MAESTRO
Ship Structural Design

DRS C3 Advanced Technology Center
149 Log Canoe Circle
Stevensville, MD 21666
MAESTRO Ship Structural Design

What is MAESTRO?

METHOD for

ANALYSIS

EVALUATION and

STRUCTURAL OPTIMIZATION
What is MAESTRO?

- ...is primarily a complete ship structural design system
- ...is primarily for design, but can be used to analyze existing structures
- ...provides a interactive graphical environment for structural design via FE modeling/analysis
- ...can model a variety of structures including monohull ships, multihull ships, offshore structures, submarines, foundations, etc.
- Ship Structural Design, Owen F. Hughes, Ph.D., SNAME
- Ultimate Limit State Design of Steel-plated Structures, Paik & Thayamballi
MAESTRO is a complete ship structural design system:

- Rapid Structural Modeling
- Ship-based Loading
- Finite Element Analysis
- Structural Evaluation
- Optimization (Scalable Solver)
- Fine mesh Analysis
- Natural Frequency
MAESTRO Ship Structural Design
Main Capabilities-Ship-based Loading
Obtain the stresses throughout the model for all defined load cases.
Evaluate the entire ship for all of the different possible failure Modes for all load cases.
Segregated-ballast Tanker

- Basis Design: 9708 Cost Units
- Large Scantlings (x3)
- Small Scantlings (x3)
- Optimized Design: 8477 Cost Units
- Standardizing Sections 8664 Cost Units 11% Cost Savings
Fully integrated fine mesh modeling and analysis capability. Also, ability to import FEMAP detailed models.
The 7200 hp escort tug “Response” experienced severe vibrations during builder’s trials.

The tug could not operate at its service speed.
Full-scale measurements showed the hull vibrating in the first mode (5.67 Hz) with several hinge points noted.
Model completed, from paper plans, in 3 weeks.
The eigenvalue analysis closely matched the full scale measurements (5.47 Hz) and mode shape.
The MAESTRO model was exported to Nastran and a forced vibration analysis was run using the engine propulsor excitation forces.

The analysis confirmed the vessel’s vibration problem was caused by the propulsor.
MAESTRO was used to re-design the tug until an acceptable change in the tug’s first mode frequency was reached. The re-design effort was conducted on-site in hours, not days or weeks.

**Bulwark was changed to a box beam girder, a shear strake double plate was added, and a keel doubler was added.**
MAESTRO is a complete ship structural design system:

- Rapid Structural Modeling
- Ship-based Loading
- Finite Element Analysis
- Structural Evaluation
- Optimization
- Detailed Stress Analysis
- Natural Frequency
1. **MODELING OF LOADS**

2. **STRUCTURAL RESPONSE ANALYSIS**
   - Calculate load effects, Q

3. **LIMIT STATE ANALYSIS**
   - Calculate limit values of load effects, Q_L

4. **EVALUATION**
   - (A) Formulate constraints
     - \( \gamma_1 \gamma_2 \gamma_3 Q \leq Q_L \)
   - (B) Evaluate adequacy

5. **OBJECTIVE**
   - Constraints satisfied?
   - Objective achieved?
     - Yes: STOP
     - No: **OPTIMIZATION**

6. **OPTIMIZATION**

   - **OBJECTIVE** (5)
   - **STOP** (6)

   - **Partial Safety Factors** \( \gamma_1 \gamma_2 \gamma_3 \)
   - **Other Constraints**

   - **All 6 are necessary**
   - **All 6 must be balanced and integrated**
Loads are ship-based and easy to apply:

- **Lightship mass distribution:**
- Hydrostatic loads:
  - Stillwater
  - Waves
- **Tank loads**
- **Cargo masses**
  - Forces
  - Moments
- **Accelerations (6 d.o.f.)**
- **Pressure loads**
- **External bending moments and shear**
  - force at ends of partial models
- **Boundary conditions**
- **Self weight mass**
- Scaled structural mass
  - Per section
  - Per module
  - Whole ship
- Individual masses
MAESTRO Ship Structural Design

R.B.D.-Modeling of Loads: Hydrostatic

- Still water
  - Height of WL above global reference point
  - Trim & Heel angle of waterplane

- Wave pressures
  - Sinusoidal or Trochodial
  - Wavelength
  - Amplitude
  - Phase angle & yaw angle
Hydrostatic loads are applied and the model is automatically balanced on the chosen wave or stillwater height.
Volume (tank) loads are applied as a percentage filled or a specific mass or a head.
R.B.D.-Modeling of Loads: Cargo masses (Nodal)

- Masses distributed (evenly) among nodes
- Large solid masses (masts, deck, cargo, etc.) with defined supporting nodes
Translation and rotational accelerations (with/without gravity)
  - Center of gravity
  - Center of flotation
  - Arbitrary point

This provides the inertial loads for all masses (lightship and cargo)
“Actual” Pressure

- Actual pressures can be constant or vary linearly across panels
- Specified as pressure, “LinPress” (positive or negative)
- Pressures resulting from a liquid mass with a designated specific gravity, either as a height above the bottom of the tank, fraction filled, or total mass (Volume loading)
- This pressure is part of the load matrix
“Design” Pressure

- Added to the panel after the FE solution/considered during evaluation
- Design pressures (additive/generic)
  - Additive – added during evaluation on top of any other pressure (e.g. ice loads)
  -Generic – are made the “lower bound” pressure on the specified panels during evaluation

![Loads interface](image)
• Apply flexural and torsional loads at the ends of the structural model
• Apply preliminary bending moment
The station values (user defined) are displayed and can be easily cut and pasted to MS-Word/Excel.
The station values (user defined) are displayed and can be easily cut and pasted to MS-Word/Excel.
**Restraints**
- Normal (6 d.o.f.) – rigid body motion
- Automatic centerplane (for half models) for symmetric or asymmetric loads

**Other BC (External Loads)**
- Vertical/horizontal BM and shear
- Torsional moment
MAESTRO Ship Structural Design

R.B.D.-Structural Response Analysis

1. MODELING OF LOADS

2. STRUCTURAL RESPONSE ANALYSIS
   CALCULATE LOAD EFFECTS, \( Q \)

3. LIMIT STATE ANALYSIS
   CALCULATE LIMIT VALUES
   OF LOAD EFFECTS, \( Q_L \)

4. EVALUATION
   (A) FORMULATE CONSTRAINTS
   \( \gamma_1 \gamma_2 \gamma_3 Q \leq Q_L \)
   (B) EVALUATE ADEQUACY
   CONSTRAINTS SATISFIED?
   OBJECTIVE ACHIEVED?

5. OBJECTIVE

6. OPTIMIZATION

   - All 6 are necessary
   - All 6 must be balanced and integrated

Partial Safety Factors \( \gamma_1 \gamma_2 \gamma_3 \)
Other Constraints

STOP
MAESTRO Ship Structural Design

R.B.D.-Structural Response Analysis

Individual modules are joined interactively to create the complete model.
MAESTRO Ship Structural Design

R.B.D.-Structural Response Analysis

Module Definitions
• Reference/Opposite Ends
• Section Spacing/Number
• Endpoints
• Strakes
• Stiffener Layout/Spacing

Reference End

Opposite End

Opposite End

Reference End

\[ \text{bbs} \]

\[ h \]

\[ t \]

\[ \text{b} \]

\[ n = \text{strake no.} \]

\[ n = \text{endpoint no.} \]

Section no.
Creating Modules: Endpoints (Nodes)

- Geometry via drawings
- FastShip
- Rhinoceros
- GHS
Creating Modules: Strakes (Elements)
- Strakes (combination of elements)
- Quads, triangles, etc.
- Compounds
- Scantling definition
- Stresses in stiffened panels:
  - Local bending of frames and girders
  - Plate and the stiffener flange
  - Combination of global and local loads
- Beam (frames/girders) moments and stresses:
  - Ends
  - Middle
Stresses are reported in the GUI; results can be queried and echoed to the output window.
Detailed stress results can also be exported to MS Excel, text file or the Grid.
The structural response analysis provides stress and deflection information about the entire vessel.
MAESTRO Verification Procedure

- QUAD4 and hybrid beam elements have been verified against theory and other FE codes (MSC-Nastran and ABAQUS)

- QUAD4 Verification
  - Tested against standard test problems published by MacNeal and Harder (“A Proposed Standard Set of Problems to Test Finite Element Accuracy”, Finite Elements in Analysis and Design 1, pp. 3-20, 1985)
    - Patch Test
    - Cantilever Beam Test
    - Curved Beam Test
    - Twisted Beam Test
    - Rectangular Plate Test
    - Scordelis-Lo Roof Test
  - The results show either similar or better level of accuracy as the results from Nastran or ABAQUS

- Beam element
  - MAESTRO obtains an exact solution for maximum displacement with two elements (the minimum possible)
  - MAESTRO obtains an exact solution for maximum bending moment with a single element

- Complete results are found in the MAESTRO Verification Manual
MAESTRO Ship Structural Design
R.B.D.-Limit State Analysis

1. MODELING OF LOADS

2. STRUCTURAL RESPONSE ANALYSIS
   CALCULATE LOAD EFFECTS, Q

3. LIMIT STATE ANALYSIS
   CALCULATE LIMIT VALUES
   OF LOAD EFFECTS, Q_L

4. EVALUATION
   (A) FORMULATE CONSTRAINTS
   \[ \gamma_1 \gamma_2 \gamma_3 Q \leq Q_L \]
   Partial Safety Factors \( \gamma_1, \gamma_2, \gamma_3 \)
   Other Constraints

5. OBJECTIVE

6. OPTIMIZATION

   • All 6 are necessary
   • All 6 must be balanced and integrated

   CONSTRAINTS SATISFIED?
   OBJECTIVE ACHIEVED?

   YES

   STOP

   NO
There are many types of limits…

- **Structural**
  - Plastic deformation
  - Instability
  - Tensile fracture
  - Fatigue

- **Other**
  - Fabrication, geometry, etc.

...and two levels

- Module (multi-member)
- Single member
Overall Structure
- Progressive collapse
- Fatigue
- Deflection limits
- Stress limits

Cylinder Collapse
- General buckling
- Bay buckling
- Local buckling

API Bulletin 2U
- Local shell buckling
- General instability
- Local stiffener buckling
- Bay instability
**MAESTRO Ship Structural Design**

**R.B.D.-Limit State Analysis-Member Level**

### PANELS
- **Collapse**
  - Stiffener flexure
  - Overall grillage buckling
  - Membrane yield
  - Stiffener tripping
  - Web buckling
- **Stiffener yield**
  - Compression, plate & flange
  - Tension, plate & flange
- **Plate unserviceability**
  - Yield
    - Transverse bending
    - Longitudinal bending
  - Local buckling
  - Allowable permanent set
    - Pressure loads
    - Concentrated loads

### GIRDER
- **Collapse**
  - Torsion buckling
  - Plastic buckling, flange
  - Plastic buckling, plate
- **Yield**
  - Compression, plate & flange
  - Tension, plate & flange

### FRAME
- **Collapse, plastic hinge**
- **Yield**
  - Compression, plate & flange
  - Tension, plate & flange
The formulation of MAESTRO's limit states is covered in Hughes, Ship Structural Design – A Rationally Based, Computer-Aided, Optimization Approach, published by SNAME.

An overview of all limit states is given in the MAESTRO's manual.

### Modes of Failure

<table>
<thead>
<tr>
<th>Name</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>PCSF</td>
<td>Panel Collapse - Stiffener Failure</td>
</tr>
<tr>
<td>PCCB</td>
<td>Panel Collapse - Combined Buckling</td>
</tr>
<tr>
<td>PCMY</td>
<td>Panel Collapse - Membrane Yield</td>
</tr>
<tr>
<td>PCSB</td>
<td>Panel Collapse - Stiffener Buckling</td>
</tr>
<tr>
<td>PITYF</td>
<td>Panel Yield - Tension, Flange</td>
</tr>
<tr>
<td>PITYP</td>
<td>Panel Yield - Tension, Plate</td>
</tr>
<tr>
<td>PYCF</td>
<td>Panel Yield - Compression, Flange</td>
</tr>
<tr>
<td>PYCP</td>
<td>Panel Yield - Compression, Plate</td>
</tr>
<tr>
<td>PSFBT</td>
<td>Panel Serviceability - Plate Bending, Transverse</td>
</tr>
<tr>
<td>PSFBL</td>
<td>Panel Serviceability - Plate Bending, Longitudinal</td>
</tr>
<tr>
<td>PFLB</td>
<td>Panel Failure - Local Buckling</td>
</tr>
<tr>
<td>GCT</td>
<td>Girder Collapse - Tripping</td>
</tr>
<tr>
<td>GCCF</td>
<td>Girder Collapse - Compression, Flange</td>
</tr>
<tr>
<td>GCCP</td>
<td>Girder Collapse - Compression, Plate</td>
</tr>
<tr>
<td>GYBF</td>
<td>Girder Yield - Bending, Flange</td>
</tr>
<tr>
<td>GYBP</td>
<td>Girder Yield - Bending, Plate</td>
</tr>
<tr>
<td>GYF</td>
<td>Girder Yield - Tension, Flange</td>
</tr>
<tr>
<td>GYTTP</td>
<td>Girder Yield - Tension, Plate</td>
</tr>
<tr>
<td>FCPH1,2,3</td>
<td>Frame Collapse - Plastic Hinge</td>
</tr>
<tr>
<td>FYCF,2,3</td>
<td>Frame Yield - Compression, Flange</td>
</tr>
<tr>
<td>FYTF,2,3</td>
<td>Frame Yield - Tension, Flange</td>
</tr>
<tr>
<td>FYCP,2,3</td>
<td>Frame Yield - Compression, Plate</td>
</tr>
<tr>
<td>FYTP,2,3</td>
<td>Frame Yield - Tension, Plate</td>
</tr>
<tr>
<td>CCBB</td>
<td>Cylinder Collapse - Bay Buckling</td>
</tr>
<tr>
<td>CGCB</td>
<td>Cylinder Collapse - General Buckling</td>
</tr>
<tr>
<td>CCGB</td>
<td>Cylinder Collapse - Local Buckling</td>
</tr>
</tbody>
</table>

The last three are used in strakes that form part of a cylinder; they replace PCSF, PCCB and PCSB respectively.

Figure 28 Modes of Failure Examined by MAESTRO
MAESTRO Ship Structural Design

R.B.D.- Evaluation

1. MODELING OF LOADS

2. STRUCTURAL RESPONSE ANALYSIS
   - CALCULATE LOAD EFFECTS, \( Q \)

3. LIMIT STATE ANALYSIS
   - CALCULATE LIMIT VALUES OF LOAD EFFECTS, \( Q_L \)

4. EVALUATION
   - (A) FORMULATE CONSTRAINTS
     \[ \gamma_1 \gamma_2 \gamma_3 Q \leq Q_L \]
   - (B) EVALUATE ADEQUACY

5. OBJECTIVE

6. OPTIMIZATION

   - CONSTRAINTS SATISFIED?
   - OBJECTIVE ACHIEVED?

   • All 6 are necessary
   • All 6 must be balanced and integrated

Partial Safety Factors \( \gamma_1 \gamma_2 \gamma_3 \)

Other Constraints

STOP
R.B.D.-Evaluation: Formulate Constraints

RESPONSE ANALYSIS

\[
\gamma \frac{Q}{Q_L} \leq 1
\]

LIMIT ANALYSIS

\[
\gamma_1 \gamma_2 \gamma_3 = \gamma
\]

Partial Safety Factors
Evaluation of the limit states is based upon the strength ratio:

\[ \frac{\gamma Q}{Q_L} = r \]

where \( \gamma = \gamma_1 \gamma_2 \gamma_3 \) (Partial Safety Factors)

The strength ratio can vary from zero to infinity, which is not useful for driving optimization, so we use an “adequacy parameter”...
The adequacy parameter, “g”:

\[
\frac{1 - r}{1 + r} = g
\]

This parameter varies from -1 to +1. Zero indicates that the structure, under the defined loads, is optimum for that particular limit state. Negative values indicate that the structure’s response, with the user defined safety factors, exceeds the limit state.
Evaluation is automatic - all structural members are evaluated to the factors of safety chosen by the user.

Different factors of safety can be specified for all “collapse” limit states and for all “serviceability” limit states, or specified on a limit state-by-limit state basis.

In addition to the strakes, frames, and girders which receive full evaluation...

- Additional panels, triangles, and additional beams receive limited evaluation,
- Struts and pillars are evaluated for Euler buckling
The entire structure can be viewed at one time...
...or only those members who have failed can be shown (negative adequacy)
Individual members can then be queried to determine their adequacy parameters and stresses. This information can be echoed to the output window.
R.B.D. - Optimization Objective

1. MODELING OF LOADS
2. STRUCTURAL RESPONSE ANALYSIS
   CALCULATE LOAD EFFECTS, Q
3. LIMIT STATE ANALYSIS
   CALCULATE LIMIT VALUES
   OF LOAD EFFECTS, Q_L
4. EVALUATION
   (A) FORMULATE CONSTRAINTS
   \( \gamma_1 \gamma_2 \gamma_3 Q \leq Q_L \)
   Partial Safety Factors \( \gamma_1 \gamma_2 \gamma_3 \)
   Other Constraints
5. OBJECTIVE
6. OPTIMIZATION

- All 6 are necessary
- All 6 must be balanced and integrated

Constraints Satisfied?
Objective Achieved?

NO
YES
STOP
MAESTRO Ship Structural Design

R.B.D. - Optimization Objective

WEIGHT

OBJECTIVE

COST
The cost function is adaptable to the user’s requirements:

<table>
<thead>
<tr>
<th>Fabrication Method</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>AUTOMATED (e.g., stiffened panels, rolled sections)</td>
<td>cost/volume, C1</td>
</tr>
<tr>
<td></td>
<td>cost/length, C2</td>
</tr>
<tr>
<td>NON-AUTOMATED (e.g., frames, girders)</td>
<td>cost/volume, C3</td>
</tr>
<tr>
<td></td>
<td>cost/length, C4</td>
</tr>
</tbody>
</table>

Cost/volume includes:
- plating cost
- transport
- handling

Cost/length includes:
- section cost
- welding
- painting

C1, C2, C3, & C4 can be varied for any part of the structure
**FUNCTIONAL**: $h_g < 0.5m$

*e.g. constraint on web height for overhead clearance*

**LOCAL**: $h_s < 30 \ t_w$

*e.g., local buckling of stiffener web*

**FABRICATION**: $h_s + 10 < 0.3 \ h_f$

*e.g., cutouts in frames*
The user defines the desired limits on the scantlings (left) as well as proportional limits on plating, stiffeners, and beams (above).
MAESTRO Ship Structural Design

R.B.D.- Optimization Objective

1. MODELING OF LOADS

2. STRUCTURAL RESPONSE ANALYSIS
   CALCULATE LOAD EFFECTS, $Q$

3. LIMIT STATE ANALYSIS
   CALCULATE LIMIT VALUES
   OF LOAD EFFECTS, $Q_L$

4. EVALUATION
   (A) FORMULATE CONSTRAINTS
   $\gamma_1 \gamma_2 \gamma_3 Q \leq Q_L$
   Partial Safety
   Factors $\gamma_1 \gamma_2 \gamma_3$
   Other
   Constraints

5. OBJECTIVE

6. OPTIMIZATION

   • All 6 are necessary
   • All 6 must be balanced
     and integrated

   CONSTRAINTS SATISFIED?
   OBJECTIVE ACHIEVED?

   NO

   YES

   STOP
Outer envelope of constant surfaces

constant cost

constant weight

slope = \frac{\text{weight saved}}{\text{extra cost}}
Examples of MAESTRO Users

---

**CLASSIFICATION SOCIETIES & SAFETY ORGANIZATIONS**
- American Bureau of Shipping
- Croatian Register
- Registro Italiano Navale (RINa)
- U.S. Coast Guard

**NAVIES**
- Australia, Brazil, Canada,
- Chile, Colombia, Germany,
- India, Italy, Japan, Mexico,
- Netherlands, New Zealand,
- Portugal, Turkey,
- United Kingdom, United States

**SHIYARDS**
- Australian Submarine Corp.
- Bath Iron Works
- Bender Shipbuilding
- Cascade General
- Northrop Grumman Shipbuilding
- Samsung Heavy Industries, South Korea
- Todd Pacific

**DESIGNERS & RESEARCH ORGANIZATIONS**
- Alion Science & Technology, USA
- Band Lavis Division of CDI Marine
- CETENA SpA, Italy
- CR Cushing & Co., USA
- BMT Designers & Planners, USA
- Downey Engineering Corp., USA
- Glosten Associates, USA
- Guido Perla & Associates, USA
- Navantia, Spain
- Keel Design Corp., USA
- MIT, USA
- Noise Control Engineering Inc., USA
- Rodriguez, Italy
- Toptech A.S., Norway
- VUYK, Netherlands
Applications of MAESTRO

- High Speed Ferries
- Warships
- SWATH Vessels
- Containerships
- Cruise Ships
- Offshore Support Vessels
- Tankers/Bulk Carriers
- Floating Dry Docks
- Barges
MAESTRO Ship Structural Design

100m Fast Ferry

Photo and model courtesy of Rodriquez Engineering, Genoa, Italy
MAESTRO Ship Structural Design

Canadian Patrol Frigate
MAESTRO Ship Structural Design

U.S. Navy AEGIS Cruisers
MAESTRO Ship Structural Design

Amphibious Assault Ship (LHD-1)
MAESTRO Ship Structural Design

SWATH Vessels: Cracking Investigation
MAESTRO Ship Structural Design

T-AGOR 26 (Kilo Moana)
MAESTRO Ship Structural Design

5500 TEU Containership
MAESTRO Ship Structural Design

Project America Cruise Ship
MAESTRO Ship Structural Design

Empress of the North Cruise Ship
MAESTRO Ship Structural Design

OSV Analysis
MAESTRO  Ship Structural Design
Pipe Laying Vessel Analysis
MAESTRO Ship Structural Design

Floating Dry Dock

Displacements: Load Case 1: Mississippi Docking, Phase 3
Working in a 3-D environment on a computer requires that you visualize an object drawn on a two dimensional computer screen.

This requires the user to manipulate the views and look at the model from different angles.

In addition to viewing the model’s geometry, we the user want to be able to view attributes of the model. Things like scantlings, loads, etc.

This is all accomplished using the Graphical User Interface (GUI). A closer look at the FEA process in the context of MAESTRO will follow.
MAESTRO Ship Structural Design

Graphical User Interface (GUI)
MAESTRO, as many 3-D modeling tools, is very mouse intensive.

It is very important to learn the mouse functionality.
<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
Changing the Model View

- **Standard Views** (right mouse click or via the View menu)
  - Bodyplan, Profile, Plan view
  - NorthEast, NorthWest
  - SouthEast, SouthWest

- **Spin, Pan, Zoom, Fit, Last** (right mouse click)

- **Heel, Pitch, Yaw View Angles**
  - HPY: 23.57, -14.11, -28.68

- **All view changing commands have no effect on the model geometry.**
MAESTRO Ship Structural Design

Displaying the MAESTRO Model

- Rendering Wire/Solid
- Nodes On/Off
- Shrink Elements
- Black/White
- View Options
MAESTRO Ship Structural Design

Displaying the MAESTRO Model

- Set View Part
- Set Current Part
- Parts Tree On/Off
- Output Window On/Off
- Groups Tree
MAESTRO Ship Structural Design

Displaying the MAESTRO Model

- Load case selection
- Launch Solver
- Dynamic Query
- Output Window On/Off
- Contour (deformation)
- Animation
Control Bars
This menu item allows the user to toggle any of the MAESTRO toolbars on or off.

Options
This menu item allows the user to control a wide variety of viewing options including element/node visibility, rendering algorithms, viewport layout, etc. Selecting this item opens the View Options dialog box.

Set View
This menu item allows the user to set the current viewing angles and view projection. Selecting this item opens a cascading submenu which allows the user to choose from a list of standard views or specify the view angles at the command line.

Set Window
This menu item allows the user to modify the current view parameters including zooming, panning, fitting the view, toggling to the previous view, changing the perspective distance, and storing and recalling views.

Cutting Planes
This menu item allows the user to create and delete cutting planes in the current view. 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.

Set View Part
This menu item allows the user to set the current view part in the active viewport.

System Sign
This menu item allows the user to toggle the system sign between plus and minus.
Element Type

This is the default view in MAESTRO, showing the default element colors:

- **COMPOUND** (dark green)
- **BEAM ELEMENT** (orange)
- **GIRDER** (yellow)
- **TRANSVERSE FRAME** (red)
- **QUAD ELEMENT** (light blue)
- **TRIANGLE ELEMENT** (light green)
- **FILLAR** (tan)
- **STRAKE PANEL** (dark blue)

**View Menu**

- Control Bars
- Options...
- Set View
- Set Window
- Cutting Planes
- Set View Part
- System Sign

**Element Type**

- Element Wettst
- Gray
- By ID
- Thickness
- Cost
- User Define
- Plate
- Edges
- Warped Quad
- Aspect Ratio
- Internal Angle
- Between Local X &

**Masters/Slaves**

**All Modules**

**Refresh**

**Right-mouse-click**
**Element Wetted**

The Wetted Elements view displays all elements that have been defined as "wetted".
By ID

This menu allows the user to view the model by Plate Property, Bar Property, Rod Property, Material, or Stiffener Layout.
**By ID**

This menu allows the user to view the model by Plate Property, Bar Property, Rod Property, Material, or Stiffener Layout.
By ID

This menu allows the user to view the model by Plate Property, Bar Property, Rod Property, Material, or Stiffener Layout.
By ID

This menu allows the user to view the model by Plate Property, Bar Property, Rod Property, Material, or Stiffener Layout.
By ID

This menu allows the user to view the model by Plate Property, Bar Property, Rod Property, Material, or Stiffener Layout.
MAESTRO Ship Structural Design

Displaying the MAESTRO Model: View Menu

- **Plate**
  This menu allows the user to view the model by **Element Pressure Side**, **Volume/Plate Pressure Side**, **Stiffener Side**, **Element Normal Side**, and **Corrosion Side**.
MAESTRO Ship Structural Design

Displaying the MAESTRO Model: View Menu

- **Plate**

  This menu allows the user to view the model by Element Pressure Side, Volume/Plate Pressure Side, Stiffener Side, Element Normal Side, and Corrosion Side.
Displaying the MAESTRO Model: View Menu

- **Edges**

  This menu allows the user to view the model by Free edges (any number of), 3 free edges, or 4 or more free edges.
Displaying the MAESTRO Model: View Menu

- **Warped Quad**
  This menu allows the user to view the model by Warped Quads.

- **Aspect Ratio**
  This menu allows the user to view the model by a specified Aspect Ratio range.

- **Internal Angle**
  This menu allows the user to view the model by a specified element edge Internal Angle.

- **Between Local X &**
  This menu allows the user to view the model between the local X axis and the Global X, Global Y, or Global Z.

- **Master/Slaves**
  This is currently under development.

- **All Modules**
  This menu allows the user to view the model by All Modules. This is useful when the MAESTRO project consists of global and fine mesh models.

- **Refresh**
  This command allows the user to refresh the graphics.
View Self Weight

The View Self Weight command under 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.
View Gross Weight

The View Gross Weight command under 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.
View Buoyancy

The View Buoyancy command under the Hull menu is used to display the FE model's buoyancy distribution for the selected load case, as shown below.
View Net Force

The View Net Force command under the Hull menu is used to display the FE model's net force distribution for the selected load case, as shown below.
View Shear Force

The View Shear Force command under the Hull menu is used to display the FE model's shear force distribution, as shown below.
**View Bending Moment**

The View Bending Moment command under the Hull menu is used to display the FE model's bending moment distribution, as shown below.
View Torsional Moment

The View Torsional Moment command under the Hull menu is used to display the FE model's torsional moment distribution, as shown below.
Displaying the MAESTRO Model: Hull Menu

- **View H. Net and Shear Force, Bending Moment**

  The View H. Net Force, H. Shear Force, and H. Bending Moment command under the Hull menu is used to display the FE model's horizontal net force, shear force, and bending moment distribution, as shown below.

  ![H. Net Force](image1)
  ![H. Shear Force](image2)
  ![H. Bending Moment](image3)
Show Properties

This menu item echoes all of the model section properties. Things like Area, Inertia, Neutral Axes, etc.
**View Element Long. Eff**

The View Element Long. Eff (longitudinally effective) command under the Hull menu is used to display structure that is "effective".
Displaying the MAESTRO Model: Hull Menu

- **View Izz and Iyy**
  The View Izz and View Iyy command under the Hull menu is used to display the FE model's inertia properties about the z-axis and y-axis respectively.

- **View Area**
  The View Area command under the Hull menu is used to display the FE model's area properties, as shown below.

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

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

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

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

The Weight Summary command under the Hull menu is used to produce weight summary tables in the Output window, as shown below.
MAESTRO is a finite element modeling and post-processing application that allows you to perform full-ship structural analyses both quickly and confidently.

- The FEA process can be broken down into the following stages:
  - Stage 1 – Geometry/Finite Element Modeling
  - Stage 2 – Checking the model
  - Stage 3 – Loading the model
  - Stage 4 – Analyzing the model
  - Stage 5 – Post-processing
  - Stage 6 – Documenting Results

- We will now look at how we can use MAESTRO to accomplish each step in the FEA process.
MAESTRO Ship Structural Design

Geometry/FE Modeling – Stage 1

- Reviewing the available structural and loading data is an important first step in planning the modeling effort.
MAESTRO Ship Structural Design

Geometry/FE Modeling – Stage 1

- Planning your model...
Imported sections to aid FE model construction (if available)…
Create a new MAESTRO model...
MAESTRO  Ship Structural Design

Geometry/FE Modeling – Stage 1

- Job Info...
Importing our IDF file (if available)…
Creating Parts...

- Frame 4 through Frame 9
- Location X=120
- Sections 4@30 inches and 1@33 inches
Endpoints…
- X, Y, Z
- Cartesian and Cylindrical
- Reference and Opposite
- 0, 20.25, 51, bilge, 43.5, deck@edge, 51, 0
MAESTRO Ship Structural Design

Geometry/FE Modeling – Stage 1

- Strakes...
  - General...
  - Plating...
  - Frames...
  - Girders...
  - Stiffeners...
  - Deletions...
MAESTRO Ship Structural Design

Geometry/FE Modeling – Stage 1

- Additional nodes
- Springs
- Rods
- Additional Beams
- Triangles
- Additional Quads
- RSplines
- Compounds
- Stiffener layout
- Materials
- Properties
- Delete
- Quick Creation
MAESTRO is a finite element modeling and post-processing application that allows you to perform full-ship structural analyses both quickly and confidently.

- The FEA process can be broken down into the following stages:
  - Stage 1 – Geometry/Finite Element Modeling
  - **Stage 2 – Checking the model**
  - Stage 3 – Loading the model
  - Stage 4 – Analyzing the model
  - Stage 5 – Post-processing
  - Stage 6 – Documenting Results

- We will now look at how we can use MAESTRO to accomplish each step in the FEA process.
Integrity check…

It is good practice to check the “integrity” of the model after completing a module.

The “integrity” of the model can be checked using particular items found in the View menu as well as the Model menu.
After a module has been completed it is usually advisable to make a “test run”, which serves to check the model.

This may require some further data: boundary conditions, loads and, if any loads involve acceleration, the definition of masses.

We will now discuss boundary conditions and constraints.
Constraints...

The General tab allows the specification of constraints (fixed nodal displacements and/or rotations) may be desired.

If only a small portion of the ship has been modeled, the user can define boundary conditions at the vessel’s “ends”. This is useful for preliminary evaluation of the ship’s structure.

Additionally, the user has the option of applying constraints to prevent nodes from moving in any of six degrees of freedom, X, Y, and Z translation, and rotations about the X, Y and Z axes.
- **Constraints…**
- **Constraining against rigid body motion…**
MAESTRO is a finite element modeling and post-processing application that allows you to perform full-ship structural analyses both quickly and confidently.

The FEA process can be broken down into the following stages:

- Stage 1 – Geometry/Finite Element Modeling
- Stage 2 – Checking the model
- Stage 3 – Loading the model
- Stage 4 – Analyzing the model
- Stage 5 – Post-processing
- Stage 6 – Documenting Results

We will now look at how we can use MAESTRO to accomplish each step in the FEA process.
Loading the model…

It is VERY important to understand the relationship between Groups and Loads in MAESTRO.

To understand this relationship, let us look at the figure below…

We will now take a close look at each Group.
Groups...

The Groups dialog is activated by clicking on the icon or by using the Model/Groups menu from the Main toolbar. The Groups menu consists of the items used to create, modify, and delete different types of groups. This is a multiple page dialog allowing the user to create groups by volume, plate, module (Scaled Mass), Section, Node, Bay, General, and Corrosion. A group is created interactively by selecting members with the mouse cursor. These members are displayed in the list box at the bottom of the groups dialog.

Upon completion of modeling the structure, it is necessary to model the weight distribution and other loading aspects…

“Groups” will aid in this task…
MAESTRO Ship Structural Design

Loading the Model – Stage 3

- VOLUME…
- Creating Volumes…
- Viewing Volumes…
- Normals and Pressure Sides…
- Loading Application…
The plate group provides a convenient way to apply a load to a collection of plate elements, for example, Deck Loading.

We will now digress for a moment to discuss:

- Mirroring, translating and rotating modules
- Issues to be aware of and how to handle them
- Planning the FE model

- Mirroring Modules (via Parts menu and right-mouse-click)
- Translating and Rotating Modules (Parts Dialog)
- Overlapped Elements
- Element normals and saving a step when creating Volume groups
- Patterns in creating an FE model (think ahead)
**MAESTRO Ship Structural Design**

**Loading the Model – Stage 3**

- **PLATE… (Back to Plate Groups)**
  The plate group provides a convenient way to apply a load to a collection of plate elements, for example, Deck Loading.
- **Creating Plate…**
- **Plate Mass…**
- **Plate Pressure…**
- **Loading Application…**
- **Open 270-WMEC_train.mdl …**
MAESTRO Ship Structural Design

Loading the Model – Stage 3

- MODULE…
  - This group is used to define a 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.

- View Weight Distribution…
- Create Module Group…
- Loading Application (matching lightship)…
- View Weight Distribution…
MAESTRO Ship Structural Design

Loading the Model – Stage 3

- SECTION...
  - A sections group is used to define a 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.

- Create Section Group...
- Loading Application (matching lightship)...
- View Weight Distribution...
MAESTRO Ship Structural Design

Loading the Model – Stage 3

- **NODE…**
  - A nodes 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.

- Create Node Group…
- Loading Application…
- View Weight Distribution…
MAESTRO Ship Structural Design

Loading the Model – Stage 3

- GENERAL…
  - 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 to create a General group which can then be refined for fine meshing.
- Create General Group…
- Run an analysis…
- View Results…
- Set General Group as Current…
- Results/View List Elements…
Now that we have looked at Groups, let us again look at the figure below.

We will now take a close look at each loading aspect.
MAESTRO Ship Structural Design

Loading the Model – Stage 3

Loads...

- The Loads dialog is activated by clicking on the icon or by using the Load/Create Load menu from the Main toolbar.
- 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 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 General tab serves as a central location for the user to select particular loading options as well as view the current status of the particular load case.

The user will find the behavior of the load case dialog is such that some option tabs are only activated upon first invoking its corresponding option check box.

- Load Safety factor…
- Options on Masses & Accelerations…
- Current Status from Other Menus…
- Advanced MAESTRO Solver Settings…
NOTE:

- In non-SI systems of units that specify densities in force units, the masses should be defined in terms of weight. Throughout this section the word "mass" should be taken to read "mass (or weight)" with the latter applying for non-SI systems. For non-SI systems the program internally converts from weight to a consistent mass unit (weight divided by gravity). Therefore in the program output (when applicable) the word "mass" should again be taken to read "mass (or weight)" with the latter applying for non-SI systems. Some exceptions occur if you specify a high level of program output (again, when applicable) because then the program will be printing the values that it is actually working with, which are the consistent mass values.

- In a half model all specified values of mass should be half values. At present, for a half model, all masses (except for Bay Set) are assumed to be symmetric and hence there is no need to define the corresponding mass in the un-modeled half. Even if a mass is used in an unsymmetrical load case, the program will assume that there is an equal mass in the un-modeled half. An exception to this is the definition of container (or other) masses in terms of bays and sets, for which it is possible to have different sets in the modeled and un-modeled halves.
**MAESTRO Ship Structural Design**

**Loading the Model – Stage 3**

- **Mass...**
  - 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
    - as large solid masses whose centroid is at an appreciable distance from the supporting nodes, such as main engines and bays of containers.
  - MAESTRO will use these values to calculate all of the inertia forces in all members throughout the structure, and apply these as loads.
Point Force...

- The Point Force group defines the location and orthogonal components of the point loads acting on the structure in the current load case.
- A point load may be either a force (with up to 3 components) or a moment (with up to 3 components) or both.
- 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 half values.
Loading the Model – Stage 3

Pressure...

- Six methods of defining the location and magnitude of pressure loads which are to be applied to the panels (quadrilaterals and triangles) of this module in the current load case.
- All pressure loads are cumulative.
- 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.
LinPress...

- The pressure is prismatic along the strake, within the length of the specified range of sections, regardless of the orientation of the strake.
- LinPress indicates an “actual” pressure, for which the nodal forces and moments will be calculated and applied to the finite element model.
- Here the pressure varies linearly across the width of the strake.
- The user can define the pressure along the strake edge 1 and strake edge 2.
The surface option is intended for hydrostatic pressure, for which the value is proportional to the depth below the free surface of a fluid.

In the Surface 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 for 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.
Automatic balance...

After defining the initial emergence values in a particular load case, the user should select Modify and then close the Loads dialog before invoking the modeler Load Balance command, via the balance icon found in the top icon bar. Selecting this icon will open the balance dialog shown below. Here the user can define convergence criteria as well as the number of iterations. If the user selects the User Control, as shown below, adjustments to the Center of Flotation and Heel/Trim Angles can be made.
MAESTRO is a finite element modeling and post-processing application that allows you to perform full-ship structural analyses both quickly and confidently.

- The FEA process can be broken down into the following stages:
  - Stage 1 – Geometry/Finite Element Modeling
  - Stage 2 – Checking the model
  - Stage 3 – Loading the model
  - Stage 4 – Analyzing the model
  - Stage 5 – Post-processing
  - Stage 6 – Documenting Results

- We will now look at how we can use MAESTRO to accomplish each step in the FEA
Running a MAESTRO analysis (Com-solver v. Scalable solver)

- The traditional MAESTRO solver (referred to as the MAESTRO Scalable solver because the arrays are automatically scaled to the required size) requires the creation of a data file, called jobname.DAT.
- Traditionally, modeler's sole purpose was to create a MAESTRO data input file that was submitted to the solver at the time of analysis.
- With the introduction of the MAESTRO Version 8.5, came MAESTRO COM Solver, the cornerstone of the next generation of MAESTRO solvers.
- With the inception of the COM Solver, there is no need to create a data file, assuming the user intends to use the COM Solver to analyze the structural system.
- MAESTRO COM Solver DOES NOT perform the calculation of limit states and the subsequent evaluation of these limit states. This area ranks very high on the development priority list but currently are not supported by COM Solver.
- Please see the Appendix B: Data Preparation Manual for a complete description of the *.DAT file.
Running a MAESTRO analysis…

Once the loads appear to be correct, you are ready to perform a finite element analysis of the current model. In the Job Info option under the File menu, select the extent and levels for the calculation of stresses and for the evaluation of structural adequacy (for Scalable Solver only). Then run MAESTRO, which will now calculate the deformations, the stresses and the adequacy parameters (Scalable Solver) in some or all of the members. Then use Modeler to plot the deformed shape and to obtain color-added displays of the stresses and adequacy parameters. Check the results carefully to see if there are any inconsistencies and if so, whether these are due to errors in the model or in the loads.
MAESTRO is a finite element modeling and post-processing application that allows you to perform full-ship structural analyses both quickly and confidently.

The FEA process can be broken down into the following stages:

- Stage 1 – Geometry/Finite Element Modeling
- Stage 2 – Checking the model
- Stage 3 – Loading the model
- Stage 4 – Analyzing the model
- **Stage 5 – Post-processing**
- Stage 6 – Documenting Results

We will now look at how we can use MAESTRO to accomplish each step in the FEA process.
Post-processing—Stage 5

MAESTRO provides a large number of Pre and Post-Processing viewing options that help to make the FEA process easier. These viewing options can be divided into five general categories and are found in MAESTRO’s main menu. They are the View, Restraints, Load, Hull, and Result menus. In combination with the Dynamic Query functionality, the user can interact with these menus to increase FEA productivity, verify model properties, and review analysis results.

- Load selection...
- Black/white...
- Animation...
- View Options...
- Gray On/Off...
- Dynamic query...
- Contour plot...

These are typically used in the post-processing of the model...
MAESTRO Ship Structural Design

Post-processing – Stage 5

- Load selection...
- Dynamic query...
- Local X...
- Local Y...
- Local XY...
- von Misses...
MAESTRO  Ship Structural Design

Post-processing – Stage 5

- Minimum Value (Plate)...
- Minimum Value (Beams)...
- Refer to documentation...

<table>
<thead>
<tr>
<th>All Load Cases</th>
</tr>
</thead>
<tbody>
<tr>
<td>Minimum Value (Plates)</td>
</tr>
<tr>
<td>PCSF (Collapse, Stiffener Flexure)</td>
</tr>
<tr>
<td>PCSB (Collapse, Combined Buckling)</td>
</tr>
<tr>
<td>PCCY (Collapse, Membrane Yield)</td>
</tr>
<tr>
<td>PCSB (Collapse, Stiffener Buckling)</td>
</tr>
<tr>
<td>PVTF (Yield, Tension in Flange)</td>
</tr>
<tr>
<td>PVTP (Yield, Tension in Plate)</td>
</tr>
<tr>
<td>PVCF (Yield, Compression in Flange)</td>
</tr>
<tr>
<td>PVCP (Yield, Compression in Plate)</td>
</tr>
<tr>
<td>PSCP (Serviceability, Plate Sliding)</td>
</tr>
<tr>
<td>PVSL (Serviceability, Plate Sliding Long)</td>
</tr>
<tr>
<td>PFLB (Failure, Local Buckling)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Minimum Value (Beams)</th>
</tr>
</thead>
<tbody>
<tr>
<td>GCT (Tension)</td>
</tr>
<tr>
<td>GCCP (Collapse, Compression in Plate)</td>
</tr>
<tr>
<td>GCCF (Collapse, Compression in Flange)</td>
</tr>
<tr>
<td>GEFYCF (Yield, Compression in Flange)</td>
</tr>
<tr>
<td>GEFYCP (Yield, Compression in Plate)</td>
</tr>
<tr>
<td>GEFYTF (Yield, Tension in Flange)</td>
</tr>
<tr>
<td>GEFYTP (Yield, Tension in Plate)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Minimum Value (Plates &amp; Beams)</th>
</tr>
</thead>
<tbody>
<tr>
<td>- (Negative)</td>
</tr>
<tr>
<td>+ (Positive)</td>
</tr>
</tbody>
</table>
- Deform…
- Dynamic query of nodes…
MAESTRO is a finite element modeling and post-processing application that allows you to perform full-ship structural analyses both quickly and confidently.

The FEA process can be broken down into the following stages:

- Stage 1 – Geometry/Finite Element Modeling
- Stage 2 – Checking the model
- Stage 3 – Loading the model
- Stage 4 – Analyzing the model
- Stage 5 – Post-processing
- Stage 6 – Documenting Results

We will now look at how we can use MAESTRO to accomplish each step in the FEA process.
Screen Capture…
Dynamic Query echo…

- Advanced Modeling Techniques
- Advanced Topics
  - Fine Meshing
  - Eigenvalue Analyses
  - Importing/Exporting FEMAP
  - Importing FastShip (idf)
- Review and questions
MAESTRO Ship Structural Design
Advanced Modeling Techniques

- Merging two models…
- Importing DXF files…
- Importing/Exporting FEMAP…
- Importing IDF files…
- Eigenvalue Analyses…
- Fine Meshing…
  - R-Splines
  - Exporting to FEMAP
MAESTRO Ship Structural Design

Review and Questions

- Review…
- Questions…
MAESTRO Ship Structural Design

MAESTRO Documentation/Tech Support

- **Documentation…**
  - MAESTRO help manual can be accessed via the Help/Contents menu item.

- **Technical Support**
  - Email: maestrosupport@orca3d.com
  - Fax: +1 (410) 643-5370
  - Telephone: +1 (410) 604-8000