

# **CONSTRUCT**

## **Conceptual Structural Design Software**

REVISION 1.0 (24.01.2010)

## TABLE OF CONTENTS

1	INTRODUCTION .....	4
	1.1 General.....	4
	1.2 Operational principles .....	4
	1.3 Modelling principles.....	5
	1.4 Architecture of software .....	6
2	SYSTEM REQUIREMENTS .....	6
	2.1 Hardware Requirements .....	6
3	SOFTWARE REQUIREMENTS.....	7
4	INSTALLATION .....	7
5	GRAPHICAL USER INTERFACE .....	8
	5.1 General.....	8
	5.2 Menu bar .....	8
	5.2.1 File Menu .....	8
	5.2.2 View menu .....	9
	5.2.3 Model menu .....	10
	5.2.4 Response menu.....	12
	5.2.5 Design Criteria menu .....	12
	5.2.6 Optimization menu .....	13
	5.2.7 Options menu.....	14
	5.2.8 Window menu .....	14
	5.2.9 Help menu.....	14
	5.3 <b>Toolbar area</b> .....	14
	5.3.1 Grid point and Section line tools .....	15
	5.3.2 Selection tool .....	15
	5.3.3 Drop Down menus .....	15
	5.4 <b>Project Explorer and Data Table areas</b> .....	16
	5.4.1 General .....	16
	5.4.2 Groups .....	16
	5.4.3 Elements.....	17
	5.4.4 Panels.....	17
	5.4.5 Openings.....	18
	5.4.6 Profiles .....	18
	5.4.7 Materials .....	19
	5.4.8 Model .....	19
	5.4.9 Full Model .....	23
	5.4.10 LCs&BCs .....	23
	5.4.11 Loads .....	23
	5.4.12 Cost.....	25
	5.4.13 Optimization .....	25
6	WORKING WITH CONSTRUCT .....	26
	6.1.1 General .....	26
	6.2 <b>Step by Step Modeling</b> .....	27
	6.2.1 Loading datasheets.....	27
	6.2.2 Creating a Model.....	27
	6.2.3 View manipulation and quick selection.....	30
	6.2.4 Adding structural entities to the model .....	31
	6.2.5 Stiffeners.....	31

6.2.6	Girders (and Frames).....	31
6.2.7	Pillars .....	32
6.2.8	Openings.....	32
6.2.9	Creating TStrakes .....	33
<b>6.3</b>	<b>Creating a full model of a ship .....</b>	<b>34</b>
6.3.1	Copying sections.....	34
6.3.2	Assembling sections into a full model .....	34
6.3.3	Generating Coupled Beams.....	34
6.3.4	Creating Groups.....	35
<b>6.4</b>	<b>Analyzing the model .....</b>	<b>35</b>
6.4.1	General .....	35
6.4.2	Creating and applying loads.....	36
6.4.3	Evaluating global response .....	36
6.4.4	Strength analysis based on global and local response .....	37
6.4.5	Running optimization process .....	37
6.4.6	Review of optimisation results.....	39
7	EXAMPLE MODELS.....	40
8	REFERENCES .....	41
9	APPENDIXES.....	42
<b>9.1</b>	<b>Datasheet file examples.....</b>	<b>42</b>
<b>9.2</b>	<b>Design criteria .....</b>	<b>43</b>
9.2.1	Plate yield .....	43
9.2.2	Stiffener yield .....	43
9.2.3	Plate buckling.....	44
9.2.4	Stiffener lateral buckling.....	44
9.2.5	Stiffener torsional buckling (tripping).....	44
9.2.6	Stiffener's web and flange buckling.....	45
<b>9.3</b>	<b>Napa steel macro.....</b>	<b>46</b>

# 1 INTRODUCTION

## 1.1 General

This document represents User's Guide for **Conceptual Structural Design Software** or CONSTRUCT. It enables the efficient structural analysis of new ship concepts using Coupled Beam theory. CONSTRUCT is focused especially on the conceptual design phase and the software enables fast generations of ship sections and assembling them into a full ship model. In addition to it, CONSTRUCT has built-in support for various ship design criteria such as buckling, yielding as well as structural optimisation. First part of this manual explains the operational principles of CONSTRUCT, the Graphical User Interface (GUI), CONSTRUCT elements and functions. The next chapter will introduce user to the working philosophy of CONSTRUCT Tool through the demonstration example.

## 1.2 Operational principles

The design procedure for CONSTRUCT is presented in Figure 1. The starting point of the conceptual structural design is the information of previous design steps, where for instance general arrangement and hull shape are created. Based on this information the steel general arrangement, loads and structural discretion for CB-analysis are determined. These initial definitions are done once, since scantlings with stiffener spacing are only varied during structural optimisation. During the optimisation process design alternatives are stored to have the wide coverage of the objective space, which is exploited for the creation of Pareto surface. CONSTRUCT allows manual design generation or automated optimisation. CONSTRUCT is composed of a central unit, ship model database and independent calculation modules. The central unit controls the database and the calculation modules, which include all methods required for techno-economical analysis.

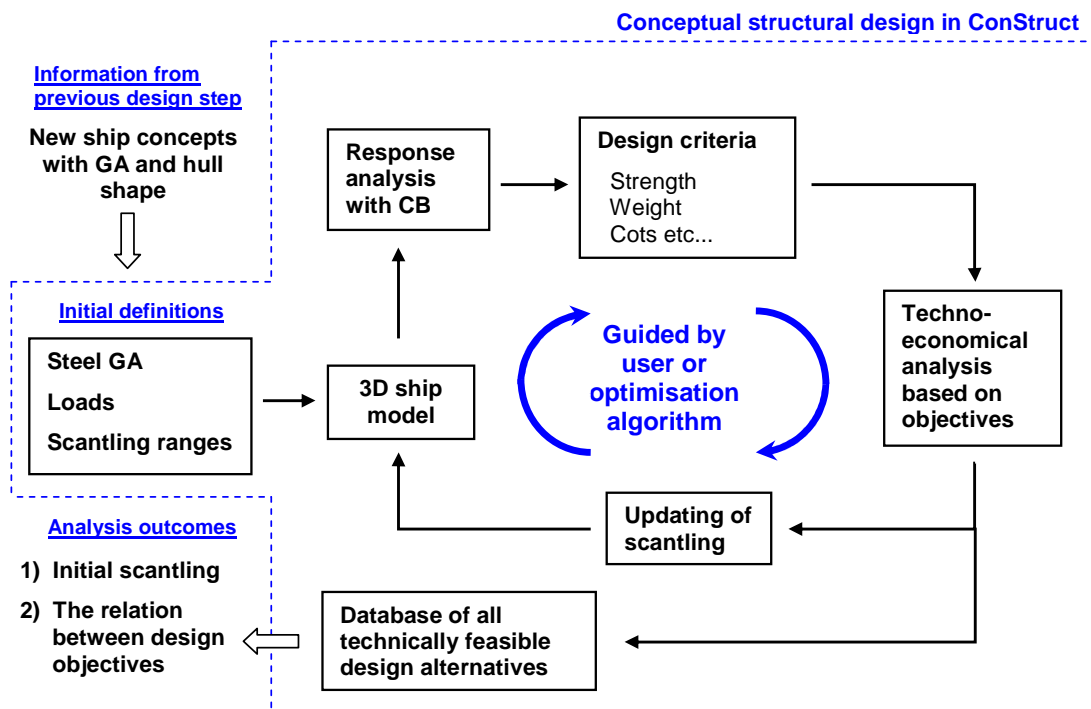


Figure 1: Design procedure for CONSTRUCT.

### 1.3 Modelling principles

The efficient utilisation of the pre-information in the iterative design process affects on structural modelling. In CONSTRUCT, the hull girder of the ship (FullModel) is built up one or more sections (Section) making it possible to study prismatic or non-prismatic hull beam problems, see Figure 2. Thus, in the iterative design process this means that the first evaluation of strength is based on the prismatic model while the more accurate analysis in next design step can rely on the non-prismatic model. For instance, to include the influence of a large deck opening or staircase the application of the non-prismatic model is necessary. Figure 3 presents the example of Section model in CONSTRUCT. The section is composed of macro-elements, which are stiffened plate (Strake), deck beam (Girder/Frame) and pillar (Pillar). Definition of each macro-element includes the information about its location, geometry and property. The geometry and property of macro-element are obtained from the pre-defined panel, material and profile tables included in CONSTRUCT. The panel is the main unit, which includes the definition of stiffener spacing, stiffener type, size of panel and plate thickness, and the panel is applied to model Strakes of the sections.

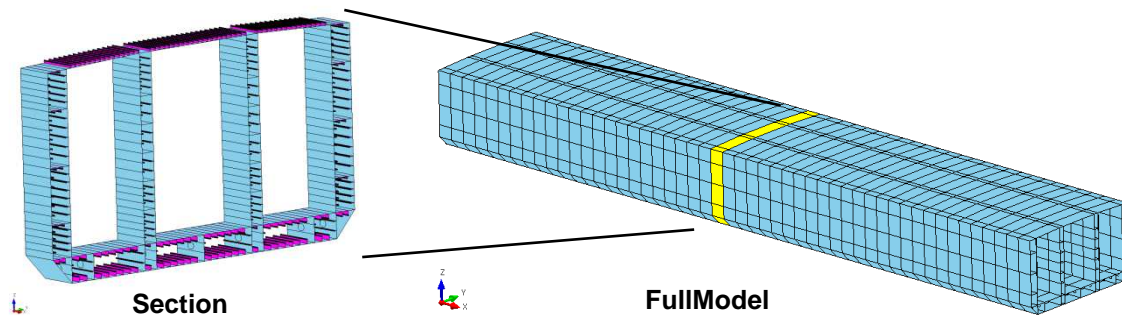


Figure 2: A section (Section) and full ship model (FullModel).

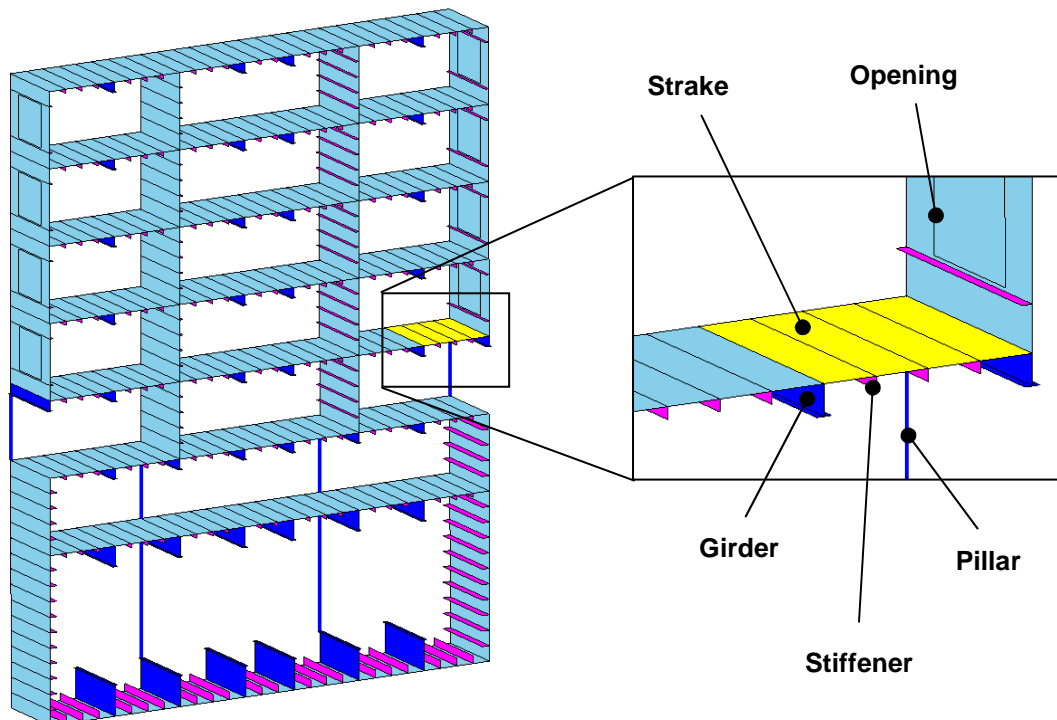


Figure 3: Example of a section model composed of Strake, Girder/Frame, Stiffener, Pillar, and Opening.

## 1.4 Architecture of software

The CONSTRUCT software is composed of a central unit (CONSTRUCT engine), ship model database and independent calculation modules; see Figure 4. The central unit controls the database and the calculation modules with the help of a functional library. Model database and calculation modules are controlled by user via Graphical User Interface (GUI).

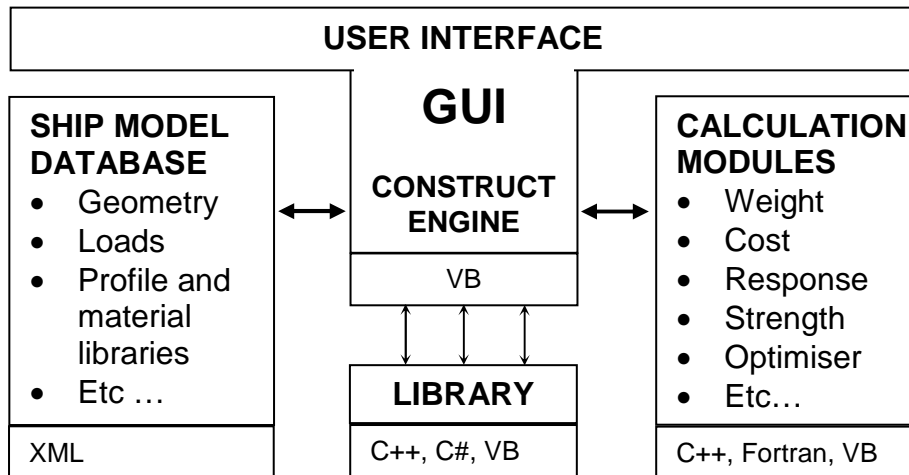


Figure 4: Architecture of CONSTRUCT software.

## 2 SYSTEM REQUIREMENTS

### 2.1 Hardware Requirements

Minimum hardware requirements to run CONSTRUCT depend on operating system one is running. These are listed below for most of the operating systems (values in brackets are recommended).

#### Windows XP family

- 600 MHz processor (1 GHz)
- 128 MB of RAM (512 MB)
- 50 MB of available hard disk space (100 MB)
- VGA with 32 MB RAM, supporting 32 bit color and minimum resolution of 1024x768 pixels including appropriate monitor (VGA with 128 MB RAM and resolution of 1400x1050 and higher, wide screen is recommended)

#### Windows Vista

- 1 GHz processor (2 GHz)
- 1 GB of RAM (2 GB)
- 100 MB of available hard disk space (200 MB)
- VGA with 128 MB RAM, supporting 32 bit color and minimum resolution of 1024x768 pixels including appropriate monitor (VGA with 512 MB RAM and resolution of 1400x1050 and higher, wide screen is recommended)

### **3 SOFTWARE REQUIREMENTS**

CONSTRUCT is based on Microsoft .NET framework version 2 and it is required to run tool. If your system doesn't have the appropriate version of .NET framework installation procedure will report that. Appropriate version of .NET framework is located on installation CD. You can also download it from Microsoft web site. In addition to .NET framework Windows Installer version 2 is required also and if not installed one can find it on installation CD. Adobe Acrobat Reader version 6 and newer is required for viewing documentation. Appropriate copy is located on installation CD. The present version of ConStruct Software does not support Window 64 bits operating systems.

### **4 INSTALLATION**

Installation of CONSTRUCT Tool is delivered on CD containing all necessary information to run it. Put CD into your optical drive and run setup.exe and follow the on screen instructions. Although one can change installation path it is recommended to keep the default one.

## 5 GRAPHICAL USER INTERFACE

### 5.1 General

CONSTRUCT Graphical user interface or simple GUI is divided into 5 areas. They are Menu bar and Toolbar, Project Explorer, Data Table area and Visualization Window, see Figure 5.

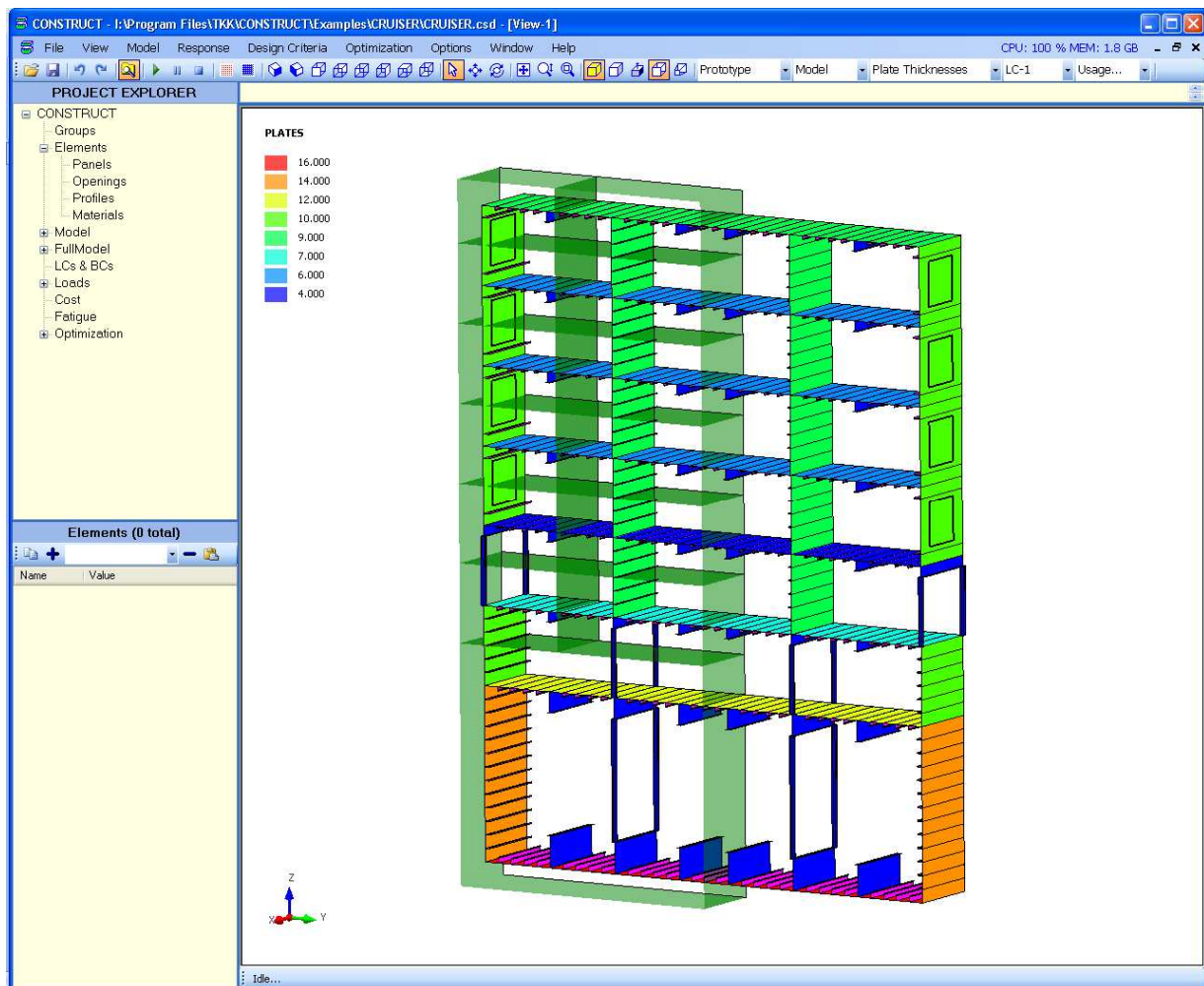


Figure 5: Graphical User Interface (model section with plate thicknesses)

### 5.2 Menu bar

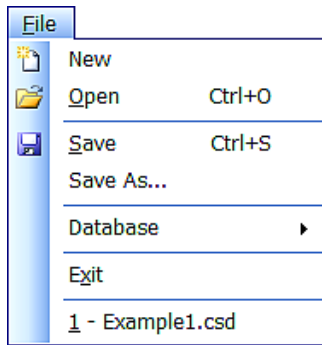
Menu bar, Located on the top of the GUI, holds commands used for file, data, view and geometry manipulation.

#### 5.2.1 File Menu

File menu is the file and data management centre of this GUI. Here's the list and a short description of the commands found in this menu:

- New: Starts a new CONSTRUCT project.



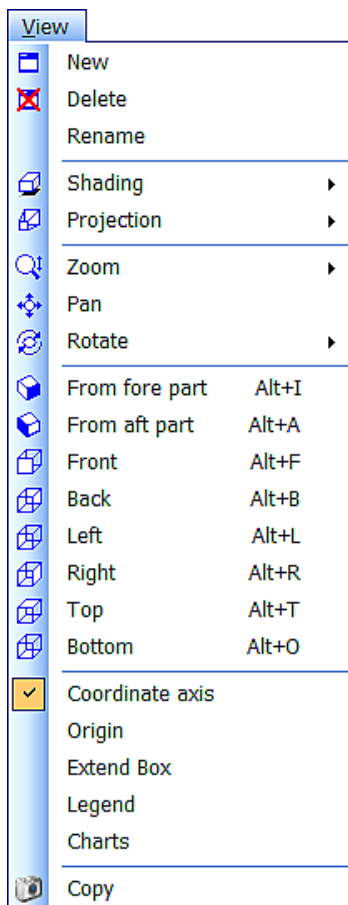


- Open: Opens an existing CONSTRUCT project file (\*.csd file extension).
- Save: Saves the working project under a current name.
- Save as: Saves the working project under a new name and location.
- Database: Database holds a sub menu used to import material and profile data sheets into the current project and model respectively. Sub menu commands will be explained in detail later in Working with CONSTRUCT chapter.
- Exit: Exits CONSTRUCT

At the end of this Menu there is a list of previously opened files (if there are any). Clicking on them will load them into CONSTRUCT.

## 5.2.2 View menu

This menu is used for view manipulation. What follows is the list of commands that can be found here.



- New: Opens a new view window
- Delete: Deletes active view window
- Rename: Renames the active view window. Selecting it, Rename View input box will appear allowing the user to specify a new window name.
- Shading: Contains a sub menu allowing the user to select from a range of model display options. Wireframe, Filled and Shaded options are included. Normals option is used to display the panel's normal orientation (red colour shows normal direction).
- Projection: Contains a submenu with 2 View projection options. Perspective and Orthogonal options are available.
- Zoom: Fit and Window options are available. Fit option will fit the whole model into the view window, and Window option allows zooming of the area of interest.
- Pan: Allows the user to pan the view up, down, left and right.
- Rotate: Allows the user to rotate the model.
- From fore part: This is an isometric view option. Displays an isometric view of the model, from the fore.
- From aft part: Displays an isometric view of the model, from the aft.

After that lists of 6 available view projections is given. Front and Back, Left and Right, Top and Bottom respectively.

- Coordinate axes: Toggles the display of View window coordinate axes.
- Origin: When checked, displays 3 orthogonal planes (XY, XZ and YZ) shaded grey. Origin is at the cross section of these 3 planes.
- Extend box: Graphically shows the total dimensions of the model. Black dashed lines show total length, height and width of the object.
- Legend: Displays the legend of the selected property in the upper left corner of the view window.
- Charts: Displays optimization results via diagrams. It shows the curves of 3 object functions ( weight, cost, C.G) or the moment-deflection curve for ultimate strength analysis
- Copy: Copies the View window area into Clipboard so it can be pasted as a picture in any other software one is using, for example Microsoft PowerPoint or Word etc.

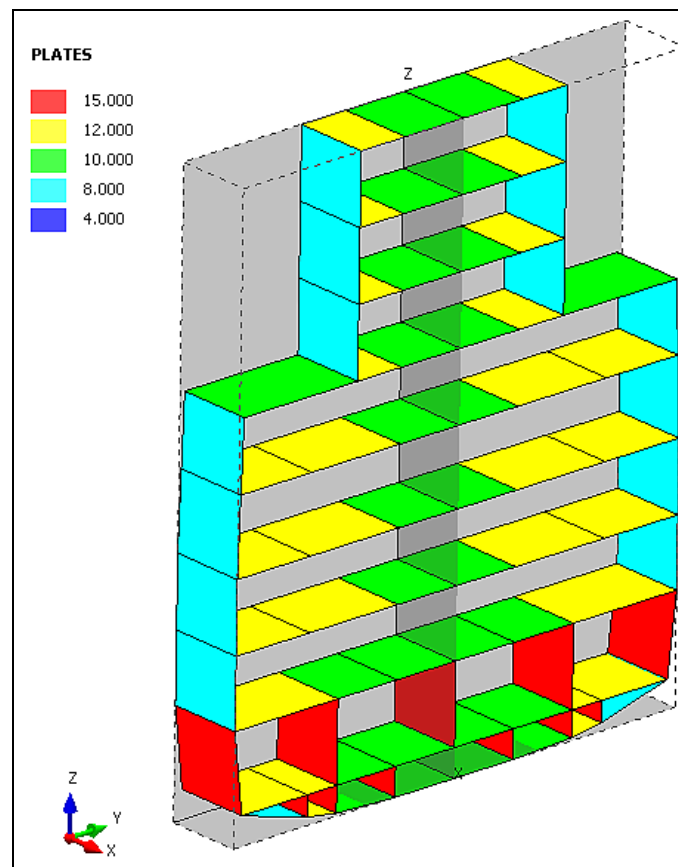


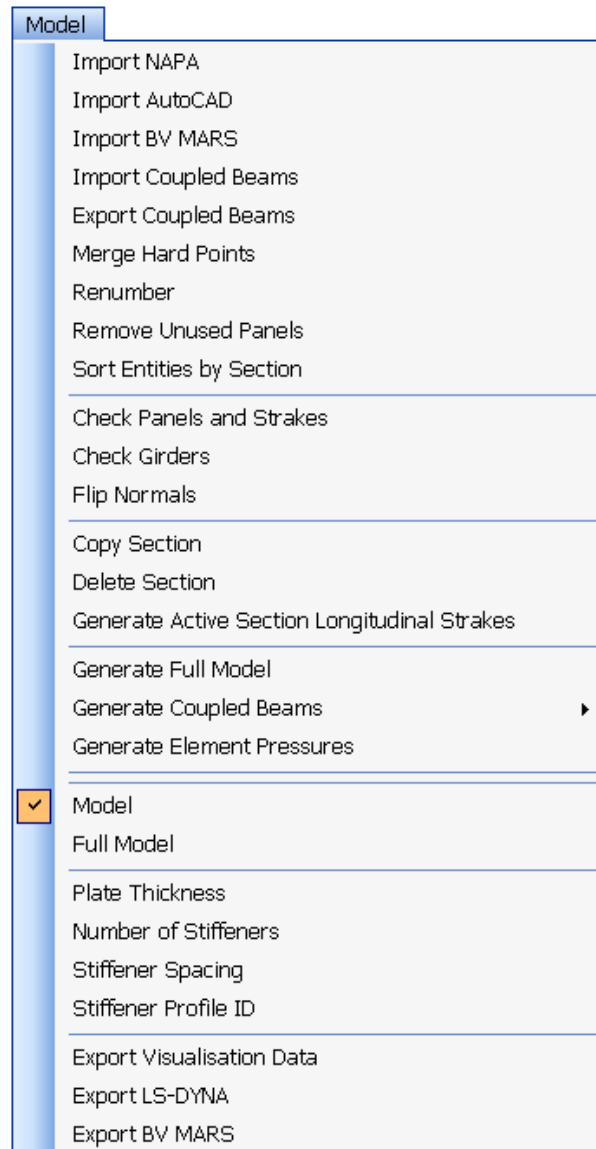
Figure 6: Model's Origin, Extend Box and plate thickness distribution

### 5.2.3 Model menu

Commands used for model creation and manipulation can be found below.

- Import NAPA: Loads NAPA standard based section model. Import NAPA data window will appear. User must specify path to the appropriate file containing NAPA data.

- Import Auto CAD: Loads points and lines from AutoCAD \*.dxf file.
- Import Coupled Beams: Loads coordinates and connection point coordinates of the coupled beam model. It is necessary in order to run a stress analysis of the model.
- Export Coupled Beams: Exports the coordinates and connection points of the coupled beams.
- Merge hard points: Merges coincident hard points.
- Renumber: Renumbers indexing of data entities. It can be utilized after entity removing.
- Remove Unused Panels: Delete panels, which are not currently applied in the ship model.
- Sort Entities by Section: Sorts data entities of the model with several sections. It is necessary after modification of the model composed of several sections.
- Check Panels and Strakes: For future use to check the validity of the input data of the panels and strakes.
- Check Girders: Check panels for difference in stiffener height which can cause particular stiffener to behaviour like girder (if stiffener height is 20% larger than other stiffeners than it becomes girder).



- Flip normals: Changes the direction of panel normals. This is useful to change the orientation of the panel's stiffeners for example or to invert the curvature of a panel.
- Copy section: Copies the active (selected) section to a new coordinate (longitudinal). User can specify the longitudinal coordinate through an input box that will appear.
- Delete Section: Deletes all data entities of the selected section.
- Generate Active Section Longitudinal Strakes: Generates longitudinal strakes from the lines drawn in the active section. Longitudinal strakes are the building blocks of the model.
- Generate Full Model: Based on the section list provided, a full model of the ship will be generated.
- Generate Coupled Beam Model: The coupled beam model, on which stress analysis is based on, can be generated in 2 ways: manually or automatically.

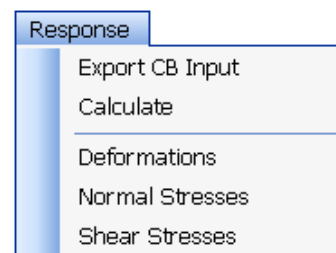
Manual generation needs Column beam connection points input. A column beam will be created after user defines 4 connection points.

- Generate Element Pressures: Distributes pressure load depending on the specified surface load and group definition.
- Calculate Cost, Weight and CG: Calculates cost, weight and the position of the CG for the model.
- Model: Activates the display of the section model (simply called model).
- Full model: Activates the display of the full model, Full model has to be generated before.
- Plate Thickness: Displays the model's plate thickness distribution. Coloring is explained by the legend in the upper left corner of the view window.
- Number of Stiffeners: Similar to the previous command, this command will activate graphical display of the number of stiffeners per panel. Also, a legend which explains the colors will appear in the upper left corner of the view window.
- Stiffener Spacing: Graphically shows the stiffener spacing distribution per panel. If the panel has no stiffeners, it's spacing value will be -1.00
- Stiffener Profile ID: Graphically shows the ID of the panel's stiffeners. That is, it shows which panel has which stiffener profile. If the panel has no stiffeners its Stiffener Profile ID value will be set to -1.00.
- Export Visualization Data: Creates a \*.txt file containing data necessary for visualization using VISUME software.
- Export LS-DYNA: Creates a LS-DYNA input file.
- Delete BV MARS: Creates a \*.txt file containing data necessary for BV MARS software.

#### 5.2.4 Response menu

Used to calculate response and load basic output data.

- Export Coupled Beam Input: Creates a file containing input data for the coupled beam solver. Before the user can do that, a set of ship's sections defining the full model must be specified.
- Calculate: Launches the Coupled Beam Solver which will calculate the models response to the applied loads.
- Deformations: Loads the deformation part of response data and graphically shows the deformation distribution along the full model of the ship.
- Normal Stresses: Loads and graphically displays calculated normal stresses.
- Shear Stresses: Loads and graphically shows the distribution of calculated shear stresses of the full model.

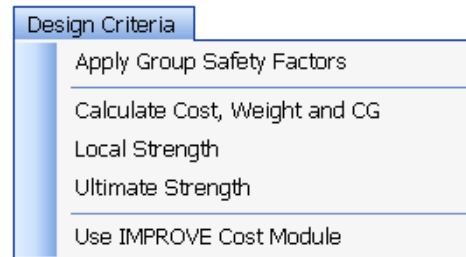


#### 5.2.5 Design Criteria menu

Commands used for calculation of design criteria and ultimate strength.

- Apply Group Safety Factors: Applies user specified safety factor for strakes. Safety factor are given in group defination.
- Calculate Cost Weight and CG: Calculates cost, weight and centre of gravity.

- Local strength: Calculates the satisfaction of particular design criteria of the ship model taking into account appropriate response.
- Ultimate strength: Calculates the ultimate strength of the ship model. As results moment deflection curves for hogging and sagging loading are printed to text files: moments\_hog.txt and moments\_sag.txt.
- Use IMPROVE Cost Module: Activates the application IMPROVE Cost Module for the cost calculation.



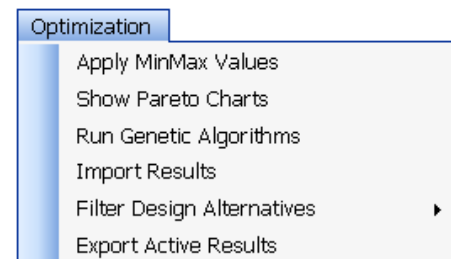
Local strength analysis includes the following design criteria:

- CROSSOVER : Overall buckling of stiffened panel, O. Hughes
- ULSDNV: Overall buckling of stiffened panel, DNV
- PBDNV: Plate bending, DNV
- STIFFBEND: Stiffener bending, beam theory
- PBUCKDNV: Plate buckling, DNV
- SBUCKFLANGE: Stiffener flange buckling, free sides simply supported, one free
- SBUCKLAT: Stiffener lateral buckling
- SBUCKLOCAL: Stiffener web local buckling, DNV
- TRIPPING: Torsion buckling of the stiffener, DNV
- FATIGUE: Fatigue criterion, DNV

### 5.2.6 Optimization menu

Used for optimization procedure manipulation.

- Apply MinMax values: Applies the specified minimum and maximum values of the panel parameters, such as: plate thickness, number of stiffeners etc.
- Show Pareto Charts: Switch the optimisation charts to objective space.
- Run: Starts the optimization procedure. Make sure that all optimization parameters have been specified.
- Import results: Loads the optimization results. An Import Optimization Data Window will appear asking the user to specify the path to the appropriate file (design alternatives).
- Filter Design Alternatives: Filters the optimisation results according to the user specification. Alternatives for filtering are All, Feasible, Infeasible, Pareto.
- Export Active Results: Exports the selected optimisation to a OptExport.txt file.



### 5.2.7 Options menu

Has only one command, Colors and Lines. It is used for altering the appearance of GUI. User can alter the background appearance, the weight and color of lines, points and other entities.

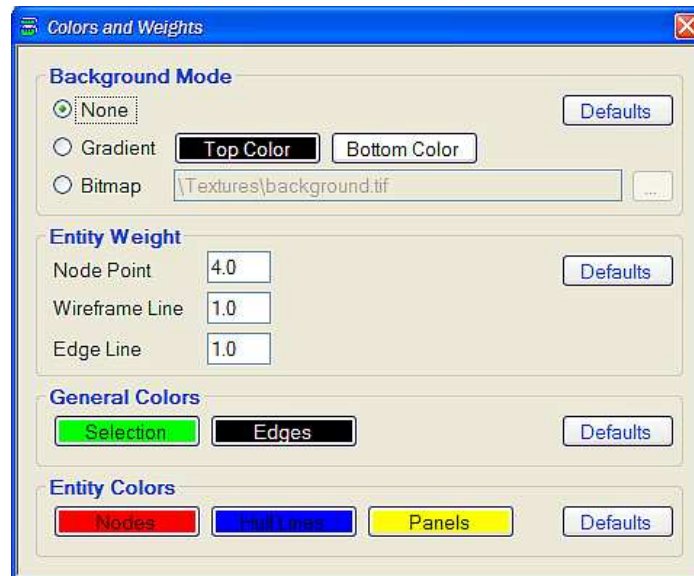
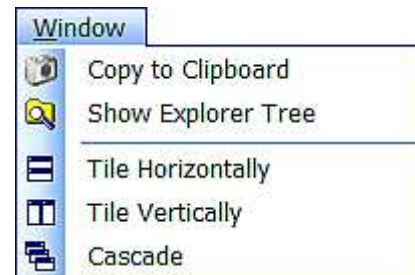


Figure 7: Colors and Line Weights Options

### 5.2.8 Window menu

Windows menu holds commands used for managing view windows.

- Copy to Clipboard: Copies the contents of the view window to Clipboard as an image. Then it can be pasted into image editing software and saved as an image file.
- Show Explorer Tree: Toggles the Project Explorer display on and off.
- Tile Horizontally/Vertically: Arranges view windows one above the other or side by side. Active view window will be placed on top or to the left.
- Cascade: Arranges view windows one in front of the other. Active view window will have the priority and be placed on top.



### 5.2.9 Help menu

- User's Guide: Launches User's Guide document in PDF format.
- About: Displays information about the software version, authors etc.

## 5.3 Toolbar area

Toolbar commands are in most cases shortcuts for Menu bar commands. This paragraph will explain only those commands and menus that can not be found among the Menu bar commands.

### 5.3.1 Grid point and Section line tools



These two buttons are used during the first stage of the modelling process. They are from left to right:

Show/Hide section grid points (the red one): Toggles the display of section grid points

Show/Hide section lines (the blue button): Toggles the display of section lines.

### 5.3.2 Selection tool

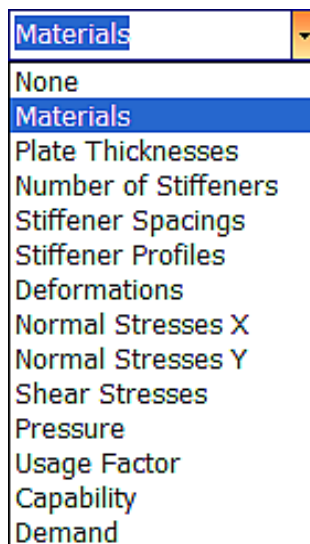
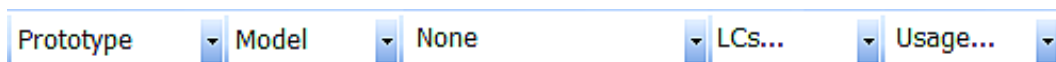


This is Selection tool button. It is used to select entities directly from the view window. Multiple selections are performed by holding the left mouse button and circling the elements of interest. To select elements that are hidden behind other elements, repeat the previous procedure while holding the Shift

key.

### 5.3.3 Drop Down menus

Drop down menus are used for manipulating response data display, they are located in the upper right corner of the GUI and they are from left to right:



- Design alternatives drop down menu: It has 2 display options, Prototype and Design Alternatives. Design Alternatives option is used for optimization results display.
- Model drop down menu: It has 3 display options, Model, FullModel and Groups. Model option will display only sections, FullModel displays the full model of a ship and Groups option displays elements which are members of a particular group.
- Properties and Response drop down menu: this menu allows user to choose the graphical display of model's properties and response (stress distribution).
- Figure on the right shows the full list of display options. The distribution of particular property (i.e. Plate thickness) is presented via colored panels. A legend assigning values to colors appears in the upper left corner of the View window.
- When displaying analysis results, user can choose the results of a particular load case through the load case drop down menu.
- Load case drop down menu: It is a list of all defined load cases. Choosing the load case from this menu will load its response, and that response data will be available for viewing.

- Usage drop down menu: It allows the user to select the particular design criterion usage factor. In order to display the usage factor, that option has to be selected from the Properties and Response drop down menu first, and of course, design criteria has to be calculated (Design Criteria menu => Calculate).

## 5.4 Project Explorer and Data Table areas

### 5.4.1 General

Project Explorer basically shows the structure of the model. User can find information about the Elements and Entities that the model is made of. Also, Project Explorer tree in conjunction with Data Table is used to specify all parameters about various entities, load cases etc. A figure on the right shows Project Explorer and Data Table area with profiles data loaded into the data table. Project Explorer allows easier navigation through the model entities, while Data Table provides information and enables creation of entities and later modification of their properties.

The screenshot shows the PROJECT EXPLORER window. The tree view on the left includes: CONSTRUCT, Groups, Elements, Panels, Openings, Profiles, Materials, Model, FullModel, LCs & BCs, Loads, Cost, and Optimization. The Data Table below is titled 'Profiles (100 total)' and shows a table with columns Name, Value, and Description. The first row is highlighted in blue and green.

Name	Value	Description
ID	3	Unique ID
Description	B*100*7	Description
Type	B	Profile type (T,L,B,FB)
Material	AH36	Material
HPW	100	Height of profile web, m
TPW	7	Thickness of profile w...
BPF	15.5	Breadth of profile flan...
TPF	14.8	Thickness of profile fl...
A	874	Cross section area, m...
ns_x	58.7	Neutral axis position, ...
ns_y	5.5	Neutral axis position, ...
Is_x	850000	Moment of inertia, x ...
Is_y	19900	Moment of inertia, y ...

Project Explorer branches are (displayed on the left):

### 5.4.2 Groups

The first Sub branch of PE (Project Explorer) is named Groups. Clicking on it will load information about the model's groups into Data Table.

- *ID*: Unique group number.
- *Description*: User can provide a description of the group, but it's not required.
- *Group Type*: There are two types available, Entity and Description type.
- *Entity Type*: Specifies what entities are in the group.
- *Color*: Specifies the Groups color.
- *Number of entities*: gives information about the number of entities in that group.
- *Entity Description*: If the group is Description based, this field specifies the description variable that all elements of that group have in common.

What follows is a set of optimization parameters. MinMax values give the range of parameters to use during optimization procedure.

- Parameters *MatIDMinMax* and *ProIDMinMax* specify the range of database ID for material and profiles respectively. During optimization, the program will use profiles and materials from the appropriate database but only those materials and profiles which ID is within specified range.
- *PlaThiMinMax* specifies the range of plate thickness (in mm) variable and *StiSpacings* specify the stiffener spacing range. *NoOfStiMinMax* specifies



the minimum and maximum number of stiffeners to be used during optimization.

### 5.4.3 Elements

Elements are the building blocks of the model. There are several types of elements: Panels, Openings, Materials and Profiles which then become different entities of the model. For instance, entities like Frames and Girders can have same Profile but different purpose in the model.

### 5.4.4 Panels

Panels are defined by their size, thickness and stiffener distribution. Panels are automatically created when importing model from NAPA for instance. Here is the list of Panel parameters and a short explanation of them:

- *ID*: Panel's ID value is a number and it is unique to every panel. It is created automatically after panel generation.
- *Description*: It is a description variable of the Panel. It is automatically generated based on the panel's length, width, plate thickness and material. User can enter its own description or erase this one and leave this field blank.
- *Length*: Panel' length, total dimension in X direction (always stiffener direction).
- *Width*: Panel's width, total dimension in Y direction (always perpendicular to stiffener direction).
- *Radius*: variable that defines curvature of a panel (only cylindrical panels can be defined, e.g. stiffener direction remains straight).
- *Thickness*: Plate thickness of a panel, in millimeters.
- *Material*: Specifies panel's material ID.
- *Stiffener ID*: Specifies the ID of profile to be used as panel's stiffener. If the panel is without stiffeners leave the box blank.
- *Stiffener Layout*: Specifies the distribution of stiffeners along its width relatively. For symmetric layout of for example 2 stiffeners, layout value would look like this: 0.33333\*0.66667. This means that the first stiffener is located at 1/3 of the panel's width and second at 2/3 of the panel's width.
- *NoOfStiffeners*: Specifies the number of stiffeners on the panel. If the user provides this value and Stiffener ID, Stiffener layout and Stiffener spacing will automatically be calculated.
- *StiSpacing*: Based on number of stiffeners and the width of the panel, this value is automatically calculated.
- *StiAngle*: This variable specifies the angle of the stiffeners. Angle 0 means that the stiffener's web is perpendicular to the panel. See figure.
- *StiOrientation*: For future use (orients stiffener flange to left or right)
- *Area*: Shows the calculated area of the panel. It is calculated using the command Calculate Cost, Weight and CG in the Model Menu.
- *Weight*: Panel's weight is calculated using the command Calculate Cost, Weight and CG from the Model menu. First number shows the weight of a single panel. Second number shows the Unit weight which is a function of

material and panel's plate thickness. Basically it shows the weight per panel's area information.

- *Cost*: The cost of the Panel.
- *C.G*: This is also a calculated data using the fore mentioned command. Shows the location of the panel's center of gravity in panel's local coordinate system.

What follows is a set of optimization parameters. MinMax values give the range of parameters to use during optimization procedure.

Parameters *MatIDMinMax* and *ProIDMinMax* specify the range of database ID for material and profiles respectively. During optimization, the program will use profiles and materials from the appropriate database but only those materials and profiles which ID is within specified range.

- *PlaThiMinMax* specifies the range of plate thickness (in mm) variable. *NoOfStiMinMax* specifies the minimum and maximum number of stiffeners to be used during optimization.
- Optimize parameter determines whether or not the optimization procedure should take this panel into account. If the panel is not to be optimized Null data should be entered.

#### 5.4.5 Openings

Openings are used to model all kinds of windows, holes, maintenance approaches, etc. They are defined by the following properties:

- *ID*: Opening's unique ID.
- *Description*: User can specify a description of every opening type.
- *TYPE*: It defines the shape of the opening. There are 2 opening types available: RR – rectangular and E- manhole (elliptical).
- *A* and *B*: Longitudinal and Transverse dimensions of the opening in mm.
- *Area*: Area of the opening. It will be calculated using the Calculate Cost, Weight and CG command in the Model menu. The area of the opening will later be subtracted from the appropriate LStrake, that is the LStrake on which the opening has been defined. This will result in the lower mass of that LStrake.

#### 5.4.6 Profiles

Profiles are elements that are used to define stiffness characteristics of all Stiffeners, Girders Frames and Pillars of the model. Profile parameters are loaded from the appropriate datasheet if available. If that is not the case, user is free to define a profile on it's own. Here is the list of the Profile parameters:

- *ID*: Profile's ID value.
- *Description*: User is free to give a description of each Profile.
- *Type*: Defines the shape of the Profile. There are 4 profile types available Depending on the data sheet : I, T, B, CH. The profile parameters are defined by it's shape, i.e. I profile has no flange like the T and B profile, so no data will be specified under the flange breadth parameter.
- *Material*: Material ID of the profile.
- List of the profile's geometrical and stiffness parameters. They are loaded from the profile datasheet.

- *HPW*: Height of the profile's web in mm. In case of a pillar type profile (CH) HPW is the diameter of the pillar.
- *TPW*: Thickness of the profile's web, mm. In case of a pillar type profile (CH) TPW is the thickness of the pillar's shell.
- *BPF*: Breadth of the profile's flange, mm.
- *TPF*: Thickness of the profile's flange, mm.
- *A*: Area of the profile, mm<sup>2</sup>.
- *Ns\_x and NS\_y* : location of the profile's neutral axis, in the profile's coordinate system.
- *Is\_x and Is\_y*: Moments of inertia of the profile about the neutral axes.

#### 5.4.7 Materials

Material properties are loaded from the proper datasheet, or can be specified by the user. CONSTRUCT supports isotropic materials only. Here's the list of them:

- *ID*: Numerical data specifying unique material ID
- *Description*: User can enter a description of the material or leave it blank
- *YM*: Young's module of elasticity, in MPa.
- *PR*: Poisson's ratio
- *TR*: Tensile strength, MPa
- *YR*: Yield strength, MPa
- *eu*: Tensile strain
- *ro*: Material's density, kg/m<sup>3</sup>.

Same data can be loaded from Material datasheets.

#### 5.4.8 Model

Model is a branch in the PE that allows modeling of sections. It shows the structure of the model's geometry. It's composed of geometrical and structural entities which are explained in the following text.

Model properties:

- *Description*: User can enter description of the model or leave it blank.
- *Length, Breadth, Depth and Draught*: These are some design parameters of the model.
- *SectionList*: Specifies the number and order of sections that make up full model of a ship. For example, user can enter 10\*1\*50\*2\*10\*1 which means that the full model will be composed of 10 sections number 1 then 50 sections number 2 and then again 10 sections number 1.
- *Weight*: Calculated weight of the model.
- *Cost*: ...
- *C.G.*: Position of the model's center of gravity (X\*Y\*Z).

#### Points

Points are connection locations between lines, girders, frames, pillars and so on. Many line entities (like the ones we've just named) depend on them. Surface entities like LStrakes are also defined by these Points. Deleting any of the points will crash the model.

Point is defined by:

- *ID*: Numerical data unique to every point
- *X, Y and Z*: These are point coordinates, in meters.

### Coupled Beam Points

Like LStrakes, Coupled Beams rely on their own set of points, the Coupled Beam Points (or CB Points). They are automatically created during automatic CB generation, or can be entered manually.

CB Point properties:

- *ID*: Numerical data unique to every CB point.
- *X, Y and Z*: CB Point coordinates, in meters.

### Stiffener Points

Stiffener Points are connection entities for panel's stiffeners. They are created automatically after stiffener generation. Their coordinates rely on the stiffener spacing property defined under the panel properties.

Stiffener Points properties are:

- *ID*: Numerical data defining the point's unique ID.
- *X, Y and Z*: Stiffener point coordinates, in meters.

Note: Changing the coordinates of a Stiffener Point will not result in stiffeners relocation. CONSTRUCT will create a new point with coordinates that are same as those of the point that has been altered. This new point will be assigned to the stiffener connected to the altered point.

### Opening Points

Opening Points define the location of the opening's center which is actually a "connection point" for all openings.

Opening Point properties are:

- *ID*: Defines the unique ID of an opening point.
- *X, Y and Z*: Coordinates of the opening point.

### Sections

Sections are something of a blueprint of the model. This is the starting point when creating a new model from scratch. It is here that the user defines points that are characteristic to the ship's section. Section properties are:

- *ID*: Numerical data that is unique to every section.
- *Description*: Section's description, it can be left blank.
- *XSection*: Longitudinal position of the particular section, in meters.
- *Length*: Specifies the length of the section, in meters.
- *Weight*: Calculated weight of the section, in metric tons.
- *Cost*: Calculated cost of a section.
- *CG*: Calculated position of the section's center of gravity, in global coordinate system. Coordinates are given in X\*Y\*Z format.
- *YGrid*: Specifies horizontal coordinates of section's characteristic points, i.e. panel intersections, girder points and so on.
- *ZGrid*: Specifies vertical coordinates of section's characteristic points,
- *NumOfDiv*: ...

### Stiffeners

Stiffeners are structural entities of the model. Almost all of their properties are defined by the panel that they are a part of. This means that in order to alter stiffener properties user has to do it through the Panel branch and Panel properties.

Stiffener properties:

- *ID*: Stiffeners all have same ID which is null because stiffeners are defined by their panel.
- *Description*: Stiffeners can have a description but it is inherited from the LStrake on which they are placed. Changing the description here will have no effect.
- *Connections*: Gives the list of stiffener's connection points.
- *Angle*: Gives the information about stiffener's angle. It is inherited from the panel.
- *Orient*: Gives the information about the stiffener's orientation which is also inherited from the panel.
- *Profile ID*: Gives information about the stiffener's shape, which is inherited from the panel.

### Coupled Beams

Response of the ship's hull to the loads is calculated through the coupled beams theory. So, in order to calculate the response, user has to define coupled beams. In the Coupled Beams branch of the PE tree, user can manually define every coupled beam, or view its properties.

Coupled Beams properties:

- *ID*: Beam's unique ID.
- *Description*: User can specify the beam's description, or leave it blank.
- *Connections*: List of Coupled Beams Points that define the Coupled beam.
- *Reference Point*: reference point is also one of the Coupled Beam Points that serves as a reference for that Coupled Beam.
- *CBCConnections*: List of connection points to other Coupled Beams.
- *CBIntStrakes*: List of LStrakes that the Coupled Beam extends over.
- *CBIntPillars*: List of Pillars that are connected with the coupled beam, if any.

### LStrakes

LStrakes are longitudinal strakes. They are created directly from section lines.

Here's the list of LStrake's properties:

- *ID*: Specifies unique LStrake ID number.
- *Description*: It can be entered by the user, or left blank.
- *Length*: Longitudinal dimension of the LStrake, inherited from the section length property.
- *Width*: Transversal dimension of the LStrake, defined by the section lines and section grid points.
- *Panel IDs*: Every LStrake has one panel ID assigned to it. That way user can quickly change the number of stiffeners or plate thickness of any LStrake by changing the Panel ID. Or changing the panel's parameters, user can change for example, the stiffener type on a number of LStrakes, that is, those LStrakes that have the panel ID of the panel that user has altered.
- *Connections*: List of LStrakes point connections. During LStrake creation, points have been created from the section line's intersections. Another set of points are created, at a section length distance from the original section lines,

defining section length (LStrake length also). LStrakes are defined by the location of those points. Note: LStrake's shape and orientation can be modified by changing the coordinate of appropriate connection points.

- *Rule Desc*: Rule description of the LStrake. For instance, Deck or Side.
- *Group IDs*: If an LStrake is a member of any groups, IDs of those groups will be displayed.
- *Openings*: Specifies openings ID, then ID of the opening point and the orientation of the opening. Opening ID defines the type and size of the opening, opening point ID determines the location of the opening and angle opening's orientation (0 to 180 deg).
- *GirderIDs*: Gives the information about the girders connected to that LStrake. Since girder's location and orientation is defined by their connection points, altering this field will have no effect.
- *Weight*: Calculated weight of the LStrake. If an LStrake has an opening it will be taken into account during weight calculation.

### TStrakes

TStrakes are transversal strakes. They are similar to LStrakes but TStrakes don't have stiffeners or girders. TStrake can have a Panel with stiffeners assigned to it, but stiffeners will not show up on a TStrake. Only plate thickness and material property of the panel will be associated with that TStrake. TStrakes are defined by their connection points. User has to enter a list of connection points and a TStrake will be created. Like LStrakes, TStrake can have an opening, and it's specified in the same way like the LStrake opening.

### Girders

Girders are massive longitudinal structural entities. Their position and orientation is defined by their connection points. Unlike stiffeners, girders allow placing of openings on them.

Girder properties:

- *ID*: Unique ID number of girder.
- *Description*: User can specify description of every girder.
- *Connections*: Like mentioned before, girder's orientation and location are defined by these points.
- *Angle*: Specifies girder's angle, that is it's web angle. If set to 0 deg, girder's web will be parallel to Z axis.
- *Orient*: Specifies girders orientation vector. Usually used to flip the girder about it's longitudinal axis. For example input 0\*0\*1 will orient the girder's web in +Z direction.
- *Profile ID*: Determines the shape of the girder. Appropriate profile ID will be loaded from the profile database.
- *Openings*: In order to add an opening on a girder, user has to specify the opening type ( appropriate ID from Openings branch), center point location (from the opening points) and orientation angle, for example 1\*7\*90.

### Frames

Frames are similar to girders, but in this case, they are transversal structural entities. The properties are completely same as those of girders.

## Pillars

Pillars are structural entities that connect decks. They rely on connection points and profile ID.

Pillar properties:

- *ID*: Pillar's unique ID number.
- *Description*: user can specify the description or leave it blank.
- *Profile ID*: Specifies the shape and dimensions of the pillar's cross section. Appropriate profile will be loaded from the profile database.
- *Connections*: Points that define the pillars position and orientation.
- *Orientation*:...

### 5.4.9 Full Model

Full model branch allows an overview of the whole ship model. User can view the ship's response and design criteria fulfillment element by element.

Full Model Properties:

- *Description*: User can enter description of the model or leave it blank.
- *Length, Breadth, Depth and Draught*: These are some design parameters of the full model. Length calculation is based on section length and number.
- *Weight*: Calculated weight of the model.

As mentioned before view of the element by element response is available. User can view the response of every L and T strake, Girder, Frame and Pillar. CONSTRUCT calculates normal (X, Y) and shear stress for every element, plus some design criteria fulfillment that are characteristic to particular element. For example buckling and yield criteria for pillars or stiffener flange buckling for LStrakes and so on.

### 5.4.10 LCs&BCs

Load cases and Boundary Conditions is the full name of this sub branch.

This is where user specifies which Loads (Load IDs) should be taken into account.

Properties:

- *ID*: Load's unique ID number. Must be specified before Solver starts.
- *Description*: A short description of the load case, optional.
- *LoadIDs*: List of loads specified under Loads branch. It is useful for combining different load types, i.e. moments plus pressure etc.
- *GroupIDs*: List of groups to which the load case applies. Useful for applying load to just a part of the ship model.

### 5.4.11 Loads

Loads branch is used to define specific loads, like moment load or wave load. Here's the list of sub branches and parameters:

#### Global

Used to define global bending moment

- *ID*: User must specify unique ID number
- *Description*: Short description of the load, optional.
- *Moment*: Specifies the bending moment which will be applied, in kNm.

#### Point

User can define load (force and moment) to a specific point of the model.

#### Point Load parameters

- *ID*: User must enter unique ID number
- *Description*: Short Description is optional.
- *X,Y and Z Force*: Force magnitude applied to the point in X,Y and direction respectively.
- *X,Y and Z Moment* : Magnitude of the moment about X,Y and Z axis respectively applied to the point.

#### Line

Line load allows applying line loads like force and moment per length.

- *ID*: Load's unique ID.
- *Description*: Short description is optional.
- *Force and Moment*: magnitude of loads to be applied, in N/m and Nm/m respectively.

#### Surface

This is where user can specify pressure type load.

- *ID*: Load's unique ID number.
- *Description*: Short, optional description of the load.
- *Pressure*: Magnitude of pressure to be applied, in Pa.

#### Displacement

Displacement can also be applied to the ship model, in X direction.

- *ID*: Load's unique ID number.
- *Description*: Short, optional description of the load
- *XDisplacement*: Enter desired displacement in X direction, in m.

#### Hydrostatic

Hydrostatic load is due to the ship's buoyancy.

- *ID*: Load's unique ID number.
- *Description*: Short, optional description of the load.
- *Density*: Specifies density of the fluid, kg/m<sup>3</sup>
- *Height...*
- *Height...*
- *Percent*: Specifies the tank fullness percentage.

#### Acceleration

Acceleration loads can be applied in different directions. Roll acceleration is also available, about 3 main axes.

- *ID*: Load's unique ID number.
- *Description*: Short, optional description of the load
- *X,Y and Z Acceleration*: Acceleration magnitude in 3 main directions, m/s<sup>2</sup>
- *X,Y and ZRoll*: Roll acceleration about 3 main axes, in 1/rad<sup>2</sup>.

#### Wave Load

Wave Load is also available, in form of a sine function.

- *ID*: Load's unique ID number.
- *Description*: Short, optional description of the load



- *Density*: Density of the fluid, kg/m<sup>3</sup>.
- *Amplitude*: Wave amplitude, in meters.
- *Phase*: Specifies phase angle in degrees.
- *Function*...

#### 5.4.12 Cost

It is used to define costs of the ship's production. Material costs, part production, assembly and welding costs are included. User can get information about the general cost and some average values of profile length and plate width and thickness.

#### 5.4.13 Optimization

CONSTRUCT has built-in optimization capabilities. Optimization branch of the Project Explorer allows the user to set optimization parameters.

Optimization parameters

- *Description*: Short description can be entered here, not necessary.
- *NV*: Number of variables taken into account.
- *NC*: Number of objective functions.
- *NCON*: Number of constraints.
- *POP*: Population size.
- *NOGEN*: Generation size.
- *PC*: Crossover probability.
- *PM*: Mutation probability.
- *W0*: Objective function ponders.
- *OPTMODE*: Toggles maximization or minimization of object function (1 or 0)
- *SMODE*: Seed mode on/off (0/1)
- *GroOptIDs*: List of group IDs on which the optimization will be applied.
- *StrGroOptIDs*: List of group IDs on which the strake optimization will be performed.

After optimization has been completed a set of design alternatives is created they can be viewed by clicking on the DesAlternatives sub branch in the Project Explorer.

## 6 WORKING WITH CONSTRUCT

### 6.1.1 General

This chapter will show how to create a ship section, model, turn it into a full ship model, define loads, analyze it and run an optimization procedure. First, a modeling procedure will be shown plus some basic operations and rules of CONSTRUCT's GUI. Prior to detailed explanation on how to create a model general idea of modeler as well as list of used terms is given.

Basic idea of mathematical modeler implemented in CONSTRUCT tool is a building block. Following blocks used in modeler: **materials, profiles, openings and panels.**

- **Material** is fundamental building block containing physical properties of materials used to build other blocks.
- **Profile** is used to define geometric characteristics of different profile sections. It also contains information about material of which particular profile can be made or delivered.
- **Openings** are used to define different kind of cutouts which are later used to define ship structure.
- **Panel** is rectangular building block which contains information about plate thickness and material of part of ship structure and its stiffening characteristics like number of stiffeners, distribution of stiffeners etc. It doesn't contain information about openings!

In order to form a ship structure previously defined building blocks have to be placed in space and additionally described if needed (adding of openings etc.). For that purposes following terms are introduced: **points, sections, strakes, girders, frames and pillars.**

- **Points** are defined by 3 coordinates, X, Y and Z respectively. They are used to position building blocks into a 3D space.
- **Section** is mathematical definition for part of the ship structure between  $XSection$  and  $XSection + Section Length$ . It is used to simplify modeling process by forming collections of entities inside of it. First section in model, usually called reference section, has zero Length and can not contain any entity. They are defined between two adjacent transverse frames.
- **Strake** becomes a panel when attached to 4 points in space together with appropriate openings if any. Depending on how they are positioned inside of model we distinguish **LStrakes** what states for Longitudinal Strakes and **TStrakes** what states for Transverse Strakes.
- **Girders** are longitudinal strong beams which divides stiffened panels into strakes. Girder becomes profile when placed between two points.
- **Frames** are transverse strong beams which divides stiffened panels into strakes. Frame becomes profile when placed between two points.
- **Pillars** are profiles connecting two different stiffened panels.

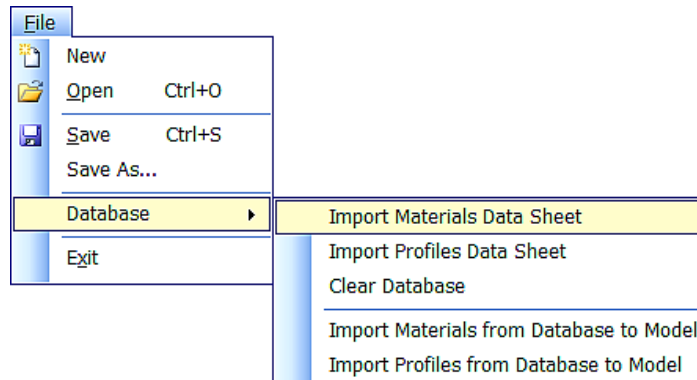
All those entities can have some additional data which are described in details later in this section.

## 6.2 Step by Step Modeling

### 6.2.1 Loading datasheets

After starting up CONSTRUCT, select New from the File menu. A new project will be opened. It is recommended to load some datasheets containing material and profiles.

These are used later on for during the automatic import of NAPA models.



- Select Database => **Import Materials Datasheet command** from the File menu. Select Data sheet window will open. Specify path to the appropriate Excel database that contains

material data. If such datasheet isn't available, materials have to be defined manually. After specifying the correct path and loading the datasheet, a material successful loading message will appear. Click OK to close the Information Window.

- Then, select Database => **Import Profiles Datasheet**. Follow the same procedure as with material data sheet.

This procedure is to be done once after installation of tool and every time when profiles datasheets are changed. Information is stored in **condb.xml** file in working directory. The materials and profiles have been loaded into the program's database, but not into the current model. To do this: Select Database => **Import Materials from Database to Model** command from the file menu. Do the same for profiles as well. Expand the Elements branch and left click on Materials sub branch. There should be several materials loaded into CONSTRUCT. You can list them in the Data Table area and view their properties. Same can be done with profiles. If the procedure was done correctly, clicking on Profiles branch will load profile's properties into Data Table (see figure), and there it will be available for listing and viewing. That's the introduction stage of the modeling process. It's necessary to define grid points and draw the section lines.

Profiles (100 total)		
Name	Value	Description
ID	16	Unique ID
Description	B*220*10	Description
Type	B	Profile type (T,L,B,FB)
Material	AH36	Material
HPW	220	Height of profile web, m
TPW	10	Thickness of profile web, mm
BPF	31	Breadth of profile flange, mm
TPF	32.1	Thickness of profile flange, mm
A	2900	Cross section area, mm <sup>2</sup>
ns_x	134	Neutral axis position, x direction, mm
ns_y	9.3	Neutral axis position, y direction, mm
Is_x	14000000	Moment of inertia, x direction, mm <sup>4</sup>
Is_y	238900	Moment of inertia, y direction, mm <sup>4</sup>

### 6.2.2 Creating a Model

A simple section of a ship which will be modeled is shown on Figure 8. First step is creating the sections. To create a section:

Sections (0 total)		
Name	Value	Description
ID		Section ID
Description		Description
XSection		X Position of section, m
Length		Section length, m
Weight		Section weight, t
Cost		Section cost, £/m
CG		Center of gravity CGX*CGY*C...
YGrid		Positions of vertical grid lines, ...
ZGrid		Positions of horizontal grid line...
NumOfDiv		Number of divisions between ...

- Expand the Model branch of PE tree by left clicking on it.
- Select Sections sub branch from the list. It's necessary to create a new, reference (zero) section. It will be located at the origin and its length is going to be zero. To do this click on the button Add (circled red in the figure). A new section with ID 0 will be created.
- Next, add another section by clicking on Add button again. Newly created section will automatically receive ID 1. Specify the X position of the section and Length, in meters.

It is necessary to define characteristic points, grid points of the section. Those points would be, for example, girder connection, compartment boundaries, pillar connections etc. Figure shows characteristic points of the section marked red. Blue points are the grid points. Grid points are grid lines intersections. There are no grid lines in CONSTRUCT, only grid points. Grid point coordinates are based on YGrid and ZGrid inputs.

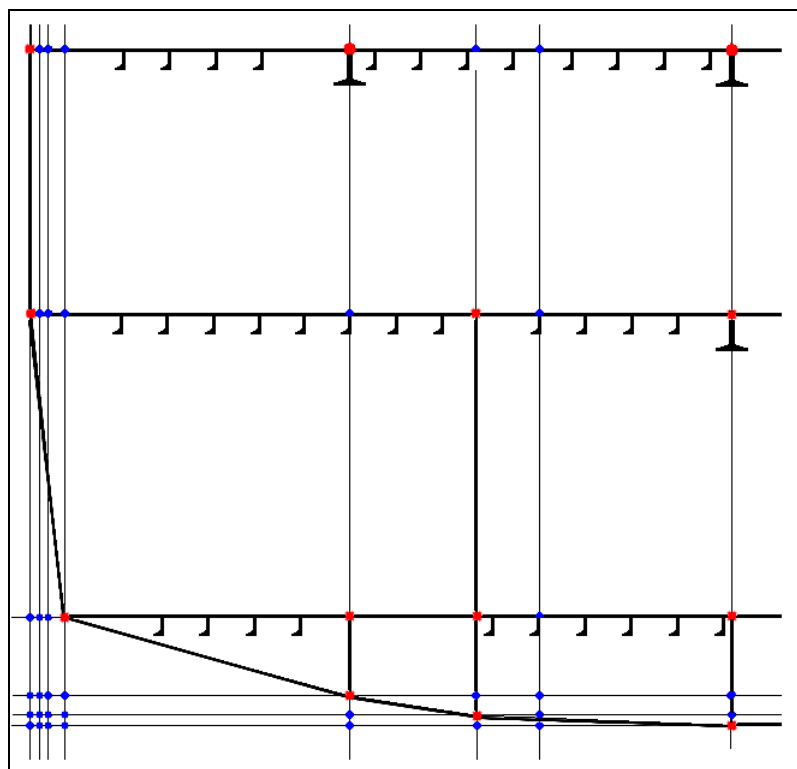


Figure 8: Ship's section, grid and characteristic points

Here's the procedure:

- Select the newly created section.
- Fill the YGrid field to specify horizontal grid point distribution. Coordinates are entered in sequence, from the highest negative coordinate towards the positive.

For example  $-9^*-7^*-4^*-1^*0^*1^*4^*7^*9$  coordinates sequence defines symmetrical section about the ZX plane. Of course, asymmetrical distribution is allowed.

- Fill the ZGrid field specifying vertical grid point distribution.
- In order to see the grid points click the **Show/Hide section grid points** button on the Toolbar. Click on the button next to it to enable the display of section lines as well.
- Now click on the Front view button to view the YZ projection of the grid points.

After the grid points have been generated, section lines can be drawn by connecting the appropriate grid points. Before you proceed make sure that the Selection tool is enabled ( Arrow button on the Toolbar), and that the **Show section lines** option is enabled also.

- Click on the first point (for example 0,0), then holding the CTRL key click on the grid point you wish to connect. A green section line should be drawn. Release the CTRL key.
- Now click on that point and holding the CTRL key connect it to the next point and so on. That is the way section lines are drawn.

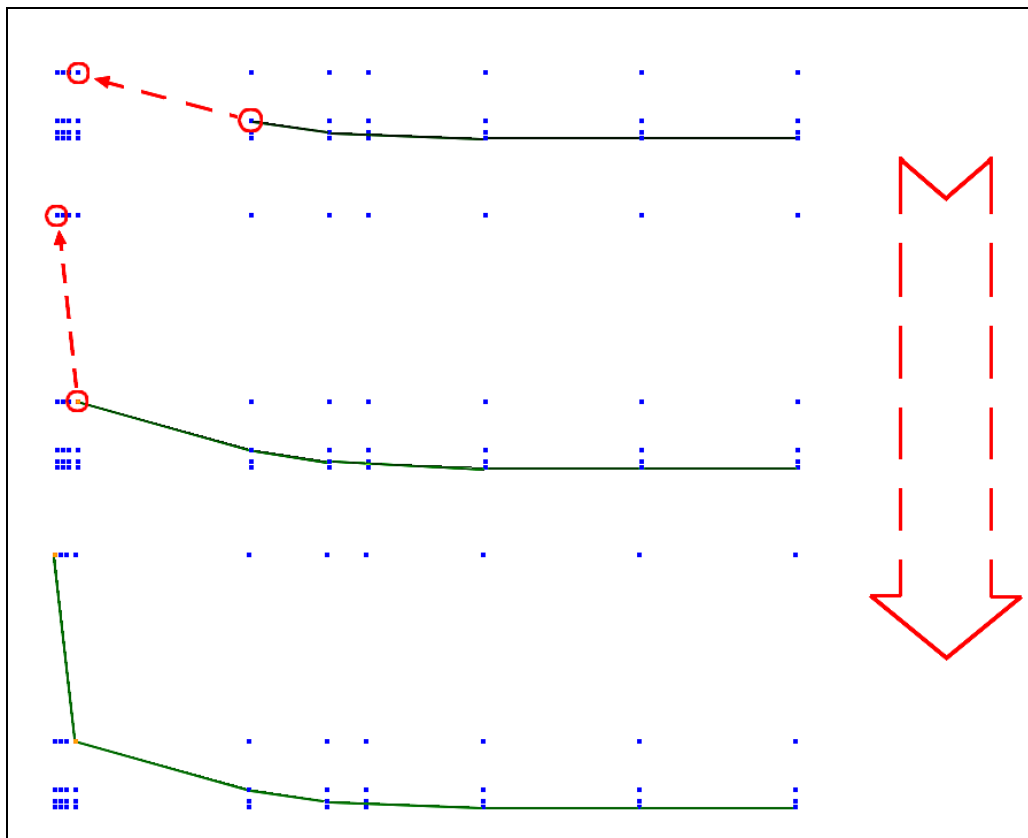


Figure 9: Drawing section lines

Now that the complete section has been drawn, LStrakes can be generated.

- Select **Generate Active Section Longitudinal Strakes** command from the Model menu. Make sure that the active section is the one with the section lines. LStrakes should be generated.
- Click on the LStrakes sub branch in the Project explorer. There should be a number of LStrakes generated, and their properties visible in the Data Table.

Double clicking on the LStrakes in the Model and FullModel branch will toggle on and off the display of that entity. Same applies to other entities in those branches.

- Double click on Points in the Model branch. Red points should appear in the view window, and point properties are loaded into Data table.

Panels are also created.

- Click on the Panels sub branch in the Elements branch of the PE. If the Selection tool is enabled, listing trough the Panels, LStrakes, Points and so on in the Data Table will highlight those entities orange, enabling the user to see which entity is in the focus at the moment.

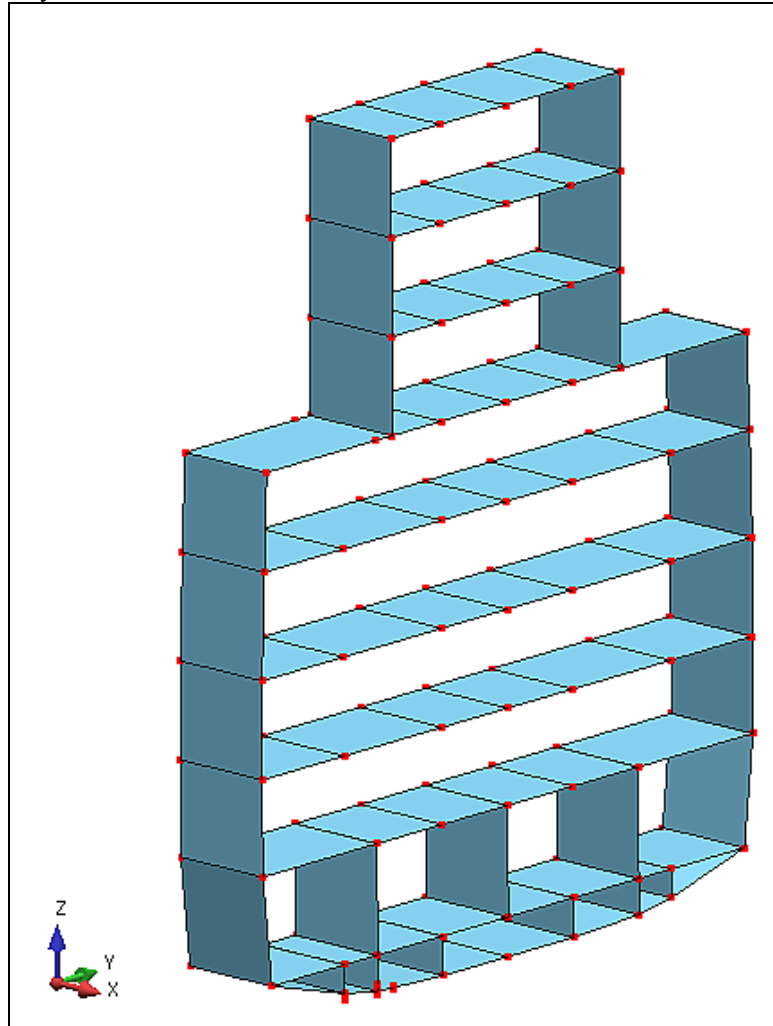


Figure 10: Shaded display of sections LStrakes and Points

### 6.2.3 View manipulation and quick selection

User can quickly manipulate the viewing angle using the mouse and CTRL/SHIFT keys.

- To rotate the model press and hold the mouse wheel and drag the mouse to rotate the view.
- To Pan the view hold the CTRL key and press and hold the mouse wheel while dragging the view.
- To Zoom out or in hold the SHIFT key and the mouse wheel combination (press and hold) while dragging the mouse forward and backward.

Panel and LStrakes are both graphically presented in the same way. This can cause problems with selection of particular entity. To select for example LStrake and not the

Panel, user has to click on Model branch (or anywhere in the Model branch) in the Project Explorer, and then on the LStrake of interest in the View window. Properties of that LStrake will be loaded into the Data Table. To select Panel instead of the LStrake, click on the Elements branch (or anywhere in it), and then on the LStrake in the View window. Now, the panel's properties will be loaded into the Data Table. Make sure that the Selection tool is enabled in both cases.

## 6.2.4 Adding structural entities to the model

Next stage of the modeling is adding structural entities into the model. Generation of stiffeners, girders and pillars will be shown

### 6.2.5 Stiffeners

As mentioned before, stiffeners are a part of panel. To add stiffeners to the panel follow the procedure:

- Click on the Panel sub branch in the Elements branch of the Project Explorer.
- Select the appropriate panel from the list in the Data Table area.
- Enter the appropriate StiffenerID to define the stiffener's shape (and it's structural characteristics). If not sure which profile to select, view the profile's characteristics under the Profiles branch first.
- Specify the number of stiffeners (NoOfStiffs field) and stiffener layout (Stiff Layout field). For example, layout for 2 stiffeners should look something like this 0.333\*0.667, which means that the first stiffener is placed at the 1/3 of width and the second at 2/3 of the panel's width. Press Enter to confirm the input then F9 to refresh the view. Stiffeners should appear on the altered panels. If stiffeners do not appear on the model, and yet they appear in the Project Explorer double click on them in the Project Explorer to activate them.
- If the stiffeners appear on the wrong side of the panel flip them around by selecting **Flip Normals** command from the model menu, but be sure to select the appropriate LStrake before that.

### 6.2.6 Girders (and Frames)

Girders rely on Points. Point's position and order define the location and orientation of the girder. To add a girder to the model to the following:

- Click on the Girders sub branch in the Model branch. Girder properties will appear in the Data Table.
- Click on the add button to add a new girder. Girder's unique ID will be created.
- Enter connection points' IDs under Connection field of the Data Table. For example enter 3\*10 in order to place a girder between points 3 and 10.
- Next specify the girder's profile shape and size by selecting the appropriate profile from the Profiles branch and entering its ID in the ProfileID field. Press Enter to confirm and F9 to refresh the view.
- To adjust the orientation or the angle of the girder modifies the appropriate field (Angle or Orient). For example, to flip the girder about its longitudinal axis enter 0\*0\*-1 under Orient or -180 deg under Angle.

To add and configure Frame just follow the procedure as with Girders. Only difference between Frame and Girder is their orientation (transversal versus longitudinal)

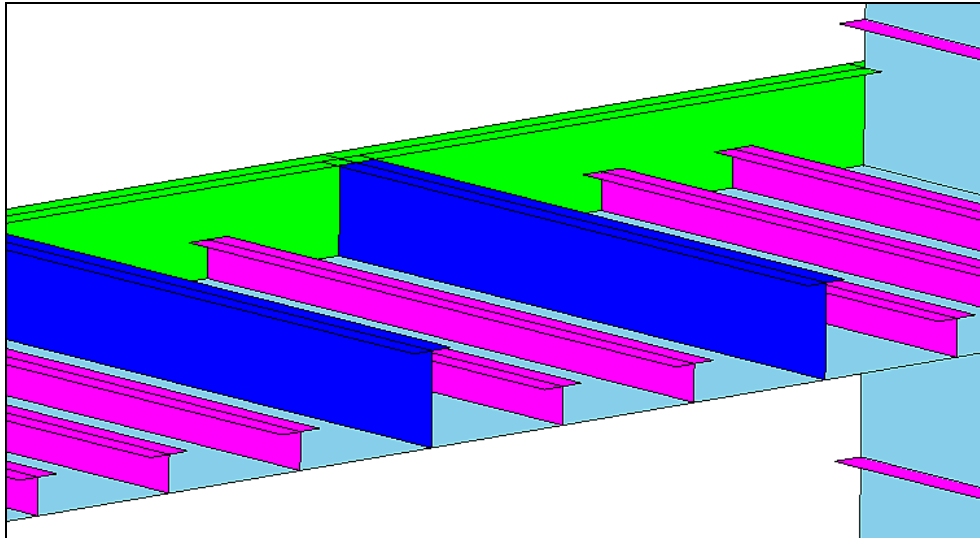


Figure 11: Stiffeners (purple) Girders and Frames (green)

### 6.2.7 Pillars

Generating pillars is same as with girders and frames. Pillars are symmetrical, so no Angle variable is available here. To add a Pillar:

- Select the Pillars sub branch in the Model branch of the PE.
- Click the add button in the Data Table to add a new pillar, New pillar ID is created.
- Assign the appropriate profile ID to the pillar. Note: Since a pillar is being created, CONSTRUCT will automatically apply a cylindrical profile from the profiles database. Since profiles in that database are ordered by type, so is their ID. Specifying 1 under profile ID here, will actually mean that profile ID 1 has been selected, but from the list of cylindrical profiles, and profile ID 1 overall.
- Then enter 2 connection points (their IDs, i.e. 23\*35). Press Enter and then F9. A pillar should appear on the model.

### 6.2.8 Openings

Openings are defined by their type and the position of their center point (opening point). To add an opening to the model:

- Select Openings from the Elements branch in the Project Explorer.
- Click Add button in the Data Table to add an opening.
- Specify the Type of the opening (rectangular or elliptical, RR or E)
- Enter the desired longitudinal and vertical dimensions in appropriate fields. Make sure to enter the dimension in millimeters. Press enter to confirm.

An opening has been defined.

Second stage is defining the center point of the opening. To do that:

- Select Opening Points from the Model branch in the Project Explorer.
- Click Add button to add a new opening point to the list
- Enter X,Y and Z coordinates respectively in the appropriate field to specify the desired location of the opening.

An opening point has been created.



Opening will not be visible until it is placed on an Entity. Openings can be placed on LStrakes, TStrakes, Girders and Frames. That's because opening's area is calculated and subtracted from the area of it's carrier, and in that way, carrier's mass and moments of inertia are reduced (LStrakes and TStrakes).

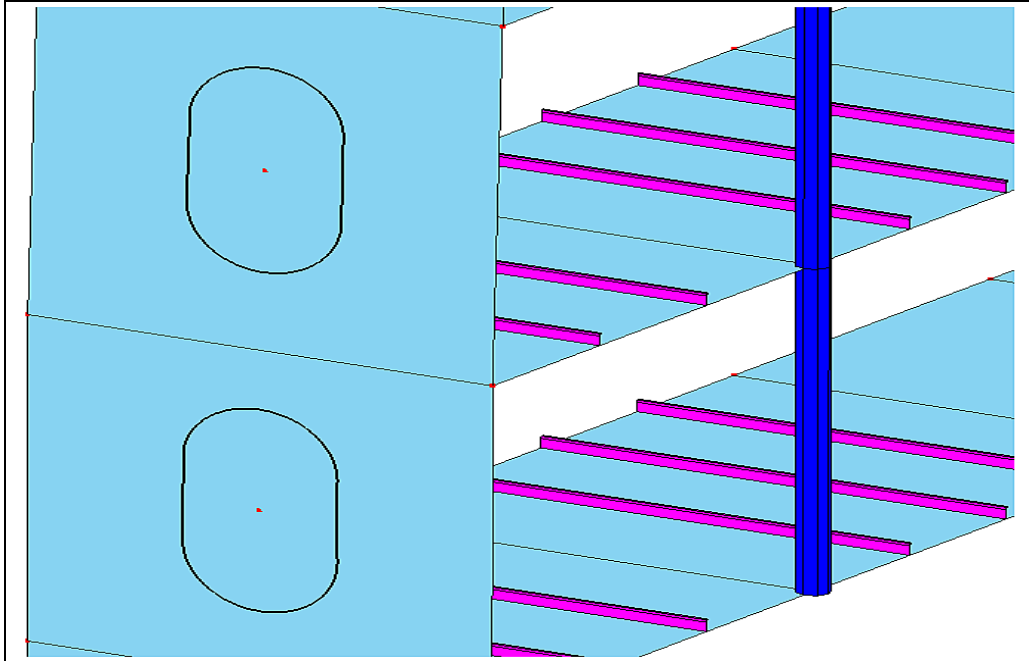


Figure 12: Pillars and Openings

To place the opening on a LStrake do the following:

- Select the appropriate LStrake (from the Project Explorer => Data Table list, or directly from View). It's properties will be loaded into the Data Table.
- Under Openings field, enter the appropriate combination of Opening ID\*Opening Point ID\* Orientation Angle, ( i.e. 0\*0\*90). Press Enter and F9 to refresh. Opening should appear at the location specified under opening point.

Note: It's not necessary for the location of the Opening Point and the LStrake surface to be coincident. The area will be subtracted from the LStrake anyway. But for esthetic reasons it is recommendable that the opening point is placed on the appropriate LStrake.

### 6.2.9 Creating TStrakes

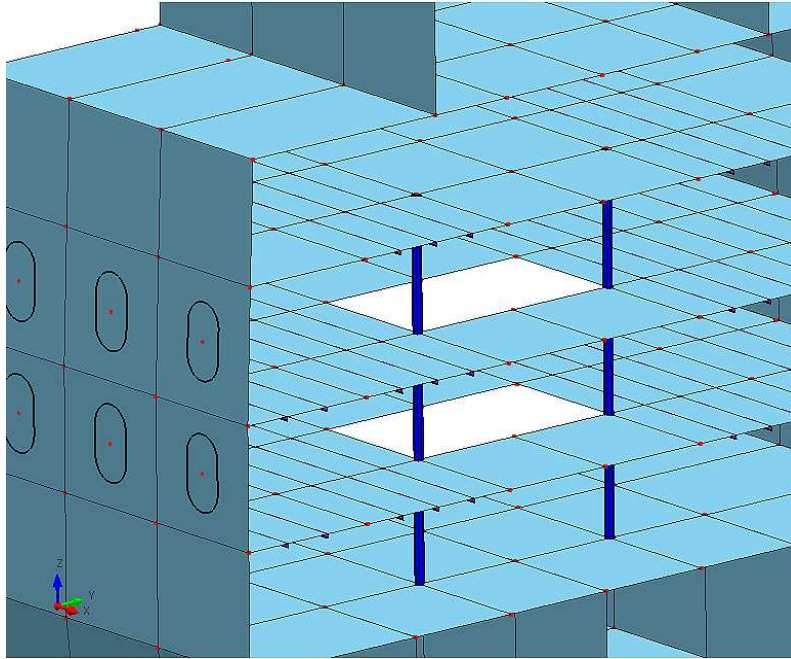
This is a simple procedure, because TStrakes rely on existing points. To create a TStrake:

- Select TStrakes from the Model branch.
- Click Add button to add a new TStrake.
- Specify 4 connection points (their IDs) of a TStrake (4 corners) under Connections field. Be sure to enter them in a clockwise or counterclockwise sequence. Press Enter and F9 when finished. TStrake should appear.

## 6.3 Creating a full model of a ship

### 6.3.1 Copying sections

Select the section you wish to copy from the Sections sub branch of the Model branch. Then, select **Copy Section** command from the Model menu. Input box will



launch. Enter longitudinal position of the new section and press enter to confirm. User can make modifications on that newly created section. For example, delete LStrakes, Girders, add openings and so on.

On the figure on the left, there is an assembly of 3 sections. This was made by copying the first section then deleting 2 strakes from the newly created section. After that first section was copied again but on a different longitudinal position (next to the second section).

### 6.3.2 Assembling sections into a full model

This is done in a simple way by lining up numbers of desired sections. To create a full ship model to the following:

- Select Model branch in the Project Explorer. Model properties will load into Data Table area.
- Under Section List field enter the desired section combination. For example 20\*1\*30\*2\*10\*1. This means that the full ship will be composed of 20 sections number 1, then 30 sections number 2 and 10 sections number 1 again, 60 sections total.
- Select **Generate Full Model** command from the Model menu.
- Select **Full Model** from the model menu or same command from the Toolbar Model drop down menu to display the full model.

### 6.3.3 Generating Coupled Beams

Before any analysis can be executed, model Coupled Beams have to be generated. It can be done automatically or manual. Automatic procedure is simple. User has to select **Generate Coupled Beams => Automatically** from the Model menu. Manual creation requires manual creation of Coupled Beam points and then coupled beams themselves. After the Coupled Beams have been created they will appear on the model shaded green.

The automatic procedure for Coupled Beams creation is not applicable to generic use, particularly in the case of complex geometry. In this case Coupled Beams can be defined based on the automatic generated points (**Generate Coupled Beams => Points**) and applying the manual definition of Coupled Beam (**Generate Coupled Beams => manual**). According to Coupled Beam Theory (Naar et al. 2004) it is recommended that the boundary of the Coupled Beam element is located into the middle of plate; see Figure 13.

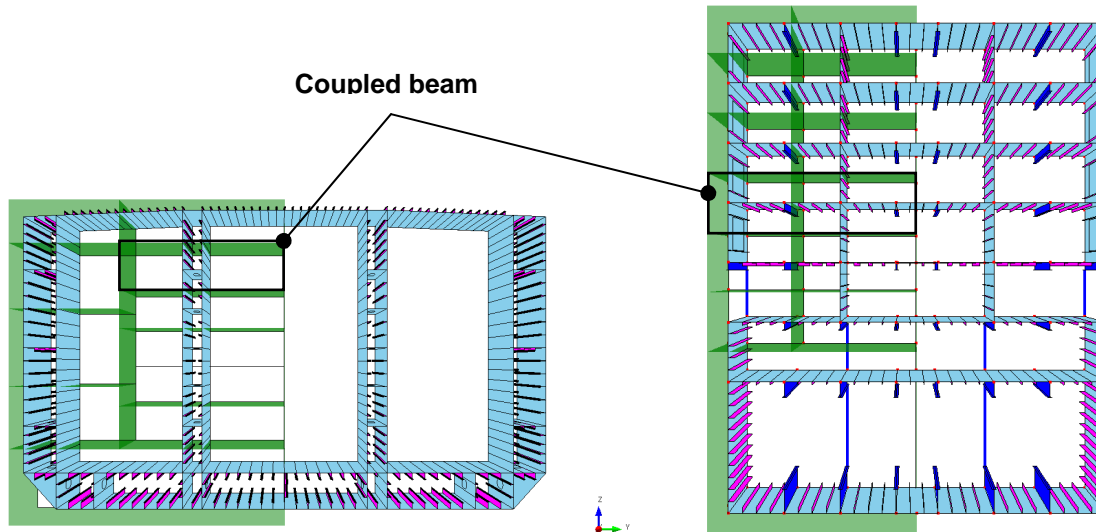


Figure 13: Examples of the deviation of Coupled Beams.

### 6.3.4 Creating Groups

It is useful to create groups of entities with same characteristics. Groups can be useful for easier viewing of the model, applying loads etc. there are two types of groups: Entity and description based. Entity groups group elements by their entity, and description by their description.

Here's the procedure for creating a group :

- Select Groups from the Project Explorer.
- Click on the Add button in the Data Table area to add a new group to the list.
- Enter Group Type, Entity or Description.
- If an Entity based group is to be created, enter appropriate entity under Entity type. For example to group LStrakes of the section input has to look like this : Model\*LStrakes. Make sure that the display option is on Model and not Full model when the grouping is done within the section.
- Select Selection tool and circle the area of interest. Press F9. Number of Entities field (Groups properties, Data Table area) should display the number of specified entities found among the selected elements.

When creating Description based groups user must provide both Entity Type and Entity Description.

## 6.4 Analyzing the model

### 6.4.1 General

This chapter will explain how to create and apply loads to the model, run a Coupled Beams analysis and display the results.

## 6.4.2 Creating and applying loads

To apply a load to the model it is necessary to create some loads.

Creating loads:

- Expand the Loads branch of the Project Explorer and select which load type to create. The simplest example is the Global load in form of a bending moment.
- Click Add button to add Global load. Its ID will be automatically generated.
- Specify the magnitude of the bending moment, then press Enter to confirm.

Now that the load has been created, it is possible to apply it to the model. To apply the load, it is necessary to create a Load Case. Do the following:

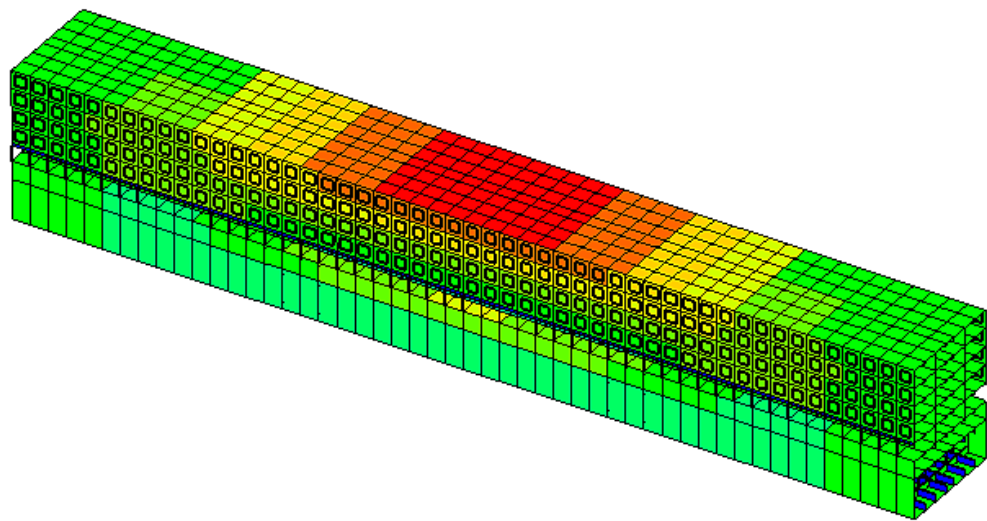
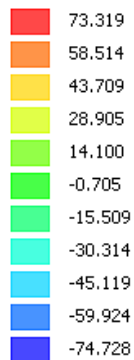
- Click on the LCs&BCs branch.
- Click Add button to create a load case. ID will be automatically assigned.
- Specify the list of loads to be applied (Load ID's field). This means that the user can combine several types of loads, for example bending moment and Hydrostatic load. Simply specify the IDs of appropriate loads and make sure those loads are created before.
- Next, if necessary, specify the groups (GroupIDs) of elements that will be subjected to the load case. If none is specified, load case will be applied to the whole model.

Grouping is important in order to apply non global loads, such as Pressure and Point Loads. After such groups of elements have been created, it is necessary to specify them when applying loads.

Load Case has been created. It's possible to create a number of load cases like this with various load combinations.

## 6.4.3 Evaluating global response

To run a Coupled Beams Solver select **Calculate** from the Response menu. Solver will calculate each load case respectively and store the results. To view the results of each load case, select the appropriate load case results from the Toolbar's load case drop down list. To view a different set of results (Normal Stress, Shear Stress etc) select the appropriate results from the Display mode drop down list. Figure below shows the response (normal stress  $x$ ) of the model subjected to a global bending moment.

**NORMAL STRESS X**

**Figure 14: Normal Stress response of a full ship model subjected to a global bending moment**

#### 6.4.4 Strength analysis based on global and local response

To calculate the fulfillment of design criteria, select Calculate from the Design Criteria menu. To view the results for a certain criterion select the appropriate results from the Usage display drop down list.

Strength criteria are calculated based on recommendations and rules from DNV (2005), Hughes (1988) and Hughes *et al.* (2004), and are described in Appendix 2. They include:

- plate yield,
- stiffener yield,
- plate buckling,
- stiffener lateral buckling,
- stiffener torsional buckling (tripping),
- stiffener's web buckling,
- stiffener's flange buckling,
- crossover.

A crossover criterion ensures controlled panel collapse due to extensive in-plane loading, where plating between stiffeners should fail first. Physically this means that the panel is not allowed to consist of thick plate and weak stiffening, and the stiffener size has to rise with plate thickness.

#### 6.4.5 Running optimization process

Before starting the optimization, the lower and upper bound of each variable must be defined in the group datasheet. They determine the possible range, where variable value can be chosen in the optimization process (e.g. setting the plate thickness of certain strake in range of 5-20 mm). However, not all of these values are permitted, since the criteria mentioned in the previous sub-chapter have certain maximum allowed value, so the design has to simultaneously satisfy all of them. Since they

restrict variable values, they are called the constraints. The task of the optimization is to find designs that are the best according to objectives (property that is to be minimized or maximized), but which satisfy all constraints, i.e. they are feasible. Objectives that are used in CONSTRUCT are weight, cost and VCG of the structure.

Optimization is performed using a robust optimization method – genetic algorithm (GA), which has been modified to efficiently handle structural optimization problems with various amount and type of constraints. More particularly, VOP (GA that is used in CONSTRUCT), handles all problems as unconstrained by actively minimizing constraint violation, alongside the original objectives, in a transformation called ‘vectorization’ (Klanac and Jelovica 2008). Therefore, the number of objectives in each problem is equal to the sum of constraints and original objectives, and the minimization importance of each of them is controlled by their weight factors. VOP can handle different number of constraints, thus being able to optimize different levels of complexity. Being in essence the genetic algorithm, it is a stochastic search method that mimics the evolution of living organisms from the nature. It requires evaluation of numerous designs which replace the calculation of gradients that are necessary in traditional optimization methods, which would be difficult or impossible to derive due to the complexity of the constraints.

The optimization is started with randomly created alternatives, using uniform distribution between variable bounds. The amount of those designs is defined by user and is known as population size. After evaluating all design criteria for every alternative in the population, the algorithm is run to produce new, the better set of designs. That completes one generation, and the same process is repeated until the determined number of generations is reached. GA’s working principle is thoroughly described in e.g. Goldberg (1989) and Deb (2001).

Optimization process can be run after specifying all necessary parameters concerning the optimization. To obtain automatically calculated initial values for the parameters user has to select **Apply MinMax Values** from the Optimisation menu. Optimisation parameters are:

- NV: number of variables, automatically calculated based on the structural model and groups used
- POP: population size, defines the number of design alternatives that will be evaluated in each generation. It should be an even number in range 50-100, increasing with the complexity of the structure.
- NCON: number of constraints, automatically calculated as 8 times of NV.
- NGEN: number of generations, defines how many times will the new set of designs (of size POP) be created and evaluated. Simultaneously, it is a stopping criterion that ends the optimization.
- PC: probability of crossover, defines the probability that two randomly selected designs from the population will swap part of their variables. Commonly, the best results are obtained using PC in range 0.7-0.9.
- PM: probability of mutation. Each design alternative is represented with binary string, and PM determines the probability of flipping each bit. This random variation is essential for the proper functionality of the optimizer. Typical PM is calculated as  $1/(\text{number of bits representing the design})$ , meaning that only one bit per design will be changed, consequently only one variable (Bäck 1993, Deb 2001). Using this expression, the value of PM is automatically provided by CONSTRUCT, however it can be enlarged to 2 times of the original PM, as used by Klanac et al. (2008) on tanker’s main frame

optimization. Defining PM outside of this recommended range can result in slow optimization progress, difficulties to retain feasible designs or even deterioration of initial designs.

- **WO:** weight factors of the original objectives. The total number of objectives in a problem is equal to the sum of constraints and original objectives ( $J+M$ ), where each of them by default receives the same weight factor equal to  $1/(J+M)$ . There is a possibility to manually change weight factors for the original objectives, where the increase in value means their greater importance. By altering them during the optimization, better designs with respect to particular objective can be obtained. However, sum of weight factors for  $J+M$  objectives is equal to one, and user should keep in mind that certain emphasis must be given to minimize constraint violation, otherwise majority of designs can become infeasible. For problems with e.g. 100 or more constraints, WOs that CONSTRUCT provides can be manually increased up to five times. Through discussion on the value of weight factors can be found in Klanac and Jelovica (2008a), where the principle that VOP is using in CONSTRUCT is named VO-2.

The optimisation can be terminated at any time by clicking the stop optimisation button in menu bar; see Figure 15.

#### 6.4.6 Review of optimisation results

Optimisation creates Optimization.txt file. User can import this file to CONSTRUCT using **Import Results** command from Optimisation menu. To review the optimisation results, user has to activate **DA** from Drop Down menus and scroll the design alternatives in **DesAlternatives datasheet**.

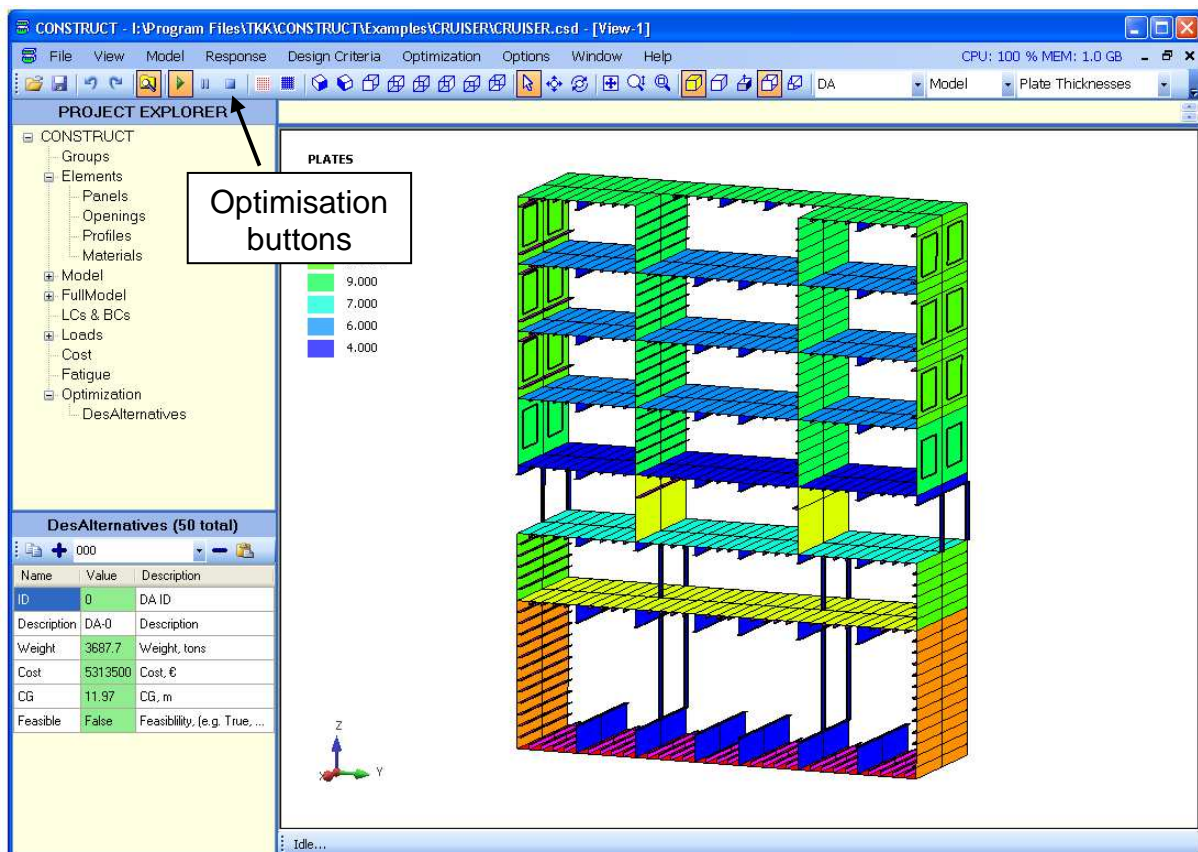


Figure 15: The reviewing of optimisation results in CONSTRUCT.

## 7 EXAMPLE MODELS

To get user familiar to CONSTRUCT, the software includes three simple example models shown in Figure 16: Box beam, Cruiser and Tanker. Model files are available in ...TKK\CONSTRUCT\Examples\ folder.

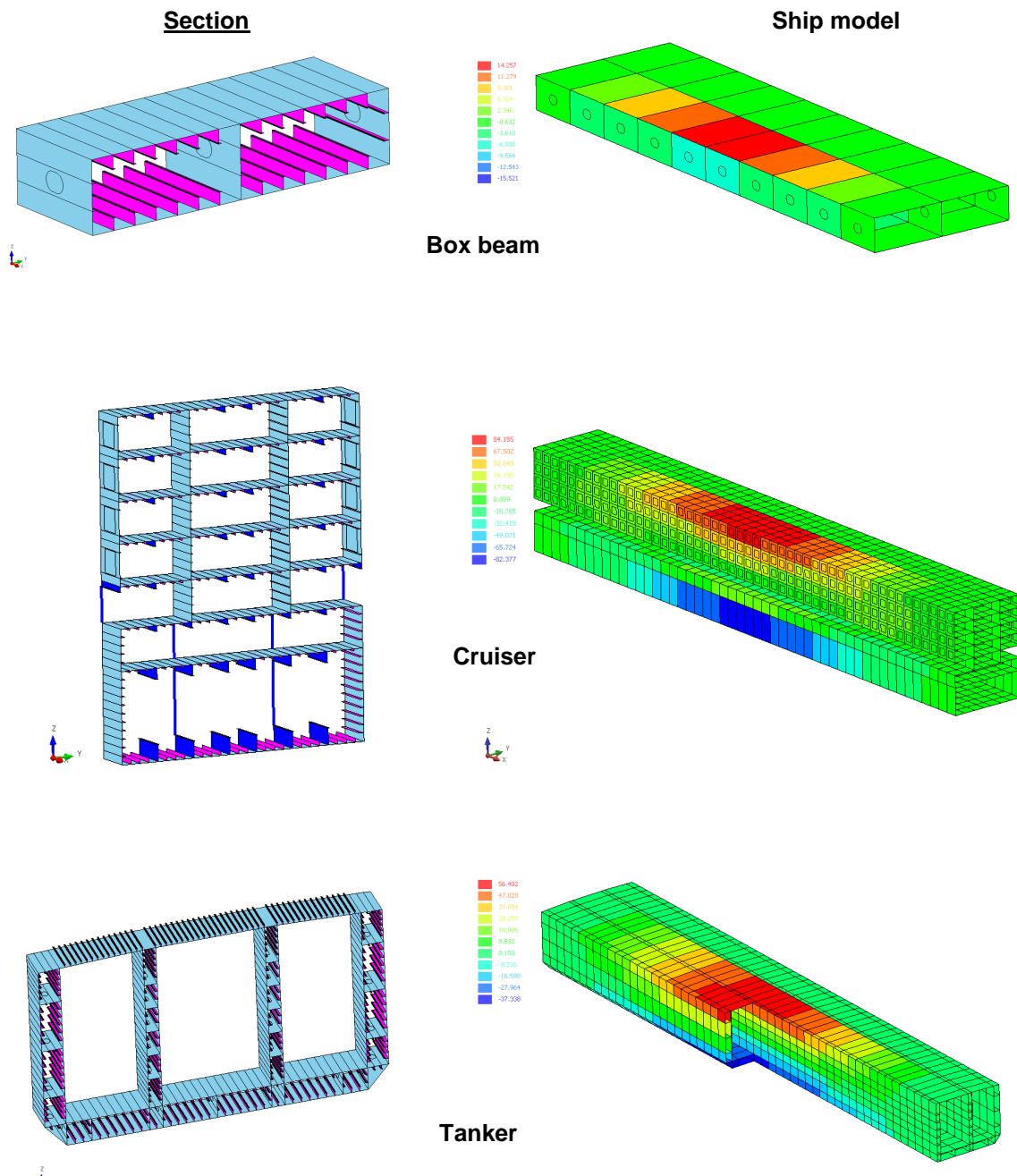


Figure 16: The example model included to CONSTRUCT software.



## 8 REFERENCES

Bäck, T., *Optimal Mutation Rates in Genetic Search*, in Proc. 15th International Conference on Genetic Algorithms, 1993.

Deb, K., *Multi-Objective Optimization using Evolutionary Algorithms*, Wiley, Chichester, UK, 2001.

Det Norske Veritas, *Rules for the classification of steel ships*, Høvik, 2005.

Goldberg, D.E., *Genetic Algorithms in Search, Optimization and Machine Learning*, Reading, MA: Addison-Wesley, 1989.

Hughes, O.F., *Ship Structural Design*, Society of Naval Architects and Marine Engineers, Wiley, New York, 1988.

Klanac, A., Jelovica, J., Niemeläinen, M., Remes, H., Romanoff, J., *Structural Omni-Optimization of a Tanker*, Proc. 7<sup>th</sup> International Conference on Computer Applications and Information Technology in the Maritime Industries – COMPIT '08, Liege, Belgium, 2008b.

Klanac, A., Jelovica, J., *Vectorization and constraint grouping to enhance optimization of marine structures*, submitted to Marine Structures, 2008a.

Mantere, H., *Strength Analysis for the Design of Local Ship Structures in Concept Stage*, Master's thesis, Helsinki University of Technology, 2007

Naar, H., Varsta, P. & Kujala, P., A theory of coupled beams for strength assessment of passenger ships. Marine Structures; Design, Construction & Safety, 2004. Vol. 17, nro 8, 590-611.

Hughes, O.F., Ghosh, B., Chen, Y., *Improved prediction of Simultaneous local and overall buckling of stiffened pannels*, Thinn-Walled Structures, 2004, Vol. 42, p. 827-856.

Remes H, Klanac A, Varsta P, Ehlers S. *ConStruct - Platform for Conceptual Structural Design*. IMPROVE- Conference, Dubrovnik, Croatia 17-19. Sept. 2009.

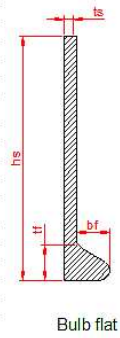
## 9 APPENDIXES

### 9.1 Datasheet file examples

Material and profile datasheet files are made in MS Office Excel format. Profile datasheets contains, along with the profile dimension data, profile's cross section which defines dimensions. User is free to add data to the datasheets, obeying the data formats.

Here's an example of a profile datasheet based on Ruukki:

1	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20
Material	hs	ts	bf	tf	As	W	ns_x	ns_y	ls_x	ls_y	PROFILE DATABASE (Based on Ruukki)							
	mm	mm	mm	mm	mm <sup>2</sup>	kg/m	mm	mm	mm <sup>4</sup>	mm <sup>4</sup>	hs = height of stiffener							
3 A	100	5	15,5	14,8	6,750E+02	5,4	61,3	4,8	6,700E+05	1,480E+04	ts = thickness of stiffener (web)							
4 A	100	6	15,5	14,8	7,740E+02	6,2	59,5	5,1	7,600E+05	1,700E+04	bf = breadth of flange							
5 A	100	7	15,5	14,8	8,740E+02	7,0	58,7	5,5	8,500E+05	1,990E+04	tf = thickness of flange							
6 A	100	8	15,5	14,8	9,750E+02	7,8	57,8	5,9	9,400E+05	2,310E+04	As = area of stiffener							
7 A	120	5	17	17,7	8,120E+02	6,5	73,9	5	1,170E+06	1,980E+04	W = weight of stiffener							
8 A	120	6	17	17,7	9,320E+02	7,5	72	5,3	1,330E+06	2,340E+04	ns_x = distance of centre of gravity in x-direction							
9 A	120	7	17	17,7	1,052E+03	8,4	70,7	5,6	1,480E+06	2,700E+04	ns_y = distance of centre of gravity in y-direction							
10 A	120	8	17	17,7	1,172E+03	9,4	69,6	6	1,640E+06	3,100E+04	ls_x = moment of inertia; axis x-x							
11 A	140	7	19	19,7	1,243E+03	9,9	83,1	5,9	2,410E+06	3,800E+04	ls_y = moment of inertia; axis y-y							
12 A	140	8	19	19,7	1,383E+03	11,1	81,8	6,3	2,660E+06	4,320E+04	Profile id: B*hs*ts							
13 A	140	10	19	19,7	1,663E+03	13,3	79,2	7	3,160E+06	5,660E+04								
14 A	160	7	22	22,2	1,460E+03	11,7	96,6	6,4	3,730E+06	5,660E+04								
15 A	160	8	22	22,2	1,620E+03	13,0	94,9	6,8	4,110E+06	6,550E+04								
16 A	160	9	22	22,2	1,780E+03	14,2	93,6	7,1	4,480E+06	7,320E+04								
17 A	180	8	25	25,5	1,886E+03	15,1	109	7,4	6,090E+06	9,900E+04								
18 A	180	9	25	25,5	2,066E+03	16,5	107	7,7	6,630E+06	1,093E+05								
19 A	180	10	25	25,5	2,246E+03	18,0	106	8,1	7,170E+06	1,205E+05								
20 A	200	9	28	28,8	2,366E+03	18,9	121	8,4	9,410E+06	1,576E+05								
21 A	200	10	28	28,8	2,566E+03	20,5	119	8,7	1,020E+07	1,721E+05								
22 A	200	12	28	28,8	2,966E+03	23,7	117	9,4	1,160E+07	2,046E+05								
23 A	220	10	31	32,1	2,900E+03	23,2	134	9,3	1,400E+07	2,389E+05								
24 A	220	12	31	32,1	3,340E+03	26,7	130	10	1,590E+07	2,798E+05								
25 A	240	10	34	35,4	3,249E+03	26,0	147	10	1,860E+07	3,234E+05								
26 A	240	11	34	35,4	3,489E+03	27,9	146	10,3	2,000E+07	3,481E+05								
27 A	240	12	34	35,4	3,729E+03	29,8	144	10,6	2,130E+07	3,743E+05								
28 A	260	10	37	38,7	3,611E+03	28,9	162	10,7	2,477E+07	4,284E+05								
29 A	260	11	37	38,7	3,871E+03	31,0	160	11	2,610E+07	4,590E+05								
30 A	260	12	37	38,7	4,131E+03	33,0	158	11,3	2,770E+07	4,911E+05								



Notice that column captions are similar to the property captions of Profiles elements. User can add profile data here or directly in CONSTRUCT. Same applies to the Material Datasheet.

## 9.2 Design criteria

Loads can be divided on primary, secondary and tertiary components, which enable separate analysis of the response. To obtain the total stress a simple superposition of the three components can be made. Primary response comes from the bending of the hull girder when exposed to external vertical loads for which the Coupled Beam method (Naar *et al.* 2004) is used. Secondary response is the consequence of possible local loads imposed on the part of the structure, e.g. cargo on the deck, while the tertiary comes from laterally loaded plates, e.g. by hydrostatic sea pressure. For better explanation of criteria presented below refer to Mantere (2007).

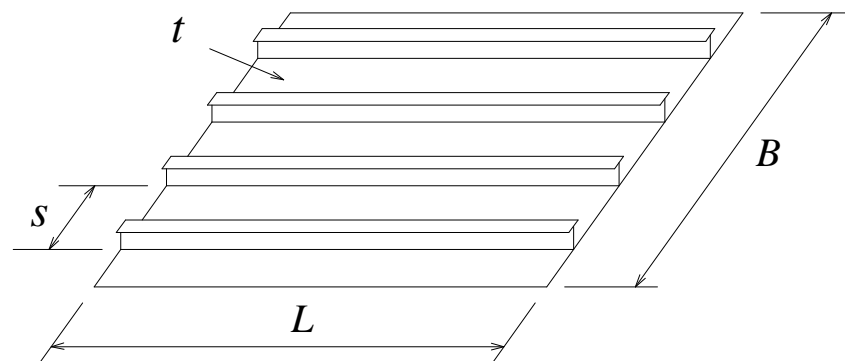


Figure 17 Dimensions of the stiffened panel: length of s. panel ( $L$ ), breadth of s. panel ( $B$ ), stiffener spacing ( $s$ ), thickness of plate ( $t$ )

### 9.2.1 Plate yield

Due to stress components that come from the vertical forces, the equivalent stress  $\sigma_{ekv}$  can be calculated using the Mises-Hencky yield criterion from which the material yields when:

$$\sigma_{ekv} = \sqrt{\sigma_x^2 - \sigma_x \sigma_y + \sigma_y^2 + 3\tau_{xy}^2} = \sigma_Y \quad (1)$$

and the components are obtained with the Coupled Beam method. Tertiary stress component is calculated using the following equation:

$$\sigma_3 = 0.25 \cdot k_a^2 \cdot p \cdot \left(\frac{s}{t}\right)^2, \quad (2)$$

where:

$k_a$  = aspect ratio correlation factor,

$p$  = lateral pressure,

$s$  = stiffener spacing,

$t$  = plate thickness.

### 9.2.2 Stiffener yield

The expression for  $\sigma_{ekv}$  from plate yield criterion is used for primary and secondary response, while the maximum bending moment for the tertiary response is calculated using:

$$M = \frac{psL}{12}, \quad (3)$$

where  $L$  is the length of the stiffener, m, and 12 is a factor defined by boundary condition (DNV 2005).

### 9.2.3 Plate buckling

Plate buckling criterion comes from:

$$\frac{\sigma_{ax}}{\sigma_{cx} \cdot q} - K \frac{\sigma_{ax} \cdot \sigma_{ay}}{\sigma_{cx} \cdot \sigma_{cy}} + \left( \frac{\sigma_{ay}}{\sigma_{cy} \cdot q} \right)^n \leq 1, \quad (4)$$

where:

$\sigma_{ax}$  = compressive stress in axial direction,

$\sigma_{ay}$  = compressive stress in transverse direction,

$\sigma_{cx}$  = critical stress in axial direction,

$\sigma_{cy}$  = critical stress in transverse direction,

$K, n$  = factors depending on the aspect ratio, thickness, yield strength and Young's modulus of the plate.

Parameter  $q$  is a function of actual shear stress  $\tau_a$  and critical shear stress  $\tau_c$ , according to expression:

$$q = 1 - \left( \frac{\tau_a}{\tau_c} \right)^2. \quad (5)$$

### 9.2.4 Stiffener lateral buckling

Euler buckling equation is used to predict critical elastic buckling stress:

$$\sigma_{el} = \pi^2 E \frac{I_x}{AL^2}, \quad (6)$$

where:

$E$  = Young's modulus,

$I_x$  = moment of inertia,

$A$  = cross-sectional area of the stiffener and with the attached plate (effective width of the stiffener plate flange is estimated to be 80% of the stiffener spacing).

### 9.2.5 Stiffener torsional buckling (tripping)

Buckling stress for the torsional mode of the stiffeners attached to the plate can be estimated using:

$$\sigma_{el} = \frac{\pi^2 EI_W}{10^4 I_P L^2} \left( m^2 + \frac{K}{m^2} \right) + 0.385 \frac{I_T}{I_P}, \quad (7)$$

Factor  $K$  is calculated as:

$$K = \frac{CL^4}{\pi^4 EI_W} 10^6. \quad (8)$$

where:

$C$  = spring stiffness exerted by supporting plate panel,

$m$  = is the number of half waves estimated between 1 and 4 on the basis on  $K$ ,

$I_W$  = sectorial moment of inertia,

$I_T$  = St Venant's moment of inertia,  
 $I_P$  = polar moment of inertia.

### 9.2.6 Stiffener's web and flange buckling

Elastic buckling stress is calculated as:

$$\sigma_{el} = 3.8E \left( \frac{t_w}{h_w} \right)^2, \quad (9)$$

where:

$t_w$  = web thickness,

$h_w$  = web height.

### 9.3 Napa steel macro

Example call of NAPA macro:

```
!add NAPA_CS_EXPORT(#99 #103 - - - 'C:/TEMP/' 'NAPA_STEEL_EXPORT.txt' 0.01)
```

NAPA macro:

```
@parameters(limits=s directory=s file=s tol=n)
@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@
@@@ THIS MACRO CREATES AN OUTPUT LIST FROM NAPA-STEEL @@@
@@@ STRUCTURE TO CONSTRUCT-TOOL. @@@
@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@

ST
LIMITS OFF

NL 'STEEL OUTPUT - CONSTRUCT'
@@ Parsing limits
@limArr=arr(3)
@n=parse(limits, limArr)

!PAGE 120,2000

LF 5
TYPE SECTION 1 (@limArr(1)_->@limArr(2)_

@@@@ STRUCTURE DATA @@@@@@
LIMITS @valueof(limArr(1))-tol @valueof(limArr(2))-tol @limArr(3) @limArr(4) @limArr(5) @limArr(6)
LIMITS ON

ST
LF 2
TYPE STRUCTURE DATA
LF 1
LQ STE NAME(F=16) STYPE MAT(F=4) PLTH XMIN(F=7.2) XMAX(F=7.2) YMIN(F=7.3) YMAX(F=7.3) ZMIN(F=7.3)
ZMAX(F=7.3) ORNT
TOO STE HD=(S, U, UL)
LIST STE ALL
LF 5

@@@@ CONNECTIONS BETWEEN STRUCTURES @@@@@
LIMITS OFF
LIMITS - - @limArr(3) @limArr(4) @limArr(5) @limArr(6)
LIMITS ON

ST
TYPE CONNECTIONS BETWEEN STRUCTURES AT XMIN
LF 1

LQ STC NAME, REFC, X, Y, Z, ANG, PRI
TOO STC HD=(S, U, UL) field=*2
LIST STC ALL X=@valueof(limArr(1))-tol P
@@@@LIST STC ALL X=@limArr(1) P
LF 5

ST
TYPE CONNECTIONS BETWEEN STRUCTURES AT XMAX
LF 1

LQ STC NAME, REFC, X, Y, Z, ANG, PRI
TOO STC HD=(S, U, UL) field=*2
LIST STC ALL X=@valueof(limArr(2))-tol P
@@@@LIST STC ALL X=@limArr(2) P
LF 5

@@@@ STIFFENERS @@@@@
LIMITS OFF
LIMITS @valueof(limArr(1))-tol @valueof(limArr(2))-tol @limArr(3) @limArr(4) @limArr(5) @limArr(6)
LIMITS ON
ST
TYPE STIFFENERS
LF 1
```

```
LQ STI NAME(F=16) PROFID(F=16) MAT(F=4) COORD(F=9.2) AXIS(F=4) IDIR(F=4) XMIN(F=6.2) XMAX(F=6.2) YMIN(F=6.2)
YMAX(F=6.2) ZMIN(F=6.2) ZMAX(F=6.2)
TOO STI HD=(S, U, UL)
LIST STI ALL
LF 5
```

```
@@@@@ OPENINGS @@@@@@@@@@
LIMITS OFF
LIMITS @valueof(limArr(1))-tol @valueof(limArr(2))-tol @limArr(3) @limArr(4) @limArr(5) @limArr(6)
LIMITS ON
```

```
ST
TYPE OPENINGS
LF 1
```

```
LQ OPN NAME(F=12) FORM(F=11) LENX LENY CGXA CGYA CGZA ANG
TOO OPN HD=(S, U, UL) field=*2
LIST OPN ALL
LF 5
```

```
@@@@@ PLATES @@@@@@@@@@
LIMITS OFF
LIMITS @valueof(limArr(1))-tol @valueof(limArr(2))-tol @limArr(3) @limArr(4) @limArr(5) @limArr(6)
LIMITS ON
```

```
ST
TYPE PLATES FROM XMIN TO XMAX
LF 1
```

```
LQ PLA NAME(F=12) PLATE(F=6) PLTH XMIN(F=7.2) XMAX(F=7.2) YMIN(F=7.3) YMAX(F=7.3) ZMIN(F=7.3) ZMAX(F=7.3)
TOO PLA HD=(S, U, UL) field=*2
LIST PLA ALL
LF 5
```

```
LIMITS OFF
LIMITS @valueof(limArr(1))-10*tol @valueof(limArr(1))-tol @limArr(3) @limArr(4) @limArr(5) @limArr(6)
LIMITS ON
```

```
ST
TYPE PLATES AT XMIN
LF 1
```

```
LQ PLA NAME(F=12) PLATE(F=6) PLTH XMIN(F=7.2) XMAX(F=7.2) YMIN(F=7.3) YMAX(F=7.3) ZMIN(F=7.3) ZMAX(F=7.3)
TOO PLA HD=(S, U, UL) field=*2
LIST PLA ALL
LF 5
```

```
LIMITS OFF
LIMITS @valueof(limArr(2))-10*tol @valueof(limArr(2))-tol @limArr(3) @limArr(4) @limArr(5) @limArr(6)
LIMITS ON
```

```
ST
TYPE PLATES AT XMAX
LF 1
```

```
LQ PLA NAME(F=12) PLATE(F=6) PLTH XMIN(F=7.2) XMAX(F=7.2) YMIN(F=7.3) YMAX(F=7.3) ZMIN(F=7.3) ZMAX(F=7.3)
TOO PLA HD=(S, U, UL) field=*2
LIST PLA ALL
LF 5
```

```
LIMITS OFF
LIMITS @valueof(limArr(1))-tol @valueof(limArr(2))-tol @limArr(3) @limArr(4) @limArr(5) @limArr(6)
LIMITS ON
```

```
SCAN
1
1
@outputFile=os.str("\",directory,"\",>',file)
send @outputFile
END
```