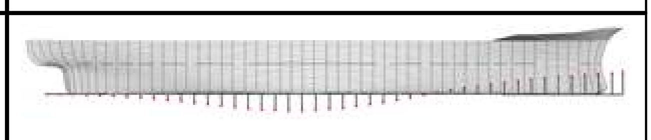
Maximum Bending Moment (Full Ship) in Structural Coordinates:
M=-1.8701e+008(Nm)

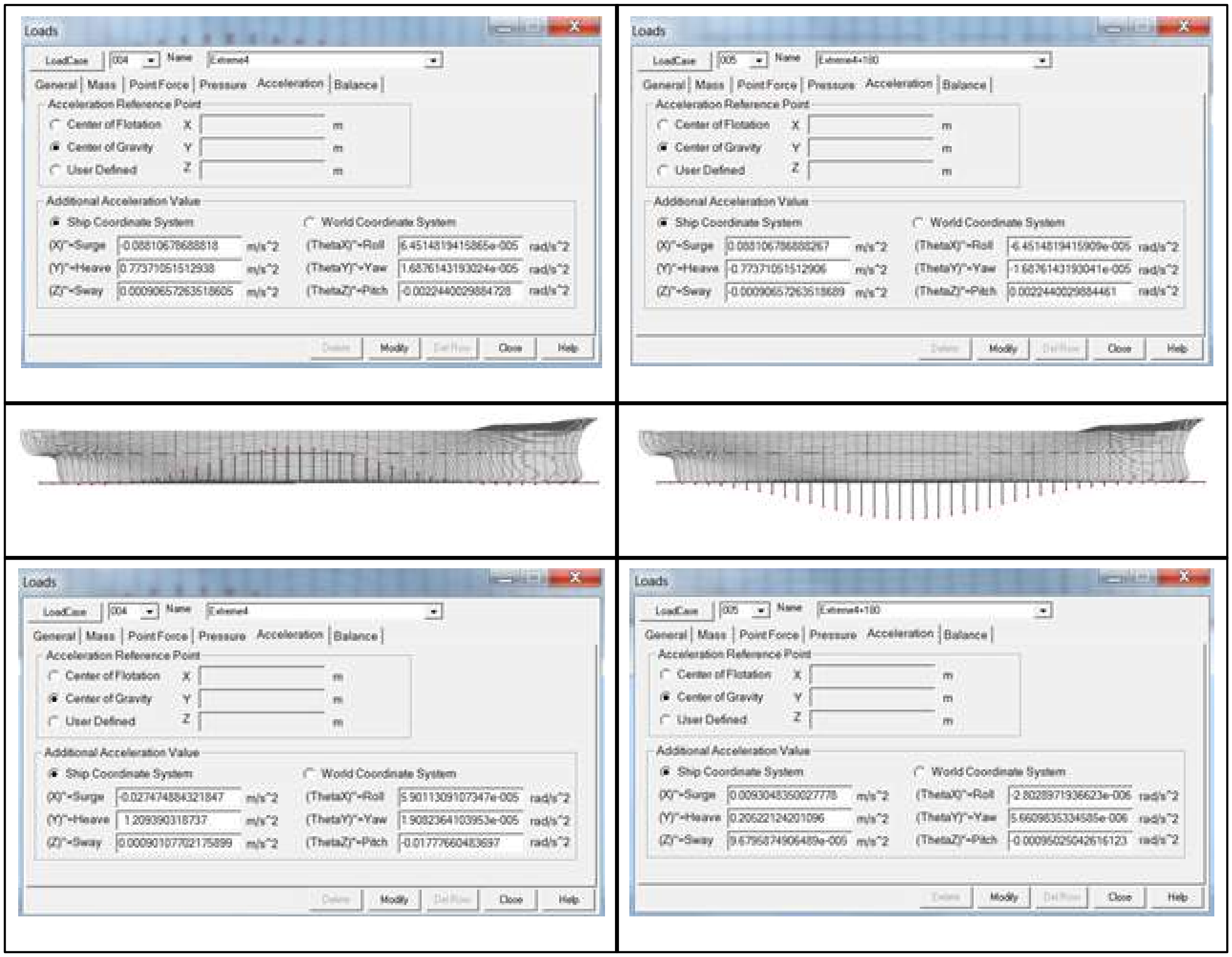
Load Case =3, "Extreme2+180"

Station	X(m)	Bending Moment(N*m)
0	-87.5	-181553
1	-83.25	-1.81032e+007
2	-79	-3.95766e+007
3	-74.75	-8.44058e+007
4	-70.5	-1.32344e+008
5	-66.25	-2.08676e+008
6	-62	-2.8754e+008
7	-57.75	-3.95702e+008
8	-53.5	-5.01362e+008
9	-49.25	-6.29086e+008
10	-45	-7.51831e+008
11	-40.75	-8.87549e+008
12	-36.5	-1.01323e+009
13	-32.25	-1.14134e+009
14	-28	-1.25284e+009
15	-23.75	-1.35248e+009
16	-19.5	-1.42843e+009
17	-15.25	-1.48712e+009
18	-11	-1.52361e+009
19	-6.75	-1.53353e+009
20	-2.5	-1.51581e+009
21	1.75	-1.48564e+009
22	6	-1.43688e+009
23	10.25	-1.35829e+009
24	14.5	-1.27121e+009
25	18.75	-1.15794e+009
26	23	-1.04473e+009
27	27.25	-9.1167e+008
28	31.5	-7.94435e+008
29	35.75	-6.6415e+008
30	40	-5.57545e+008
31	44.25	-4.42159e+008
32	48.5	-3.56335e+008
33	52.75	-2.63491e+008
34	57	-2.03053e+008
35	61.25	-1.37939e+008
36	65.5	-1.02187e+008
37	69.75	-6.26728e+007
38	74	-4.42389e+007
39	78.25	-2.33895e+007
40	82.5	-1.4194e+007
41	86.75	-2.61133e+006
42	91	4.14998e+006
43	94.423	4.21013e+006

Maximum Bending Moment (Full Ship) in Structural Coordinates:
M=-1.53353e+009(Nm)

S175: Dynamic + Sine Wave

Sagging	Hogging
	
	


Loads
LoadCase 004 Name Extreme4
General | Mass | Point Force | Pressure Acceleration | Balance
Acceleration Reference Point
Center of Flotation X m
Center of Gravity Y m
User Defined Z m
Additional Acceleration Value
Ship Coordinate System
World Coordinate System
(X)''=Surge -0.08810678688818 m/s^2
(Y)''=Heave 0.77371051512938 m/s^2
(Z)''=Sway 0.00090657263518605 m/s^2
(ThetaX)''=Roll 6.4514819415865e-005 rad/s^2
(ThetaY)''=Yaw 1.6876143193024e-005 rad/s^2
(ThetaZ)''=Pitch -0.0022440029884728 rad/s^2
Delete Modify Del Row Close Help
Loads
LoadCase 005 Name Extreme4+180
General | Mass | Point Force | Pressure Acceleration | Balance
Acceleration Reference Point
Center of Flotation X m
Center of Gravity Y m
User Defined Z m
Additional Acceleration Value
Ship Coordinate System
World Coordinate System
(X)''=Surge 0.088106786888267 m/s^2
(Y)''=Heave -0.77371051512906 m/s^2
(Z)''=Sway -0.00090657263518689 m/s^2
(ThetaX)''=Roll -6.4514819415909e-005 rad/s^2
(ThetaY)''=Yaw -1.6876143193041e-005 rad/s^2
(ThetaZ)''=Pitch 0.0022440029884461 rad/s^2
Delete Modify Del Row Close Help
Loads
LoadCase 004 Name Extreme4
General | Mass | Point Force | Pressure Acceleration | Balance
Acceleration Reference Point
Center of Flotation X m
Center of Gravity Y m
User Defined Z m
Additional Acceleration Value
Ship Coordinate System
World Coordinate System
(X)''=Surge -0.0274748884321847 m/s^2
(Y)''=Heave 1.209390318737 m/s^2
(Z)''=Sway 0.00090107702175899 m/s^2
(ThetaX)''=Roll 5.9011309107347e-005 rad/s^2
(ThetaY)''=Yaw 1.9082364103953e-005 rad/s^2
(ThetaZ)''=Pitch -0.01777660483697 rad/s^2
Delete Modify Del Row Close Help
Loads
LoadCase 005 Name Extreme4+180
General | Mass | Point Force | Pressure Acceleration | Balance
Acceleration Reference Point
Center of Flotation X m
Center of Gravity Y m
User Defined Z m
Additional Acceleration Value
Ship Coordinate System
World Coordinate System
(X)''=Surge 0.0093048350027778 m/s^2
(Y)''=Heave 0.20522124201096 m/s^2
(Z)''=Sway 9.6795874906489e-005 m/s^2
(ThetaX)''=Roll -2.8028971936623e-006 rad/s^2
(ThetaY)''=Yaw 5.6609835334585e-006 rad/s^2
(ThetaZ)''=Pitch -0.00095025042616123 rad/s^2
Delete Modify Del Row Close Help

Load Case =4, "Extreme4"		
Station	X(m)	Bending Moment(N*m)
0	-87.5	694928
1	-83.25	-1.71342e+007
2	-79	-3.04362e+007
3	-74.75	-5.53546e+007
4	-70.5	-5.56614e+007
5	-66.25	-5.55367e+007
6	-62	-2.1123e+007
7	-57.75	1.05062e+007
8	-53.5	9.60105e+007
9	-49.25	1.88418e+008
10	-45	3.16424e+008
11	-40.75	4.48834e+008
12	-36.5	6.04086e+008
13	-32.25	7.55034e+008
14	-28	9.14736e+008
15	-23.75	1.05942e+009
16	-19.5	1.19409e+009
17	-15.25	1.30106e+009
18	-11	1.38013e+009
19	-6.75	1.42907e+009
20	-2.5	1.44134e+009
21	1.75	1.40173e+009
22	6	1.32238e+009
23	10.25	1.2253e+009
24	14.5	1.09212e+009
25	18.75	9.55468e+008
26	23	7.93496e+008
27	27.25	6.48087e+008
28	31.5	4.88972e+008
29	35.75	3.58906e+008
30	40	2.26178e+008
31	44.25	1.33013e+008
32	48.5	4.20755e+007
33	52.75	4.8907e+006
34	57	-4.02928e+007
35	61.25	-4.87815e+007
36	65.5	-6.3334e+007
37	69.75	-5.23521e+007
38	74	-4.62831e+007
39	78.25	-2.9959e+007
40	82.5	-2.1455e+007
41	86.75	-8.33652e+006
42	91	-6.93597e-006
43	94.423	-2.15402e-005

Maximum Bending Moment(Full Ship) in Structural Coordinates:
M=1.44134e+009(Nm)

Load Case =5, "Extreme4+180"		
Station	X(m)	Bending Moment(N*m)
0	-87.5	-46752.8
1	-83.25	-1.90289e+007
2	-79	-3.82319e+007
3	-74.75	-8.21733e+007
4	-70.5	-1.28525e+008
5	-66.25	-2.09795e+008
6	-62	-2.97187e+008
7	-57.75	-4.21871e+008
8	-53.5	-5.55379e+008
9	-49.25	-7.24841e+008
10	-45	-9.0073e+008
11	-40.75	-1.1037e+009
12	-36.5	-1.30505e+009
13	-32.25	-1.51654e+009
14	-28	-1.71258e+009
15	-23.75	-1.89429e+009
16	-19.5	-2.0442e+009
17	-15.25	-2.16604e+009
18	-11	-2.25197e+009
19	-6.75	-2.29431e+009
20	-2.5	-2.2893e+009
21	1.75	-2.25384e+009
22	6	-2.18181e+009
23	10.25	-2.06234e+009
24	14.5	-1.92074e+009
25	18.75	-1.74044e+009
26	23	-1.55295e+009
27	27.25	-1.34177e+009
28	31.5	-1.14967e+009
29	35.75	-9.47717e+008
30	40	-7.78877e+008
31	44.25	-6.08481e+008
32	48.5	-4.78778e+008
33	52.75	-3.51234e+008
34	57	-2.64024e+008
35	61.25	-1.77506e+008
36	65.5	-1.27443e+008
37	69.75	-7.72739e+007
38	74	-5.34294e+007
39	78.25	-2.88872e+007
40	82.5	-1.81938e+007
41	86.75	-6.04138e+006
42	91	-1.45747e-005
43	94.423	7.65373e-006

Maximum Bending Moment(Full Ship) in Structural Coordinates:
M=-2.29431e+009(Nm)

© 2014 Optimum Structural Design, Inc.

Spectral Fatigue Analysis

14

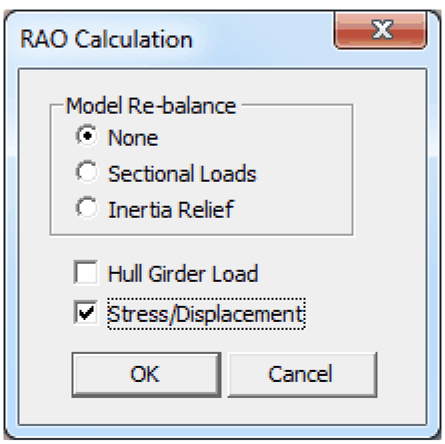
14 Spectral Fatigue Analysis

The Spectral Fatigue Analysis (SFA) module provides the ability to perform global fatigue screening of the vessel. The SFA module introduces additional functionality to the ELA module to compute Stress RAOs, displacement RAOs, define and associate structural groups to SN curves and Stress Concentration Factors (SCFs), and compute fatigue damage based on the Miner cumulative damage principle.

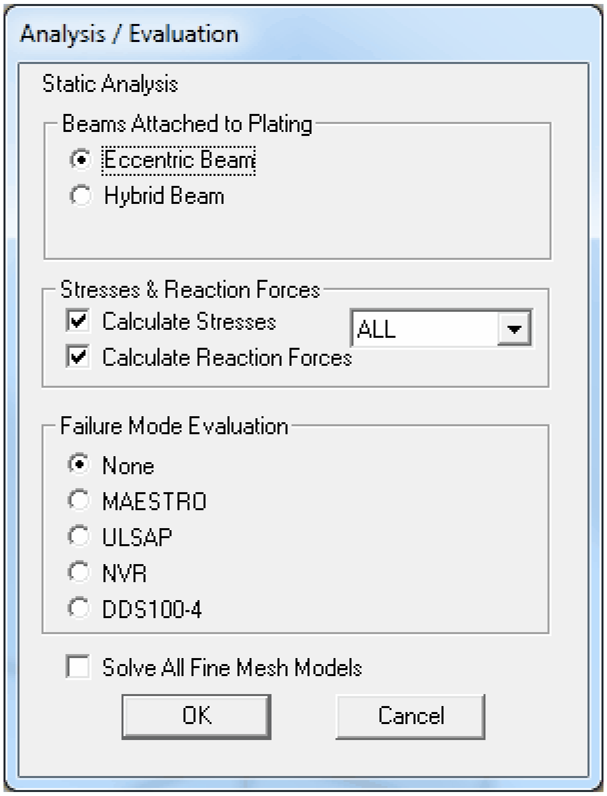
Table of Contents

14.1 Defining a SFA Load Case

Before creating a fatigue load case, the hydrodynamic loads need to be imported either from MAESTRO-Wave or a 3rd party seakeeping tool. For more information, see the importing *.smn file section. Once the loads are imported, the Stress/Displacement RAOs need to be calculated. Select ELA/SFA > Compute RAOs from the menu.

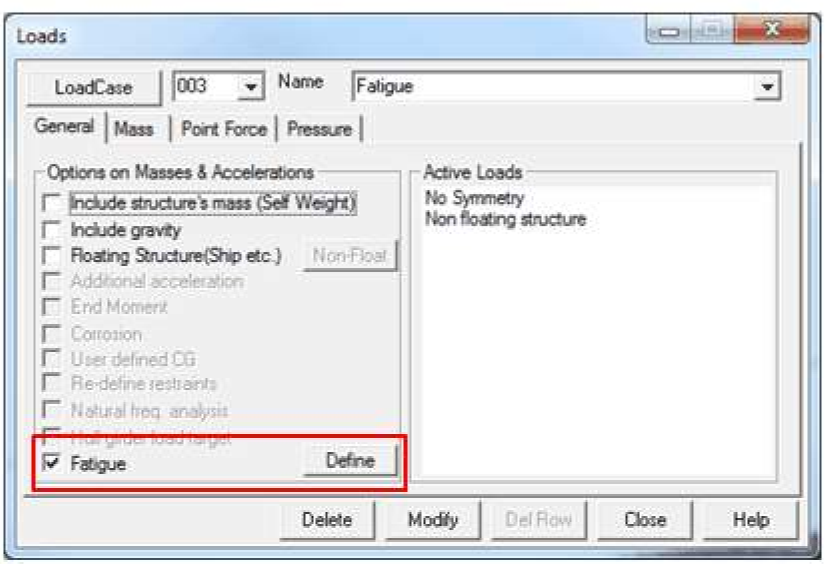


If the loads are imported from a 3rd party tool, it is recommended that the model be re-balanced using the "Sectional Loads" option. Ensure the "Stress/Displacement" box is checked and click "OK". This will open the Analysis/Evaluation dialog.



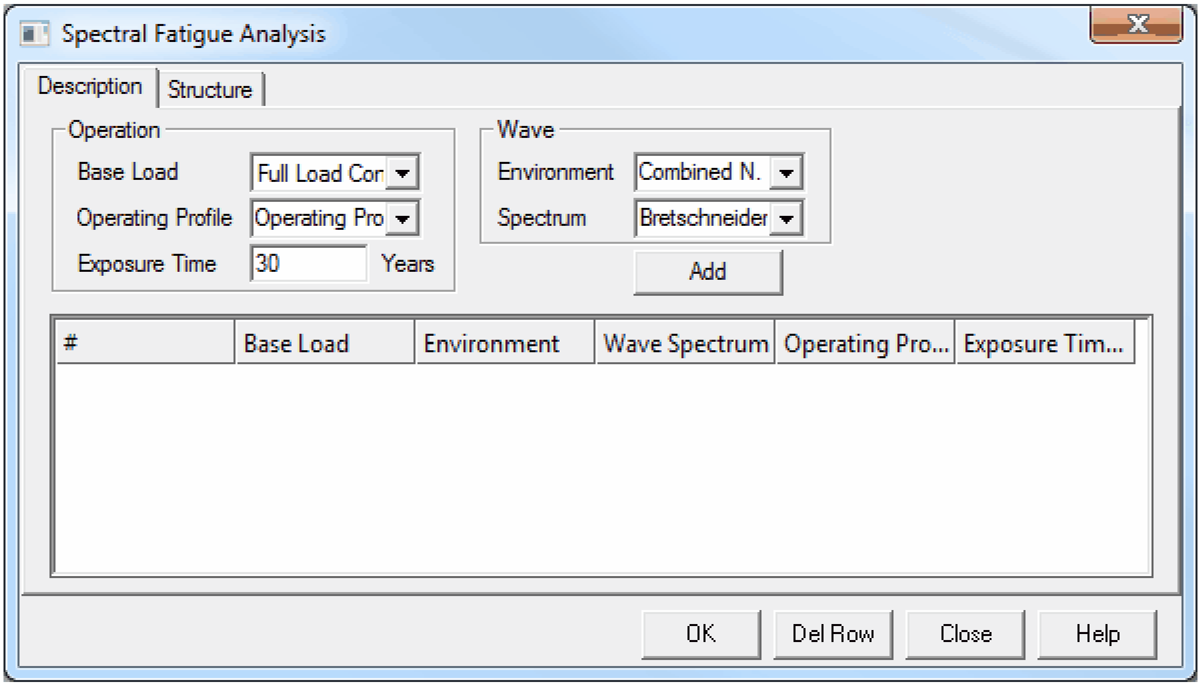
Click "OK" to calculate the RAOs.

Once the RAOs are calculated, the spectral fatigue analysis load case can be created using the standard MAESTRO load case dialog.



© 2014 Optimum Structural Design, Inc.

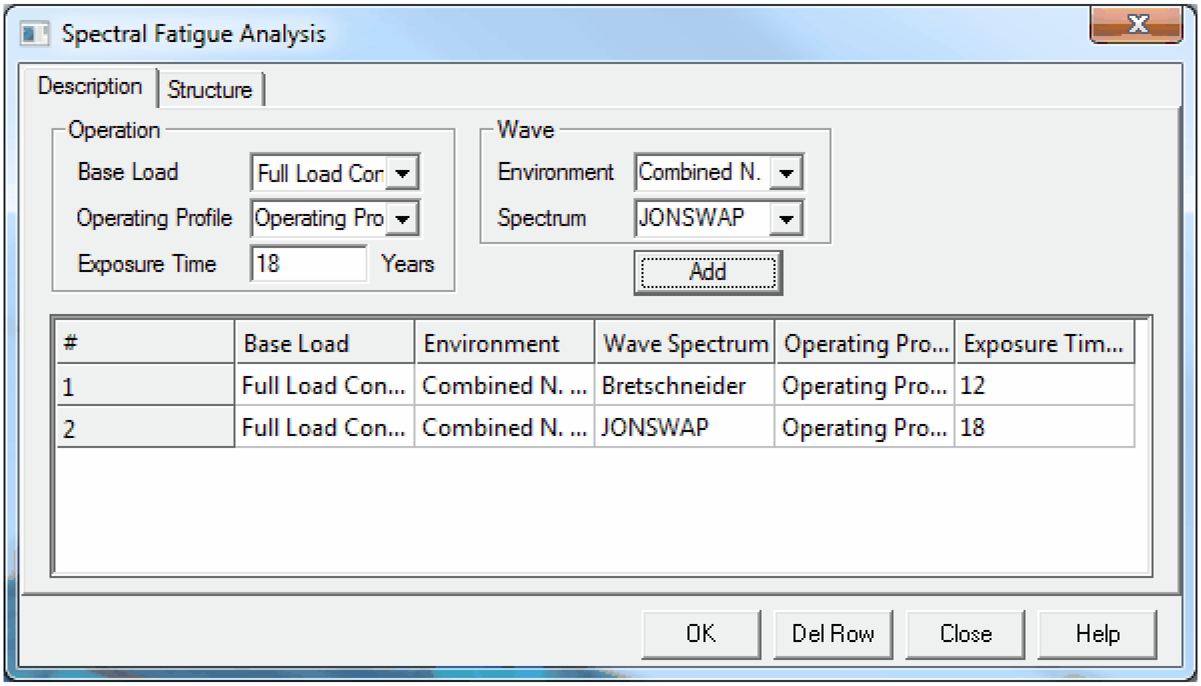
First create a new "Fatigue" load case, check only the "Fatigue" option, and click Create. (The actual loading is defined in the "base" load cases.) This will expose the "Define" button allowing the user to specify the fatigue load case parameters. Clicking "Define" will open the Spectral Fatigue Analysis dialog:



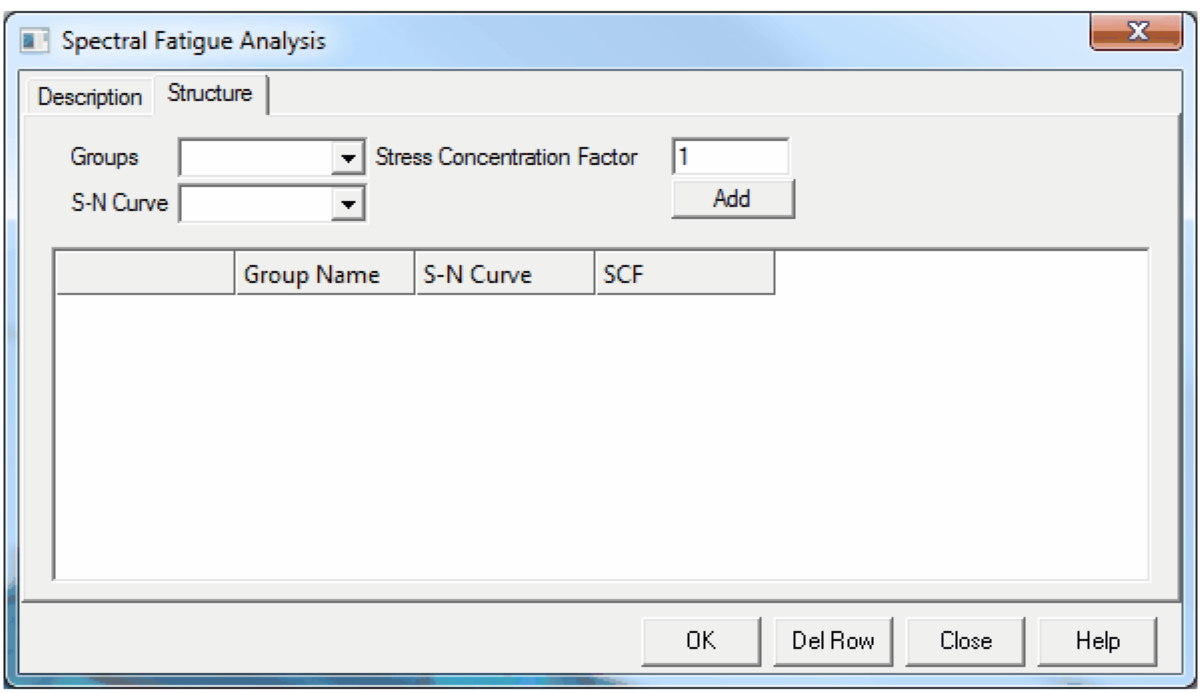
The Spectral Fatigue Analysis dialog is split into two tabs. The "Description" tab allows the user to specify the operation and wave environment. The base load case will be automatically populated with existing MAESTRO load cases. Multiple "base" load cases can be added to the "Fatigue" load case to describe the operations of a ship over its lifetime. For example, the ship's lifetime could be 30 years defined by the following conditions:

Load Case	**Exposure Time**
Full Load Departure	10 years
Full Load Arrival	4 years
Min Op Departure	12 years
Min Op Arrival	4 years

The same base load cases could also be combined with different wave environments. Once the Operation and Environment parameters are selected, click the "Add" button to add to the list of operating descriptions. The operating profile and wave environments can be created or imported. Please see this section for more details on these options.



The "Structure" tab allows the user to associate structural groups (i.e. MAESTRO groups) and associate S-N curves and stress concentration factors (SCF).



The groups drop-down will automatically be populated with the available Plate, Volume, General, Section, and Module groups defined in the model. Additional groups can be created after the initial definition of the load case and added at a later time. The S-N curve drop-down is populated with the available S-N curves from the classification societies: American Bureau of Shipping (ABS) and Det Norske Veritas (DNV):

ABS_B

ABS_C

ABS_D

ABS_E

ABS_F

ABS_F2

ABS_G

ABS_W

DNV_I

DNV_II

DNV_II

DNV_IV

DNV_Ib

DNV_IIIb

The SCF is the ratio of the "hot spot" stress to the nominal stress computed for that location where the "hot spot" stress is defined as the surface value of the structural stress at a given "hot spot" and the nominal stress is the stress at a cross section of the structural detail, away from the "hot spot".

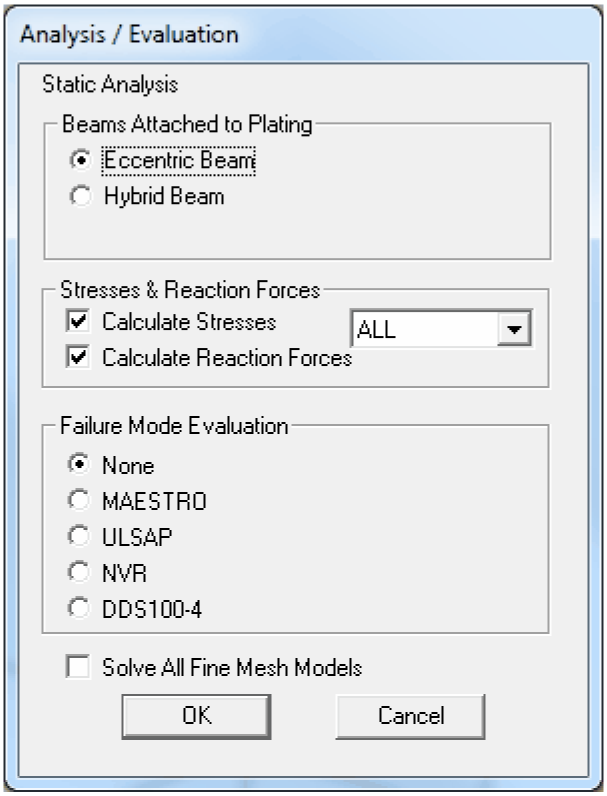
When a group is added, the S-N curve and SCF is applied to all the elements included in the group. Similar to the "Description" tab, multiple groups with different S-N and SCFs can be added and a summary will be displayed in the table.

Once the "Description" and "Structure" tab are completed, click "OK" to save the load case parameters. You can return to this dialog at any time to make changes to the load case.

© 2014 Optimum Structural Design, Inc.

14.2 Fatigue Screening

The Fatigue load case is run and analyzed in the same process as a standard MAESTRO load case. Using the Analysis/Evaluation dialog, the user can run the analysis once their load case is defined. If you plan to run the fatigue screening on a top-down fine mesh model, make sure to select the "Solve All Fine Mesh Models" option in the dialog below.



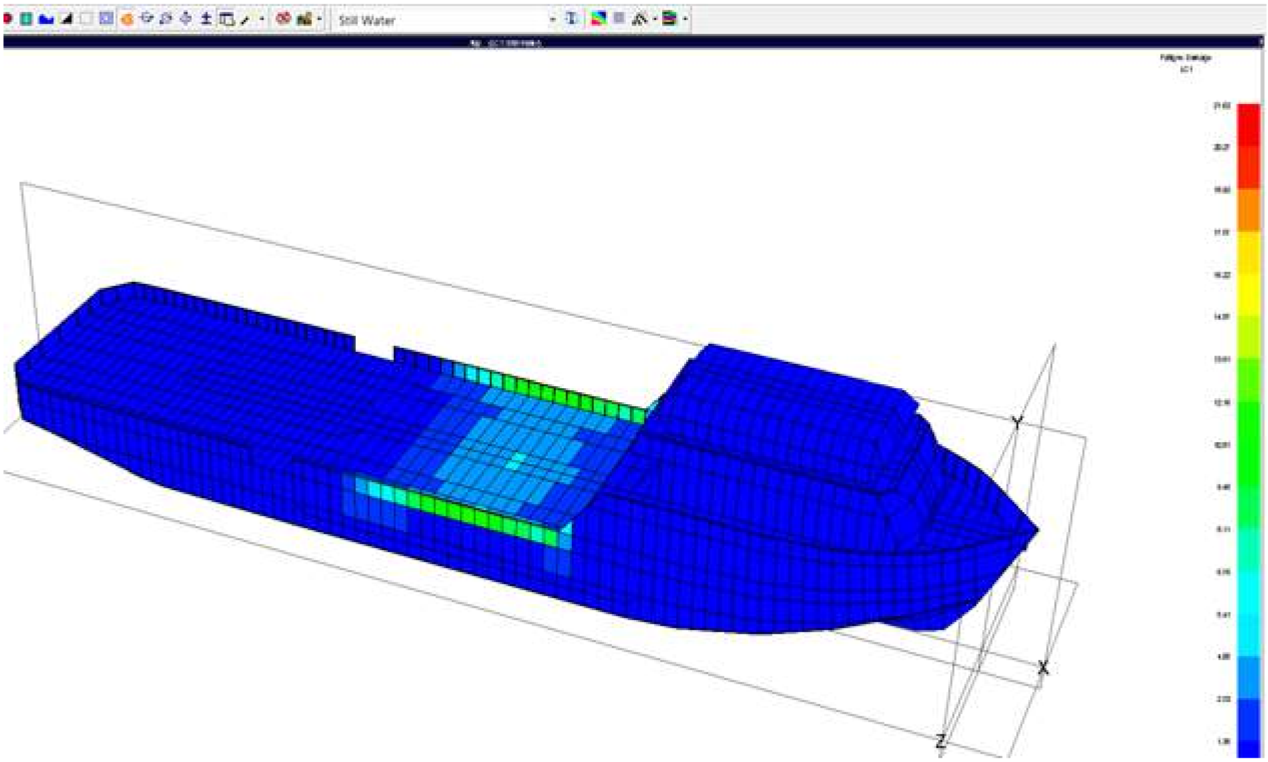
Using the calculated stress RAOs, the fatigue damage is calculated using Miner's Rule and stored as the fatigue damage ratio in the standard MAESTRO results file (*.rlt). If the fatigue damage, D, is greater than 1.0, then the structure has a fatigue problem. The fatigue life, FL, is then calculated using the fatigue damage value as:

$$FL = \text{service life}/D$$

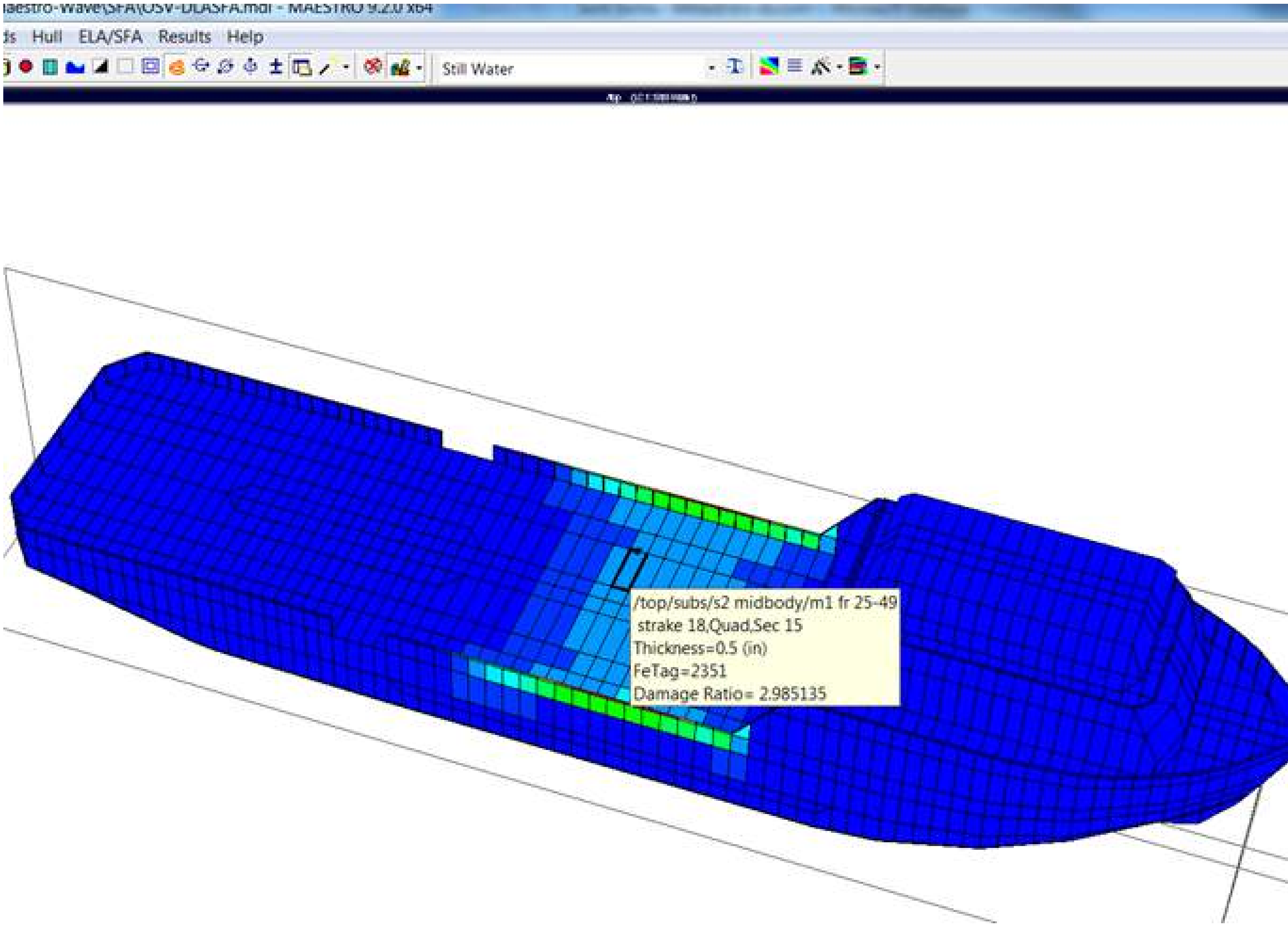
This means any structure with a damage ratio greater than 1.0 will have a fatigue life shorter than the service life.

Once the analysis is complete, the damage ratio is plotted for each element.

© 2014 Optimum Structural Design, Inc.



Using the dynamic query, the damage ratio for each element can be individually queried.



14.3 Validation

The 44,000 ton tanker model from the ELA validation is also used for the spectral fatigue validation. The quad element "/top/s2 midbody hull/m4 tank4 strake 12 section 7" is selected for a hand calculation, and is then compared to the calculation using the MAESTRO SFA module.

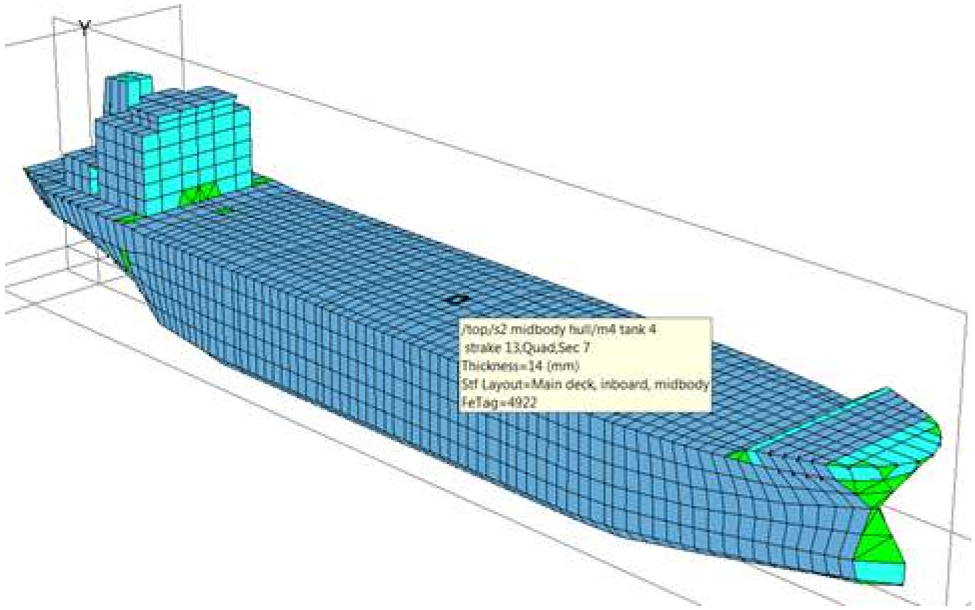
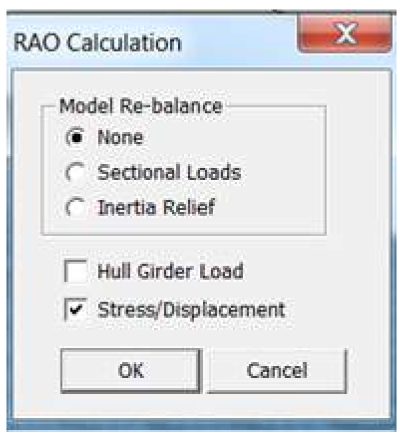


Figure 1. Quad element "/top/s2 midbody hull/m4 tank4 strake 12 section 7"

To perform a spectral fatigue analysis, the stress RAO database (*.sfa) has to first be generated.



Once the database is created, the "real" and "imaginary" stress and deflection of a selected speed, heading and wave frequency can be obtained from the "Frequency Load Database" dialog for a sanity check. When the result of a frequency load case is selected, all regular post process functions, such as query, stress, deflection and the result lists are available to the user.

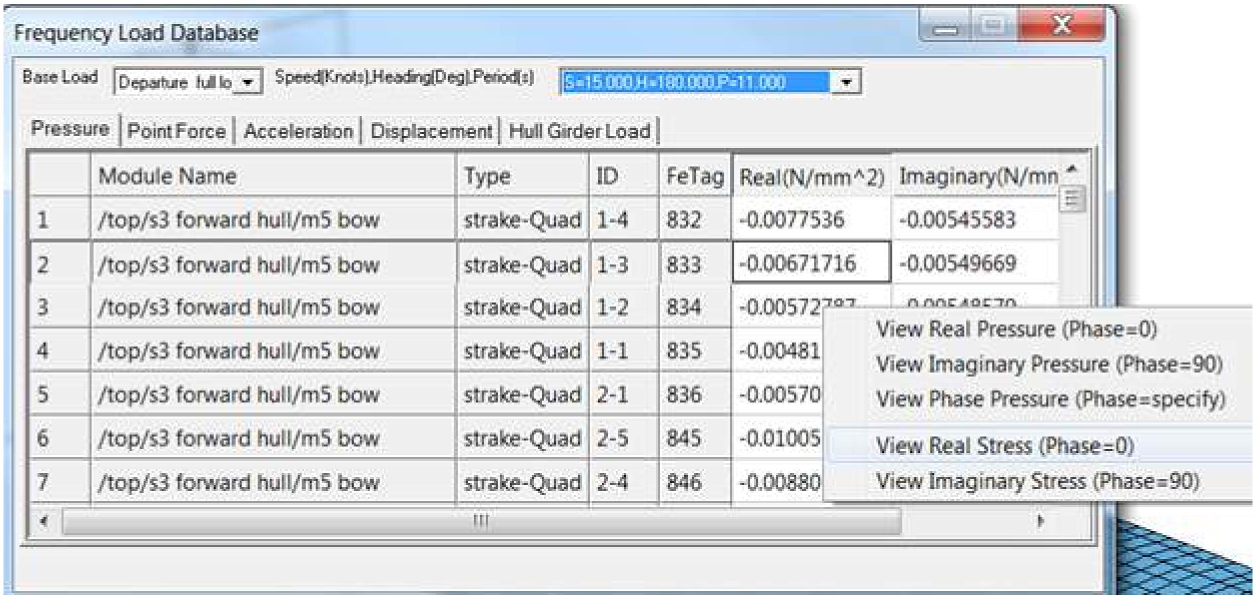
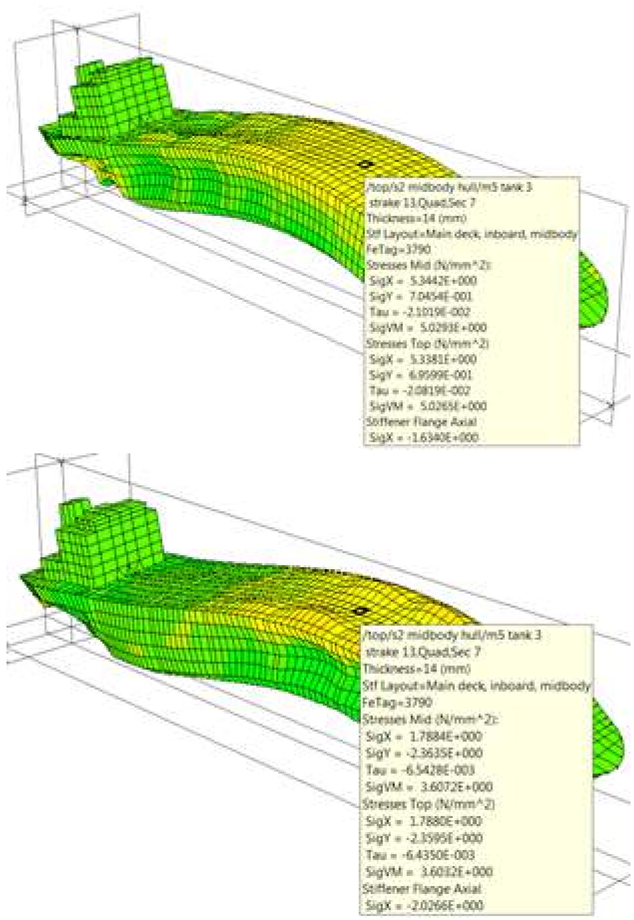


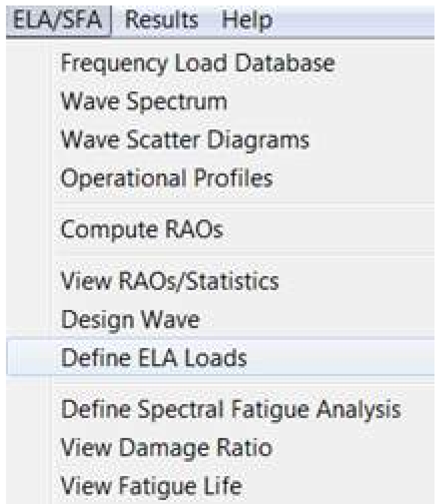
Figure 2. Selection of the full model stress and deflection

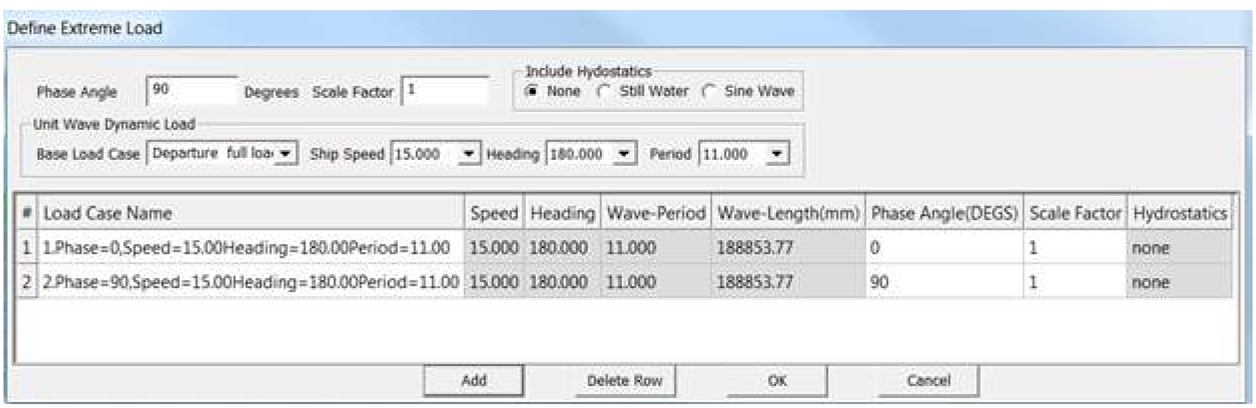


Heading=180, Wave Period=11sec, Real   Heading=180, Wave Period=11sec, Imaginary

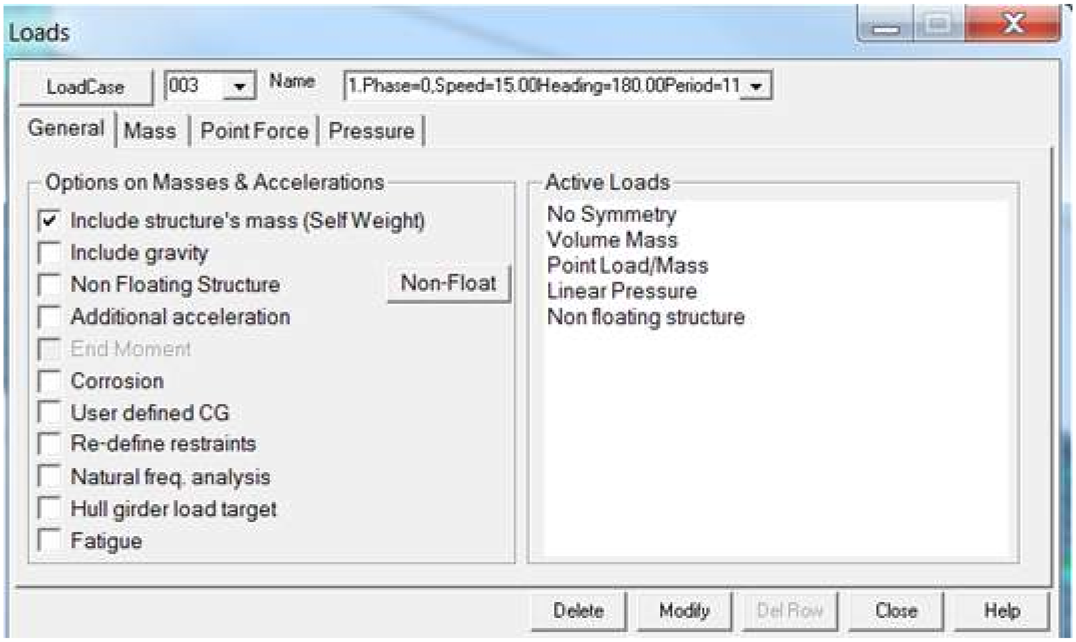
Figure 3. Full model of Stress and deflection

To verify the stress RAO calculation of a specific case, the “Define ELA Loads” command was used to create regular MAESTRO load cases. Note that, in order to match the automatic stress RAO calculation, the hydrostatics should not be included when generating an ELA load case, as shown below.





Once a regular MAESTRO load case is created, the loads (either individual elemental loads or hull girder loads) can be reviewed and examined. The corresponding finite element analysis results can then be compared to the automatic stress RAO results.



Element stress RAOs can be obtained from the MAESTRO "Response Amplitude Operator" dialog, as shown in Figure 4.

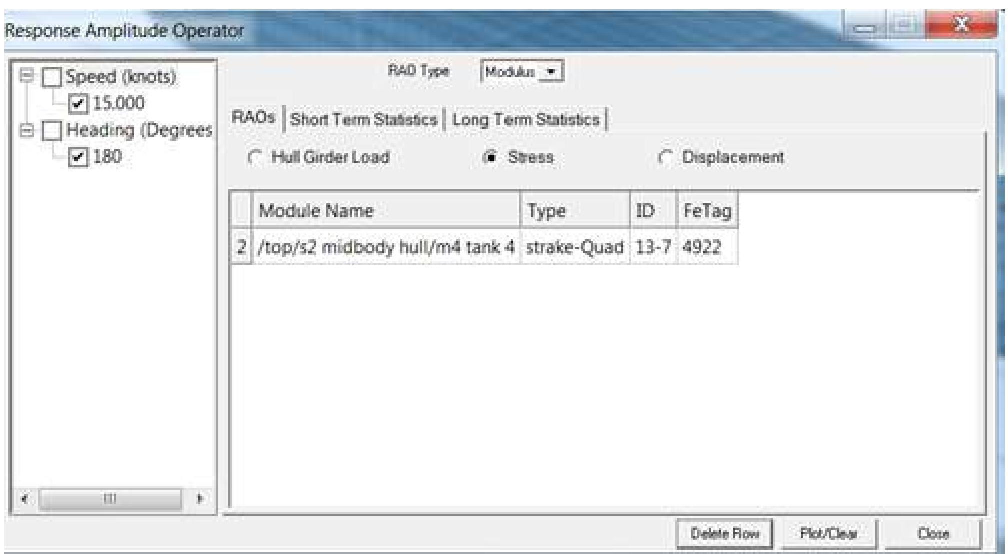


Figure 4. Retrieve selected element stress RAO

Table 1 lists the principal stress RAO of quad element 4922, computed by MAESTRO. Table 2 gives the hand calculations of the principal stress for a wave period at 11 seconds. Figure 5 shows the principal stress RAO of quad element 4922.

T(sec)	Omega (rad/s)	RAO:Stress(N/mm^2) FeTag=4922,S=15.00(Knot):H= 180DEG

30.000	0.209	0.479
29.000	0.217	0.420
28.000	0.224	0.351
27.000	0.233	0.270
26.000	0.242	0.176
25.000	0.251	0.071
24.000	0.262	0.099
23.000	0.273	0.232
22.000	0.286	0.414
21.000	0.299	0.638
20.000	0.314	0.909
19.000	0.331	1.249
18.000	0.349	1.693
17.000	0.370	2.316
16.000	0.393	3.235
15.000	0.419	4.607
14.000	0.449	6.600
13.000	0.483	8.530
12.000	0.524	7.429
11.000	0.571	5.636
10.000	0.628	7.876
9.000	0.698	6.094
8.000	0.785	1.150
7.000	0.898	2.610
6.000	1.047	1.314
5.000	1.257	0.726
4.000	1.571	0.834
3.000	2.094	2.630

Table 1. Stress RAO of quad element 4922

© 2014 Optimum Structural Design, Inc.

Stress (MPa)	Real (Phase angle=0)	Imaginary(Phase angle=90)	Modulus
σ_x	5.3442	1.7884	5.6355
σ_y	0.70454	-2.3635	2.46627
τ_{xy}	-2.10E-02	-6.54E-03	0.02201
σ_1	5.344295	1.78841	5.6356
σ_2	0.704445	-2.36351	2.46626

Table 2. Hand calculation of the principal stress of quad element 4922 for wave period at 11 seconds

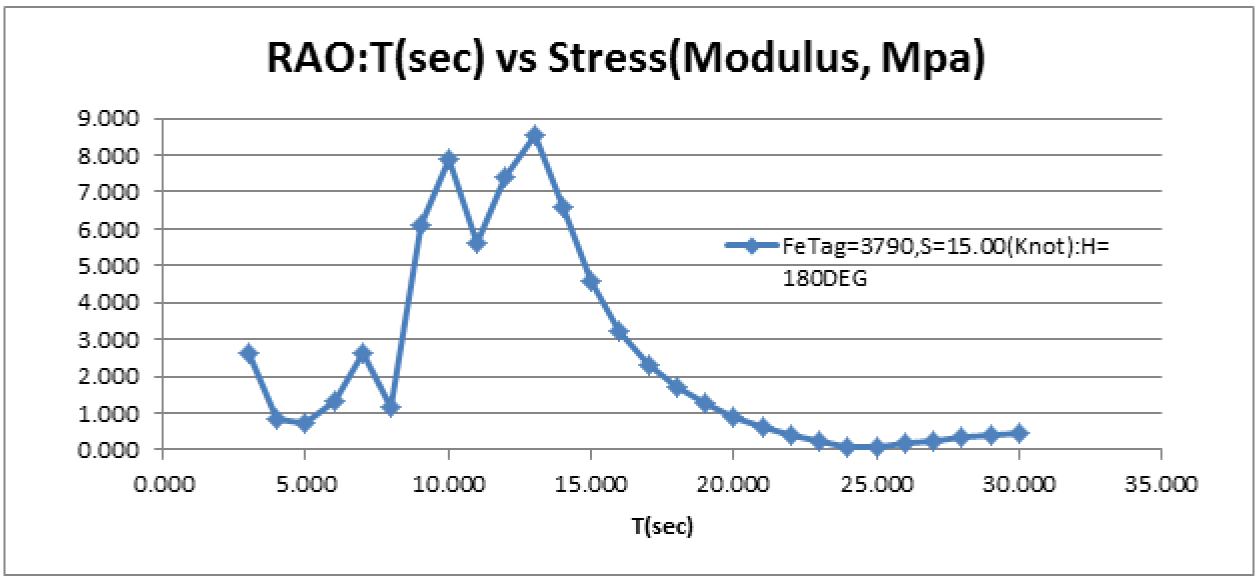
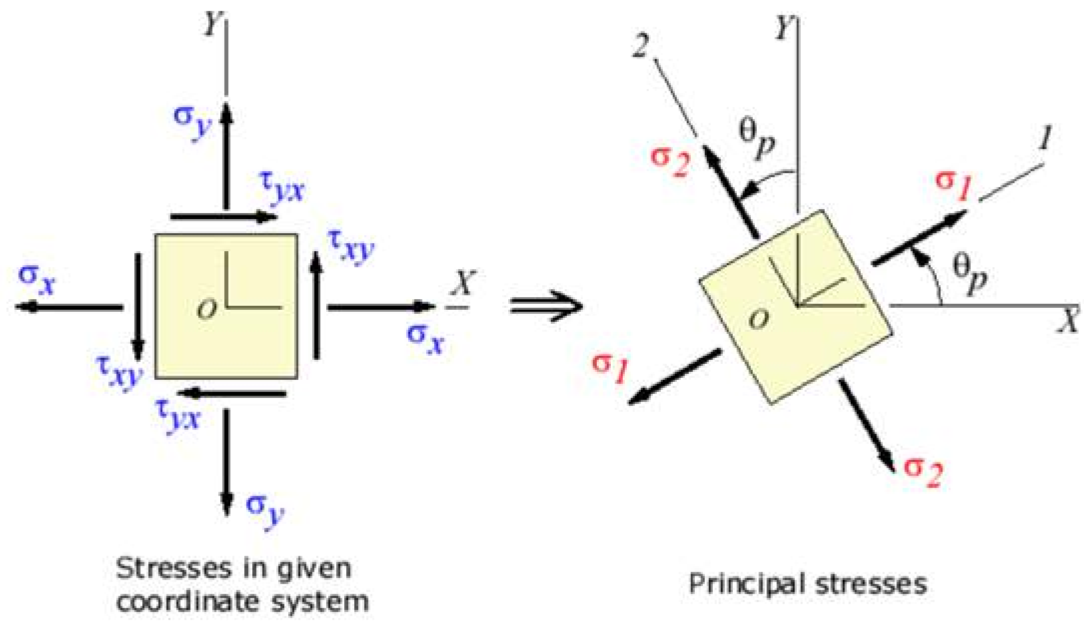


Figure 5. Quad element 4922 stress RAO

The transformation to the principal directions can be illustrated as:



$$\sigma_{1,2} = \frac{\sigma_x + \sigma_y}{2} \pm \sqrt{\left(\frac{\sigma_x - \sigma_y}{2}\right)^2 + \tau_{xy}^2}$$

$$tan2\theta_p = \frac{\tau_{xy}}{\sigma_x - \sigma_y}$$

Two sea states, shown below, are selected for the stress spectrum calculation. The stress spectra of the two sea states are given in Table 3 and 4.

Zero-crossing frequency=0.150166 Hz
Sea State: Hs=1200.000(mm) Tp=8.000s Probability=0.40000

Wave-Freq-e(Omega)	Wave-Period-e(s)	Wave-Length-e(mm)	Response(N/mm^2)^2
0.243957	25.7553	1035.32	3.96E-27
0.2536	24.776	958.08	5.60E-24
0.264022	23.7979	883.933	2.51E-21
0.275324	22.8211	812.853	6.00E-19
0.287615	21.8458	744.865	2.85E-16

© 2014 Optimum Structural Design, Inc.

0.301031	20.8723	679.954	1.40E-13
0.315731	19.9004	618.111	2.92E-11
0.331906	18.9306	559.334	2.88E-09
0.349782	17.9631	503.622	1.51E-07
0.369641	16.9981	450.963	4.63E-06
0.391821	16.0359	401.352	8.84E-05
0.416746	15.0768	354.779	0.001114
0.444945	14.1213	311.234	0.009806
0.47709	13.1698	270.708	0.063876
0.514046	12.223	233.183	0.324081
0.556945	11.2815	198.644	1.32606
0.607293	10.3462	167.072	4.46027
0.667138	9.41813	138.443	10.4081
0.739327	8.49852	112.727	8.8981
0.827933	7.589	89.8897	3.02143
0.938968	6.69159	69.8875	9.8995
1.08165	5.8089	52.6657	5.62666
1.27079	4.94433	38.1554	0.24561
1.53157	4.10244	26.2679	0.279134
1.91011	3.28944	16.8882	0.041644
2.49923	2.51405	9.8648	0.001522
3.51235	1.78889	4.99466	0.002185
5.54604	1.13291	2.00325	0.004808

Table 3. Stress RMS=2.09537(N/mm^2)

Zero-crossing frequency=0.167133 Hz

Sea State: H_s=1500.000(mm) T_p=7.000s Probability= 0.60000

Wave-Freq-e(Omega)	Wave-Period-e(s)	Wave-Length-e(mm)	Response(N/mm^2)^2
0.243957	25.7553	1035.32	5.31E-46

© 2014 Optimum Structural Design, Inc.

0.2536	24.776	958.08	2.10E-40
0.264022	23.7979	883.933	1.51E-35
0.275324	22.8211	812.853	3.48E-31
0.287615	21.8458	744.865	9.83E-27
0.301031	20.8723	679.954	1.84E-22
0.315731	19.9004	618.111	9.70E-19
0.331906	18.9306	559.334	1.65E-15
0.349782	17.9631	503.622	1.06E-12
0.369641	16.9981	450.963	2.87E-10
0.391821	16.0359	401.352	3.63E-08
0.416746	15.0768	354.779	2.33E-06
0.444945	14.1213	311.234	8.24E-05
0.47709	13.1698	270.708	0.001742
0.514046	12.223	233.183	0.023712
0.556945	11.2815	198.644	0.219866
0.607293	10.3462	167.072	1.44493
0.667138	9.41813	138.443	5.78983
0.739327	8.49852	112.727	7.60377
0.827933	7.589	89.8897	3.60714
0.938968	6.69159	69.8875	15.2455
1.08165	5.8089	52.6657	10.4644
1.27079	4.94433	38.1554	0.522929
1.53157	4.10244	26.2679	0.65222
1.91011	3.28944	16.8882	0.103386
2.49923	2.51405	9.8648	0.003919
3.51235	1.78889	4.99466	0.005742
5.54604	1.13291	2.00325	0.012759

Table 4. Stress RMS=2.34854(N/mm^2)

© 2014 Optimum Structural Design, Inc.

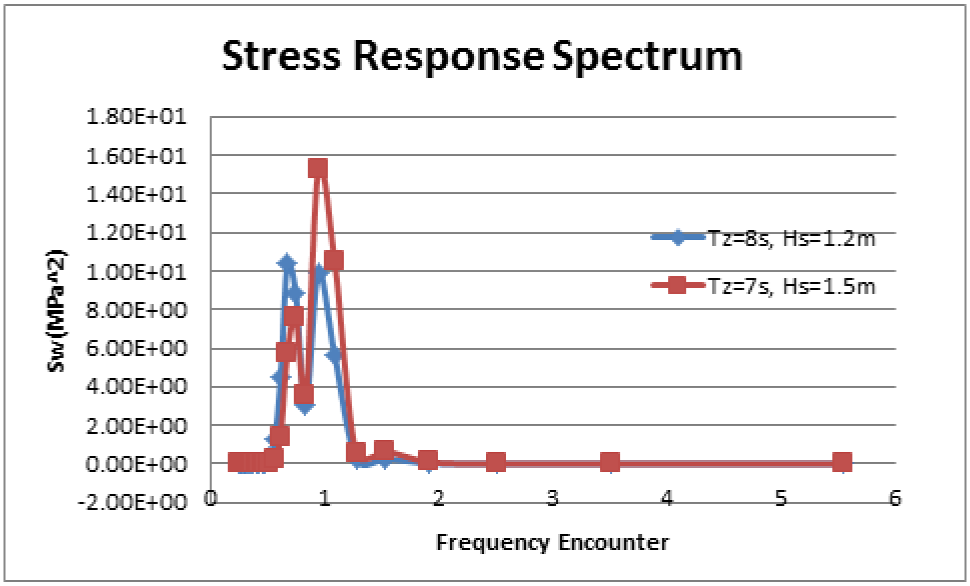


Figure 6. Stress spectra of Quad element 4922

S-N Curve	Material	a	Log a	m
Ib	Welded joint	5.75E12	12.76	3.0
IIIb	Base material	1.0E13	13.00	3.0

Table 5. DNV One Slope S-N Curve

The damage ratio of quad element 4922 over a design life of 5 years is calculated according to the equation below:

$$D = \frac{T}{a}\left(2\sqrt{2}\right)^m \Gamma\left(1+\frac{m}{2}\right) \sum_{i=1}^{all\ sea\ states} f_{0i}\, p_i \left(\sqrt{m_{0i}}\right)^m$$

One slope S-N curve, DNV-Ib, is used. The probability of sea state 1 is 0.4, and the probability of sea state 2 is 0.6.

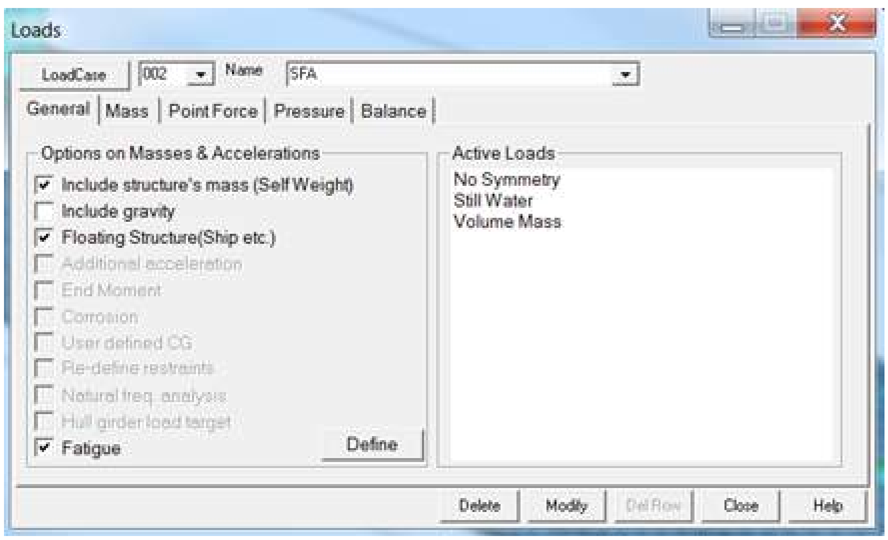
$$D_1 = \frac{5\cdot 365\cdot 24\cdot 3600}{5.75E12}\left(2\sqrt{2}\right)^3 \Gamma\left(1+\frac{3}{2}\right)\cdot 0.15016\cdot 0.4\cdot (2.09537)^3 = 0.000456$$

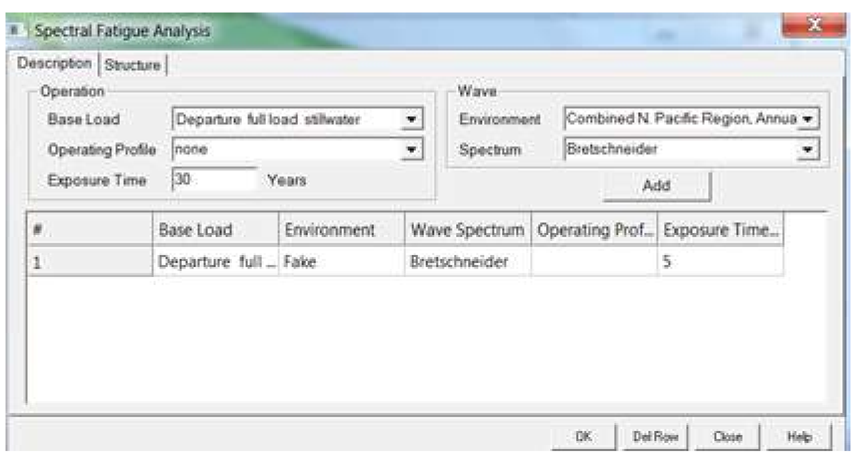
© 2014 Optimum Structural Design, Inc.

$$D_2 = \frac{5 \cdot 365 \cdot 24 \cdot 3600}{5.75E12} \left(2\sqrt{2}\right)^3 \Gamma\left(1 + \frac{3}{2}\right) \cdot 0.167133 \cdot 0.6 \cdot (2.34854)^3 = 0.00107202$$

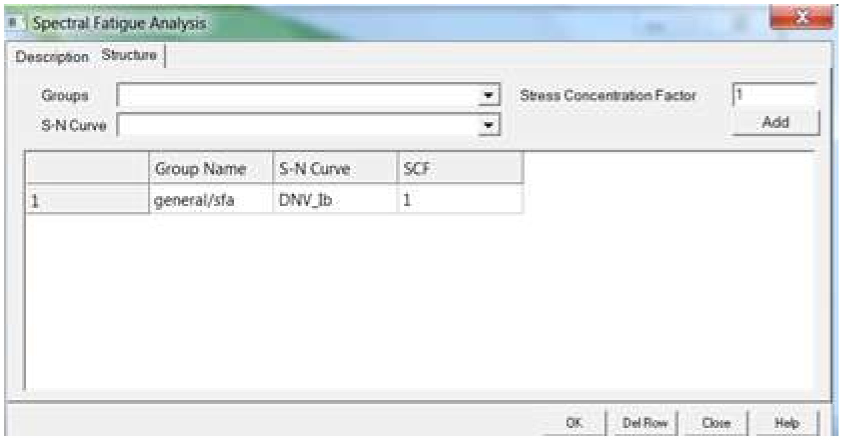
$$D = D_1 + D_2 = 0.00153$$

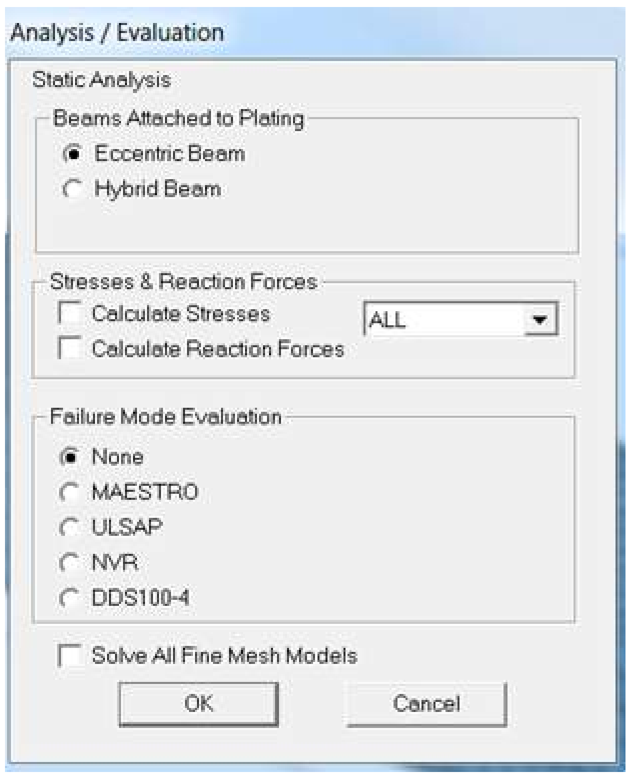
To calculate fatigue damage using MAESTRO, first define a spectral fatigue load case, as shown below, and then run a MAESTRO analysis. Note that for the spectral fatigue load case, the solution dialog is just a place holder. The options for stress and adequacy calculation do not apply.



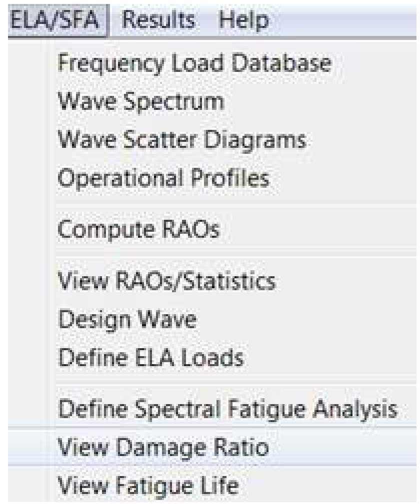


© 2014 Optimum Structural Design, Inc.





To view the element fatigue damage, select ELA/SFA > View damage Ratio, as shown below:



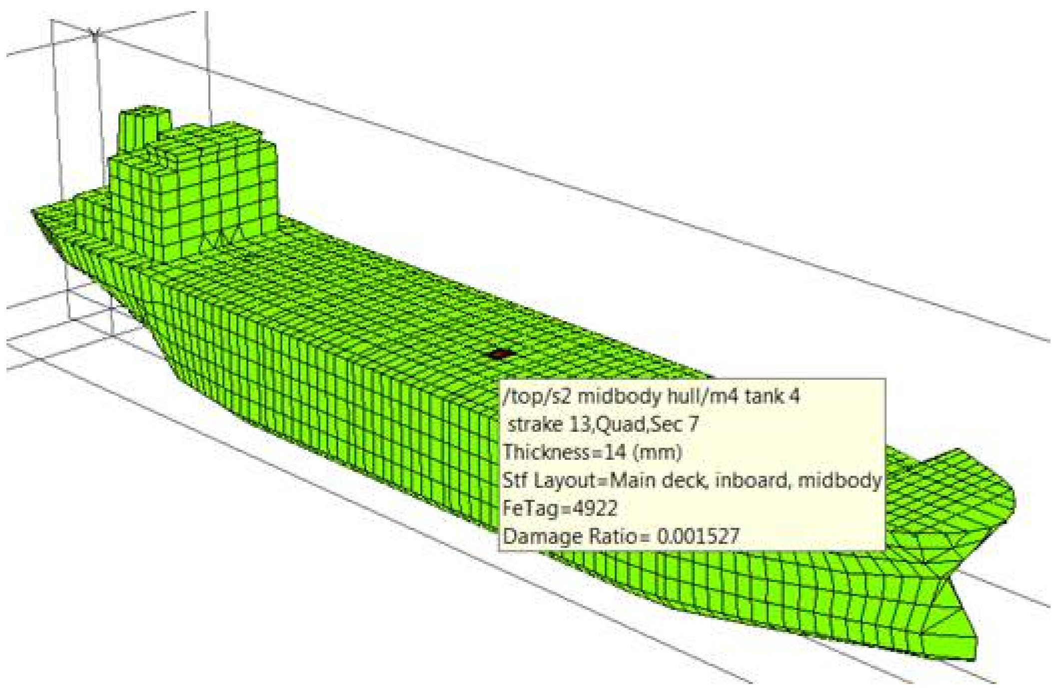


Figure 7. Quad 4922 Fatigue Damage Ratio

The fatigue damage ratio calculated by MAESTRO matches the hand-calculated value.

© 2014 Optimum Structural Design, Inc.

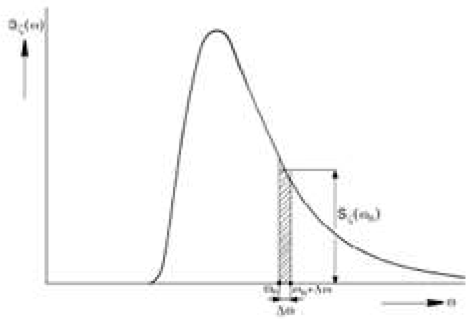
14.4 Extreme Stress Analysis

In addition to calculating stress RAOs for each element in the model, MAESTRO is also able to compute mean stress and extreme stress values. The mean stress is a statistical measure of each element's stress RAO and is defined as:

$$\sigma_m = \sqrt{m_0}$$

where

$$m_0 = \int_0^\infty S_\sigma(\omega,\phi,V)\,d\omega$$



The extreme stress is a measure of each element's extreme stress value given a probability of exceedence or exposure time. Using short term statistics, the extreme stress is defined as:

$$\bar{y}_n = \sqrt{2\ln\left(\frac{2\sqrt{1-\varepsilon^2}}{1+\sqrt{1-\varepsilon^2}}\,\frac{n}{\alpha}\right)}\sqrt{m_0}$$

Where

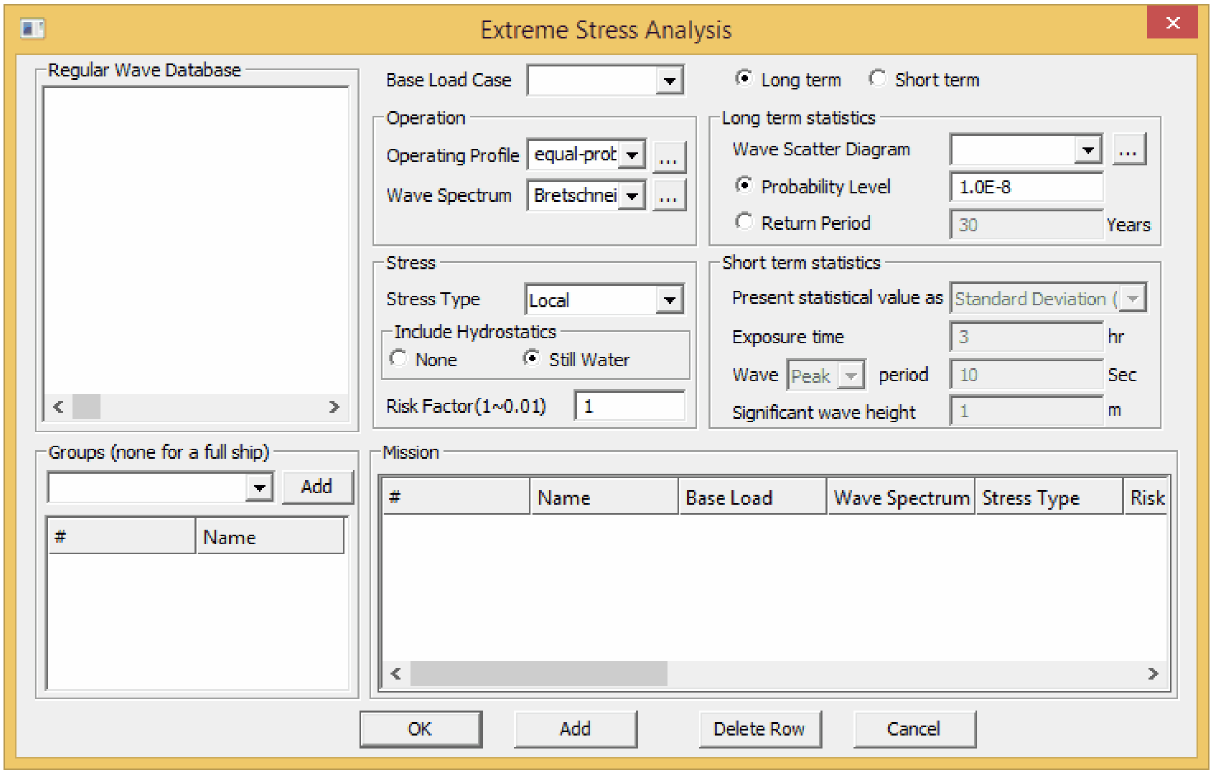
$\alpha = a\ risk\ parameter$
$n = total\ number\ of\ wave\ encounters\ for\ the\ particular\ sea\ state$
$T = time\ in\ hours$

$$n = \frac{1}{2\pi}\sqrt{\frac{m_2}{m_0}}60^2T \qquad \varepsilon = \sqrt{1-\frac{m_2^2}{m_0 m_4}}$$

Using long term statistics, the extreme value is calculated using a probability of exceedence equal to $\frac{1}{N}$ in which N is the number of wave load peaks determined from a scatter diagram and operating profile.

Similar to the design wave calculations in which the unit wave load RAOs are used to determine the design wave corresponding to the maximum hull girder loads, the extreme stress value is calculated using the element stress RAOs. The benefit of the extreme stress calculation is that it enables the user to screen the maximum expected stress for each element given a short term exposure or long term design life.

The extreme stress analysis setup can be accessed from Wave > Extreme Stress Analysis:



Similar to the design wave computation, the extreme stress analysis will create a new static load case based on the dialog settings. For using either short or long term statistics, the base load case and regular wave runs must be selected for the analysis in addition to an operating profile and wave spectrum. The default operating profile assumes equal probability of all speeds and headings selected. For long term statistics, a wave scatter diagram and either a probability level or return period must be selected. For the short term statistics, an exposure time and spectrum definition must be provided.

To create the extreme stress analysis load case, click "Add". The calculation can be performed on the entire ship or on a selected group or groups only. Once the load case is created, it can be analyzed like any other static load case. Since the stresses are computed using statistics, the results of the extreme stress analysis does not include deflections or loads. These stresses can also be used as input into MAESTRO's limit state evaluation framework.

© 2014 Optimum Structural Design, Inc.

Advanced Processing and Programming

15

15 Advanced Processing and Programming

The following topics discuss advance capabilities offered with MAESTRO. These include batch processing, accessing results via a COM interface and importing 3rd party hydrodynamic loads.

15.1 Batch Processing

Introduction

MAESTRO has the ability to perform batch processing. This is useful, for example, when the user would like to sequentially solve multiple models with many load cases without having to manually launch the process. Further, several post-processing keywords are available for users to compare a result to a known value.

The batch file will consist of at least three tokens separated by a space: first to launch (behind the scene) MAESTRO, second to identify the model, and third to define a keyword. Note, the second and third tokens may need to be bracketed by quotation marks if there are multiple sub-keywords present. The presence of a third token will allow MAESTRO to start the batch process. If a third token is not present, MAESTRO will simply launch the GUI with the selected model open. An example batch file (*maestro.bat*) is found in the Models and Samples/Advanced/Batch Processing directory of the MAESTRO installation directory.

An example would be:

first_token second_token third_token

cd ..\..\system modeler92.exe "ex1_ulsap.mdl" run

Keywords

The following keywords are available to support batch processing and accessing results for a given *fetag*. The keywords have been categorized into Processing and Post-processing.

Processing

The batch processing is initiated by the presence of a third token as described above. The example above uses *run* as the third token, which would initiate the batch process with the default Analysis and Evaluation setup parameters. If the model consists of fine-mesh analysis models, the Sparse solver will be used as the solver method.

The default Analysis and Evaluation setup parameters and behaviors are:

- Equation Solver Method = Sparse
- Beams Attached to Plating = Eccentric Beam

© 2014 Optimum Structural Design, Inc.

- Stresses & Reaction Forces = Calculate Stress & Calculate Reaction Forces
- Failure Mode Evaluation = None
- A log file containing all of the results will be generated

Keyword: *hybrid*

Used to change the method of Beams Attached to Plating from the default, Eccentric Beam, to Hybrid Beam

Example: *cd ..\..\system modeler92.exe "ex1.mdl" hybrid*

Keyword: *iterative*

Used to launch the Iterative solver. Note the user is to define the number of iterations and tolerance. If an iteration number and tolerance are omitted, the number of iterations and tolerance default to 20,000 and 1.0E-5 respectively.

Example: *cd ..\..\system modeler92.exe "ex1_iter.mdl" "iterative 2000 2.0E-6"*

Keyword: *maestro*

Used to perform limit state analysis using MAESTRO.

Example: *cd ..\..\system modeler92.exe "ex1_maestro.mdl" maestro*

Keyword: *nostress*

Used to turn off stress calculations during batch processing.

Example: *cd ..\..\system modeler92.exe "ex1.mdl" nostress*

Keyword: *noreaction*

Used to turn off reaction force calculations during batch processing.

Example: *cd ..\..\system modeler92.exe "ex1.mdl" noreaction*

Keyword: *nolog*

Used to turn off the functionality that generates a log file containing the results.

Example: *cd ..\..\system modeler92.exe "ex1.mdl" nolog*

Keyword: *ulsap*

Used to perform limit state analysis using ALPS/ULSAP.

© 2014 Optimum Structural Design, Inc.

Example: *cd ..\..\system modeler92.exe "ex1_ulsap.mdl" ulsap*

Post-processing

The following keywords are used to access results for a given model, load case, and fetag so the user can compare the values to a known quantity. If the solution quantities do not match the prescribed quantities, MAESTRO will produce an error message such as the on below. This is useful from a quality checking perspective.

Keyword: *adq*

Used to access a variety of adequacy parameters associated with structural evaluation using the MAESTRO and ALPS/ULSAP limit states. Note the adequacy comparisons have a built in tolerance of 0.001. Sub-keywords associated ONLY with ALPS/ULSAP are noted below. Examples is provided below.

Sub-keywords to *adq* for MAESTRO and ALPS/ULSAP

- Load Case Specification: *LC=*
- FeTag Specification: *FETAG=*
- Panel Limit States: *pcsf* | *pccb* | *pcmy* | *pcsb* | *pytf* | *pytp* | *pycp* | *pspbl* | *pspbt* | *pflb*
- Beam Limit States: *bct* | *byc* | *bcwb* | *bcc* | *bccf* | *bccp* | *bycf* | *bycp* | *bytf* | *bytp* | *bcph*
- Rod Limit State: *euler*

Example: *cd ..\..\system modeler92.exe "ex1_maestro.mdl" maestro "adq LC=1 FETAG=6 pccb=0.535"*

Sub-keywords to *adq* for ALPS/ULSAP only

- Panel Limit States: *pcpm* | *pcpe*

Example: *cd ..\..\system modeler92.exe "ex1_maestro.mdl" ulsap "adq LC=1 FETAG=6 pcpe=0.502"*

Keyword: *disp*

Used to access the displacement results of a given load case and fetag. Note the user is to define (via sub-keywords) the load case, fetag, type of displacement, and tolerance. An example is provided below.

Sub-keywords to *disp:*

© 2014 Optimum Structural Design, Inc.

- Load Case Specification: *LC=*
- FeTag Specification: *FETAG=*
- Displacement Type: *dx* | *dy* | *dz* | *rx* | *ry* | *rz* | *dt* (total translation)
- Comparison Tolerance: *tol=*

Example: *cd ..\..\system modeler92.exe "ex1_maestro.mdl" maestro "disp LC=1 FETAG=37 dt=0.001 tol=1.0E-6"*

Keyword: *stress*

Used to access the mid-plane stress results of a given load case and fetag. Note the user is to define (via sub-keywords) the load case, fetag, type of mid-plane stress, and tolerance. An example is provided below.

Sub-keywords to *stress:*

- Load Case Specification: *LC=*
- FeTag Specification: *FETAG=*
- Mid-plane Stress Type: *sxx* | *syy* | *syx* | *svm*
- Comparison Tolerance: *tol=*

Example: *cd ..\..\system modeler92.exe "ex1_maestro.mdl" maestro "stress LC=1 FETAG=51 sxx=3518734.5 tol=0.01"*

Keyword: *freq*

Used to perform a natural frequency analysis.

Sub-keywords to *freq:*

- Load Case Specification: *LC=*
- FeTag Specification: *FETAG=*
- Frequency Mode Selection: *iMode=*
- Frequency: *freq=*
- Comparison Tolerance: *tol=*

Example: cd ..\..\system modeler92.exe "Natural Frequency\w4_13_girder_free.mdl" run "freq LC=1 iMode=7 freq=73.2816 tol=1.0E-4"

Keyword: *alpshull*

© 2014 Optimum Structural Design, Inc.

Used to access the results of an ALPS/HULL analysis. Note: all values entered should be in SI (N, m) units.

Sub-keywords to *alpshull:*

- Maximum Bending Moment: *maxbm=*
- Comparison Tolerance: *tol=*
- Incremental Loading Steps: *steps=*
- Bending Moment Curvature: *curve={vertical rotation increment, horizontal rotation increment}*
- Shear Force & Torsional Moment: *shear={vertical shear force, horizontal shear force, torsional moment}*
- Plate Initial Condition: *plateinit={Max deflection/thickness, residual stress/yield stress}*
- Stiffener Initial Condition: *stiffinit={Max deflection/length, residual stress/yield stress}*
- Aluminum Heat Affected Zone: *haz={breadth, yield stress ratio}*

Example: cd ..\..\system modeler92.exe "Dow.mdl" alpshull maxbm=-1.00007E10 tol=1.0E20 steps=200 curv={0.00001,0} shear={0,0,0} plateinit={0.001,0.01} stiffinit={0.01,0.2} haz={0.5,0.3}"

15.2 Programming

MAESTRO exposes the results file (*.rlt) through COM interfaces. Two samples, on written in C# and one written in VC++6 are provided in the Models and Samples/Advanced/ Programming directory.

MAESTRO also provides an example program in C# that shows users how to compare

15.2.1 Read Results

MAESTRO offers extensive post-processing functionality. However, there are specific needs a user may have that is not included in the product. To assist the user in extracting results, MAESTRO exposes the results (*.rlt file) through a COM interface.

To use the MAESTRO COM interface successfully you need to understand:

- C++/C# Programming
- How to access the particular data.

© 2014 Optimum Structural Design, Inc.

Two sample programs are provided in the *Models and Samples/Advanced/Programming/Read Results* directory.

15.2.2 Run MAESTRO Solver

MAESTRO offers batch processing functionality as described in the Batch Processing section. The batch processing described in this particular section describes how a user can access multiple results for custom processing. To assist the user processing multiple entities and load case, a sample program has been provided.

A sample program, written in C# is provided in the *Models and Samples/Advanced/Programming/Run MAESTRO Solver* directory.

© 2014 Optimum Structural Design, Inc.

Tutorials

16

16 Tutorials

Much of the help file is mini-tutorial or how-to based. The following full tutorials present a broader range of MAESTRO features.

16.1 Creating a Mesh in Rhino

Generating a Mesh using Rhino

Preface to Creating Meshes in Rhino to be Used in MAESTRO
It should be noted that aspect ratio affects MAESTRO's Finite Element Analysis results. Therefore, the auto-meshing procedures used in this walk-through should only be used with MAESTRO's hydrostatics solver and MAESTRO Wave.

The recommended procedure using Rhino is listed below:

1. Set the model placement/orientation and units
2. Join surfaces together
3. Create mesh from surfaces
4. Join meshes (if needed)
5. Check for naked edges
6. Fix naked edges where gaps/overlaps in the geometry exist
   a. "Match Mesh Edge" command
   b. Manually adjust mesh vertices
   c. Manually Delete/Add faces to fix mesh
7. Mirror meshes about centerline (if needed)
8. Check for naked edges on centerline; fix if necessary
9. Cull Degenerate Mesh Faces
10. Unify Mesh Normals
11. Check normal direction
12. Save mesh

Step 1. Set the Model Placement/Orientation and Units
Placement/orientation: The hull should be oriented with the X-direction as the longitudinal axis. MAESTRO interprets +Y as the up direction, so it is beneficial to also rotate the model -90 degrees from the typical orientation of +Z as the up direction. The coordinate rotations can also be executed in MAESTRO, however this will only rotate the model geometry; it will not rotate the module's local coordinate system. Hence, all coordinates entered into MAESTRO will be subsequently transformed into the new orientation.

Units: The .ply import setting in MAESTRO allows permits a change to the incoming units in the same dialogue as the orientation. For information on available unit systems see the Help File under **Geometry/Finite Element Modeling > Defining Units**.

Step 2. Join Surfaces Together
All surfaces that are to be meshed should be joined together if possible. This will force the meshing algorithm to use the same points on the common edge(s) between joined surfaces and prevent gaps/naked edges from appearing in the mesh. Any naked edges resulting from surfaces that cannot be joined will be fixed later. Select all surfaces to be joined and execute the Join command or select **Edit > Join** from the menu.

© 2014 Optimum Structural Design, Inc.

Step 3. Create Mesh from Surfaces

Select all surfaces to be meshed and execute the Mesh command or select **Mesh > From NURBS Object** from the menu. A dialog with the meshing parameters will appear where the user can make adjustments and preview the generated mesh. An image of the dialog with suggested settings for a ship (with units set to meters) is shown below in Figure 3.1:

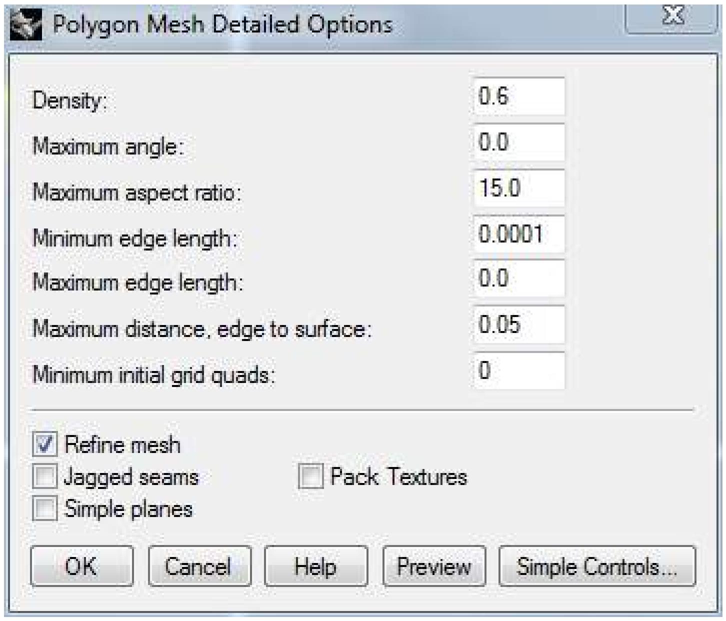


Figure 1.3a: Suggested Meshing Parameters

In addition to this dialogue, clicking simple controls will give an option to use a slider bar and make the process easier. Again, you may preview the meshing precision level before creating the mesh. The listed values were found to generate a reasonable balance between mesh density and accurate representation of the hull shape. Additional information on selecting parameter values can be found in the section **Picking a Mesh Density**, found later in this Appendix. Additional information on the individual meshing parameters can be found in the Rhino Help.

Step 4. Join Meshes (if needed)

Unless all of the ships surfaces were able to be joined prior to creating the mesh, there will be multiple meshes. Select all meshes to be joined and execute the Join command or select **Edit > Join** from the menu.

Step 5. Check for Naked Edges

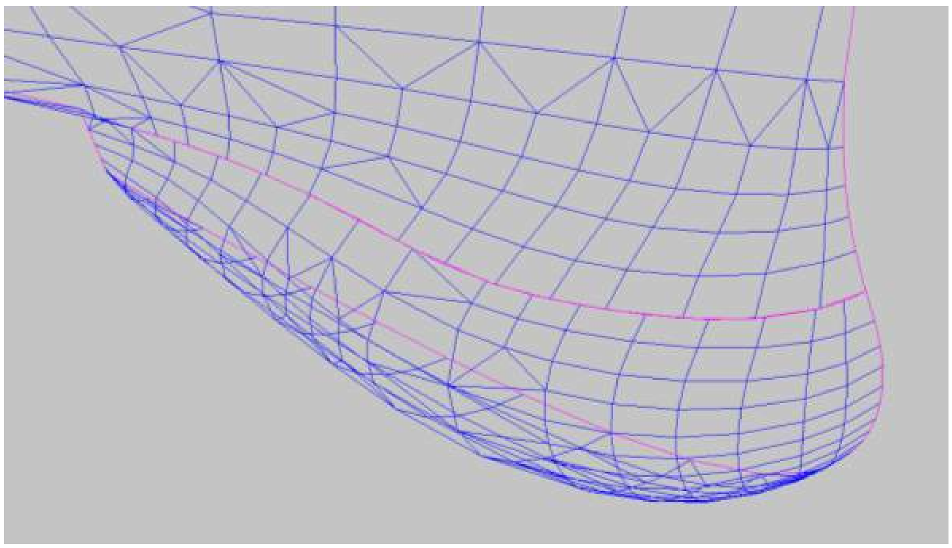
Select the mesh and execute the ShowEdges command or select **Analyze > Edge Tools > Show Edges** from the menu. The number of naked edges found in the mesh will be shown in the command window. The radio button in the dialog should be set to Naked Edges which will be shown in the selected highlight color (usually magenta).

Step 6. Fix Naked Edges Where Gaps/Overlaps in the Geometry Exist

If naked edges are present in the mesh, there are several methods to repair them. Three of the methods are described below and can be used exclusively or in combination.

a. “Match Mesh Edge” command

This is the most automated of the three methods described, but it may require some trial-and-error in setting the DistanceToAdjust parameter. An appropriate DistanceToAdjust will depend on the mesh panel size with the value being only large enough to achieve the desired result (e.g. try 50%-75% of an average panel edge). Additional details about this command can be found in the Rhino help. To repair specific edges, do not pre-select the entire mesh. Execute the MatchMeshEdge command or select **Mesh > Mesh Repair Tools > Match Mesh Edge** from the menu and left-click on PickEdges in the command window or type “p” and press Enter. Select the naked edge(s) to be repaired. Below in Figures 1.6a and 1.6b are before and after images of using the Match Mesh Edge command.



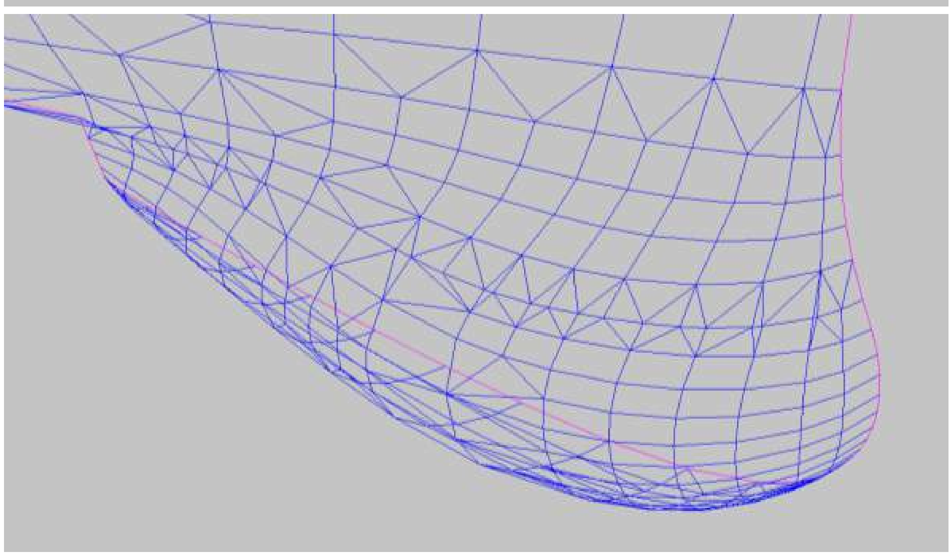
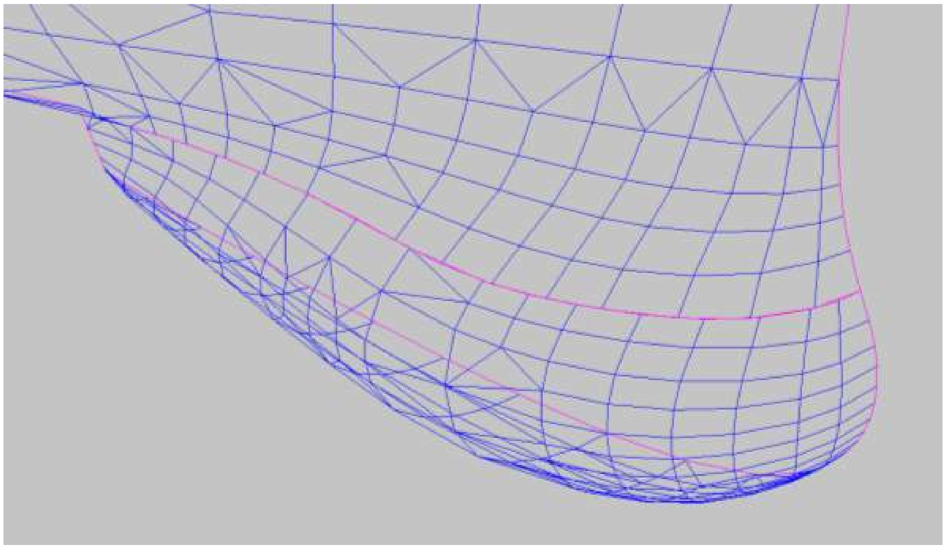


Figure 1.6a: Bulb with Mesh Discontinuity Figure 1.6b: Bulb after Match Mesh Edge Command

b. Manually Adjust Mesh Vertices

The user can manually adjust the locations of the mesh vertices along the naked edge to create a point-to-point connection between the two mesh edges. To turn on the mesh vertices, select the mesh and use either the PointsOn command, select **Edit > Control Points > Control Points On** from the menu or simply press F10. The mesh vertices along the naked edge can now be dragged and dropped onto other existing vertices. (Make sure that Object Snap for points is enabled.) Below in Figures 1.6c and 1.6d are images of the naked edge before and after manually relocating the vertices.



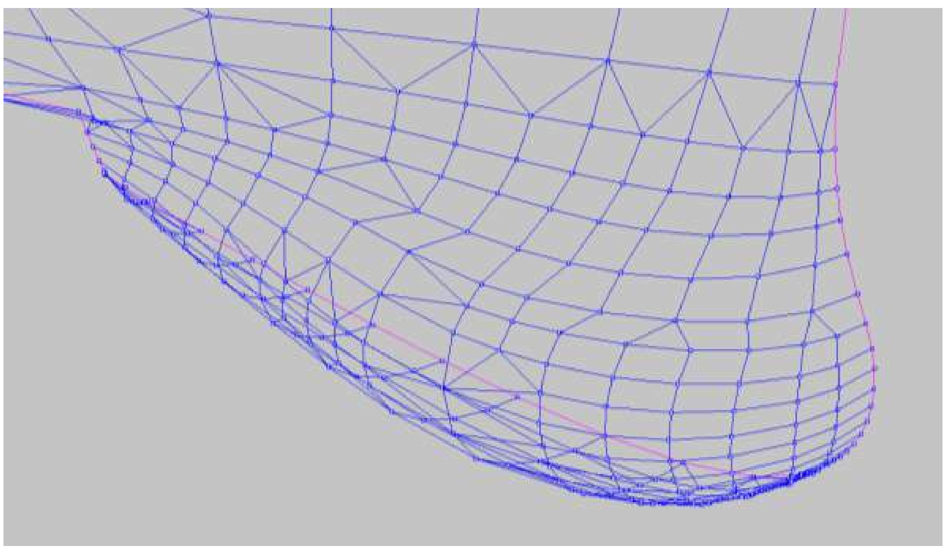


Figure 1.6c: Bulb with Mesh Discontinuity Figure 1.6d: Bulb after Manually Adjusting Mesh Vertices

c. Manually Delete/Add faces to Fix Mesh

The user can manually delete a row of faces along the naked edge and recreate them either manually with the Patch Single Face command or automatically by using the Fill Hole command. It is easiest to select faces for deletion if viewing the shaded model. Meshes can be deleted by using the DeleteMeshFaces command or by selecting **Mesh > Mesh Edit Tools > Delete Mesh Faces** from the menu. Below in Figure1.6e is an image of a naked edge where a row of faces has been deleted.

© 2014 Optimum Structural Design, Inc.

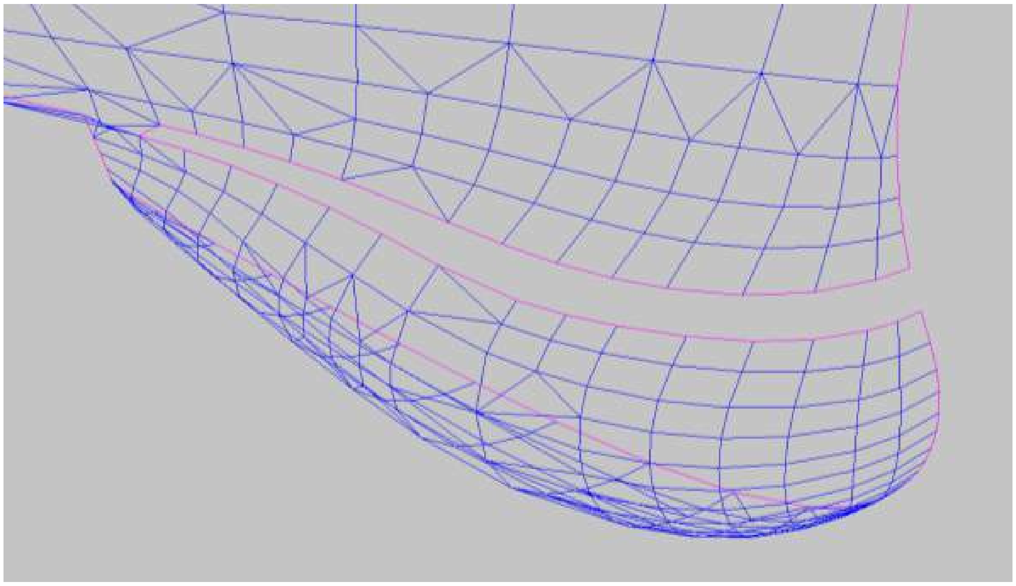
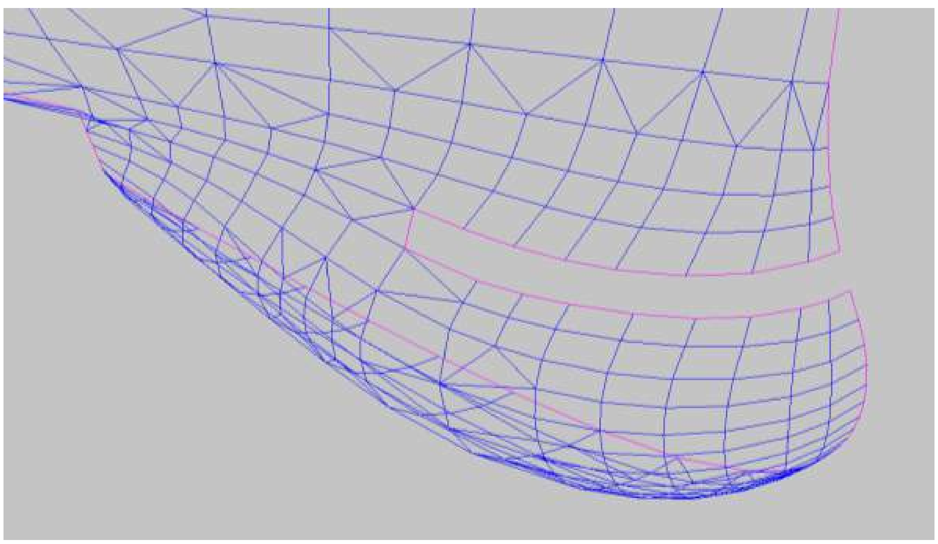


Figure 1.6e: Naked Edge with 1 Row of Mesh Faces above Edge Deleted

For areas where the user may want direct control, mesh faces can be created manually using the PatchSingleFace command or by selecting **Mesh > Mesh Repair Tools > Patch Single Face** from the menu. When prompted, the user selects two existing mesh face edges to define a new mesh face. If the edges share a vertex, a triangle will be created, otherwise a quad will appear. (Only one face is created each time the command is run, but right-clicking in a viewport will repeat the last command to speed up the process.) Below in Figures 1.6f and 1.6g are images of the mesh being patched manually.



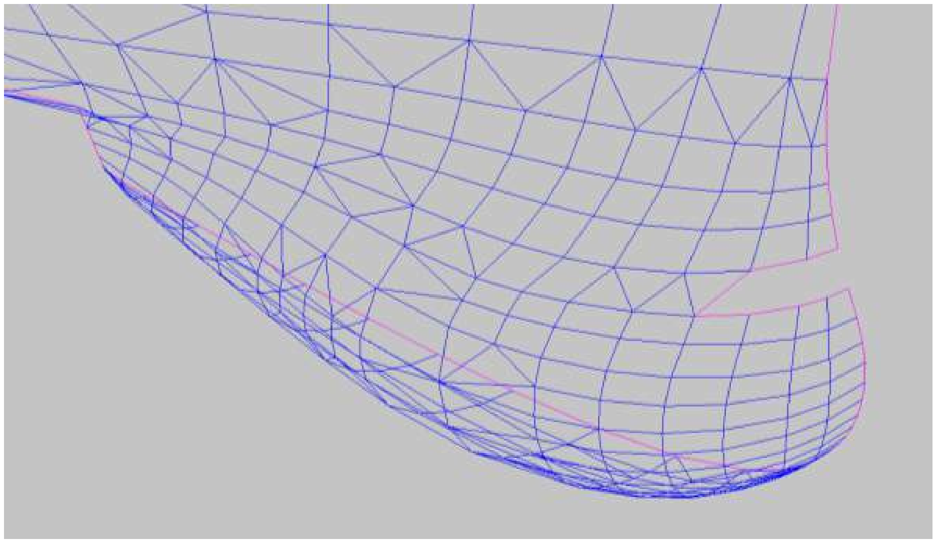
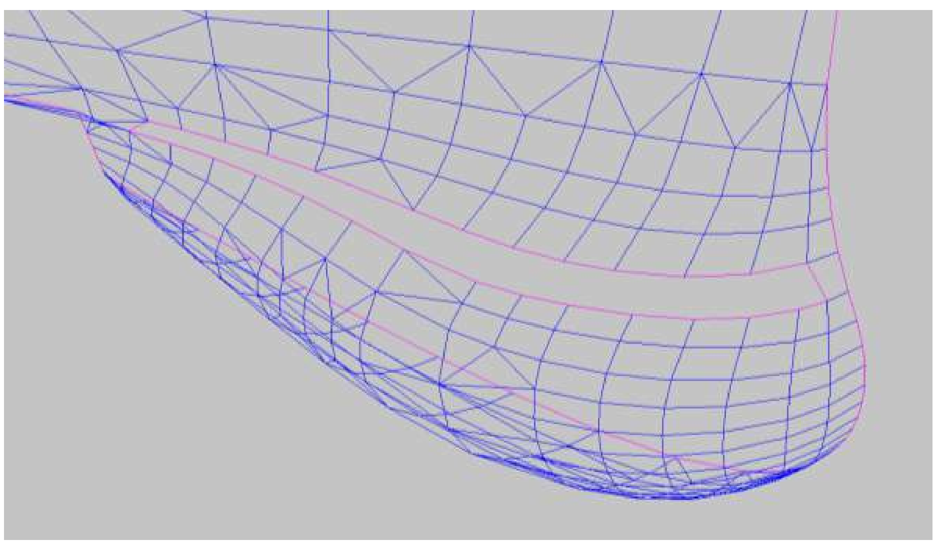


Figure 1.6f: Manually Patching Figure 1.6g: Manually Patching All the Way to the Perimeter

We may continue to mesh all of the way to perimeter by creating individual mesh faces or now we may use an alternate method. We choose to take advantage of the FillMeshHole by calling it from the command line or selecting **Mesh > Mesh Repair Tools > Fill Hole**from the menu. When prompted, select an edge along the boundary of the hole in the mesh. Below in Figures 1.6h and 1.6i are images of the manually added face to enclose the hole and of the result from the FillMeshHolecommand



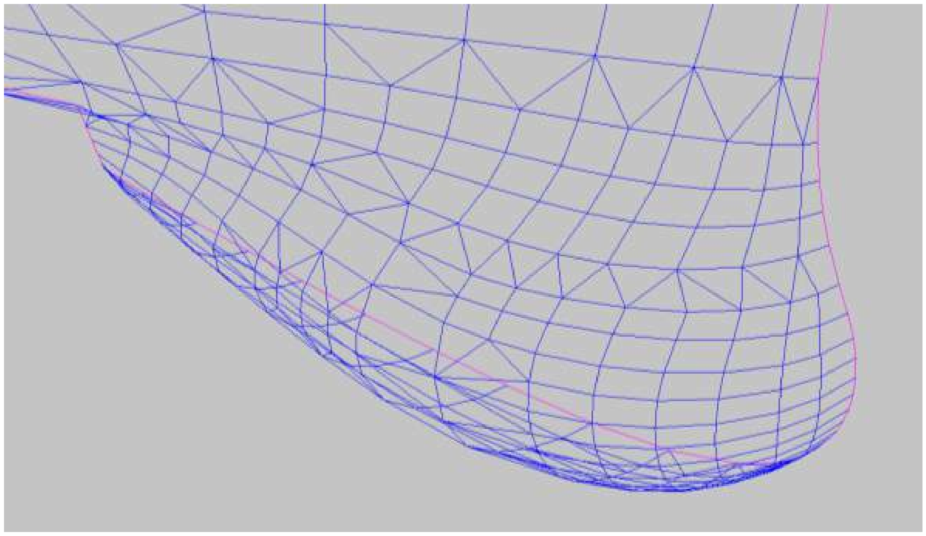


Figure 1.6h: Closing Perimeter with 1 Mesh Face Figure 1.6i: Mesh after Fill Mesh Hole

Note: If a mesh edges appears to be joined and have point-to-point connectivity, but still reports a naked edge, this can normally be fixed with the AlignMeshVertices command (**Mesh > Mesh Repair Tools > Align Mesh Vertices** in the menu). See the Rhino help for details on this command. Just execute the command, click on SelectNakedEdges and then select the edge to repair.

Step 7. Mirror Meshes about Centerline and Join (if needed)
If only half of the ship was modeled, the user will have to mirror the mesh about the centerplane to create a closed volume representing the full ship. This can be done by selecting the newly generated mesh(es) and typing Mirror at the command prompt or selecting **Transform > Mirror** from the menu.

Step 8. Check for naked edges on centerline; fix if necessary
If the hull was mirrored and joined in Step 7, check again for naked edges. If the centerline mesh edges appear to be joined and to have point-to-point connectivity, but still reports a naked edge, this can be fixed with the AlignMeshVertices command (**Mesh > Mesh Repair Tools > Align Mesh Vertices** in the menu). See the Rhino help for details on this command. Just execute the command, click on SelectNakedEdges and then select the edge to repair.

Step 9. Cull Degenerate Mesh Faces
Once all naked edges have been eliminated and the mesh forms a closed volume, select the mesh and execute the CullDegenerateMeshFaces command or select **Mesh > Mesh Edit Tools > Cull Degenerate Mesh Faces** from the menu. This will remove mesh faces with zero area and edges with zero length. While the geometry will not change, there will be a message in the command window indicating if any mesh faces were modified or removed.

Step 10. Unify Mesh Normals
Unlike joining surfaces in Rhino, joining meshes does not reset the normal direction of each component mesh to a consistent direction. To ensure the mesh normals are consistent, select the mesh and execute the UnifyMeshNormals command or select **Mesh > Mesh Repair Tools > Unify Normals** from the menu. A message in the command window will

© 2014 Optimum Structural Design, Inc.

indicate how many mesh faces were updated. It is best to rerun this command until the message indicates: “All face normals are already oriented in the same direction.” (Mesh normals toggling back and forth may indicate the presence of a degenerate mesh face or edge.)

Step 11. Check normal direction

Mesh normals can be checked by selecting the mesh and executing the Dir command or selecting **Analyze > Direction** from the menu. Vectors representing the mesh normal direction will appear intermittently on the mesh. Mesh normals should be pointing inward, towards hull, because MAESTRO automatically flips surface normal from Rhino. If they are not, left-clicking on the mesh or typing “f” on the command line with Enter will reverse the normal direction of the mesh. Right-clicking or pressing Enter at an empty command line will complete the command.

Since each individual mesh face can have its own normal direction, it is safest to check the normal direction using shading. To turn on shading, enter the perspective view and execute the Shade command or select **View > Shaded** from the menu. To change the color of the backfaces, execute the Options command or select **Tools > Options…** from the menu. In the tree, navigate to **Rhino Options > Appearance > Advanced Settings > Shaded**. Change the Backface settings to use “Single Color for all backfaces” and set the color to something conspicuous like Magenta as shown below in Figure 1.11a.

© 2014 Optimum Structural Design, Inc.

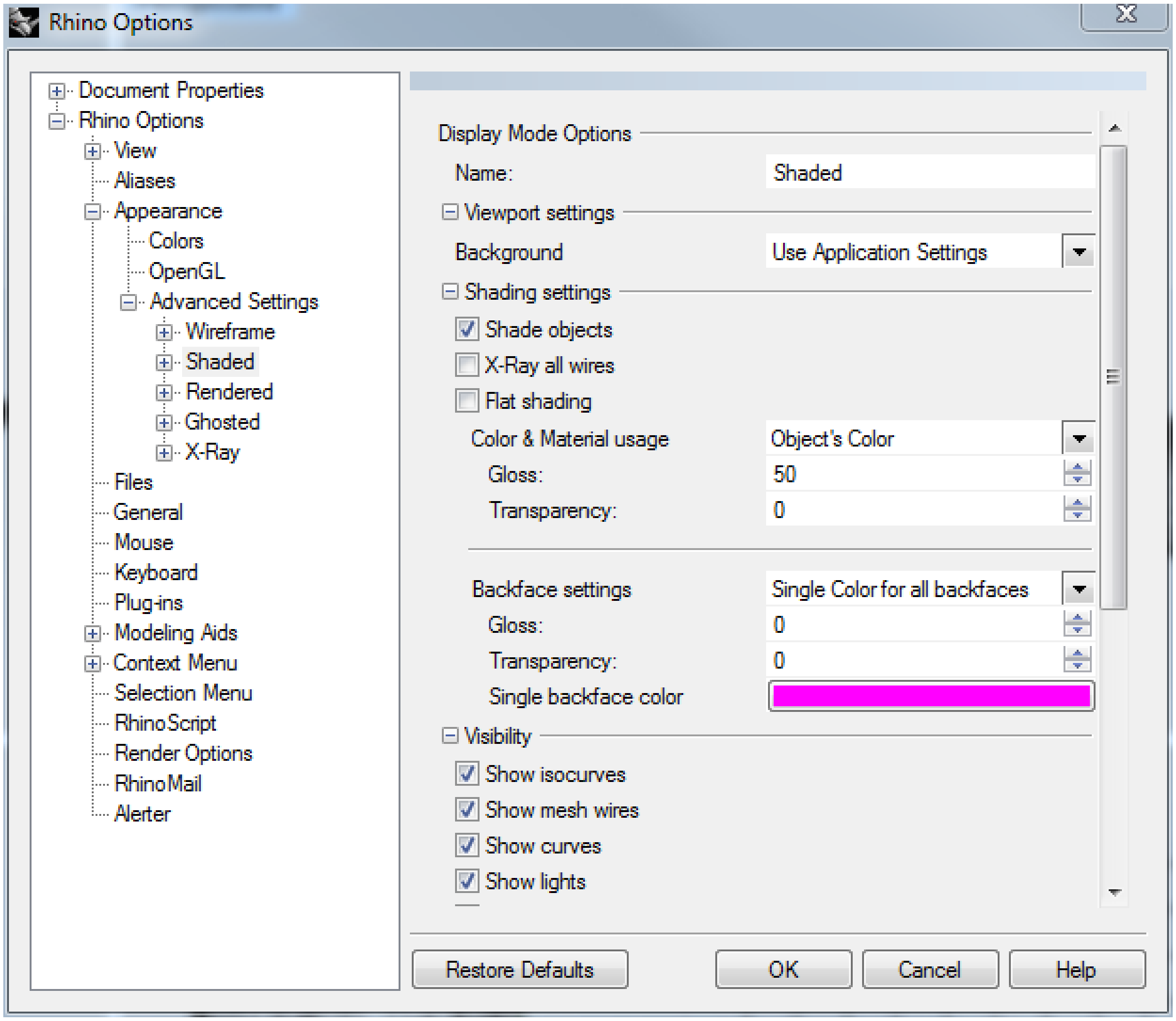


Figure 1.11a: Backface Color Settings

Again, as MAESTRO auto flips surface normal, the backface should be facing outward (towards the water) and the entire hull should be shaded magenta. It is worthwhile to rotate the hull and verify that all panels are magenta.

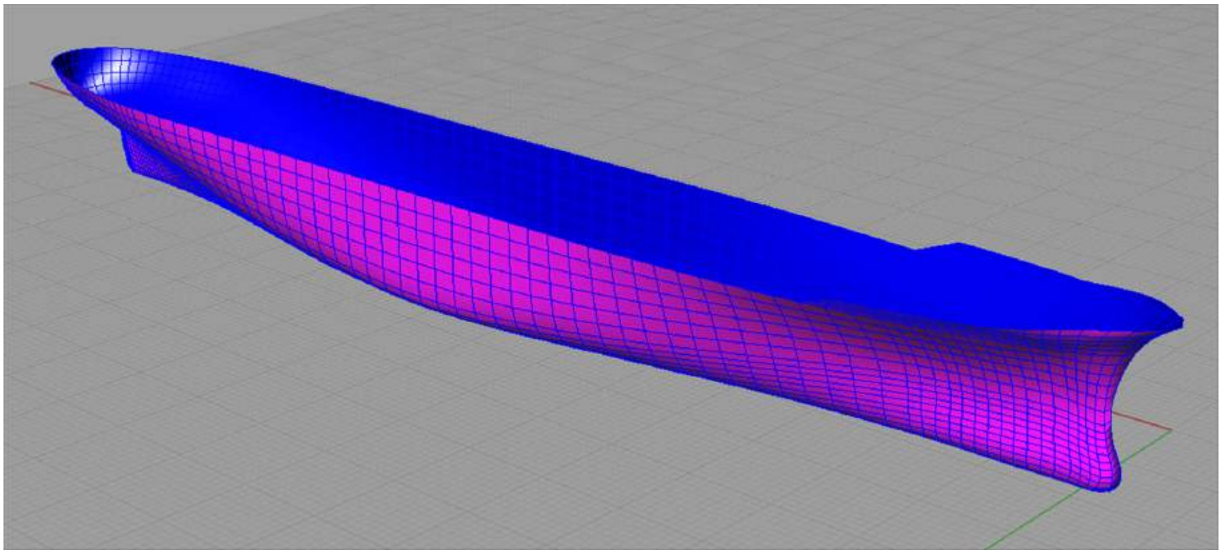


Figure 1.11b: Backface Oriented Outwards Showing Inward Normal

Step 12. Save mesh
If the mesh is the only item in the model (including hidden items or layers that are turned off), it can be saved using the SaveAs command or **File > Save As…** from the menu. Otherwise, select the mesh and execute the Export command or **File > Export Selected…** from the menu. The file type should be set to "PLY – Polygon File Format (*.ply)". After clicking the Save button, an Export PLY dialog will appear. In this final dialog, ASCII should be the selected format with no additional options checked as shown below in Figure 1.12a.

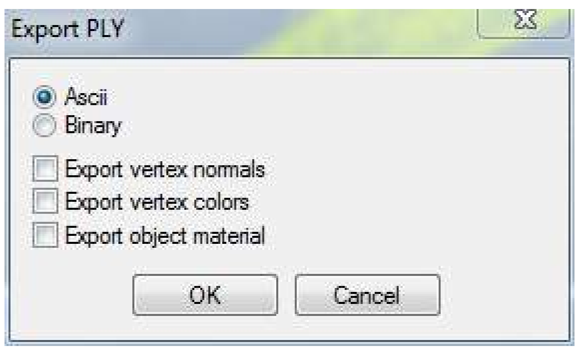


Figure 1.12a: .PLY Export Settings

16.2 Midship Design Tutorial

Click here to launch the Midship Design Tutorial document.

This tutorial will walk through the complete process of creating and analyzing a midship section in MAESTRO given a basic set of design parameters and hull geometry. To use this tutorial, you are expected to have a general knowledge of the MAESTRO graphic user interface (GUI) and MAESTRO elements (i.e., substructures, modules, strakes, etc.). If you are not comfortable with these areas, please first review the General section of the

MAESTRO help file.

16.3 Basic Features

This Tutorial is intended to do two things simultaneously:

- to demonstrate and explain some of the basic features and options of the Modeler, and
- to go through the steps of actually building a small model, starting with ¼ of the engine room of a small naval vessel.

16.4 ALPS/ULSAP

The following tutorials will walk through using and modifying an existing ULSAP model provided in the MAESTRO distribution directory and creating an ULSAP model from scratch. For more information, see the ALPS/ULSAP section above.

Modifying and Using an Existing ULSAP Model

Creating a new ULSAP Model

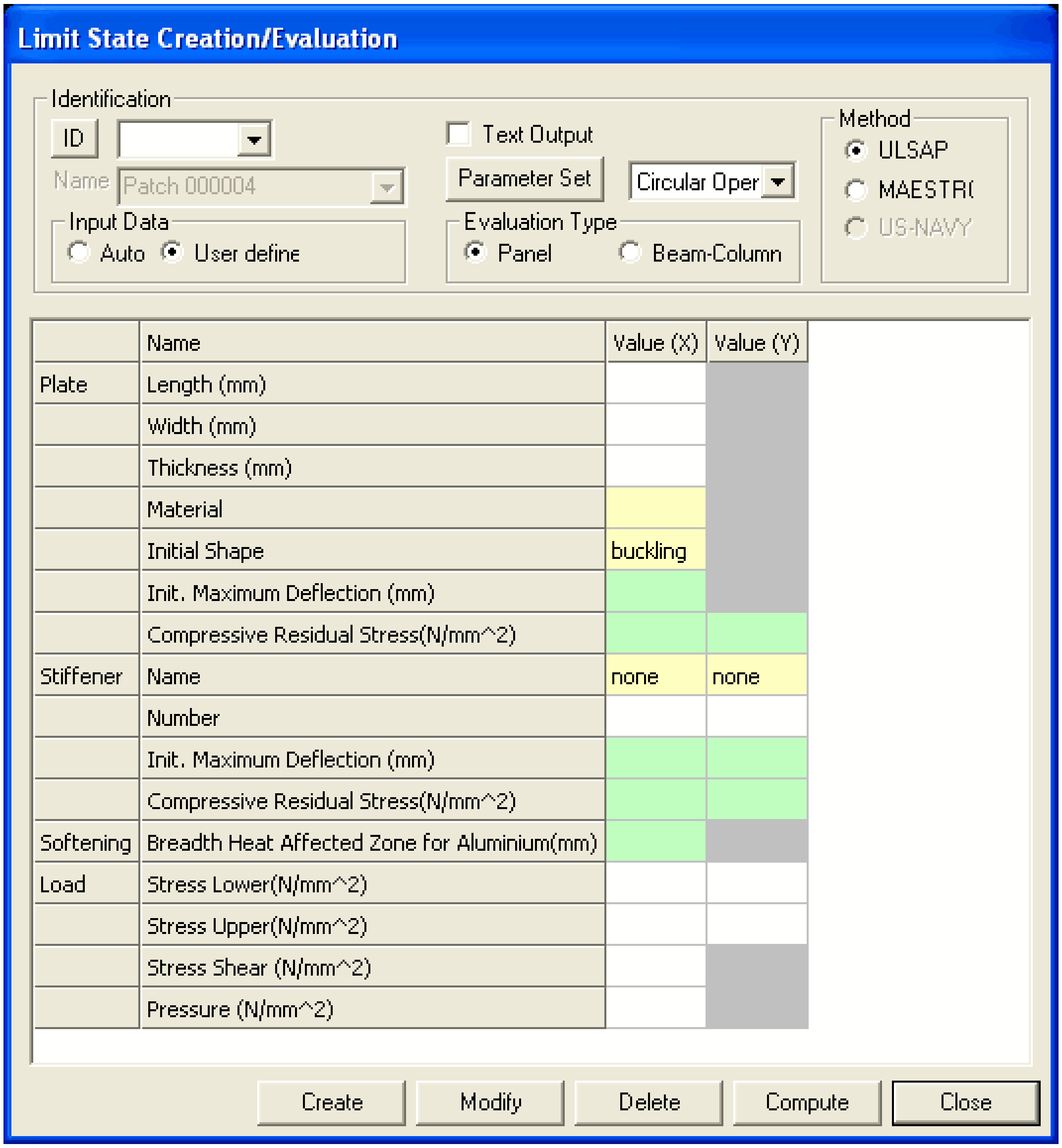
Text Out (All Panels) Option

Modifying and Using an existing ULSAP Model

1. Open the model UlsapSample.mdl from ...\MAESTRO\Models and Samples\ALPS \ULSAP directory.

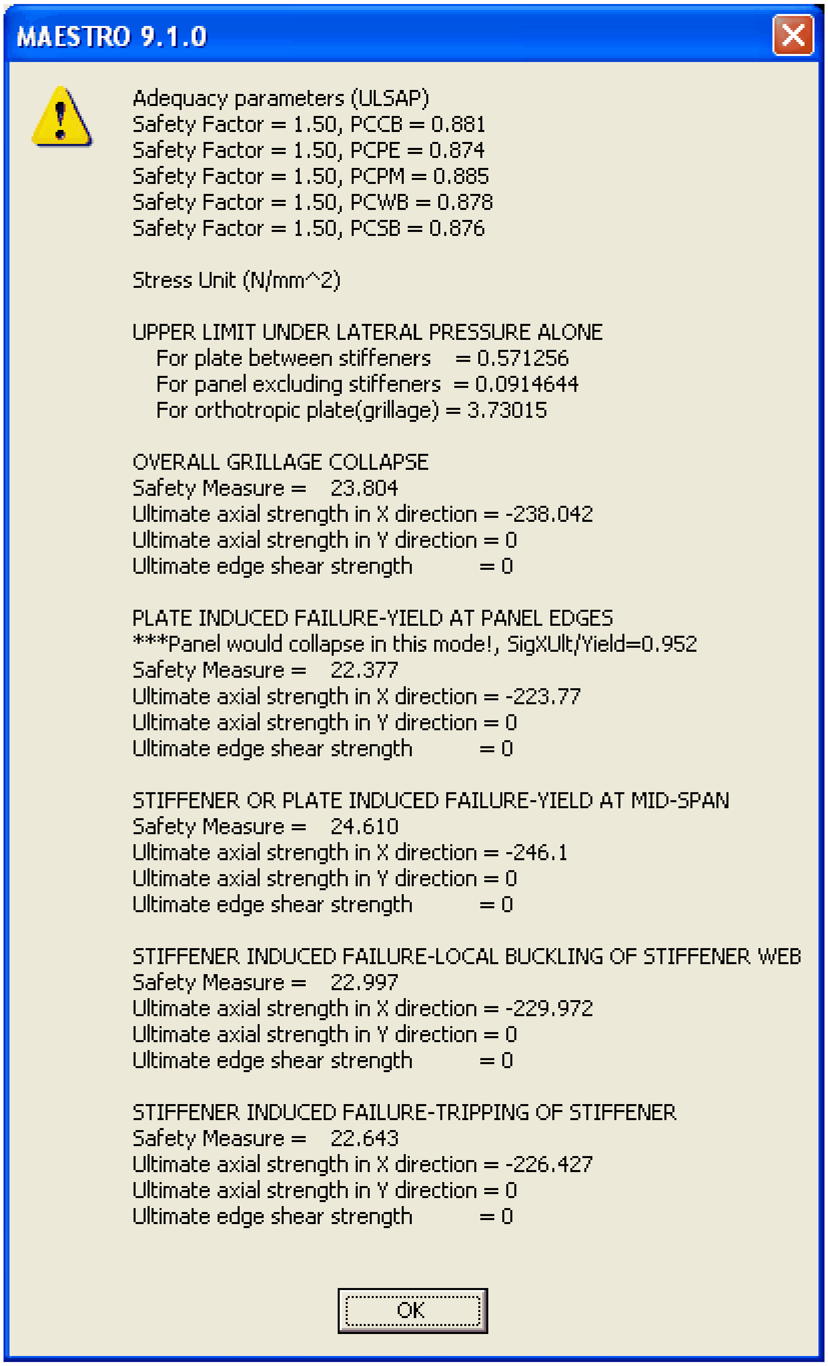
Note: the Modeler window appears to be empty since the model is not associated with any visible geometry.

2. Launch the ULSAP Evaluation Dialog from Model > Evaluation Patch > Create/Evaluate.



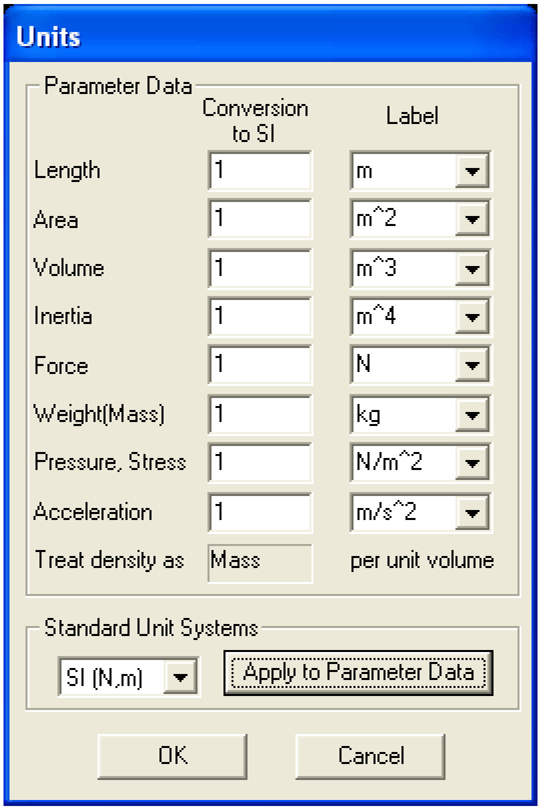
3. Select a panel from the drop-down box next to ID.
4. Click the "Compute" button. The following results dialog for that patch will open.

© 2014 Optimum Structural Design, Inc.

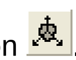


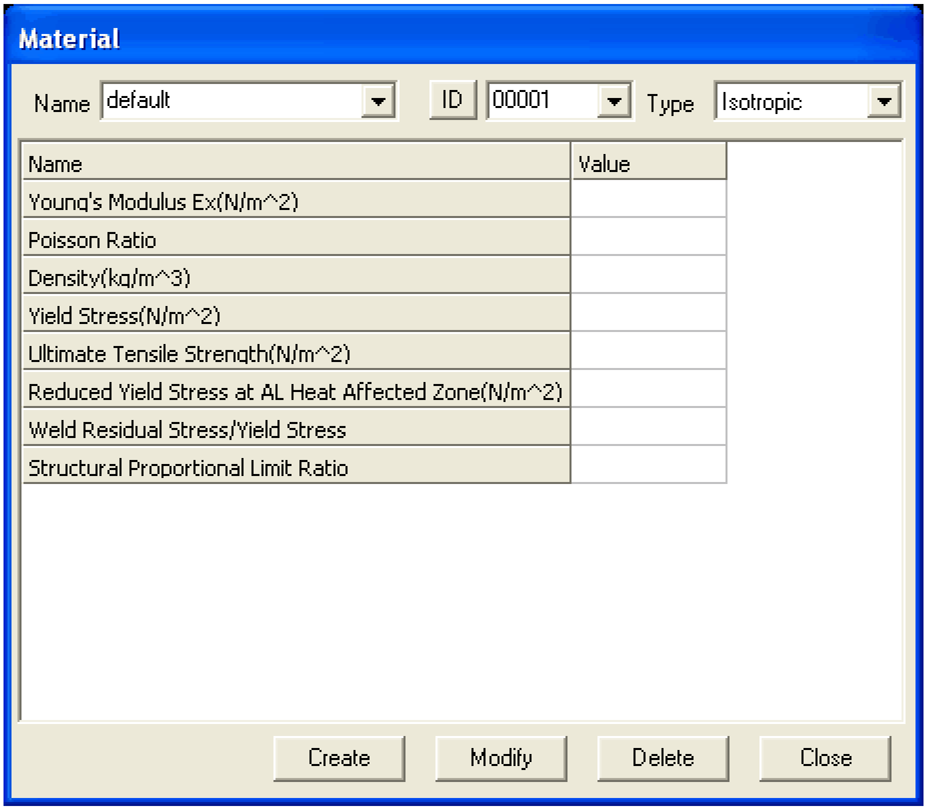
Creating a New ULSAP Model

1. Open a new file from File > New.
2. Open the Units dialog from File > Units...



3. From the drop-down box under Standard Unit Systems, select "SI (N,mm)" and click Apply to Parameter Data. This will set the units to SI using Newtowns and meters.

4. Open the Material dialog from Model > Materials... or clicking the material icon .



5. Click the ID button to create a new material.
6. Enter the following data and then click Create:

Name: ULSAP Material

Type: Isotropic

Young's Modulus: 2.04e+011

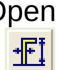
Poisson Ration: 0.3

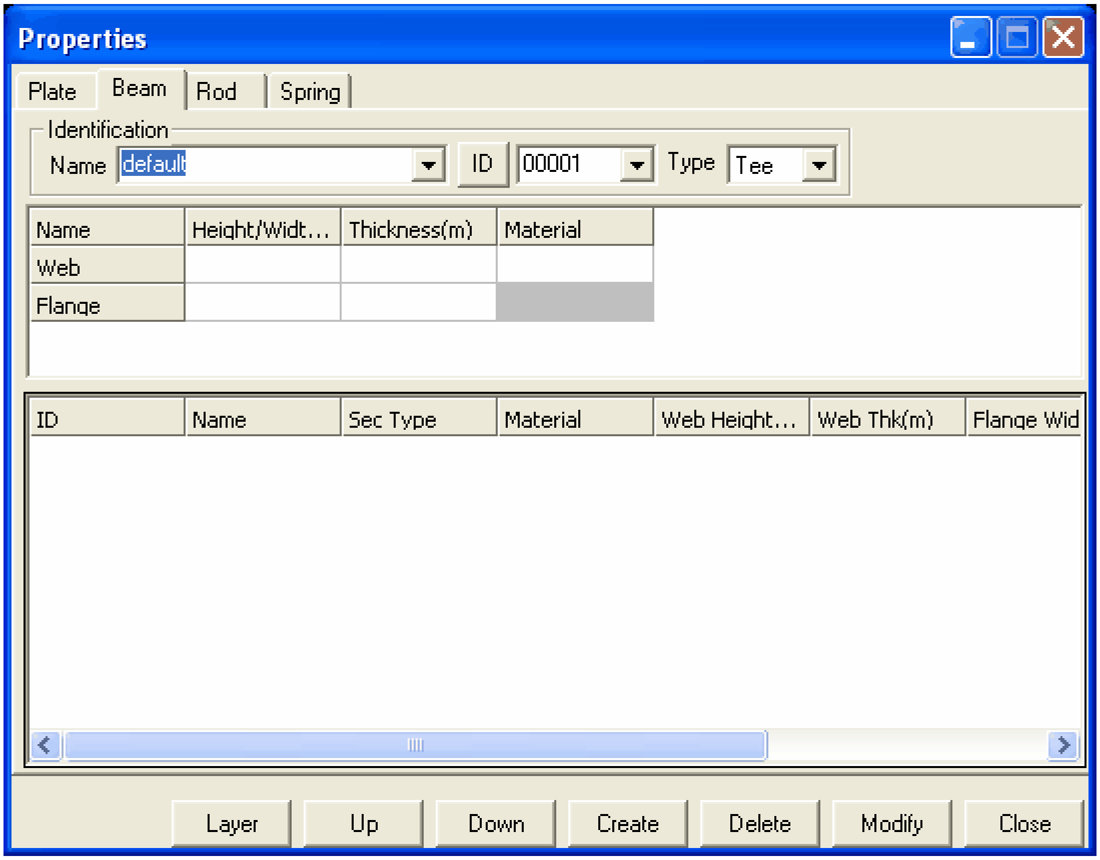
Density(kg/m^3): 7850

Yield Stress(N/m^2): 2.35e+008

Ultimate Tensile Strength(N/m^2): 4e+008

Reduced Yield Stress at AL Heat Affected Zone(N/m^2): 2.35e+008, Right-clicking in the value box will pop up a dialog with typical values:

7. Open the Properties dialog from Model > Properties > Beam or clicking the Properties icon .



8. Click ID to create a new beam property. Click No when prompted whether you would like to copy an existing property.

9. Enter the following data and click Create:

Name: 360x12-100x15-T(mm)

Type: select "Tee"

Web Height: 0.360

Web Thickness: 0.012

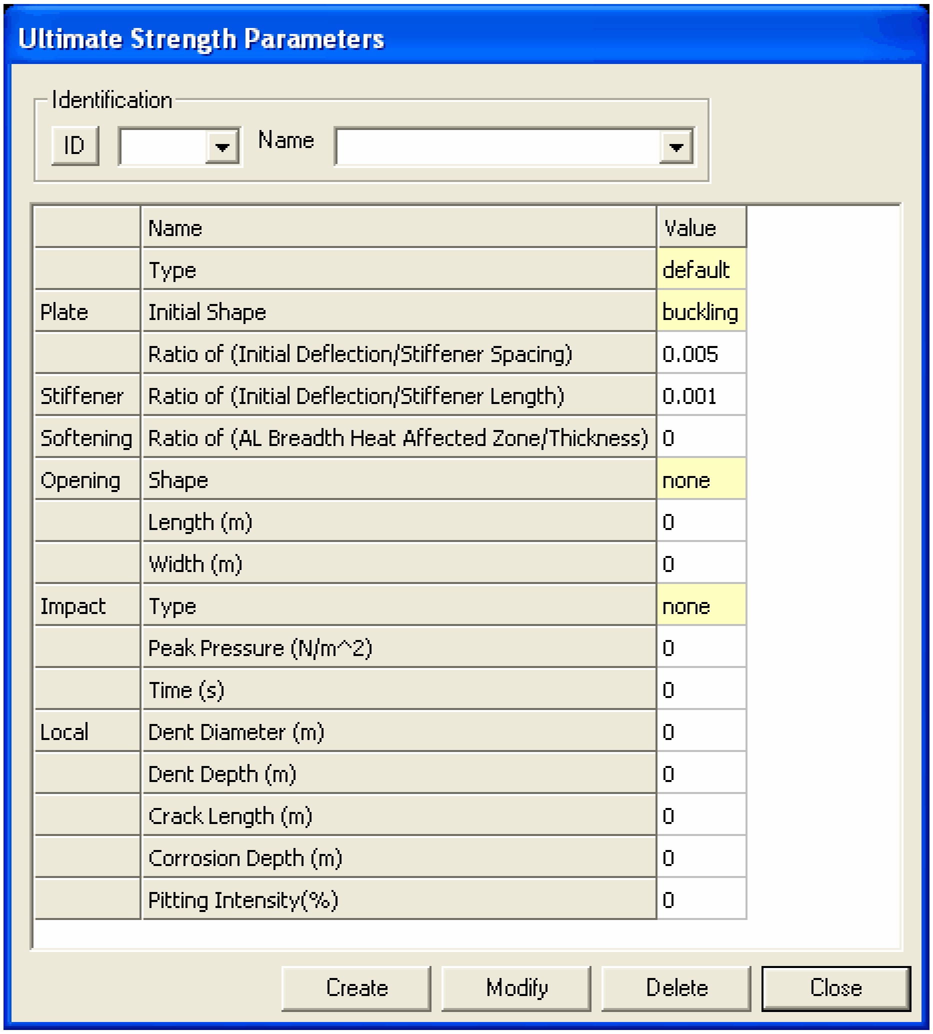
Flange Width: 0.10

Flange Thickness: 0.015

Material: Select "ULSAP Material"

10. The ULSAP Parameters can be changed by selecting Model > ULSAP Parameters.

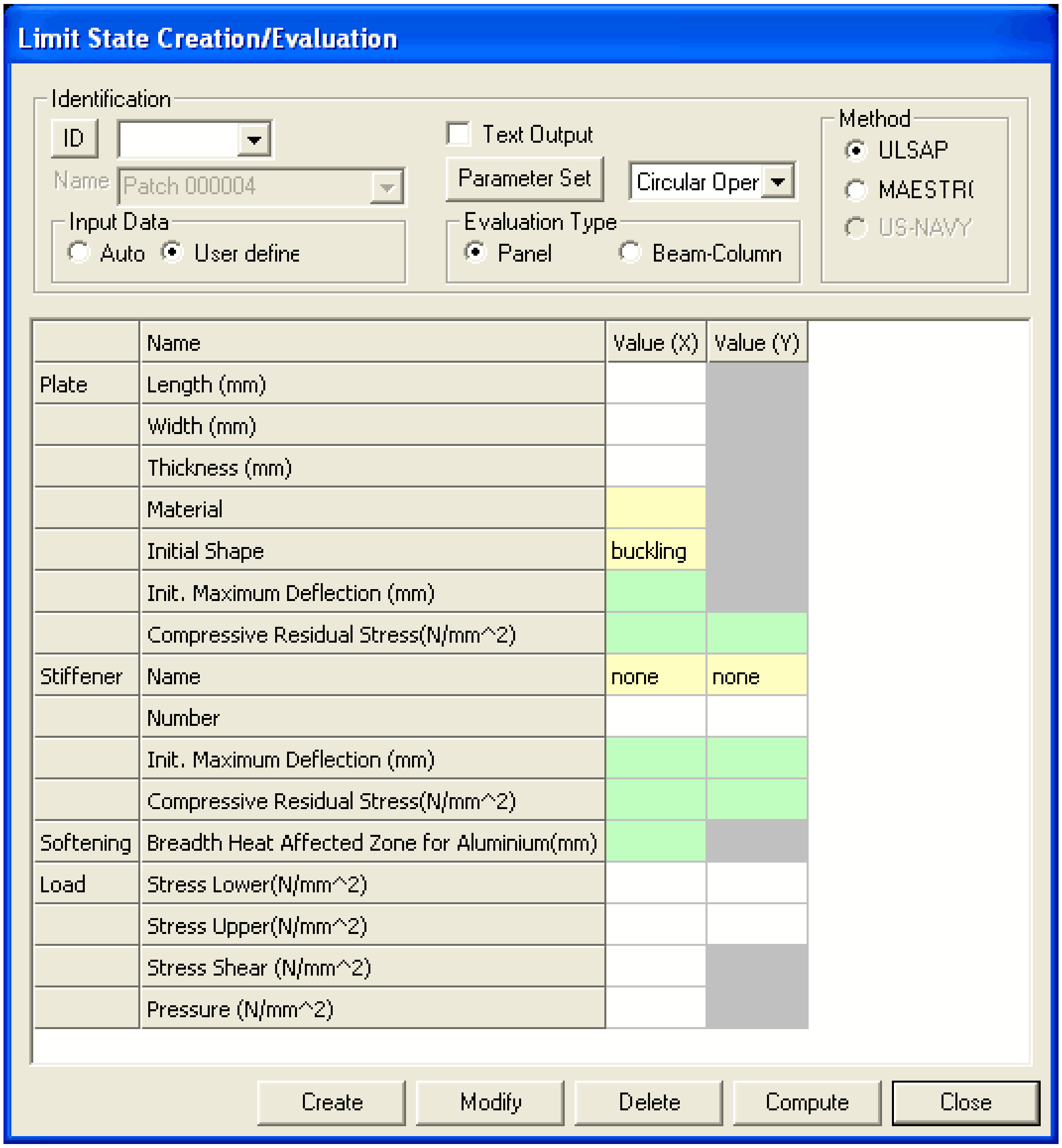
© 2014 Optimum Structural Design, Inc.



We will use the default values for this tutorial, so click Close.

11. Launch the ULSAP dialog from Model > Evaluation Patch > Create/Evaluate.

© 2014 Optimum Structural Design, Inc.



12. Click the ID button to create a new panel.

13. Select "User Define" under Input Data.

14. Select "None" under Parameter Set. If ULSAP parameters were previously defined in step 10, you could select those here.

15. Select the Text Output checkbox to get a complete text result. (Double-click this checkbox for the "Text Out (All Panels)" option)

16. Enter the following data and click Create:

Plate

Length: 2.64

Width: 3.6

© 2014 Optimum Structural Design, Inc.

Thickness: 0.021

Material: ULSAP Material

Initial Shape: buckling

Init. Maximum Deflection: 0.0045

Compressive Residual Stress:

Stiffener

Name: Select 360x12-100x15-T(mm)

Number: 3

Init. Maximum Deflection: 0.00264

Compressive Residual Stress:

Breadth Heat Affected Zone for Aluminum:

Load (A figure of these stress definitions can be found below)

Stress Lower: -1e+07

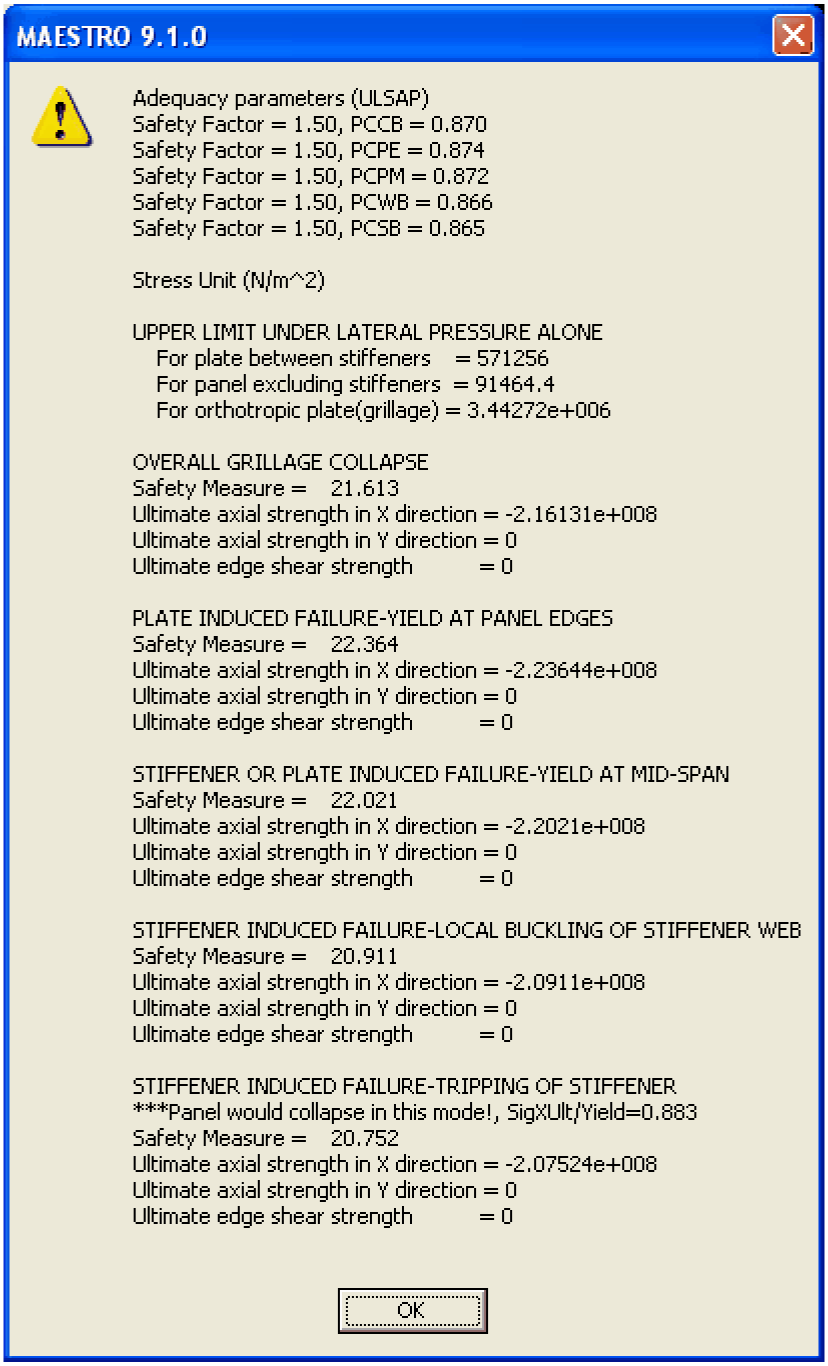
Stress Upper: -1e+07

Stress Shear:

Pressure:

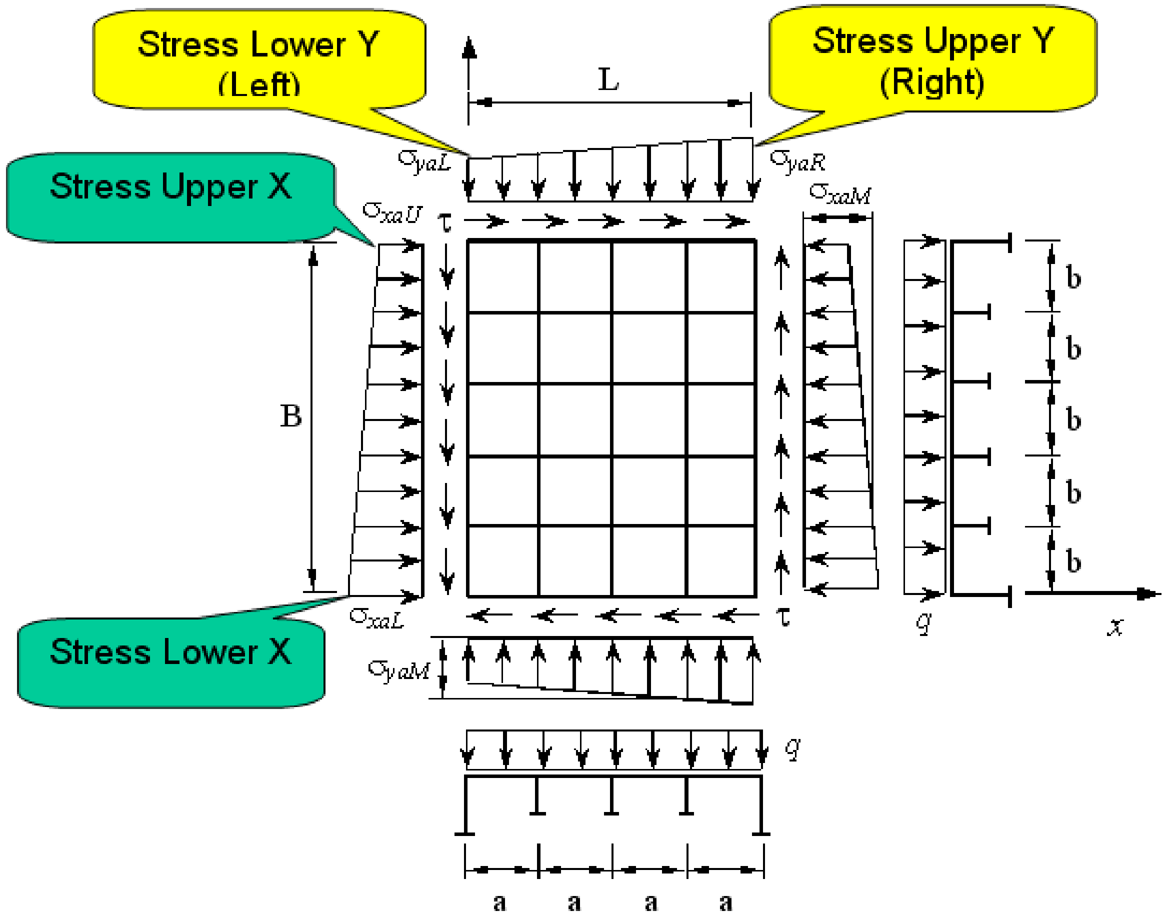
The material and stiffener properties can be queried within the dialog by right-clicking inside the appropriate cell.

17. Click the Compute button to obtain the results.



The output tab will provide a full text output of the input data and the results.

Stress Definition:



Text Out (All Panels) Options

The *Text Out (All Panels)* option all generates a list of all the panels results into the *Output* tab. The results are generated when the user clicks the *Compute* button. The results of the evaluation patch are presented first and followed by the beam evaluations.

16.5 ALPS/HULL

NOTE:

The sample ALPS/HULL analysis models used in this tutorial are *DryDock_1.mdl* and *DryDock_2.mdl,* can be found in the Models and Sample directory. For more information, see the ALPS/Hull section above.

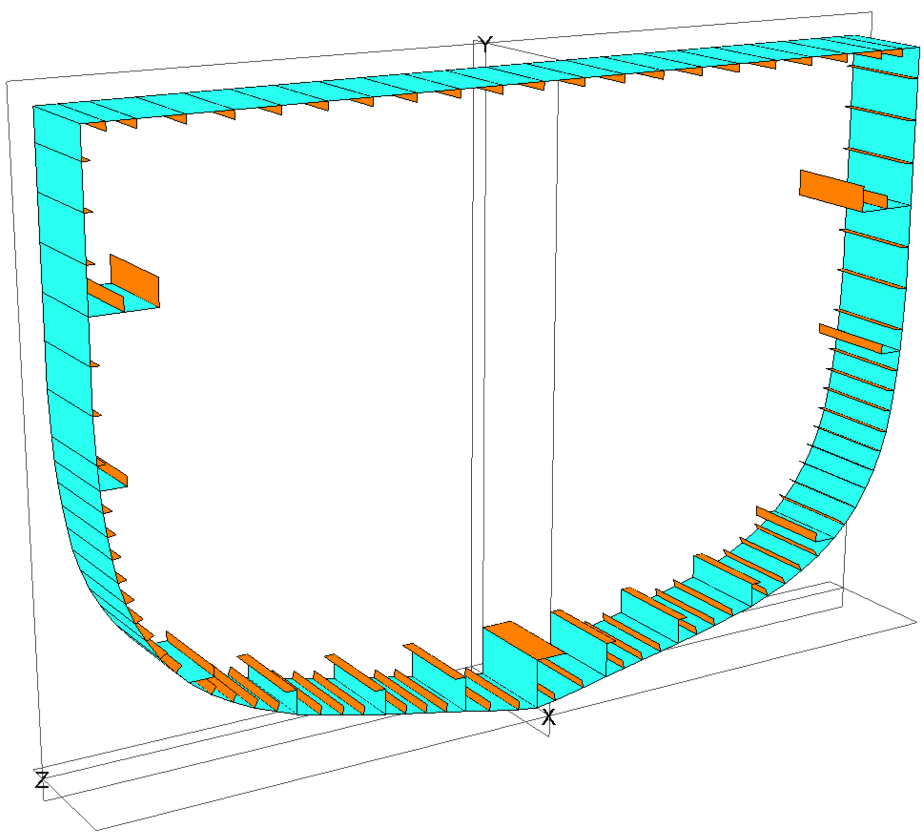
Analysis Setup

Additional Pressure Loading

Post-Processing

Creating an ALPS/HULL Model

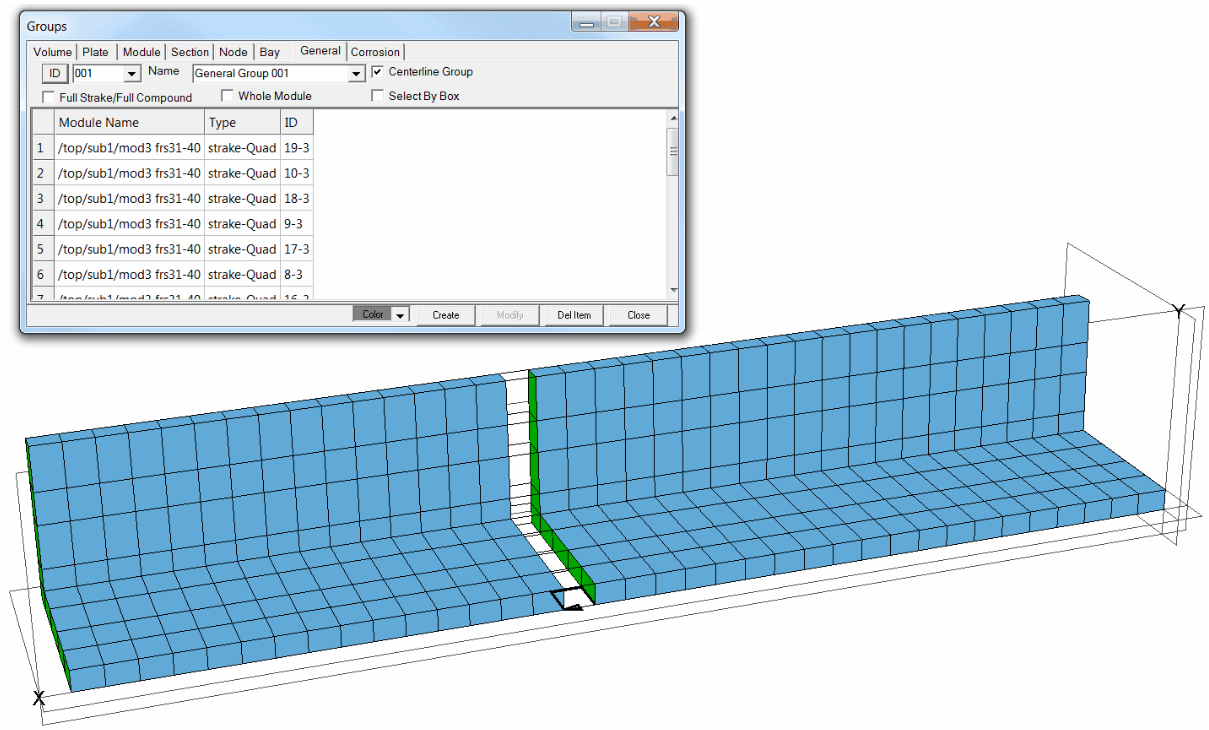
The progressive hull girder collapse model is a hull girder cross sectional module with one frame length. This "one bay" module should only consist of quad and bar elements, such as the one shown below. The plates and webs of the longitudinal girders should be modeled as non-stiffened quads (bare plate) while stiffeners and girder flanges can be modeled as bar elements. The hull girder collapse model can be generated manually or automatically from the global model by using MAESTRO's refinement functionality.



1. Open the sample *DryDock_1.mdl,* which is located in the *...\MAESTRO\Models and Samples/ALPS/Hull* directory. You will notice that this model is a half-model, which is a common starting point for users. The steps below will demonstrate how to arrive to a full-model, which is required for the progressive hull girder collapse analysis.

© 2014 Optimum Structural Design, Inc.

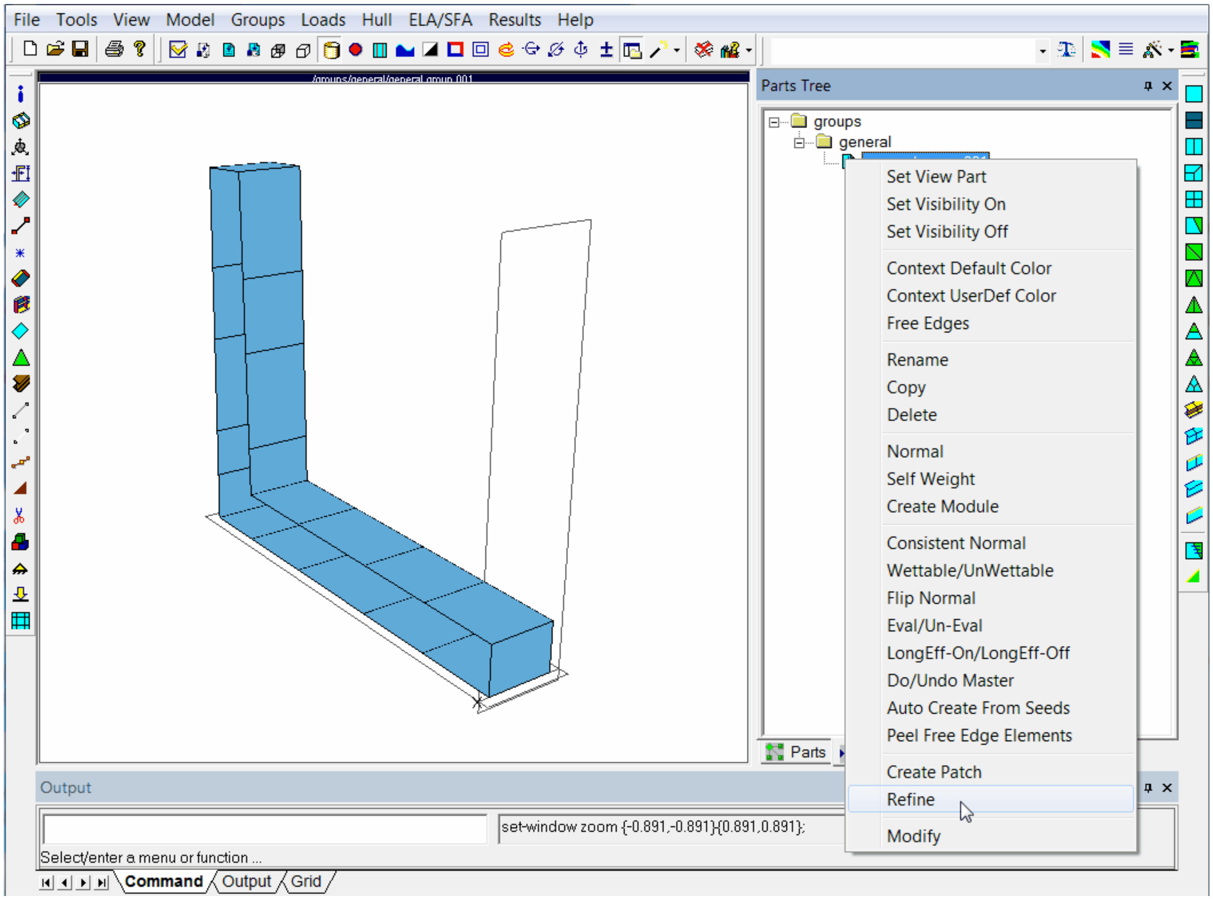
2. Create a general group of the section (i.e., one frame spacing) of interest from the global model as shown below. Be sure to check the "Centerline Group" box before creating the group. Click here to learn about creating general groups.

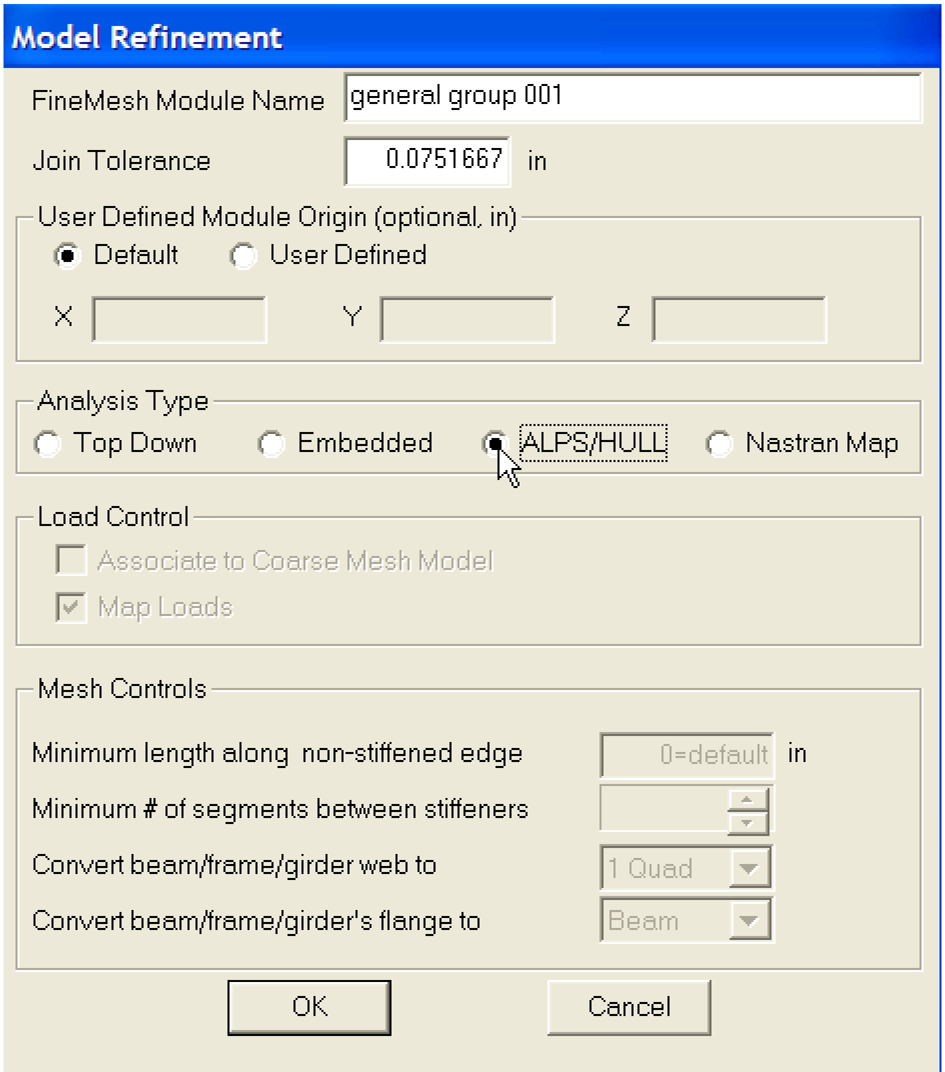


3. View the newly created general group by right-clicking on the group in the Groups Tree and click on "Set View Part." If necessary, modify the general group to capture all the elements participating in the progressive hull girder collapse analysis.

4. Right-click on the general group again and choose **Refine** from the menu, as shown below. This will bring up the Model Refinement dialog box, shown in the second figure below.

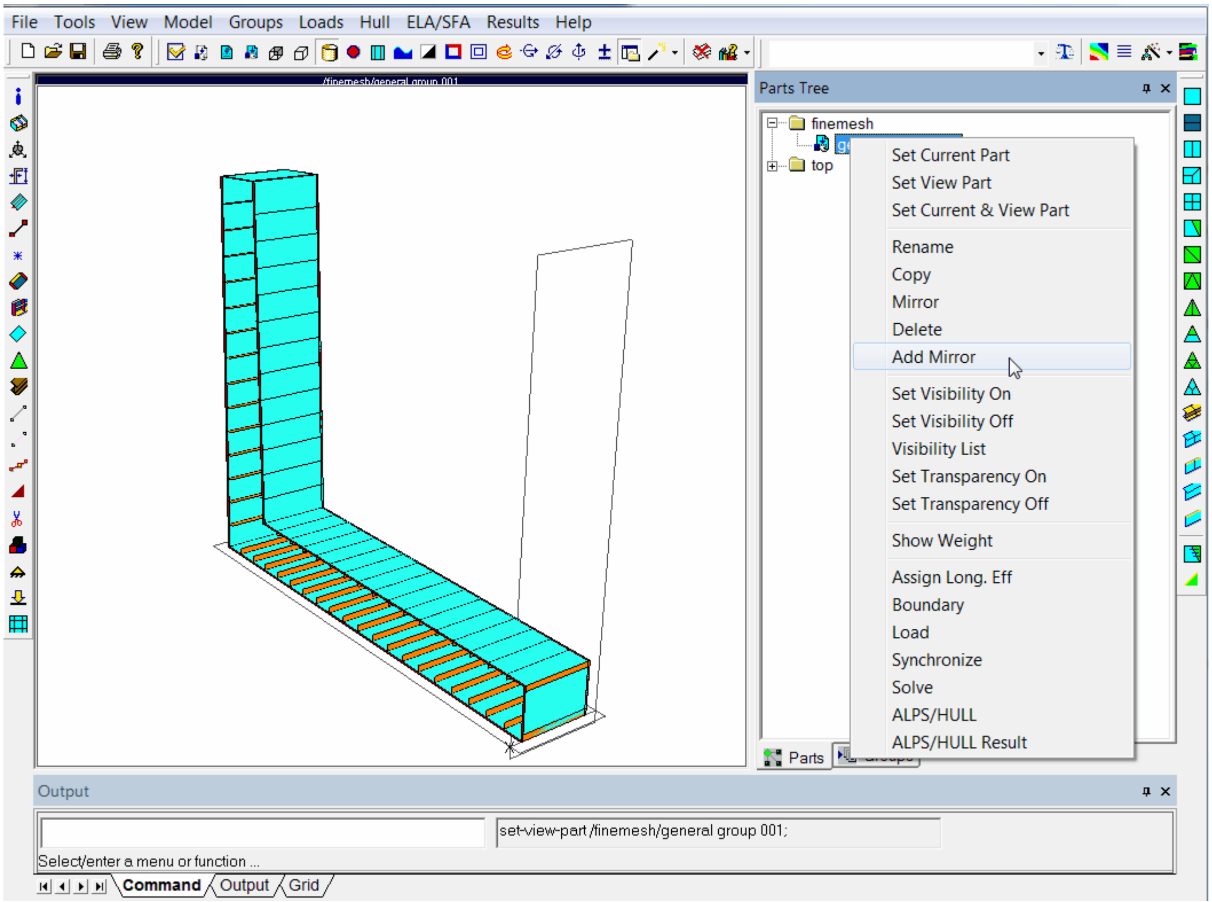
© 2014 Optimum Structural Design, Inc.


File Tools View Model Groups Loads Hull ELA/SFA Results Help
Parts Tree
groups
general
Set View Part
Set Visibility On
Set Visibility Off
Context Default Color
Context UserDef Color
Free Edges
Rename
Copy
Delete
Normal
Self Weight
Create Module
Consistent Normal
Wettable/UnWettable
Flip Normal
Eval/Un-Eval
LongEff-On/LongEff-Off
Do/Undo Master
Auto Create From Seeds
Peel Free Edge Elements
Create Patch
Refine
Modify
Output
set-window zoom {-0.891,-0.891}{0.891,0.891};
Select/enter a menu or function ...
Command
Output
Grid
Parts

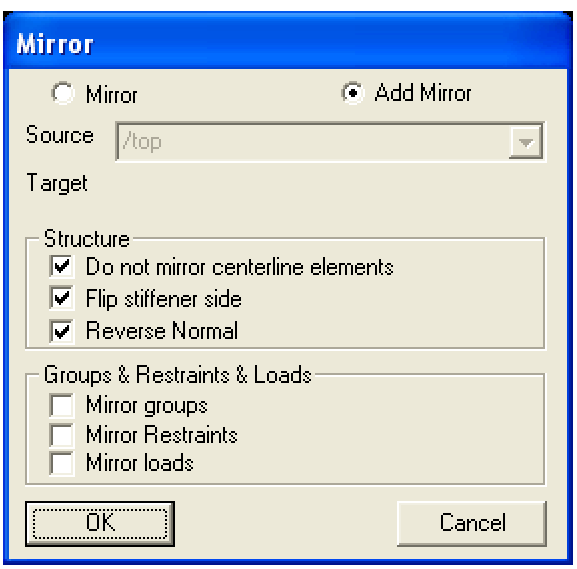


5. Choose the *ALPS/HULL* analysis type and click OK. This will automatically convert the existing strakes and stiffener layouts to the appropriate elements as described above and will generate the ALPS/HULL analysis model and place it under the *finemesh* object in the Parts Tree.

© 2014 Optimum Structural Design, Inc.



6. In the Parts Tree expand the *finemesh* object and right-click on the ALPS/HULL analysis model that was generated in the previous step. Choose **Add Mirror** from the menu, which will access the *Mirror* dialog.

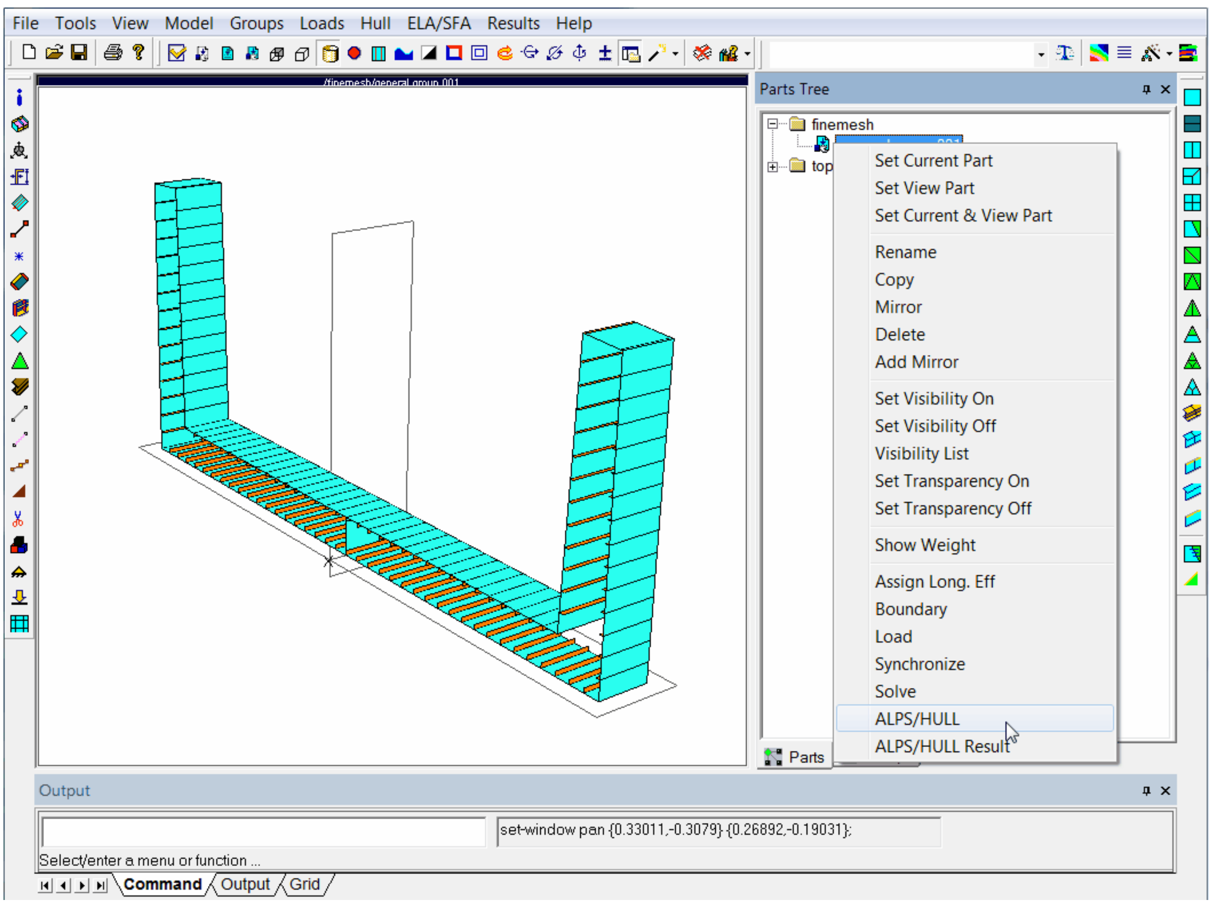


Be sure the correct boxes are checked, as shown above. Finally, click **OK**.

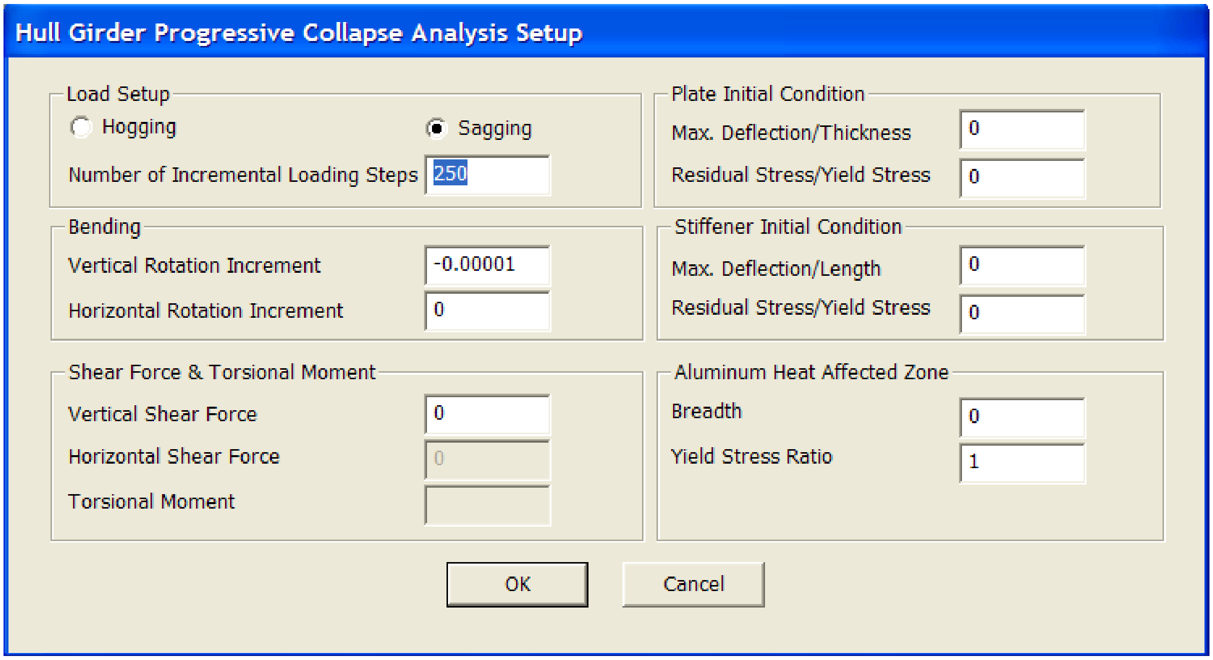
7. To view the full width model, expand the *finemesh* folder and right-click on the fine mesh model and "Set View Part".

Analysis Setup

Now that the full-model is successfully generated, the user can now proceed with the ALPS/Hull analysis. Note that the user can also add additional *localized* pressures. This is described the next section.



1. Expand the *finemesh* object again if necessary and right-click on the analysis model. Choose **ALPS/HULL** from the menu, which will launch the Analysis Setup dialog.



2. The Hull Girder Progressive Collapse Analysis Setup dialog provides several user input fields. For this dry-dock structure, select *Sagging*, enter *250* for the Number of Incremental Loading Steps, and enter *-0.0001* for the Vertical Rotational Increment.

A description of all the Hull Girder Progressive Collapse Analysis input parameters are provided below.

Load Setup

Select either a *Hogging* or *Sagging* scenario to be analyzed as well as define the number of incremental loading steps for the analysis. The user can increase the number of incremental steps if complete hull girder collapse was not obtained.

Bending

The Vertical and Horizontal Rotation Increment represents the value at which each load step will be executed, either in the Vertical Rotation or Horizontal Rotation plane.

Shear Force & Torsional Moment

The user can impose an initial Vertical Shear Force on the hull girder collapse analysis model.

Plate Initial Condition

© 2014 Optimum Structural Design, Inc.

The user can impose known initial out-of-plane deformations or residual stress.

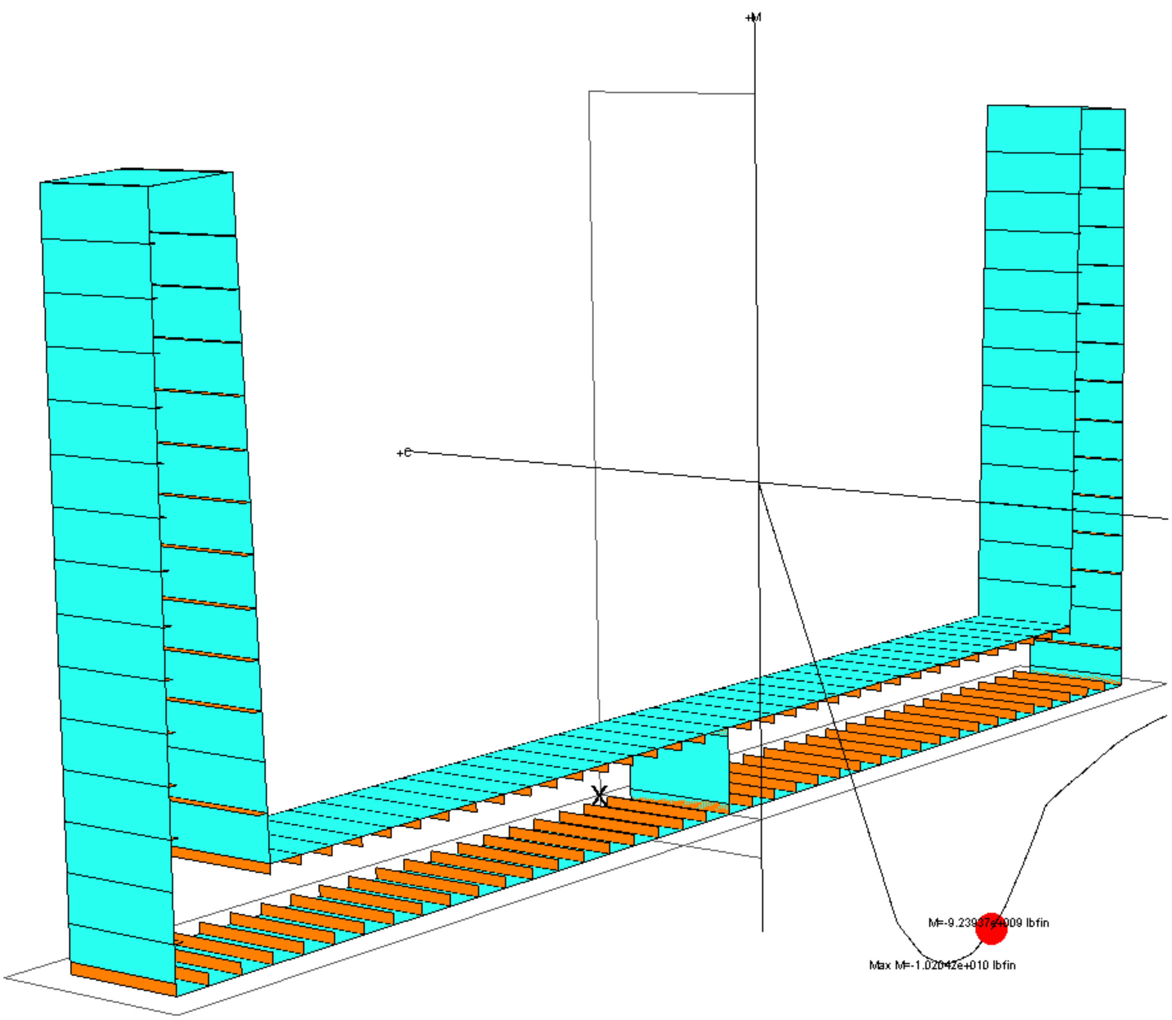
Stiffener Initial Condition

The user can impose known initial out-of-plane deformations or residual stress.

Aluminum Heat Affected Zone

The user can define the heat affected zone for aluminum structures.

3. When all the parameters have been defined, click **OK**. When the analysis is complete, i.e. all *Incremental Loading Steps* are done, the user can begin to post-process the results (as described in Post-Processing). At this point the user can proceed to Post-processing the current results or can go through the steps of the next section, which address Additional Pressure Loading.



Additional Pressure Loading

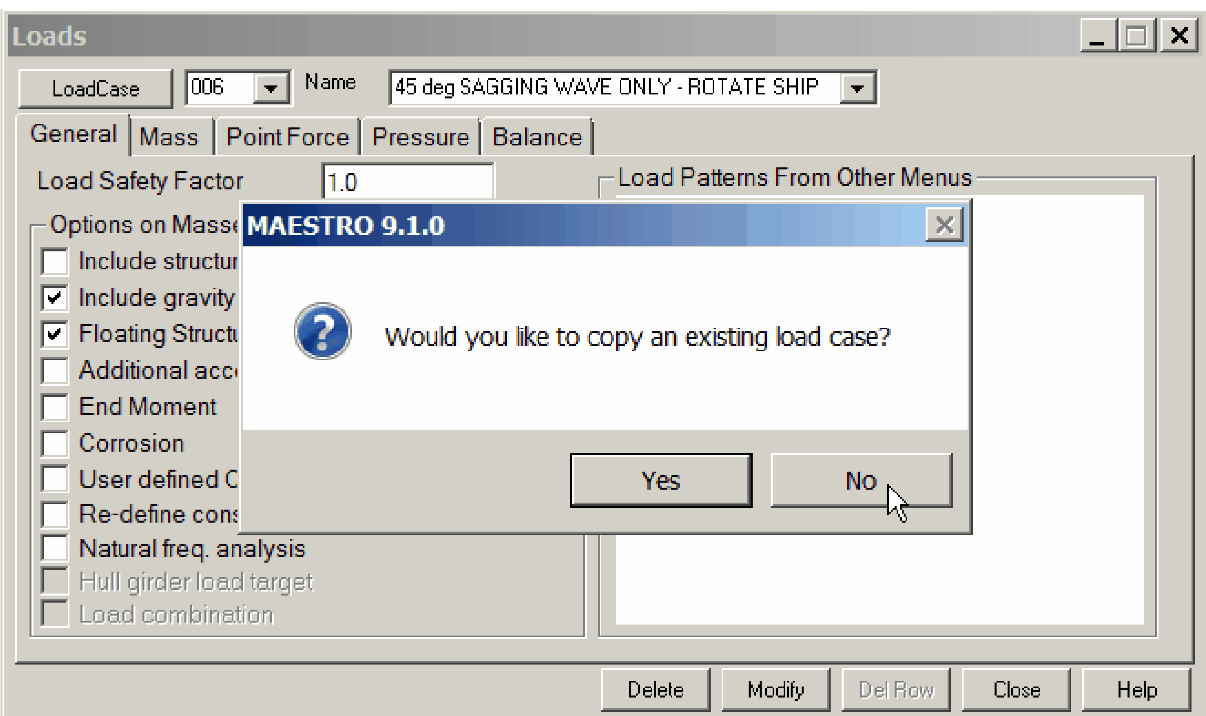
In addition to defining global ALP/HULL analysis parameters (see above), the user has the ability to impose *local* pressure loading. If the user elects to impose *local* pressure loads on the hull girder collapse model, these loads will be combined with the global analysis parameters. See Defining Pressure for complete details on creating and imposing pressure loads.

The following steps demonstrate how the user can add additional internal pressure and hydrostatic pressure to the ALPS/HULL model. To impose this additional pressure, the user will leverage the Surface Head method and the automatic Immersion pressure. The user also has the ability to leverage other local pressure loading methods such as Surface Zero or LinPress.

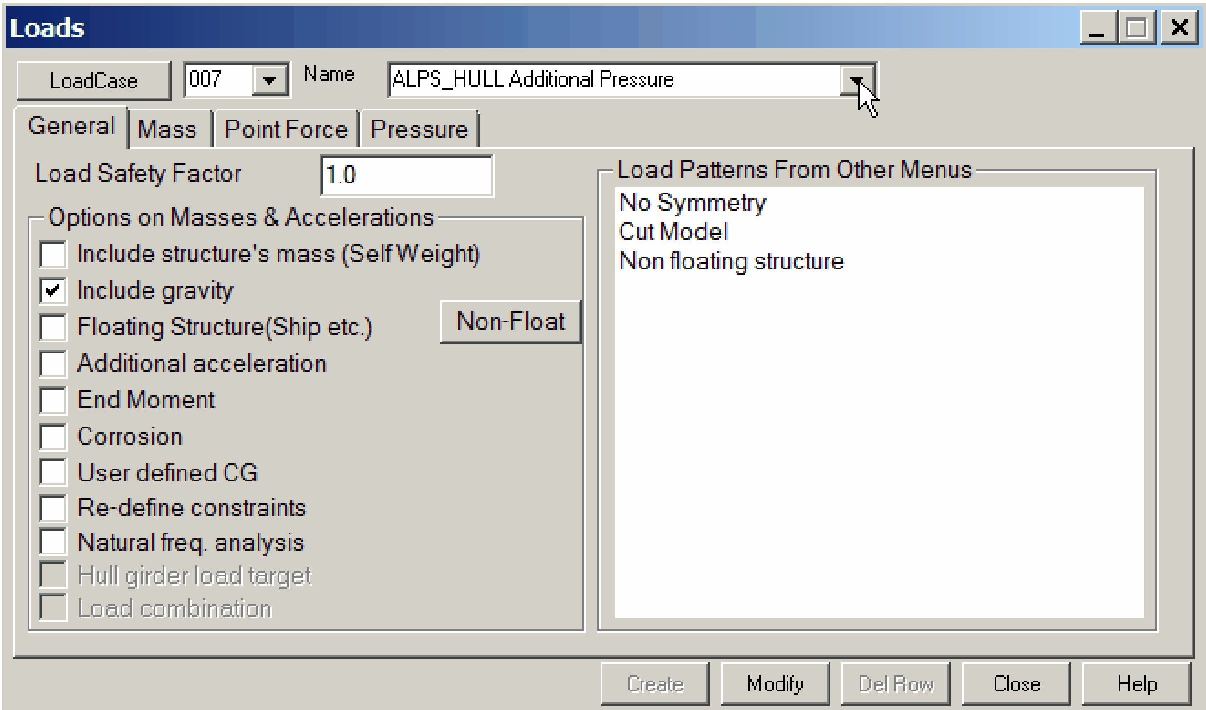
If you wish to do this part of the tutorial, when you perform Step 1, you will be asked wether you want to save the current model. Since this is a tutorial, you should click on *NO.*

1. Open the sample *DryDock_2.mdl,* which is located in the *...\MAESTRO\Models and Samples/ALPS/Hull* directory.

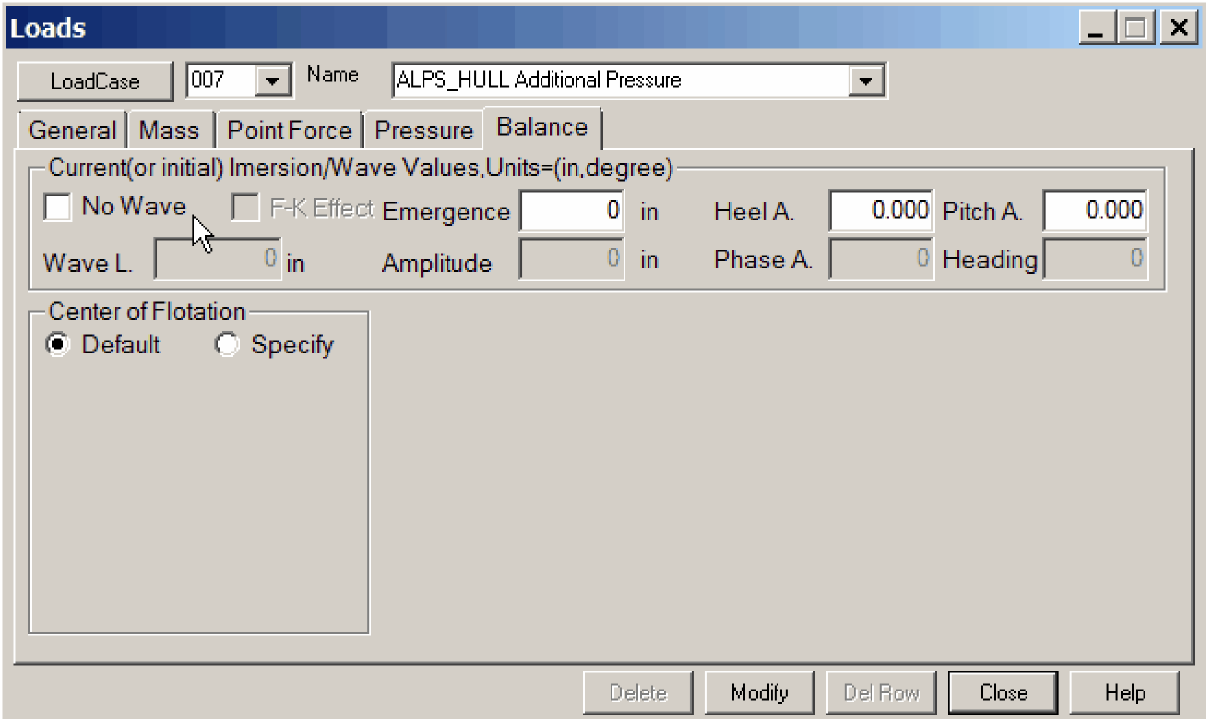
2. The existing model has a total of six load cases, but the user only has the ability to perform ALPS/HULL calculations on a single load case; therefore, the user must create a new load case. See *Creating A Load Case* for a complete description of creating and/or modifying load cases. Create a new load case and title it *ALPS_HULL Additional Pressure.*

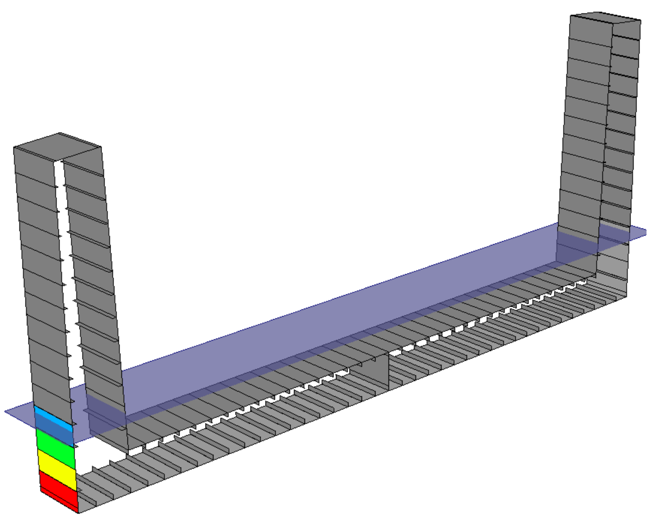


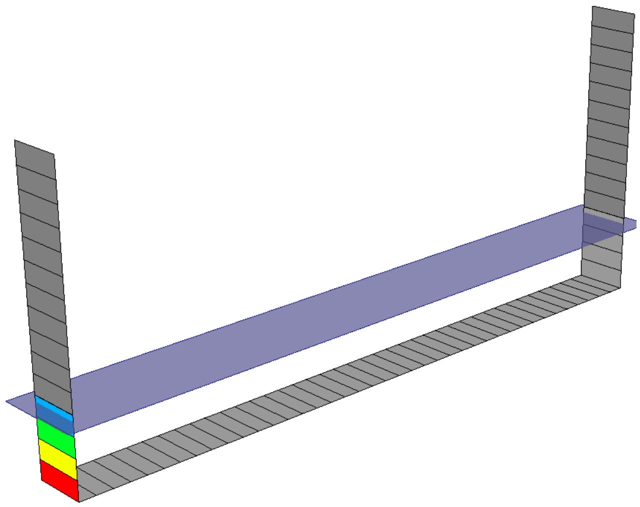
© 2014 Optimum Structural Design, Inc.



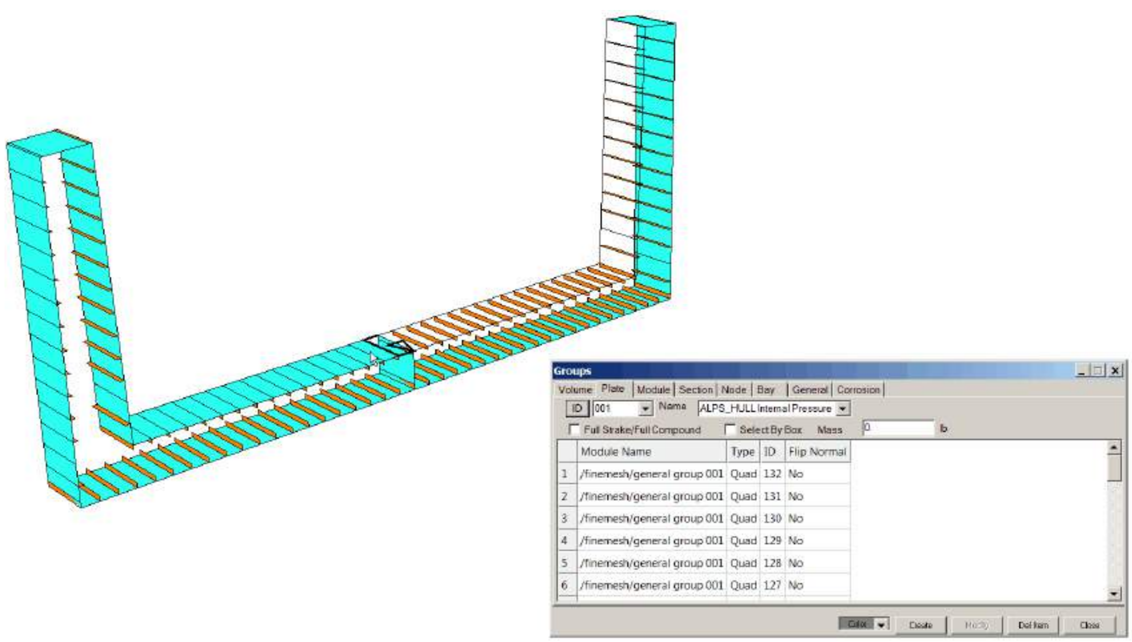
3. The next step will be to impose the hydrostatic pressure on the wetted elements associated with the ALPS/HULL analysis model. We do this by selecting the *Floating Structure* load option followed by clicking *Modify.* Upon clicking *Modify*, the loads dialog should expose the *Balance* tab. The Balance tab should have no wave (i.e. a stillwater condition); this is shown below. To view the hydrostatic pressure caused by the immersion, go to the main menu and click on *Load/View Pressure/Immersion* (shown in the next two figures). The second of these figures was obtained by viewing the Wettable Elements with the hydrostatic pressure. To view the Wettable Elements, go to the main menu and click *View/Wettable Elements*.

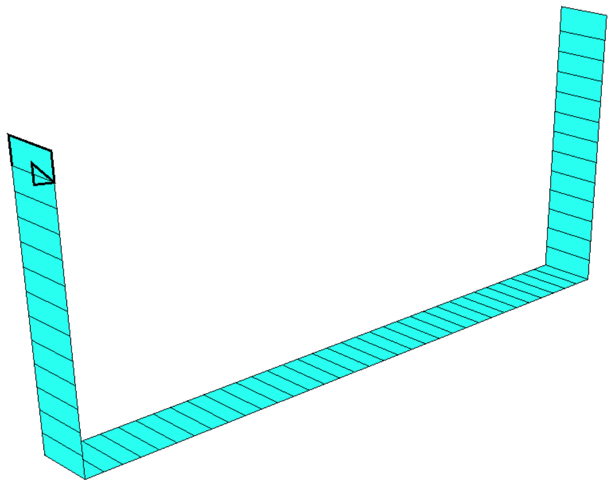

Loads
LoadCase
007
Name
ALPS_HULL Additional Pressure
General
Mass
Point Force
Pressure
Balance
Current(or initial) Imersion/Wave Values,Units=(in,degree)
No Wave
F-K Effect
Emergence
0
in
Heel A.
0.000
Pitch A.
0.000
Wave L.
0
in
Amplitude
0
in
Phase A.
0
Heading
0
Center of Flotation
Default
Specify
Delete
Modify
Del Row
Close
Help





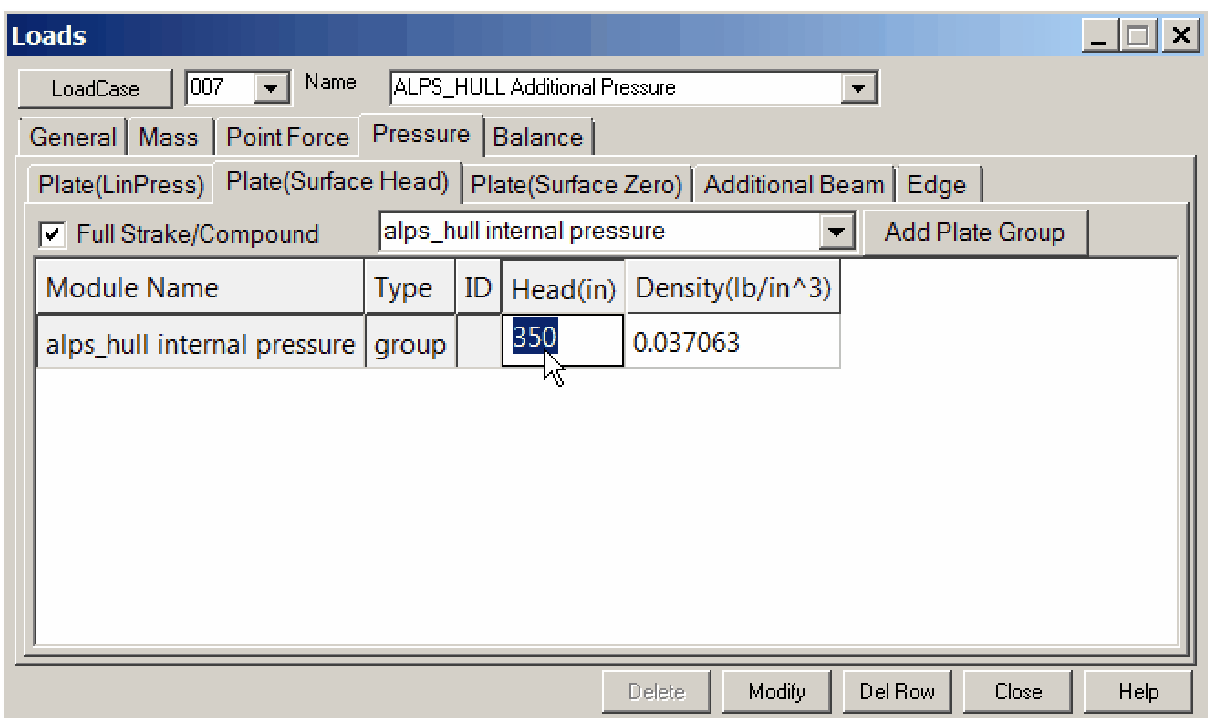
4. Now that the hydrostatic pressure is imposed, we focus on imposing the internal pressure. To do this we must first create a Plate group in the area of interest. For this tutorial it is assumed that this is a dry-dock and it is filled to a height of 350 inches. Therefore the plate group will consist of all the plate elements in the inner sides and the inner bottom of the dry-dock. For this tutorial, name the plate group *alps_hull internal pressure.* The next figure shows the process of creating the plate group with one side and half of the inner-bottom having been selected. The next figure shows the complete plate group.





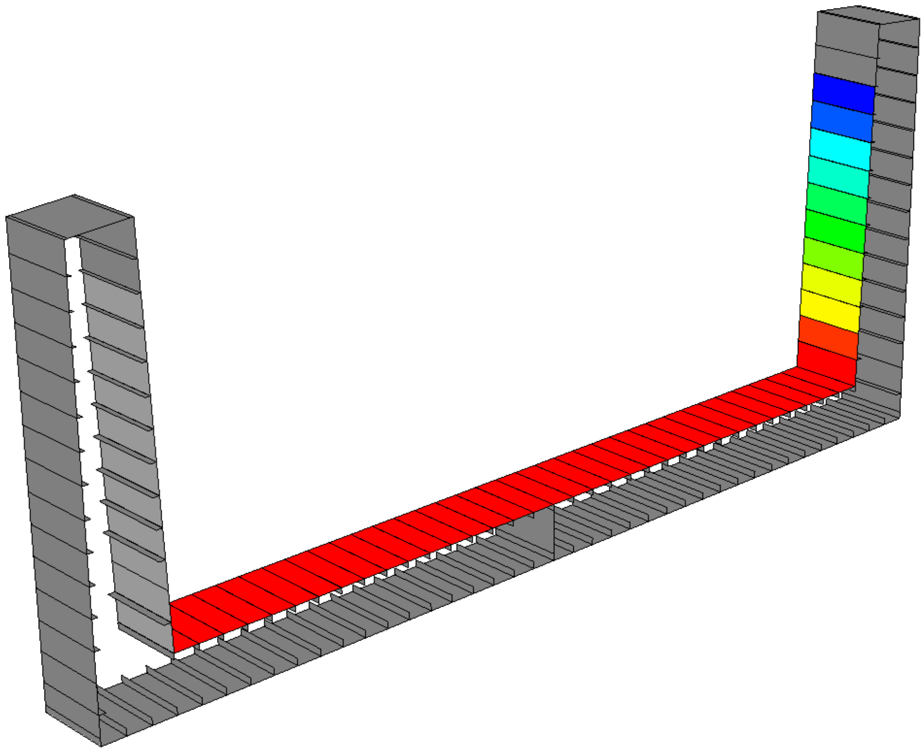
5. After creating the internal plate group, the user can include it in the current load case (i.e. *ALPS_HULL Additional Pressure).* See *Creating A Load Case* for a complete description of how to modify existing load cases. The intent is to add the additional internal pressure by using the *Plate(Surface Head)*loading pattern. Within the Loads dialog, open the *Plate(Surface Head)* tab under the *Pressure* tab.

6. Add the *alps_hull internal pressure* plate group and input *350* for the Head value (see below). Make certain to click *Modify* after inputting the Head value. We are now finished adding additional pressure.

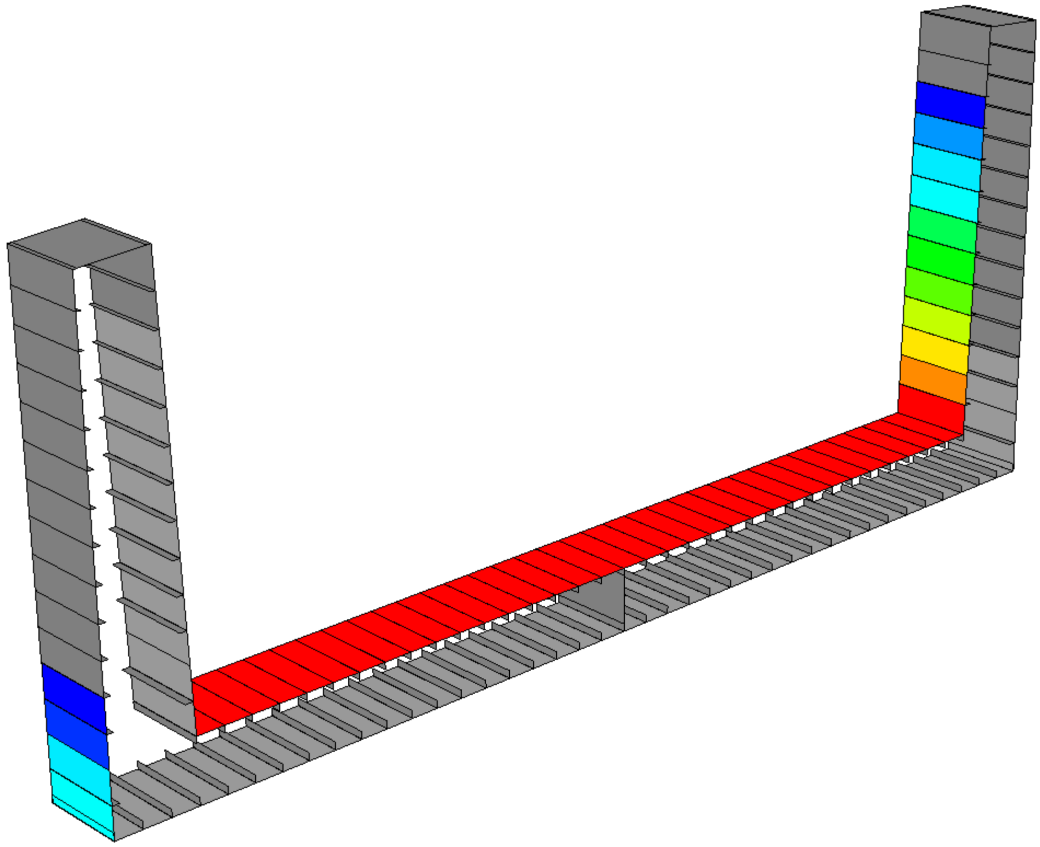


7. The next step is to verify the application of the Surface Head, which is done by selecting the *Loads/View Pressure/Surface Head* menu item. The model should look like the image

below.



8. The combined pressure load can be viewed by selecting the *Loads/View Pressure/Combined Pressure on Plate* menu item. The model should look like the image below. Dynamically spin the model around to view the pressure on the bottom shell.



9. The model is now ready to be solved and includes additional pressure. To solve the ALP/HULL model, go to the Parts tree, expand the *finemesh* object, right-click on the *general group 001 part,* and then select **ALPS/HULL** from the menu. Now, using the steps described in Analysis Setup, the user can solve the ALPS/HULL model.

Post-processing

The user can review the following results from the progressive collapse analysis at a given *Step*:

- SigXX Stress to Yield Stress Ratio
- VonMises Stress to Yield Stress Ratio
- The particular Failure Mode, which are summarized in the table below:

Failure Mode Acronym	*Failure Mode Description*
OC	Overall Collapse
PB	Collapse of plating between support members (stiffeners)

© 2014 Optimum Structural Design, Inc.

BCC	Beam-column Type Collapse
SWB	Local Buckling of Stiffener Web
TR	Flexural-torsional Buckling of Stiffener
GY	Gross Yielding
RT	Rupture due to Tension
CC	Crushing due to Compression

- SigXX, SigYY, SigXY, SigVM

The user can graphically plot these various values by right-clicking in the modeling window, which will launch the menu shown below. Once the menu appears the user first chooses the result parameter of interest followed by the Step of interest. The contour plot will reflect the current selection.

The **Define Range From...** provides a mechanism to choose what Step first appears in the menu post-processing menu. The user can also echo the complete Curvature versus Moment results to the Output tab, which can then be plotted using 3rd party applications.

Finally, the user can animate the progressive collapse analysis, i.e. step through each increment, by clicking the  icon.

The ALPS/HULL results can be turned on and off by checking or unchecking the "ALPS/HULL" box in the View Options dialog.

ALPS/HULL Report

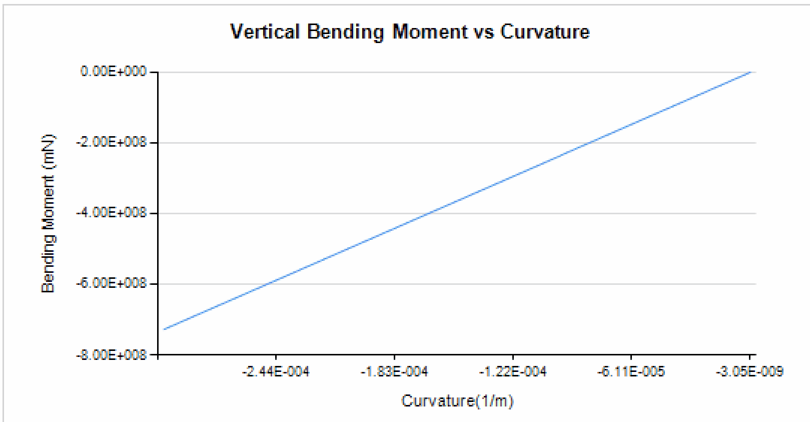
The results of an ALPS/HULL analysis can also be viewed in report form by right-clicking during the ALPS/HULL results view and selecting *Create Report...*

This will launch a new window with a report of the results:

© 2014 Optimum Structural Design, Inc.

FormALPSHULL

ALPS/HULL Report



Step	Curvature(1/m)	Bending Moment (mN)
1	-3.052503E-09	-7298.541
2	-3.055556E-06	-7306086
3	-6.108059E-06	-14681380
4	-9.160562E-06	-22018420
5	-1.221306E-05	-29355470
6	-1.526557E-05	-36692520
7	-1.831807E-05	-44029590
8	-2.137057E-05	-51366660
9	-2.442308E-05	-58703740
10	-2.747558E-05	-66040830
11	-3.052808E-05	-73377930
12	-3.358059E-05	-80715030
13	-3.663309E-05	-88052150
14	-3.968559E-05	-95389280
15	-4.27381E-05	-102726400
16	-4.57906E-05	-110063600
17	-4.88431E-05	-117400700
18	-5.18956E-05	-124737900
19	-5.494811E-05	-132075000
20	-5.800061E-05	-139412200
21	-6.105311E-05	-146749400
22	-6.410562E-05	-154086600

This report can be exported to PDF, Microsoft Excel, or Microsoft Word.

© 2014 Optimum Structural Design, Inc.

16.6 Optimization

The following tutorial will walk through an example process of optimizing a box girder structure.

16.6.1 Defining the Model

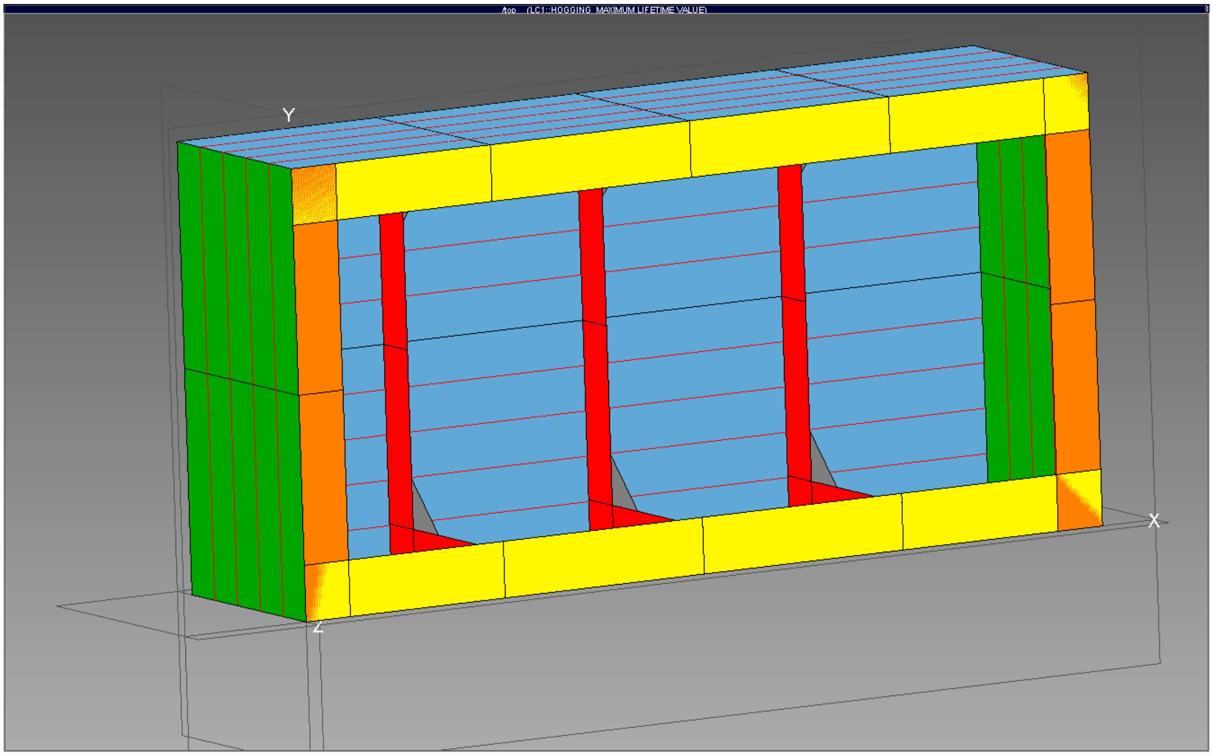
DEFINITION OF MODEL, BOUNDARY CONDITIONS AND LOADS

1.1 Making a Backup Copy of the Modeler File

This document shows how to prepare the input data and interpret the output for optimization jobs. This document assumes that you have already gone through the Tutorial that is available through the Help button. In effect it is a second tutorial and you are invited to open the relevant Modeler file – ex2.mdl – and actually fill in the Dialog Boxes as shown herein. To do this will require deleting the existing entries in the Dialog Boxes and then re-entering the data. To safeguard the original file you should first make a backup copy, say ex2backup.mdl. Later in the tutorial you will also want to display the output file – ex2.out – and that is achieved from within the Modeler by going to the Main Menu and clicking **File > Analysis / Evaluation > Legacy Version of MAESTRO > View Output File**. If you do not have this file or if it becomes corrupted you can generate a new copy by running the ex2 job, simply by clicking on the execute button at the right end of the upper toolbar.

1.2 Description Of The Structural Model

This example demonstrates optimization of a simple but realistic module- a rectangular box girder with vertical and transverse symmetry, both of structure and of loading. The structure might, for example, be a segment of a pontoon in a semi-submersible. The one-module model of the structure is shown in the following figure. If you open the Modeler file ex2.mdl you will have a color picture of the structure. You may wish to shrink and move the Modeler display so that you can see it side by side with this document. As in the earlier tutorial, you can gain more display space by closing the Parts Tree window and hiding the Refine toolbar (under **Tools > Toolbars** in the Main Menu).



The model consists of four strakes. To simplify the example, the maximum design value of transverse pressure is the same for all strakes. This means that the module has vertical symmetry (about the half height) of loading as well as of geometry. Because of this, the results for strake 3 should be the same as for strake 2, and the results for strake 4 should be the same as for strake 1.

In this module, there is a transverse bulkhead at each end, modeled with two “additional” panels and stiffened by two "additional" beams (as always, the word "additional" means "not strake-related").

1.3 Boundary Conditions For Modules

In this example, the boundary forces are the bending moment and shear force that act at the end of the module. In ships and in box girder bridges this information can be obtained from a prior and relatively simple analysis of the overall structure, because it is essentially a beam. This is not possible with a semi-submersible because of the interconnection between pontoons, columns and deck structure. For such a structure the MAESTRO model would, initially at least, include however much of the structure is necessary in order to have known boundary conditions. This would be either the entire structure or, depending on the degree of symmetry, one half or one quarter of it.

In most applications of MAESTRO this overall model is the only model that is required. Since all of the modules are in the model they interact fully, just as in the real structure.

Hence each module always has the correct boundary conditions.

1.4 Boundary Forces For This Example

The above explanation is given because in this example, for the sake of simplicity, the boundary conditions for the relevant portion of the structure are assumed to be already known, which is not the usual situation. Normally the analysis portion of the design process would deal with all of the modules simultaneously, such that in the analysis, and in the subsequent module-by-module evaluation and optimization, all of the modules have the correct boundary conditions, automatically and at all times. But for this simple introductory example we restrict ourselves to just one module.

The boundary loads and external pressures are:

(1) a lateral pressure on each strake of 12 psi;

(2) an alternating (hog, sag) vertical bending moment, for which the characteristic value (expected maximum lifetime value, with an acceptably small probability of exceedance) is 1.95×10^8 lb-in.

To be consistent with these loads, and to preserve their symmetry, the shear force at the ends is taken as zero. The program's echo print of the loads is given on pages 8-13 of the output, which you obtain by going to the Main Menu and clicking **File > Analysis / Evaluation > Legacy Version of MAESTRO > View Output File.**

1.5 Partial Safety Factors

The partial safety factors for this example are obtained from the American Petroleum Institute Bulletin 2V, "Design of Plane Structures". The loads described above correspond to the "extreme service condition", for which the factors are 1.25 for serviceability limit states and 1.50 for ultimate (or collapse) limit states. These values are specified in the criteria (page 2 of the output).

16.6.2 Optimization Data

OPTIMIZATION DATA

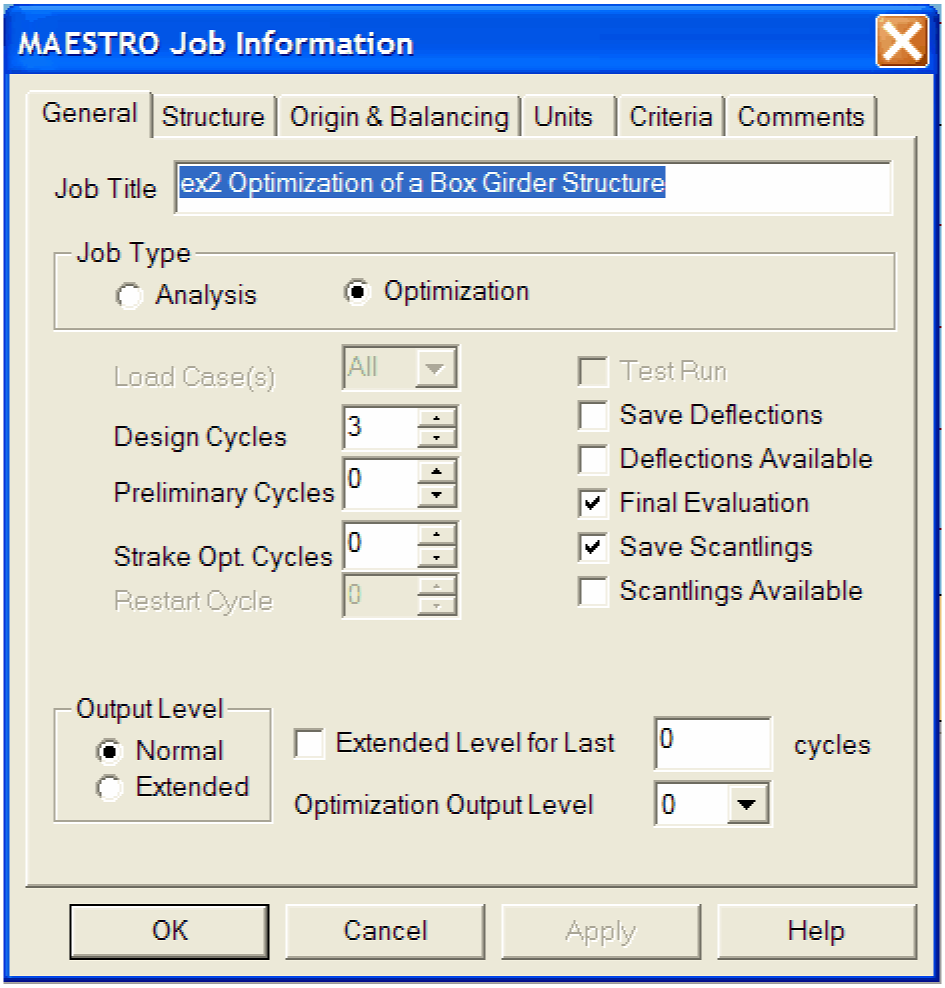
This chapter shows how to use the Groups Dialog Box and the Optimization Dialog Box to define the data needed for optimization.

2.1 Optimization Initial Settings – Job Information Dialog Box

Before beginning any optimization task, you should always perform a MAESTRO structural analysis on the entire model and be sure to correct all errors and serious inadequacies. After an analysis job in which there are no serious inadequacies, you should open the Job Information Dialog Box from **File > Analysis / Evaluation > Legacy Version of**

© 2014 Optimum Structural Design, Inc.

MAESTRO > Job Information.



The General Page in the Job Info Dialog Box is the only page that needs to be altered to initiate the optimizer. The Optimization radio button in the Job Type section must be selected, which unlocks options in the lower half of the Dialog Box.

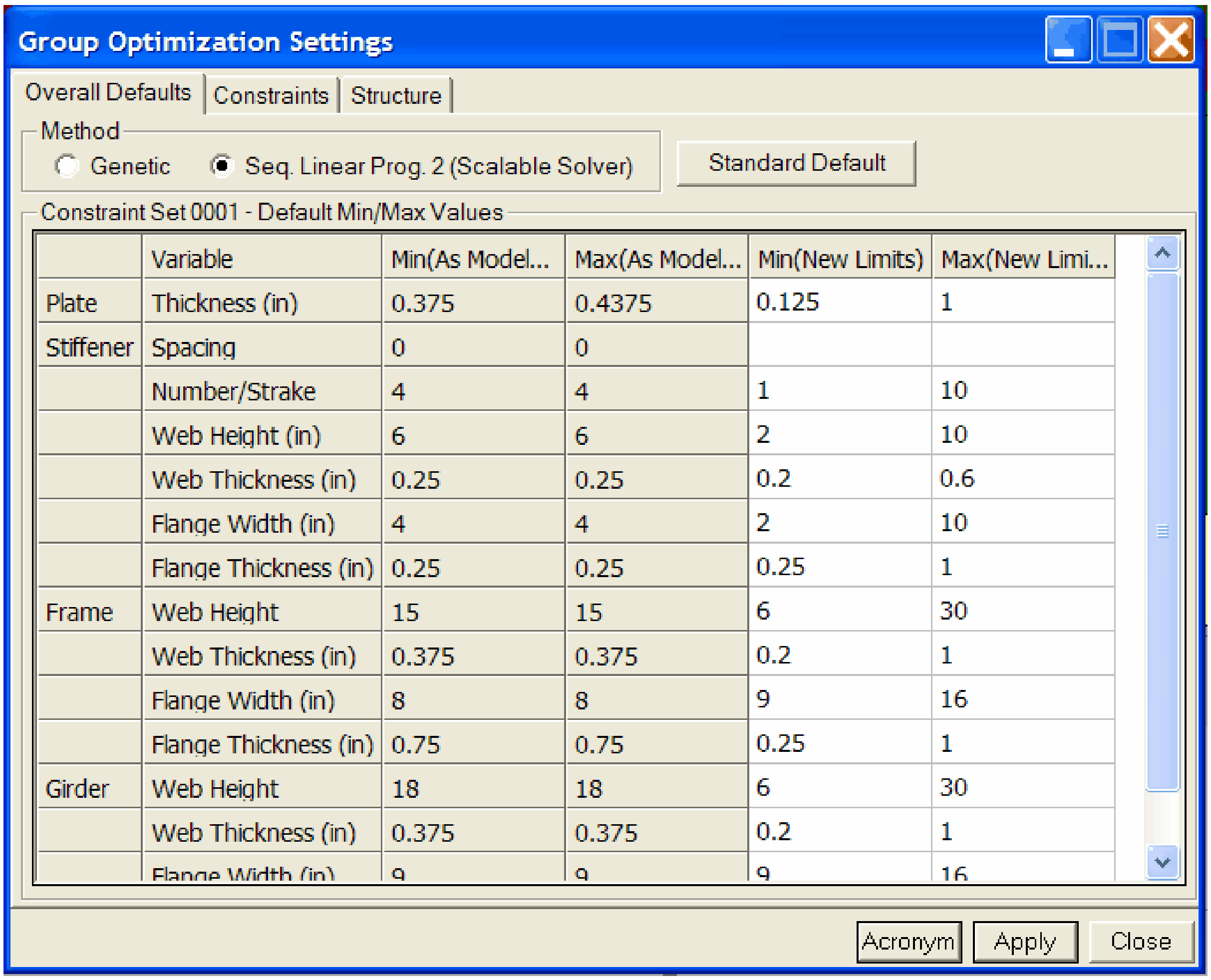
The above figure shows that the user has told MAESTRO go through 3 design cycles. The checked “Final Evaluation” box tells it to perform an analysis of the final design cycle scantlings and to put the results into the ex2.OUT file. The checked “Save Scantlings” box instructs MAESTRO to create a “restart” file that records the scantling values at each design cycle for each module that is being optimized. Restart files are explained in a later section.

2.2 Optimization Dialog Box - Page 1, Optimization Data And Overall Defaults

In MAESTRO the optimization is performed separately for each module. The optimization data is specified in two Groups:

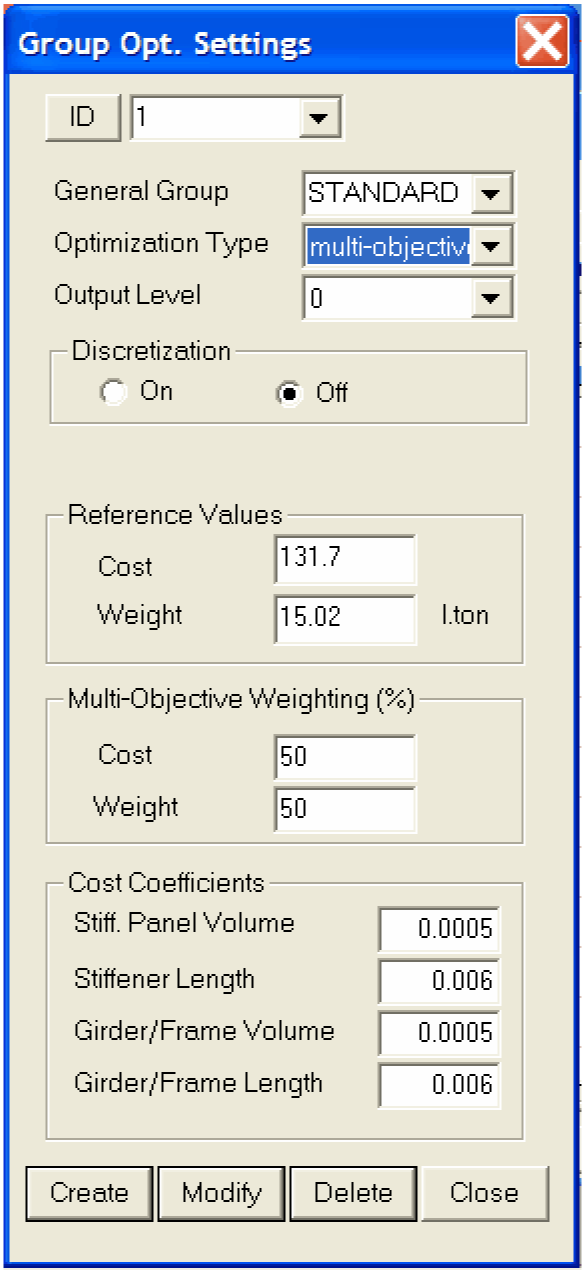
1. one Standard Group - a module-independent group of Min/Max constraints and other Constraint Sets which serve as default Constraint Sets for all modules,
2. any number of module-specific General Groups, containing Constraint Sets which can selectively override the default Constraint Sets, and which can also include other types of Constraint Sets that are not in the Standard Group.

These two Groups and their component Constraint Sets are defined in the Optimization Dialog Box pictured below. There is no toolbar icon for this Dialog; you obtain it from the Main Menu by clicking on **File > Analysis / Evaluation > Legacy Version of MAESTRO > Optimization Settings**.



This picture shows the Overall Defaults page, which defines the overall information and Constraint Set 1, which specifies the default Min/Max constraints.

2.2.1 Type of Optimization



The Standard Default button on the Overall Defaults page of the Optimization Dialog Box brings up the Group Opt. Settings dialog box shown to the left. This dialog allows the user to define different Optimization Types for each of the module-specific General Groups. MAESTRO can optimize based on Weight only, Cost only or Multi-objective: a non-dimensional weighted sum of weight and cost. The latter is nearly always the true objective in structural design. Least weight by itself (i.e. at any cost) is appropriate only for aircraft and aerospace vehicles. With the multi-objective approach the cost implications of weight can be explicitly included; for example, a weight reduction may mean more cargo capacity and hence more revenue.

The non-dimensional weighted sum is defined as

$$U = P_1 \frac{C}{C_0} + P_2 \frac{W}{W_0}$$

where C_0 and W_0 are "reference" values. These are simply typical values for the structure. In this example they are C_0 = 131.7 K-DOLLARS and W_0 = 15.02 long tons, which are nothing more than the starting values of cost and weight, obtained by making an Analysis run before starting the optimization. The units were specified in the Units page of the Job Information Dialog Box.

The resulting non-dimensional values are combined by means of weighting parameters, P_1 and P_2, which are simply fractions that add to 1.0. In this example, for simplicity, the values are 0.5 and 0.5; i.e. cost reduction and weight reduction are given equal importance. In the Dialog Box they are specified as percentages.

2.2.2 Discretization

MAESTRO provides a feature whereby the designer can specify a list of standard (or available) thicknesses, the criteria as to how round off is to be done for the various member types, and the desired (or permissible) increments in web depths and flange breadths of the various member types.

During the optimization process the design variables are treated as continuous variables, in order to avoid the high computational cost of discrete variable optimization. But when the design is seen to be converging the designer can specify that the next run should include discretization of the design variables. In that run, the discretization will be done during the last of the design cycles requested for that run. The discretization is not simply a rounding up all of the design variables to the next standard size.

2.2.3 Cost Function

For structural optimization the only costs that are relevant are those that are influenced by scantlings. The two types of cost that are most directly related to scantlings are the cost of materials and the cost of fabrication, and for these MAESTRO provides the following "general purpose" cost function

$$Cost = \sum^{NS} (C_{sp} + C_g + C_f)$$

where NS = number of strakes, and all of the four cost terms are on a per-strake basis.

© 2014 Optimum Structural Design, Inc.

Stiffened Panel Cost

C_{sp} = cost of the stiffened panel for each strake. In the general-purpose cost function this is defined as

$$C_{sp} = \rho_1 \left[ABt_p + \frac{A}{d} ls \left(h_{sw} t_{sw} + b_{sf} \; t_{sf} \right) + \rho_2 \frac{A}{d} ls \right.$$

in which

s is the number of stiffeners:

$$s = \frac{B}{b} - 1$$

t_p, h_{sw}, t_{sw}, b_{sf} and t_{sf} are the panel scantlings

A is the length of the module

d is the section spacing

B is the width of the strake (mean value if tapered)

l is the stiffener length in each panel (l = B if transversely stiffened)

ρ_1 is the volumetric cost coefficient (cost per unit volume) for a stiffened panel

ρ_2 is the lineal cost coefficient (cost per unit length of stiffener) for a stiffened panel

Girder Cost

C_g = cost of girder for each strake, given by

$$C_g = dN_g \left[\rho_3 \left(h_{gw} t_{gw} + b_{gf} \; t_{gf} \right) + \rho_4 \right]$$

where

ρ_3 is the volumetric cost coefficient for fabricated beams

ρ_4 is the lineal cost coefficient for fabricated beams

$$N_g = \frac{A}{d}$$

© 2014 Optimum Structural Design, Inc.

N_g is the number of girder segments (elements) in the current strake; if no elements are deleted this is

Frame Cost

C_f = cost of the frames for each strake, given by

$$C_f = N_f \; B\,[\rho_3 \,(h_{fw} t_{fw} + b_{ff} \; t_{ff}) + \rho_4]$$

where N_f is the number of frames per strake.

2.2.4 Cost Coefficients

The above function contains four coefficients, ρ_1, ρ_2, ρ_3 and ρ_4. Their default values are defined in the Module Opt. Settings Dialog Box, and alternative values can be specified for individual strakes. The lineal (cost per unit length) coefficients would reflect such items as welding costs and would influence the optimum number of stiffeners in each strake of plating. These coefficients might vary according to which shipyard is building the ship. For example, one shipyard might have more automatic welding machines so that its cost per unit length of stiffener weld might be cheaper than at another shipyard. In this case, the optimum design would probably have more stiffeners and less steel than at another shipyard with higher welding costs.

Note that for optimization there is no need for absolute values for the coefficients; they need only have the correct relative magnitudes.

In this example the cost unit is K-DOLLARS ($1000) but this is arbitrary. The volumetric cost coefficient will therefore be in units of K-DOLLARS per in^3. Let us say that a rough "order of magnitude" figure for steel cost (including handling and other costs that are related to the amount of steel) is $1.75 per pound. Translating to the example units gives $0.50 per in^3 or 0.0005 K-DOLLARS per in^3. Similarly let us say that a rough "order of magnitude" figure for welding costs is $6.00 per inch of weld, which is 0.006 K-DOLLARS per inch. These values are inserted as the volumetric and lineal cost coefficients in the first page of the Dialog Box. The cost function allows for different values to be used for rolled "panel line" members (plating and stiffeners) and fabricated members (frames and girders) but for simplicity we here use the same values for both.

2.3 Constraints Imposed Directly On Design Variables

In addition to the inbuilt constraints relating to structural failure and other limit states, MAESTRO allows the designer to specify any number of other constraints directly on the design variables. These constraints arise from many factors such as avoidance of local failures, production and fabrication considerations, and operational requirements.

There are three types of such constraints:

© 2014 Optimum Structural Design, Inc.

A. those involving only one design variable and which specify a minimum, and/or a maximum value for it, or a fixed value for it;

B. those involving a pair of design variables and which impose a linear relationship (equality or inequality) between them.

C. those which "freeze" any number of design variables to the values that have already been assigned to them in the finite element data.

NOTE: It is mandatory that all design variables either be assigned a fixed value, be "frozen", or have minimum and maximum values specified for them.

The default minimum/maximum constraints are always specified in the first page (Overall Defaults) of the Dialog Box and they constitute Constraint Set 1 of the optimization data. The proportionality constraints are defined in the second page of the Dialog Box. In this example they are all defined in Constraint Set 2.

The “acronym” button at the lower right of the Dialog Box provides a pop-up list that explains the various acronyms.

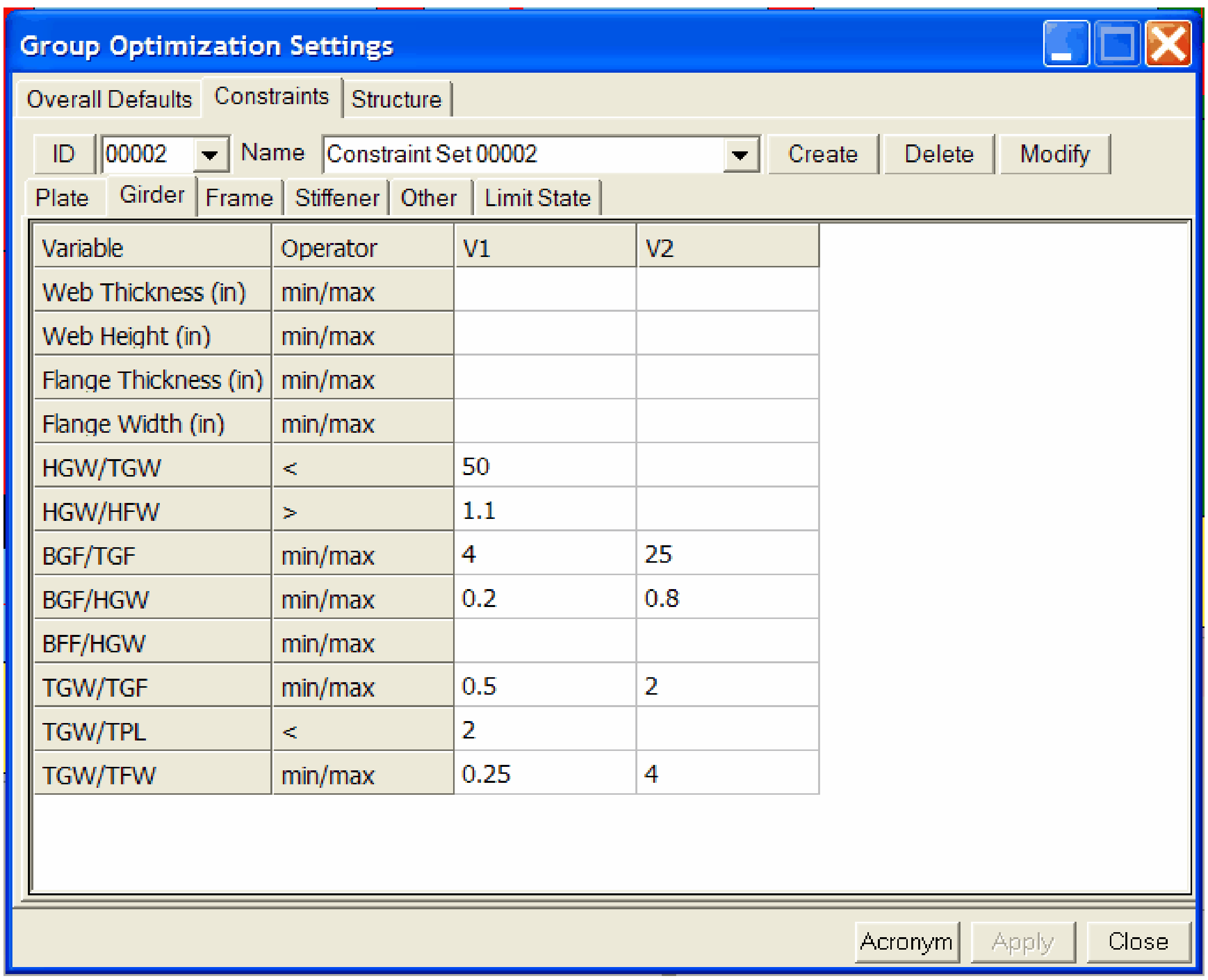
TPL...Plate Thickness
STF...Number of Stiffeners
BBS...Breadth Between Stiffeners
HSW...Stiffener Web Height
TSW...Stiffener Web Thickness
BSF...Stiffener Flange Breadth
TSF...Stiffener Flange Thickness
HGW...Girder Web Height
TGW...Girder Web Thickness
BGF...Girder Flange Breadth
TGF...Girder Flange Thickness
HFW...Frame Web Height
TFW...Frame Web Thickness
BFF...Frame Flange Breadth
TFF...Frame Flange Thickness
ZMOD...Section Modulus of Module
IMOD...Moment of Inertia of Module

In this example the minimum values of flange width (8 inches for the frames and 9 inches for the girders) are chosen not because of fabrication but rather to prevent flexural-torsional (or lateral-torsional) buckling for these members. This is more than just a local type of failure, and in the future it will be added to those limit states which are examined explicitly by MAESTRO. The stipulation of a minimum value (which might depend on the member length and other factors) is more typical of code based design, and this example shows that, if desired, the requirements of such codes can be incorporated into the constraint set.

© 2014 Optimum Structural Design, Inc.

2.4 Optimization Dialog Box – Page 2, Other Constraint Sets

The first page – Overall Defaults – defines only the default min/max constraints and these always constitute Constraint Set 1. The rest of the constraints are defined in the second page, labeled Constraints. The following figure shows the second page, with Constraint Set 2 selected (in the ID box) and after clicking on the Girder tab. Each new Constraint Set is created by clicking the ID button and then clicking on Create. Then you go to the subpages (Plate, Girder, Frame, Other) and define the constraints that make up that Constraint Set. As you define them you should click on Modify. As a minimum you must click on Modify before leaving a subpage or your definitions will be lost.



The constraints of Constraint Set 2 all impose limits on the proportions of individual members and on the relative sizes and thicknesses between members, in order to ensure a balanced and buildable structure. Most of the constraints are direct proportions, with the left hand side being a simple ratio and the right hand side being the limit value (V1).

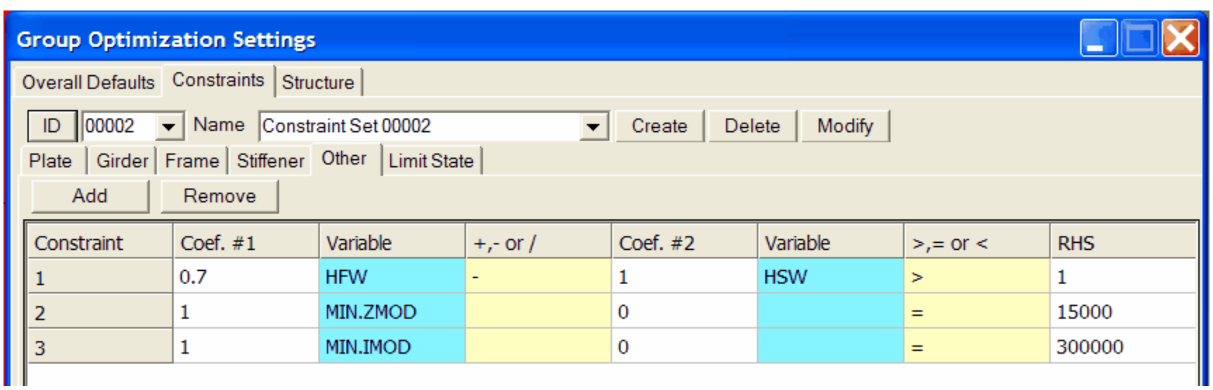
Double-click in the Operator cell to toggle between min/max, >, = and <. Some sample constraints are given in the Girder subpage shown above. The first constraint prevents the height of the girder web from being more than 50 times the web thickness, to prevent web buckling. The second constraint requires that the girder web height must be at least 10% larger than the frame web height, in order that the flanges do not intersect, thereby avoiding a complicated welded joint. The third, fourth, fifth and last constraints involve both a lower and an upper limit on a ratio (V1 and V2). The second to last constraint prevents the stiffener web thickness from being more than twice the thickness of the plating, because a large difference in thickness causes an unbalanced heat input during welding.

If a fixed value is to be imposed this may done by means of an equals sign. For example, if it was desired to fix the girder web thickness at 0.25 inches, this could be done by adding the following line

TGW = 0.25

An alternative is to “freeze” any design variable to its current value. For constraints in which the left hand side is a design variable, “freeze” is an additional operator, as shown in the first line of the above figure.

The Frame and Stiffener subpages are similar to the Girder subpage, whereas the Other subpage allows the definition of more complicated constraints on the design variables, and some special constraints relating to the entire module. Examples are given in the following figure.



Constraint	Coef. #1	Variable	+,- or /	Coef. #2	Variable	>,= or <	RHS
1	0.7	HFW	-	1	HSW	>	1
2	1	MIN.ZMOD		0		=	15000
3	1	MIN.IMOD		0		=	300000

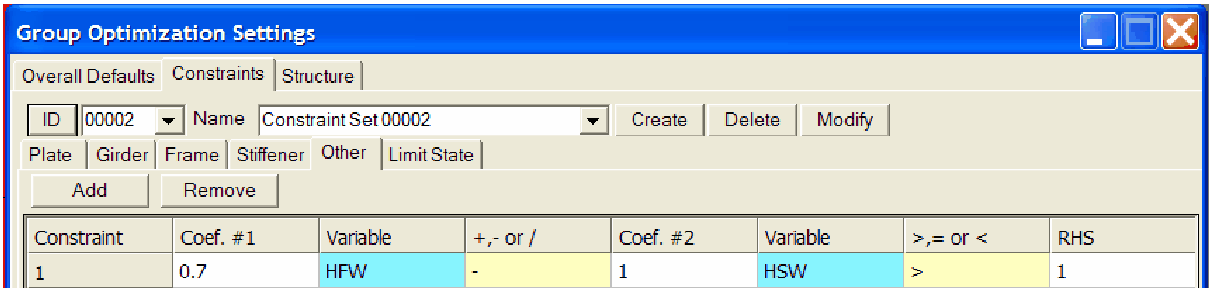
The first constraint illustrates the more general form that is available. This constraint requires that the total height of the cutout in the frame web for the stiffener penetration must not exceed 70% of the frame web height. The height of the cutout is taken as the stiffener web height plus a semicircle 1 inch in radius. The constraint is then

$$h_{sw} + 1.0 < 0.7\ h_{fw}$$

In the data input the two design variables must be on the left hand side and the constant term on the right hand side. Hence the MAESTRO form of the constraint is

$$0.7\ \mathrm{HFW} - 1.0\ \mathrm{HSW} > 1.0$$

© 2014 Optimum Structural Design, Inc.



The above figure shows the Other subpage after the frame web cutout constraint has been generated by the following 9 steps. If you actually perform these steps you will be generating a new fourth line of data, and the 10^{th} step will Remove it.

1. Click on Add. A new blank line appears in the List Box.
2. Click in the Coef. #1 cell and type in 0.7
3. Double click in the first Variable cell to get a drop-down list of variables. Scroll down and select HFW.
4. Define the first operator. You can toggle through the three operators by double clicking. Double click on the + symbol to change it to -.
5. Double click in the second Variable cell to get a drop-down list of variables. Scroll down and select HSW.
6. Click in the Coef. #2 cell and type in 1.
7. Go to the second operator cell, toggle (double click) until you get the > symbol.
8. Click in the RHS cell and type in 1.
9. Click on Modify.
10. Since we don't want this line, click on Remove.

2.5 Data Concerning Module-Related Load Effects

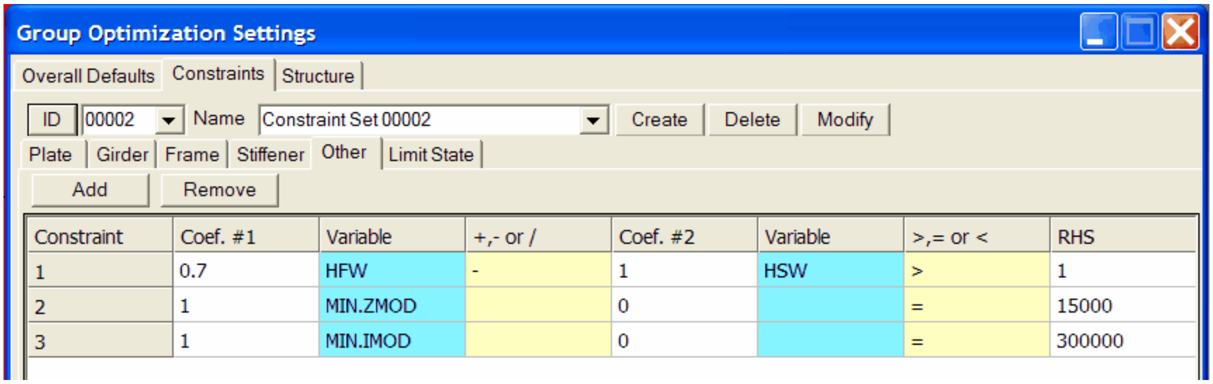
2.5.1 Module Level Design Limits (Constraints)

As shown in the next figure, this example contains two constraints that pertain to the entire module rather than to an individual strake.

(1) Minimum required value of moment of inertia of the module (MIN.IMOD)
Here the limit or constraint is some maximum permissible value of module deflection due to vertical bending. Such a limit is unlikely to be relevant for a pontoon, but naval vessels can have such limits arising from radardirected guns, and in aircraft structures such limits are universal and are dominant in the design. Therefore, in order to demonstrate how MAESTRO can accommodate such a limit, the pontoon will be required to have a moment of inertia of at least $3.0x10^5$ in^4. This is specified in the "MIN.IMOD" line of the Other subpage.

(2) Minimum required value of section modulus of the module (MIN.ZMOD)
This constraint limits the maximum stress due to vertical bending of the module. For example, a ship's hull is subjected to long term cyclic bending due to waves, and it may be desired to limit the cyclic stress to avoid fatigue. In this example the module is required to

have a minimum section modulus of 15000 in^3. This value is specified in the "MIN.ZMOD" line of the Other subpage.



Constraint	Coef. #1	Variable	+,- or /	Coef. #2	Variable	>,= or <	RHS
1	0.7	HFW	-	1	HSW	>	1
2	1	MIN.ZMOD		0		=	15000
3	1	MIN.IMOD		0		=	300000

The above figure shows the Other subpage after the MIN.IMOD and the MIN.ZMOD constraints have been defined. The first of these was generated by the following six steps. Again, if you perform these steps you will be generating a new line, and the 7th step will delete it.

1. Click on Add. A new blank line appears in the List Box.
2. Click on Coef. #1 and type in 1
3. Double click in the first Variable cell to get a drop-down list. Scroll down and select MIN.IMOD.
4. Double click on the > symbol to change it to =.
5. Click in the RHS cell and type in 300000.
6. Click on Modify.
7. Since we don’t want this line, click on Remove.

2.5.2 Dual Level Optimization

Some loads and load effects involve the entire structure - for example, the hull girder bending, shear and torsion of a ship hull. These load effects cannot be controlled adequately by a local resizing of a few structural members, but rather by the coordinated resizing of all of the members that make up the cross section of the ship. In MAESTRO terminology, it requires a coordinated resizing of all of the strakes that make up the cross section of the module. Likewise, many of the more serious types of structural failure are caused principally by these overall load effects. For example, one of MAESTRO’s failure modes for a stiffened panel is Panel Collapse, Membrane Yield (PCMY). Of course in its scope this failure mode relates to (occurs in) individual panels, and for this reason it is checked as part of the strake-by-strake analysis and adequacy evaluation. But if a panel was found to be inadequate, it would be inefficient to cure the problem only by a local strengthening of that panel. In order to achieve an optimum solution all such panels must be redesigned simultaneously, and the optimization must account for the influence that the redesign has on the overall load effect. In other words, the requirement (constraint) that such failure must be cured is part of the module level optimization problem, in which the variables are the strake sectional areas. At this second level of optimization MAESTRO determines the precise distribution of strake areas that best (optimally) satisfies the PCMY constraint and all other module level constraints. Consequently, for any structure in which the module level load effects, combined with other load effects, might be sufficient to cause membrane yield of plating, "dual level" optimization should be used. This is done by inserting the keyword "DUAL" as part of the "Further Constraints" data. Alternatively if the input data includes either of the other two "module level" limit states (MIN.IMOD and MIN.ZMOD) then MAESTRO automatically uses dual level optimization. (To give the designer maximum flexibility, the PCMY constraint does not automatically invoke dual level optimization.)

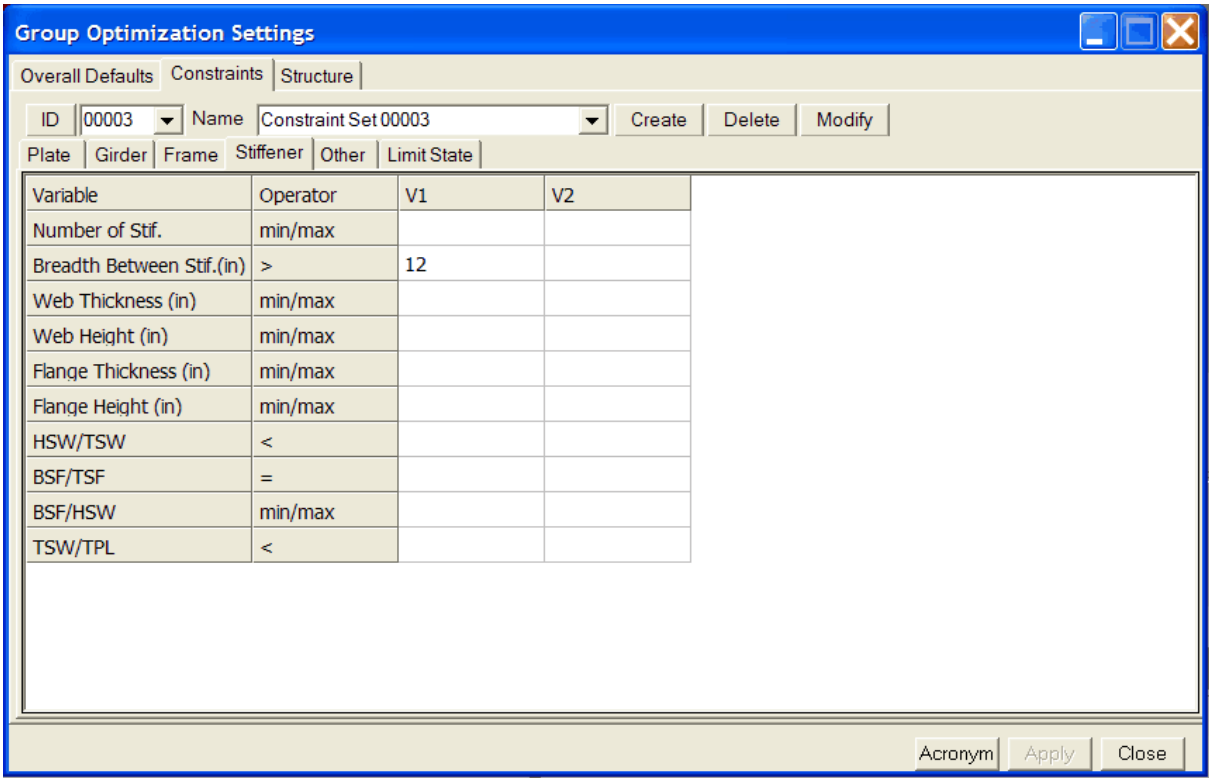
2.5.3 Nullifying a Structural Limit State

The Limit States subpage allows the designer to "nullify" a structural failure mode, which means that MAESTRO will ignore any inadequacy that might occur due to that failure mode, and that Limit State will not influence the optimization. Obviously, this should only be done when the designer is certain that the inadequacy will be dealt with by some other means, outside of the optimization process. In the Limit States subpage, double clicking to the right of a limit state acronym will generate a "nullify" tag.

2.6 Defining Constraint Sets That Are Strake-Specific

Thus far the Constraint Sets that have been defined are intended to be used for all strakes. Therefore, when we come to the third page of the Dialog Box, they will be placed in the Standard Group. But it often happens that there are some strakes for which we want to define some special (different) data, such as a special constraint. To do this we create a new Constraint Set in which we define that new data. Then, when we come to the third page of the Dialog Box, we will place that Constraint Set in a "General Group" which is associated with those strakes.

© 2014 Optimum Structural Design, Inc.



For example, the above figure shows a new Constraint Set (number 3) in which there is a constraint that the stiffener spacing must not be less than 12 inches, as might be needed to accommodate automatic welding machines. The Breadth Between Stiffeners, BBS, is an alternative design variable to the number of stiffeners, STF. The following four steps were used to create this Constraint Set. If you want to create it yourself then you must first delete it by using the down-arrow to select Constraint Set 3 and clicking on Delete. Then do the following:

1. Click on the ID button to generate a new Constraint Set, having the next available ID number.

2. Go to the line labeled Breadth Between Stif. and toggle (double click) the Operator until the > symbol appears.

3. Click in the cell under V1 and type in 12.

4. Click on Create. If you make any subsequent changes, be sure to click on Modify before leaving the page.

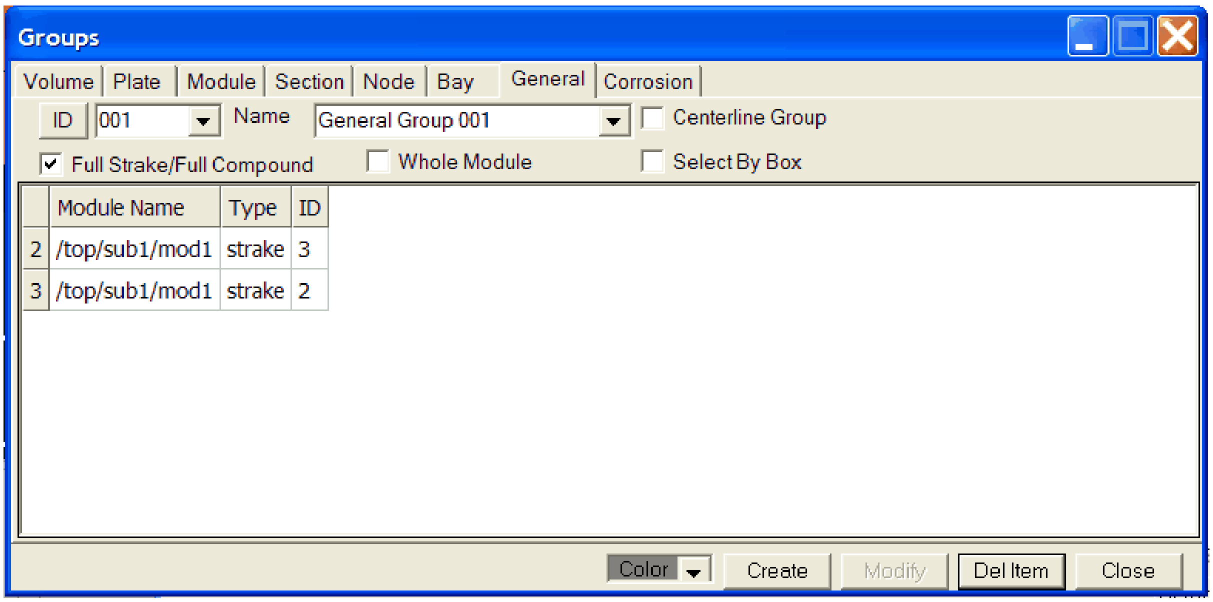
Later (in the third page of the Dialog Box) when we assign the Constraint Sets to Groups, we will include Constraint Set 3 in a Group that refers to the strakes for which this constraint is to be imposed. These will be strakes 2 and 3, which are the side strakes in the structure.

2.7 Using General Groups to Identify Modules and Strakes

The Standard Group of constraint sets is a module-independent group which serves as a

© 2014 Optimum Structural Design, Inc.

default group for all modules. We also need a way of identifying individual modules, and strakes within those modules, for which we want to associate some module-specific (or strake-specific) constraint sets. In this example we want to associate constraint set 3 – the set containing the BBS > 12 constraint that we just defined – with strakes 2 and 3 of module 1. We identify these modules by means of the Group Dialog Box, and the button for that is near the bottom of the Pre-Processing Toolbar, with a picture of thre stacked blocks.  **Click on this to open the Dialog Box and you will see that one of the types of groups is the General group. Click the General tab and then click the down-arrow in the ID box and select group 1. This will fill the List Box with the two strakes of module 1 that constitute General Group 1, as shown in the following figure.**



To see how this group was defined let's make a second group, using the following steps:

(1) Click the ID button to get the next ID number (002)

(2) Click the Full Strake/Full Compound check box, since we want to specify all of strakes 2 and 3

(3) Click anywhere in the List Box and then move your cursor to the model and click on the two side strakes of the structure (strakes 2 and 3). They will disappear to confirm that they have been selected.

(4) Normally at this point you would click on Create and then on Close, and you would have created a second General Group consisting of strakes 2 and 3 of module 1. But since we already have General Group 1 we don't want this second group, so click only on Close.

(5) Re-open the Optimization Dialog Box.

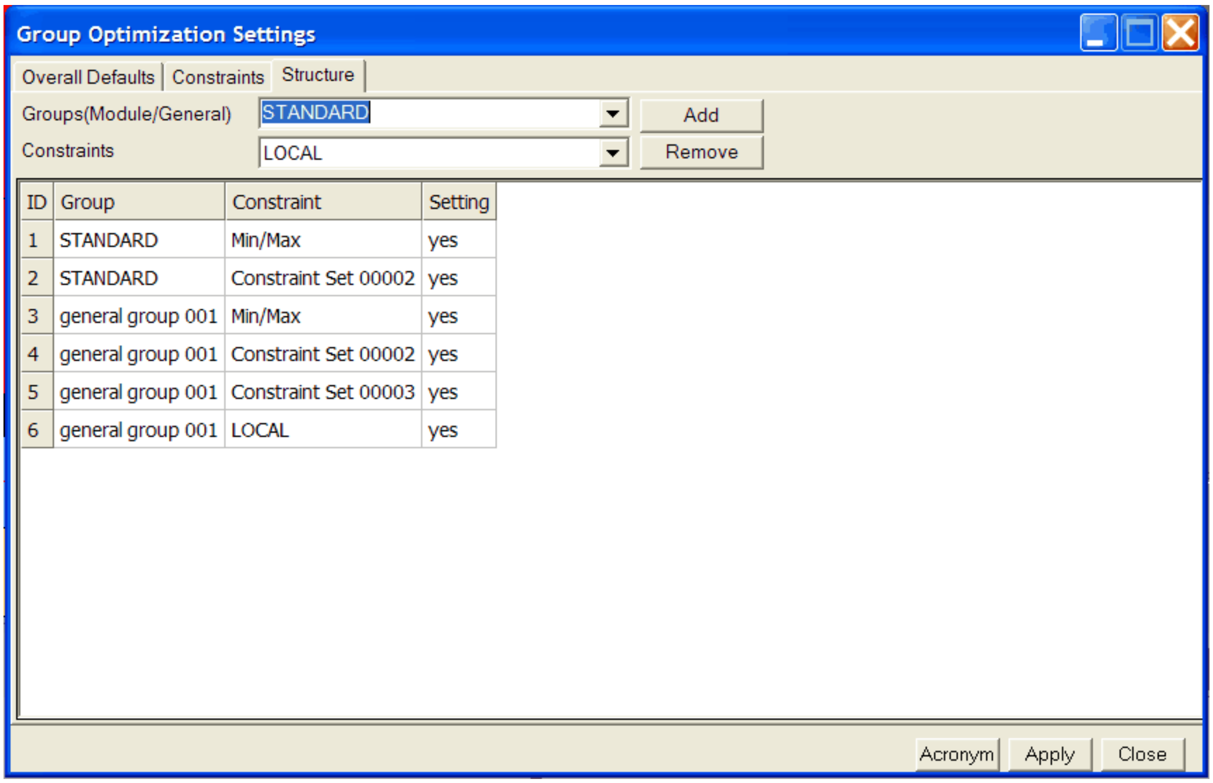
2.8 Optimization Dialog Box - Page 3, Creating Groups Of Constraint Sets

As noted earlier, optimization is performed separately for each module. The optimization data is specified in two Groups:

© 2014 Optimum Structural Design, Inc.

1. one Standard Group - a module-independent group of Min/Max constraints and other Constraint Sets which serve as default Constraint Sets for all modules,
2. any number of module-specific General Groups, containing Constraint Sets which can selectively override the default Constraint Sets, and which can also include other types of Constraint Sets that are not in the Standard Group.

The association of Groups and Constraint Sets is done in the third page of the Optimization Dialog Box, with “Structure” as its tab. In the first box you select which type of group (Standard or General) and in the second box you select, one by one, which Constraint Sets are to be in that group. The next figure shows the completed Dialog Box.



If you want to create this data yourself you should clear the six lines by highlighting them and clicking on Remove. Then recreate them as follows:

1. In the first box select Standard
2. In the second box select Min/Max
3. Click on Add, which generates the first line in the List Box, saying that Min/Max is the first Constraint Set in the Standard Group.
4. In the second box select Constraint Set 2
5. Click on Add, which generates the second line in the List Box, saying that Constraint Set 2 is the second Constraint Set in the Standard Group. This completes the Standard Group.
6. In the first box select General Group 1. This is the group that was defined using the Groups Dialog Box, and it consists of strakes 2 and 3 of module 1.
7. In the second box select Min/Max

© 2014 Optimum Structural Design, Inc.

8. Click on Add, which generates the third line in the List Box, saying that Min/Max is the first Constraint Set in General Group 1.
9. In the second box select Constraint Set 2
10. Click on Add, which generates the fourth line in the List Box, saying that Constraint Set 2 is the second Constraint Set in General Group 1
11. In the second box select Constraint Set 3 (recall that this was BBS > 12)
12. Click on Add, which generates the fifth line in the List Box, saying that Constraint Set 3 is the third Constraint Set in General Group 1
13. In the second box select LOCAL
14. Click on Add, which generates the sixth line in the List Box, saying that the LOCAL option is the fourth Constraint Set in General Group 1. This means that the strakes defined in General Group 1 (the side panels, strakes 2 and 3) will have only Local optimization (not Dual Level optimization).
15. Click on Apply, to preserve this definition of these two groups. This is like the Modify button in other Dialog Boxes.

2.9 Choice of Initial Values of the Design Variables

With MAESTRO the initial scantlings are arbitrary. They do not need to satisfy all or indeed any of the constraints. But it is preferable to select values that at least appear to be reasonable, because to do otherwise increases the number of design cycles required for convergence. If anything it is best to be conservative, because convergence is slightly faster from the feasible side. A good approach is to use whatever values may be available from existing similar structures, and to make a "best estimate" for the other values. In fact, the latter provides an interesting and challenging opportunity to try out one's "design eye", and to then gauge its accuracy from the extent of MAESTRO's changes either an improvement in the measure of merit due to the optimization, or the identification and correction of any inadequacies in the starting design, or both.

Oftentimes, from experience with a given type of structure it is known beforehand that a particular limit state is likely to be one of the governing constraints in the design. For example, some ship structural design rules require a certain minimum value of section modulus, and for some ship types this requirement is one of the governing constraints. In such cases a common technique in rulebased design is to first obtain member sizes that satisfy the local strength rules (which are relatively straightforward) and then to "scale up" the design until it just satisfies the overall strength rules, such as minimum section modulus of the structure. This is about the best that rulebased design can do; to do more would require the calculation of the load effects (stresses, etc.) and of all of the limit values of the load effects, and both of these tasks require a computer.

One of the aims of this example is to illustrate, qualitatively, the savings that rationally based design can achieve compared to rule based design. Therefore the starting design will be a typical "good" rulebased design one which just fulfills any governing overall (i.e. module level) constraints. The relevant constraint is the

© 2014 Optimum Structural Design, Inc.

minimum section modulus requirement, the second of the two module level constraints discussed earlier.

2.10 Running the Job and Viewing the Output File

We are now ready to run the job. The progress of the job, and later the job output file, will be displayed in the Output pane of the Modeler display, which can be viewed by clicking on the Output tab at the bottom left corner of the display. Then make the Output pane larger by placing the cursor on the border between the Viewport and the Output pane until you get a double-headed arrow and move the border up so that the Output pane nearly fills the entire display.

Now go to the execution toolbar (at the far right of all the toolbars along the top) and click on the Run Coarse Mesh button **(the MAESTRO logo).**

You will be asked if you want to overwrite the input file EX2.DAT. The usual response is Yes, in order that the file will reflect any and all changes you have made in the model by means of the Modeler. However, sometimes there are reasons to make a change to the DAT file by editing it directly, instead of using the Modeler. In that case you would answer No. Click on Yes.

In the Output pane MAESTRO will display its progress as it runs through the three optimization cycles and the final evaluation cycle that we requested (in the Job Info Dialog Box – see Section 2.1). When finished it will ask if you want to load the result. Click on Yes. If it asks about overwriting the ex2.rlt file, say Yes (always).

This loads the results into the Modeler so that the Modeler can display them graphically. However, most of the results of an Optimization Job are text and tables rather than graphics, and so we first want to look at the output file, ex2.out

In the Main Menu click on Edit and then on Maestro Out.

The output file consists of “pages” (although they are not separated – it is one continuous file). Each page starts with two lines containing the Job Title and other information, with the page number at the far right.

Page 20 of the output shows the initial design variables. The flange breadth of the frames and girders is set to the minimum value required to avoid lateral buckling. Here again the aim is to make the initial design be a typical "good" rulebased design, which fulfills the requirements but not to excess.

© 2014 Optimum Structural Design, Inc.

2.10.1 Initial Adequacy Parameters - Strake Level

After the finite element analysis, MAESTRO's next major task is the evaluation of all of the strakes calculating the 33 strakebased limit values, combining these with the actual values to obtain the adequacy parameters and then, for each limit state and for each strake, searching through all bays and all loadcases to find the worst (lowest) value of each adequacy parameter. The program then prints these lowest values in four tables, which in this example are on pages 2124 of the MAESTRO output. As mentioned elsewhere, before commencing any MAESTRO "DESIGN" job, an "ANALYSIS" job should always be performed first because it does several data checking operations that are not done in a "DESIGN" job, because of the cyclic nature of the latter. This example presupposes such an "ANALYSIS" job, which, among other things, would have established the adequacy of all of the strakes.

As indicated in these four tables, all of the constraints are satisfied. On page 21 it can be seen that one of the most important of them Panel Collapse, Stiffener Flexure is well satisfied, having an adequacy parameter of 0.194. This means that its ratio of strength/load (or capability/demand) is above the minimum required value of 1.5 (if it was right at 1.5 then the adequacy parameter would be exactly 0). This again shows that the starting design is already as good or better than a typical rulebased design. Such design does not calculate the ultimate strength of each panel because this would require a computer, even with the simplest algorithm (the algorithm used by MAESTRO for panel strength is given in Section 14.2 of Ship Structural Design). However, through a combination of theory and accumulated experience with a given type of structure, rules can be developed such that the design variables which they prescribe do give an ultimate strength that is close to the target value.

Page 24 gives the adequacy parameter values for some nonfailure constraints. The "area" constraint is the requirement that the strake crosssectional area be at least as large as the value that is determined by the module level optimization, in which the design variables are the strake areas. Since module level optimization has not yet occurred this constraint is not yet relevant.

The other columns in that table are not used in this example.

2.10.2 Initial Adequacy Parameters Module Level

After the strake evaluation the program then evaluates the limit states at the module level.

(1) PCMY Panel Collapse, Membrane Yield

As shown earlier (page 21 of the output) the initial design satisfies the PCMY constraint for both deck and bottom. Since this constraint actually

© 2014 Optimum Structural Design, Inc.

belongs to the module optimization problem, the PCMY adequacy parameters are printed again on page 25 (including, in this case, the values for the side strakes). The value of 0.2706 for the deck and bottom shows that their von Mises stress, even after being multiplied by 1.5, is well below the yield value.

(2) Minimum Required Moment of Inertia

In this example the required value is $3.0\text{x}10^5$ in^4. On page 19 of the output the actual value is shown to be $1.209\text{x}10^6$, and therefore on page 25 the adequacy parameter is well above zero, being 0.602.

(3) Minimum Required Section Modulus

In this example the required value is 15000 in^3. The actual value is $1.209\times10^6/72$ = 16,792 and so, as mentioned earlier, the starting design satisfies this constraint almost exactly, with an adequacy parameter that is close to zero (.056, on page 25). This shows that, as desired, the starting design is reasonably efficient. Because of the symmetry the value is the same for both deck and bottom.

2.10.3 Module Optimization

Page 26 of the output gives the results of the first cycle of module optimization under the heading “Values at start of module opt’n. cycle 2”. Because the initial design had slightly more than the required section modulus, the strake areas have been decreased slightly. The program performs two cycles of module optimization. The program obtains the new values of TPL, HSW, TSW, BSF and TSF by scaling the old values so as to match the new strake area. The scaling is applied only to these five variables; the other nine variables remain the same, including those that could have contributed to the strake area increase: the number of stiffeners (STF) and the girder variables. They are never changed at the module level because their strongest influence is on the strakebased limit states. The program then recalculates the adequacy parameters using the new panel scantlings, and using the same load effects as before (the finite element analysis is done only once in each overall cycle).

2.10.4 Strake Optimization

At this point the program performs the strake optimizations, in whatever sequence of strakes was requested. If none was requested (as here) the strakes are optimized in numerical order. The order is only relevant if the "LINK" option is being used, whereby some design variables in some strakes can be required to match, or bear a fixed relationship to, the (already optimized) variables in other strakes, in a "masterslave" manner.

© 2014 Optimum Structural Design, Inc.

2.11 Results Of The First Design Cycle

If the Job Type data specified the normal level of output, as in this example, there is no output during the entire strake optimization process. When it is finished the results are presented in three tables:

(1) the final set of active constraints for each strake (page 27)

(2) the new values of the module flexural properties (page 28)

(3) the new values of the design variables for each strake and the corresponding value of the objective function (page 29).

This information can be very useful since it indicates where the design is heading, precisely which constraints are "driving" it, and how much benefit has been achieved. This gives the designer some valuable insight and a better grasp of the situation in many ways, such as a better appreciation of what is important and what is not. In many cases it also gives ideas for improving the design in more general ways, such as by adding or deleting members, or by changing the geometry of the structure. (As will be shown later, this occurs in the present example). Also, some of the results may be different to what was anticipated, perhaps leading to the realization that a load has been forgotten, or that another constraint is needed. As will be shown, MAESTRO's "restart" feature makes it relatively easy to make such changes, without having to start all over from the initial design.

In the present case there are no unusual changes and the results are quite satisfactory. The objective function and its ingredients (cost and weight) have decreased. Compared with the original value (1.000 on page 20) the new value (0.830, on page 29) shows a 17% improvement. Figure 1 (the numbered figures are at the end of the text, just before Appendix A) gives a plot of the objective, cost and weight for all of the cycles that we will be doing.

2.12 Subsequent Design Cycles And Evaluation Cycle

The program now begins the second design cycle, first performing a new finite element analysis, then a new evaluation at both strake level (pages 30 to 33) and module level (page 34), then a module level optimization (35), and finally the strakebystrake re-evaluation and optimization (which is not printed). The active constraints are given on page 36 and the results are given on page 38. The objective function is now 0.790.

As requested (in the Job Information data) the program next performs a third design cycle (pages 39 to 47. The objective function has converged to a final value of

0.782 (page 47) a change of 21.8% from the original value of 1.000.

Finally, as requested in the Job Information data, MAESTRO performs an "evaluation cycle": a finite element analysis followed by an evaluation at both levels. This should normally be requested, because it is the only way of being sure that the results of the last design cycle satisfy all of the constraints. But this final evaluation also has a broader purpose. Since the specified number of design cycles has now been completed, the designer would usually want to review the results in more detail than before, to see whether any further cycles are required and to examine aspects of special interest. For example, for some of the limit states especially those that were the active constraints it may be of interest to learn just where in the structure each limit state has its lowest margin, and for what loadcase. Are several of them clustered in one location? Do they occur in the same loadcase? What particular combination of load effects is involved and what are their magnitudes? As with the optimization results, this type of information gives the designer a clearer and more complete view of the structure, and may generate ideas for further improvements.

In order to provide this information the output of the evaluation cycle is usually more detailed than any previous evaluation. The precise level of detail is specified in the Job Information data. In this example the output is on pages 49 to 67. (Note that the stress values and other results on these pages are superseded by designerspecified alterations to the structure, to be described in Section 2.13). The output for the additional members is on pages 68-70, and the output for the module evaluation is on page 71.

2.13 Review of the Design

Besides monitoring the cost and the adequacy of the design, it is important to examine its principal features what are the main changes and what constraints are directing the design? Figure 2 compares the initial design and the current design (defined on page 47 of the output) and Figure 2(b) shows some of the active constraints. Instead of four stiffeners there are now only two (1.7 at this stage; it is still too early to discretize) and the plating is thicker.

2.13.1 Sample Investigation Of An Active Constraint

This section shows briefly how to obtain information about the active constraints, taking the FYCF2 constraint as an example.

The active constraints are given on page 45. For strake 1 one of the active constraints arising from a limit state (or failure mode) is FYCF2, which means Frame Yield, Compression, Flange (the R prefix simply means it was active repeatedly). For a frame failure mode the number at the end is always 1, 2 or 3, signifying the worst location: strake edge 1, strake edge 2, or the middle. Therefore FYCF2 means that the relevant location (the place where the total compressive stress (axial + bending) in the frame flange has reached (approximately) the maximum permissible value of $_{Y}/1.25$) is at strake edge 2. Since there is a bracket at this edge, the precise location is at the toe of the bracket.

The required information can be obtained from the evaluation cycle output for strake 1: pages 49 to 52. For example, the summary on page 52 shows that for the FYCF2 limit state the relevant section is section 2 and the relevant loadcase is 2. Turning back to this section and loadcase on page 50, the value of the bending moment at the bracket toe at strake edge 2 (the second of the three values) is seen to be 3.778E05. The bending moment value at the far right, MFBKT2, is the value at the junction of the flexible length and the rigid length (see Section 8.6 of reference 1). This is larger than the value at the bracket toe, but since it is within the bracket it is not used in optimizing the frame. But it is supplied by MAESTRO so that it can be used later, in designing the bracket. The output also gives the frame stresses. The stress in the frame flange (axial + bending) is ASIGFF2 = 11,230 psi and the shear stress in the web is TAUF2 = 12,320 psi. Together these cause the frame flange to be just at the maximum allowable stress condition, which is 80% of yield (p.s.f. = 1.25). For that reason the FYCF2 adequacy parameter is exactly zero (page 52).

2.13.2 Individual Influence of the Active Constraints

The first eight active constraints (page 45) two "minimum" and six "proportionality" are all explicit functions of the design variables, and so they can be shown directly on Figure 2(b) as the "governing dimensions" of those design variables. The constraints relating to structural failure do not have any such explicit relationship, but it is clear from the nature of each of them that FYCF2 would mainly control the frame web height, and PSPBL and AREA would mainly control the number and height of the stiffeners and the plate thickness. The R in front of PSPBL, FYCF2 and AREA indicates that the constraint was "repeated" (i.e. was a strong influence).

2.14 Example Of Design Modification Deletion Of Girders

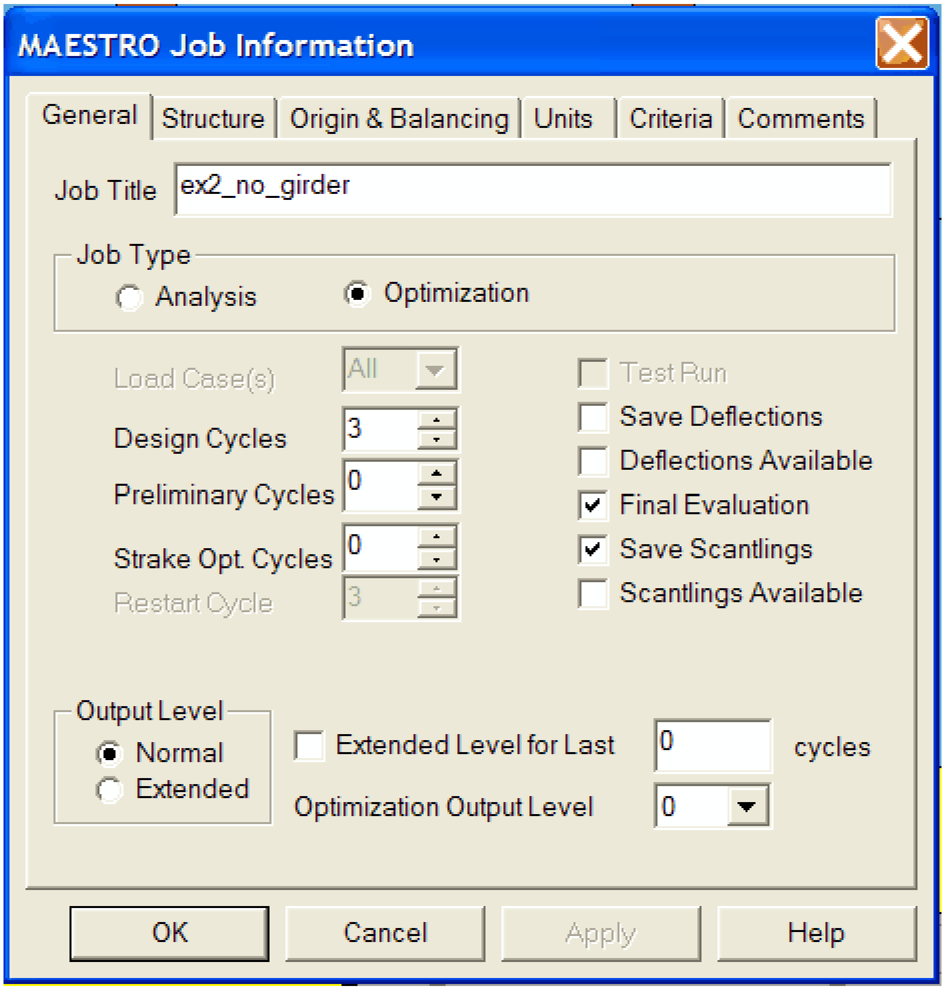
Inspection of Figure 2(b) reveals an interesting fact: the girder is as small as it can possibly be. The girder web is as small as it can be (10% higher than the frame web, and a thickness that is 1/50 of the height) and the flange is at minimum breadth (the 9 inches required for flexuraltorsional buckling) and minimum thickness (limited by the 1:25 proportion). Also, none of the girder limit states was an active constraint, and all of the girder adequacy parameters are well above zero (page 52 for strake 1). It therefore appears that the girders may not be needed.

© 2014 Optimum Structural Design, Inc.

This possibility was investigated by deleting the girders. However, it is important to note that any change in the number of strakes or the number of girders requires that the optimization job be performed again, from the beginning (cycle number 1) in order that the restart file has the correct number of strakes and girders. The job can be repeated for the same number of design cycles, or if the convergence had been slow or rapid, for more or fewer cycles.

Another important consideration is that up to now all of the text of this document refers to the output of the original job (EX2.OUT) and so we certainly do not want to overwrite that file. Therefore we will now create a new Modeler "mdl" file by copying and modifying the model in the following steps:

(1) Create a new model by going to the Main Menu, clicking on "File / Save As", entering ex2_no_girder as the file name and clicking on Save.
(2) Delete the girders by going to the Strake Dialog Box and for strakes 1 and 4 uncheck the "Enable Girders" check box and click on Modify.
(3) To avoid confusion between the outputs of the original model and the "no girders" model, it will be helpful to change the job title to ex2_no_girder, because the title is printed at the top of each page. Go to the Job Information Dialog Box and replace the existing title by ex2_no_girder.
(4)



(5) Also in the Job Information Dialog Box, specify that a restart file should be generated by checking the Save Scantlings check box, as shown in the next figure it is probably already checked). Keep the number of design cycles at 3 so that the new restart file has all 3 cycles. Click OK.

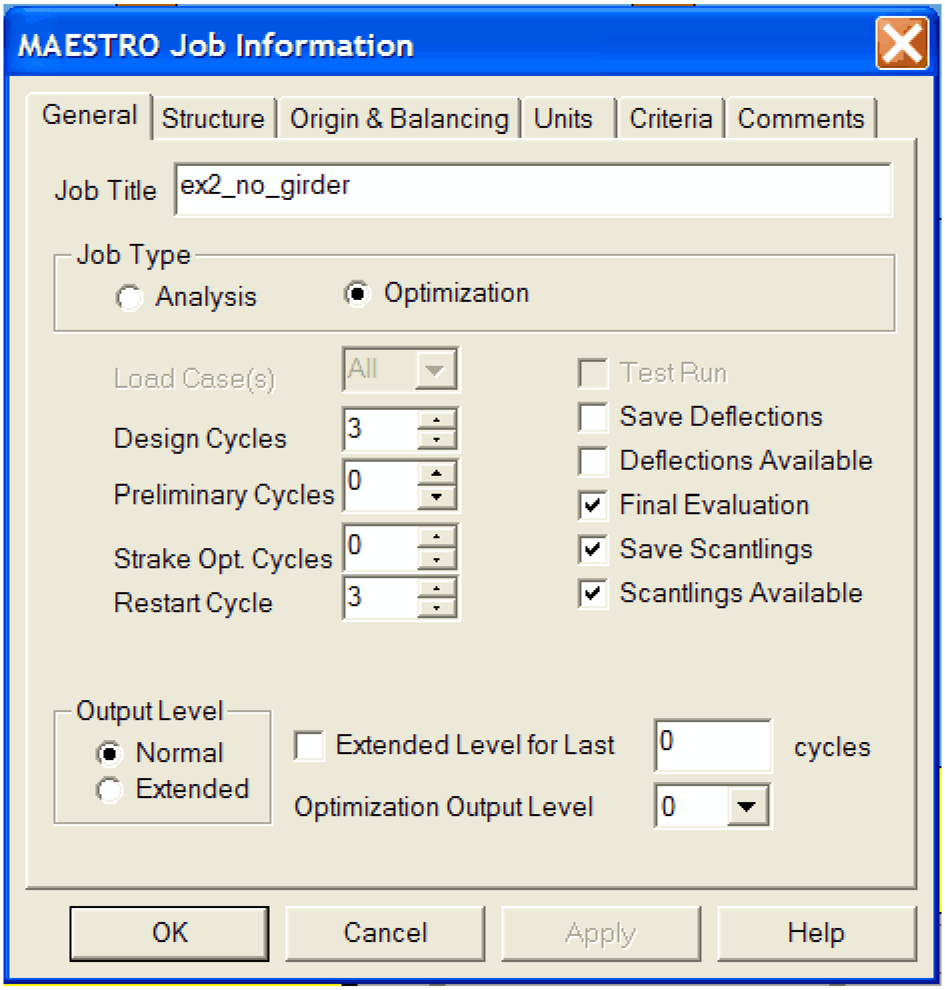
Now we must run MAESTRO in order to create the new restart file, consistent with the deletion of the girders. That is, we are repeating the first three cycles, but with no girders.

(6) Click on the “execute” button at the far right of the upper toolbar (the MAESTRO logo). When asked about overwriting click Yes because we have changed the model (no girders).

2.15 Job No. 2 (Restart: Cycles 4 And 5) - No Girders

A restart file is a file created by MAESTRO (if requested, as we just did) for each module that is currently being optimized. Each restart file contains the initial scantlings and the new scantlings that are obtained in each design cycle. Each restart file has the name *jobname*.sMN, where MN is the cumulative module sequence number, numbering all the modules sequentially among all the substructures. The new (and consistent) restart file that we just created is ex2_no_girder.s01.

Once a (consistent) restart file has been created, the optimization can be continued from any of the previous cycles that have been performed. The user specifies a restart run through the Job Information Dialog Box, and the modeler automatically pulls the chosen scantlings from the restart file. To make a restart run, open the Job Information Dialog Box and check the "Scantlings Available" box, as shown in the next figure. This will cause the Restart box to be changed from gray to white, and you can enter the starting cycle. For example the next figure shows that the second run will started from the latest (cycle 3) design. Enter the number of new design cycles in the Design Cycles box. In the next figure a further two cycles are specified, plus an evaluation cycle.



NOTE: After deleting the girders it would have been easier to simply rerun the job from the beginning, for the full 5 cycles; i.e. ignore the Restart Cycles box (leave it at zero) and in the Design Cycles box click the up arrow to obtain the number 5. Then there would be no need to create a new restart file for the first three cycles; it would have been created as part of the restart file for the 5-cycle job. The only reason for restarting from cycle 3 was to demonstrate how to restart a job from any cycle, rather than starting all over. For a full ship model this can save a considerable amount of execution time.

Before making a restart run the designer can make further changes to the model, as long they do not change the number of strakes or the number of girders. For example changes could be made to the loads, the optimization constraints, and the objective function.

Run the restart job. This starts with the results of cycle 3 and produces cycles 4 and 5. You will be asked about overwriting ex2_no_girder.DAT. That DAT file was made when we repeated the first three cycles. We now want to do cycles 4 and 5, and so we do need to update the DAT file. Click Yes.

We will now look through the output of this second job, which is the file ex2_no_girder.OUT.

Open the output file by going to the Main Menu and clicking on Edit / MAESTRO Out. If necessary enlarge the Output pane by dragging its upper boundary upward. Right click and select Find. Enter “objective function” and click on Find Next. This will take you to page 19 of the output which has a table of scantlings at the start of cycle 4 (a restart job maintains the original cycle numbering). The scantlings are all the same as at the end of the third cycle of the first job, except that there are now no girders. Therefore the weight, cost and objective function are all less than before. For example the objective function is now 0.753 compared to 0.782 before, or about 3% less.

As can be seen in Figure 1, this 3% savings is reflected in both the cost and the weight, because they were given equal importance. Most of this savings is due to the elimination of the girders. (In reality the savings in cost would be still greater because of the elimination of the connections between frames and girders, which is not considered in the simple cost function being used here).

The results of design cycle 5 are given on pages 35-37. There are now only 10 active constraints (page 35) because there are only 10 design variables. As shown on page 37, the objective function is now 0.748, only slightly less than the starting value, and so we conclude that the optimization has essentially converged.

The Final Evaluation results are given on pages 38-54. Page 55 is the histogram from the evaluation cycle, showing that all but two constraints are satisfied. The table on page 51 shows that they are PYCF in strakes 1 and 4, and that the adequacy parameter is very small, being only -0.012. This result verifes that the girders were indeed not needed.

Up to five modules can be optimized at once, but because a typical module is in itself a rather large structure, with many design variables and involving a large amount of information, it is recommended that only one or two modules be optimized at one time. Once a module has been sufficiently optimized (usually in a series of runs, each having any number of design cycles and restarting from the results of any previous cycle) two final steps are needed:

(1) the General Group for that module should be removed from the List Box on the Structure page of the Optimization Dialog Box.

(2) the final scantlings should be copied from the restart file of that module using a text editor (as explained in the next section) and pasted into the job data (jobname.DAT) in place of the original scantlings.

The optimization process is then repeated for another one or two modules, again involving a series of runs.

© 2014 Optimum Structural Design, Inc.

2.16 Job No. 3 (Restart: Cycle 6) Discretize At End

From the plots of Figure 1, it can be seen that at this point (end of cycle 5) the optimization process is close to convergence and hence it is time to discretize the design variables; i.e. to convert them to standard sizes. During most of the design cycles, MAESTRO deliberately treats all design variables, including the number of stiffeners, as continuous variables, in order to avoid the enormous computation and complexity of discrete variable optimization. This also has the advantage of allowing the designer to see what the idealized optimum is, and thus perhaps obtain new ideas regarding the degree of standardization and commonality, or at least be better able to guide and direct the discretization process.

In the DAT file the input data for discretization consists of a list of available thicknesses, the roundoff fractions and the size increments. This information is supplied in the DISCRETE data group, at the beginning of the optimization data. At present the Modeler does not yet have a Dialog Box for the user to enter this data. Instead it has just one box for entering a fixed increment in plate thickness, and in producing the DAT file it simply adopts the default values for all other items in the discretization data. If you want to use other values you must edit the DAT file and type them in. Editing the DAT file is similar to editing the OUT file – in the Main Menu click on Edit / MAESTRO Dat and the Modeler will bring up the DAT file using the Notepad editor. The format for typing the discretization data is given in Appendix A, taken from the Data Preparation Manual.

In a DESIGN job for which discretization is requested, it is performed as part of the last design cycle, and the subsequent evaluation cycle then provides a check that the resulting scantlings still satisfy all of the constraints. The discretization is not simply a rounding up all of the design variables to the next standard size. Rather it consists of six cycles, which are referred to as "rounds" in order to distinguish them from design cycles. Each round begins with the rounding off (up or down, according to the roundoff fractions specified by the designer) of a further one, two or three design variables, followed by a full optimization of the remaining free (undiscretized) variables. In this way, if a design variables is rounded down the remaining free variables will adjust themselves so as to prevent whatever slight degree of inadequacy might otherwise occur.

We now want to perform one more cycle of optimization, starting with the results of cycle 5 and obtaining discretized results. In Job No. 2 (labelled ex2_no_girder and consisting of cycles 4 and 5) we did not make any changes to the structural model. Therefore we have a restart file (ex2_no_girder.s01) that is valid and complete, and we could now perform a 6th cycle using the same Modeler file (ex2_no_girder.mdl). In the Job Information Dialog Box we would enter 5 in the Restart Cycle box and enter 1 in the Design Cycles box (to do one more cycle).

However, if we did use the same Modeler file then the previous output file (ex2_no_girder.OUT) would be overwritten and the page numbering would change. We don't want that because this tutorial has referred to some of those pages. Therefore we will make a new Modeler file and then enter the data for cycle 6, as follows:

(1) Go to the Main Menu, click on "File / Save As" and enter ex2_cycle6 as the file name. Click on Save.

(2) Since the job title is printed at the top of each page we should also change it to ex2_cycle6. Go to the Job Information Dialog Box and in the Job Title box replace ex2_no_girder by ex2_cycle6.

(3) In that same Job Information Dialog go to the Restart Cycle box and click the up arrow twice to change 3 to 5. The go to the Design Cycles box and change 2 to 1 (to do just one more cycle).

(4) Click on OK.

(5) Bring up the Optimization Dialog Box (Model / Optimization) click on Group Opt. Settings. In that Dialog Box the ID will be 1 and the Group will be STANDARD.

(6) In the Discretization section select On.

(7) Let's say that plating is available in 1/16 inch increments. Therefore in the Plate Thickness Increment box enter 0.0625.

(8) Click on Modify.

(9) Steps (6) – (8) were for the STANDARD Group. Click the ID down-arrow and select general group 1. Then repeat steps (6) – (8).

(10) Save this new Modeler file – go to the Main Menu and click on File / Save.

(11) We also need to make a new restart file. Recall that in order to avoid overwriting the output file ex2_no_girder.OUT (which would disturb the page numbering) we made a new Modeler file for cycle 6 called ex2_cycle6.mdl. Normally this would not be necessary – it was only done because of the Tutorial. But since we did make a new Modeler file we must also make a new restart file that has the same name. We cannot do this by "File / Save As" because the Modeler can only read mdl files. We will do it by renaming the current restart file.

In the Main Menu click on File / Open. In the Open dialog box go to the bottom, click the down-arrow on Files of type, and select All Files. Select the file called ex2_no_girder.s01, right click and select Rename. Rename the file to ex2_cycle6.s01. Close the Open dialog box.

© 2014 Optimum Structural Design, Inc.

Now we are ready to perform the sixth cycle, during which the scantlings will be discretized.

In the upper toolbar click on the Execute icon. Select Yes to generate a new DAT file. After the job has completed enlarge the Output pane by dragging its upper border. Bring up the output file (Edit / MAESTRO Out).

The final scantlings, all of which are now standard sizes, are given in the table on page 28 of the output. Notice that the number of stiffeners is now an integer. The new design is illustrated in Figure 2. As shown in Figure 1, the objective function and the cost have increased (from 0.748 to 0.766 - about 2.4%) whereas the weight has stayed essentially the same. The reason for this is that most of the weight is in the plating, which stayed the same thickness, whereas the number of stiffeners increased, which increased the welding cost and hence also the objective.

2.17 Active Constraints in the Final Design

After deleting the girders in strakes 1 and 4, these strakes have 10 design variables. Since MAESTRO uses (an improved version of) linear programming to perform the optimization, the number of active constraints is always equal to the number of design variables. For the Final Design these constraints are listed in the table on page 26 of the output. There is one minimum constraint and four proportionality constraints. For min/max and proportionality constraints it is possible to show graphically how these constraints govern the final scantlings, and Figure 2 (c) illustrates this. The other five constraints are three failure modes and two Area constraints. The latter are minimum values of the strake cross sectional area to satisfy the overall (hull girder) constraints (MIN.IMOD and MIN.ZMOD). From the nature of the failure modes it is possible to make a logical connection between the design variables and the constraints that most influenced them, as shown in the following table.

Design Variables	Governing Constraints
BFF	MIN.BFF
TSW, BSF, TSF, TFW	HFW/TFW, BFF/TFF, BSF/TSF, HSW/TSW
HSW	PYCF (Panel Yield, Compression, Flange)
TPL	AREA

© 2014 Optimum Structural Design, Inc.

HFW	RFYCF2 (Frame Yield, Compression, Flange)
TFW	RAREA
STF	RPSPBL (Panel Serviceability, Plate Bending, Longitudinal)

2.18 Updating the DAT file by Copying and Pasting from the Restart File

The restart files are text files, similar to the input data file (*jobname.DAT*). When a module has been sufficiently optimized, the designer should transfer its final optimum scantlings from the restart file into the appropriate place in the job data file. This transfer is accomplished by copying and pasting using a text editor such as NotePad. For example, with the NotePad editor the procedure would be as follows for Example 2:

(1) Within the "File" option, "Open" the restart file, ex2_no_girder.S01.
(2) Highlight the lines containing the final optimum scantlings (do not include the last line containing the objective function, cost and weight).
(3) Within the "Edit" option, "Copy" these lines (to the Paste buffer).
(4) Go back to the "File" option and "Open" the original dataset file EX2.DAT.
(5) Go to the beginning of Data Group IX of the module corresponding to this restart file (in this example there is only one module).
(6) Highlight the lines containing the old scantlings.
(7) Within the "Edit" option, "Paste" the lines from the restart file into the data file.

The result is that data groups IX, X and XI of the job data file now contain the final optimum scantlings.

2.19 Updating the Modeler file by Importing the DAT File

Finally, it is necessary to update the Modeler file (in this case ex2.mdl) because if MAESTRO was run again and was allowed to write a new DAT file the above updates would be lost. To update the Modeler file go to the Main Menu, click on File / Import and select the newly updated DAT file. Since the new scantlings are not in the Properties lists, the Modeler will create new properties for all of them.

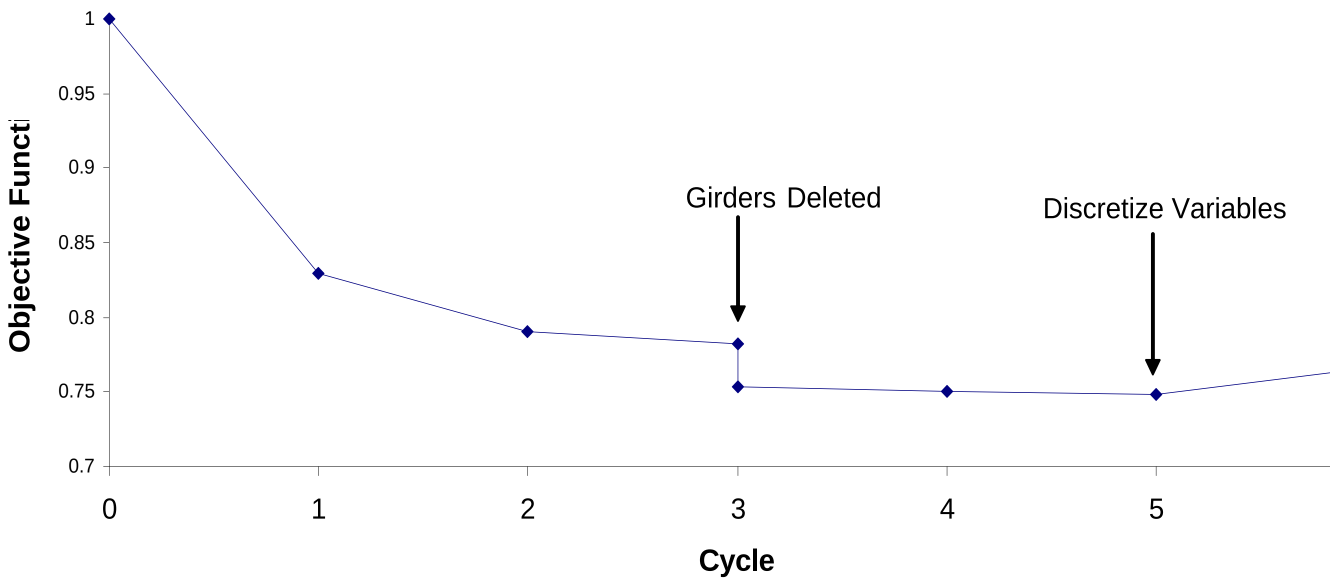
NOTE: At the time of writing (July 2005) the Modeler's Import option has not been extended to include the optimization data. It will be extended in a future update.

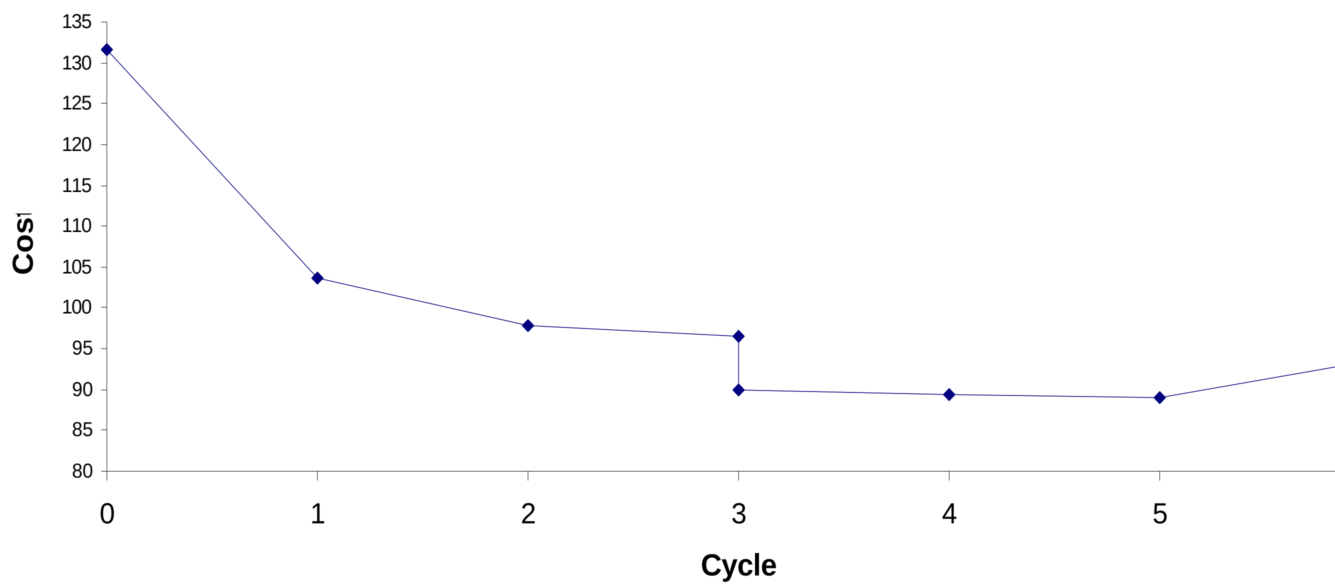
© 2014 Optimum Structural Design, Inc.

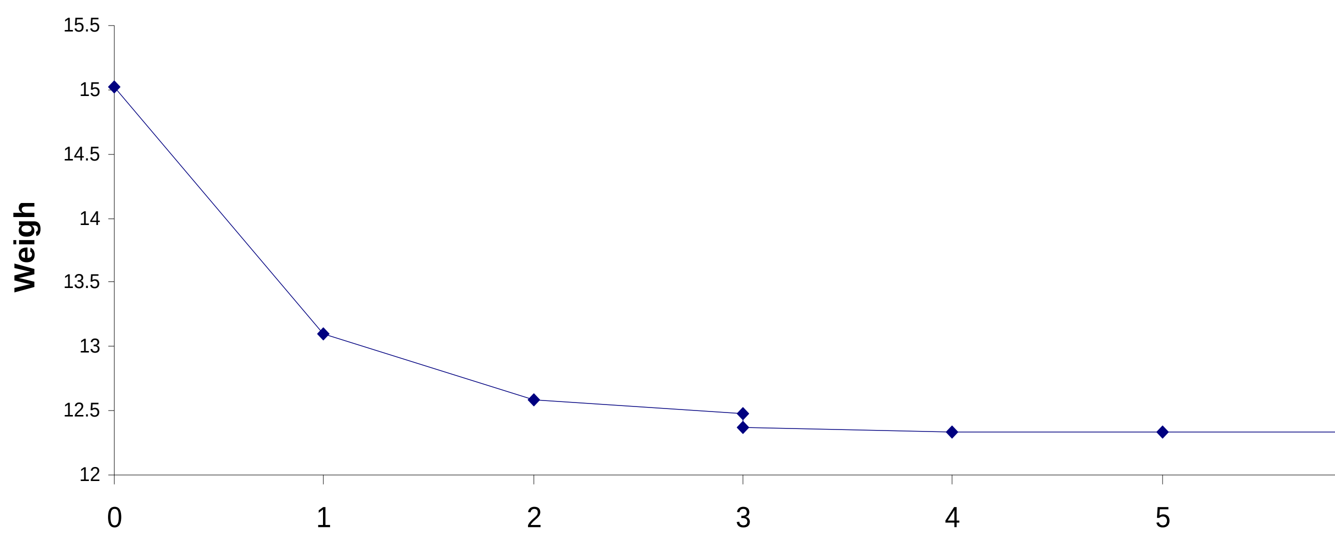
16.6.3 References and Figures

References and Figures

1. Hughes, O.F., *Ship Structural Design*, Society of Naval Architects and Marine Engineers, Jersey City, NJ, 1988.

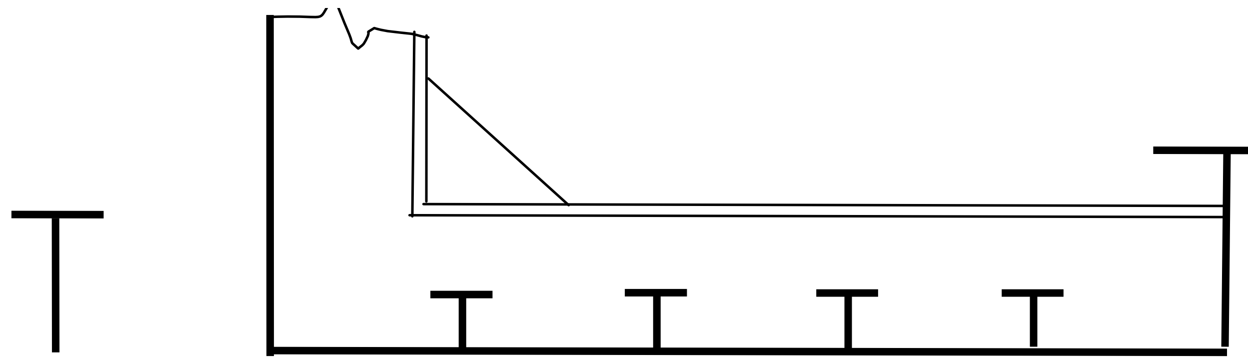

Objective Functi
Girders Deleted
Discretize Variables
Cycle


Cost
Cycle

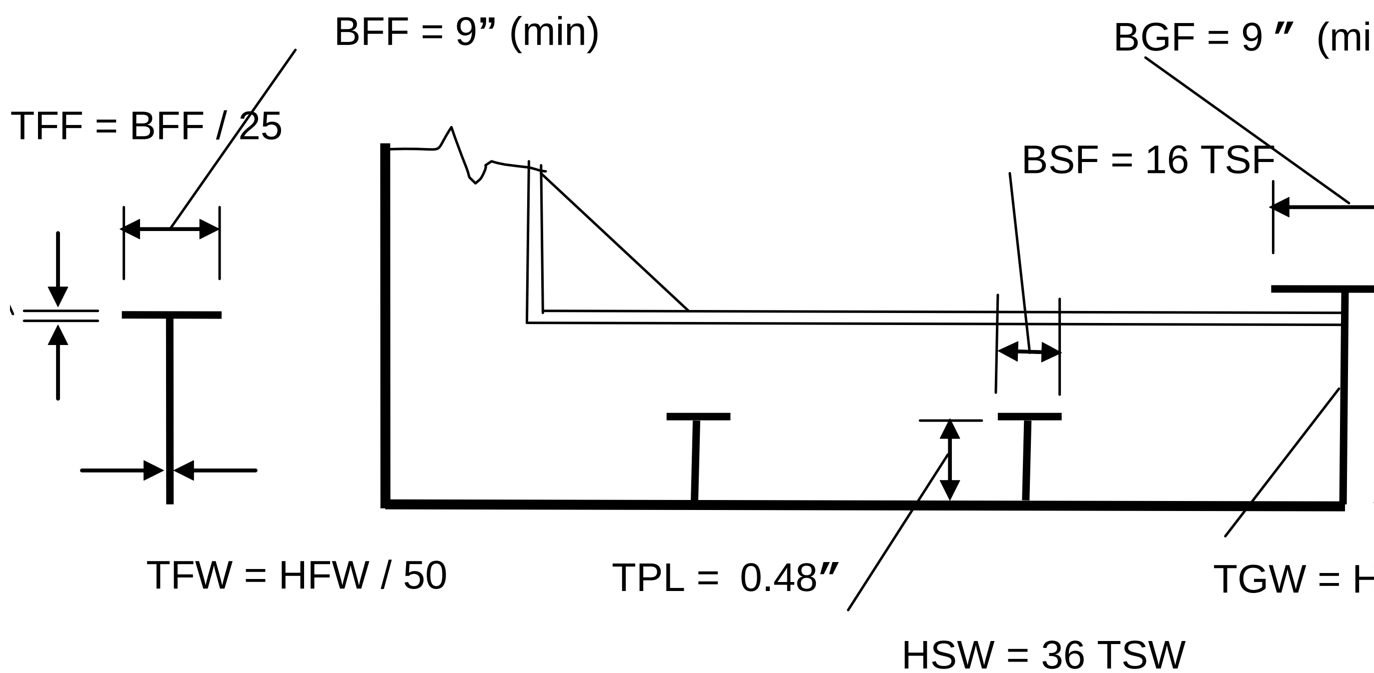

Weigh

© 2014 Optim

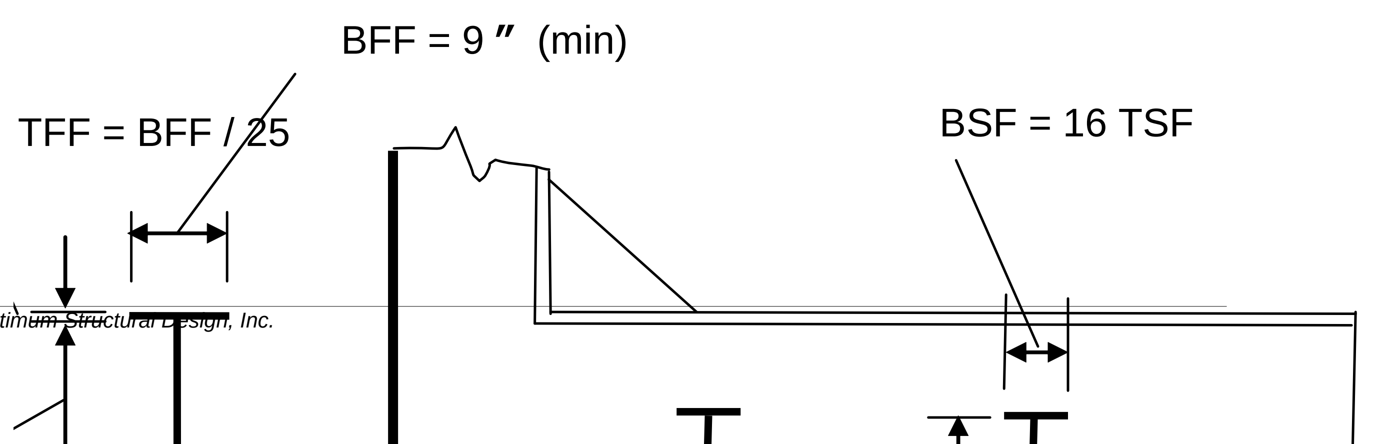
Figure 1- History of Objective Function, Cost and Weight



(a) Initial Design



(b) Scantlings and Constraints at End of Cycle 3



© 2014 Optimum Structural Design, Inc.

16.6.4 Appendix A

Appendix A

Format for Specifying the Discretization Data

(excerpt from the Data Preparation Manual)

9.1.3 DISCRETIZE Data Group

This optional group allows the designer to specify allowable thicknesses and size increments for the final values of the design variables. If this group is included, then in the last design cycle the program will round off each of the hitherto continuous values of the design variables to one of the allowable discrete values. The evaluation cycle is then performed on these discrete values. Each rounding off may be either up or down, depending on the current structural adequacy of the affected structure, and on "roundoff fractions" specified by the designer.

FIRST LINE:

ITEM 1 DISCRETIZE or NODISCRETIZE (keyword; the first four letters are sufficient).

NODISCRETIZE indicates that strake design variables will not be rounded to the specified allowable values. If no roundoff of strake variables is desired, there is no need for this data line. But once the DISCRETIZE data has been created, adding the two-letter prefix is a convenient way of switching off the roundoff feature and avoids having to remove or "comment out" the data for it.

ITEM 2 The number of discrete plate thickness values. These values will be used for the strake plating and also for the webs and flanges of the stiffeners, girders, and frames.

ITEM 3 Type of roundoff. Normally the roundoff is done progressively, modifying only a few design variables at a time, in order that a decrease in a design variable can be compensated for, if necessary, by an increase (rounding up) of a subsequent design variable. This is the default, and will be done unless it is overridden by entering the keyword SIMPLE for this item. In that case the program simply rounds off all design variables simultaneously, according to the roundoff fractions. A null or any other entry will give the ordinary (progressive) roundoff.

© 2014 Optimum Structural Design, Inc.

ITEM 4 Debug output flag. A 1 will produce additional output during the roundoff process. This item should normally be null or zero.

SECOND LINE:

ITEMS 18

The first eight plate thickness values, in order of increasing plate thickness.

THIRD LINE:

ITEMS 18 The next group of eight plate thickness values, again in order of increasing plate thickness.

etc. As many lines are used as are required to input the total number of values specified by Item 2 of the first line. (The size parameter MPLTSZ defines the maximum number of allowable values.) All lines except the last line (of plate thicknesses) must have eight values.

LAST LINE:

ITEM 1 Roundoff fraction for the number of panel stiffeners, STF. When the fractional part of the number of stiffeners is greater than this value (STFROF), the number of stiffeners is rounded up to the next integer value. For values less than STFROF, the number is rounded down to the next lower integer. The default value is 0.25.

To switch off discretization of STF, specify a value of 1 for this item.

ITEM 2 Plate roundoff fraction (PLTROF) for the strake plate thickness and for the web and flange thicknesses of the stiffeners, girders, and frames. For a thickness value TPL between discrete values T(N) and T(N+1) the value of TPL will be rounded up to the value T(N+1) if the ratio

is greater than PLTROF, and rounded down to T(N) if the ratio is less. The default value is 0.25.

To switch off discretization for plate thickness, specify a value of 1 for this item.

© 2014 Optimum Structural Design, Inc.

ITEM 3 Panel roundoff fraction (PANROF) for the web height HSW and flange breadth BSF of the panel stiffeners. This item is used in a similar fashion to Item 2 except that the appropriate stiffener values are used instead of the plate thicknesses. The default is 0.25.

To switch off discretization for HSW and BSF, specify a value of 1 for this item.

ITEM 4 Girder roundoff fraction (GIRROF) for the web height HGW and flange breadth BGF of the girders. This item is identical to Item 3 except that this value is applied to the girders. The default value is 0.25.

To switch off discretization of HGW and BGF, specify a value of 1 for this item.

ITEM 5 Frame roundoff fraction (FRMROF) for the web height HFW and flange breadth BFF of the frames. This item is identical to Items 3 and 4 except that this value is applied to the frames. The default value is 0.25.

To switch off discretization of HFW and BFF, specify a value of 1 for this item.

ITEM 6 Dimensional increment (PNDELT) for the web heights and the flange breadths of the panel stiffeners. If the Nth discrete value of the web height is HW(N), then the N+1st value is defined as HW(N+1) = HW(N) + PNDELT. Note that this item has dimensions so the appropriate units as previously defined must be used. The default value is 10.0 for mm, 0.010 for m and 0.25 for inches.

ITEM 7 Dimensional increment (GRDELT) for the web heights and flange breadths of the girders. This item is similar to Item 6 except that this value applies to girders. The default value is 20.0 for mm, 0.020 for m and 0.5 for inches.

ITEM 8 Dimensional increment (FRDELT) for the web heights and flange breadths of the frames. This item is similar to Items 6 and 7 except that it applies to the frames. The default value is 20.0 for mm, 0.020 for m and 0.5 for inches.

© 2014 Optimum Structural Design, Inc.

Verification and Validation

17

17 Verification and Validation

The topics in this section provide test results for MAESTRO element verification studies and compare these results to theoretical and other FEA software results.

All data files referenced in the following sections can be found in the MAESTRO installation directory under MAESTRO/Models & Samples/Verification Models.

17.1 CQUAD4R

The following sections provide various verification tests for MAESTRO's quad element.

17.1.1 Patch Test

For the patch tests, there are five files in all. Three of them are set up for the constant in plane strains, corresponding to tensions in X, Y direction and shear in X-Y plane. The other three files are for constant bending curvature, corresponding to pure bending in X, Y direction and pure twist. Table 2 gives the boundary conditions, loads and the theoretical strains and stresses for the patch elements for the five tests. The MAESTRO recovered stress and displacements are presented along with the results from MSC/Nastran for Windows V3.0 for comparison.

The patch test model geometry is presented in Figure 1. The location of the nodes are presented in Table 1.

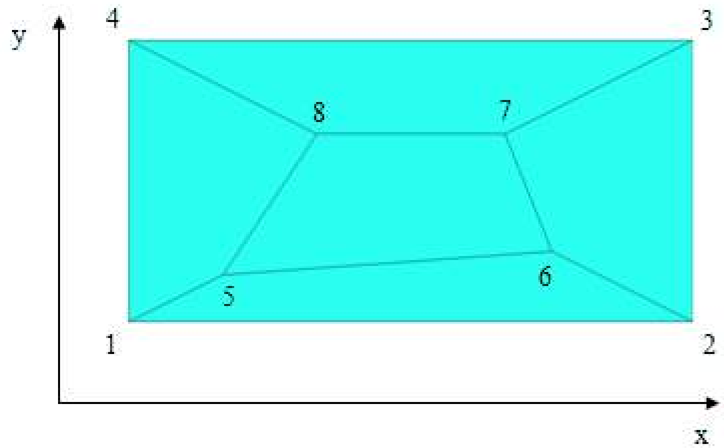


Figure 1 Patch Test Model, E = 1.0e^{06}, v = 0.25

Node	X	Y

1	0.00	0.00
2	0.24	0.00
3	0.24	0.12
4	0.00	0.12
5	0.04	0.02
6	0.18	0.03
7	0.16	0.08
8	0.08	0.08

Table 1 Patch Test Model Node Locations

	BCs	**Loads**	**Theoretical Strain**	**Theoretical Stress**	**MAESTRO Solver**	**MSC/Nastran for Windows V3.0**	**Input Data Files**
Constant Strain in X-Direction	1: 111001 2: 011000 4: 101000	2: P_x = 0.06 M_z = -1.2e-03 3: P_x = 0.06 M_z = 1.2e-03	$e_x = 10^{-03}$ e_y = 0.25e-03 e_{xy} = 0	s_x = 1,000 s_y = 0 s_{xy} = 0	U_2 = 0.000289978 U_3 = 0.000287805 V_3 = -0.000123489 s_x = 1013.8	U_2 = 0.00028987 U_3 = 0.00028765 V_3 = -0.00012322 s_x = 1012.42	patchx.mdl patchx.nas patchx.mod patchx.f06
Constant Strain in Y-Direction	1: 111001 2: 011000 4: 101000	3: P_y = 0.12 M_z = -4.8e-03 4: P_y = 0.12 M_z = 4.8e-03 2: M_z = 4.8e-03	e_x = 0.25e-03 $e_y = 10^{-03}$ e_{xy} = 0	s_x = 0 s_y = 1,000 s_{xy} = 0	V_3 = 0.00012 V_4 = 0.00012 $U_3 = -6.0e^{-05}$ s_{VM} = 1,000	V_3 = 0.00012 V_4 = 0.00012 $U_3 = -6.0e^{-05}$ s_{VM} = 1,000	patchy.mdl patchy.nas patchy.mod patchy.f06
Constant Shear in X-Y Plane	1: 111011 4: 101000	2: P_x = -0.048 P_y = 0.024 3: P_x = 0.048 P_y = 0.024 4: P_y = -0.024	e_x = 0 e_y = 0 $e_{xy} = 10^{-03}$	s_x = 0 s_y = 0 s_{xy} = 400	V_3 = 0.00024 s_{xy} = 395.94	V_3 = 0.000235 s_{xy} = 399.72	patchs.mdl patchs.nas patchs.mod patchs.f06

© 2014 Optimum Structural Design, Inc.

	BCs	Loads	Theoretical Strain	Theoretical Stress	MAESTRO Solver	MSC/Nastran for Windows V3.0	Input Data Files
Constant Bending m_y= 8.889e-08	1: 111111 2: 000100 3: 000100 4: 101111	2: M_y = 5.33e-09 3: M_y = 5.33e-09	Curvature: 1.0e^{-03} Slopes: 2: q_y = 2.4e^{-03} 3: q_y = 2.4e^{-03}	Surface Stress: s_x = 0.533	q_{3y}: 0.00023985	q_{3y}: 0.00023985	patchb.mdl patchb.nas patchb.mod patchb.f06
Constant Biaxial Bending m_{xy} = 3.33e-08	1: 111111 2: 011010 4: 101101	2&3: Mx = 0.2e-08 3&4: My = -0.4e-08	Twist: 0.5e^{-03} Slopes: 2: q_x = 1.2e^{-03} q_y = 0 3: q_x = 1.2e^{-04} q_y = -6.0e^{-05} 4: q_y = -6.0e^{-05}	Surface Stress: s_{xy} = 0.2	q_{3x}: 0.00012 q_{3y}: -6.0e^{-05} W_3: 1.44e^{-05}	q_{3x}: 0.00012 q_{3y}: -6.0e^{-05} W_3: 1.44e^{-05}	patchws.mdl patchws.nas patchws.mod patchws.f06

Table 2 Patch Test MAESTRO Quad Element vs. MSC/Nastran for Windows V3.0 QuadR Element Results

17.1.2 Cantilever Beam

The second test is a cantilever beam modeled with six trapezoidal (or parallelogram) shell elements. The dimensions and the material properties are given in Figure 1. Three loads are applied at the free end of the beam: a unit force in the Y direction (in the plane of the element), a unit force in the Z direction (out of plane) and a unit twisting moment. The in-plane force causes in-plane shear. The out of plane force causes shell bending. The critical part of this test is the in-plane shear. The MSC/Nastran QUAD4 element completely failed the in-plane test because of shear locking. The MAESTRO element is only 1.2% different from the theoretical value. For the out of plane bending, the errors are 2.3% and 1.6% respectively. For the twist the theoretical value is 0.0233 radians and not 0.0321 as given in [2] and MAESTRO matches this value exactly. Table 1 presents the results.

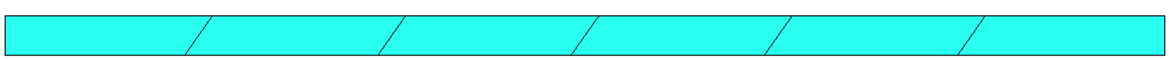


Figure 1 Straight Cantilever Beam, Length = 6.0, Height = 0.2, Depth = 0.1, E = 1.0e^{07}, v = 0.3, mesh = 6 x 1

	BCs	Loads	MAESTRO Solver	Theoretical	Relative Error	MSC/ Nastran V3.0	Input Data Files
In-Plane Shear	Clamped at one end of beam	Unit force in Y-direction at free end	v = 0.1068	v = 0.1081	1.20%	0.1068	trapz.mdl trapz1.dat trapz1.mod trapz1.f06
Out-of-Plane Shear	Clamped at one end of beam	Unit force in Z-direction at free end	w = 0.4252	w = 0.4321	1.62%	w = 0.4264	trapz.mdl trapz2.dat trapz2.mod trapz2.f06
Twist	Clamped at one end of beam	Unit twisting moment at free end	q_x = 0.00233	q_x = 0.00233	0%	q_x = .00306	trapz.mdl trapz3.dat trapz3.mod trapz3.f06

Table 1 Cantilever Beam Results

17.1.3 Curved Beam

The third test is the curved beam problem. The geometry, dimensions, material properties, and loading conditions are shown in Figure 1. The element shape in this test is not exactly rectangular and so this test includes the effect of a small irregularity in the element. Table 1 presents the results of this test, which shows that the MAESTRO QUAD4 element has about the same accuracy as the MSC QUAD4 element.

© 2014 Optimum Structural Design, Inc.

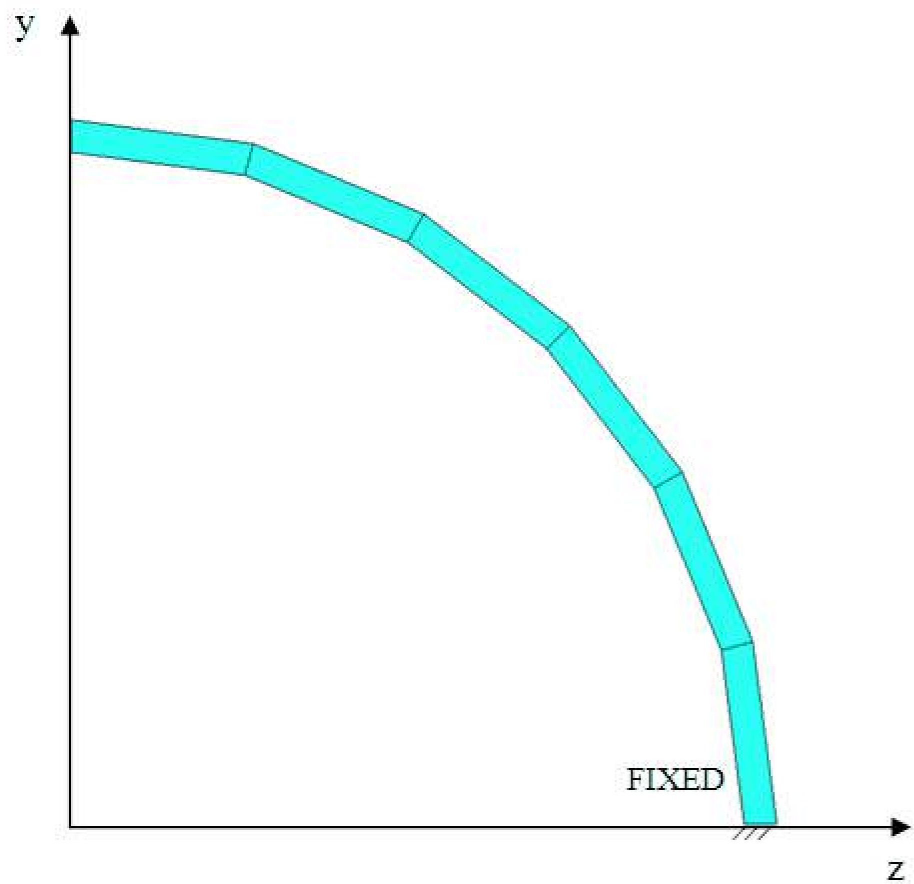


Figure 1 Curved Beam, Inner Radius = 4.12, Outer Radius = 4.12, arc = 90 degrees, thickness = 0.1, E = 1.0e^{07}, v = 0.25, mesh = 6 x 1

	BCs	Loads	MAESTRO Solver	Theoretical	MAESTRO Error	MSC Quad4 Error	Input Data Files
In Plane, Shear	Clamped at one end of the curved beam	Unit force in Y-direction at free end	v = 0.0880	v = 0.0873	0.8%	0.8%	curvedbeam.mdl curvbb.nas curvbb.mod curvbb.f06
Out of Plane, Bending	Clamped at one end of the curved beam	Unit force in X-direction at free end	u = 0.4492	u = 0.5022	10.5%	4.9%	curvedbeam.mdl curvbs.nas curvbs.mod curvbs.f06

17.1.4 Twisted Beam

The fourth test is the twisted beam problem. The size of the beam, element mesh and material properties are given in Figure 1. The purpose of this test is to study the ability of an

© 2014 Optimum Structural Design, Inc.

element to treat the coupling of in-plane and out-of-plane strain when there is a warped element mesh. In this test, the warp of each element is 7.5 degrees. As shown in Table 1, the MAESTRO QUAD4 gives results that are in good agreement with theoretical results.

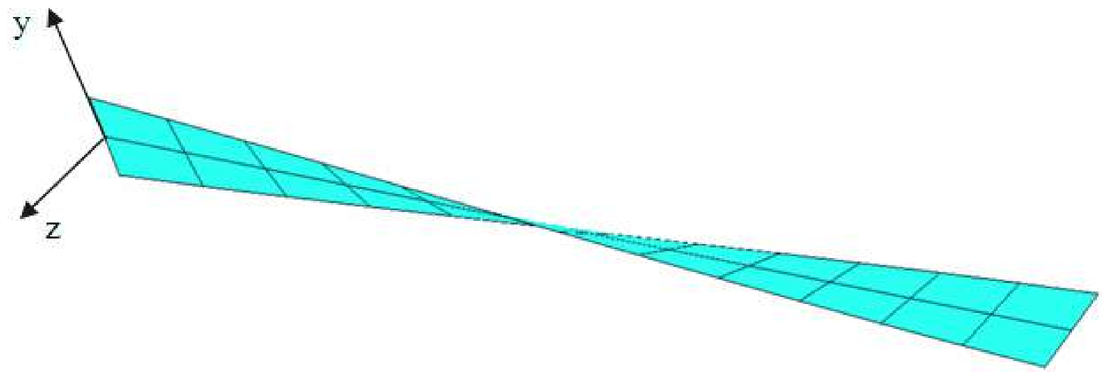


Figure 1 Twisted Beam, Length = 12.0, Width = 1.1, Depth = 0.32, Twist = 90 degrees, E = 2.9e^{07}, v = 0.22, mesh = 12 x 2

	BCs	Loads	MAESTRO Solver	Theoretical	Relative Error	MSC/Nastran	Input Data Files
Out-of-Plane Shear	Clamped at one end of the beam	Unit Force in Y-direction at free end	v = 1.728e-03	v = 1.754e-03	1.5%	v = 1.727e-03	twsb.mdl twsb1.nas twsb1.mod twsb1.f06
In-Plane Shear	Clamped at one end of the beam	Unit Force in Z-direction at free end	w = 5.382e-03	w = 5.424e-03	0.8%	w = 5.388e-03	twsb.mdl twsb2.nas twsb2.mod twsb2.f06

Table 1 Twisted Beam Results

17.1.5 Rectangular Plate Under Lateral Load

The fifth test investigates the accuracy of the plate elements bending response for the case of a rectangular plate. A lateral load is applied to a rectangular plate of a given aspect ratio. The plate is tested separately for each of the two types of loads: a uniform pressure of 1.0e^{-04} and a central concentrated load of 4.0e^{-04}. It is tested for two types of boundary conditions: simply supported edges and clamped edges, and two aspect ratios, yielding a total of eight test problems. The results are presented in Table 1, showing that the MAESTRO QUAD4 element gives quite good results. The two models are presented in Figures 1 and 2. Because of symmetry, only one quarter of the plate is modeled for each aspect ratio.

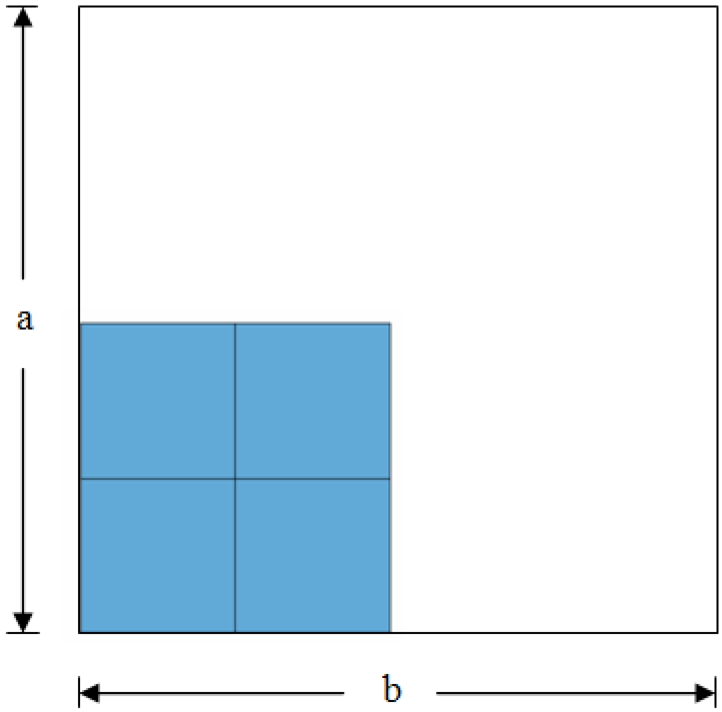


Figure 1 Rectangular Plate, Aspect Ratio 1, a = 2.0, b = 2.0, t = 0.0001, E = 1.7472e^{07}, v = 0.3

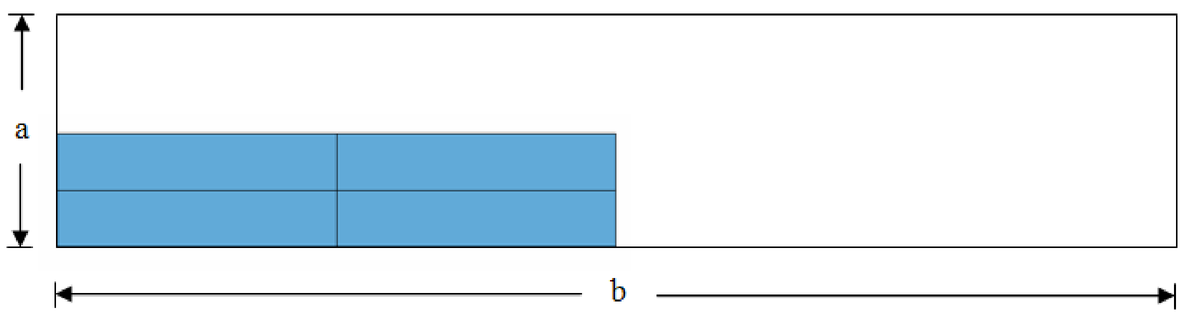


Figure 2 Rectangular Plate, Aspect Ratio 5, a = 2.0, b = 10.0, t = 0.0001, E = 1.7472e^{07}, v = 0.3

BCs	Aspect Ratio	Loads	Theoretical Deflection	MAESTRO Solver	MAESTRO Error	MSC/Nastran	MSC/Nastran Error	Input Data Files

Simply Supported All Sides	1	Pressure	4.06	4.07	0.2	4.15	2.2	sp4.mdl sp4pt.mod sp4pt.nas sp4pt.f06 sp4pres.mod sp4pre.f06 sp4pre.nas
		Point Load	11.6	12.02	3.6	12.36	6.5	
	5	Pressure	12.97	12.98	0.1	13.08	0.8	rpl4.mdl rpl4pt.mod rpl4pt.nas rpl4pt.f06 rpl4pre.mod rpl4pre.nas rpl4pre.f06
		Point Load	16.96	16.22	4.4	16.11	5.0	
Clamped All Sides	1	Pressure	1.26	1.30	3.2	1.31	4.0	spl4cl.mdl spl4clpr.mod spl4clpr.f06 spl4clpr.nas spl4clpt.mod spl4clpt.f06 spl4clpt.nas
		Point Load	5.60	5.34	4.6	5.61	0.2	
	5	Pressure	2.56	2.86	11.7	3.41	33.2	rpl4cl.mdl rpl4clpr.mod rpl4clpr.f06 rpl4clpr.nas rpl4clpt.mod rpl4clpt.nas rpl4clpt.f06
		Point Load	7.23	4.97	31.3	3.80	47.4	

Table 1 Rectangular Plate Results

© 2014 Optimum Structural Design, Inc.

17.1.6 Scordelis-Lo Roof

The sixth test is the Scordelis-Lo roof problem. The structure is loaded by its own weight. From the given conditions in Figure 1, the specific weight is 360/unit volume. This problem has the combination of in plane and out of plane loads. This is the only slightly curved shell problem in the test set. It is required to correctly distribute the body force to four grid nodes. The results of this test are listed in Table 1.

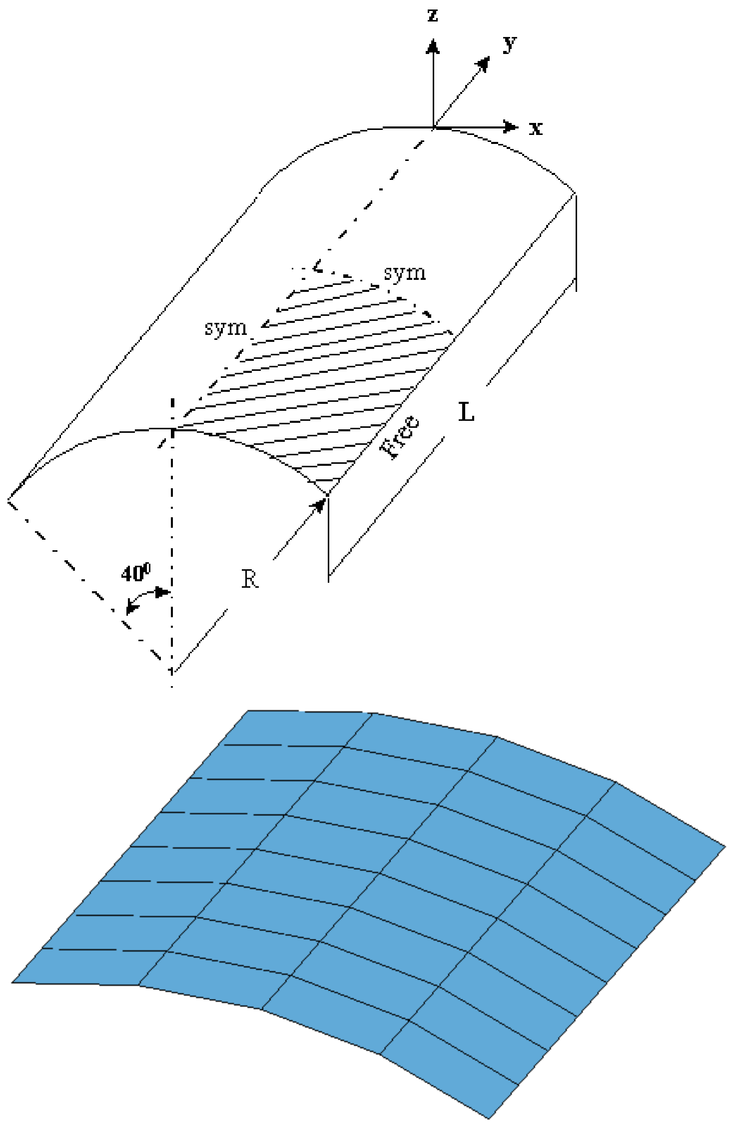


Figure 1 Scordelis-Lo Roof, Radius = 25.0, Length = 50.0, t = 0.25, E = 4.32e^{08}, v = 0, Loading = 90.0 per unit area in Z-direction

BCs	Load	Mesh Size	Theoretical Max Displacement	MAESTRO Max Displacement	MSC/ Nastran Max Displacement	ABAQUS Max Displacement	Input Data Files
Simple Support at Both Circular Ends	Self-Weight. r = 360/ Unit Volume	4 x 4	v = -0.3086	v = -0.3197	v = -0.3218	v = -0.3880	scod.mdl scod.mod scod.nas scod.f06
		8 x 8	v = -0.3086	--	--	v = -0.3226	--
		16 x 16	v = -0.3086	--	--	v = -0.3190	--

Table 1 Scordelis-Lo Roof Results

As shown in Figure 2, MAESTRO's typical coarse mesh modeling (4 elements per side, for a total of 16 elements) has an order of accuracy (3.6%) that ABAQUS only achieves with 16 elements per side, for a total of 256 elements (3.4%).

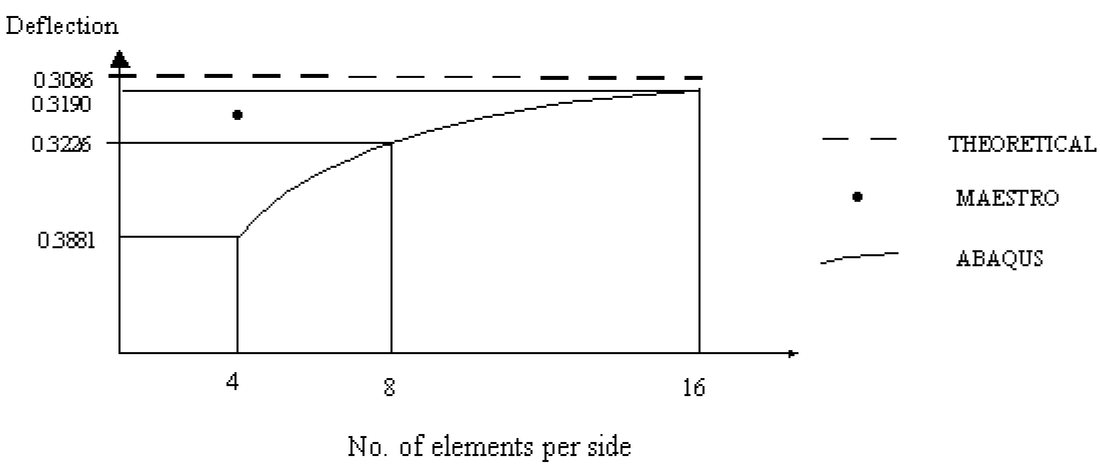


Figure 2 Comparison of MAESTRO and ABAQUS Results with the Theoretical Solution

17.1.7 Hemispherical Shell

The final test is the hemispherical shell problem. This test gives an opportunity to study the solution accuracy of a doubly-curved shell. The geometric size of the hemispherical shell and the material properties are given in Figure 1. The results are listed in the table below.

© 2014 Optimum Structural Design, Inc.

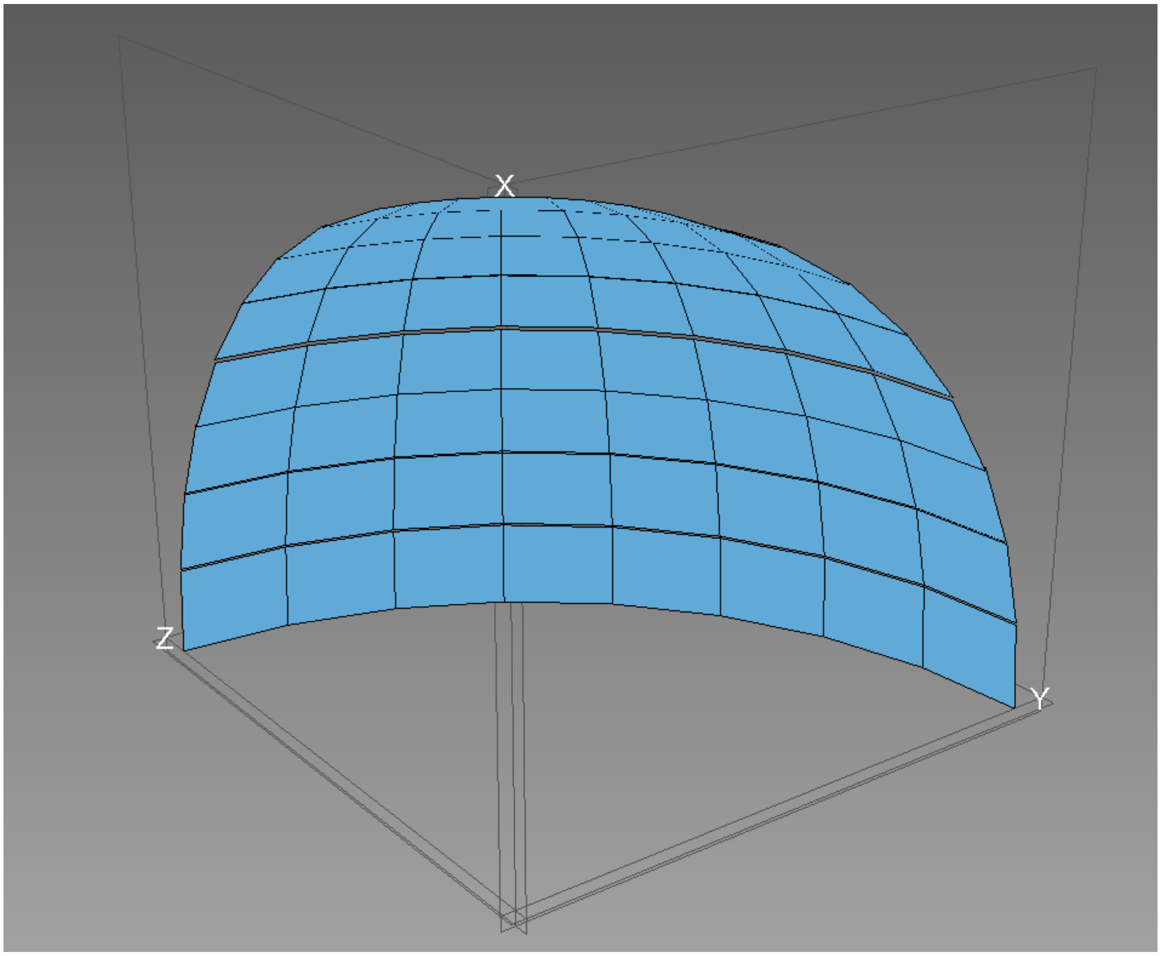


Figure 1 Sphere8 Model, Transverse Symmetry, Radius = 10.0, thickness = 0.04, E = 6.825e^{07}, v = 0.3

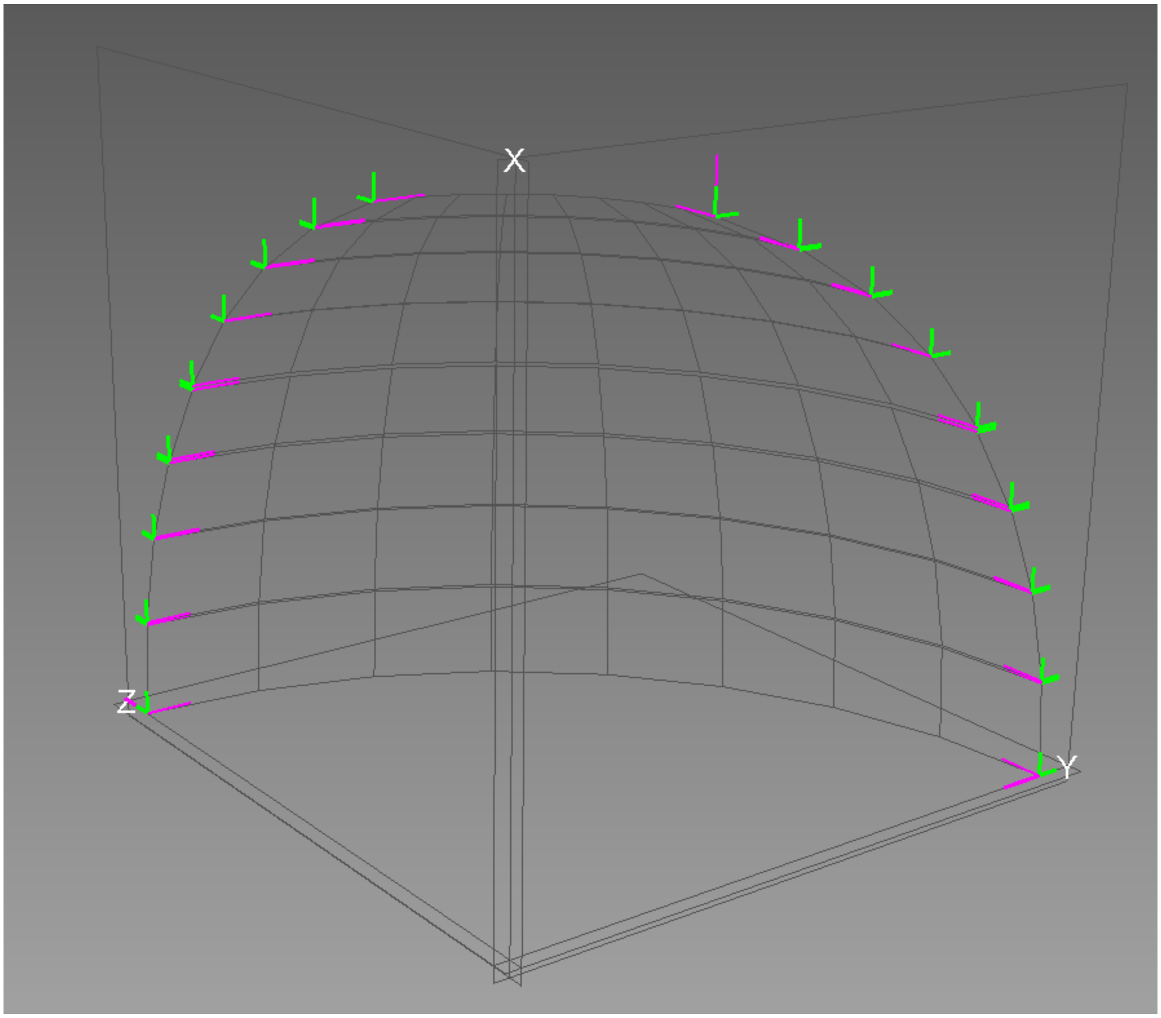


Figure 2 Sphere8 Boundary Conditions

	BCs	Loads	MAESTRO Displacement	Theoretical Displacement	% Error	Input Data Files
	Refer to Figure 2	Unit Forces in Symmetric Planes. $P_z = -1$, $P_y = 2$	v = 0.090	v = 0.094	4.3 %	sphere8.mdl sphere8a.mod sphere8a.f06 sphere8a.nas

© 2014 Optimum Structural Design, Inc.

	Refer to Figure 2	Unit Forces in Symmetric Planes. P_z = 1, P_y = -2	v = 0.090	v = 0.094	4.3%	sphere8.mdl

17.2 CROD

The rod test is a rod modeled with two solid rod elements of different cross sections. The rod is fixed at node 1. The model and node locations are presented in Figure 1 and Table 1. The rod model is applied a vertical compressive force of 1,000 psi at node 3.

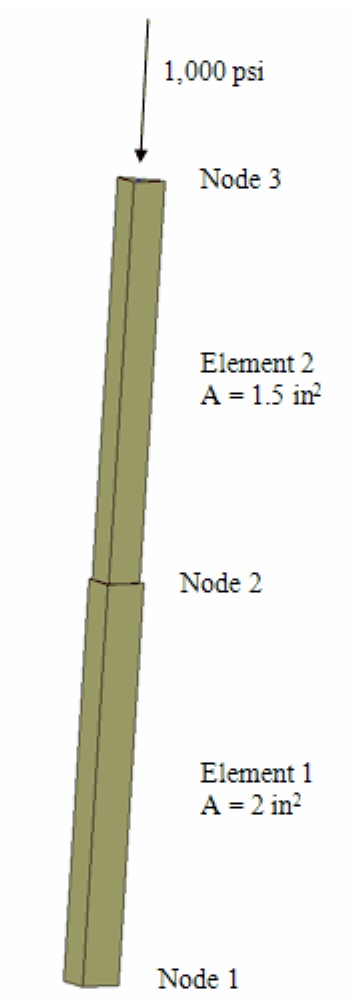


Figure 1 Rod Test, E = 1.0e07 psi

Node	X	Y
1	0	0
2	0	10
3	0	20

The MAESTRO calculated axial stresses recovered in each element match exactly to the theoretical values.

Element 1 axial stress = -500.00 psi

Element 2 axial stress = -666.67 psi

Model: rodtest.mdl

17.3 RBE3

RBE3 implementation was compared to NEi Nastran. The results of the two multipoint constraint coefficients are provided below. The MAESTRO model, RBE3.mdl, and the related NEi Nastran files can be found in the Program Files\MAESTRO\Models and Samples \Verification Models\RBE3 directory.

© 2014 Optimum Structural Design, Inc.

M U L T I P O I N T   C O N S T R A I N T   D E F I N I T I O N

DEPENDENT DEGREES OF FREEDOM			INDEPENDENT DEGREES OF FREEDOM					
GRID ID	COMPONENT NUMBER	COEFFICIENT	GRID ID	COMPONENT NUMBER	COEFFICIENT	GRID ID	COMPONENT NUMBER	COEFFICIENT
17	1	1.000E+00	9	1	-2.062E-01	9	3	3.088E-01
			11	1	-2.062E-01	13	1	-1.753E-01
			13	3	-3.088E-01	15	1	-2.062E-01
			16	1	-2.062E-01			
17	2	1.000E+00	9	2	-1.949E-01	11	1	-1.177E-02
			11	2	-2.097E-01	11	3	3.860E-01
			13	2	-1.761E-01	15	2	-2.097E-01
			16	1	1.177E-02	16	2	-2.097E-01
			16	3	-3.860E-01			
17	3	1.000E+00	9	3	-2.000E-01	11	3	-2.000E-01
			13	3	-2.000E-01	15	3	-2.000E-01
			16	3	-2.000E-01			
17	4	1.000E+00	9	2	-1.632E-04	11	1	3.796E-04
			11	2	3.113E-04	11	3	-1.245E-02
			13	2	-7.706E-04	15	2	3.113E-04
			16	1	-3.796E-04	16	2	3.113E-04
			16	3	1.245E-02			
17	5	1.000E+00	9	1	-1.992E-04	9	3	9.960E-03
			11	1	-1.992E-04	13	1	7.968E-04
			13	3	-9.960E-03	15	1	-1.992E-04
			16	1	-1.992E-04			
17	6	1.000E+00	9	2	-6.103E-03	11	1	4.890E-03
			11	2	9.491E-06	11	3	-3.796E-04
			13	2	6.074E-03	15	2	9.491E-06
			16	1	-4.890E-03	16	2	9.491E-06
			16	3	3.796E-04			

Figure 1 MPC Definition from NEi Nastrain

© 2014 Optimum Structural Design, Inc.

```
Begin to assemble sparse matrix...
 S-ID DOF       M-ID DOF         Coefficient==>RBE3's MPC equation
   17   1          9  1           -0.206175
                   9  3            0.308765
                  11  1           -0.206175
                  13  1           -0.175299
                  13  3           -0.308765
                  15  1           -0.206175
                  16  1           -0.206175
   17   2          9  2            -0.19494
                  11  1          -0.0117685
                  11  2            -0.20965
                  11  3            0.386007
                  13  2            -0.17611
                  15  2            -0.20965
                  16  1           0.0117685
                  16  2            -0.20965
                  16  3           -0.386007
   17   4          9  2        -0.000163241
                  11  1         0.000379629
                  11  2         0.000311296
                  11  3          -0.0124518
                  13  2        -0.000770647
                  15  2         0.000311296
                  16  1        -0.000379629
                  16  2         0.000311296
                  16  3           0.0124518
   17   5          9  1        -0.000199203
                   9  3          0.00996016
                  11  1        -0.000199203
                  13  1         0.000796813
                  13  3         -0.00996016
                  15  1        -0.000199203
                  16  1        -0.000199203
   17   6          9  2         -0.00610254
                  11  1          0.00488962
                  11  2        9.49073e-006
                  11  3        -0.000379629
                  13  2          0.00607407
                  15  2        9.49073e-006
                  16  1         -0.00488962
                  16  2        9.49073e-006
                  16  3         0.000379629
```

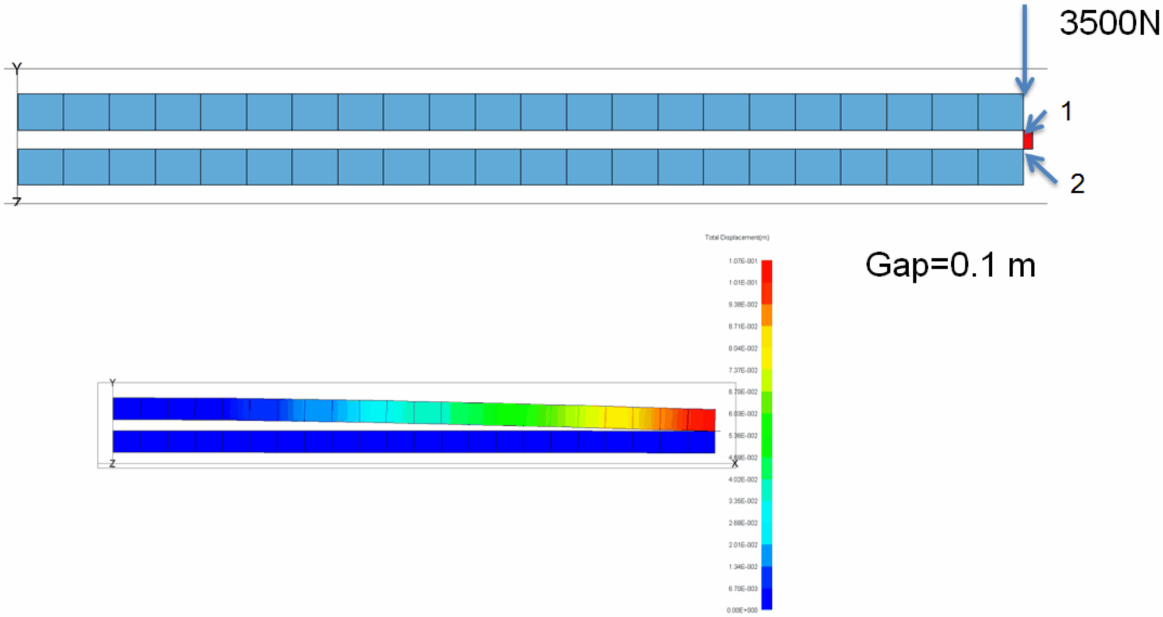
Figure 2 MPC Definition from MAESTRO

17.4 Gap Element

The sample models shown below can be found under C:\Program Data\MAESTRO\Models and Samples\Gap

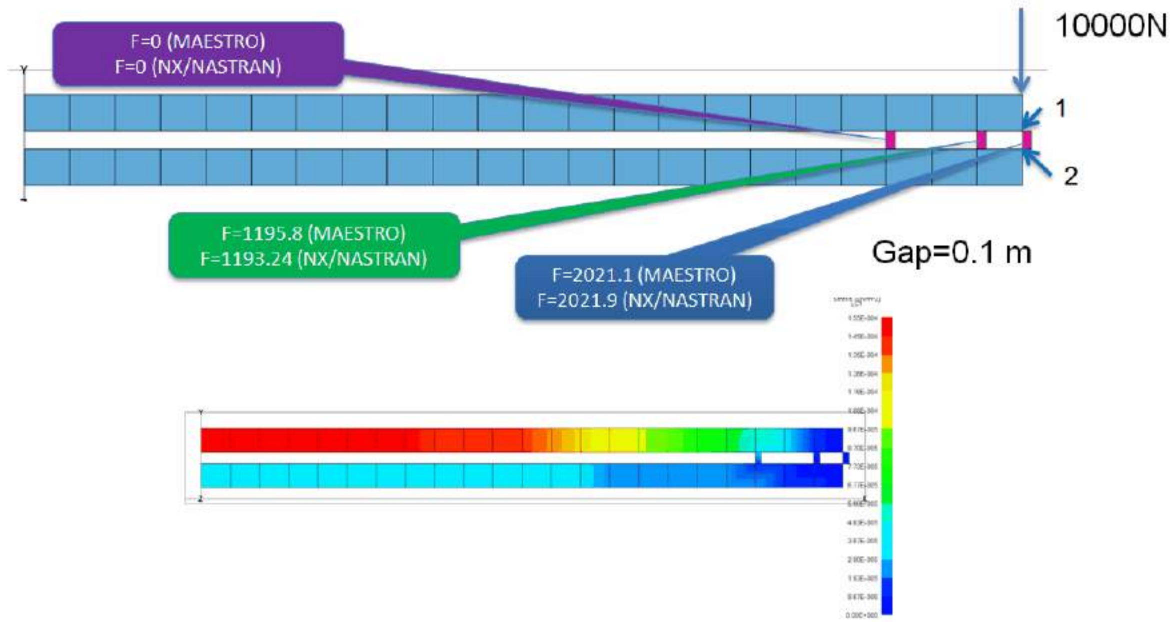
The first example compares two beams separated by 0.1m with a GAP element connecting the two ends and single vertical force applied at the top beam. The results of the displacement of the two nodes is compared to results from Nastran.

© 2014 Optimum Structural Design, Inc.



Node	NX/Nastran	MAESTRO
1	-0.10723	-0.107188
2	-0.0055165	-0.005566

The second example shows the same two beams and a single vertical force, but now with 3 gap elements modeled along the beam. The force and nodal displacement results are compared between MAESTRO and Nastran.



Node	NX/Nastran	MAESTRO
1	-0.22117	-0221132
2	-0.10096	-0.101008

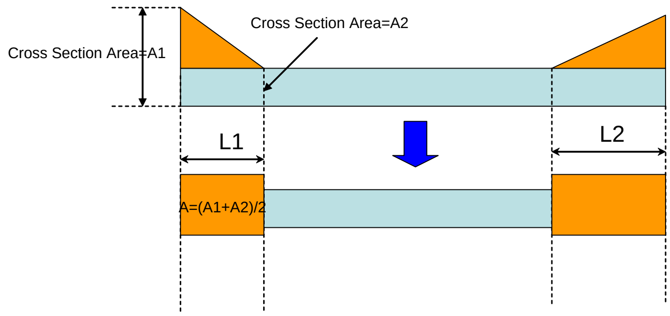
17.5 Bracket

Bracket Element

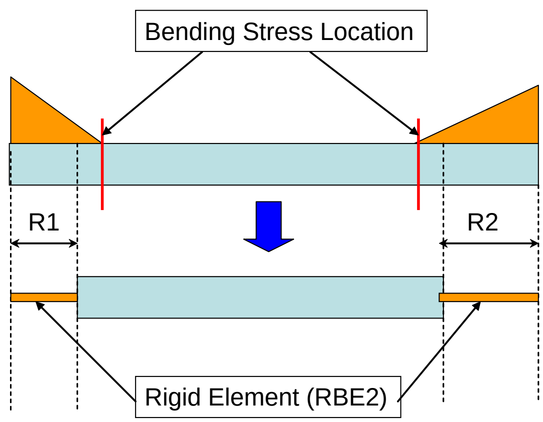
1. Background

MAESTRO bracket element is used for a simplified coarse mesh analysis. It can be attached to an end of a beam element to get stress reduction. The technique involves separate modeling of the axial and bending stiffness. The axial stiffness is modeled by a super-element method. It is assembled with tapered axial elements for the bracketed portion and a standard axial element for the rest of the beam. Internal nodes are eliminated by condensation of the stiffness matrix.

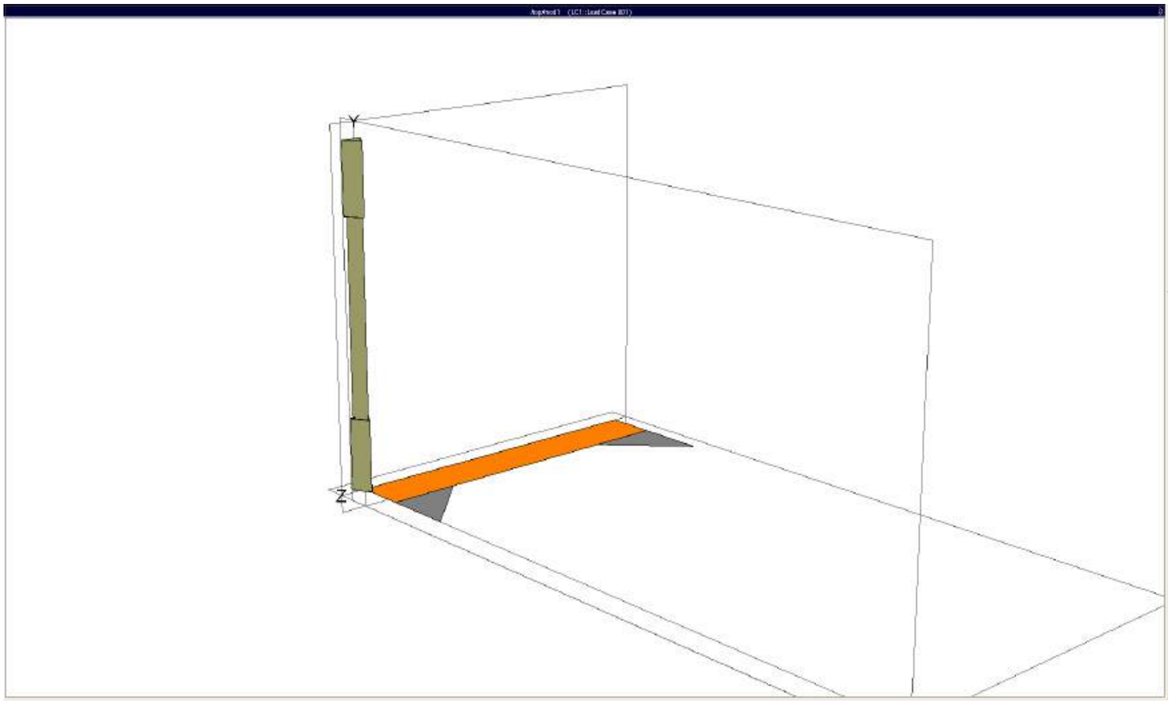
© 2014 Optimum Structural Design, Inc.



The bending stiffness of the assembly is obtained by matrix transformation. A portion of the bracket is modeled as a rigid link element, and the remaining part of the beam is modeled as a regular beam element.

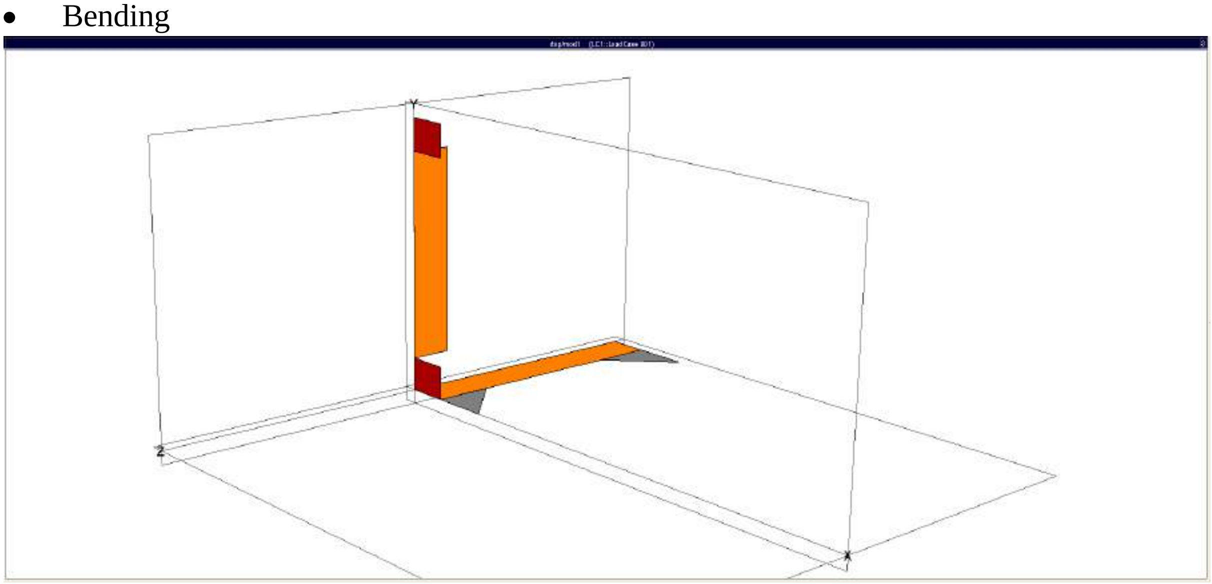


2. Verification
   - Axial



Model	Beam with Brackets	3 Rods
Beam_Brk_Tension.mdl	Dz=-1.6413e-006	Dy=-1.6413e-006

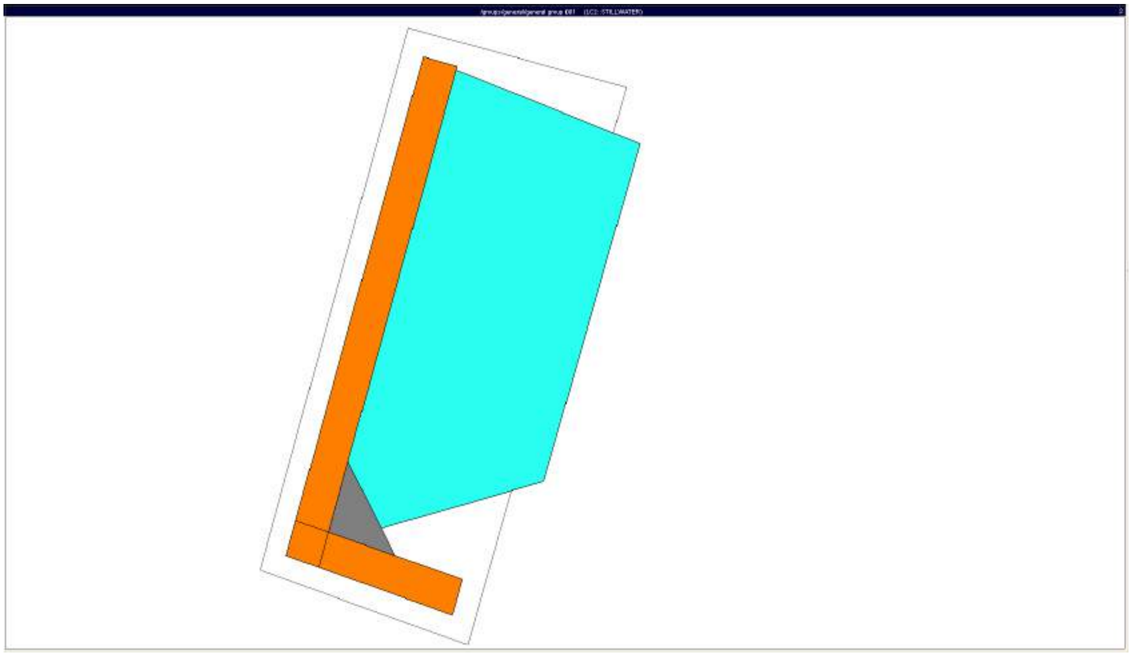
   - Bending



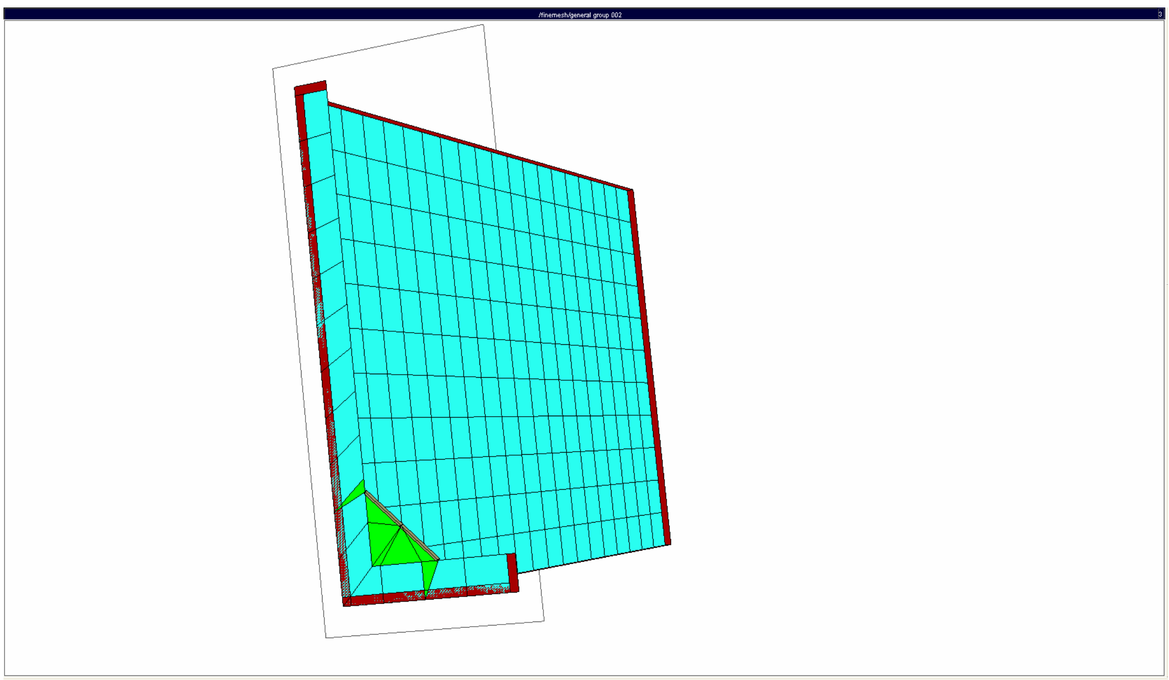
Model	Beam with Brackets	2 RBE2+ 1 Beam
Beam_Brk_Bending.mdl	Dx=7.08468e-005	Dz=7.08469e-005

© 2014 Optimum Structural Design, Inc.

3. Refinement



Coarse Mesh

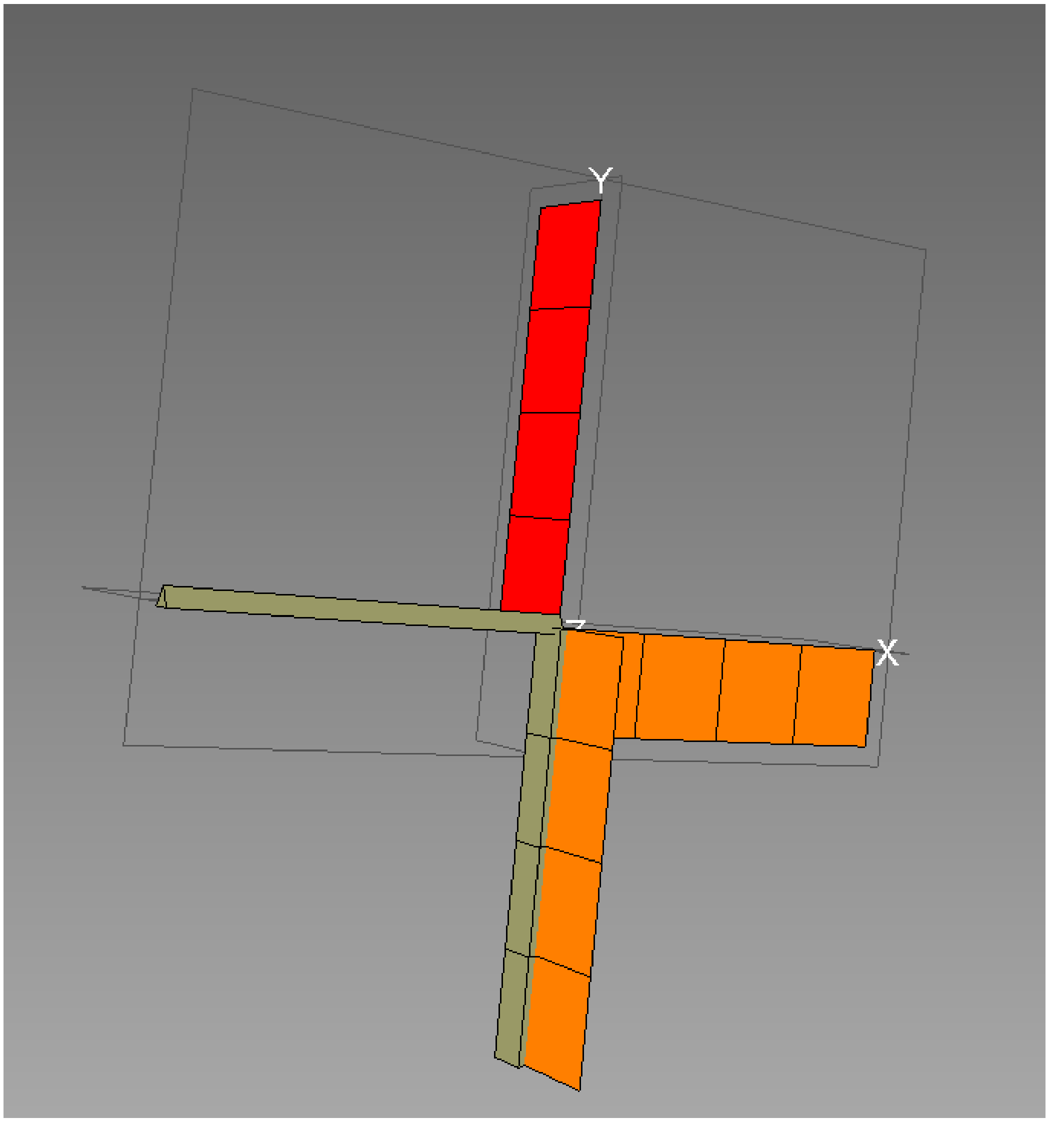


Top-down auto-refinement

17.6 Second Flange

The following second flange examples demonstrate the three methods for modeling a second flange: the second flange strake, defining a second flange on a beam, and creating a T-beam with a rod element as the second flange.

Example 1



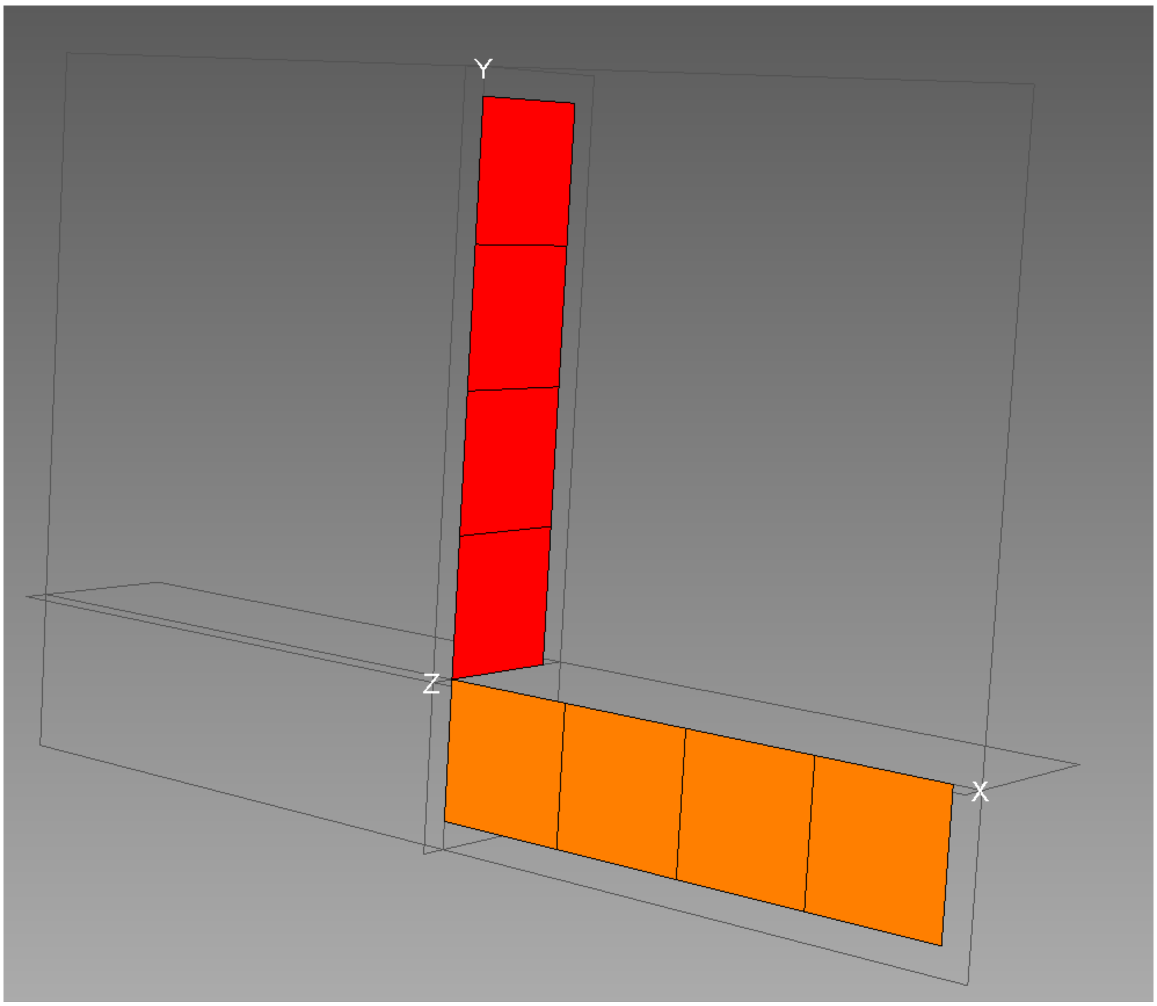
This example compares the displacement of a beam with a second flange, a second flange strake, a beam with a rod as the second flange, and a rod with equivalent cross sectional area all subjected to a tensile force of 1,000 Newtons. The intersection of all 4 elements is fixed and the opposite end of the beams, frame, and rod are constrained such that only axial motion is allowed. This model, SecondFlangeTension.mdl, can be found in the Program Files\MAESTRO\Models and Samples\Verification Models\Second Flange directory.

Element	Force	Displacement

© 2014 Optimum Structural Design, Inc.

Beam with second flange defined	F_x = 1000 N	D_x = 1.08948e^{-6}
Second flange strake	F_y = 1000 N	D_y = 1.08948e^{-6}
Rod with equivalent cross sectional area	F_x = -1000 N	D_x = -1.08948e^{-6}
Beam with rod defined as second flange	F_y = -1000 N	Dy = - 1.08948e^{-6}

Example 2



This example compares the deformation of a beam with a second flange defined and a second flange strake under a lateral load. The ends of both beams are fixed and a later load of 1000 N is applied at each beam's center. This model, SecondFlangeLateral.mdl, can be found in the Program Files\MAESTRO\Models and Samples\Verification Models\Second Flange directory.

Element	**Force**	**Displacement**
Beam with second flange defined	$F_y = 1000$ N	$D_x = 5.70311e^{-7}$
Second flange strake	$F_z = 1000$ N	$D_x = 5.70311e^{-7}$

17.7 Added Mass

Panel Method for Added Mass

1. Basic Theory

The added mass can be expressed as

$$m^k = \oiint_S \varphi^k n^k ds \qquad k = 1(Surge), 2(Sway), 3(Heave) \quad (1)$$

Where $\vec{n} = (n^1, n^2, n^3)$ is the unit normal vector on the body surface

φ^k is the solution of the following problem

$$\nabla^2 \varphi^k = 0 \quad \text{in the field} \quad (2)$$

$$grad\varphi^k \cdot \vec{n} = \frac{\partial \varphi^k}{\partial n} = n^k \quad (3)$$

The solution to the Laplace equation (2) is obtained by covering the body surface with local source strengths $\sigma(\xi, \eta, \zeta)$ at point $q(\xi, \eta, \zeta)$,

Thus, the velocity potential can be expressed as

$$\varphi^k(\vec{p}) = \varphi^k(x, y, z) = \iint_S \sigma^k(\vec{q}) G^k(\vec{p}, \vec{q}) ds = \iint_S \sigma^k(\xi, \eta, \zeta) \cdot G^k(x, y, z, \xi, \eta, \zeta) ds \quad (3)$$

in which $$G(\vec{p},\vec{q}) = G(x,y,z,\xi,\eta,\zeta) = \frac{-1}{4\pi} \cdot \frac{1}{\left|\vec{p}(x,y,z) - \vec{q}(\xi,\eta,\zeta)\right|}$$

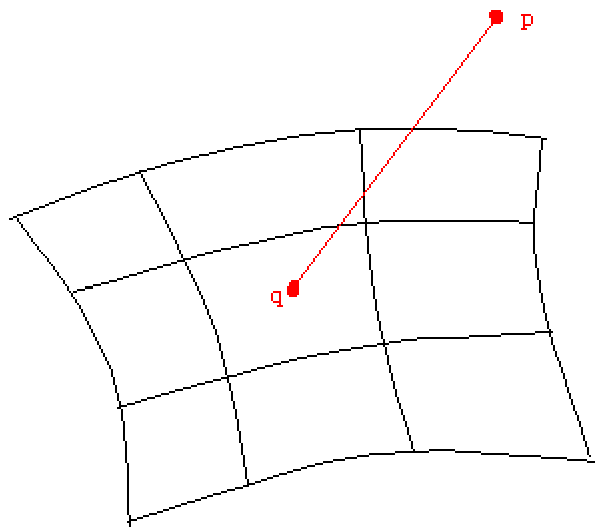


Fig 1

According to boundary condition(3), the normal velocity of the fluid at the surface of the body can be solved by

$$\frac{\partial \varphi^k(\vec{p})}{\partial n} = -\frac{1}{2}\sigma^k(\vec{p}) + \iint_s \sigma^k(\vec{q})\vec{n}(\vec{p}) \bullet \nabla G^k(\vec{p},\vec{q})ds = n^k \qquad k = 1,2,3$$
(4)

2. Panel Method

The problem to determine the continuous function $\sigma(x,y,z)$ is replaced by the problem to determine a finite number N of value $\sigma_i (i = 1,2,3,4,\cdots N)$. The integer equation become

© 2014 Optimum Structural Design, Inc.

$$\frac{\partial \varphi_i^k}{\partial n_i} = -\frac{1}{2}\sigma_i^k + \sum_{\substack{j=1 \\ j\neq i}}^{N} \sigma_j^k \iint_{S_j} \vec{n}_i \bullet \nabla G^k{}_{ij} ds \quad \begin{array}{l} i = 1,2,3,\cdots N \\ j = 1,2,3,\cdots N \\ k = 1,2,3 \end{array} \tag{5}$$

Whereas this is a finite number of equations, only in a finite number of points this boundary condition can be satisfied.

After σ_i^k are determined , the velocity potential can be obtained by

$$\varphi_i^k = \sum_{j=1}^{N} \sigma_j^k \iint_{s_j} G^k (x_i, y_i, z_i, \xi, \eta, \zeta) ds \quad \begin{array}{l} i = 1,2,3,\cdots N \\ k = 1,2,3 \end{array} \tag{6}$$

The each panel added mass can be get by

$$m_i^k = \varphi_i^k n^k{}_i A_i \qquad \begin{array}{l} i = 1,2,3,\cdots N \\ k = 1,2,3 \end{array}$$

The added mass of the body is

$$m^k = \sum_{i=1}^{N} m_i^k$$

3. Application

3.1 Sphere no free surface

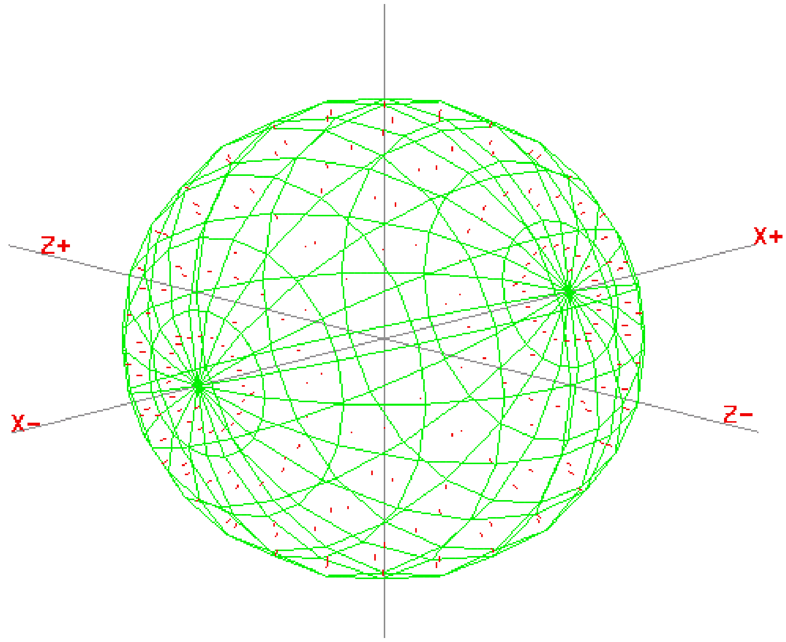


Figure 2

The sphere can be expressed by panels (See Figure 2 above). The theoretical value[1] of the added mass is $\frac{2}{3}\pi \cdot \rho \cdot r^3$ (The surge,sway and heave are same).

The ρ and r are the density of water and the radius of sphere respectively.

The numeric results are showed in the Table 1 below.

Table 1 $\rho =$ 1025(kg/m3) $r = 1.0(m)$

Panel Number	Theoretical			Numeric		
	Surge	Sway	Heave	Surge	Sway	Heave
50	2146.7594	2146.7594	2146.7594	2117.4725	2270.3245	2270.3245
800				2212.6462	2235.6909	2235.6911
1800				2195.8930	2209.7752	2209.7754

3.2 Half Sphere with free surface

© 2014 Optimum Structural Design, Inc.

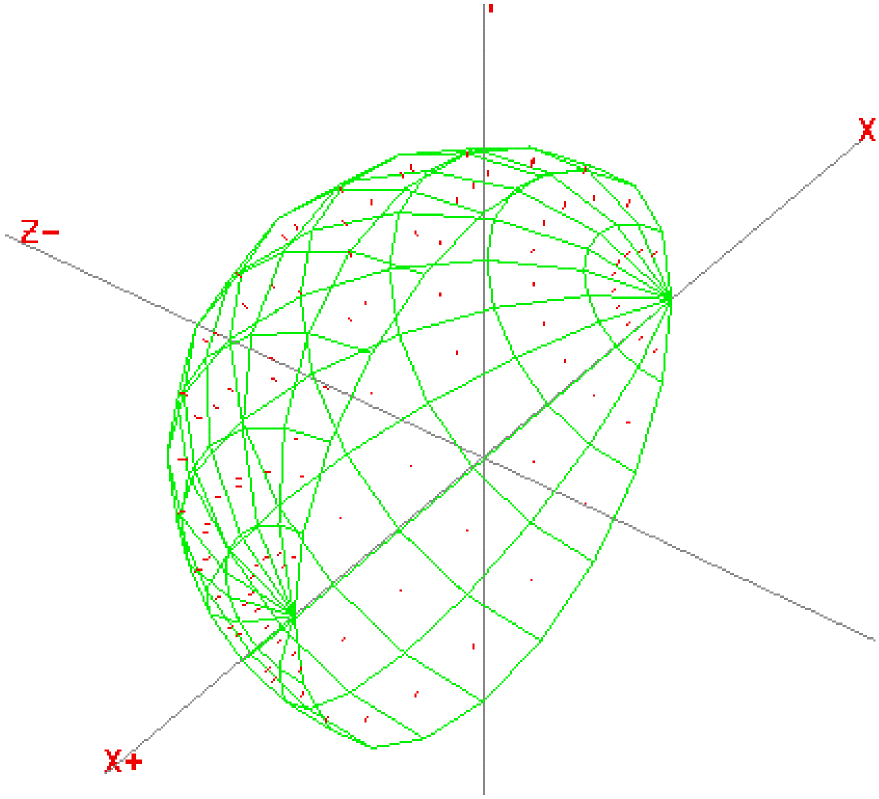


Figure 3

The half sphere can be expressed by panels(See Figure 3 above).

The Landweber value[2] of the added mass is $\frac{1}{3}\pi \cdot \rho \cdot r^3$ for heave and

$(\frac{4}{\pi}-1) \cdot \frac{2}{3}\pi \cdot \rho \cdot r^3$ for surge and sway.

The ρ and r are the density of water and the radius of sphere respectively.

The numeric results are showed in the Table 2 below.

Table 2 $\rho =$ 1025(kg/m3) $r = 1.0(m)$

Panel Number	Theoretical			Numeric		
	Surge	Sway	Heave	Surge	Sway	Heave
100	586.5784	586.5784	1073.3775	622.825	646.769	1144.62
462				609	617	1115

1458				600	605	1099

3.3 Quad Sphere with free surface

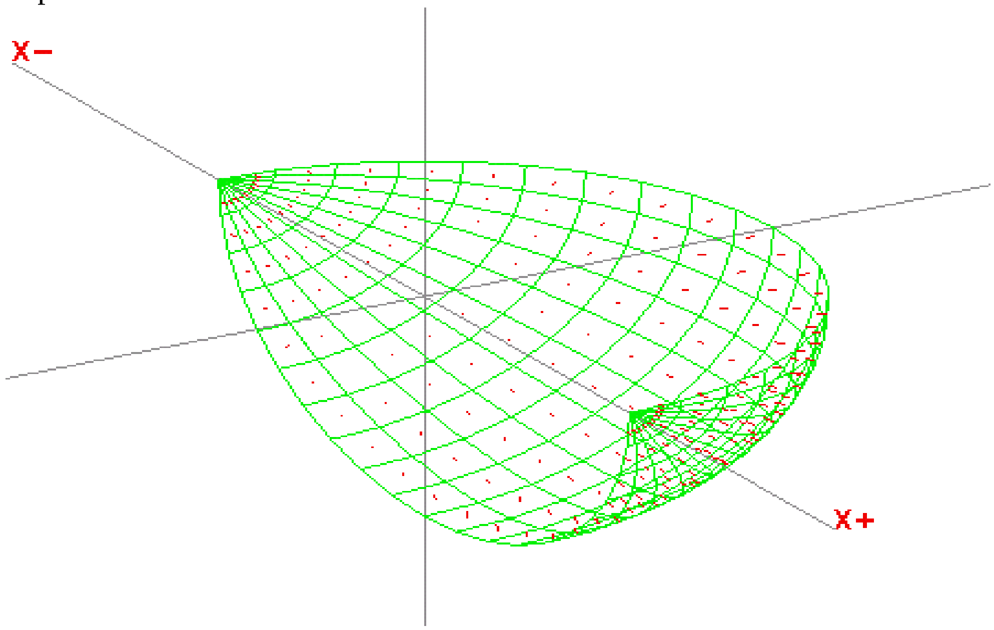


Figure 4

The quad sphere can be expressed by panels(See Figure 4 above).

The Landweber value[2] of the added mass is $\frac{1}{6}\pi \cdot \rho \cdot r^3$ for heave and $(\frac{4}{\pi} - 1) \cdot \frac{1}{3}\pi \cdot \rho \cdot r^3$ for surge and sway.

The ρ and r are the density of water and the radius of sphere respectively.

The numeric results are showed in the Table 3 below.

Tab 3 $\rho =$ 1025(kg/m3) $r = 1.0(m)$

Panel Number	Theoretical			Numeric		
	Surge	Sway	Heave	Surge	Sway	Heave
50	293.2892	293.2892	536.6887	311.3752	323.3339	572.2338
200				305.2384	309.8359	558.7456

© 2014 Optimum Structural Design, Inc.

3.4 Cylinder no free surface

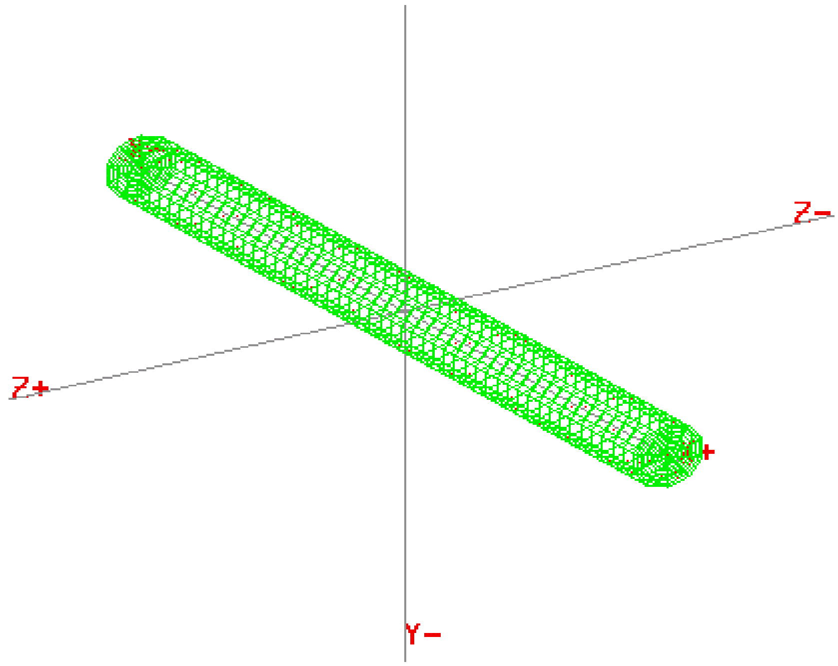


Figure 5

The Cylinder can be expressed by panels(See Figure 5 above).

The theoretical value[1] of the added mass is $\pi \cdot \rho \cdot r^2 \cdot L$ (for sway and heave).

The ρ , r and L are the density of water , the radius of cylinder and the length of cylinder respectively.

The numeric results are showed in the Table 4 below.

Table 4 $\rho =$ 1025(kg/m3) $r = 4.2(m)$ $L = 96(m)$

Panel Number	Theoretical		Numerical	
	Sway	Heave	Sway	Heave
200	5453101.0369	5453101.0369	5630603.9968	5630603.5989
300			5599916.6043	5599917.0003

3.5 Half Cylinder with free surface

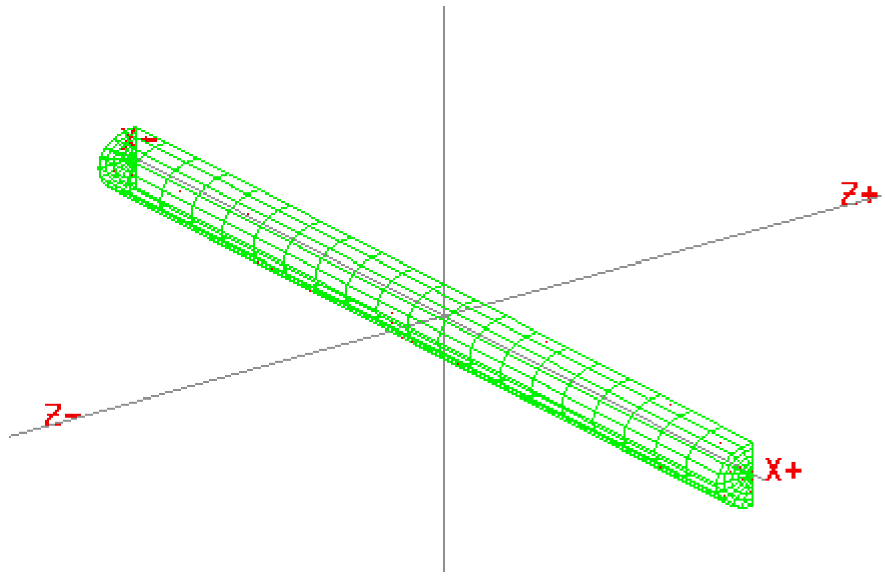


Figure 6

The half cylinder can be expressed by panels(above Fig.6).

The theoretical value[3] of the added mass is $\frac{\pi}{2}\cdot\rho\cdot r^2\cdot L$ (for heave), $\frac{2}{\pi}\cdot\rho\cdot r^2\cdot L$ (for sway) and $\frac{4}{3}k\rho\pi\cdot r^3$ (for surge $k=0.2$).

The ρ , r and L are the density of water , the radius of cylinder and the length of cylinder respectively.

The numeric results are showed in the Table 5 below.

Tab 5 $\rho=$ 1025(kg/m3) $r=4.2(m)$ $L=96(m)$

Panel Number	Theoretical			Numeric		
	Surge	Sway	Heave	Surge	Sway	Heave
300	63619.5121	1105029.3409	2726550.5184	68414.6968	1173607.3222	2725225.6351
150				72652.9161	1252297.6302	2802519.8030

3.6 Quad Cylinder with free surface

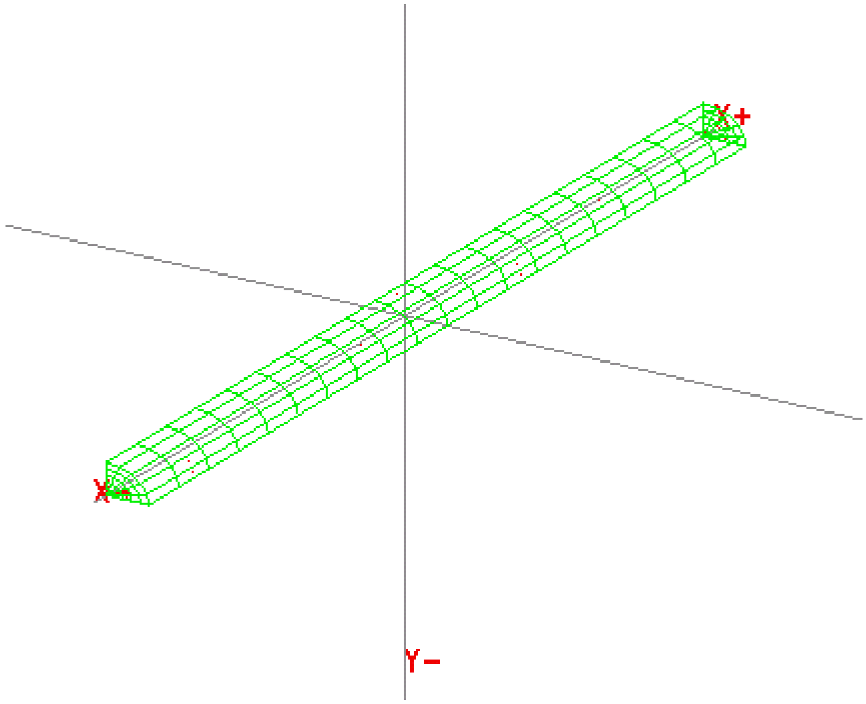


Figure 7

The quad cylinder can be expressed by panels (See Figure 7 above).

The theoretical value[3] of the added mass is $\frac{\pi}{4}\cdot\rho\cdot r^2\cdot L$ (for heave), $\frac{1}{\pi}\cdot\rho\cdot r^2\cdot L$ (for sway) and $\frac{2}{3}k\rho\pi\cdot r^3$ (for surge $k=0.2$).

The ρ , r and L are the density of water , the radius of cylinder and the length of cylinder respectively.

The numeric results are showed in the Table 6 below.

Tab 6 $\rho =$ 1025(kg/m3) $r = 4.2(m)$ $L = 96(m)$

Panel Number	Theoretical			Numeric		
	Surge	Sway	Heave	Surge	Sway	Heave
100	31809.7560	552514.6704	1363275.2592	36799.7577	590738.4908	1369071.0764
150				34201.8993	587695.3912	1361613.3078

3.7 Cube no free surface

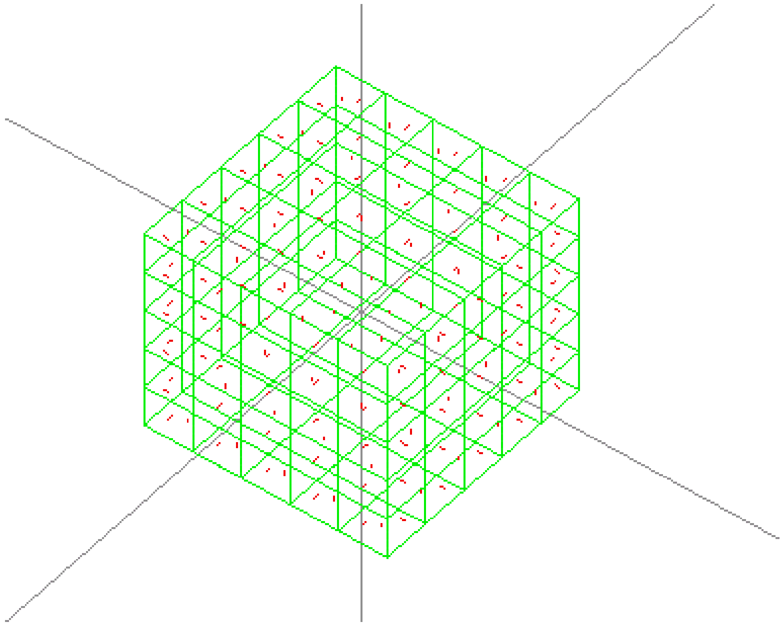


Figure 8

The cube can be expressed by panels(See Figure 8 above).

The theoretical value[1] of the added mass is $0.7 \cdot \rho \cdot a^3$ (The surge,sway and heave are same).

The ρ and a are the density of water and the length of cube respectively.

The numeric results are showed in the Table 7.

Tab 7 $\rho =$ 1025(kg/m3) $a = 1.0(m)$

Panel Number	Theoretical			Numeric		
	Surge	Sway	Heave	Surge	Sway	Heave
54	717.5000	717.5000	717.5000	726.0585	726.0585	726.0585

References

1. Sarpkaya, T., Isaacson, M., Mechanics of Wave Forces on Offshore Structures, Van Nostrand Reinhold, New York, 1981
2. L.Landweber and Matilde Macagno,Added Mass of a Rigid Prolate Spheroid Oscillating Horizontally in a Free Surface, JSR, Vol 3, Number 4,March, 1960

© 2014 Optimum Structural Design, Inc.

3. L.Landweber and Matilde Macagno, Added Mass of Two-Dimensional Form Oscillating in a Free Surface, JSR, Vol 1, Number 3, Nov. , 1957

17.8 Load Balance

This section provides verification data for MAESTRO two methods of balancing a model:

Hydrostatic Balance

Primitive Shapes Balance

S175 Hull Form Balance

Inertia Relief Balance

Hydrostatic Balance Validation

To validate MAESTRO's hydrostatic balance, a simple box-shaped vessel 100 m long, 20 m beam and 12 m depth is built. The Modeler file is called BalanceTest, located in ".../MAESTRO/Models and Samples/Verification Models/Hydrostatic Balance" folder. For simplicity there is no wave, and the mass (9.0E6 kg) gives a mean draft of 4.5 m. The product ρg to convert from a volume to a force is taken as 10,000 N/m^3. The mass is concentrated at the ¾ length position (X = 75 m) and at a height (KG) of 4 m, which causes a trim of -9.0903 degrees. In this position the stern is out of the water and the bow is immersed to exactly the full depth (12 m). The center of gravity and the center of buoyancy are co-located at the same point (x=75, y=4, z=10).

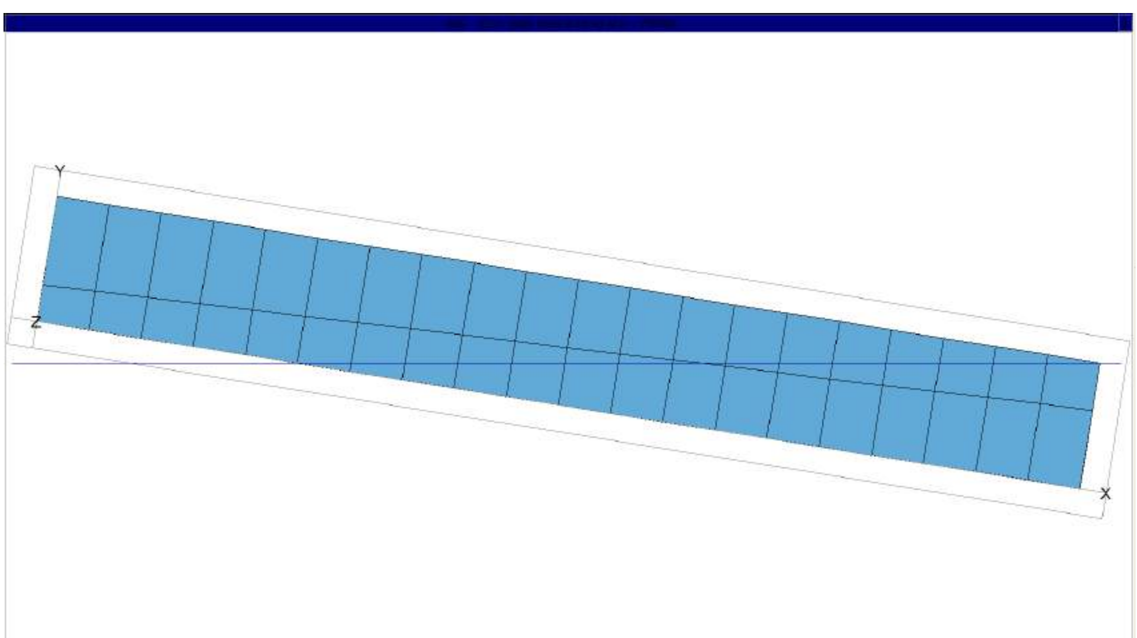


Figure 1

As shown in Figure 2 the model is very coarse mesh – much coarser than a normal ship model. All strakes are tapered, and strakes on the port side are tapered oppositely than those on the starboard.

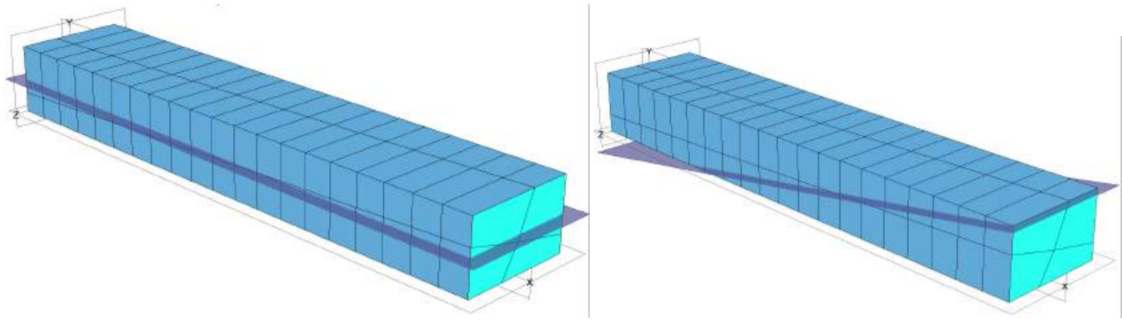
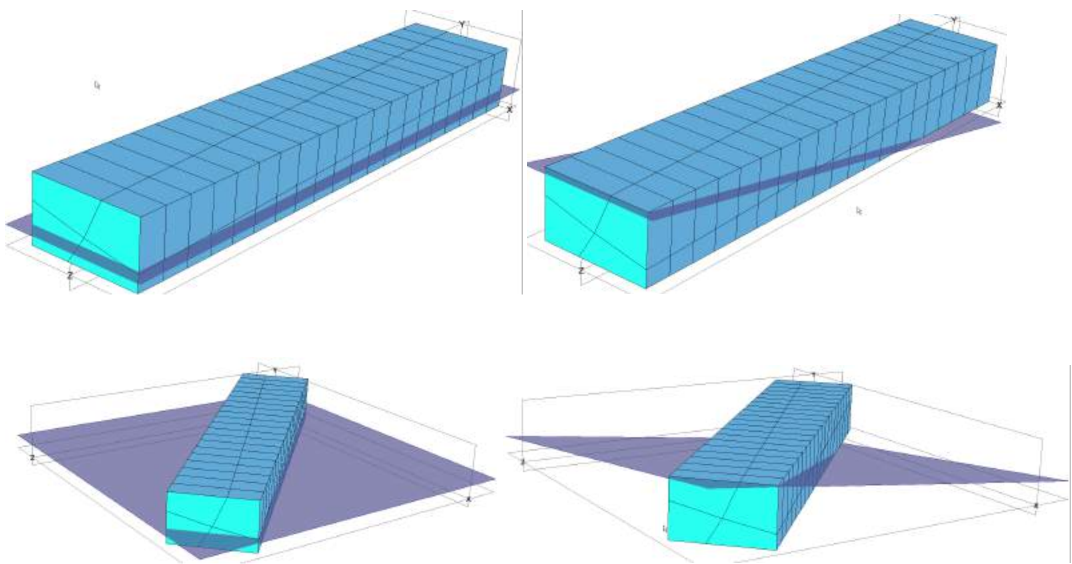


Figure 2

The initial position of the model is horizontal, at a uniform draft of 4.5 m. This corresponds to the model's displacement force of 90.E6 N, but it is incorrect since there is as yet no trim. In this position the LCF is at amidships. As the model rotates about the initial LCF the stern rises out of the water, and the model must sink a total of 1.5 m in order to maintain a vertical force balance. In MAESTRO terminology this sinkage is a negative "emergence".

Two other models, BalanceTest_Trans.mdl and BalanceTest_rot45.mdl, which were rotated 90 degrees and 45 degrees from model BalanceTest.mdl, were also tested. The result is shown in table 1.



Model	Item	MAESTRO	Theory (exact)

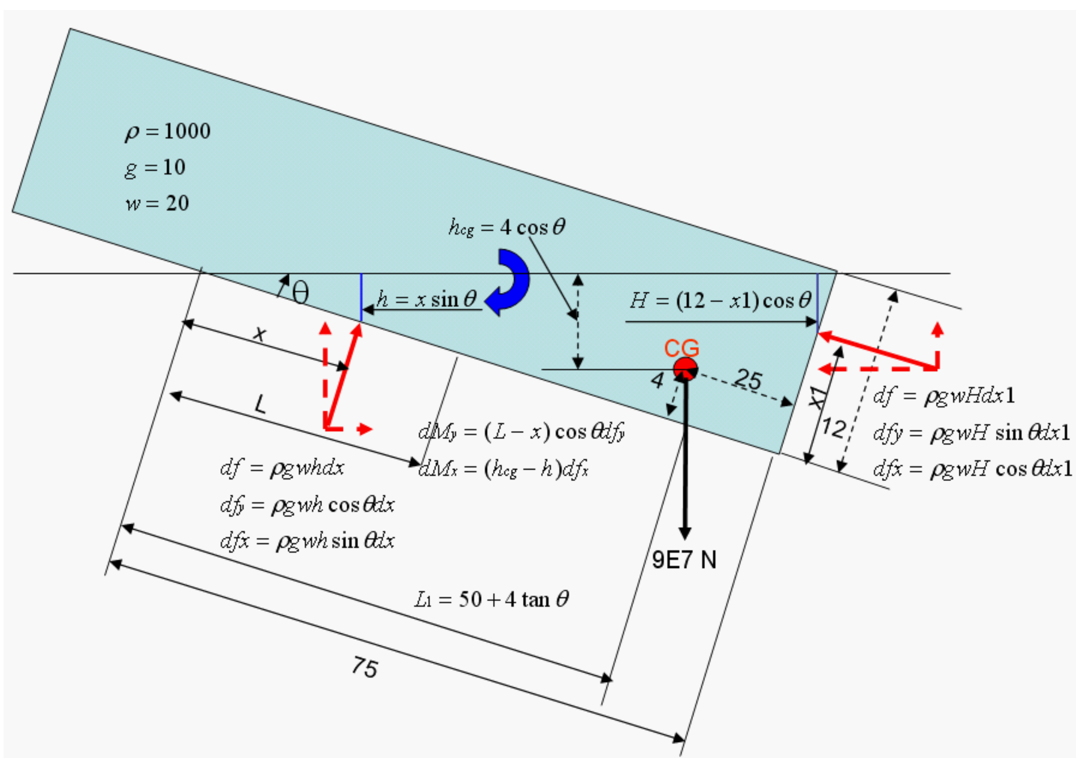
BalanceTest.mdl	Emergence (m)	-1.500	-1.500
	Heel (deg.)	0	0
	Trim (deg.)	-9.09	-9.0903
BalanceTest_Trans.mdl	Emergence (m)	-1.500	-1.500
	Heel (deg.)	-9.09	-9.0903
	Trim (deg.)	0	0
BalanceTest_Rot45.mdl	Emergence (m)	-1.500	-1.500
	Heel (deg.)	-6.455	Visual
	Trim (deg.)	-6.414	Visual

Table 1

Exact solution derivation in the global (earth) coordinate system

To further validate MAESTRO's balance tool, an exact solution of BalanceTest is presented below. Since the model is hydrostatic balanced, the following equilibrium or balance conditions in earth coordinate system should be fulfilled,

$$\begin{cases} \sum M_z(CG) = 0 \\ \sum F_x(CG) = 0 \\ \sum F_y(CG) = 0 \end{cases}$$



Free Body Diagram

Vertical Force:

$$F_{y1} = \int_0^{75} \rho g w h dx \cos\theta = \int_0^{75} \rho g w x \sin\theta \cos\theta dx = \rho g w \sin\theta \cos\theta \frac{75^2}{2}$$

$$F_{y2} = \int_0^{12} \rho g w H dx1 \sin\theta = \int_0^{12} \rho g w (12 - x1) \cos\theta \sin\theta dx1 = \rho g w \cos\theta \sin\theta (12 \cdot 12 - \frac{12^2}{2})$$

$$F_y(CG) = F_{y1} + F_{y2} - 9E7 = \rho g w \sin\theta \cos\theta (\frac{75^2}{2} + \frac{12^2}{2}) - 9E7 = \rho g w \frac{12 \cdot 75}{2} - 9E7 = 0$$

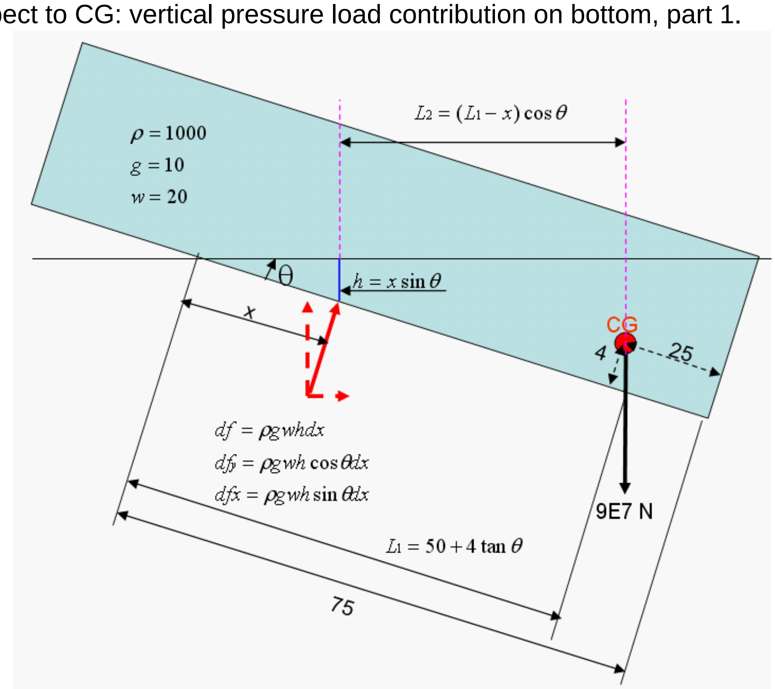
Horizontal Force:

$$F_{x1} = \int_0^{75} \rho g w h dx \sin\theta = \int_0^{75} \rho g w x \sin\theta \sin\theta dx = \rho g w \sin\theta \sin\theta \frac{75^2}{2}$$

$$F_{x2} = \int_0^{12} \rho g w H dx1 \cos\theta = \int_0^{12} \rho g w (12 - x1) \cos\theta \cos\theta dx1 = \rho g w \cos\theta \cos\theta (12 \cdot 12 - \frac{12^2}{2})$$

$$F_x(CG) = F_{x1} - F_{x2} = \rho g w \sin\theta \sin\theta \frac{75^2}{2} - \rho g w \cos\theta \cos\theta \frac{12^2}{2} = 0$$

Moment with respect to CG: vertical pressure load contribution on bottom, part 1.



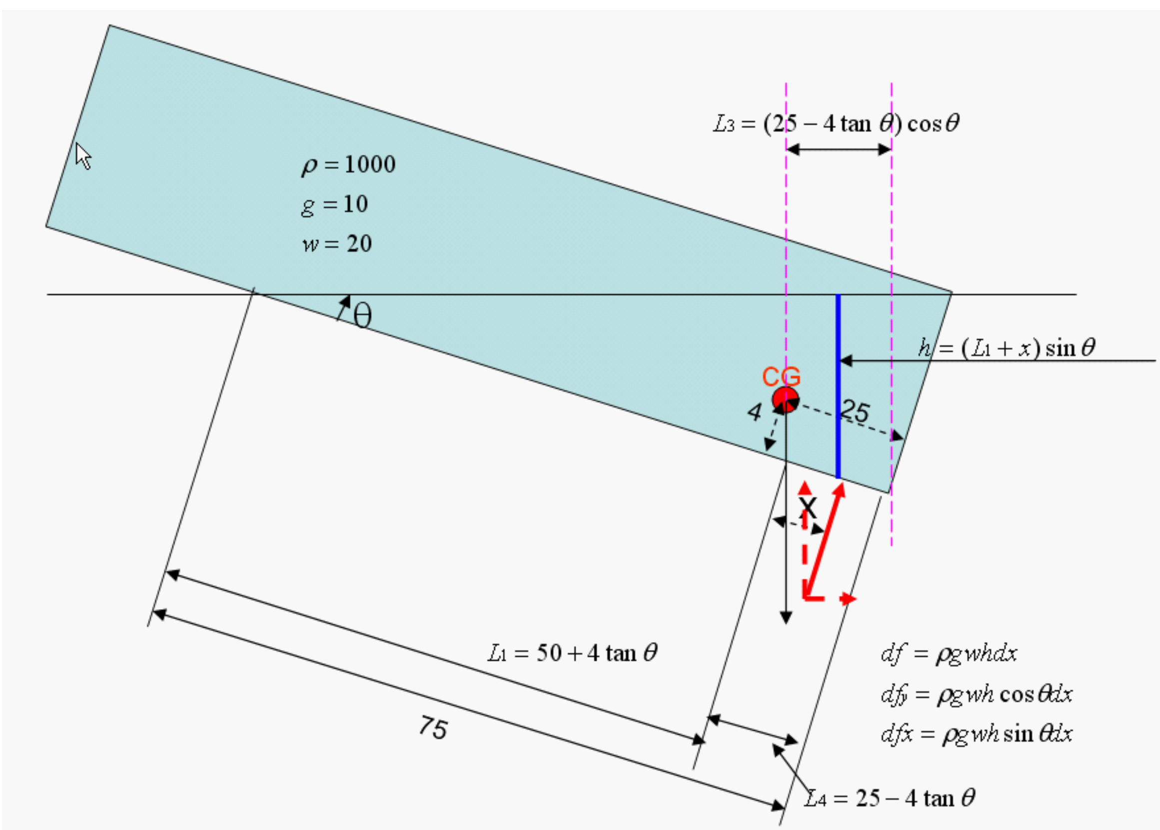
:

$$M_{y1}(L) == \int_0^L f_{y1} L_2 dx$$
$$= \int_0^L f_{y1}(L_1 - x)\cos\theta dx$$
$$= \int_0^L \rho g w h \cos\theta (L_1 - x)\cos\theta dx$$
$$= \int_0^L \rho g w x \sin\theta \cos\theta (L_1 - x)\cos\theta dx$$
$$= \rho g w \sin\theta \cos\theta^2 \left(L_1 \frac{L^2}{2} - \frac{L^3}{3} \right)$$

where

$$0 < L < L_1$$
$$L_1 = 50 + 4\tan\theta$$

Moment with respect to CG: vertical pressure load contribution on bottom, part 2.



$$M_{y2}(L) = \int_0^L f_{y2} \underline{x \cos\theta} dx$$

$$= \int_0^L \rho g w h \cos\theta x \cos\theta dx$$

$$= \int_0^L \rho g w (L_1 + x) \sin\theta \cos\theta x \cos\theta dx$$

$$= \int_0^L \rho g w (L_1 x + x^2) \sin\theta \cos\theta^2 dx$$

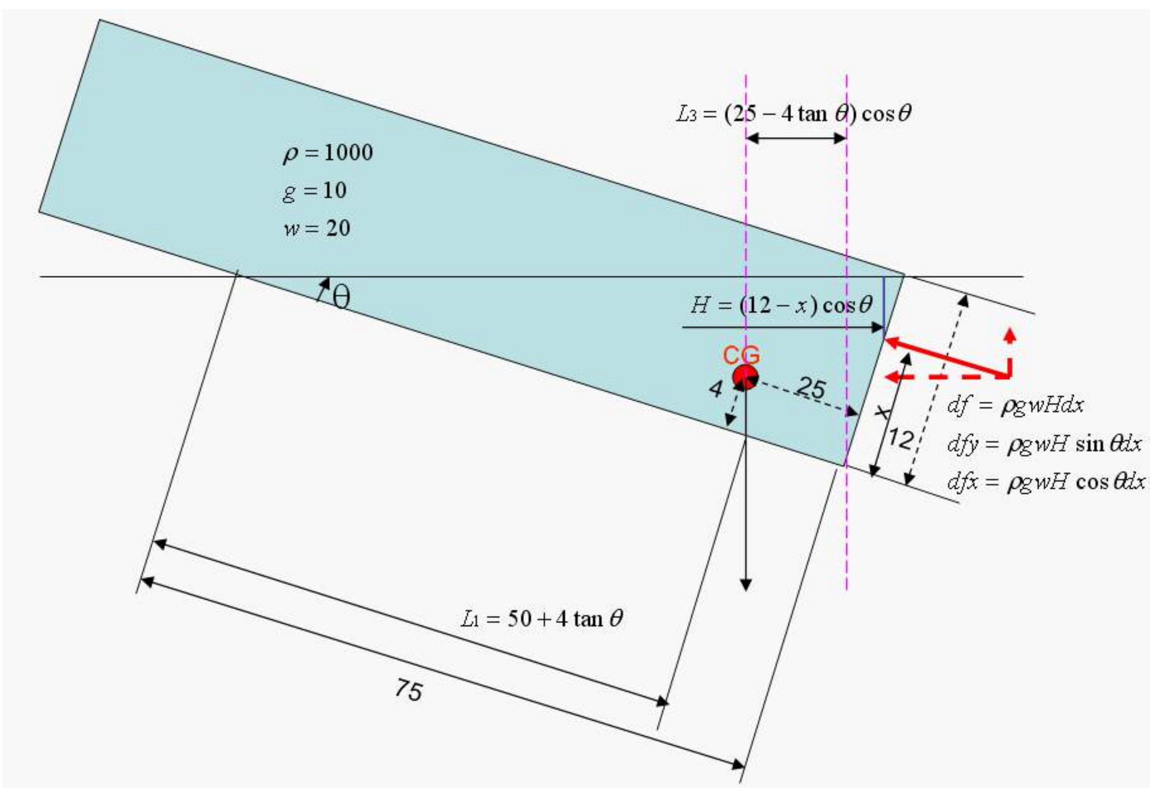
$$= \rho g w \sin\theta \cos\theta^2 \left(L_1 \frac{L^2}{2} + \frac{L^3}{3} \right)$$

where

$$0 < L < L_4$$

$$L_4 = 25 - 4\tan\theta$$

Moment with respect to CG: vertical pressure load contribution on bow.



$$M_{y3}(L) = \int_0^L f_{y3}(L_3 + x\sin\theta)dx$$

$$= \int_0^L \rho g w H(L_3 + x\sin\theta)\sin\theta dx$$

$$= \int_0^L \rho g w(12 - x)\cos\theta(L_3 + x\sin\theta)\sin\theta dx$$

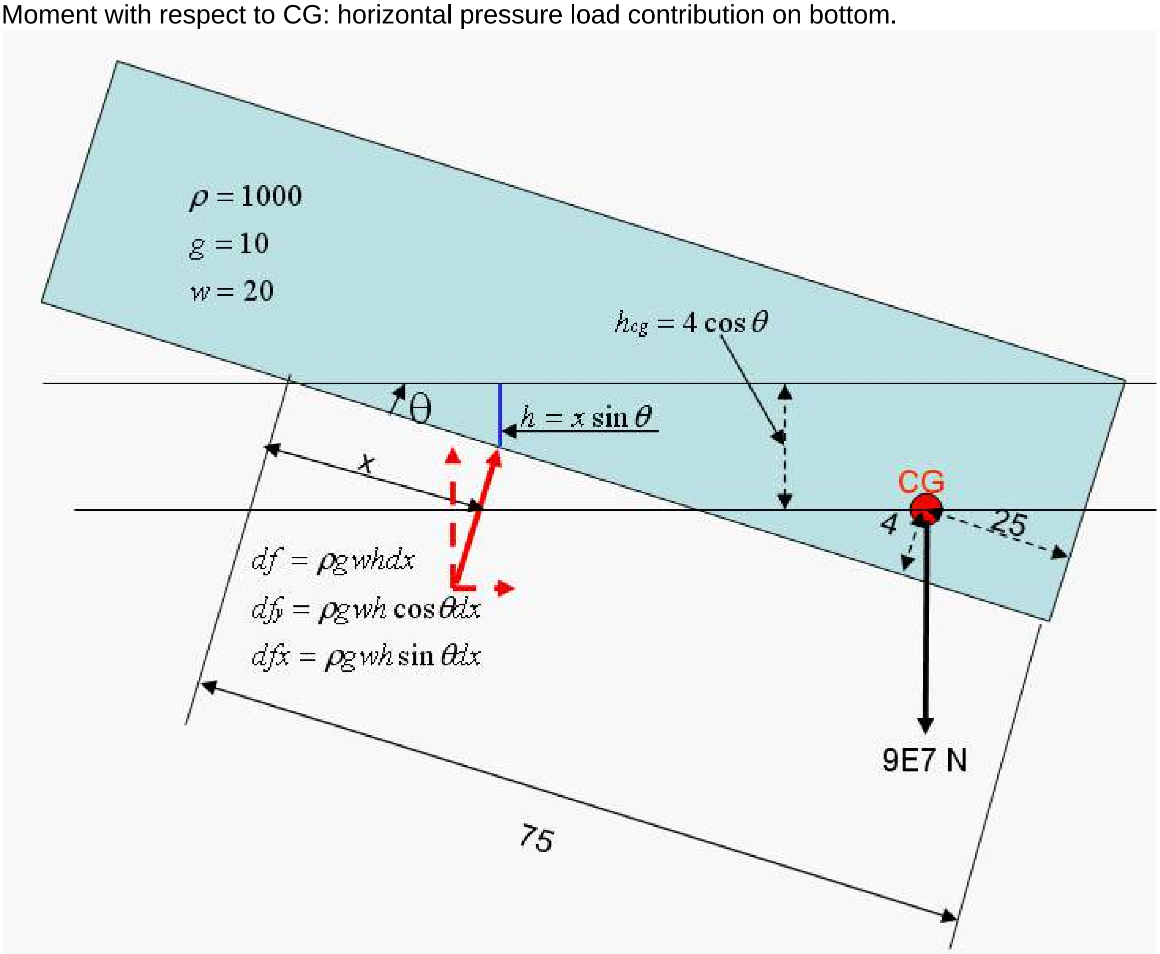
$$= \rho g w \cos\theta \sin\theta\left(12L_3L + \frac{L^2}{2}(12\sin\theta - L_3) - \sin\theta\frac{L^3}{3}\right)$$

where

$0 < L < 12$

$L_3 = (25 - 4\tan\theta)\cos\theta$

Moment with respect to CG: horizontal pressure load contribution on bottom.



$$M_{x1}(L)=\int_0^L f_{x1}(h_{cg}-h)dx$$

$$=\int_0^L \rho g w h \sin\theta\,(h_{cg}-h)dx$$

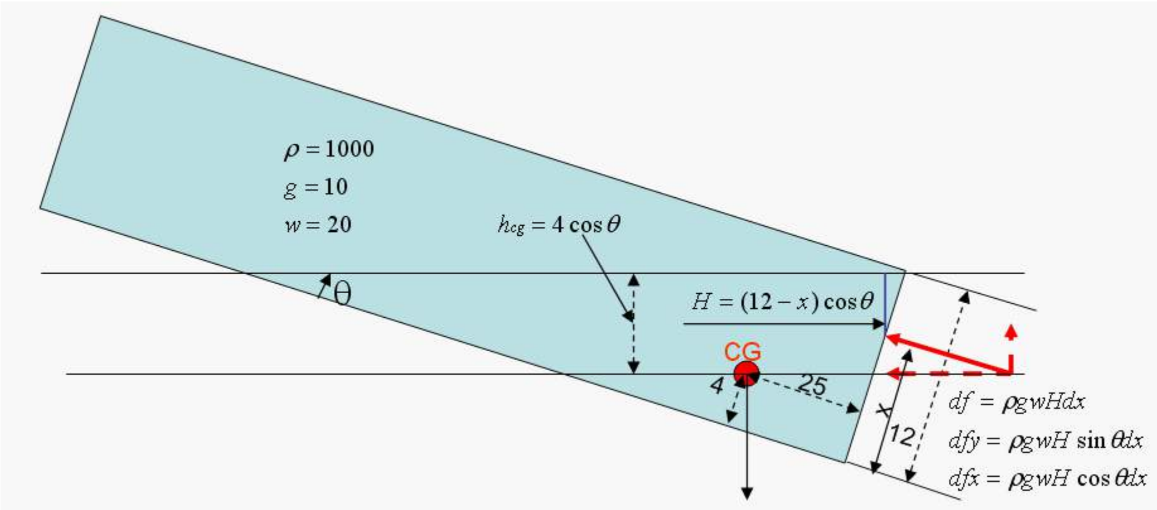
$$=\int_0^L \rho g w x \sin\theta\,\sin\theta\,(h_{cg}-x\sin\theta)dx$$

$$=\rho g w \sin\theta^{2}\left(h_{cg}\frac{L^2}{2}-\frac{L^3}{3}\sin\theta\right)$$

where

$0<L<75$

Moment with respect to CG: horizontal pressure load contribution on bow



$$M_{x2}(L)=\int_0^L f_{x2}(h_{cg}-H)dx$$

$$=\int_0^L \rho g w H\cos\theta\,(h_{cg}-H)dx$$

$$=\int_0^L \rho g w(12-x)\cos\theta\,\cos\theta\,(h_{cg}-(12-x)\cos\theta)dx$$

$$=\rho g w \cos\theta^{2}\left(h_{cg}\left(12L-\frac{L^2}{2}\right)-\left(12^2L-12L^2+\frac{L^3}{3}\right)\cos\theta\right)$$

where

$0<L<12$

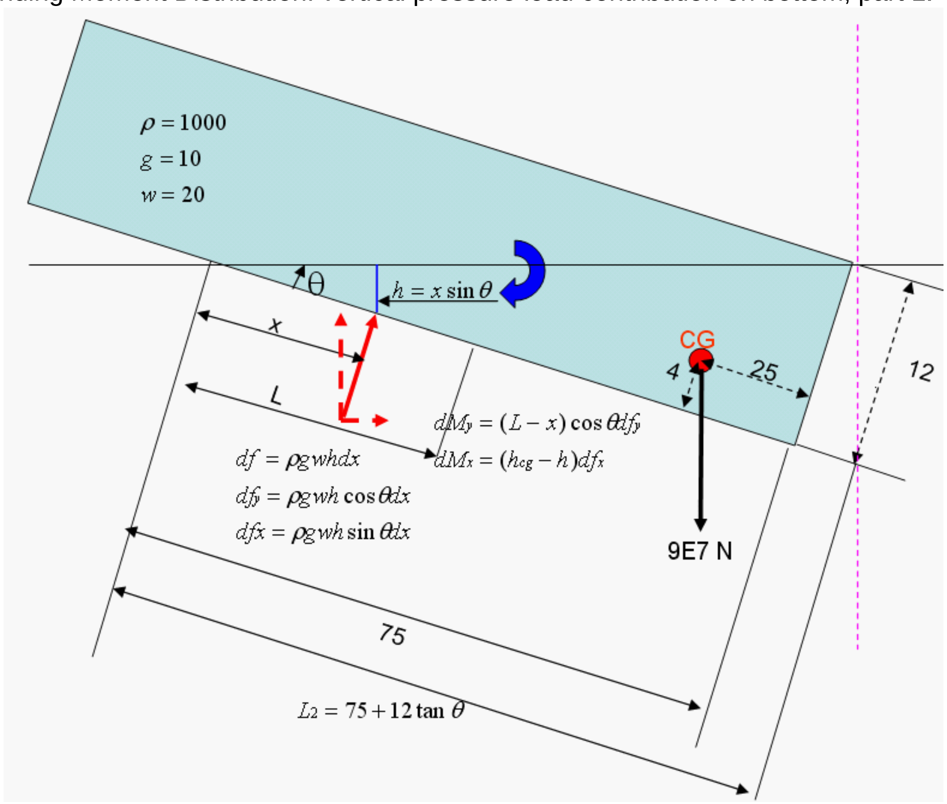
The combined moment at CG

$$\sum M_z(CG) = M_{y1}(50 + 4\tan\theta) + M_{y2}(25 - 4\tan\theta) + M_{y3}(12) + M_{xq}(75) + M_{x2}(12) = 0$$

Vertical Bending Moment Distribution

While the static equilibrium equations are formulated with respect to the center of gravity, naval architects often prefer to check vertical bending moment and shear force distribution. The closure of bending moment and shear force distribution is an alternative way to validate equilibrium.

Vertical Bending Moment Distribution: vertical pressure load contribution on bottom, part 1.



$$M_{y1}(L) == \int_0^L f_{y1} L_2 dx$$

$$= \int_0^L f_{y1}(L - x)\cos\theta dx$$

$$= \int_0^L \rho g w h \cos\theta (L - x)\cos\theta dx$$

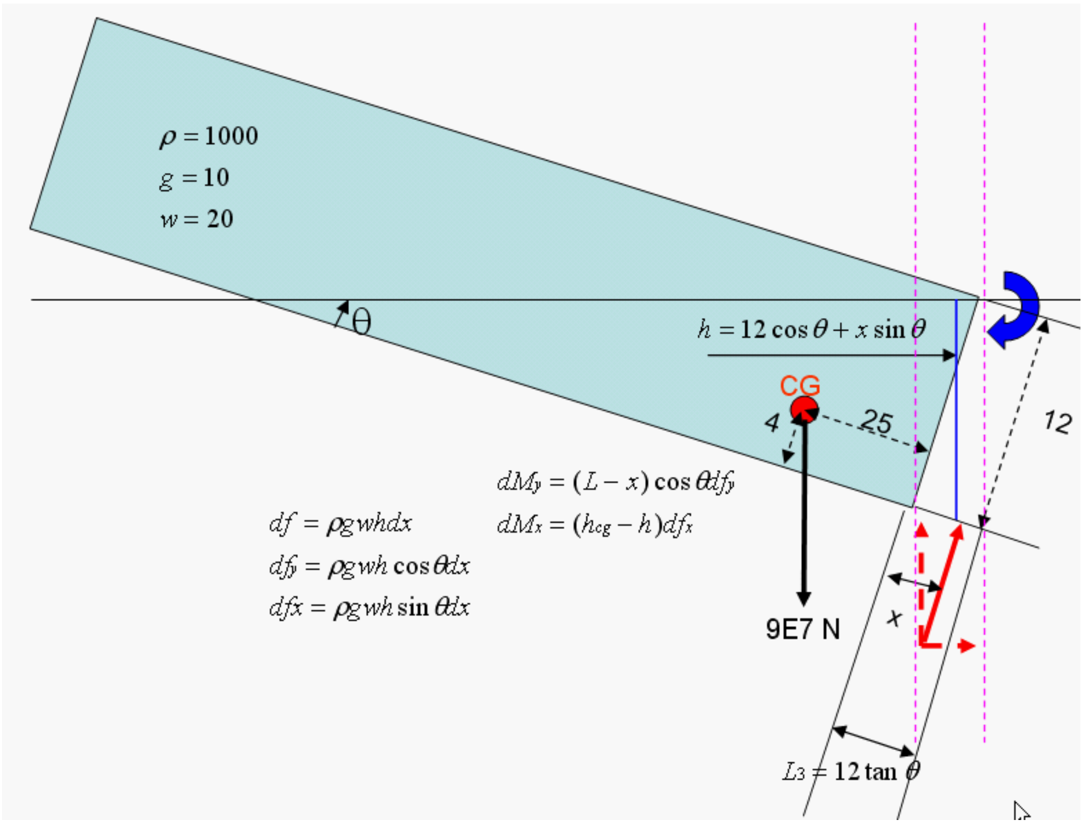
$$= \int_0^L \rho g w x \sin\theta \cos\theta (L - x)\cos\theta dx$$

$$= \rho g w \sin\theta \cos\theta^2 \frac{L^3}{6}$$

where

$$0 < L < L_2$$
$$L_2 = 75 + 12\tan\theta$$

Vertical Bending Moment Distribution: vertical pressure load contribution on bottom, part 2.

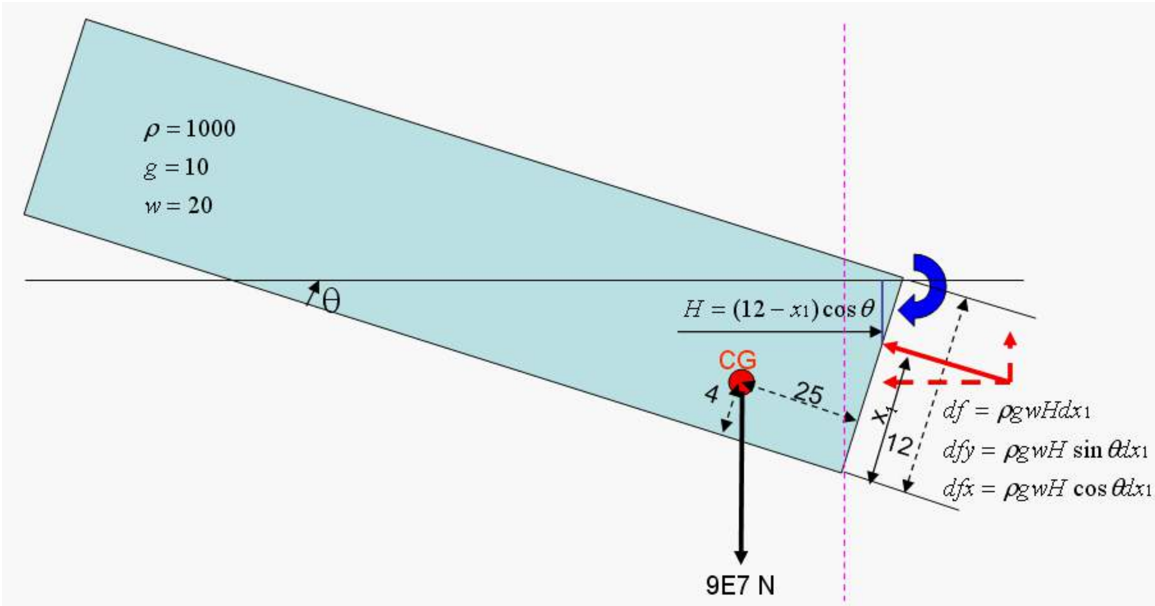


$$M_{y2}(L) = -\int_0^L f_{y2}(L-x)\cos\theta dx$$
$$= -\int_0^L \rho gwh\cos\theta(L-x)\cos\theta dx$$
$$= -\int_0^L \rho gw(12\cos\theta + x\sin\theta)\cos\theta(L-x)\cos\theta dx$$
$$= -\rho gw\cos\theta^2\left(12L^2\cos\theta + (-12\cos\theta + L\sin\theta)\frac{L^2}{2} - \frac{L^3}{3}\sin\theta\right)$$

where
$$0 < L < 12\tan\theta$$

Vertical Bending Moment Distribution: vertical pressure load contribution on bow.

© 2014 Optimum Structural Design, Inc.



$$M_{y3}(L) = \int_0^L f_{y3}(L-x)\sin\theta dx$$

$$= \int_0^L \rho g w H \sin\theta (L-x)\sin\theta dx$$

$$= \int_0^L \rho g w(12-x)\cos\theta \sin\theta (L-x)\sin\theta dx$$

$$= \rho g w \cos\theta \sin\theta^2 \left(12L^2 - \frac{L^2}{2}(12+L) + \frac{L^3}{3}\right)$$

where

$$0 < L < 12$$

$$M_{y4}(L) = -9E7L$$

where

$$0 < L < (25 - 4\tan\theta)\cos\theta + 12\sin\theta$$

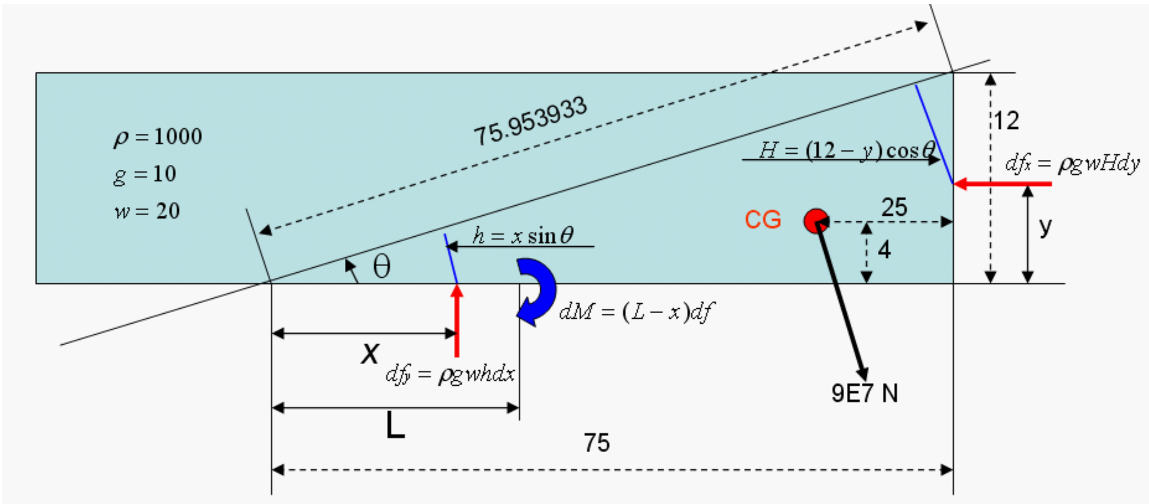
Combined bending moment

$$M_y(End) = M_{y1}(75 + 12\tan\theta) + M_{y2}(12\tan\theta) + M_{y3}(12) + M_{y4}((25-4\tan\theta)\cos\theta + 12\sin\theta) = 0$$

Exact solution derivation in the local (structural) coordinate system

The above derivation is based on the global coordinate system. The static equilibrium should also be fulfilled in the local coordinate system. MAESTRO's graphical display and tabulate data are based on the local coordinate system.

© 2014 Optimum Structural Design, Inc.



$$M_{y1}(L) == \int_0^L f_y(L-x)dx$$

$$= \int_0^L \rho g w h(L-x)dx$$

$$= \int_0^L \rho g w x \sin\theta (L-x)dx$$

$$= \rho g w \sin\theta \frac{L^3}{6}$$

where

$$0 < L < 75$$

Bending Moment due to concentrated load

$$M_{y2} = \begin{cases} 0 & 0 < L < 50 \\ 9E7\cos\theta L & 50 < L < 75 \end{cases}$$

$$M_x = \int_0^{12} f_x(y-4)dy$$

$$= \int_0^{12} \rho g w H(y-4)dy$$

$$= \int_0^{12} \rho g w(12-y)\cos\theta (y-4)dy$$

$$= \rho g w \cos\theta \left(-12 \cdot 4 \cdot 12 + 16\frac{12^2}{2} - \frac{12^3}{3} \right)$$

$$= 0$$

© 2014 Optimum Structural Design, Inc.

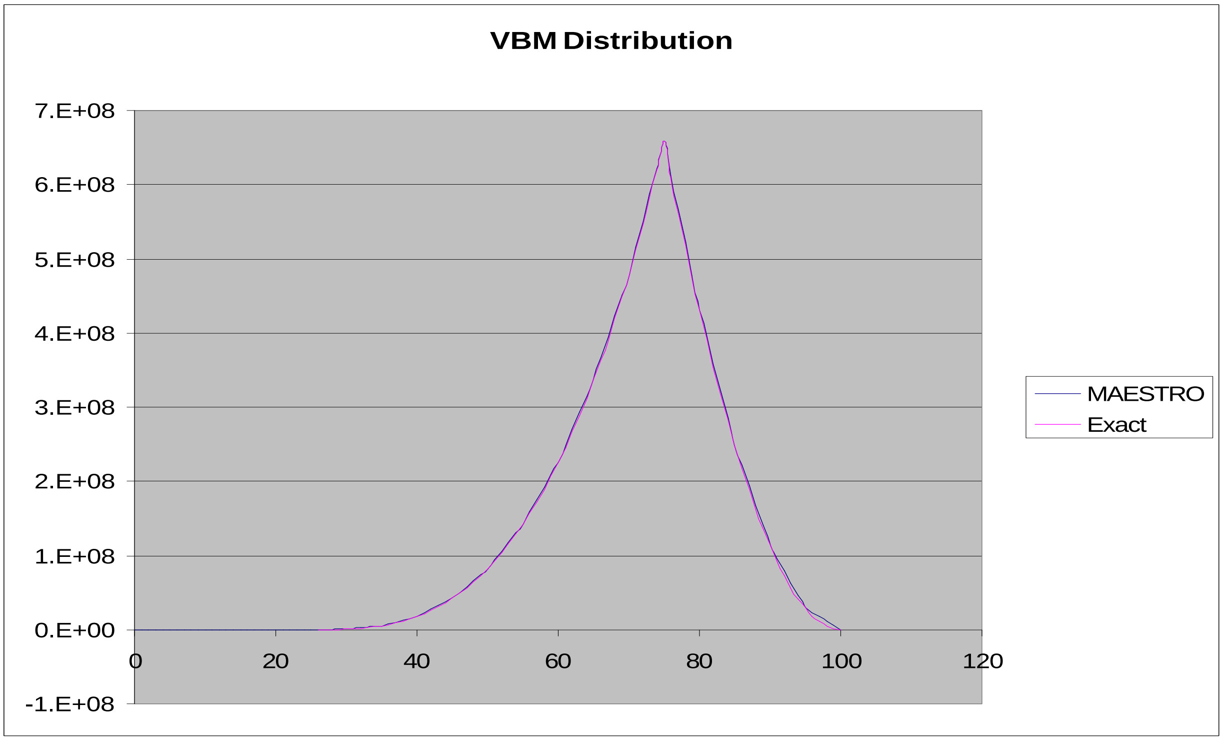
Bending Moment at the end

$$M = M_{y1} - M_{y2} + M_x$$

$$= \rho g w \sin\theta \frac{75^3}{6} - 9E7\cos\theta \bullet 75 + 0$$

$$= 2E5\sin\theta \frac{75^3}{6} - 9E7\cos\theta \bullet 75$$

$$= 0$$



X	MAESTRO	Exact	X	MAESTRO	Exact
0	0		71	5.16E+08	512604607.2
25	0		72	5.51E+08	546767363.9
26	131658	5266.341407	73	5.87E+08	582415228.9
27	263316	42130.73126	74	6.23E+08	619579800.2
28	394973	142191.218	75	6.58E+08	658292675.9
29	526631	337045.8501	76	6.13E+08	609716945.1
30	658289	658292.6759	77	5.68E+08	562751635.3
31	1.58E+06	1137529.744	78	5.22E+08	517429426.6
32	2.50E+06	1806355.103	79	4.77E+08	473781917.3

33	3.42E+06	2696366.8	80	4.32E+08	431840705.3
34	4.34E+06	3839162.886	81	3.95E+08	391637388.9
35	5.27E+06	5266341.407	82	3.59E+08	353203566.2
36	7.77E+06	7009500.413	83	3.22E+08	316570835.1
37	1.03E+07	9100237.951	84	2.85E+08	281770793.9
38	1.28E+07	11570152.07	85	2.49E+08	248835040.6
39	1.53E+07	14450840.82	86	2.22E+08	217795173.3
40	1.78E+07	17773902.25	87	1.95E+08	188682790.1
41	2.26E+07	21570934.4	88	1.67E+08	161529489.2
42	2.75E+07	25873535.33	89	1.40E+08	136366868.6
43	3.24E+07	30713303.09	90	1.13E+08	113226526.4
44	3.73E+07	36121835.71	91	9.64E+07	92140060.74
45	4.21E+07	42130731.26	92	7.95E+07	73139069.7
46	5.02E+07	48771587.77	93	6.27E+07	56255151.4
47	5.82E+07	56076003.3	94	4.58E+07	41519903.91
48	6.62E+07	64075575.9	95	2.90E+07	28964925.36
49	7.43E+07	72801903.61	96	2.32E+07	18621813.83
50	8.23E+07	82286584.49	97	1.74E+07	10522167.43
51	9.43E+07	92561216.57	98	1.16E+07	4697584.258
52	1.06E+08	103657397.9	99	5.79E+06	1179662.415
53	1.18E+08	115606726.6	100	-389.622	0
54	1.30E+08	128440800.6			
55	1.42E+08	142191218			
56	1.59E+08	156889576.9			
57	1.76E+08	172567475.2			
58	1.92E+08	189256511.1			
59	2.09E+08	206988282.7			
60	2.26E+08	225794387.8			
61	2.48E+08	245706424.7			
62	2.70E+08	266755991.3			

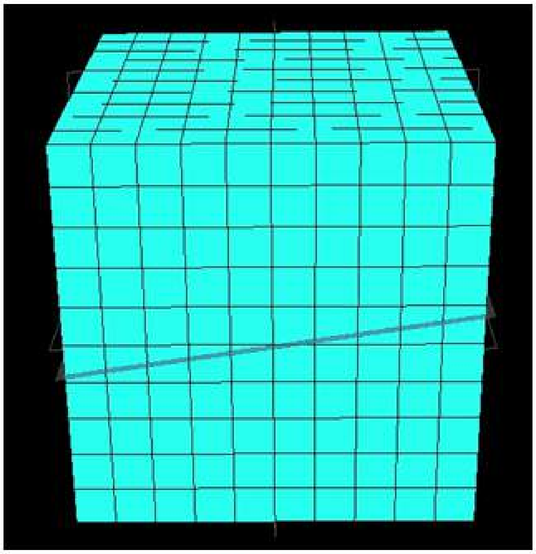
63	2.93E+08	288974685.7
64	3.15E+08	312394105.9
65	3.37E+08	337045850.1
66	3.66E+08	362961516.1
67	3.94E+08	390172702.2
68	4.23E+08	418711006.3
69	4.51E+08	448608026.4
70	4.80E+08	479895360.7

Primitive Shapes Balance

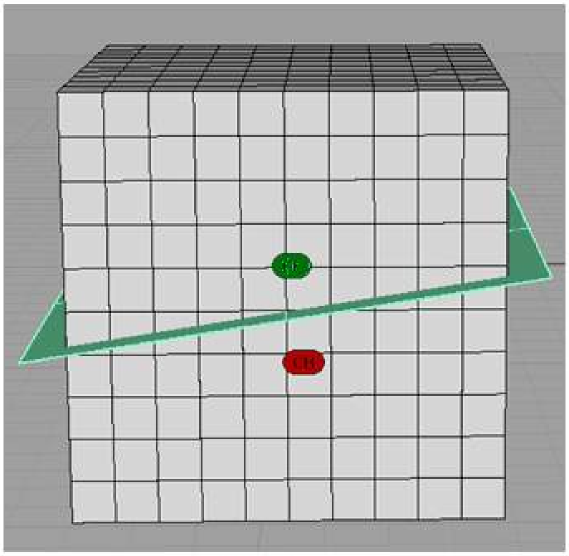
The following presents the hydrostatic verification results for a cube, cylinder, and sphere. These problems demonstrate the hydrostatic balance capabilities of MAESTRO by applying them to a polygon cube, cylinder, and sphere with geometric centers placed at the origin. Rhino was used to create these primitives by using the *Mesh>Polygonal Mesh Primitives>Box*. The number of panels per side were created by using the Rhino Command Line. Next, the geometry was selected in Rhino and the *File>Export Selected* was chosen, which allows a *.ply file to be exported for consumption into MAESTRO. An upright unstable condition and a heeled stable condition were tested and the results presented below.

Cube

See the ...\MAESTRO\Models and Samples\Verification Models\Hydrostatic Balance directory for model data and results.



MAESTRO Heeled Case



Orca Heeled Case

10m x 10m Cube								
Load Case	Analytical Results for 5m Draft	LC1 – 5m Draft		Percent Difference from Analytical		LC2 – Heeled Steady Case at mean draft of 5m		
	Orca3D	Orca3D	MAESTRO	Orca3D	MAESTR	Orca3D	MAESTRO	GHS
Displacement (kgf)	512950.000	512950.000	512950.000	0.000	0.000	512950.000	512950.000	512950.000
Volume Displaced (m^3)	500.000	500.000	500.000	0.000	0.000	500.000	500.000	500.000
LCB (m) (+ is aft of origin)	0.000	0.000	0.000	0*	0*	0.261	0.261	0.261
TCB (m) (+ is starboard)	0.000	0.000	0.000	0*	0*	0.000	0.000	0.000
VCB (m) (- is below design wate	-2.500	-2.500	-2.500	0.000	0.000	-2.479	-2.480	-2.477
Area Moment of Inertia (m^4)	833.333	833.333	833.333	0.000	0.000	843.526	843.519	842.5**
Heel Angle (deg)	0.000	0.000	0.000	0*	0*	0.000	0.000	0.000
Trim Angle (deg)	0.000	0.000	0.000	0*	0*	8.916	-8.913	8.920
Bmt (m)	1.667	1.667	1.667	0.020	0.020	1.687	1.687	1.685
Bml (m)	1.667	1.667	1.667	0.020	0.020	1.729	1.729	1.727
GMt (m)	-0.833	-0.83	-0.833	0.360	0.000	3.23	3.226	3.227
GMl (m)	-0.833	-0.83	-0.833	0.360	0.000	3.27	3.268	3.268
LCG (m) (+ is aft of origin)	0.000	0.000	0.000	0*	0*	0.500	0.500	0.500
TCG (m) (+ is starboard)	0.000	0.000	0.000	0*	0*	0.000	0.000	0.000
VCG (m) (- is below design wate	0.000	0.000	0.000	0*	0*	-4.000	-4.000	-4.000

* Absolute error in the case that the expected value is undefined.

** Transverse Area Moment of Inertia was backed out of BMt. Expect rounding error.

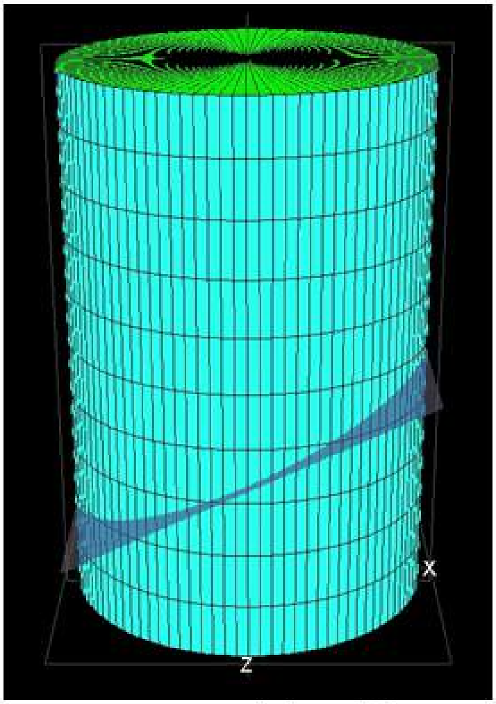
NOTE -90 degree Rotation about x-axis in MAESTRO

Waterplane Area (m^2)	100.000	100.000				101.223	
Wetted Surface (m^2)	300.000	300.000				300.000	

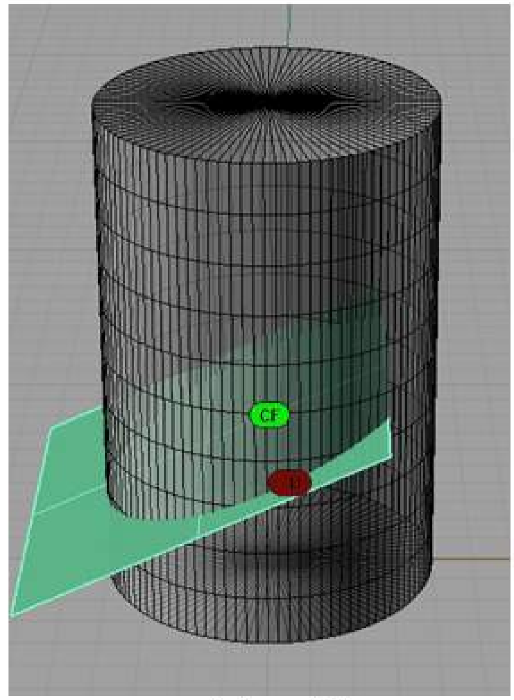
Cylinder

See the ...\MAESTRO\Models and Samples\Verification Models\Hydrostatic Balance directory for model data and results.

© 2014 Optimum Structural Design, Inc.



MAESTRO Heeled Condition



Orca Heeled Condition

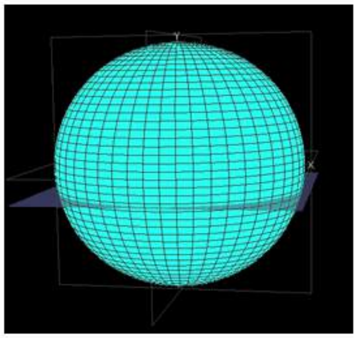
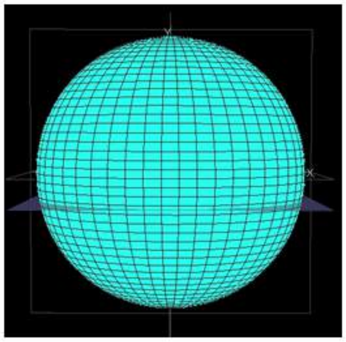
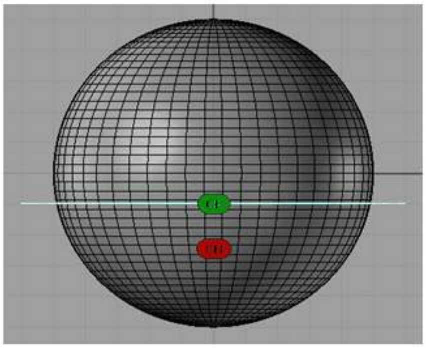
12 m by 8 m dia. Cylinder									
Load Case	Analytical Results for 4m Draft		LC1 – 4m Draft		Percent Difference from Analytical		LC2 – Heeled Condition		
	Orca3D	MAESTR	Orca3D	MAESTRO	Orca3D	MAESTR	Orca3D	MAESTR	GHS
Displacement (kgf)	206269.434		206269.434	206269.215	0.000	0.000	206269.43	206267.176	206280**
Volume Displaced (m^3)	201.062		201.062	201.061	0.000	0.000	201.062	201.059	201.072***
LCB (m)	0.000		0.000	0.000	0*	0*	0.473	0.473	0.471
TCB (m)	0.000		0.000	0.000	0*	0*	0.000	0.000	0.000
VCB (m)	2.000		2.001	2.000	0.050	0.000	-3.887	-3.887	-3.883
Transverse Area Moment of Inertia	201.062		200.798	200.798	0.131	0.131	222.169	222.167	221.381****
Heel Angle (deg)	0.000		0.000	0.000	0*	0*	0.000	0.000	0.000
Trim Angle (deg)	0.000		0.000	0.000	0*	0*	25.337	-25.336	25.350
Bmt (m)	1.000		0.999	0.999	0.100	0.100	1.105	1.105	1.101
Bml (m)	1.000		0.999	0.999	0.100	0.100	1.353	1.353	1.348
GMt (m)	-3.000		-3.00	-3.000	0.000	0.000	2.34	2.337	2.337
GMl (m)	-3.000		-3.00	-3.000	0.000	0.000	2.58	2.584	2.585
LCG (m)	0.000		0.000	0.000	0*	0*	1.000	1.000	1.000
TCG (m)	0.000		0.000	0.000	0*	0*	0.000	0.000	0.000
VCG (m)	0.000		0.000	0.000	0*	0*	-5.000	-5.000	-5.000

* Absolute error in the case that the expected value is undefined.
** GHS requires weight input in Metric Tonnes for larger vessels. Thus the accuracy of loading has been reduced to 206,280.
*** Volume was derived from displacement divided by specific gravity of 1.0259 MT/m^3
**** Transverse Area Moment of Inertia was backed out of Bmt. Expect rounding error.

Weight (kg) or Sinkage			4m Sinkage	206269.434			206269.43	206269.43
Waterplane Area (m^2)	50.265		50.232		0.066	100.000	55.579	
Wetted Surface (m^2)	150.796		150.813		0.011	100.000	150.813	

Sphere

See the ...\MAESTRO\Models and Samples\Verification Models\Hydrostatic Balance directory for model data and results.

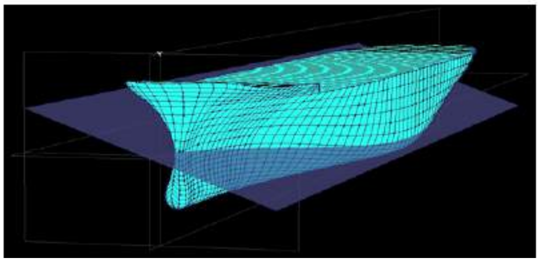
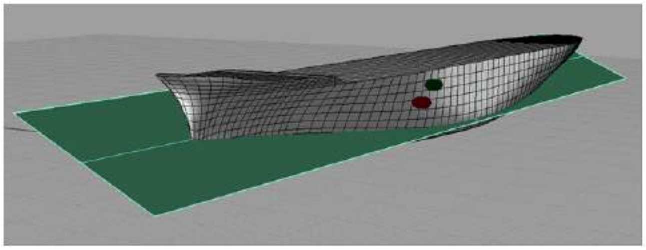




5 Meter Radius Sphere (50 Vertical and 50 Around Faces)												
Load Case	Analytical Results for 5m Draft		LC1 – 5m Draft with CG at center of sphere		Percent Difference from Analytical		Analytical Results for 4m Draft		LC2 – 4m Draft with CG at center of sphere		Percent Difference from Analytical	
	Orca3D	MAESTRO	Orca3D	MAESTRO	Orca3D	MAESTRO	Orca3D	MAESTRO	Orca3D	MAESTRO	Orca3D	MAESTRO
Displacement (kgf)	268579.992		268579.992	268579.994	0.000	0.000	189080.314		189080.3	189080.551	0.000	0.000
Volume Displaced (m^3)	261.799		261.560	261.799	0.091	0.000	184.307		184.307	184.307	0.000	0.000
VCB (m) (- is below design waterlin	-1.875		-1.867	-1.867	0.427	0.427	-2.455		-2.448	-2.448	0.265	0.252
Transverse Area Moment of Inertia	490.874		488.280	488.144	0.528	0.556	452.389		449.868	449.868	0.557	0.557
Heel Angle (deg)	0.000		0.000	0.000	0.000	0.000	0.000		0.000	0.000	0.000	0.000
Trim Angle (deg)	0.000		0.000	0.000	0.000	0.000	0.000		0.000	0.000	0.000	0.000
Bmt (m)	1.875		1.865	1.865	0.533	0.556	2.455		2.441	2.441	0.550	0.555
Bml (m)	1.875		1.865	1.865	0.533	0.556	2.455		2.441	2.441	0.550	0.555
GMt (m)	0.000		0.00	-0.003	0.00	0.003	0.000		-0.01	-0.007	0.01	0.007
GMl (m)	0.000		0.00	-0.003	0.00	0.003	0.000		-0.01	-0.007	0.01	0.007
Weight (kg) or free float			268579.99	268579.992					189080.3	189080.314		
Waterplane Area (m^2)	78.540		78.332		0.265		75.398		75.128		0.358	
Wetted Surface (m^2)	157.080		157.176		0.061		125.664		125.692		0.022	

S175 Hull Form Balance

The following presents the hydrostatic verification results for the S175 hull form. This hull was generated in Rhino and then exported using the Export Selected command in Rhino to generate a *.ply file, which was then consumed by MAESTRO. An upright and heeled condition was tested and the results presented below.

See the ...\MAESTRO\Models and Samples\Verification Models\Hydrostatic Balance directory for model data and results.

S175 Hullform						
Load Case	LC1 – Design Draft at Design Waterline			LC2 – Heeled Condition		
	Orca3D	MAESTRO	GHS	Orca3D	MAESTRO	GHS
Displacement (kgf)	24587986.544	24586001.270	24587990.000	24588001.353	24584154.650	24587990.000
Volume Displaced (m^3)	23967.235	23963.500	23967*	23967.250	23963.500	23967*
LCB (m) (+ is aft of forwardmost extent)	96.937	96.924	96.937	96.926	96.921	96.924
TCB (m) (+ is starboard)	0.000	0.000	0.000	2.497	2.494	2.493
VCB (m) (- is below design waterline)	-4.284	-4.284	-4.274	-3.698	-3.698	-3.688
Transverse Area Moment of Inertia (m^4)	127680.132	127680	127170**	168094.442	168040	167579**
Heel Angle (deg)	0.000	0.000	0.00	25.285	25.269	25.30
Trim Angle (deg)	0.000	0.000	0.01	0.158	0.157	0.15
Bmt (m)	5.327	5.328	5.306	7.014	7.012	6.992
Bml (m)	209.212	215.609	209.160	238.386	238.392	238.510
GMt (m)	1.04	1.044	1.032	2.92	2.923	2.913
GMl (m)	204.93	211.325	204.880	234.30	234.303	234.430
LCG (m) (+ is aft of forwardmost extent)	96.937	96.937	96.937	96.937	96.937	96.937
TCG (m) (+ is starboard)	0.000	0.000	0.000	0.750	0.750	0.750
VCG (m) (- is below design waterline)	0.000	0.000	0.000	0.000	0.000	0.000

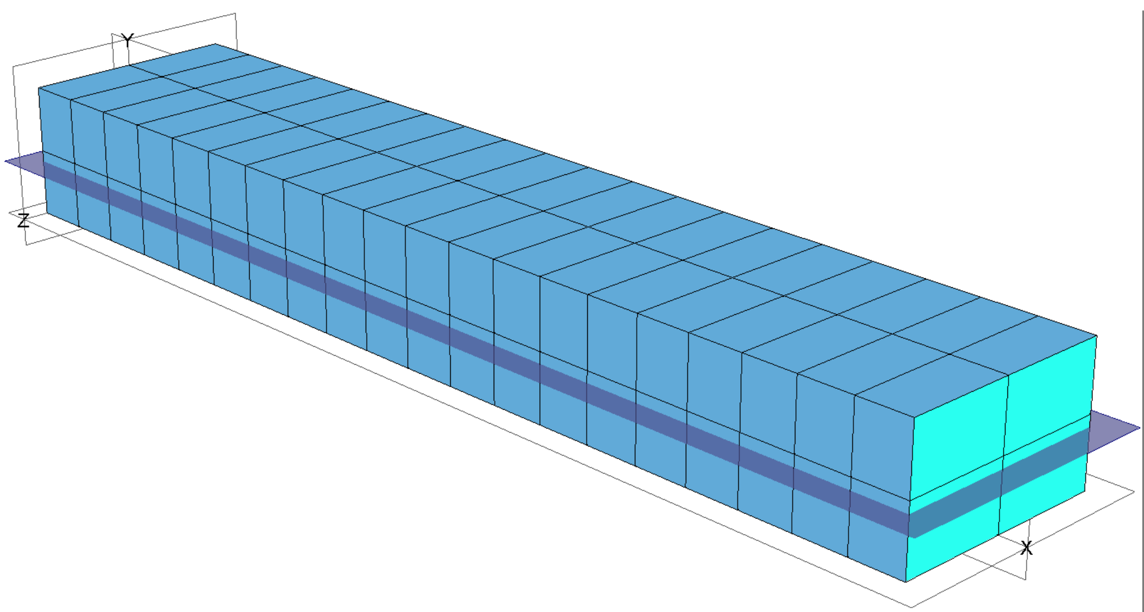
* Volume was derived from displacement divided by specific gravity of 1.0259 MT/m^3

** Transverse Area Moment of Inertia was backed out of Bmt. Expect rounding error.

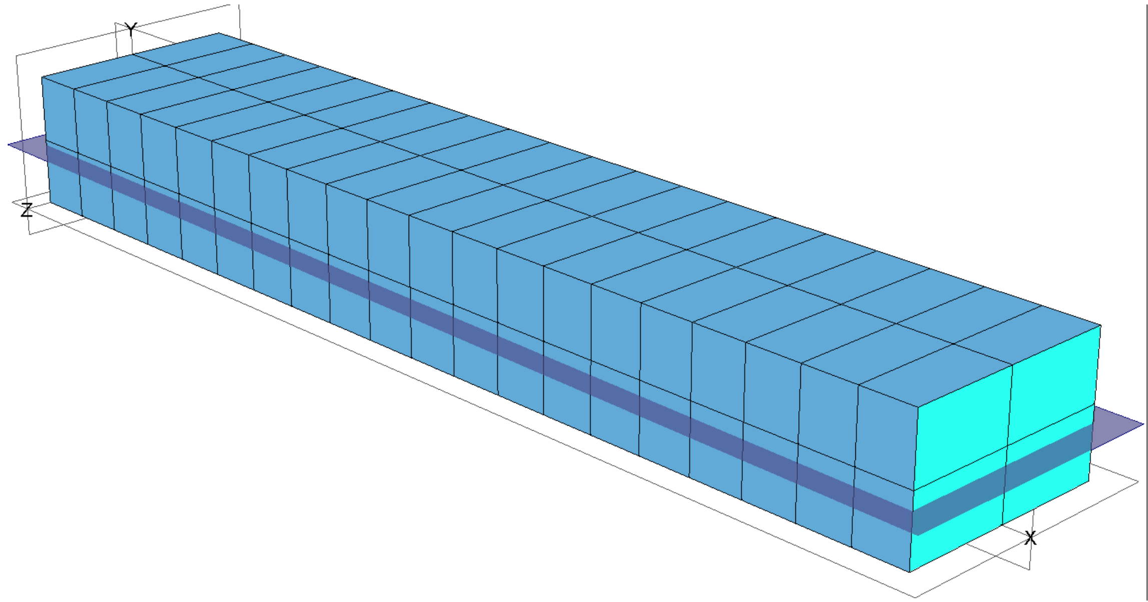
Weight (kg) or free float	free float	24587986.54	24587986.54	24587986.544	24587986.544	24587986.54
Waterplane Area (m^2)	3158.153			3481.640		
Wetted Surface (m^2)	5360.846			5469.693		

Inertia Relief Balance Validation

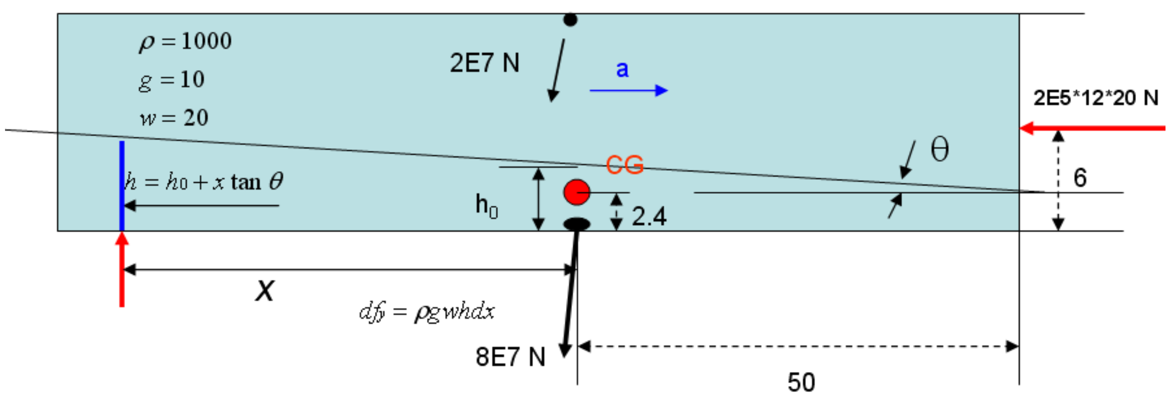
To validate MAESTRO's inertia relief balance, a variation of the above model is used. The Modeler file is called *BalanceTest-acc.mdl*. A point mass of 4E6 kg is located at the 1/4 length position (X = 25 m) on bottom shell, a point mass of 2E6 kg is located at the 1/2 length position (X = 50 m) on deck shell, and a point mass of 4E6 kg is located at the 3/4 length position (X = 75 m) on bottom shell. The bow is subjected to 20E5N uniform slamming pressure.



Because the model is a floating structure, hydrostatic balance is performed first, which results a small trim (0.594 degrees). The model now has equilibrium in heave, roll and pitch. Note that the slamming force (x-direction, surge) can not be balanced by the hydrostatic balance. Next, an inertia relief balance is performed, which automatically generates a -4.9 surge acceleration to balance the slamming load. The model is now fully balanced in 6 degrees of freedom.



Exact solution derivation in the local (structural) coordinate system



$$\begin{cases} \sum M_z = 0 \\ \sum F_x = 0 \\ \sum F_y = 0 \end{cases}$$

$$\begin{cases} \rho g w\left(\int_0^{50}(h_0 + x\tan\theta)x dx - \int_0^{50}(h_0 - x\tan\theta)x dx\right) + 2E6(a - g\sin\theta)\cdot 9.6 - 8E6(a - g\sin\theta)\cdot 2.4 - 4.8E7\cdot 2.4 = 0 \\ 1E7(a - g\sin\theta) - 2E5\cdot 12\cdot 20 = 0 \\ \int_0^{50}\rho g w(h_0 + x\tan\theta)dx + \int_0^{50}\rho g w(h_0 - x\tan\theta)dx - 1E7 g\cos\theta = 0 \end{cases}$$

$$\begin{cases} \theta = 0.594021367875417^{\circ} \\ a = 4.903674427886449 \\ h_0 = 4.999731283104183 \end{cases}$$

Bending moment distribution due to buoyancy

$$M_1(L) = \int_0^L \rho g w\big(\big(100 - x\big)\tan\theta + \big(h_0 - 50\tan\theta\big)\big)\big(L - x\big)dx$$

Where

$0 < L < 100$

Bending moment distribution due to gravity contribution in y direction

$$M_2(L) = -4E6g\cos\theta(L - 25)$$

Where

$25 < L < 100$

$$M_3(L) = -2E6g\cos\theta(L - 50)$$

Where

$50 < L < 100$

$$M_4(L) = -4E6g\cos\theta(L - 75)$$

Where

$75 < L < 100$

Bending moment distribution due to gravity and additional acceleration contribution in x direction

$$M_6(L) = -4E6\big(a - g\sin\theta\big)\cdot 2.4$$

Where

$25 < L < 100$

$$M_7(L) = 2E6\big(a - g\sin\theta\big)\cdot(12 - 2.4)$$

Where

$50 < L < 100$

$$M_7(L) = -4E6\big(a - g\sin\theta\big)\cdot 2.4$$

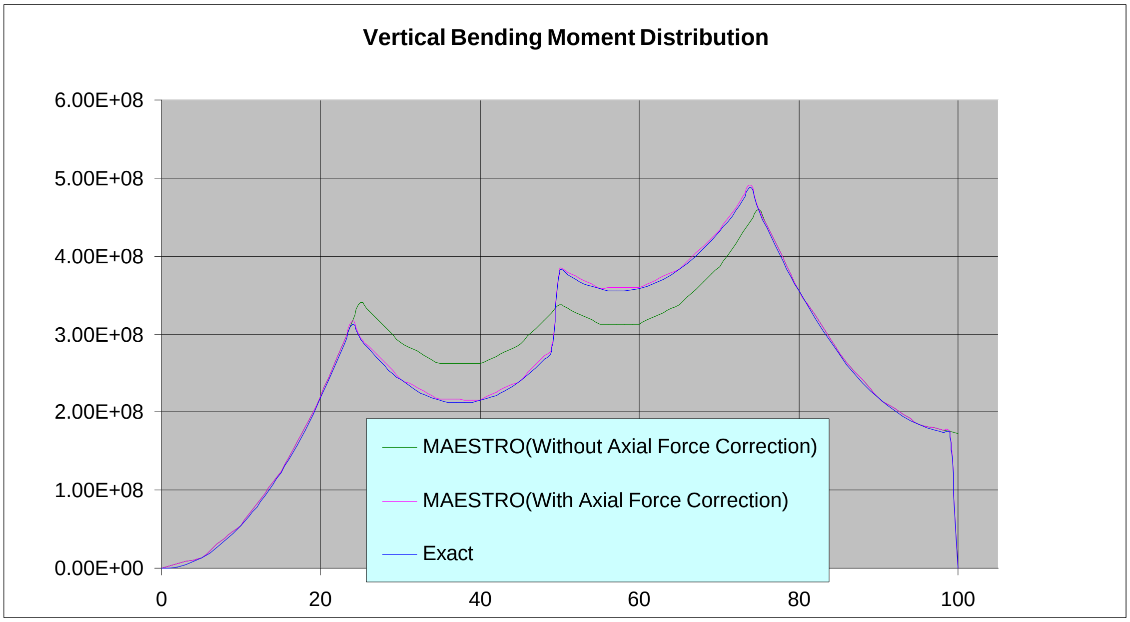
Where

$50 < L < 100$

Bending moment distribution due to end bulkhead pressure load

$$M_8(100) = -2E5\cdot 12\cdot 20\cdot\big(6 - 2.4\big)$$

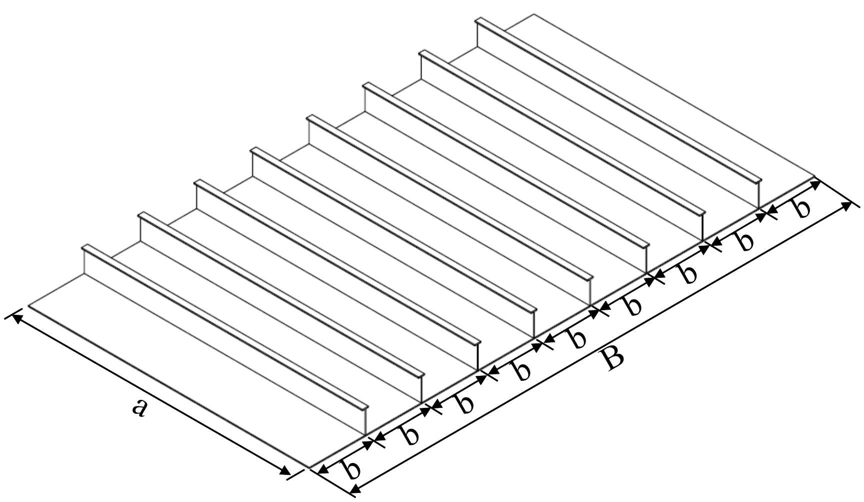
© 2014 Optimum Structural Design, Inc.



17.9 ULSAP

The following model has been created in MAESTRO using user-defined evaluation patches. The *.mdl files can be found in the *Models and Samples/Verification Models/ULSAP* directory. The model properties are shown below and he two tables describe the input data and results for the 15 load cases.

Model properties



- Density, ? = 7.850e-06 kg/mm^3
- Plate length, a = 4750 mm
- Plate breath, b = 950 mm
- Panel breath, B = 8550 mm
- Number of stiffeners, n_s = 8

Initial deflections (ISO formula)

- $w_{opl} = 0.1ß^2t_p$
- $w_{os} = 0.0015a$

Input data

Case	Lateral Pressure (MPa)	Load type $\sigma_x : \sigma_y$	Plate Thickness, t_p (mm)	Stiffener type	h_w (mm)	t_w (mm)	b_f (mm)	b_t (mm)	w_{opl} (mm)	w_{os} (mm)
01	0.0	1:0	15	Flat	250	25	-	-	9.168	7.125
02	0.0	1:0	15	Angle	235	10	90	15	9.168	7.125
03	0.0	1:0	15	Tee	235	10	90	15	9.168	7.125
04	0.0	1:0	15	Flat	350	35	-	-	9.168	7.125
05	0.0	1:0	15	Angle	383	12	100	17	9.168	7.125
06	0.0	1:0	15	Tee	383	12	100	17	9.168	7.125
07	0.0	1:0	18.5	Flat	250	25	-	-	7.434	7.125
08	0.0	1:0	18.5	Angle	235	10	90	15	7.434	7.125
09	0.0	1:0	18.5	Tee	235	10	90	15	7.434	7.125
10	0.0	1:1	15	Flat	250	25	-	-	9.168	7.125
11	0.0	1:1	15	Angle	235	10	90	15	9.168	7.125
12	0.0	1:7	15	Tee	235	10	90	15	9.168	7.125

© 2014 Optimum Structural Design, Inc.

13	0.1	1:0	15	Flat	250	25	-	-	9.168	7.125
14	0.1	1:0	15	Angle	235	10	90	15	9.168	7.125
15	0.1	1:0	15	Tee	235	10	90	15	9.168	7.125

*** S_x and S_y are longitudinal and transverse compression. There are no residual stress**

Ultimate limit strength using ALPS/ULSAP

Case	ALPS/ULSAP			FEA (ANSYS)		ULSAP/FEA	
	S_{xu}/S_{Yeq}	S_{yu}/S_{Yeq}	Mode	S_{xu}/S_{Yeq}	S_{yu}/S_{Yeq}	S_{xu}/S_{Yeq}	S_{yu}/S_{Yeq}
01	0.642	0.000	III	0.640	0.000	1.003	-
02	0.625	0.000	V	0.594	0.000	1.052	-
03	0.624	0.000	V	0.585	0.000	1.067	-
04	0.734	0.000	IV	0.794	0.000	0.924	-
05	0.684	0.000	V	0.702	0.000	0.974	-
06	0.673	0.000	V	0.655	0.000	1.027	-
07	0.616	0.000	III	0.647	0.000	0.952	-
08	0.651	0.000	III	0.640	0.000	1.017	-
09	0.651	0.000	III	0.605	0.000	1.076	-
10	0.250	0.250	III	0.224	0.224	1.116	1.116
11	0.253	0.253	III	0.217	0.217	1.164	1.164
12	0.253	0.253	III	0.215	0.215	1.174	1.174
13	0.268	0.000	III	-	-	-	-
14	0.332	0.000	III	-	-	-	-
15	0.332	0.000	III	0.436* 0.371**	-	0.762* 0.894**	-

*** is plate sided pressure, ** is stiffener sided pressure.**

© 2014 Optimum Structural Design, Inc.

Frequently Asked Questions

18

18 Frequently Asked Questions

This section covers some problems that are frequently encountered by users of MAESTRO. The questions are organized by category and where necessary, links are provided to relevant sections of the help file.

18.1 General Questions

- **What are the recommended graphics card settings?**

The recommend settings in the View Options dialog are:

Rendering Options: Hardware Z-buffer

Video Driver: OpenGL Driver

Driver Options: Double Buffering

- **If the MAESTRO auto-recovery functionality fails, can I still try to recover my model?**

Open the location where the original model was saved. You will see a *filename*.mdl and *filename*.bck. The *filename*.bck file can be opened directly by MAESTRO; this is your recovered modeler file.

- **How can I download the latest version of MAESTRO?**

The latest version of MAESTRO and the security driver can be downloaded from http://www.maestromarine.com/download.php. Note, you must have a valid maintenance and support expiration date to run the newest release of MAESTRO.

- **Where can I find additional support for MAESTRO?**

In addition to contacting technical support at support@maestromarine.com, you may visit the MAESTRO forum at MAESTRO Forum to add and review posts from MAESTRO users and technical support personnel.

- **Why aren't MAESTRO folders removed from Start > All Programs when MAESTRO is uninstalled?**

This is due to the autosave files that are added to the directory. These folders and menu items can be manually deleted after the uninstall process.

© 2014 Optimum Structural Design, Inc.

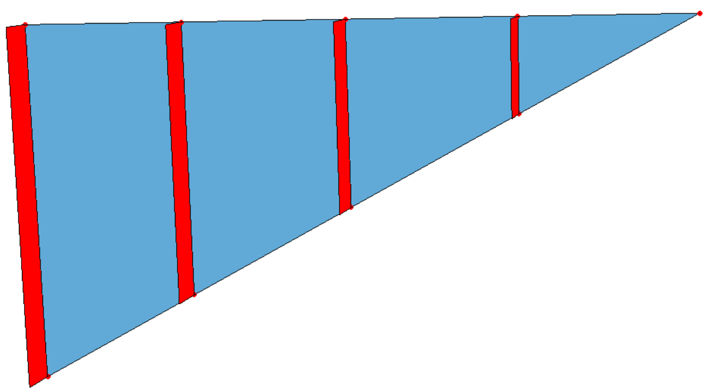
18.2 Pre-Processing

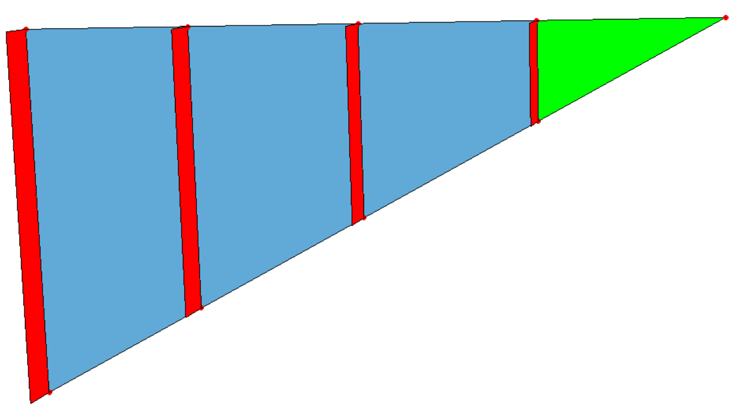
- **How do the stiffeners in a stiffener layout differ from a beam element?**

Stiffeners are defined as beam elements, but they are not actual finite elements in a coarse mesh model. Instead, they are treated as additional stiffness in the defined direction for the panel element they are defined on, thus converting the material to orthotropic. In a coarse mesh model, all stiffeners are treated as "internal" regardless of their defined location. The number of stiffeners, whether defined as internal or edge, is the key to how the mechanical properties of the panel element are changed. However, the location is relevant when creating a fine mesh model from the coarse mesh model. At this point, an actual beam element will be created representing the properties of the stiffener at the defined location

- **When two end points coincide at the reference or opposite end, is there a difference between using a strake triangle or an additional triangle element?**

No, in the case of a strake element with coinciding reference of opposite end points, the triangular element is degenerated to a simple triangle.





- **Do I need to define the "second strake" for girders when running the sparse solver?**

No. The second strake is used to identify the girder's effective breadth for the hybrid beam formulation in the scalable solver.

- **How is the additional stiffness treated when a quad or strake has a stiffener layout?**

The axial and bending stiffness of the stiffener(s) defined on that plate element are "smeared" into the panel's property. If an edge stiffener is defined, MAESTRO adds the axial and bending stiffness of the whole stiffener to the panel on which the stiffener is defined.

- **Can I sort the plate and beam properties in the strake definition dialog or the additional elements dialogs?**

Yes, element properties can be sorted in the properties dialog by name, thickness, etc.

- **How is a model balanced in MAESTRO? How does this compare to Orca3D, for example?**

MAESTRO is a force based balance. The equivalent forces are calculated for each panel element. Orca3D, however uses a volume/buoyancy based balance.

- **Can I hide particular substructures and modules?**

Using a right-mouse-click on a particular Part (via the Parts Tree), the user has the ability to make particular parts visible or invisible. This works well when you are turning the visibility of a few parts off, but is a little inconvenient when turning many parts off. When you are interested in turning the visibility off/on of many parts (substructures or modules), use the Visibility List command, which allows you to toggle the visibility on/off of Parts quickly.

1. Right-click on any part in the Parts Tree
2. Choose Visibility List, which opens a separate Parts Tree list
3. Left-click on a part to toggle the visibility on or off.

This is especially useful when the model becomes sufficiently large and many parts exist.

- **How do I model an I-beam element?**

MAESTRO can create an equivalent I-beam element by adding a second flange to an existing T-beam element. This second flange is defined in the Beams tab of the Finite Elements dialog, the Beam tab of the Compounds dialog, or a "second flange" strake can be created. The second flange is defined by a width and thickness. Please see the Second Flange section under Verification and Validation for further demonstration of this.

- **How do I reset zCF?**

© 2014 Optimum Structural Design, Inc.

1. Navigate to the balance tab in the load dialog
2. Click "Specify" under the Center of Flotation section.
3. In the Define Floating Center dialog, select "Do not Change Wave Emergence".
4. Change the CF Structure Coord and Click OK.
5. On the balance tab, set Center of Floatioan back to "Default" and click Modify.

- **Can I modify the property of more than one element at a time?**

Yes. You can create a general group of all the elements desired and modify the entire group's element's properties at once. For example, if you would like to change the hull thickness you can create a general group of all the hull plate elements. Right-click on the group in the groups tree and select Modify. In the Properties dialog that opens, select a new plate property from the drop-down list to change all plate elements to this property.

18.3 Post-Processing

- **With respect to stiffener tripping calculations, are the stiffeners defined in the stiffener layouts taken to be normal to the strake?**

Yes, MAESTRO's stiffeners are normal to the plate.

- **How does Yield Stress (as defined in the Materials dialog) affect the analysis?**

Yield stress only affects the analysis when using the limit state analysis.

- **What is the "Master Elements" option used for in the View Options dialog?**

Master elements are the parent elements of fine mesh elements. This can be used when a user wants to view the fine mesh results in the context of the global model by clicking View > All Modules. Turning "off" Master Elements allows the fine mesh elements to be more clearly seen.

- **When performing a natural frequency analysis, how do you view the participation factors and what do they indicate?**

A summary of the analysis can be seen in the output tab at the bottom of the screen. "ACU-%" is the accumulated modal participation factor. Modal effective mass and participation factor are relative measurement to the total vibration. For example, if the "ACU-%" is 99% in the first five modes, it implies the first five modes are dominant modes, and the rest of the modes can often be ignored. Modal effective mass and participation factors are often used in constraint vibration such as Dynamic Design Analysis Method (DDAM).

- **How do I eliminate local distortion of boundary modules when applying end**

© 2014 Optimum Structural Design, Inc.

moments?

End moments applied in MAESTRO are simply a combination of end nodal forces that collectively sum to the user defined moments. Unless additional local structure, for example a transverse bulkhead, is applied, we would expect to see local deformation.

- **How do I copy the data from the MAESTRO grid into a program such as Microsoft Excel?**

1. Right-click inside the grid and select "Change to Fixed Rows."
2. Change fixed row number from 1 to 0.
3. Highlight the desired data and right-click and select Copy.
4. Paste the data into the desired program.

18.4 Licensing and Security Device

- **How do I reprogram my hardware lock with new passwords?**

To reprogram a lock, please execute the following steps:

- Remove all other hardware locks from your computer. The MAESTRO hardware lock should be the only device attached to your computer for this operation.
- Run the MAESTRO Fast Lock program from your MAESTRO program group.
- Select: Edit/Set Passwords.
- Enter the new passwords and click OK
- The new support date should be shown

- **How do I transfer my license to another computer?**

MAESTRO licenses can be easily transferred between computers with the USB security device. Before plugging the security device into the new computer, verify that MAESTRO has been installed. After this, the security device can be plugged into the new computer and MAESTRO can be launched. Security drivers and network servers are integrated into the MAESTRO installation process. Therefore, if MAESTRO version 9.0 or newer is installed, licenses can be moved between computers by simply moving the security device.

- **How do I renew my maintenance and support contract or add modules to my license?**

To renew your maintenance and support contract, or to add additional modules to your license, please send an email to support@maestromarine.com with your contact information, license number and request for update or addition and you will receive a quote for these services.

- **I have installed MAESTRO 9.0 or newer; why doesn't my Sentinel System Driver and Protection Server show in the Add/Remove Programs dialog box?**

The Sentinel Driver and Protection Server are now part of the MAESTRO installation and therefore it is no longer a separate product.

- **Where can I get more information for troubleshooting issues with a Network lock?**

Please see the *ReadMe.pdf* file located in the MAESTRO installation directory under System > Sentinel for more information regarding MAESTRO network locks.

© 2014 Optimum Structural Design, Inc.

Appendices

19

19 Appendices

Enter topic text here.

19.1 A: References

1. Hughes, O.F., "Ship Structural Design," Society of Naval Architects and Marine Engineers, Jersey City, NJ, 1988.
2. Paik, Thayamballi, "Ultimate Limit State Design of Steel-Plated Structures," John Wiley & Sons, LTD, England, 2003
3. Mistree, F., Hughes, O.F., and Phuoc, H.B., "An Optimization Method for the Design of Large, Highly Constrained Complex Systems," Engineering Optimization, vol.5, no.3, August 1981.
4. Mistree, F., Hughes, O.F., Bras, B., "The Compromise Decision Support Problem and the Adaptive Linear Programming Algorithm", Structural Optimizaiton: Status and Promise, Kamat, M.P., ed., A.I.A.A., Washington, DC, 1993, pp. 247-286.
5. Liu, D., Hughes, O.F., and Mahowald, J.E., "Applications of a Computer-Aided Optimal Preliminary Ship Structural Design Method." Trans. SNAME, 1981.
6. Hughes, O.F., "A General Method for Computer-Aided Optimum Structural Design of Ocean Structures", (Banda and Kuo, eds.) North-Holland (Elsevier) Amsterdam, 1985, pp. 13-26.
7. Hughes, O.F., "Computer-Aided Optimum Structrual Design of Tension Leg Platforms," International Conference on Computer-Aided Design in Marine and Offshore Industries, Washington, D.C., 1986, (Keramidas and Murthy, eds.) Springer-Verdag, Berlin, 1986.
8. Bathe, K.J., "Finite Element Procedures in Engineering Analysis", Prentice-Hall Inc., Englewood Cliffs, New Jersey (1982).
9. Vernon, T.A., Bara, B., and Hally, D., "A Surface Panel Method for the Calculation of Added Mass Matrices for Finite Element Models", Defense Research Establishment Atlantic, Technical Memorandum 88/203 (February 1988).
10. MacNeal, R.H., "A Simle Quadrilateral Shell Element," Computers and Structures, Vol. 8, pp. 175-183, 1978.
11. MacNeal, R.H. and Harder, R.L., "A Fefined Four-Noded Membrane Element with Rotational Degrees of Freedom," Computers and Structures, Vol. 28, Vo. 1, pp. 75-84, 1988.
12. American Petroleumm Institute, Bulletin on Stability Design of Cylindrical Shells, API Bulletin 2U, May 1987.
13. Computer Science Department, University of Basel Switzerland, "Parallel Sparse Direct Solver PARDISO User Guide Version 3.2", pp. 3
14. Intel, "Intel Math Kernal Library (Intel MKL) 10.1, In-Depth", pp. 10
15. Richard H. MacNeal and Robert L. Harder, “A Proposed Standard Set of Problems to Test Finite Element Accuracy”, Finite Elements in Analysis and Design 1, pp. 3-20, 1985.

16. Cowper,G.,“The Shear Coefficient in Timoshenko’s Beam Theory”, Journal of Applied Mechanics, Vol. 33, 1966, pp. 335-340.

17. O.F.Hughes and Ming Ma, “Elastic Tripping Analysis of Asymmetric Stiffeners”, Computers and Structures, Vol. 60, No.3, 1996, pp 369-389

18. O.F. Hughes and Ming Ma, “Inelastic Stiffener Buckling and Panel Collapse”, Computers and Structures, Vol. 61, No. 1, 1997, pp 107-117

19. O.F. Hughes, "Two First Principles Structural Designs of a Fast Ferry - All-Aluminum and All-Composite", Fourth International Conference on Fast Sea Transportation, Volume I, pages 91-98, July 1998, Sydney, Australia

20.Ming Ma, Owen Hughes, Chengbi Zhao, "Applying Sectional Seakeeping Loads to Tull Ship Structural Models Using Quadratic Programming", International Conference on Maritime Technology (ICMT2012), June 2012

19.2 B: Data Prep Manual

For the legacy Data Preparation Manual, click here.

19.3 C: IDF Specification

INTERNATIONAL MARINE SOFTWARE ASSOCIATES

INTERFACE DEFINITION FILE (.IDF)

REVISION 3.03

5 May, 1997

The IMSA IDF is intended to be a neutral file format for exchange of hull description data between marine programs, without the generality or complexity of standards such as IGES and DXF, and without the specific traits of a particular program's native format.

The file is designed to be easily human-readable. Compactness is sometimes sacrificed for this goal.

3.01 NOTES

1. Revision 3.01 includes a new sectional AREA entity, at the request of the US Navy and other users.

2. All data tags (items preceded with $) must exist in the header, in the order and format given.

3. Following the $UNITS data tag must be a line that reads either SI or User Defined.

4. The HYDRO entity has been reduced to a subset of the ITTC computer symbols, called the

Interim Standard Transfer Set (ISTS). The list of supported terms is included in the description of the HYDRO entity. At their own risk, programs may output other ITTC values; however these are not strictly supported, and may or may not be read by other programs.

5. It is suggested that IDF interfaces be tested by trading files with other programmers who have IDF interfaces. Please contact the IMSA Technical coordinator above to arrange this.

3.02 NOTES

1. The General Form showed $COORDINATE SYSTEM preceeding $COMMENTS, while the specific entity defintions had these reversed. The specific entity definitions have been revised to be the same as the General Form.

3.03 NOTES

1. Added the PROPSECTS entity, for describing propeller geometry.

GENERAL FORM

$IDF

3.01 (or greater)

$ENTITY

entity type

$VESSEL NAME

identifier for this vessel

$DATA SOURCE

name of program that wrote the file

$DATE

date

$TIME

time

$UNITS

units

$COORDINATE SYSTEM

© 2014 Optimum Structural Design, Inc.

coordinates of a point one unit forward,starboard,down ("coordinate gnomon")

e.g. for FAST SHIP 1,1,1

$COMMENTS

comments

comments

$GEOMETRY

(data format specific to geometry type from here down)

$END ENTITY

Current Entity Types:

Entity Type	Description
SECTIONS	Sectional Data (Stations, Buttocks, Wls, 3d curves)
MESH	Surface Mesh data
NURBS	NURBS Surface data
HYDRO	Hull Parameter data
AREA	Sectional Area Data
PROPSECTS	Propeller Geometry Data

General Comments

This standard contains only one interface file. This file can contain one or more entities, where each entity is a specific data type (e.g. hull sectional data, NURBS surface data, etc.). This avoids having many different files, and allows new entities to be added as necessary. It also means that one file can contain different types of data for a single ship (sectional data, surface data, etc.), thus avoiding many files describing the same ship.

The file will be a simple ASCII file, so that it will be transportable across different hardware platforms, as well as being easily human-readable. While this does not result in the most compact format, it does result in a format that is easy to produce, read, add to, and modify.

Data for each line item are to be separated by commas. Comments may be added on any line following an exclamation mark (!). End of line sequence is to be appropriate to the operating system. Text strings may be up to 79 characters long, and are limited to ASCII characters 1 through 127.

© 2014 Optimum Structural Design, Inc.

Units

Units must be specified as either: SI or User Defined. If User Defined, then the following lines must be given:

# of user units/meter

# of user units/square meter

# of user units/cubic meter

# of user units/kg

Some entities may not require all of the conversion factors, and the entity's definition will specify which should be included.

Coordinate System

Since different programs use different coordinate systems (e.g. some have positive X aft, some have positive X forward, some use Z for the longitudinal coordinate, etc.), the coordinates of a fixed point in space is required. This point is one unit forward of the origin, one unit to starboard, and one unit down from the origin. Then, as data is read in from the file, by multiplying the data by the given vector and by your own vector, the sign will be correct. All data in the formats is given in the order longitudinal, athwartships, and height. Not all entities will have a coordinate system associated with them. If not, the entity definition will leave this section out.

Data Tags

Data tags (e.g. $ENTITY), while not absolutely required in a fixed format file, make the file easily human-readable, and can simplify the computer-reading process. Import programs that are searching for a particular ENTITY type, can search the file for the string "$ENTITY", and then read the next line to see if the type is correct, and go on from there.

Data tags (items preceeded with $) must have the $ in column 1, i.e. no white space is allowed before a data tag. Leading white space (tabs, spaces) is allowed on lines containing data. Blank lines are allowed between data and the next data tag.

Any data that is shown in the entity definitions is required; if not known, dummy data should be substituted.

Where entities allow for more than one body or surface, it is subdivided into parts (each part may represent a body or surface, or a group of bodies).

Entity #1: Sectional Data (SECTIONS)

Note: Indenting is for clarity only; not used in actual data file.

$IDF

3.01 (or greater)

$ENTITY

SECTIONS

$VESSEL NAME

Identifier for this vessel

$DATA SOURCE

program that wrote the file

$DATE

mm/dd/yy

$TIME

hh:mm:ss

$UNITS

This line must be either SI or User Defined

If User Defined, then the following line(s) must be specified:

# of user units/meter

$COORDINATE SYSTEM

coordinates of a point one unit forward,starboard,down ("coordinate gnomon")

e.g. for FAST SHIP 1,1,1

$COMMENTS

This is a comment about the ship about to be described. Can be any # of 79 character lines.

$GEOMETRY

n (number of parts or bodies)

part 1

© 2014 Optimum Structural Design, Inc.

.

.

part n

where each part format is:

$PART

part name

m (number of curves)

curve 1

.

.

curve m

where each curve format is:

$CURVE

curve name

Curve type (station, buttock, waterline, cant, incline,diagonal) diagonal, general plane, three-d)

j=integer number of points on curve

point 1

.

.

point j

where points are coordinate triplets (long'l, trans ,vert), breakpoint indicator (unknown, fair, knuckle)

for example: 10.15, 3.25, 1.50, fair

$END ENTITY

Entity #2: Surface Mesh Data (MESH)

Note: Indenting is used for clarity only; does not exist in actual file

$IDF

3.01 (or greater)

$ENTITY

MESH

$VESSEL NAME

Identifier for this vessel

$DATA SOURCE

program that wrote the file

$DATE

mm/dd/yy

$TIME

hh:mm:ss

$UNITS

This line must be either SI or User Defined

If User Defined, then the following line(s) must be specified:

# of user units/meter

$COORDINATE SYSTEM

coordinates of a point one unit forward,starboard,down ("coordinate gnomon")

e.g. for FAST SHIP 1,1,1

$COMMENTS

This is a comment about the ship about to be described. Can be any # of 79 character lines.
$COORDINATE

$GEOMETRY

© 2014 Optimum Structural Design, Inc.

n (number of parts or surfaces)

part 1

.

.

part n

where each part is

$PART

part name

#rows,#columns in surface mesh

long'l,trans,vert coords of mesh points: B(row,col) where col varies fastest

.

.

long'l,trans,vert coords

$END ENTITY

Entity #3: NURBS Surface Data (NURBS)

Note: Indenting is used for clarity only; not used in actual file.

$IDF

3.01 (or greater)

$ENTITY

NURBS

$VESSEL NAME

Identifier for this vessel

$DATA SOURCE

program that wrote the file

© 2014 Optimum Structural Design, Inc.

$DATE

mm/dd/yy

$TIME

hh:mm:ss

$UNITS

This line must be either SI or User Defined

If User Defined, then the following line(s) must be specified:

# of user units/meter

$COORDINATE SYSTEM

coordinates of a point one unit forward,starboard,down ("coordinate gnomon")

e.g. for FAST SHIP 1,1,1

$COMMENTS

This is a comment about the ship about to be described. Can be any # of 79 character lines.

$GEOMETRY

n (number of parts or surfaces)

part 1

.

.

part n

where each part is

$PART

part name

nonrational or rational

basis function in u,w directions (e.g. open or periodic) order in u,w directions (integers)

i,j (#rows,#columns in defining polygon net, integers)

knot vector in u direction (floating point)

knot vector in v direction (floating point)

long'l, trans, vert coords, weight of net points: B(i,j) where j varies fastest

.

.

long'l, trans, vert coords, weight (floating point)

$END ENTITY

Entity #4: Hull Parameter Data (HYDRO)

Note: Indenting is used for clarity only; not used in actual file.

$IDF

3.01 (or greater)

$ENTITY

HYDRO

$VESSEL NAME

Identifier for this vessel

$DATA SOURCE

program that wrote the file

$DATE

mm/dd/yy

$TIME

hh:mm:ss

$UNITS

© 2014 Optimum Structural Design, Inc.

This line must be either SI or User Defined

If User Defined, then the following line(s) must be specified:

# of user units/meter

# of user units/square meter

# of user units/cubic meter

# of user units/kg

$COORDINATE SYSTEM

coordinates of a point one unit forward,starboard,down ("coordinate gnomon")

e.g. for FAST SHIP 1,1,1

$COMMENTS

This is a comment about the ship about to be described. Can be any # of 79 character lines.

$GEOMETRY

n (number of parts or surfaces)

part 1

.

.

part n

where each part is

$PART

part name

entry 1

.

.

entry n

$END ENTITY

Each entry is an ITTC computer symbol from the Interim Standard Transfer Set (ISTS), a subset of the ITTC list of Standard Symbols and Terminology entered in the following form:

computer symbol=value

For example, the ITTC computer symbol for length of waterline is LWL. For a ship with a waterline length of 451.5, the data would be entered in the file as:

LWL=451.5

As many entries as desired may be made in this form using the ITTC ISTS standard computer symbols.

NOTES ON THE IDF HYDRO ENTITY

The 'HYDRO' entity of the IDF file is used to pass parametric information about hull geometry between programs. This information - length on waterline, displacement volume and trim, for example - is representative of a single vessel load condition.

The symbols are derived from ITTC computer symbols and terminology. In January 1993, the ITTC Symbols and Terminology Group identified the need for an Interim Standard Transfer Set (ISTS) as a subset to their comprehensive database-oriented collection of computer symbols. Dr. Bruce Johnson, Chairman, has asked IMSA to prepare and define the ISTS. The symbols used in the IDF HYDRO entity will form the ISTS.

To eliminate redundancy and potential confusion, the ISTS philosophy will be to use only those symbols based on geometric items (displaced volume, for example), rather than parameters (such as Cb). Also, a number of geometric and conversion references are defined.

Abbreviations used in this document:

FP - forward perpendicular.

Reference datum for the forward point of the length between perpendiculars (LPP).

AP - after perpendiculars.

Reference datum for the aft point of the length between perpendiculars (LPP).

MIDP - midship.

Located midway between FP and AP.

References:

International nautical mile = 6076.1155 feet, 1852.00 meters.

© 2014 Optimum Structural Design, Inc.

Gravitational constant, G = 32.1740 feet/sec^2, 9.80665 meters/sec^2.

Supported Symbols and Definitions:

ABT - total area of transverse cross-section of a bulbous bow. Full (port and starboard) cross sectional area at the FP.

AM - midship section area.

Immersed transverse sectional area located at MIDP.

APB - planing bottom area.

Horizontally projected planing bottom area (at rest), excluding area of external spray strips. (Area outlined by the chine as projected onto a horizontal plane.)

ATR - total area of immersed transom.

Full (port and starboard) cross-sectional area of a transom stern below the waterline.

AVL - longitudinal area exposed to wind.

Area of portion of ship above waterline projected onto a longitudinal plane (as viewed from the side).

AVT - transverse area exposed to wind.

Area of portion of ship above waterline projected onto a transverse plane (as viewed from ahead).

AW - area of the waterplane.

Area enclosed by the outline of the waterplane.

AX - maximum transverse section area.

Maximum immersed transverse sectional area.

BETD - principal deadrise angle of planing bottom.

Angle of the tangent slope of the planing bottom. (For a temporary solution, the tangent slope of the planing bottom at a point BPX/4 off the centerline, located at the mid-point of LPRC, is recommended.)

BETTR - deadrise angle of planing bottom at transom. Angle of the tangent slope of the planing bottom at the transom. (For a temporary solution, the tangent slope of the planing bottom at a point BTR/4 off the centerline, located at the aft-most point of LPRC, is recommended.)

© 2014 Optimum Structural Design, Inc.

BM - midship breadth on waterline.

Molded breadth on the waterline located at MIDP.

BPX - maximum breadth over chines.

Maximum breadth of the outside of the chine (excluding external spray strips).

BTR - breadth of the chine at the transom.

Breadth of the outside of the chine (excluding external spray strips) at the transom (aftmost point of LPRC).

BX - maximum breadth on waterline.

Maximum breadth of the waterplane.

DISV - displacement volume.

Immersed volume of the hull, neglecting appendages. (Large added volumes such as skegs may have a contribution to hull volume and there should be data agreement between SWH and DISV.)

ENTA - half angle of entrance.

Angle of waterline at the bow with reference to centerplane, neglecting local shape at stem. (For a temporary solution, the tangent slope of the waterplane at a point BX/10 off the centerline is recommended.)

LOS - overall submerged length.

Entire length of the submerged portion of the vessel, including items such as bulbs that extend beyond the limits of LWL.

LPP - length between perpendiculars.

Reference length that defines the distance between FP and AP.

LPRC - projected chine length.

Overall longitudinal length of chine projected onto a horizontal plane. (Longitudinal limit of APB.)

LWL - length of waterline.

Overall longitudinal length of the waterplane.

RHOW - mass density of water.

© 2014 Optimum Structural Design, Inc.

Standard ITTC values at 15 deg C, 59 deg F are:

Fresh: 1.9384 lb-sec^2/ft^4, 101.87 kg-sec^2/m^4

(specific gravity of 0.9990)

Salt: 1.9905 lb-sec^2/ft^4, 104.61 kg-sec^2/m^4

(specific gravity of 1.0259, 3.5% salinity)

Note: specific gravity uses international convention of distilled

water at 3.98 deg C (1.9403 lb-sec^2/ft^4, 101.97 kg-sec^2/m4).

SWH - wetted surface of the hull.

Entire immersed surface of the hull, neglecting appendages. (Large added volumes such as skegs may have a contribution to hull wetted surface and there should be data agreement between SWH and DISV.)

TM - draft at midship.

Molded hull draft on centerline, located at MIDP. Value reflects the principal hull volume and should not be confused with a keel draft that includes the effect of appendages or skegs.

TR - trim.

Vessel trim by the stern. Equals the draft at AP less the draft at FP.

XFB - longitudinal center of buoyancy from FP.

Longitudinal distance of the center of buoyancy aft of the FP.

XFG - Longitudinal center of gravity from FP.

Longitudinal distance of the center of gravity aft of the FP.

XLWL - location of length on waterline

Distance of the forward-most point of LWL aft of FP. (Registers location of LWL with respect to FP.)

XLPRC - location of projected chine length.

Distance of the forward-most point of LPRC aft of FP. (Registers location of LPRC with respect to FP.)

Entity #5: Sectional Area Data (AREA)

© 2014 Optimum Structural Design, Inc.

Note: Indenting is used for clarity only; not used in actual file.

$IDF

3.01 (or greater)

$ENTITY

AREA

$VESSEL NAME

Identifier for this vessel

$DATA SOURCE

program that wrote the file

$DATE

mm/dd/yy

$TIME

hh:mm:ss

$UNITS

This line must be either SI or User Defined

If User Defined, then the following line(s) must be specified:

# of user units/meter

# of user units/square meter

$COORDINATE SYSTEM

coordinates of a point one unit forward,starboard,down ("coordinate gnomon")

e.g. for FAST SHIP 1,1,1

$COMMENTS

This is a comment about the ship about to be described. Can be any # of 79 character lines.

$GEOMETRY

n (number of parts or surfaces)

part 1

.

.

part n

where each part is

$PART

part name

nsta

x1, a1

x2, a2

.

.

xnsta, ansta

where nsta is the number of stations cutting the part, xi is the ith station's longitudinal coordinate, and ai is it's immersed area.

$END ENTITY

Entity #6: Propeller Sectional Data (PROPSECTS)

Note: Indenting is for clarity only; not used in actual data file.

$IDF

3.03 (or greater)

$ENTITY

PROPSECTS

$VESSEL NAME

Identifier for this vessel (or job)

© 2014 Optimum Structural Design, Inc.

$DATA SOURCE

Program that wrote the file

$DATE

mm/dd/yy

$TIME

hh:mm:ss

$UNITS

This line must be either SI (meters) or User Defined

If User Defined, then the following line(s) must be specified:

# of user units/meter

$COMMENTS

This is a comment about the propeller about to be described. Can be any # of 79 character lines.

$GEOMETRY

rotation = Right or Left hand

number of blades , expanded blade area ratio , propeller diameter [dim], nominal pitch [dim], hub diameter [dim]

blade thickness ratio (may be zero), rake at tip (positive downstream) [dim]

n = number of sections

type = XY Offsets or Camber Thickness

section 1

.

section n

where each expanded section format is:

$SECTION

section name (e.g., 0.7R), radial position [dim], chord length [dim], maximum thickness [dim], location of max.

thickness from leading edge [dim]

local pitch [dim], distance of generator line to mid-chord (positive in the direction of the leading edge) [dim], rake

(positive downstream) [dim]

maximum camber [dim], location of max. camber from leading edge [dim]

j = number of radial sections

point 1 where points are ordered triplets of

. XY: chord position, ordinate suction side, ordinate press side [dim], or

. CT: chord position, ordinate of camber, thickness (normal to camber line) [dim]

point j

$END ENTITY

Comments

These files may contain any number of these entities. However, when writing to files, entities are typically appended to existing files. Therefore, only the last found entity is typically used when reading entities from files.

Example of PROPSECTS entity in units of meters (SI):

$IDF

3.03

$ENTITY

PROPSECTS

$VESSEL NAME

Sample propeller

$DATA SOURCE

PropCad 2.00

© 2014 Optimum Structural Design, Inc.

$DATE

03/25/97

$TIME

08:22:46

$UNITS

SI

$COMMENTS

Example of the PROPSECTS entity.

$GEOMETRY

Right

4, 0.65, 1.0, 0.9, 0.18

0.0702704, 0

10

XY Offsets

$SECTION

0.2R, 0.1, 0.253418, 0.0406, 0.0794288

0.72, 0.233917, 0.0140541

0.0203, 0.0794288

15

0.0, 0.0142046, 0.0142046

0.00633546, 0.0219650, 0.00923921

0.0126709, 0.0256052, 0.00684665

...

0.228076, 0.0102679, 0.0

© 2014 Optimum Structural Design, Inc.

0.240747, 0.00608771, 0.0

0.253418, 0.00182631, 0.0

$SECTION

0.3R

...

0.4R

...

...

$END ENTITY

© 2014 Optimum Structural Design, Inc.

License & Copyright

20

20 License & Copyright

Information included herein is categorized as ECCN 8D992 under the Export Administration Regulations (15 CFR § 730-774) issued by the U.S. Department of Commerce. An export license issued by the U.S. Department of Commerce or an EAR exception may be required prior to export or transfer of this information to certain parties or end uses. Public release of this document is not authorized. Diversion contrary to U.S. law is prohibited.

MAESTRO Software License Agreement

NOTICE: PLEASE READ THIS LICENSE AGREEMENT CAREFULLY BEFORE INSTALLING OR USING THE SOFTWARE. BY INSTALLING OR USING THE SOFTWARE, YOU ARE AGREEING TO BE BOUND BY THE TERMS AND CONDITIONS OF THIS LICENSE AGREEMENT. IF YOU DO NOT AGREE WITH THESE TERMS AND CONDITIONS, PROMPTLY RETURN THE UNUSED SOFTWARE.

1. RESTRICTED LICENSE. This License Agreement grants you (whether an entity or a person, hereinafter referred to as the "Customer") the non-exclusive, non-transferable, perpetual right to use MAESTRO (hereinafter referred to as the "SOFTWARE") for the stipulated License Fee.

2. LICENSE RESTRICTIONS. The SOFTWARE, along with all security features, shall be used only as stated herein. Customer may not transfer or assign to another party or location the rights under this License Agreement, the SOFTWARE, or any accompanying Documentation without Licensor's prior written consent. This license includes the right to use one copy of the SOFTWARE on any single computer, provided the SOFTWARE is only used on one computer at a time. The SOFTWARE is "in use" on a computer when it is loaded into temporary memory (RAM) or installed into the permanent memory of a computer (e.g., a hard disk, CD-ROM, or other storage device). Customer may not use the SOFTWARE for commercial time-sharing or rental use. Customer may make one (1) copy of the SOFTWARE solely for backup, archival or disaster recovery purposes. Customer may not modify, decompile, disassemble, reverse engineer, reverse translate or prepare derivative works of the SOFTWARE in whole or in part at any time for any reason.

3. COPYRIGHT. The SOFTWARE is owned by Optimum Structural Design, Inc. (hereinafter referred to as "Licensor") and is protected by U.S. copyright laws and international treaty provisions. This License Agreement does not transfer any ownership in the SOFTWARE, Documentation, trademarks or other Licensor proprietary property rights.

4. WARRANTY. Licensor warrants that it is the owner of the SOFTWARE and Documentation and that the SOFTWARE and Documentation do not infringe any U.S. patent, copyright or trade secret rights of any third party. Licensor further

warrants for a period of thirty (30) calendar days from the date of shipment that (i) the media on which a copy of the SOFTWARE is provided to Customer will be free from defects in material and workmanship under normal use, and (ii) the SOFTWARE will perform substantially in accordance with the Documentation.

5. CUSTOMER REMEDIES. Licensor's entire liability for breach of warranty and the Customer's exclusive remedy for breach of warranty shall be, at Licensor's option and expense, either (a) repair or replacement of the SOFTWARE or media that does not meet the warranty or (b) return of the price paid. This warranty is void if failure of the SOFTWARE has resulted from accident, abuse or misapplication from any party other than Licensor. **EXCEPT AS EXPRESSLY STATED IN ARTICLE 4, LICENSOR DISCLAIMS ALL OTHER WARRANTIES, WHETHER EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. NO AGENT, DEALER OR DISTRIBUTOR IS AUTHORIZED TO MODIFY OR REVISE THIS WARRANTY.**

6. **LIMITATION OF LIABILITY. IN NO EVENT SHALL LICENSOR BE LIABLE FOR ANY INDIRECT, INCIDENTAL, SPECIAL, PUNITIVE OR CONSEQUENTIAL DAMAGES, DAMAGES FOR LOSS OF BUSINESS PROFITS, BUSINESS INTERRUPTION, LOSS OF BUSINESS INFORMATION, LOSS OF DATA, OR LOSS OF GOODWILL ARISING OUT OF THE USE OF OR INABILITY TO USE THE SOFTWARE. NOTWITHSTANDING ANYTHING HEREIN TO THE CONTRARY, LICENSOR'S TOTAL LIABILITY TO CUSTOMER AND ANY THIRD PARTIES SHALL NOT EXCEED THE AMOUNTS PAID BY CUSTOMER FOR THE LICENSED SOFTWARE HEREUNDER.**

7. INDEMNITY. In the event that a claim is brought against Customer alleging that the SOFTWARE or Documentation infringes a U.S. patent, copyright or trade secret, Licensor agrees to defend such claim and to indemnify and hold Customer harmless for any damages or costs awarded against Customer. Customer shall give Licensor prompt written notice of any such claim, shall allow Licensor to control the defense and settlement of such claim, and shall reasonably cooperate with Licensor in the defense and settlement thereof. If an injunction or order is obtained against Customer's use of the SOFTWARE, or if, in Licensor's reasonable opinion, the SOFTWARE is likely to become the subject of a claim of infringement or violation of a U. S. patent, copyright, or trade secret, then Licensor will, at its option and expense: (i) procure for Customer the right to continue using the SOFTWARE; (ii) replace or modify the SOFTWARE so that it becomes noninfringing; or (iii) return the unamortized cost of the SOFTWARE to Customer based on a thirty-six (36) month pro rata schedule.

8. TECHNICAL SUPPORT/SOFTWARE MAINTENANCE. User Support and Upgrades are available through the *MAESTRO* Maintenance and Support Agreement, which can be purchased from Licensor or one of its designated

© 2014 Optimum Structural Design, Inc.

Support/Sales Dealers in accordance with the terms and conditions provided.

9. TERMINATION OF LICENSE. This License Agreement may only be terminated by Licensor upon providing at least ten (10) calendar days prior written notice to Customer of any violation or default of the terms of this License Agreement by Customer. Customer shall be allowed to remedy any such violation or default within the 10-day period to Licensor's reasonable satisfaction. Upon termination for Customer's default, all copies of the SOFTWARE and Documentation shall be immediately returned to Licensor or destroyed. If Customer destroys all SOFTWARE and Documentation, it will notify Licensor in writing that such destruction has taken place and that the SOFTWARE will not be used or reactivated by Customer in the future. Customer may terminate this License Agreement at any time by returning all SOFTWARE and Documentation to Licensor or destroying all SOFTWARE and Documentation and notifying Licensor as above. Any termination under this provision (whether by Licensor or Customer) shall be without rebate or any reduction in the amount owed or paid. Any cause of action or claim accrued or to accrue because of any breach or default shall survive termination of this License Agreement. Articles 2, 3, 4, 5, 6, 7, 12, 14, 15, and 16 shall survive any expiration or termination of this License Agreement.

10. TAXES. Any prices and fees for SOFTWARE or services exclude any applicable sales, use, excise, value added or other taxes or assessments which are or may hereinafter be levied or imposed by any federal, state, local or other public taxing authority, howsoever designated. Any such taxes or assessments (except for Licensor income taxes) levied or imposed as a result of this License Agreement shall be the Customer's obligation solely, and the Customer shall be responsible for all such payments.

11. FORCE MAJEURE. Except for any payment obligations, neither party shall be in default for any delay or failure to perform hereunder due to causes beyond its reasonable control and without its fault or negligence.

12. GOVERNING LAW/ARBITRATION. This License Agreement shall be governed by and construed under the laws of the State of New York, U.S.A., excluding its conflict of laws principles. Any dispute, controversy or claim, arising out of or relating to this License Agreement or a breach thereof, shall be finally resolved by arbitration. Such arbitration shall be the parties' exclusive remedy (except for cases of urgent equitable relief). The arbitration shall be in accordance with the rules of the American Arbitration Association (AAA) then in effect on the date of this Agreement by one arbitrator appointed in accordance with such rules. In the event of any conflict between the rules and this clause, the provisions of this clause shall govern. Unless the parties otherwise mutually agree in writing, the place of arbitration shall be New York, New York, U.S.A. The arbitration shall be conducted in the English language. The parties shall pay their own arbitration expenses and shall equally share the arbitrator's costs and fees and the arbitrator shall allocate such costs equally between the parties as part of the award.

13. LIMITED LICENSE FOR TRIAL RELEASE. Licensor hereby grants Customer a limited, non-exclusive, non-transferable right to use the SOFTWARE at no-charge for the limited purpose of evaluating whether to purchase the SOFTWARE. This trial release license contains the following requirements/restrictions: The SOFTWARE will run for fifteen (15) calendar days after installation. Upon the expiration of this 15-day period, the Customer shall either (i) purchase an ongoing license by paying the stipulated License Fee, which shall be subject to the terms and conditions of this License Agreement, or (ii) promptly remove or uninstall the SOFTWARE and return it to Licensor, or, at the Licensor’s option, destroy the SOFTWARE and certify to such destruction.

14. EXPORT COMPLIANCE. The SOFTWARE is subject to the United States Export Regulations, as administered by the Department of Commerce, Bureau of Industry Security. The SOFTWARE may not be used in or exported/re-exported to Cuba, Libya, North Korea, Sudan, Syria, or any country to which the United States embargoes goods. The same restriction applies to persons, no matter where they are located, that appear on the Table of Denial Orders, the Entity List, or the List of Specially Designated Nationals.

15. NOTICE TO U.S. GOVERNMENT END USERS. The SOFTWARE and Documentation are "commercial items," as that term is defined at 48 C.F.R. Part 2.101, consisting of "Commercial Computer Software" and "Computer Software Documentation," as such terms are defined in 48 C.F.R. Part 252.227-7014(a)(1) and 48 C.F.R. Part 252.227-7014(a)(5), and used in 48 C.F.R. Part 12.212 and 48 C.F.R. Part 227.7202, as applicable. Consistent with 48 C.F.R. Part 12.212, 48 C.F.R. Part 252.227-7015, 48 C.F.R. Part 227.7202-1 through 227.7202-4, 48 C.F.R. Part 52.227-19, and other relevant sections of the Code of Federal Regulations, as applicable, the Commercial Computer Software and Computer Software Documentation are distributed and licensed to U.S. Government end users (a) only as commercial items and (b) with only those rights as are granted to all other end users pursuant to the terms and conditions herein.

16. MISCELLANEOUS. If any portion of this License Agreement shall be held to be illegal or otherwise void and invalid, the remaining portion of the License Agreement shall not be affected and it shall remain in full force and effect. This License Agreement constitutes the exclusive and entire understanding between Licensor and Customer with respect to the SOFTWARE. This License Agreement supersedes any prior proposals, bids, quotes, representations, agreements, or any other understandings, whether oral or written, regarding the SOFTWARE or the relationship between Licensor and Customer, and may only be modified by a written agreement executed by authorized representatives of both parties. Licensor hereby rejects any additional or inconsistent terms and conditions offered by Customer at any time and irrespective of Licensor’s commencement of performance or shipment, or the acceptance of payment, hereunder. Purchase orders or other similar unilateral documents issued by the Customer shall be for the Customer’s internal use only and

© 2014 Optimum Structural Design, Inc.

shall not be binding on Licensor or otherwise affect or amend this License Agreement whatsoever.

09/24/12