BUREAU
VERITAS

MARS2000

User's guide

MARS 2000

1. **INTRODUCTION SHELL / BASIC SHIP DATA**
2. **DEFINITION OF A SECTION**
3. **CALCULATION OF A SECTION**
4. **DEFINITION OF A BULKHEAD ARRANGEMENT**
5. **CALCULATION OF A BULKHEAD ARRANGEMENT**
6. **DEFINITION AND CALCULATION OF A TORSION MODEL**

MARS2000
User's guide

Booklet 1

INTRODUCTION
SHELL / BASIC SHIP DATA

February 2000 /1
Dev/UG/MARS2000/01

MARINE DIVISION
DEVELOPMENT DEPARTMENT
17 bis, place des Reflets – La Défense 2
92077 Paris – La Défense Cedex
Tel.: + 33 (0)1 42 91 52 91
Fax: + 33 (0)1 42 91 52 98

TABLE OF CONTENTS

Chapter 1 : GENERAL COMMENTS

1.1 GENERAL

MARS2000 is a software tool that allows to input sections, bulkheads and torsion models; it is organized around the following seven modules:

- MARS2000 (Shell).
- MIRE2000 (Basic Ship Data).
- MarsIn2000 (Section's Input).
- MarsRule2000 (Section's Check).
- BhaIn (Bulkhead Arrangement's Input).
- BhaRule (Bulkhead Arrangement's Check).
- Torsion (Input of torsion model and check).

The geometry and the scantling are defined using an user friendly process. MARS2000 checks that the actual local scantlings (Hull girder strength, plating and ordinary stiffeners) are in accordance with the Rules.

The program is updated each time the Rules are modified.

Chapter 2 : SHELL

2.1 MAIN FEATURES

The Mars shell module allows creating a new database or choice an existing database. The module is organized around the following application:

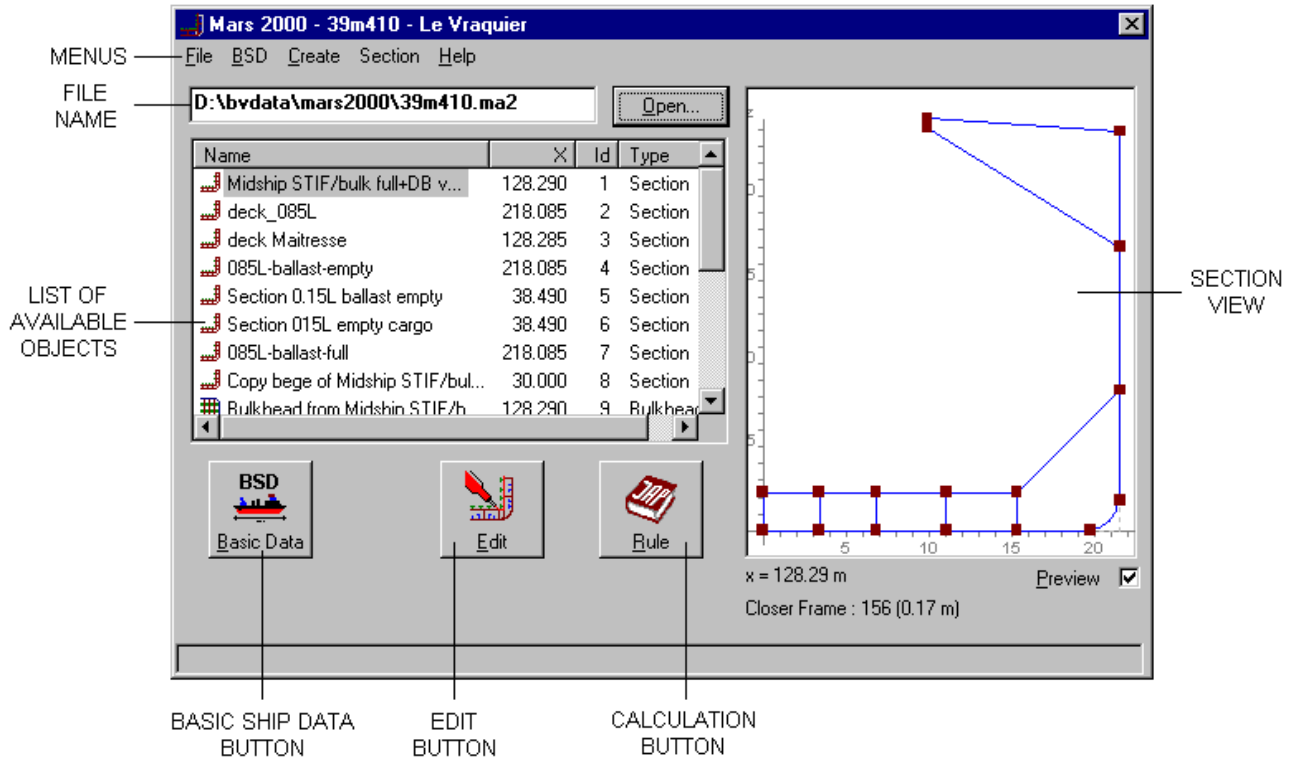


Figure 1 : MARSSHELL

List of available objects: displays all the existing objects in the database.

Basic Ship Data button: run the Basic Ship Data program.

Edit button: run the MarsIn2000 program if the user have choice a section in the list of available objects. In case of Bulkhead Arrangement the button run the BhaIn program. In case of torsion model the button run the Torsion program.

Rule button: run the MarsRule2000 program if the user have choice a section in the list of available objects. In case of Bulkhead Arrangement the button run the BhaRule program.
WARNING: BUTTON NOT AVAILABLE IN TORSION MODEL CASE

Section view: displays a view of the section (not available in case of torsion model).

2.2 MENUS

File Menu
It allows to manage the databases (save, open…) and to quit MARS2000.

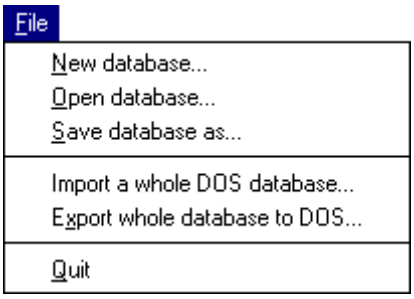


Figure 2 : FILE MENU

Item	Use
New database…	Creates a new database.
Open database…	Opens an existing database.
Save database as…	Saves the opened database.
Import a whole DOS database…	Imports from a old MARS DOS database (.mar)
Export whole database to DOS…	Exports to a old MARS DOS database (.mar)
Quit	Quits MARS2000

BSD Menu
It allows undoing the last action or copy the section drawing to clipboard.

Figure 3 : BSD MENU

Item	Use
Open…	Runs Basic Ship Data program.
Import …	imports BSD from a file .mar, .maw, .ma2, .bsd

Create Menu
It allows creating a new section, bulkhead arrangement or model for torsion.

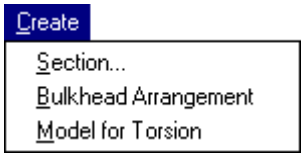


Figure 4 : CREATE MENU

Item	Use
Section…	Creates a new section.
Bulkhead Arrangement	Creates a new bulkhead arrangement.
Model for Torsion	Creates a new model for torsion.

Section Menu
It allows managing the objects.

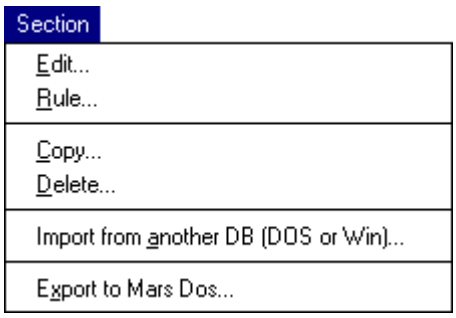


Figure 5 : SECTION MENU

Item	Use
Edit…	Runs input module related to selected object.
Rule…	Runs check module related to selected object.
Copy…	Copy the selected object.
Delete…	Delete the selected object.
Import from another DB (DOS or Win)…	Imports section from a file .mar, .maw, .ma2
Export to Mars Dos…	Exports to a old section DOS (.mar)

Chapter 3 : BASIC SHIP DATA

3.1 GENERAL COMMENTS

3.1.1 MAIN FEATURES

The Basic Ship Data module allows the input of general data common for all the transverse sections, bulkhead arrangements and models of torsion. It also performs calculations that may be done from those data. The module is organized around the following application:

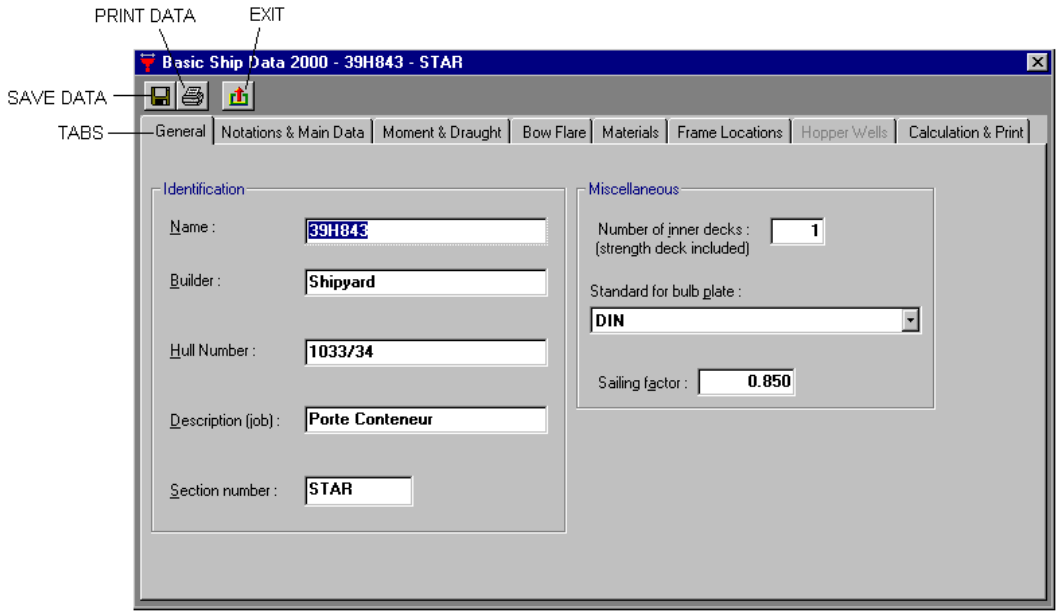


Figure 6 : BASIC SHIP DATA

The BSD module is divided into eight main parts:

- General.
- Notations & Main Data.
- Moment & Draught.
- Bow Flare.
- Materials
- Frame Locations.
- Hopper wells (Hopper dredgers & split hopper dredgers only)
- Calculation & Print.

3.2 GENERAL

The first tab of the basic ship data window allows defining the identifiers of the ship and the general data.

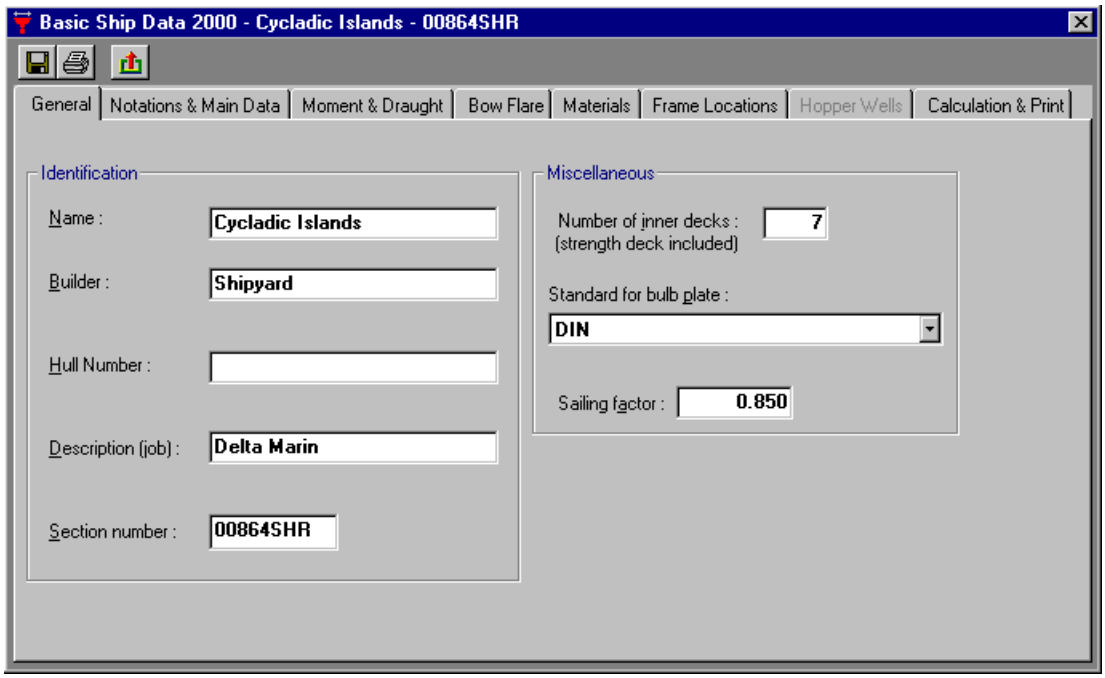


Figure 7 : GENERAL TAB

Miscellaneous

- *Number of inner decks*: the strength deck has to be included in this number. So the minimum value is 1.
- *Standard for bulb plate*: select one of the 5 standards in the list. The geometrical properties of the bulb plates are stored on internal files corresponding to the 5 cases of the list.
- *Sailing factor*: ratio of sailing time of the ship in its life.

3.3 NOTATION AND MAIN DATA

The data defined in this tab are the main ship dimensions, notations, depths and location of hold regions.

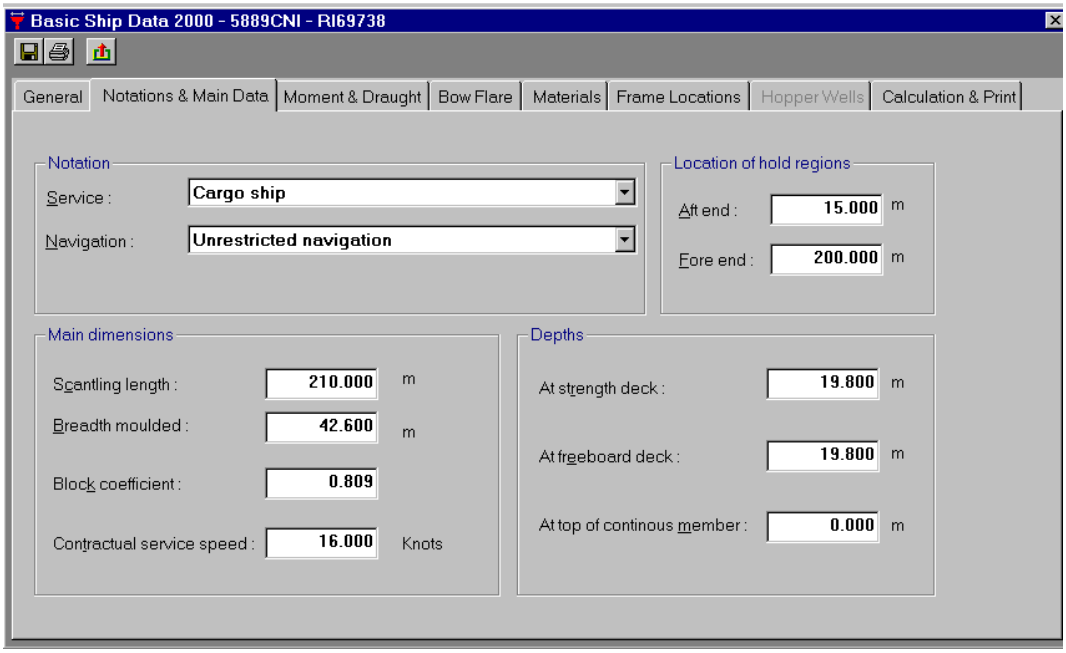


Figure 8 : NOTATION & MAIN DATA TAB

3.3.1 NOTATION

- *Service*: identifies the service notation of the ship. This notation is used in the local rule scantling to determine, for example, the load heights and the minimum rule thickness.

The possible service notations are:

1	Cargo ship	12	Tanker
2	Container ship	13	Dredger
3	Ro-ro cargo ship	14	Hopper dredger/barge
4	Ro-ro passenger ship	15	Split hopper dredger/barge
5	Passenger ship	16	Fishing vessel
6	Oil tanker (and easy chemical)	17	Launch
7	Bulk carrier	18	Sea going launch
8	Ore carrier	19	Yacht
9	Combination carrier	20	Tug
10	Liquefied gas carrier	21	Supply vessel
11	Chemical tanker		

- *Navigation*: identifies the area where the ship may navigate. This notation is used to calculate some rule coefficient.

The possible navigation notations are :

1	Unrestricted navigation
2	Summer zone
3	Tropical zone
4	Coastal area
5	Sheltered waters

If the service notation is equal to Hopper dredger/barge or Split hopper dredger/barge the window become:

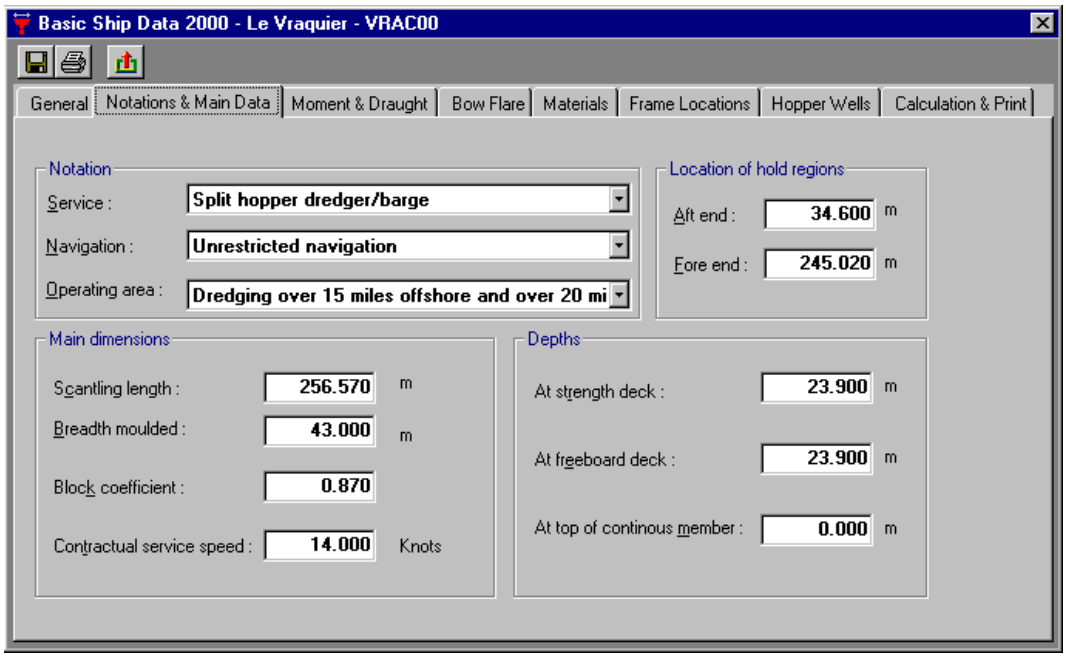


Figure 9 : NOTATION & MAIN DATA TAB –2

- *Operating area*: indicates where the dredger may operate.

The possible operating areas are:

1	Dredging over 15 miles offshore and over 20 miles from port
2	Dredging within 15 miles from shore or within 20 miles from port
3	Dredging within 8 miles from shore

3.3.2 MAIN DIMENSION

- *Scantling length*: length of the ship as defined in Rules for ship scantlings.
- *Breadth moulded*: breadth moulded of the ship as defined in Rules.
- *Block coefficient*: block coefficient as defined in Rules, corresponding to the scantling draught amidship.
- *Contractual service speed*: contractual service speed used to calculate the ship motions.

3.3.3 LOCATION OF HOLD REGIONS

- *Aft end*: It is the longitudinal location of the aft end of the hold region measured from the aft perpendicular in metres
- *Fore end*: It is the longitudinal location of the fore end of the hold region measured from the aft perpendicular in metres

3.3.4 DEPTHS

- *At strength deck*: depth moulded at strength deck in metres. The strength deck is defined in Rules.

- *At freeboard deck*: depth moulded at freeboard deck. When this value is not defined the strength deck is supposed to be freeboard deck also.

- *At top of continuous member*: this depth has to be defined when there are continuous member above the strength deck taking part into the longitudinal strength. This value is useful to determine the scale of plotting. But even when it is no completed the program searches for continuous member above the strength deck to calculate a modulus at top if necessary.

3.4 MOMENT AND DRAUGHT

The third tab of the basic ship data window allows defining moments and draughts for scantling or ballast.

3.4.1 SCANTLING CASE

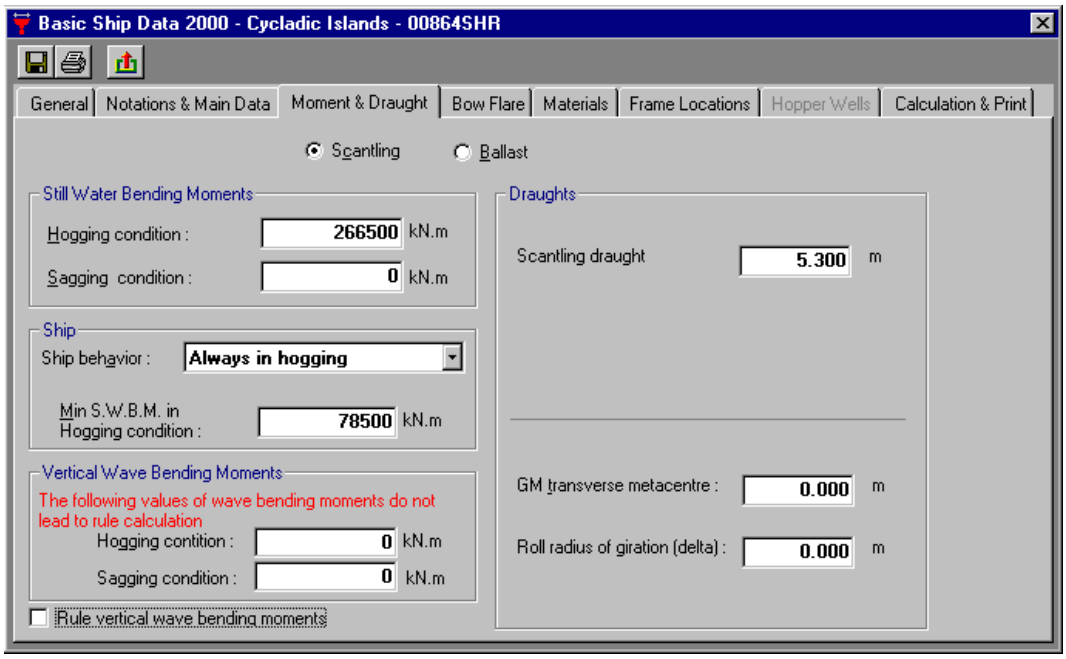


Figure 10 : MOMENT & DRAUGHT TAB – SCANTLING CASE

Still water bending moments
This item allows defining the still water bending moment in midship region, in hogging and sagging conditions, for which the ship will be calculated. If these value are yet unknown, the scantling will be done using permissible rule value.

- *Hogging condition*: still water bending moment in hogging condition and calculated in midship region.

- *Sagging condition*: still water bending moment in sagging condition and calculated in midship region.

Ship

- *Ship behavior*: this data is used in buckling calculation.

The possible ship behaviors are:

1	Both Hogging / Sagging	The different loadings of the ship cause both hogging and sagging condition
2	Always in sagging	The ship is always in sagging condition whatever the loadings
3	Always in hogging	The ship is always in hogging condition whatever the loadings

Vertical wave bending moments

The vertical wave bending moments are rule values that are determined automatically by the program. However the user can define other bending moments clicking on Rule vertical wave bending moments but it is important to know that **WAVE BENDING MOMENTS DEFINED BY THE USER LEAD TO NON RULE SCANTLINGS CALCULATION**

This option may be used in a vue of experimental calculation.

Draughts

This item is used to define the scantling draught.

- *Scantling draught*: at midship perpendicular used for scantlings. This data is compulsory.

- *GM transverse metacentre*: GM corresponding to scantling draught or minimum ballast draught defined hereabove.

3.4.2 BALLAST CASE

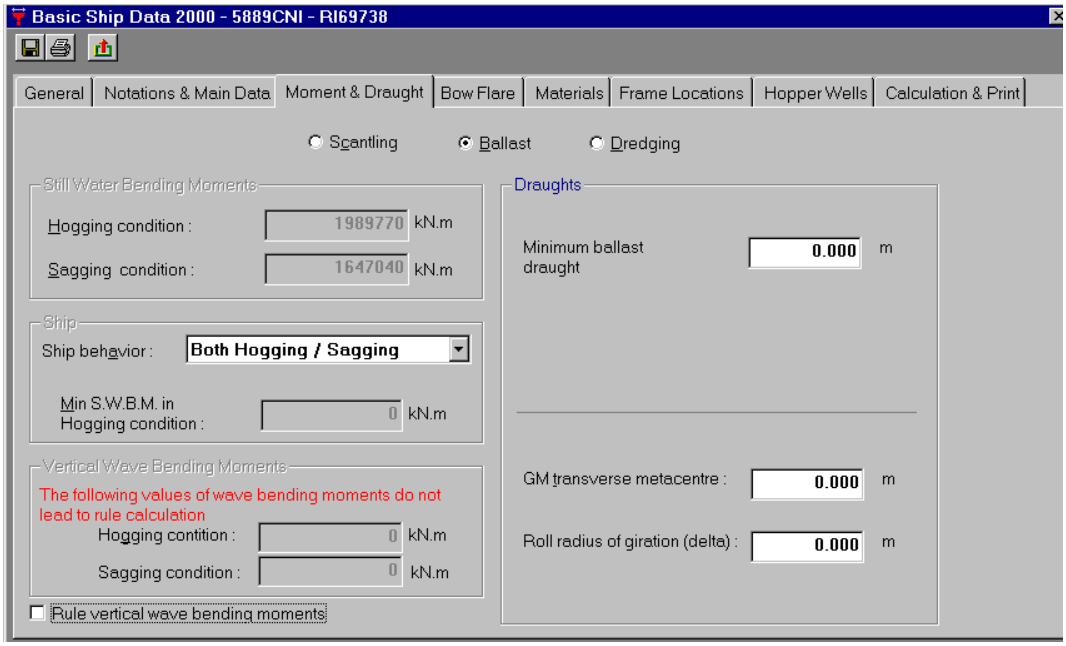


Figure 11 : MOMENT & DRAUGHT TAB – BALLAST CASE

Draughts

This item is used to define the Minimum ballast draught and the Heavy ballast draught.

- *Minimum ballast draught*: if it is not defined the program assumes a rule default value.

3.4.3 DREDGING CASE

If the service notation is equal to Hopper dredger/barge or Split hopper dredger/barge the window become:

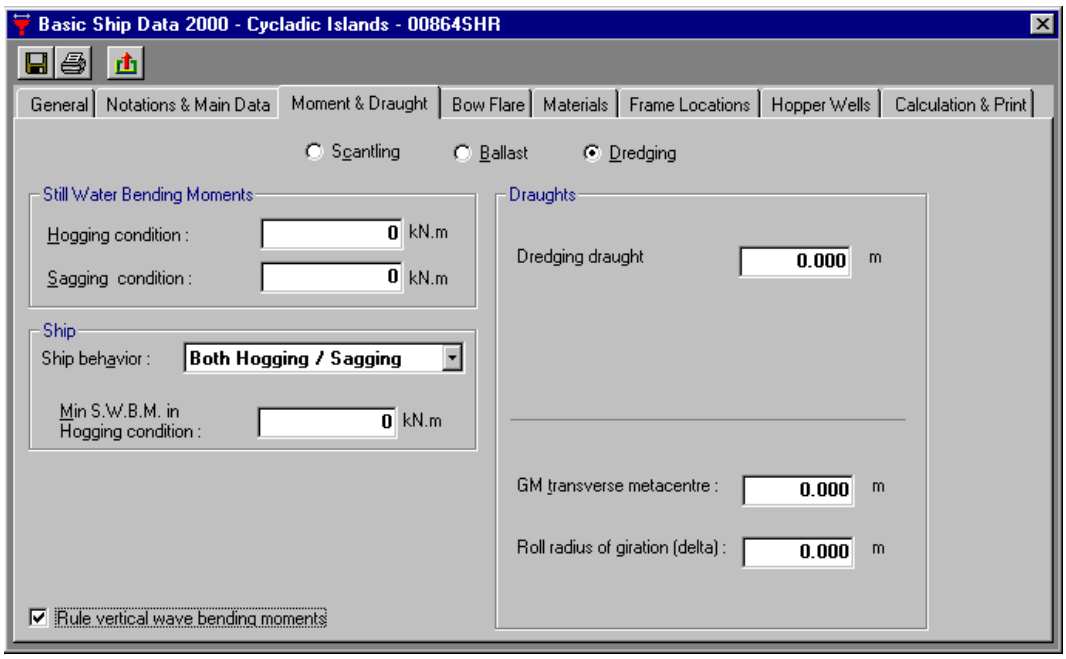


Figure 12 : MOMENT & DRAUGHT TAB – DREDGING CASE

In this case a new field is displayed:

- Dredging draught: draught in dredging condition.

3.5 BOW FLARE

Clicking on the *Bow Flare* tab, you can define the dimension of the bow flare:

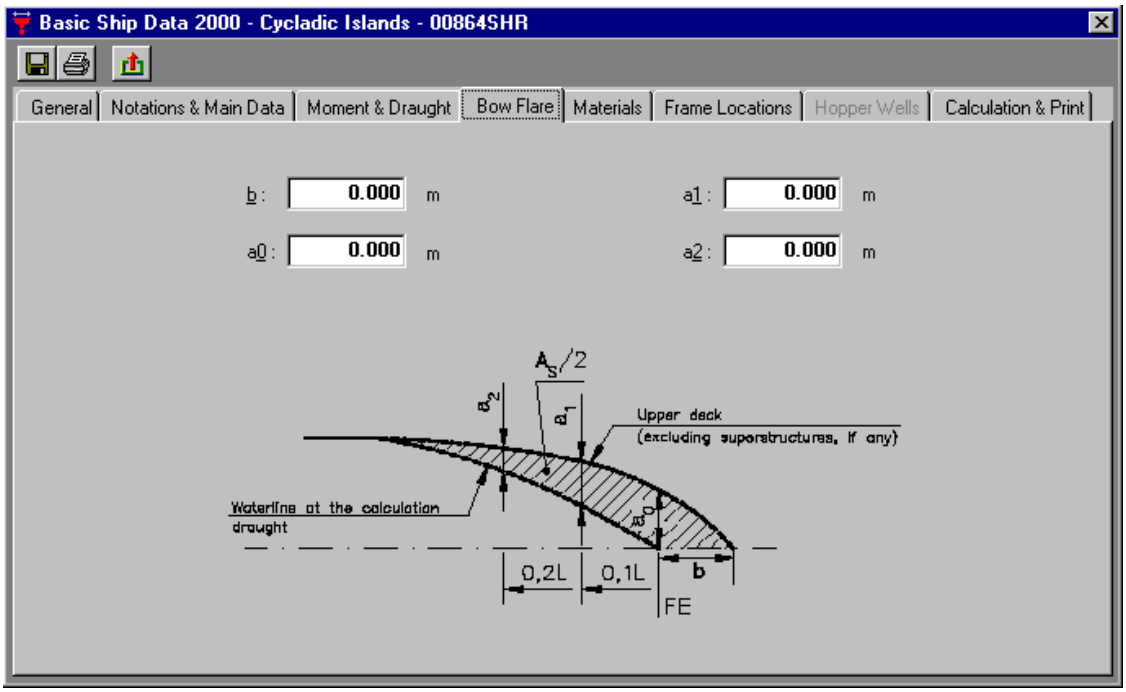


Figure 13 : BOW FLARE TAB

The effects of bow flare impact are to be considered where all the following conditions occur:

- Ship length between 120 m and 180 m.
- Maximum ahead service speed greater than 17.5 knots.
- $\frac{100FAs}{LB} > 1$

,where As is twice the shaded area shown inFigure 13.

3.6 MATERIALS

The item *Materials* allows to define up to 6 different materials used in the ship and their corresponding mechanical properties.

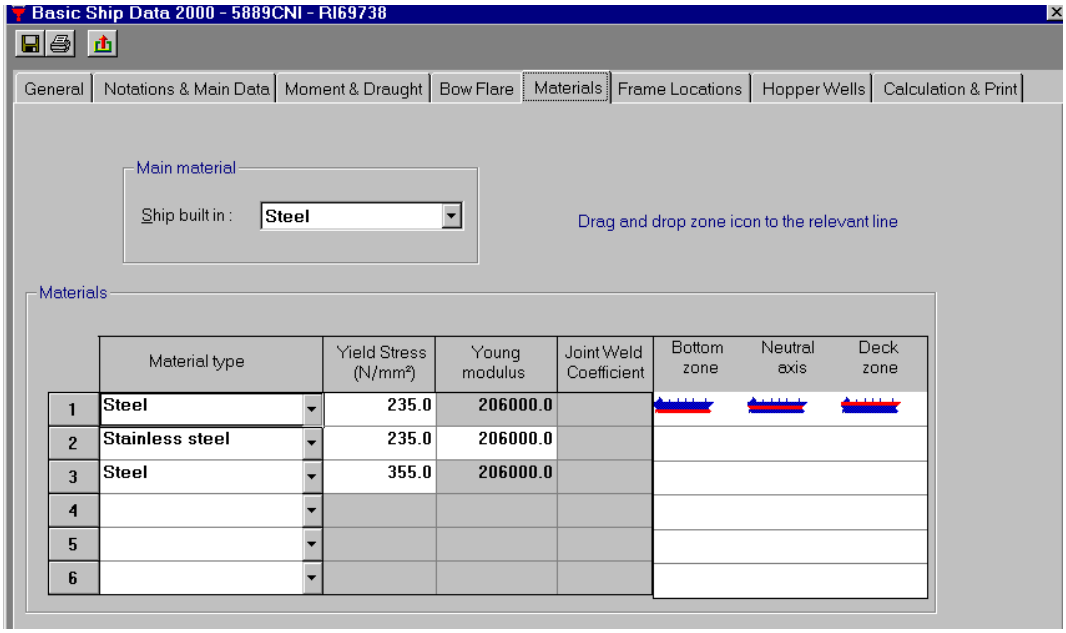


Figure 14 : MATERIALS TAB

- *Ship built in*: indicates if the ship is a steel ship or an aluminium ship.

- *Material type*: type of the material. The possible materials are :

1	Steel
2	Stainless steel
3	Aluminium rolled
4	Aluminium extruded

- *Yield stress*: Yield stress of the material in newtons per square millimetres.

- *Young modulus*: Young modulus in newtons per square millimetres. For steel the program assumes a standard value of 206.000 N/mm².

- *Joint Weld coefficient*: have to be defined only for aluminium

- *Bottom zone, Neutral axis, Deck zone*: this data allows to define the material used in their 3 horizontal zones of the ship. This data is considered, for example, in the evaluation of global strength criteria, to calculate the rule moduli at top, bottom and top of continuous member if any.

3.7 FRAME LOCATIONS

The frame locations need to be defined, only when the user asks for the calculation of longitudinal strength distributions. In this case the calculation points may be defined by the frame number or by distance from aft perpendicular in meters. The frame location definition is done by means of the following window:

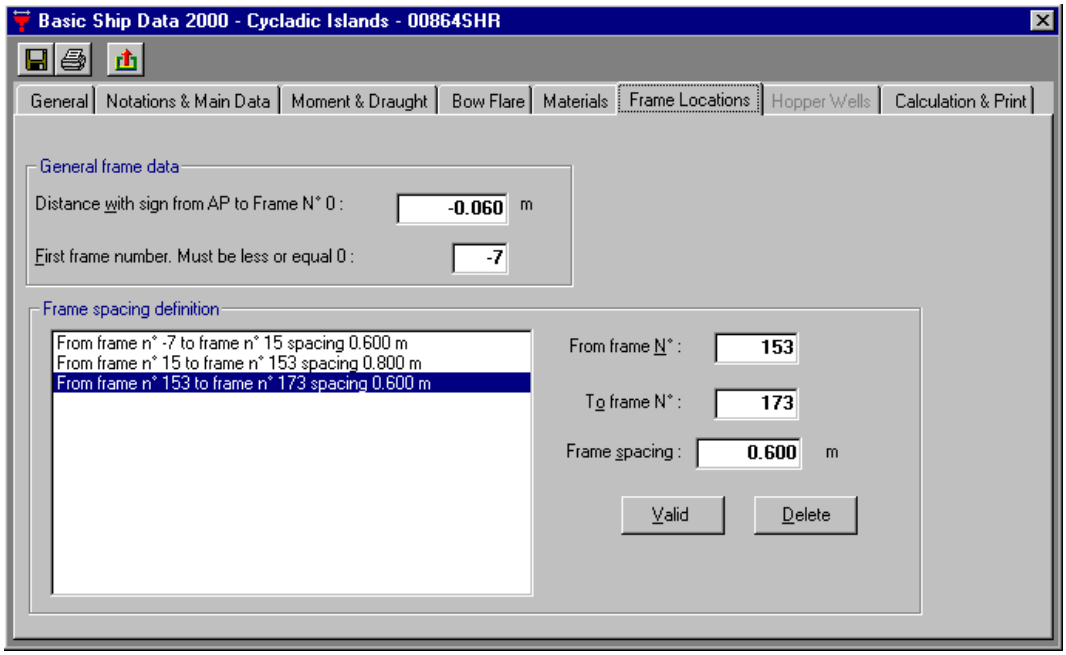


Figure 15 : FRAME LOCATIONS TAB

General frame data

- *Distance with sign from AP to frame N°0*: location of frame number 0 from the aft perpendicular. This value is positive when the frame 0 is forward the aft perpendicular and negative when it is backward.

- *First frame number*: this value may be negative if there are frames located backward the frame 0.

Frame spacing definition

- *From frame N°*: indicates the number of the first frame of the group.

- *To frame N°*: indicates the number of the last frame of the group.

- *Frame spacing*: frame spacing of the group.

- Clicking *Valid* button you may enter a new group of frame with the relevant properties; using *Delete* button you may erase a group of frame previously selected on the frame list.

3.8 HOPPER WELLS

If the service notation is equal to Hopper dredger/barge or Split hopper dredger/barge the Hopper Wells tab become active:

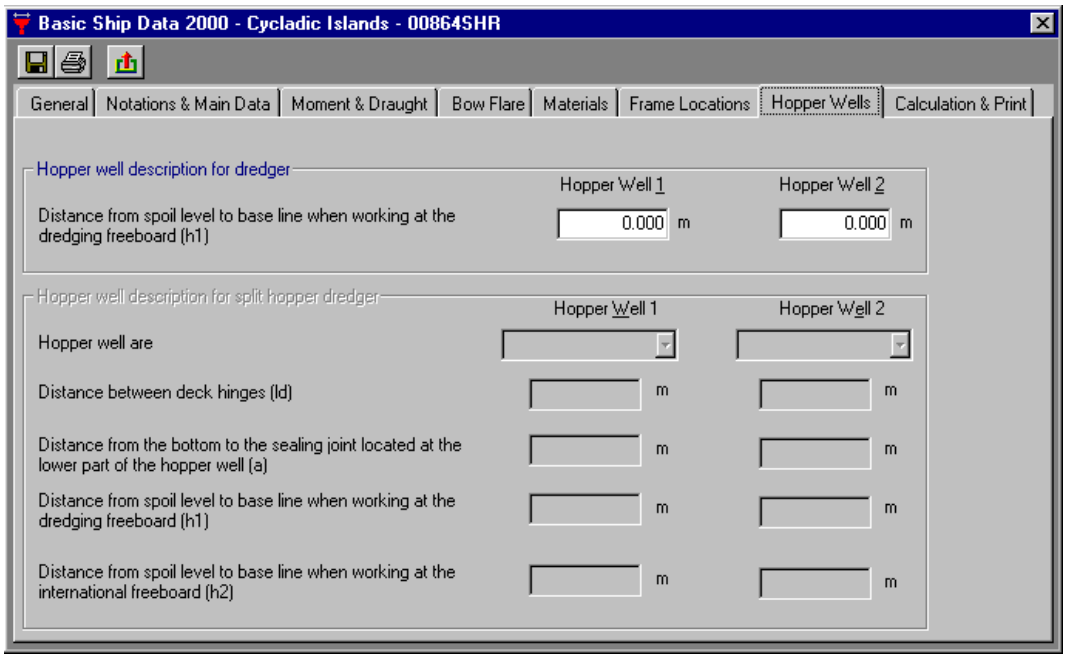


Figure 16 : HOPPER WELLS TAB

Warning: Rule calculations for dredgers have not yet been implemented.

3.8.1 HOPPER DREDGER CASE

Hopper well description for dredger

- h1: Distance from base line to highest weir level, when working at the dredging freeboard.

3.8.2 SPLIT HOPPER DREDGER CASE

Hopper well description for split hopper dredger

- *Hopper well are*: the possible hopper wells are :

1	Partly fixed
2	Simply supported

- h1: Distance from base line to highest weir level, when working at the dredging freeboard.
- h2: Distance from base line to highest weir level, when working at the international freeboard.

3.9 CALCULATION AND PRINT

3.9.1 PRINTING DATA

Clicking on  you enter the Print Data management window:

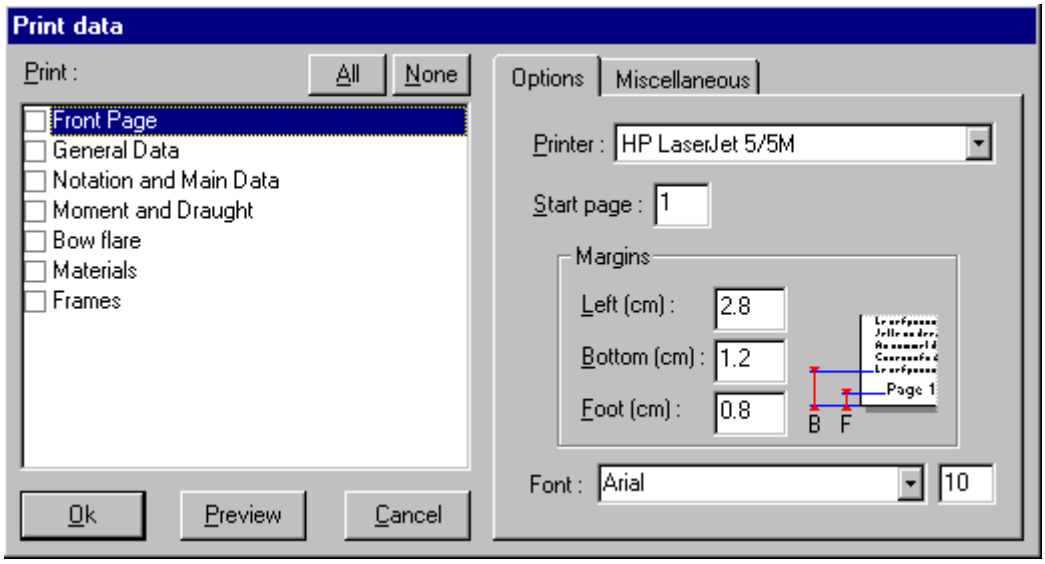


Figure 17 : PRINT DATA MANAGEMENT WINDOW

This window allows you to select what you want to print. The *All* (*None)* button selects (deselects) all the items of the list.

Front page produces the cover page of a report.

3.9.2 CALCULATION

The program is able to perform basic calculations from the basic ship data only. The window with the available possibilities is the following:

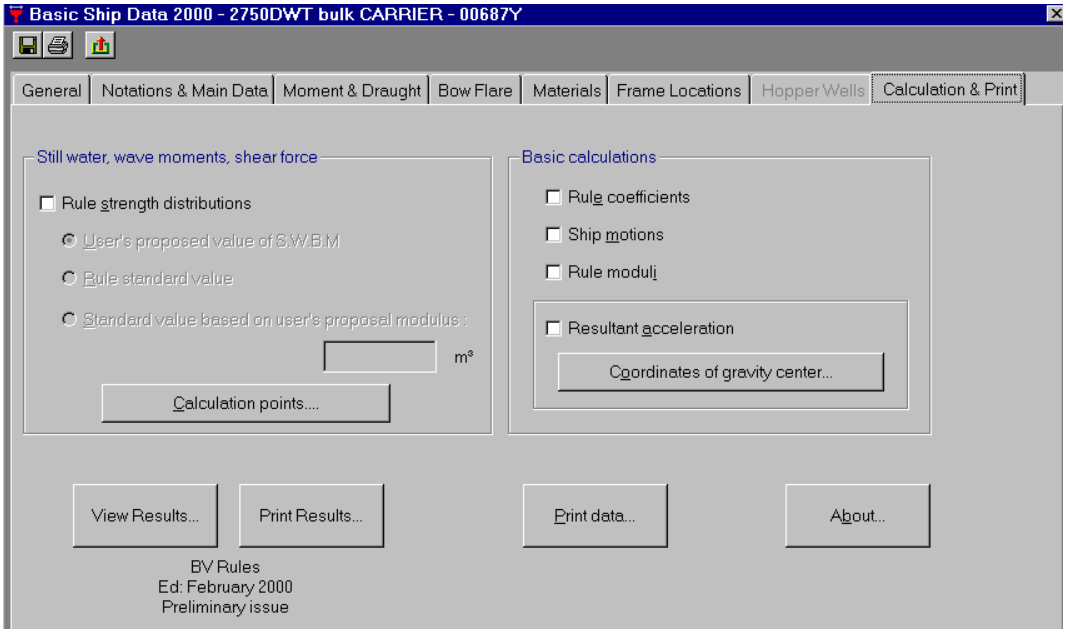


Figure 18 : CALCULATION & PRINT TAB

3.9.2.1 STILL WATER, WAVE MOMENTS, SHEAR FORCE

The longitudinal strength distributions which can be obtained clicking on *Rule strength distributions* are :

- Vertical still water bending moments, in hogging and sagging conditions.
- Vertical wave bending moments, in hogging and sagging conditions.
- The resultant moments, in hogging and sagging conditions corresponding to the combined effect of the still water bending moments and wave bending moments, as defined in Rules.
- The horizontal wave bending moment.
- The horizontal wave shear force.

The distribution of still water bending moments is based on values in hogging and sagging conditions in midship region with the rule longitudinal distribution applied. Those values may have different origine selected from the window:

- *User's proposal value of S.W.B.M.*: distribution based on user's proposal values in midship region in hogging and sagging conditions.
- *Rule standard value*: distribution based on the rule permissible still water bending moments.
- *Standard value based on user's proposal modulus*: the section modulus amidships is a user's proposal value. The still water bending moment amidships are calculated from this user's proposal modulus amidships and from the rule horizontal wave bending moment amidships.

The longitudinal strength distributions are displayed for calculation points selected by the user. There **are 3 ways to define those points:**

- Direct input of their X coordinates.
- Selection of a list of frame numbers.
- Both, X coordinates and list of frames.
- Clicking on *Calculation points* button you enter the following window which allows the definition of the longitudinal calculation points:

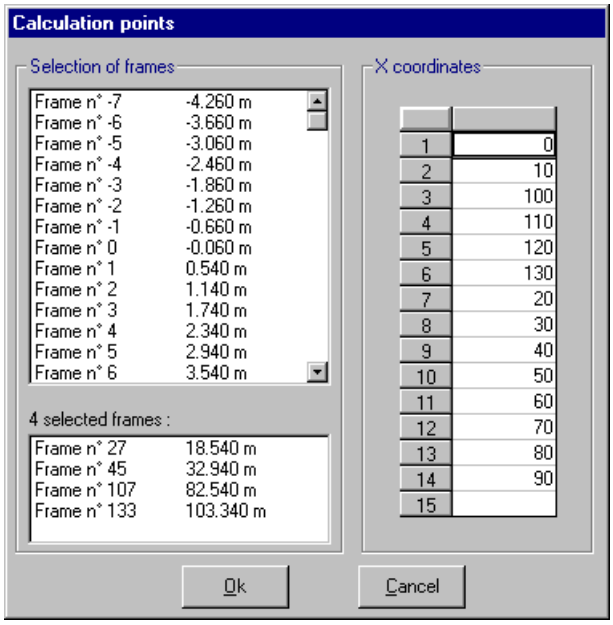


Figure 19 : CALCULATION POINTS WINDOW

3.9.2.2 BASIC CALCULATIONS

- *Rule coefficients*: provides the rule coefficients, parameters and material factors.
- *Ship motions*: provides the ship acceleration and amplitude of motions.
- *Rule moduli*: provides the rule section moduli at midship perpendicular.
- Resultant acceleration: this item allows to calculate the components of the resultant acceleration in upright ship condition and inclined ship condition at scantling and minimum ballast draughts for a list of points the coordinate of which are defined with the following window:

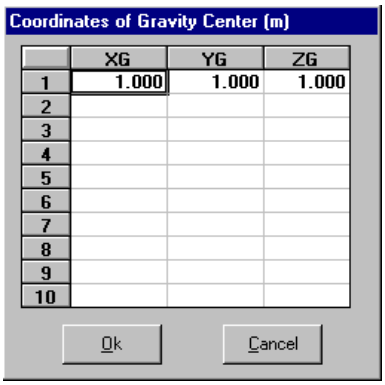


Figure 20 : COORDINATES OF GRAVITY CENTER WINDOW

Up to 10 points may be defined in the same calculation. Each point may be considered as the center of gravity of a compartment.

3.9.3 PRINTING RESULT

Clicking on *Print Result* button you enter the Print Result management window:

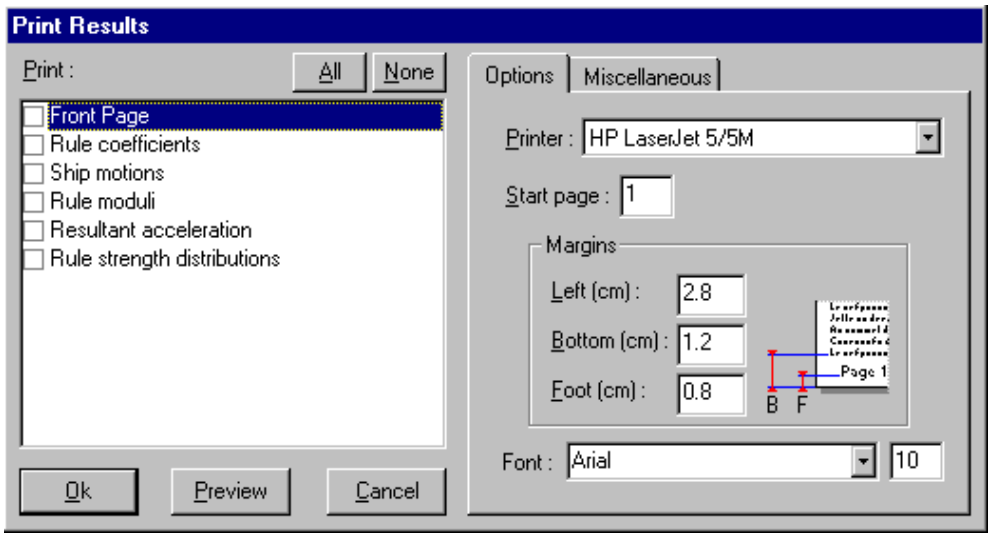


Figure 21 : PRINT RESULT MANAGEMENT WINDOW

This window allows you to select what you want to print. The *All* (*None)* button selects (deselects) all the items of the list.

Front page produces the cover page of a report.

Table of Figure

MARS2000
User's guide

Booklet 2

DEFINITION OF A SECTION

February 2000 /1
Dev/UG/MARS2000/02

MARINE DIVISION
DEVELOPMENT DEPARTMENT
17 bis, place des Reflets – La Défense 2
92077 Paris – La Défense Cedex
Tel.: + 33 (0)1 42 91 52 91
Fax: + 33 (0)1 42 91 52 98

TABLE OF CONTENTS

Chapter 1 : GENERAL COMMENTS

1.1 INTRODUCTION

MarsIn allows the input of any section along the ship length.

The section is described by:

- Longitudinal elements contributing to the hull girder strength.
- Transverse stiffeners.
- Compartments

1.2 MARSIN INTERFACE

The module allowing to input the data of a section is organized around the following application:

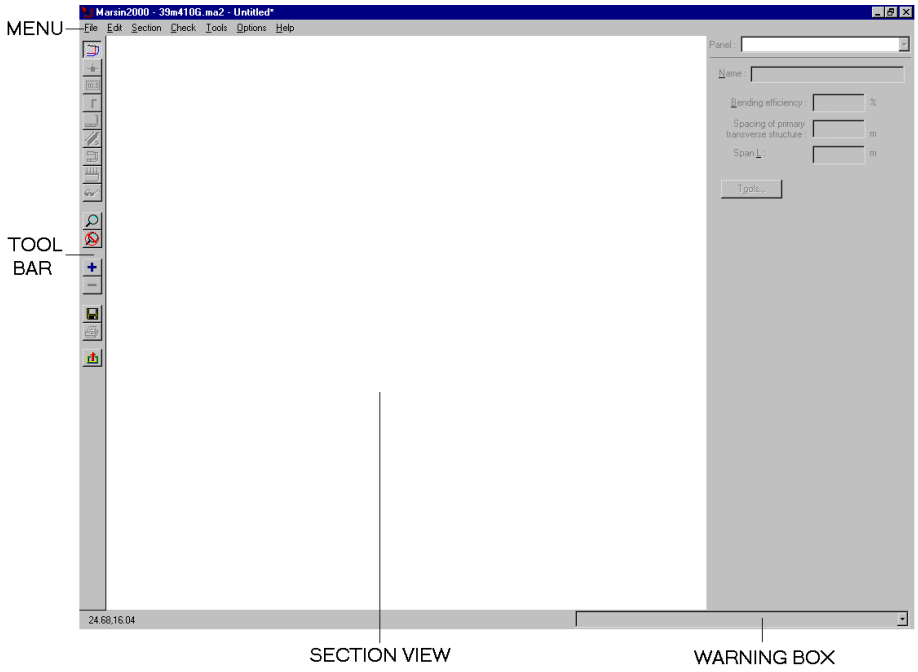


Figure 1: MARSIN

Warning Box: displays warning message when MARSIN detects incoherence in the Section definition.

Section view: displays a view of the section.

1.3 A GOOD WAY TO MODELISE A SECTION

The geometry of a section is panel oriented: the section has to be divided into several panels, each one corresponding to a physical entity such as the outside shell, the strength deck, the inner bottom or a longitudinal bulkhead...

Each panel is made of contiguous segments described by two nodes. Those segments have eventually different geometry (straight, circular or corrugated line).
The way of the description defines a direction on the panel. This direction is commonly used to define flange direction of longitudinals...
Each panel is an independent entity, even for calculation.

Once the panels are fully described, welding joints and longitudinal stiffeners (with their scantling) can be located.

If necessary, it is also possible to define transverse stiffening zones.

1.4 MAIN DIFFERENCES WITH MARSWIN

1.4.1 Compartments data

A new data has been added: main destination of the compartment. This data leads to the evaluation of net scantling of the section.

The top of air pipe is replaced by the deck level to which the air pipe extends.

When openning an old MarsWin database, those data are initialized:

- For the main destination: on the basis of the compartment load type. It should be checked carefully.
- For the air deck: minimum between the top of air pipe and the depth at strength deck.

1.4.2 Span

The SpanS value is recovered through the Spacing of primary transverse structure (see 3.3 and 3.8)."

The SpanL value is recovered through the stiffeners span reduction.

Chapter 2 : MAIN SECTION DATA

On the *Section* menu, click on *Main Data - Moments…* (Figure 46) to display the Main Section Data Window.

2.1 MAIN DATA

The first tab of the main section data window allows defining the X longitudinal location of the section and also to precise main data that might be dependent of this location.
Moreover it allows defining the type of section (e.g. Half section, Full section, Thin walled structure).

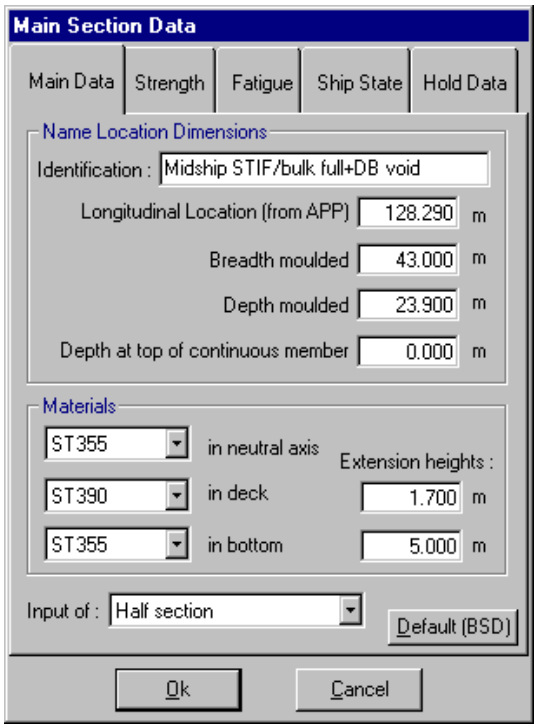


Figure 2: MAIN DATA WINDOW

The *Default (BSD*) button of this window initializes the Main Data with the values already defined in the Basic Ship Data or with default values. It is only an initialization: a change in the Basic Ship Data doesn't modify the Main Section Data

It initializes:

- *Identification* = Midship section
- *Longitudinal Location (from APP)* = Scantling length / 2
- *Extension heights* = 0.000 m
- All the others data are initialized with their BSD equivalent.

Extension heights (in m): used to define the zone in which the materials are defined.

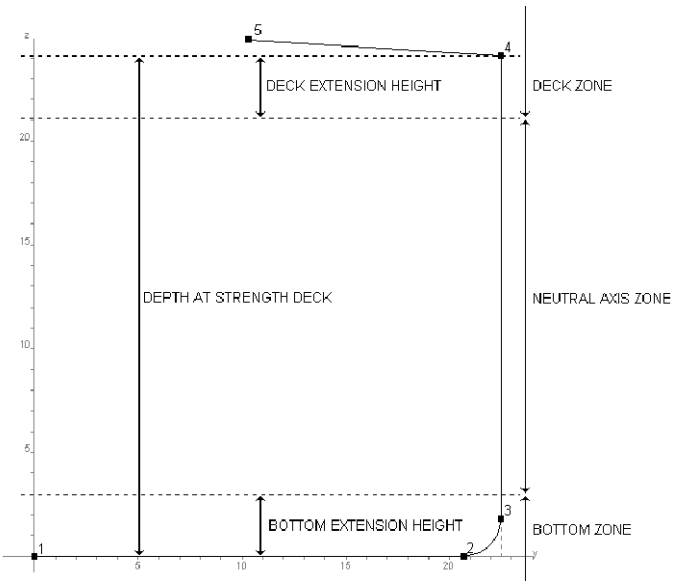


Figure 3: EXTENSION HEIGHTS

2.2 STRENGTH

The second tab of the main section data window allows entering the design values of the still water bending moments at X, still water shear force and vertical wave bending moments.

If you prefer don't enter a SWBM value for a section, the calculations are based on:

- The values defined in the BSD taken as midship section values and modified using the rule distribution law.

- If no value has been defined in the BSD, the rule distribution will be applied to the admissible rule values.

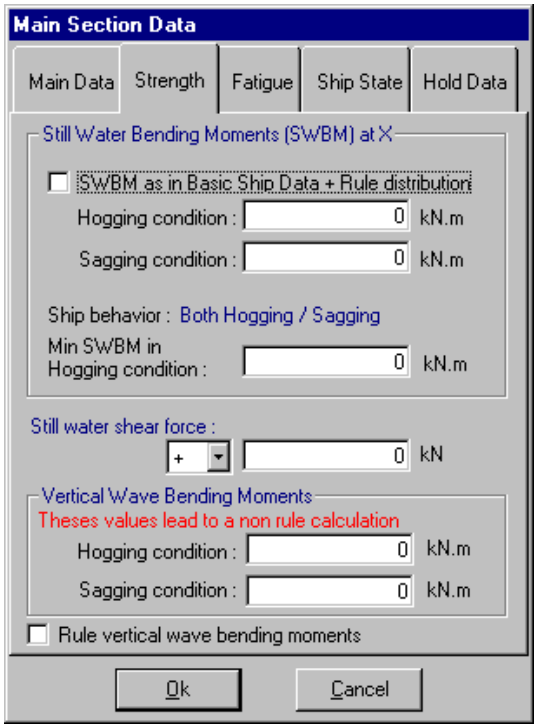


Figure 4: STRENGTH WINDOW

The ship behavior defined in the BSD (both hogging/sagging, hogging only, sagging only) is displayed on screen. Obviously, it can not be corrected at this level.

The rule calculations are based on rule values of vertical wave bending moments automatically calculated by the program.
However it is possible to perform special calculations with vertical wave bending moments defined by the user. If so, it is reminded that the obtained results are not according to the rules.

2.3 FATIGUE

The third tab of the main section data window allows defining the data needed for fatigue calculation.

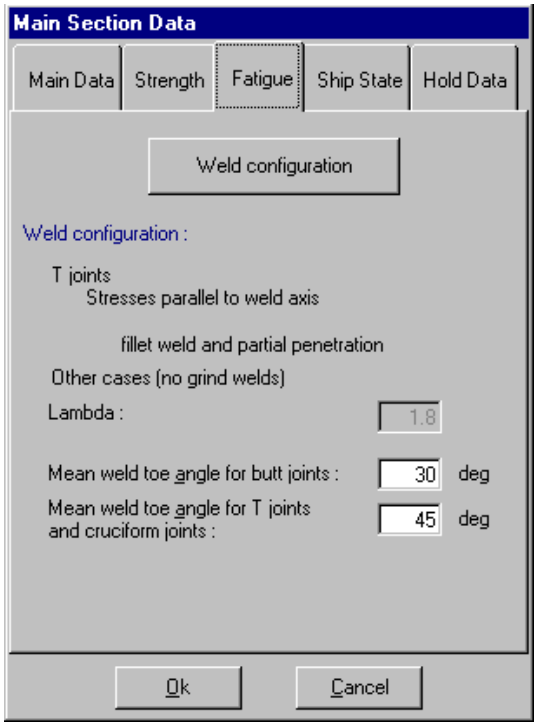


Figure 5: FATIGUE DATA WINDOW

This tab allows precising:

- *Mean weld toe angle for butt joints* (30° default value)

- *Mean weld toe angle for T joints* and cruciform joints (45° default value)

On this tab (Figure 5), the Weld configuration button shows a window which allows to initialize the lambda (λ) value:

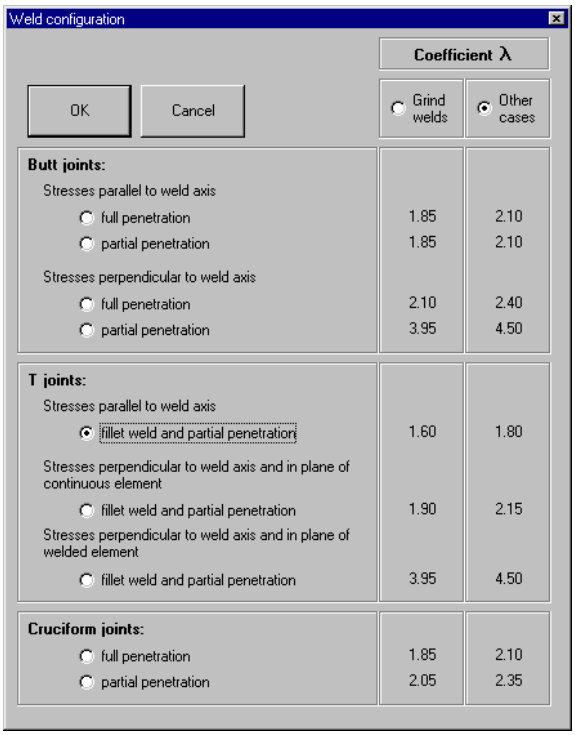


Figure 6: WELD CONFIGURATION WINDOW

2.4 SHIP STATE

The fourth tab of the main section data window allows defining the Ship State.

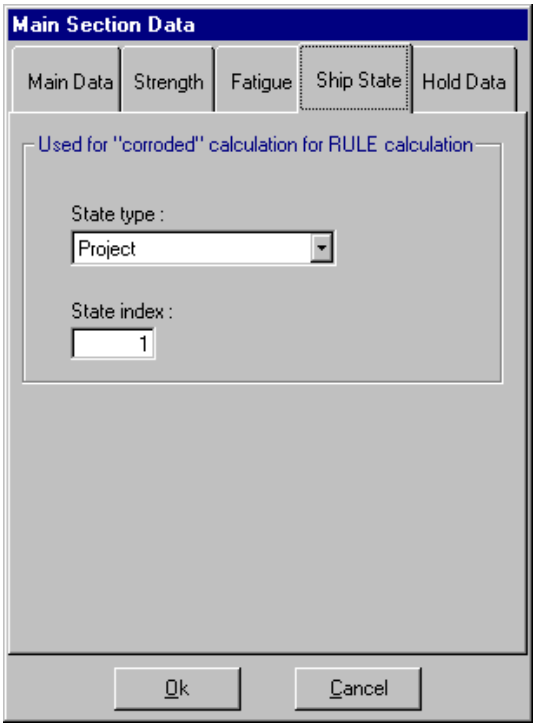


Figure 7: SHIP STATE WINDOW

The State type can be:

- Project
- As Built
- Survey

The *state type* survey allows performing a corroded section calculation.

The state index has no impact on the calculation. It is an help to manage a section for different surveys.

2.5 HOLD DATA

The last tab of the main section data window allows defining the hold data. These values have to be filled to calculate bulk pressure in case of hold having geometry shows in figure:

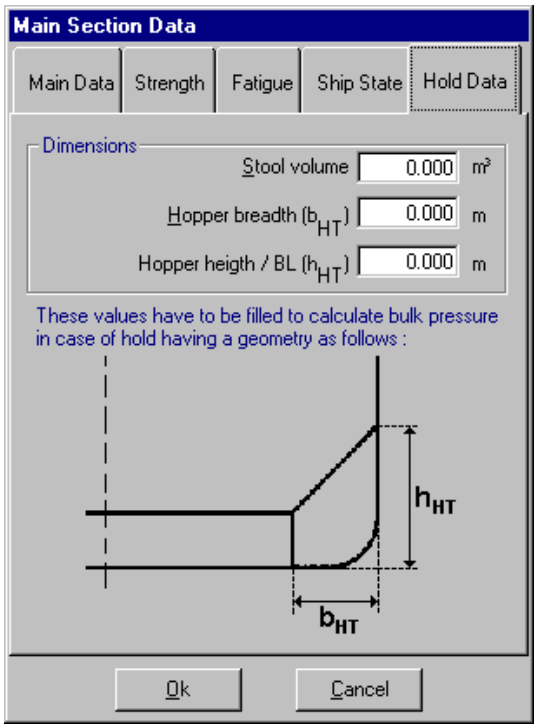


Figure 8: HOLD DATA WINDOW

Chapter 3 : SECTION GEOMETRY

3.1 INPUT SEQUENCE ORGANIZED BY PANEL

There are four main sequences to define the geometry of a section:

- *Panels*
- *Nodes*
- *Strakes*
- *Stiffeners*

These different sequences of the geometry description may be accessed by the following toolbar as follows:

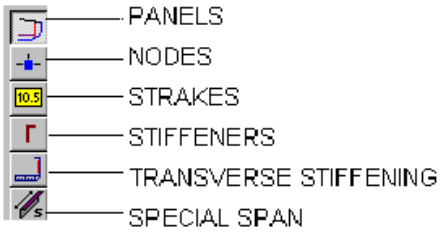


Figure 9: MARSIN TOOLBAR

Panels: allows defining the panel name and characteristics.

Nodes: allows defining the geometry of the panel by a succession of segments.

Strakes: allows to locate welding joints and to enter the actual thickness of every strake on a panel.

Stiffeners: allows to locate all the longitudinals and to define their scantlings.

Transverse stiffening: allows defining the areas transversally stiffened and the secondary transverse stiffeners

Special Span: allows defining panel areas where a special span is to be considered

All these input sequences are panel-oriented. It means that, inside an input sequence, the data are available panel by panel. When an input sequence is selected in the toolbar, the program displays generally the data corresponding to the current panel. The current panel is the last selected panel.

They are five ways to move from one panel to another:

- a direct click on the desired panel in the section view,
- the *Next Panel* and *Previous Panel* items on the *Tools* menu (Figure 51),
- the F5 key to jump to the previous panel,
- the F6 key to jump to the next panel,
- using the panel list placed on the right-hand corner of MARSIN :

Figure 10: PANEL LIST

3.2 CREATION AND DELETION OF DATA

In each input sequence, you can create or delete data:

- Nodes : creation or deletion of a segment
- Strakes : creation or deletion of a welding joint
- Stiffeners : creation or deletion of a group of stiffeners regularly spaced

Each object has to be created or deleted using the following toolbar:

Figure 11 : CREATION-DELETION TOOLBAR

For example, if you want to create a new panel

- click on the *panel* button (Figure 9),
  the *Panel management window* (Figure 12) is displayed,
- click on the *creation* button (Figure 11).

Inversely, if you want to delete an existing panel,

- click on the *panel* button (Figure 9),
- select the panel you want to delete,
- click on the *deletion* button (Figure 11).

To create data, it is also possible to hit the F12 key instead of clicking the *creation* button.
Another way to create or delete objects is to use *Create object* or *Delete object* on the *Tools* menu (Figure 51)

3.3 PANEL DATA

Clicking on the panel button  or on Panel on the Section menu (Figure 46), you enter the Panel management window:

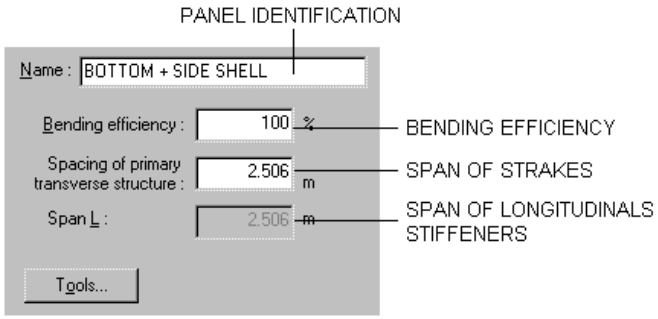


Figure 12: PANEL MANAGEMENT WINDOW

In this window, you can create or delete panels using the *creation* or the *deletion* buttons (Figure 11).
Each panel is defined by identification, a percentage of bending efficiency and spans. These values are assigned to every strake and every longitudinal stiffener of the panel.

Panel Identification: up to 40 characters.
The panel has to be clearly identified. This identification will appear in input sequences and in output of data and results.

Bending Efficiency (in %): percentage of contribution to the overall longitudinal strength.
DO NOT ENTER 50% FOR PLATINGS ON THE CENTER LINE

Spacing of primary transverse structure (in m): span of the strakes, in the ship longitudinal direction, between 2 consecutive transverse stiffeners.
This value is used to calculate the rule thickness of the strakes longitudinally stiffened.

Span L (in m): effective span of the longitudinal stiffeners on the panel.
This value is only a reminder in case of old MarsWin database (see 1.4.2).

Spacing and Span L can be locally changed (see 3.8.).

It is also possible to assign particular values of bending efficiency to groups of stiffeners (see 3.6.2).

Clicking on Tools button you can:

- Duplicate a panel.
- Transform a panel.
- Duplicate and mirror section.

3.4 NODE BUTTON : GEOMETRY OF THE PANEL

3.4.1 Nodes and segments

A panel is made of contiguous segments of different geometry (straight, circular or corrugated line). Each segment is described by:

- Its ending node
- Its type of curve.
- Its position code.

First node of a panel
When you define a panel, a first node (segment 0) is automatically created with coordinates by default: you have to correct these Y and Z coordinates.

Clicking on the node button  or on Nodes - Plates on the Section menu (Figure 46), you enter the Node management window where these inputs are performed:

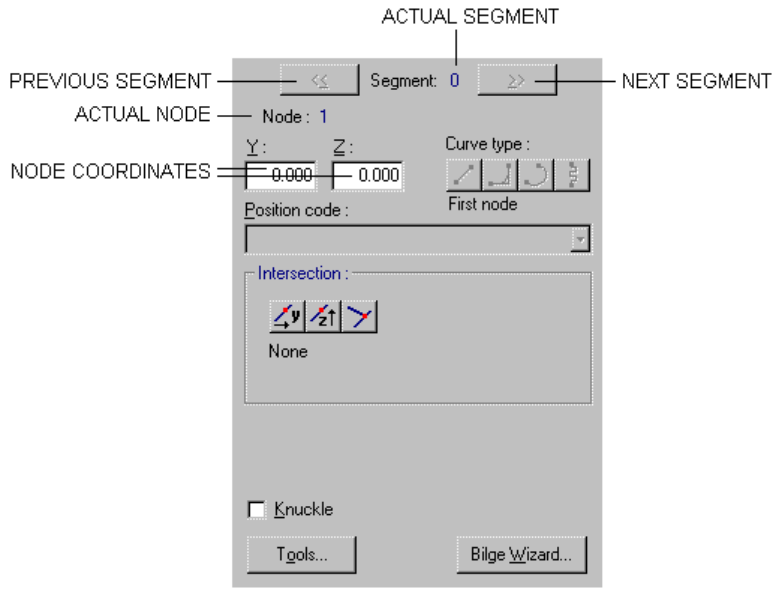


Figure 13: NODE MANAGEMENT WINDOW

Node creation

The *creation* button  allows you to create a segment extending the current panel.

The *deletion* button  is used to delete nodes.

It is also possible to insert a node between two existing nodes by clicking on the *Tools…* button (Figure 13) and selecting *Insert node* (Figure 21). See 3.4.5 for more information.

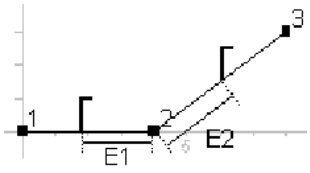
Next segment - Previous segment: Those two buttons allow navigating segment by segment within a panel. A direct click on the desired segment in the section view is also possible.

Node characteristics

Coordinates of the node (in m): The Y and Z coordinates of the current segment ending node. Sometimes, it is not necessary (see 3.4.2.1, 3.4.4 and 3.4.5) to define them.

Knuckle Code: It is used in the rule scantling calculation of strakes and stiffeners. The corresponding node is considered as a limit between two segments with a significant variation of curvature. A knuckle node affects the calculation of the spacing (E) between longitudinals or of the span of transverse plating.

For instance, with this following detail:



If the node 2 has the knuckle code, the spacing will be calculated with E1 or E2
If the node 2 hasn't the knuckle code, the spacing will be calculated with E1+E2

Some facilities are provided to make easier the input of the coordinates:

- The bilge wizard (see 3.4.2.1).

- The tools button (see 3.4.5) to align or to insert nodes or to set the node position with the mouse.

Segment characteristics
The characteristics of the segment between the current node and the previous one consist in :
Curve Type: it specifies the type of the segment.

 : Straight line

 : Tangent arc (see 3.4.2.2)

 : Arc (see 3.4.2.3)

 : Corrugated segment (see 3.4.3)

Position Code: definition of the allocation of the segment inside the section. A position code in the list is to be selected.
The possible positions are:

1	Undefined	13	Inner hull
2	Keel plate	14	Double hull girder
3	Bottom	15	Keelson or other girder
4	Bilge	16	Tank and watertight bulkhead
5	Side shell	17	Vertical corrugation
6	Sheer strake	18	Wash bulkhead
7	Upper strength deck (weather)	19	Cellular keel
8	Upper strength deck (no weather)	20	Hopper well bulkhead
9	Trunk deck	21	Solid bar keel
10	Lower deck	22	Garboard plate
11	Inner bottom	23	Margin plate
12	Double bottom girder	24	Miscellaneous

The more precise position code has to be selected.
For the bottom plate at the centerline side, the position code Keel plate or Bottom can be equally chosen.

3.4.2 Circles

There is two ways to define circle:

- the Bilge Wizard

- the Curve Type buttons

3.4.2.1 Bilge wizard

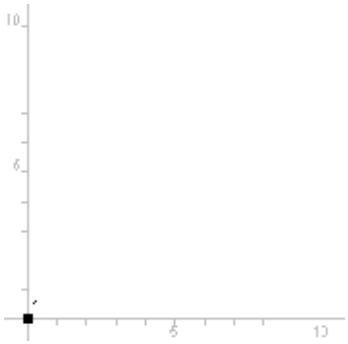
The bilge wizard is the best way to easily define circular plate tangent to the adjacent segments. Using this tool will create a straight segment, a circular segment and another straight segment.

The bilge wizard has to be used very early into the process: click the bilge wizard immediately after the coordinate definition of the first tangent line-starting node.

Example of input of bilge with a flat bottom
For a flat bottom, you enter:

Node	Y Coordinate	Z Coordinate	Click On
1	0.	0.	Bilge Wizard...

You should obtained this kind of section:



and see the Round Shape Wizard displayed :

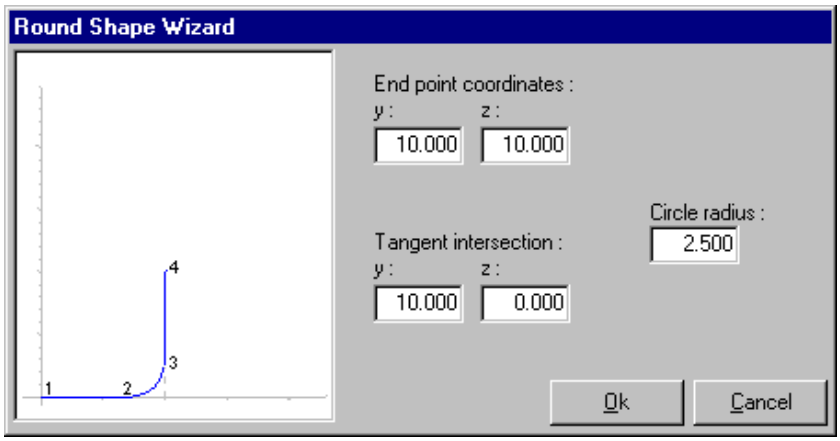


Figure 14: ROUND SHAPE WIZARD

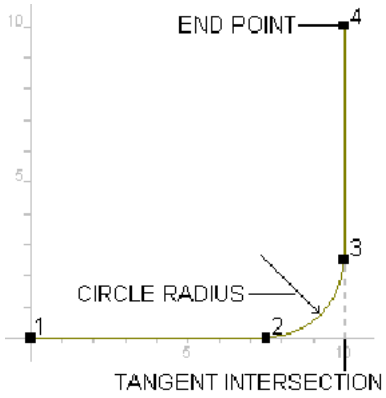
End point coordinates (in m): coordinates of ending node of the second tangent line (node 4 on Figure 14).

Tangent intersection (in m): coordinates of the intersection between the tangent lines.

Circle radius (in m): Radius of the bilge

Preview: Refresh the section view of the Round Shape Wizard without any effect on your section.

Clicking on *Ok* will then create the bilge:

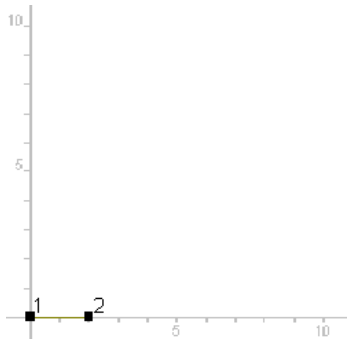


Example of input of bilge with rise of keel

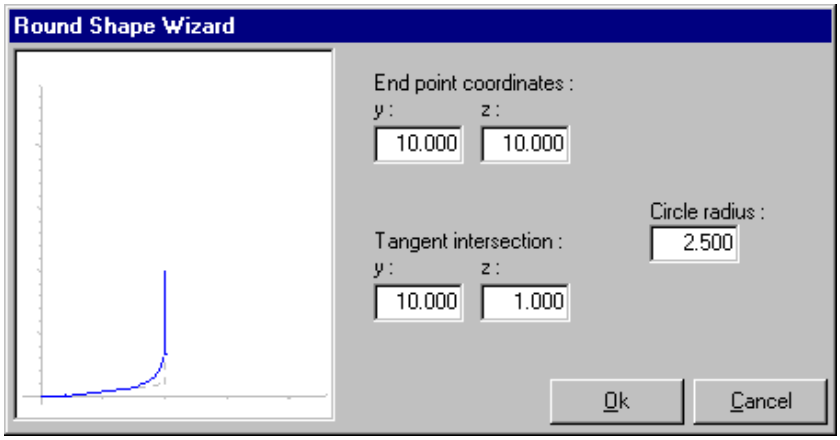
For a rise of keel, you enter:

Node	Y Coordinate	Z Coordinate	Click On
1	0.	0.	+ to create a node.
2	2.	0.	Bilge Wizard...

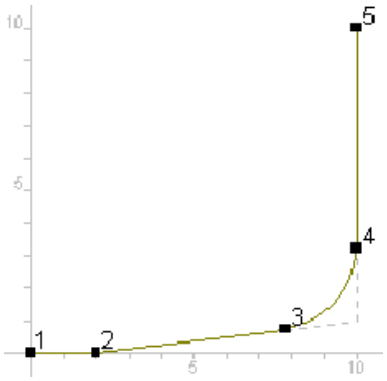
You should obtained this kind of section:



and see the Round Shape Wizard displayed :



Clicking on *Ok* will then create the bilge:



3.4.2.2 Tangent arc

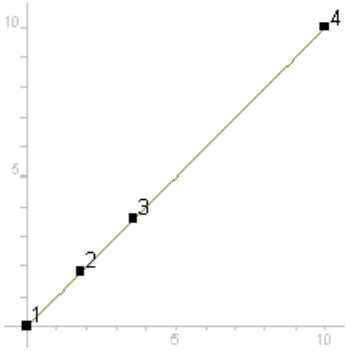
The *Tangent Arc* button is another way of defining circular segment tangent to both segments enclosing it. Therefore you have to create the nodes for the three segments (the circular one end its adjacent).

Example of input of bilge with a flat bottom

For a rise of keel, you enter:

Node	Y Coordinate	Z Coordinate	Click On
1	0.	0.	+ to create a node.
2	-	-	+ to create a node.
3	-	-	+ to create a node.
4	10.	10.	

You should obtained this kind of section:



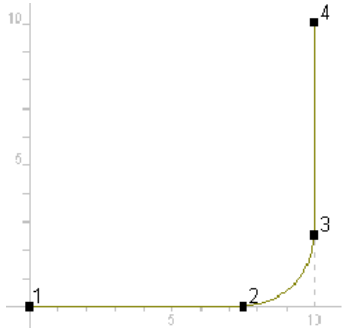
Click on segment 2 (node 3) and hit on the *Arc Tangent* button (Figure 13). The window here after is displayed on screen:

Figure 15: TANGENT ARC DATA

R (in m): Radius of the circle

YTan and ZTan (in m): Y and Z coordinates of the tangent intersection of the enclosing segments.

The coordinates of the first and the last nodes on this segment are automatically calculated and cannot be changed. The result is:

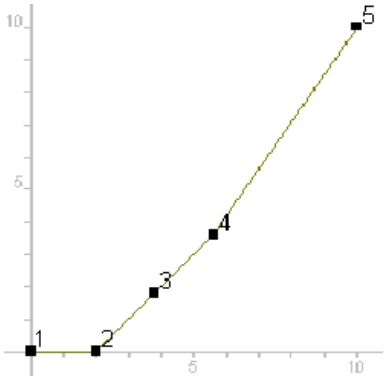


Example of input of bilge with rise of keel

For example, for a rise of keel, you enter:

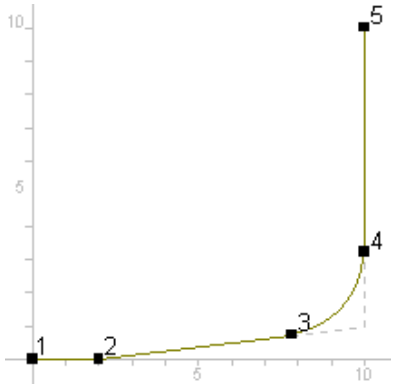
Node	Y Coordinate	Z Coordinate	Click On
1	0.	0.	+ to create a node.
2	2.	0.	+ to create a node.
3	-	-	+ to create a node.
4	-	-	+ to create a node.
5	10.	10.	

You should obtained this kind of section:



Click on segment 3 (node 4) and hit on the *Arc Tangent* button (Figure 13). The window here after is displayed on screen:

The result is:



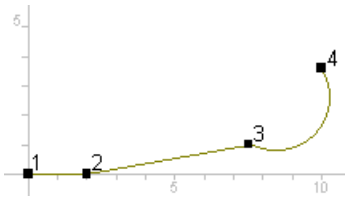
3.4.2.3 Arc

The *Arc* button can be used to define any circular segment.
The coordinates of the first and the last nodes on this segment have to be input.
The definition of an arc is completed by means of the window here after. It is displayed on screen, hitting arc button when the current node is the last node of the circle:

Figure 16: ARC DATA

R (in m) : Radius of the circle.

With this method, you can define any kind of circular segment. For example:



3.4.3 Corrugated segment

Choosing a corrugated segment makes this button appear: 

The definition of a corrugated segment is completed by means of the window hereafter obtained by clicking on the last button:

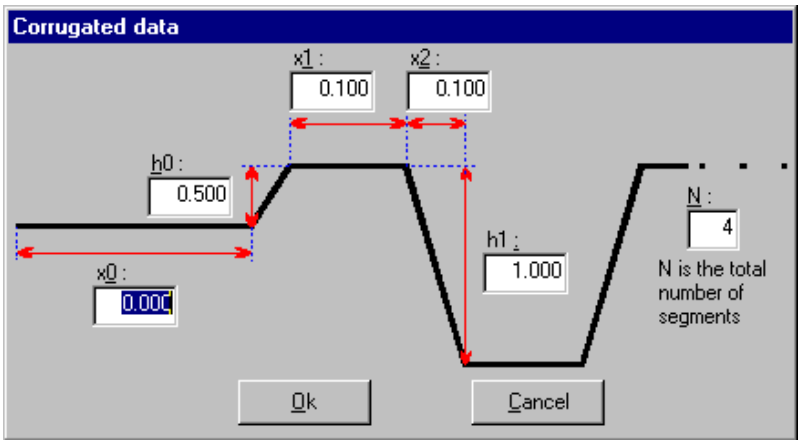


Figure 17: CORRUGATED DATA

h0 (in m): transverse width of the first corrugation.

h1 (in m): height of corrugations.

x0 (in m): distance from the first node of the corrugated segment (see here-above) to the first corrugation.

x1 (in m): width of the flange.

x2 (in m): projection of the inclined part on the support line (see here-above).

N: number of straight parts of the corrugated segment: from the first node to the last node (including the first and the last straight part on the support line).

A corrugated bulkhead is defined as a segment (with corrugations) located between two nodes.
Those two nodes that start and end the "corrugated segment" define the support line of the corrugated bulkhead.

This support straight line is the reference for all locations of welding joints, stiffeners, … This means that the curvilinear abscises must be projected on the support line.

3.4.4 Intersection

During the definition of a panel, you have to create nodes that are intersection nodes with a panel previously defined. To do that:

- Click on  as usual,
- select the type of intersection :

   : Intersection with a panel defined by the Y coordinate

   : Intersection with a panel defined by the Z coordinate

   : Intersection with a node
- click on the relevant panel,

- enter the relevant coordinates.

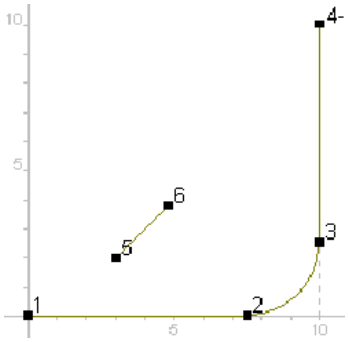
What is an intersection node?
It is a node defined on a panel 2 and being located as intersecting with a panel 1.
This node is not seeable in the list of nodes that define panel 1, but it may be used on panel 1 as an entry to locate stiffeners for example.

3.4.4.1 Intersection with Y entered

This method is to be used when the intersection node may be defined by:

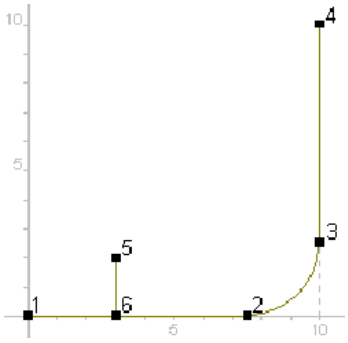
- the selection of the intersection panel
- a Y coordinate.

For example, this is the case of a horizontal segment:



To define the node 6 as an intersection with the segment 1-2:

- click on the *intersection with Y entered* button (Figure 13),
- the cursor change in : ⟷ and a message "Click on panel…" appears,
- click on the segment 1-2 to attach the node 6 with it.



The following boxes are displayed:

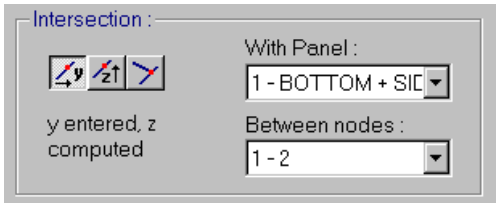


Figure 18: INTERSECTION Y ENTERED

With Panel: panel on which the intersection is defined.

Between nodes: nodes between which the intersection is defined.

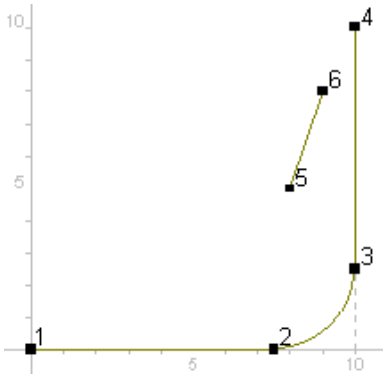
The Y coordinate of the intersection is available, but its Z coordinate is automatically calculated.

3.4.4.2 Intersection with Z entered

This method is to be used when the intersection node may be defined by:

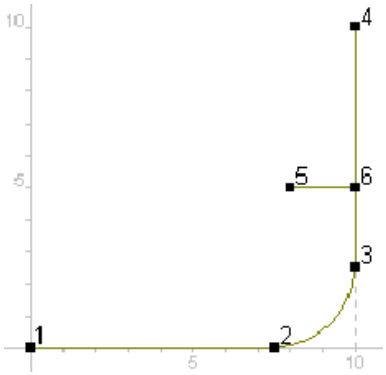
- the selection of the intersection panel
- a Z coordinate.

For example, this is the case of a vertical segment:



To define the node 6 as an intersection with the segment 3-4:

- click on the *intersection with Z entered* button (Figure 13),
- the cursor change in : ↕ and a message "Click on panel…" appears,
- Click on the segment 3-4 to attach the node 6 with it.



The followings boxes are displayed:

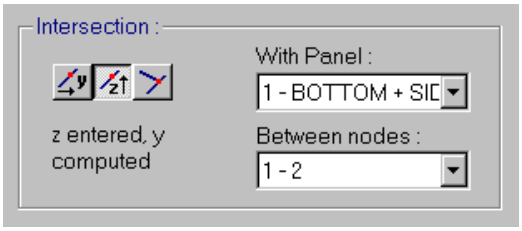


Figure 19: INTERSECTION Z ENTERED

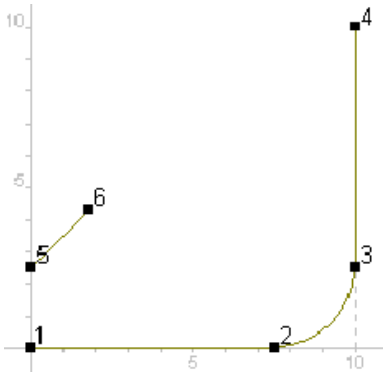
With Panel: panel on which the intersection is defined.

Between nodes: nodes between which the intersection is defined.

The Z coordinate of the intersection is available, but its Y coordinate is automatically calculated.

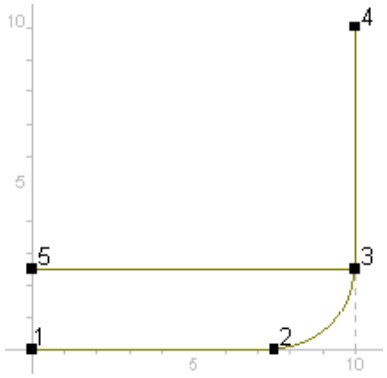
3.4.4.3 Intersection with a node

This method is to be used when the new node to define is an exiting node (generally on another panel).



To put the node 6 of panel 2 in coincidence with the node 3 of panel 1:

- click on the *intersection with node* button (Figure 13),
- the cursor change in : ↑ and a message "Click on node…" appears,
- click on the Node 3 to attach the node 6 with it.



The following box is displayed:

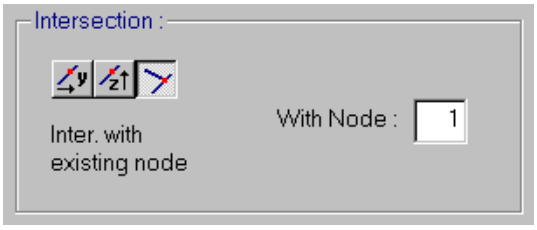


Figure 20: INTERSECTION WITH NODE

With Node: Node on which the intersection is defined.

The intersection node coordinates are not available when the current segment is the one used to defined the intersection (segment 5-3 in the previous example).

3.4.5 Insert node

The *creation* button (Figure 11) permits to create nodes, but only at a panel end. With the *Insert node* tool, it is possible to create a new node on the segment you want. Using this tool changes the cursor in I. A simple click on the segment on which you want to insert the node creates it.

This tool is also reachable on the menu *Tool* (Figure 51).

3.4.6 Tools

There are some tools that make easier the input of nodes. They can be reached via the *Other Tools* button (Figure 13). This button display the following menu:

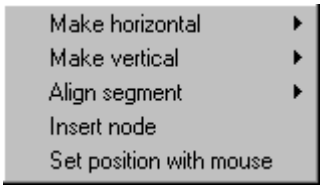
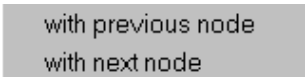


Figure 21: OTHER TOOL MENU

Those tools are just an help to define the node position: they don't define properties for the nodes. For example, making a segment horizontal just set the position of the nodes, but the horizontality of the segment is not a recorded data.

3.4.6.1 Make horizontal

The choice of the tool *Make horizontal* (Figure 21) will display the following sub-menu :

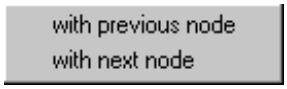


- *with previous node* make the Z coordinates of the current node equal to the previous node. For example, with the node 2 as current node :

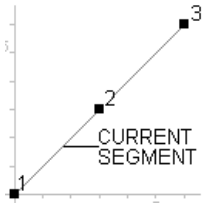
- *with next node* make the Z coordinates of the current node equal to the next node. For example, with the node 2 as current node :

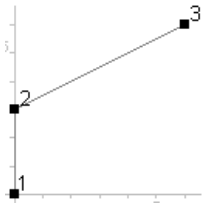
3.4.6.2 Make vertical

The choice of the tool *Make vertical* (Figure 21) will display the following sub-menu:

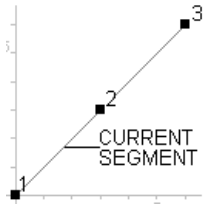


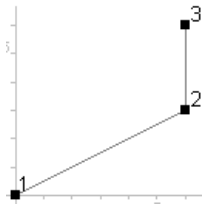
- *with previous node* make the Y coordinates of the current node equal to the previous node. For example, with the node 2 as current node :





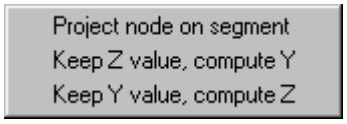
- *with next node* make the Z coordinates of the current node equal to the next node. For example, with the node 2 as current node :



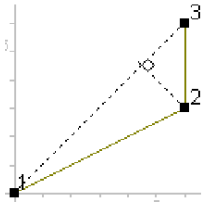


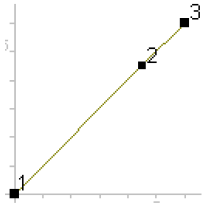
3.4.6.3 Align segment

The choice of the tool *Align segment* (Figure 21) will display the following sub-menu:

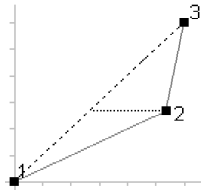


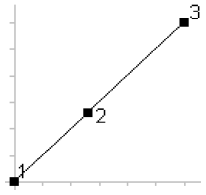
- *Project node on segment* places the current node at the nearest position on the line passing by the previous and the next nodes. For example, if the current node is the node 2 :





- *Keep Z value, compute Y* places the current node on the line passing by the previous and the next nodes keeping the old Z value. For example, if the current node is the node 2 :





- *Keep Y value, compute Z* places the current node on the line passing by the previous and the next nodes keeping the old Y value. For example, if the current node is the node 2 :

3.4.6.4 Set position with mouse

The *Set position with mouse* tool (Figure 21) is another way to define the coordinates of a node. Choosing it change the cursor in ┼. A simple click on the section view where you want to place the node position set it at this exact location.

3.5 STRAKE

All the strakes constituting the panel have to be defined by their width and their thickness.
They are automatically numbered, starting at number 1 on the first panel, and with an increasing numbering along the successive panels.

Clicking on the *Strake* button (Figure 9) or on *Strakes - Welding joints* on the *Section* menu (Figure 46), you enter the *Strake management window* where these inputs are performed:

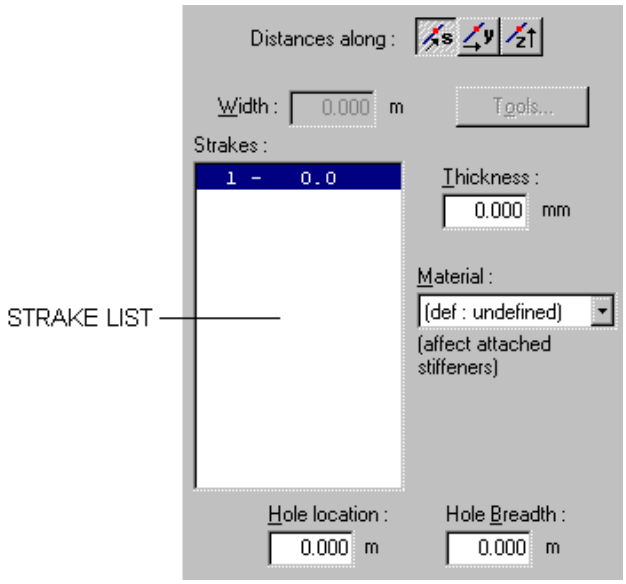


Figure 22: STRAKE MANAGEMENT WINDOW

During a first input, there is a strake automatically created on each panel with width equal to the width of the panel.

The *creation* and the *deletion* buttons (Figure 11) allow you to create or delete strakes.
Creating a strake means to divide the strake selected in the *strake list* (Figure 22) into two strakes of same characteristic, thickness and material, and with width equal to the half width.

Strake List: display of the number and thickness of the strakes defined on the current panel. The strake selected is the current strake.
To select a strake, click on a strake in the list or in the section view.

Width (in m): width of the current strake. As general, this value is also the distance between 2 consecutive welding joints. Nevertheless the first strake is the distance between the origin of the panel and the first welding joint.
The software always recalculates the width of the last defined strake so that the ending joint of the strake coincides with the last node of the panel.

Distance along: specifies how the widths are measured. There are 3 possibilities:

-  : along the curve
-  : along the Y axis
-  : along the Z axis

Thickness (in mm): thickness of the current strake.

Material: material of the current strake.
By default, the material defined in the main section data (see 2.1) is assigned to the strake.

Hole Location (in m): distance from the beginning of the strake to the beginning of the hole (measured with respect of the *Distance Type*).

Hole Breadth (in m): distance from the beginning to the end of the hole (measured with respect of the *Distance Type*).

To perform an efficient entry of strakes:

Press F12 to create a new strake,	the current strake is divided into two, the first part is selected and the focus is on the width.
Define the width and press Enter,	the strake thickness is selected.
Input the thickness and press Enter,	the next strake is selected.
Press F12 to create a new strake	…

3.6 STIFFENER

3.6.1 Location

This chapter concerns the description of longitudinal secondary stiffeners. For transverse stiffeners, refer to 3.7 part.

Clicking on the *Stiffener* button (Figure 9) or on *Stiffeners* on the *Section* menu (Figure 46), you enter the *Stiffener management window* where these inputs are performed:

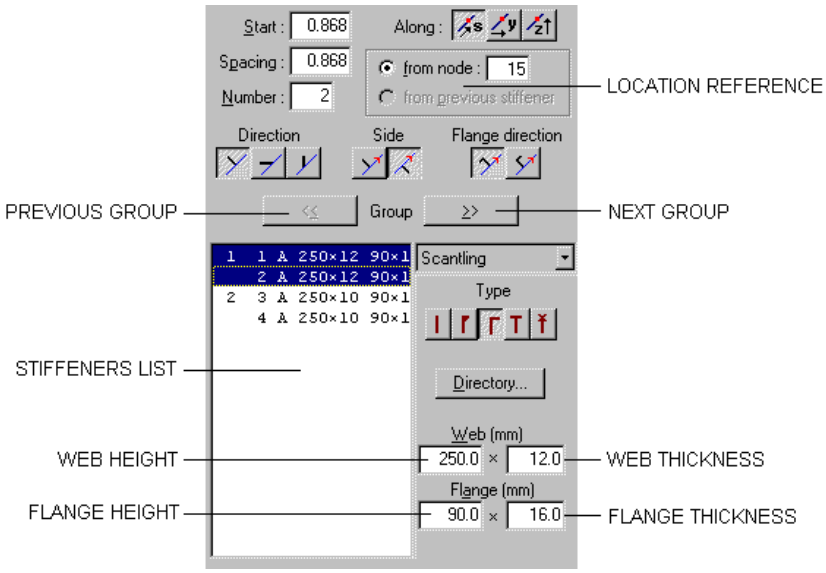


Figure 23: STIFFENER MANAGEMENT WINDOW

The longitudinal stiffeners have to be located on a panel. The successive locations of stiffeners must be given in an increasing order along the panel description.

This location is done by defining groups of stiffeners having the same spacing. The *creation* and the *deletion* buttons (Figure 9) allow you to create or delete groups of stiffeners.

It is necessary to define sequentially all the stiffener groups as they appear moving along the panel, without skipping any of them.

Stiffener List: Display of the stiffeners defined on the current panel and of their scantling. They're divided in groups of location which are numbered.
The list shows: the group number, the stiffener number and its scantling.
The stiffeners selected are the current stiffeners. They can belong to different groups.

Next group - Previous group: Those two buttons allow navigating group by group within a panel.

Selection of one stiffener
Click the stiffener in the *Stiffener List* or in the section view.

Selection of a group of stiffeners
Click a stiffener of the previous (next) group and push the *Next group* (*Previous group*) button.

Selection of stiffeners that are next to each other
Select the first stiffener you want to select; hold down the SHIFT key and click the last stiffener you want to select.

Select in the list the first stiffener you want to select, hold down the mouse button and drag the selection to the last stiffener you want.

Selection of stiffeners that are not next to each other
Hold down the CTRL key, and then click each stiffener you want to select.

Location Reference: It is either a node (select *from node* (Figure 23) and enter the node number), or the last stiffener of the previous group (select *from previous stiffener* (Figure 23)).

Start (in m): Distance from the location reference (node or last stiffener of previous group) to the first stiffener of the group.

Spacing (in m): spacing of the stiffeners for the group.

Number: number of stiffeners in the group.
A group of stiffener is a set of successive stiffeners having the same spacing, even if their scantlings differ.

Along: the starting distance and the stiffener spacing can be measured along

-  : the curve
-  : the Y axis
-  : the Z axis

Direction: allows to precise if the stiffener webs are:

-  : perpendicular to the curve
-  : parallel to the Y axis
-  : parallel to the Z axis

Side: allows to precise, according to the panel direction, if the stiffeners are:

-  : on the left side of the panel
-  : on the right side of the panel

Flange Direction: allows precising if the stiffener flanges are oriented:

-  : in the description direction of the panel
-  : in the opposite direction of the panel

3.6.2 Scantling

Scantling item
Stiffener Type: allows selecting the stiffener type:

-  : flat stiffener
-  : bulb stiffener
-  : angle stiffener
-  : T-bar stiffener

In case of flat or bulb type, flange characteristics aren't available.
If none type is chosen, the stiffener is considered as a NULL type, which is defined in 3.6.3.

Web Height (in mm): Height of stiffener web (H1).

Web Thickness (in mm): Thickness of stiffener web (E1).

Flange Height (in mm): Width of stiffener flange (H2).

Flange Thickness (in mm): Thickness of stiffener flange (E2).

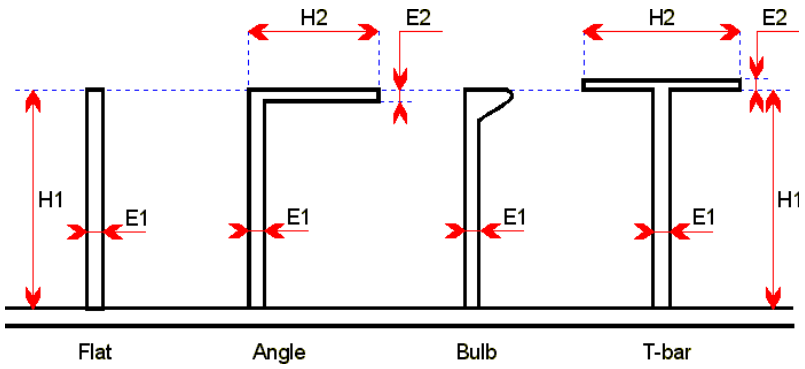


Figure 24: STIFFENER TYPE

Special item
A click on it displays the following windows:

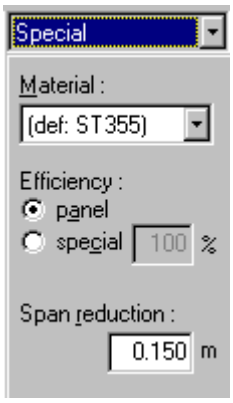


Figure 25: SPECIAL ITEM

Material: allows changing the current stiffener material, which, by default, is the supporting strake one.

Efficiency (in %): allows changing the bending efficiency of current stiffeners.

DO NOT ENTER 50% FOR STIFFENERS ON THE CENTERLINE
DO NOT ENTER 50% FOR STIFFENERS LOCATED ON A PANEL ON THE CENTERLINE

In that case, enter 100 %: the program detects their location and takes them into account only once

Span reduction (in m): allows defining the span reduction of current stiffeners.

Brackets item
A click on it displays the following windows:

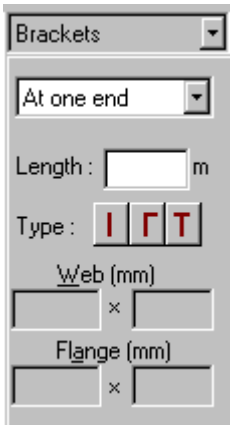


Figure 26: BRACKETS ITEM

The brackets may be defined:

- *No bracket ;*
- *At one end ;*
- *At both end.*

Length (in m): allows defining the bracket length.

Type: (see 3.6.2. – *Scantling item*)

3.6.3 Null type

A stiffener with NULL type has no effect on the geometric characteristics (areas, inertia, moduli) of the section.

But it may be used in different ways with program convention as follows:

- Type NULL and a bending efficiency coefficient equal to 0 %.

  This type of stiffener allows defining an intermediate support for a transverse stiffener without affecting the span of the plating. It is a convenient way to enter a strut linking transverse frames in inner hull, for instance.

  Plating longitudinally stiffened: no action.

  Plating transversally stiffened: support of the transverse member only.

- Type NULL and a bending efficiency coefficient not equal to 0 %.

  This type of stiffener allows in the considered transverse section to reduce the plate extension without modification of the supporting conditions of the transverse members.

  Plating longitudinally stiffened: this stiffener reduces the spacing E used to calculate the scantlings of the plating and of the adjacent longitudinals.

  Plating transversally stiffened: this stiffener reduces the span used to calculate the scantlings of the plating. It is not considered as an intermediate support for the transverse stiffener.

3.7 TRANSVERSE STIFFENING

This sequence allows basically defining the areas transversally stiffened so that MARS is able to distinguish plating longitudinally stiffened from plating transversally stiffened.

But the same sequence may be considered as defining the secondary transverse stiffeners whose scantlings may be evaluated in the calculation modulus.

The areas transversally stiffened have to be described by giving:

- their location
- the corresponding spacing of secondary transverse stiffeners
- the type of secondary transverse stiffener

In a more precise definition, the areas transversally stiffened have to be located with respect of the starting and ending points of the transverse stiffeners.
The lengths of associated brackets may be defined for span correction as stated in the Rules.

Every plating not considered in this sequence is supposed to be longitudinally stiffened.

The locations are performed by giving:

- a START point (node or stiffener) and
- an END point (node or stiffener).

The START point must be located before the END point in way of the panel description.

Clicking on the *Transverse Stiffening* button (Figure 9) or on *Transverse Stiffening* on the *Section* menu (Figure 46), you enter the *Transverse Stiffening management window* where these inputs are performed:

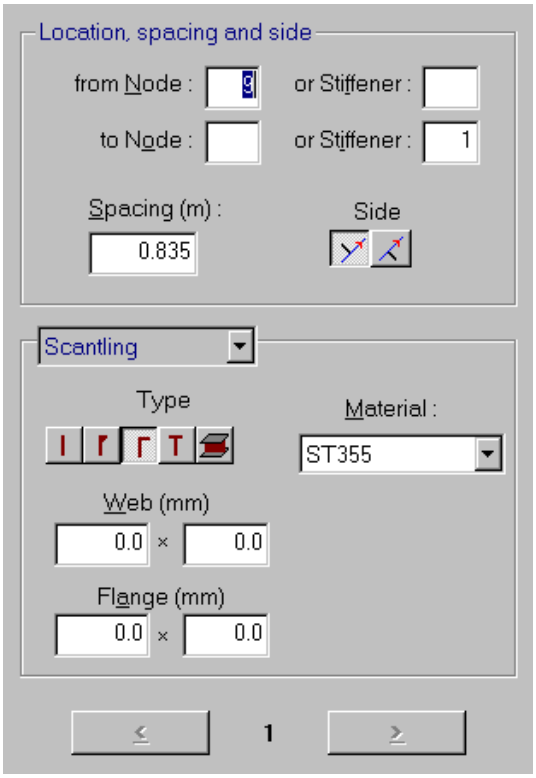


Figure 27: TRANSVERSE STIFFENING MANAGEMENT WINDOW

Location: enter either a node or a stiffener number in the corresponding input fields to define the start and end of stiffener. The stiffener numbers are always related to their location numbering on the panel.

Spacing (in m): the transverse frame spacing to be considered for all members (stiffener or part of strake) located inside this transverse area.

Side: allows precising, according to the panel direction, if the stiffeners are:

-  : on the left side of the panel
-  : on the right side of the panel

Scantling item

Type: allows selecting the stiffener type:

-  : Primary supporting members.
- See 3.6.2 for the other scantling types.

Material: allows defining the current stiffener material.

Bracket at start item
A click on it displays the following windows:

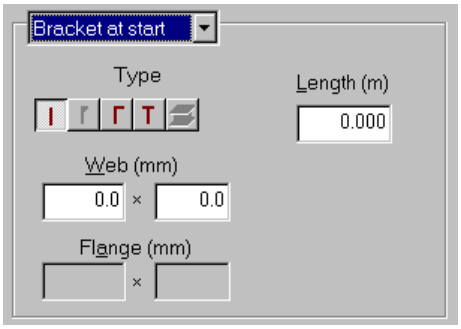


Figure 28: BRACKET AT START ITEM

Length (in m): the bracket length are useful to get the rule scantling of the transverse stiffener.

Type: (see 3.6.2. – *Scantling item*).

Bracket at end item
A click on it displays the following windows:

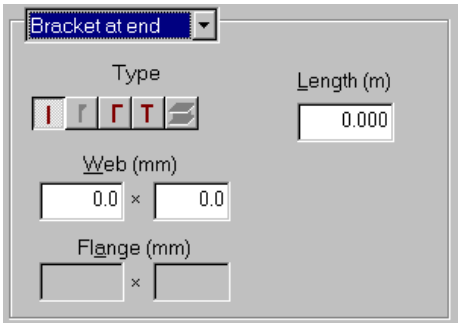


Figure 29: BRACKET AT END ITEM

Length (in m): the bracket length are useful to get the rule scantling of the transverse stiffener.

Type: (see 3.6.2. – *Scantling item*).

3.8 SPECIAL SPAN

The rule scantling of elements (strakes and longitudinal stiffeners) is based on a longitudinal span. The default values are those defined for the panel (in general panel description).
This input allows defining panel areas where a special span is to be considered (for strakes and/or longitudinals).

Clicking on the *Special Span* button (Figure 9) or on *Special Span Areas* on the *Section* menu (Figure 46), you enter the *Special Span management window* where these inputs are performed:

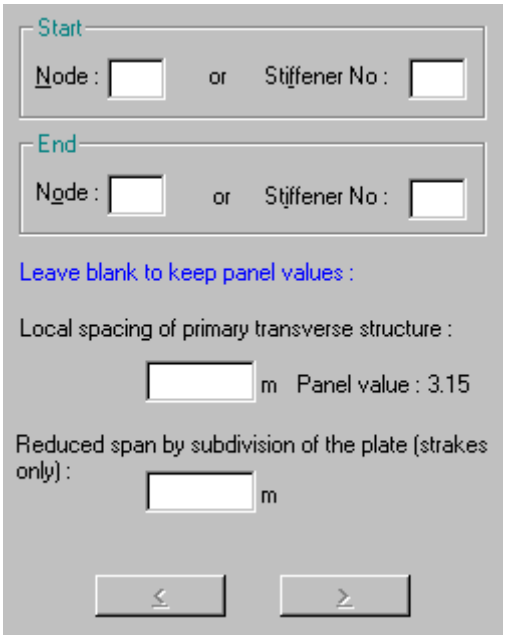


Figure 30: SPECIAL SPAN MANAGEMENT WINDOW

Start: enter either a node or a stiffener number in the corresponding input field.

End: enter either a node or a stiffener number in the corresponding input field.

Local spacing of primary transverse structure (in m): special span value for primary transverse structure.

Reduced span by subdivision of the plate (in m): special span value for strakes only.

IF YOU WANT TO KEEP PANEL VALUES LEAVE BLANK THE TWO INPUT FIELDS.

Chapter 4 : COMPARTMENTS AND LOADS

The compartments and loads descriptions are managed by the toolbar as follows :

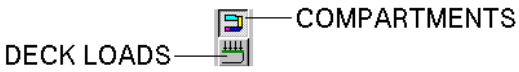


Figure 31 : COMPARTMENTS AND LOADS TOOLBAR

4.1 COMPARTMENTS

Clicking on the *compartment* button (Figure 31) or on *Compartments - Loading cases* on the *Section* menu (Figure 46), you enter the *Compartments management window*:

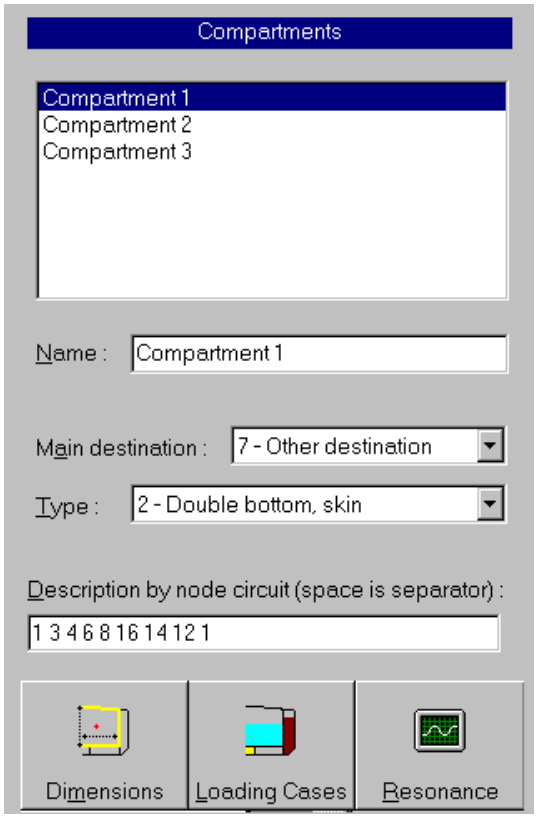


Figure 32 : COMPARTMENTS MANAGEMENT WINDOW

In this window, you can create or delete compartments using the *creation* or the *deletion* buttons (Figure 11). Each compartment is defined by a name, a main destination, a type and a list of node.

Name: used to identify the compartment.

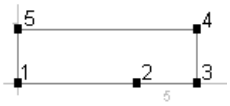
Main destination: choose a compartment main destination in the list.
The possible destinations are:

1	Ballast
2	Cargo oil / Fuel oil
3	Other Liquid Cargo
4	Dry bulk
5	Spoil
6	Accomodation space
7	Other destination

Type: choose a compartment type in the list.
The possible types are:

1	Tank
2	Double bottom, skin
3	Dry compartment
4	Engine room
5	Boiler compartment
6	Tunnel
7	Hopper well 1
8	Hopper well 2

Description by node circuit: enter the list of nodes of compartment contour.
If the compartment is entirely located on one side of the section, the first and last nodes are to be the same.
You don't have to enter all the nodes along the compartment contour. For instance:



can be describe with: 1 4 5 1. The result is:

The list of nodes is also used in the calculation modulus to determine the highest point after rolling in tanks.
The contour of an opened hold is generally not complete because the hatch cover has not to be defined in MARS. So the upper node on the vertical axis is missing. To obtain a correct calculation of the center of gravity, it is necessary to start the description of the contour by the low node located on the vertical axis: MARS will close the contour with an horizontal line at the level of the last defined node.

4.1.1 Dimensions

Clicking on the *Dimensions* button (Figure 32), the compartment-dimension window is displayed:

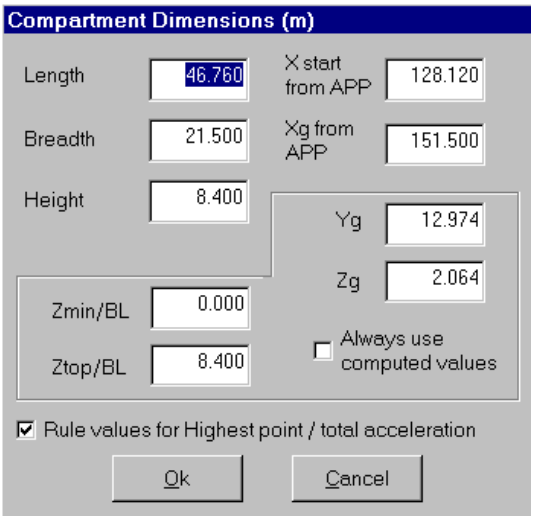


Figure 33: COMPARTMENT-DIMENSION WINDOW

Length (in m): compartment length.

Breadth (in m): compartment breadth.

Height (in m): compartment height.

X start (in m): longitudinal location of start compartment from APP.

Xg (in m): longitudinal location of compartment center of gravity from APP.

The following data may be obtained by calculation clicking on *Always use compute values* check or by direct input:

Yg (in m): center of gravity (Centerline).

Zg (in m): center of gravity above base line.

Zmin/BL (in m): min of compartment from base line.

Ztop/BL (in m): top of compartment from base line.

If you don't want use the Rule values for highest point / total acceleration click in the related check. The window become:

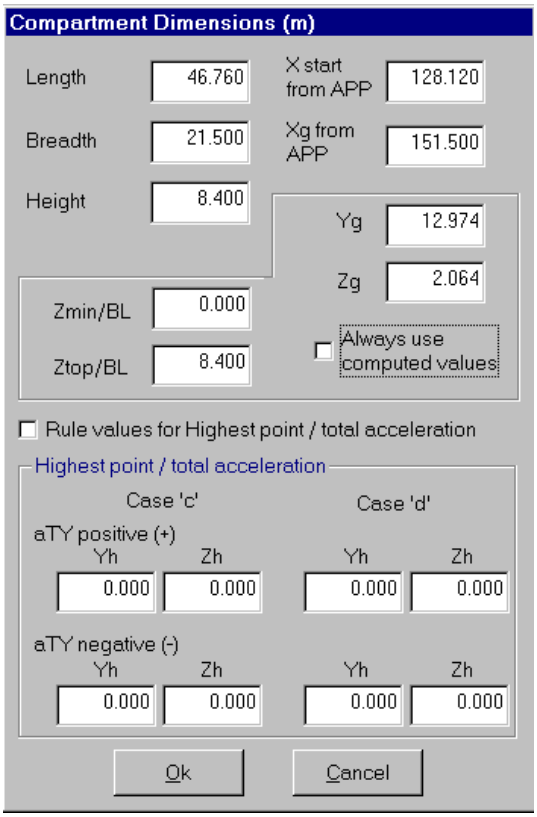


Figure 34: COMPARTMENT-DIMENSION WINDOW (2)

4.1.2 Loading cases

Clicking on the *Loading Cases* button (Figure 32), the compartment load window is displayed:

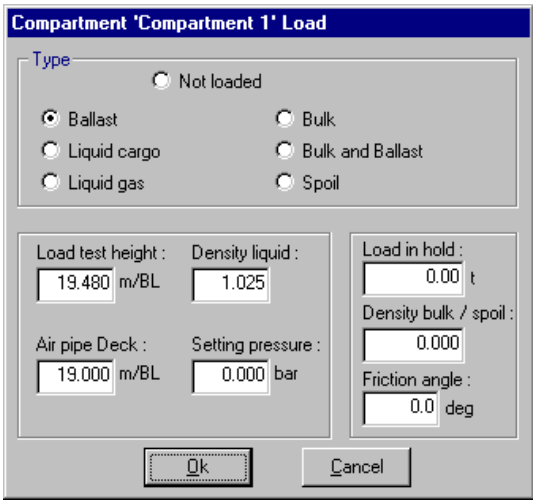


Figure 35: COMPARTMENT LOAD WINDOW

Type: select a load type.

Liquid cargo
Load test height (in m): tank testing load height from base line.

Air Pipe (in m): distance from top of air pipe height to base line.

Density liquid: density of liquid cargo.

Setting pressure (in bar): setting pressure of safety valves.

Bulk cargo
Load in hold (in t): load in hold (bulk or spoil).

Density of bulk or spoil: density of bulk or spoil.

Friction angle (in deg): internal friction angles for bulk.

This input allows defining for the same compartment a liquid cargo and a bulk cargo. This facility may be used for ballastable hold in bulk carrier.

4.1.3 Resonance

Clicking on the *Resonance* button (Figure 32), the Resonance window is displayed.

The user may ask to the program to calculate in any case the risk of resonance due to roll or sway checking the relative checkbox on the window.

In Tank geometry frame the user may choose the case to calculate the coefficient for reference pressure for calculation of sloshing loads.

The first tab of the Resonance window allows defining the values in inclined ship condition.

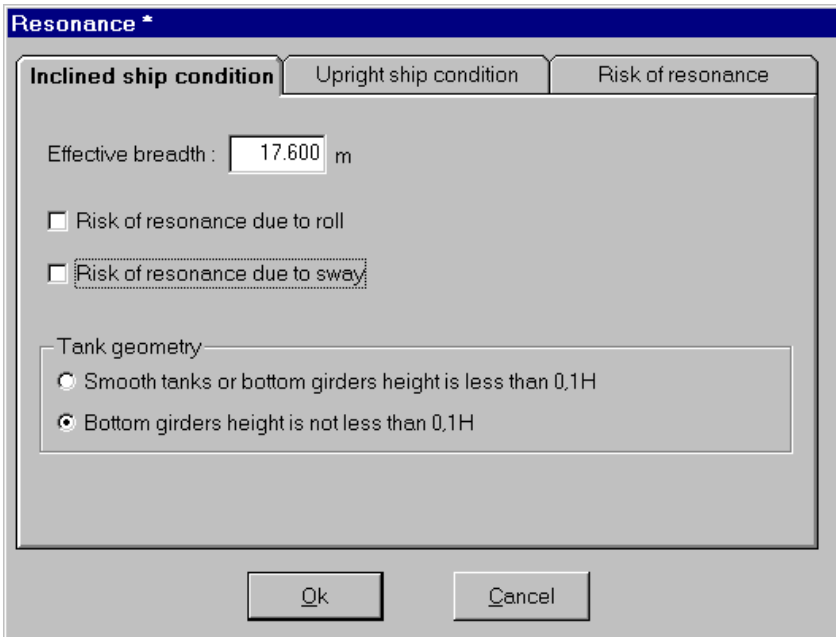


Figure 36: RESONANCE - INCLINED SHIP CONDITION

Effective breadth (in m): breadth of the free surface of the liquid, measured horizontally with the ship at rest and depending on the filling level d_F for ships without watertight or wash longitudinal bulkheads; for ships fitted with watertight or wash longitudinal bulkheads the effective breadth is delimited by these bulkheads.

The second tab of the Resonance window allows defining the values in upright ship condition.

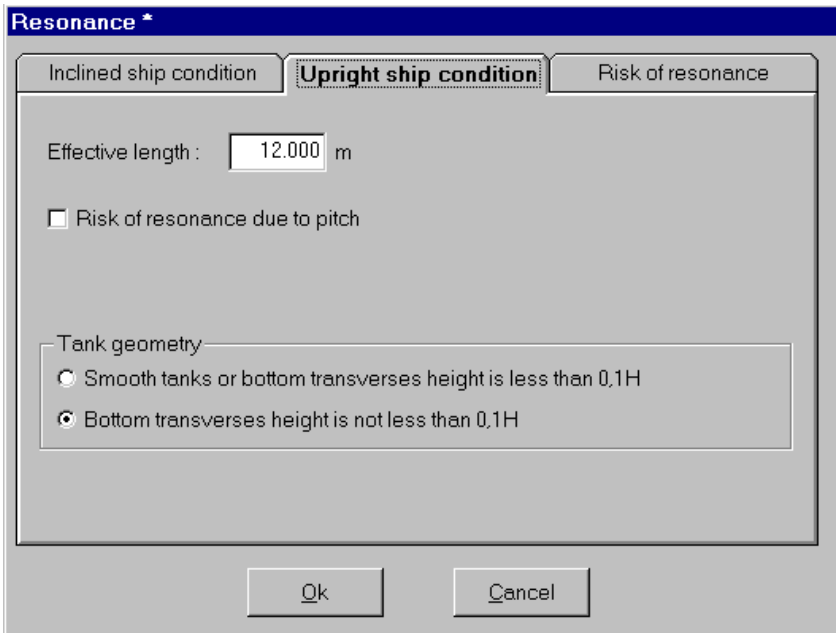


Figure 37: RESONANCE - UPRIGHT SHIP CONDITION

Effective length (in m): length of the free surface of the liquid, measured horizontally with the ship at rest and depending on the filling level d_F.

The last tab of the Resonance window allows to define the filling level and the corresponding free surface length and free surface breadth to evaluate the risk of resonance for each of these.

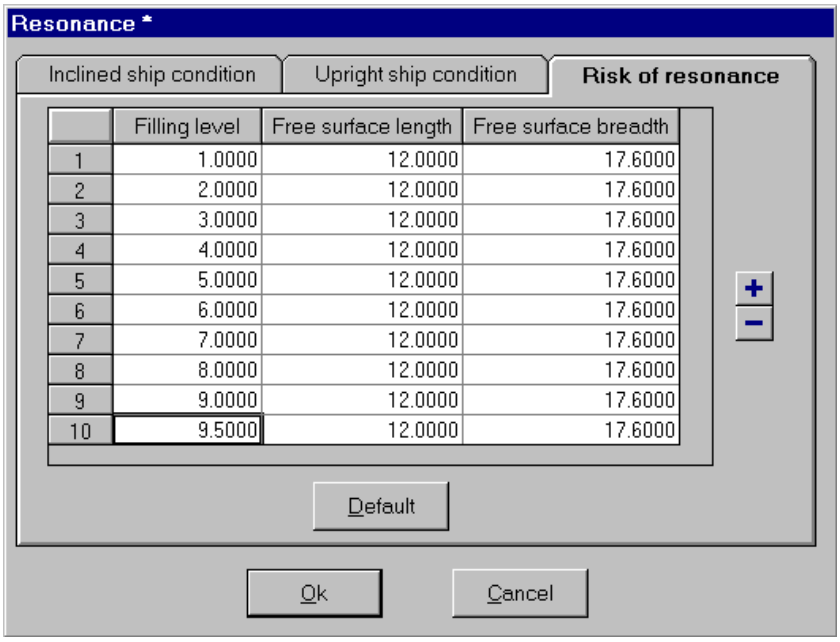


Figure 38: RISK OF RESONANCE (2)

These data are used to evaluate the risk of resonance of the compartment in roll, sway and pitch. This check is reachable through the prints (see 6.2.1).

4.2 DECK LOAD

The definition of deck load is panel oriented. It means that a deck load is defined between two nodes in a panel.

Clicking on the *Deck Load* button (Figure 31) or on *Deck Load* on the *Section* menu (Figure 46), you enter the *Deck Load management window*:

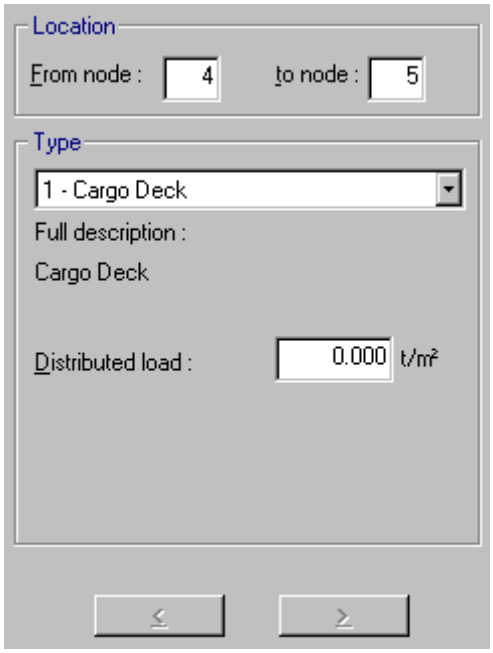


Figure 39: DECK LOAD MANAGEMENT WINDOW

From Node: node number for the start of load extension.
The starting point of the load extension must be located before ending point (in way of the panel description).

To Node: node number for the end of load extension.

Type: choose the load type in the list proposed.
The possible types are:

1	Cargo Deck
2	Accommodation deck : large public rooms
3	Accommodation deck : large rooms with fixed furniture or cabins
4	Accommodation deck : other spaces
5	Wheel load

Distributed load (in t/m²): in case of cargo deck only.

In case of Wheel load type the window become:

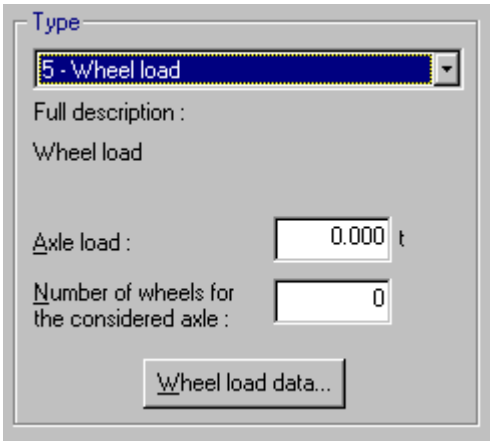


Figure 40: DECK LOAD MANAGEMENT WINDOW (2)

Axle load (in t): load for considered axle.

Number of wheels for the considered axle: wheel number.

The definition of a wheel load data is completed by means the window here-after obtained by clicking on the *Wheel load data…* button:

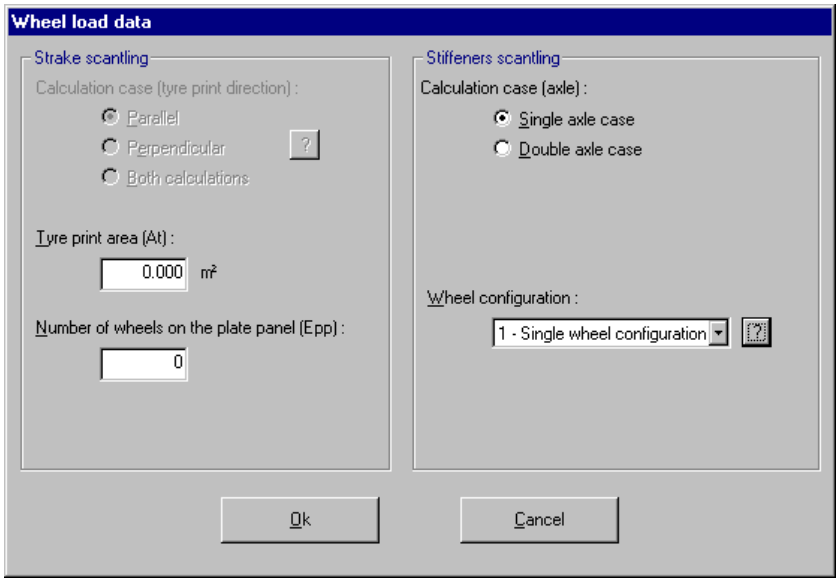


Figure 41 : WHEEL LOAD DATA WINDOW

Wheel configuration: choose a configuration in the list.

The possible configurations are:

1	Single wheel configuration
2	Double wheel configuration
3	Triple wheel configuration

Chapter 5 : FATIGUE

Clicking on the *Fatigue* button  or on *Fatigue* on the *Section* menu (Figure 46), you enter the *Fatigue management window*:

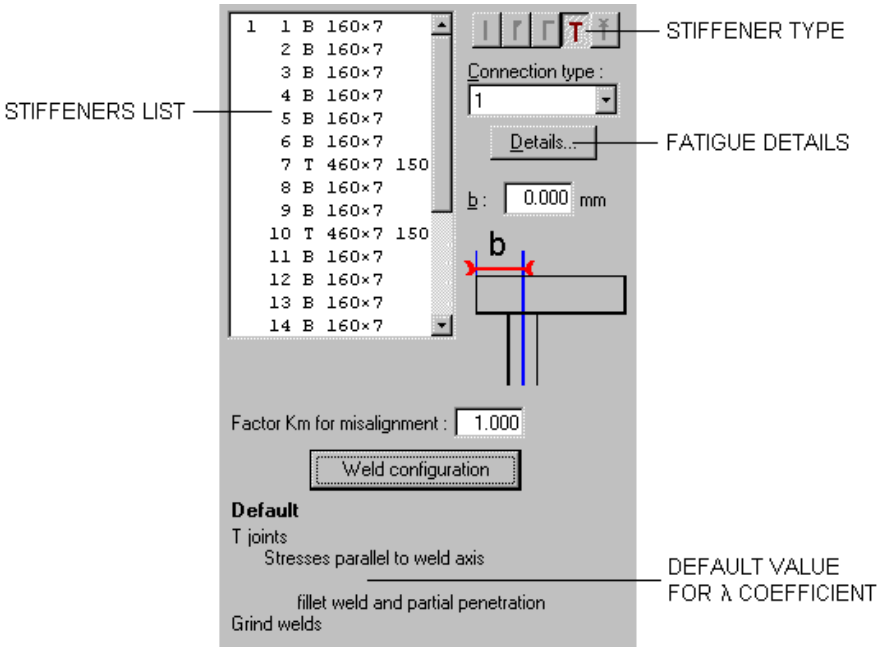


Figure 42: FATIGUE MANAGEMENT WINDOW

Stiffeners list: allows selecting the stiffeners to be fatigued.

Stiffeners type: displays the type of the selected stiffeners.

The fatigue details can be selected by:

- the details… button (Figure 42) which display a view of the connection type and of the collar plate with the relevant Kh and Kl coefficients ;

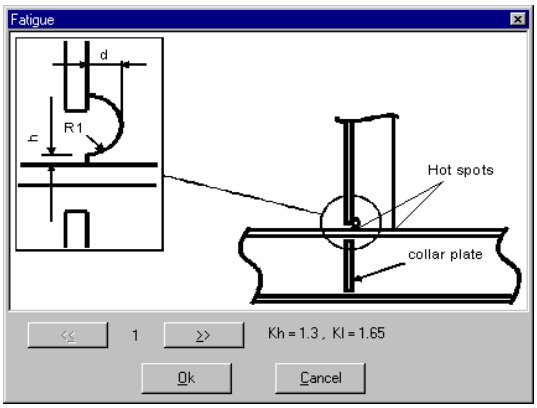


Figure 43: FATIGUE DETAILS

- *the* Connection type *list (*Figure 42*)*

b (in mm): geometric characteristic of a stiffener for fatigue.

Factor Km for misalignment: allows defining the stress concentration factor for misalignment.

Clicking on Weld configuration button you can change for the stiffener selected the default value for λ coefficient defined in Main Section data (see 2.3. - *fatigue tab*).

Chapter 6 : GENERAL FEATURES

6.1 MENUS

File Menu
It allows managing the sections (save, open), to print and to quit MARSIN.

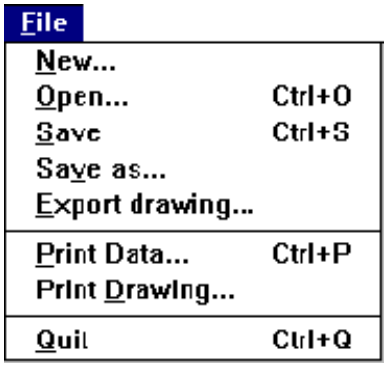


Figure 44: FILE MENU

Item	Use	Shortcut
New…	Creates a new section.	
Open..	Opens an existing section.	You can also press Ctrl + O.
Save	Saves the opened section.	You can also press Ctrl + S or
Save as…	Saves a copy of the section with another name.	
Export drawing	Creates a bitmap file from the section drawing.	
Print Data…	Prints the data of the section (see 6.2.1).	You can also press Ctrl + S or
Print Drawing…	Prints a drawing of a section (see 6.2.2).	
Quit	Quits MARSIN to return to MARSHELL.	You can also press Ctrl + Q or

Edit Menu
It allows undoing the last action or copy the section drawing to clipboard.

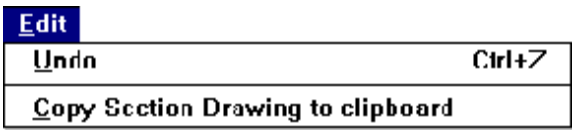


Figure 45: EDIT MENU

Item	Use	Shortcut
Undo	Cancels the last action.	Ctrl + Z
Copy Section Drawing to clipboard	Allows pasting the section drawing in any other application.	

Section menu
It gathers the entries to the input fields.

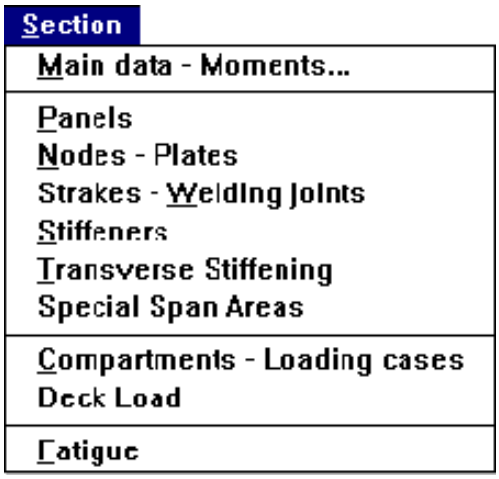


Figure 46: SECTION MENU

Item	Use	Shortcut
Main data - Moments…	Displays the Main Section Data Window (see Chapter 2 :).	
Panels	Displays the Panels management window (see 3.3).	
Nodes - Plates	Displays the Nodes management window (see 3.4).	
Strakes - Welding joints	Displays the Strakes management window (see 3.5).	
Stiffeners	Displays the Stiffeners management window (see 3.6).	
Transverse Stiffening	Displays the Transverse Stiffening management window (see 3.8).	
Special Span Areas	Displays the Special Span management window (see 3.8).	
Compartments - Loading cases	Displays the Compartments management window (see 4.1).	
Deck Load	Displays the Deck Load management window (see 4.2).	
Fatigue	Displays the Fatigue management window (see Chapter 5 :).	

Check menu
It includes checking tools.

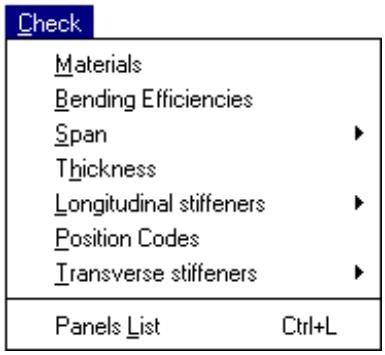


Figure 47: CHECK MENU

Item	Use	Shortcut
Materials	Displays the strakes and stiffeners with a different color for each material.	
Bending Efficiencies	Displays the strakes and stiffeners with a different color for each bending efficiency.	
Span	(see Figure 48)	
Thickness	Displays the strakes with a different color for each thickness.	
Longitudinal stiffener	(see Figure 49)	
Positions Codes	Displays the segments with a different color for each position code.	
Transverse stiffener	(see Figure 50)	
Panel List	Displays a list of the current entry data. These lists are the exact reflect of what would be printed with the *Print Data* function (see 6.2.1).	Ctrl + L

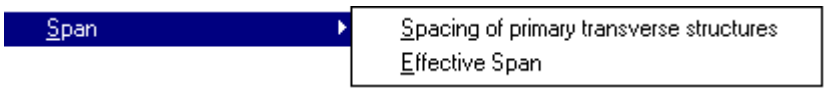


Figure 48: SPAN SUB-MENU

Item	Use	Shortcut
Spacing of primary transverse structures	Displays the strakes with a different color for each span.	
Effective Span	Displays the strakes and stiffeners with a different color for each span.	

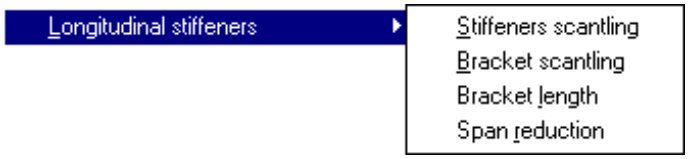


Figure 49: LONGITUDINAL STIFFENERS SUB-MENU

Item	Use	Shortcut
Stiffeners scantling	Displays the stiffeners with a different color for each stiffener scantling.	
Bracket scantling	Displays the brackets with a different color for each bracket scantling.	
Bracket length	Displays the bracket length.	
Span reduction	Displays the span reduction.	

Figure 50: TRANSVERSE STIFFENERS SUB-MENU

Item	Use	Shortcut
Stiffeners scantling	Displays the transverse stiffening zones with a different color for each scantling of those transverse.	
Stiffeners spacing	Displays the transverse stiffening zones with a different color for each spacing of those transverse.	

***Tools* menu:**

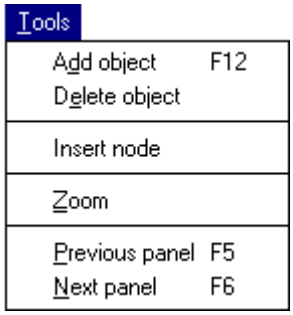


Figure 51: TOOLS MENU

Item	Use	Shortcut
Add object	Creates data (see 3.2).	**or F12**
Delete object	Deletes data (see 3.2).	
Insert node…	Inserts a node (see 3.4.5), this item is available only during the node input sequence.	
Zoom	Allows zooming in (see 6.3).	
Previous panel	Changes the current panel to the previous one (see 3.1).	F5
Next panel	Changes the current panel to the next one (see 3.1).	F6

Options menu

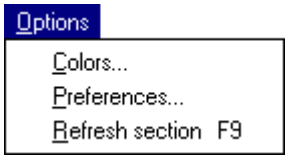


Figure 52: OPTIONS MENU

Item	Use	Shortcut
Colors…	Displays a set up window for the colors on the screen or a printer.	
Preferences	Displays a set up window for the drawing preferences on the screen or a printer.	
Refresh section	Refreshes the screen in case of display anomalies.	F9

6.2 PRINTING

6.2.1 Printing data

Clicking on  or on Print Data… on the File menu (Figure 44) or pressing Ctrl + S, you enter the Print Data management window:

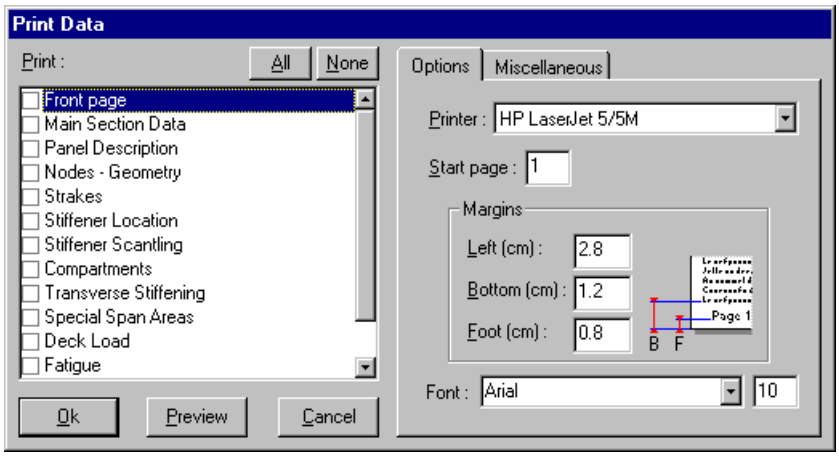


Figure 53: PRINT DATA MANAGEMENT WINDOW

This window allows you to select what you want to print. The *All* (*None)* button selects (deselects) all the items of the list.

Front page produces the cover page of a report.

6.2.2 Printing drawing

Clicking on *Print Drawing…* on the *File* menu (Figure 44), you enter the *Print Drawing management window*:

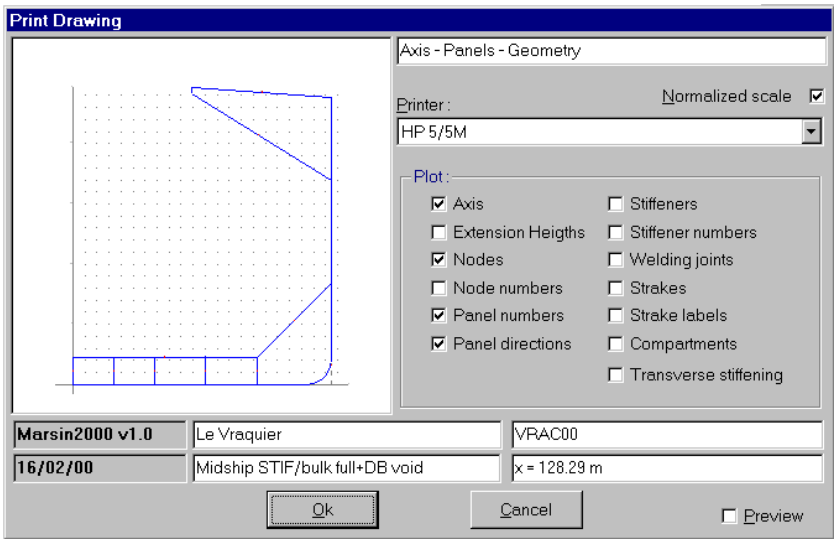


Figure 54: PRINT DRAWING MANAGEMENT WINDOW

This window allows you to select which item will be printed on the item.

The *Normalized scale* check box will make the drawing printed with a regular scale (e.g. 1/50, 1/100…).

6.3 ZOOM

It is possible to Zoom in on or out of the Section view thanks to the *Zoom* Toolbar:

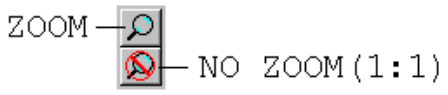


Figure 55: ZOOM TOOLBAR

A first click on the *Zoom* button (Figure 55) or on *Zoom* on the *Tools* menu (Figure 51) changes the cursor in  and allows you to zoom in the section view by simple click on it.

A second click on the *Zoom* button (Figure 55) or on *Zoom* on the *Tools* menu (Figure 51) changes back the cursor in  and allows you to work on your zoomed section view.

To zoom out of the section view, you can:

- Click on the *No Zoom (1:1)* button (Figure 55) to bring back the view to the initial size.
- Right-click on the section view when the *Zoom* button is down.

Table of Figure

MARS2000
User's guide

Booklet 3

CALCULATION OF A SECTION

February 2000 /1
Dev/UG/MARS2000/03

MARINE DIVISION
DEVELOPMENT DEPARTMENT
17 bis, place des Reflets – La Défense 2
92077 Paris – La Défense Cedex
Tel.: + 33 (0)1 42 91 52 91
Fax: + 33 (0)1 42 91 52 98

TABLE OF CONTENTS

Chapter 1 : GENERAL COMMENTS

1.1 MAIN FEATURES

The MARSRULE module allows to check a section according to Bureau Veritas Rules for Classification of Steel Ships. It checks:

- the strength characteristics of the hull girder,
- the scantling of the continuous longitudinal members – strakes and longitudinal ordinary stiffeners,
- the scantling of the transverse ordinary stiffeners.

MARSRULE is able to perform calculations in any section all along the ship length. The sections are to be defined as described in the booklet 'Definition of a section'.

The module allowing to perform the calculations for a given section is organized around the following application:

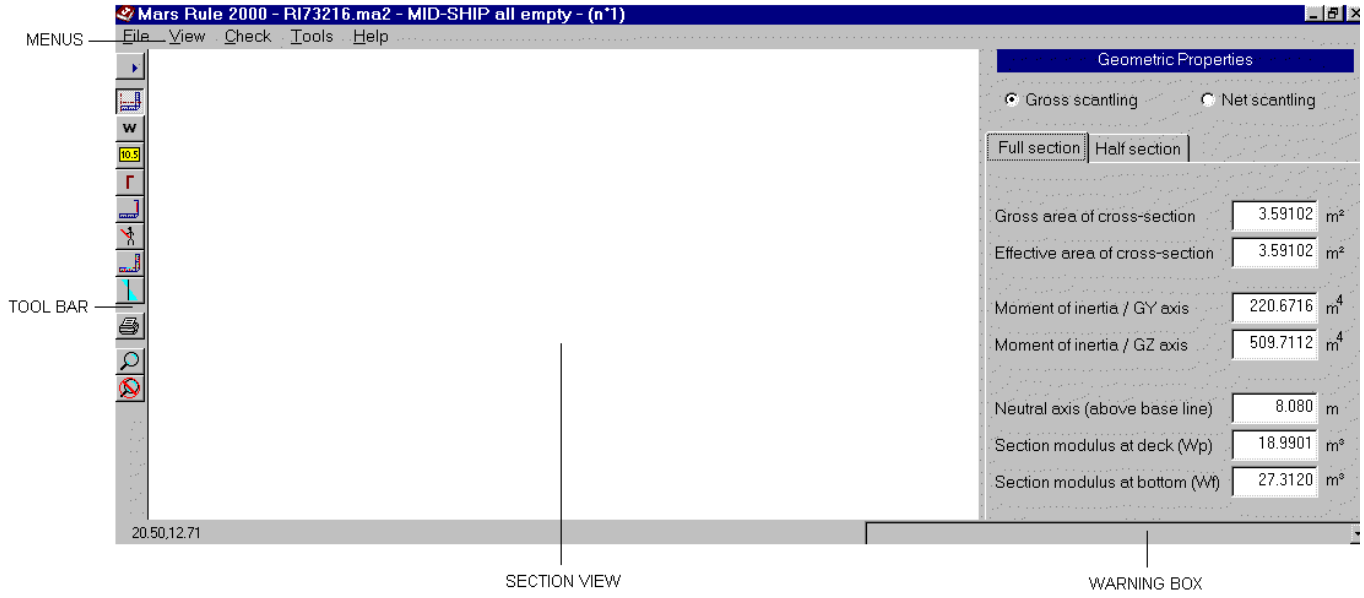


Figure 1: MARSRULE

Warning Box: displays warning message when MARSRULE detects incoherence in the Section definition.

Section View: displays a view of the section.

1.2 COMPUTE SECTION

When you launch MARSRULE module or you click on the Compute section button or on Compute section on the File menu (Figure 18), the Compute section window is displayed:

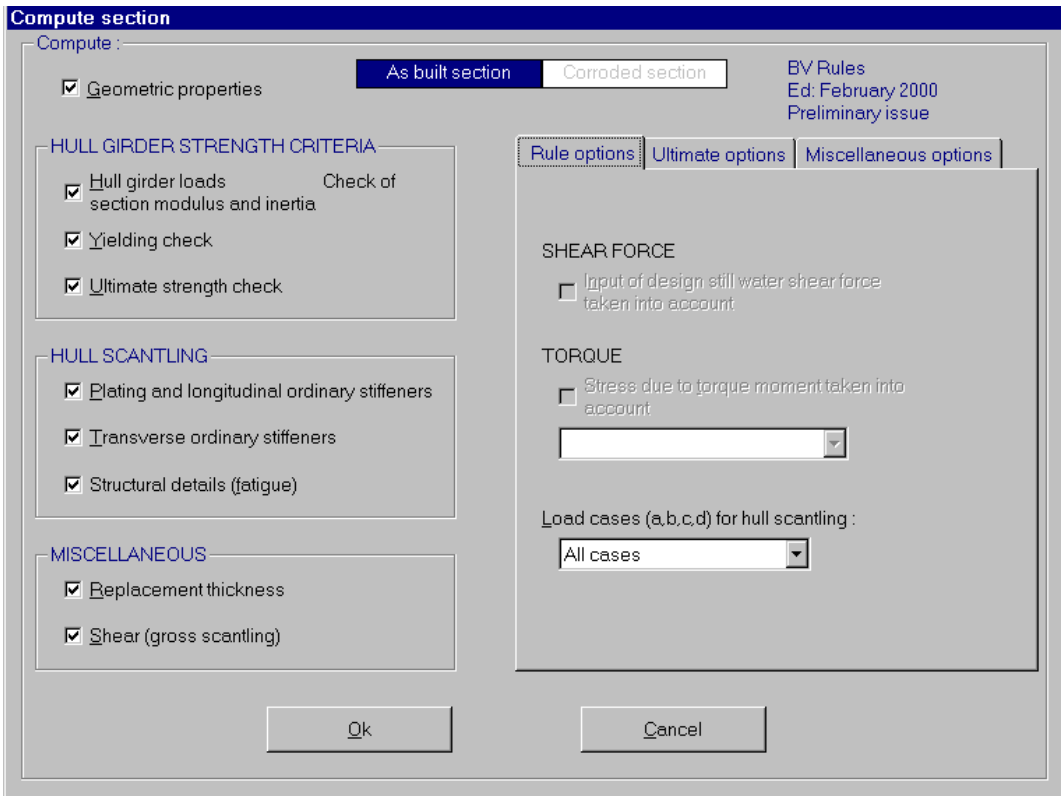


Figure 2: COMPUTE SECTION WINDOW

This window allows selecting one or several types of calculation from the following list:

- Geometric properties.
- Hull girder loads; check of section modulus and inertia.
- Yielding check.
- Ultimate strength check.
- Plating and longitudinal ordinary stiffeners.
- Transverse ordinary stiffeners.
- Structural details (fatigue)
- Replacement thickness.
- Shear (gross scantling).

Warning: Rule calculations for dredgers have not yet been implemented.

Rule options tab

SHEAR FORCE:
As required by the Rules, the shear stress is taken into account in criteria applied to strakes and longitudinal ordinary stiffeners (ships greater than 90 m in length). Consequently, exhaustive rule verification assumes that the user has defined a value of design still water shear force.

If this value is not entered, Mars uses a default shear stress as defined in Rules (ships greater than 90 m in length).
Clicking on Shear Force check box the calculation takes into account the input value of design still water shear force. If this value has not been defined (it means, equal to rezo), there is a warning.

TORQUE:
Clicking on Torque check box the calculation takes into account stresses due to torque moment. To do that, a torsion model, for which the calculations have been performed and saved into the database, has to be selected.

Load cases (a, b, c, d)
This list allows carrying out the hull scantling calculations for all the load cases (normal rule scantling) or only for a particular one, it means:

- load case a,
- load case b,
- load case c,
- load case d,
- dynamic pressure (resonance due to Roll),
- dynamic pressure (resonance due to Sway).

Ultimate options tab
Choosing the Ultimate strength check calculation it is possible to require calculations based on different assumptions.
The rule calculation is based on:

- the net scantling for the section,
- the Standard control for "Solution" item,
- the "Fixed horizontal/vertical curvatures ratio" for "Moment" item, with a ratio value equal to zero.

Following parameters may be tested:

- Solution

Elastic ideally plastic behaviour
Standard control
Plate failure mode
Beam-column failure mode
Flexural-torsional failure mode
Local (web) failure mode

- Moment

Fixed vertical bending moment
Fixed horizontal bending moment
Fixed horizontal/vertical bending moments ratio
Fixed vertical/horizontal bending moments ratio
Fixed horizontal/vertical curvatures ratio
Fixed vertical/horizontal curvatures ratio

- Ratio value
- Gross scantling calculation check

Miscellaneous options tab
Clicking on User defined value option button launchs the calculation with the user defined vertical wave bending moment, in case of the user has entered correspondinge values in the Main Section data.

In that case, the section is not evaluated according to the Rules.

1.3 CONCEPTS

The following section gives some guidance about two concepts widely used in MARS, the net scantling and the Elementary Plate Panel.

1.3.1 Net Scantling

Depending on the rule criteria, the calculations have to be performed with the gross or with the net scantling of the section.

Gross scantling: hull girder strength criteria (bottom and deck moduli; yielding check)

Net scantling: verification of the scantling of the strakes and ordinary stiffeners, including fatigue calculation for structural details; ultimate strength check of the hull girder.

In MARS, Actual Net Scantling refers to a scantling automatically deduced by the program from the Actual Gross Scantling.This evaluation, for every stake and every stiffener, is based on the location of the considered element in the section and on the attribute "Main destination" of the compartments surrounding the element.

The "Main destination" defines the corrosion margin to be applied to the element.

1.3.2 Elementary Plate Panel - EPP

The rule criteria apply to every strake and every stiffener.
But, even is the approval process has to be performed at the stake level, the calculations are performed at a more elementary level, the Elementary Plate Panel which is a unit of plating stiffened on its four sides.
So, local calculations for plating are always performed E.P.P by E.P.P. The syntheses of calculations are made at the strake level.

We may distinguish two cases:

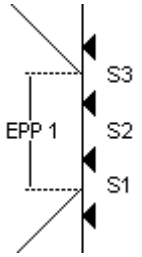
PLATING LONGITUDINALLY STIFFENED

An elementary plate panel (E.P.P.) is the unit of plating bordered by two longitudinal ordinary stiffeners and limited by the primary transverse structure.

To check the strake S2, MARSRULE checks the EPP's 1,2, 3 and 4. The EPP 1 and 4 are assumed to have the thickness of strake S2.

But, when checking the strake S3, MARSRULE reuses, as first EPP of the strake S3, EPP 4 with an assumed thickness equal to S3 thickness.

PLATING TRANSVERSALLY STIFFENED



An elementary plate panel (E.P.P.) is the unit of plating bordered by two transverse ordinary stiffeners and limited by the primary longitudinal structure.

To check strake S2, MARSRULE calculates the large EPP 1 assumed to have an homogeneous thickness equal to the thickness of strake S2.

Chapter 2 : GEOMETRIC PROPERTIES

Clicking on the *Geometric properties* button  or on *Geometric properties* on the *View* menu (Figure 19), The user enter the G*eometric properties result* window:

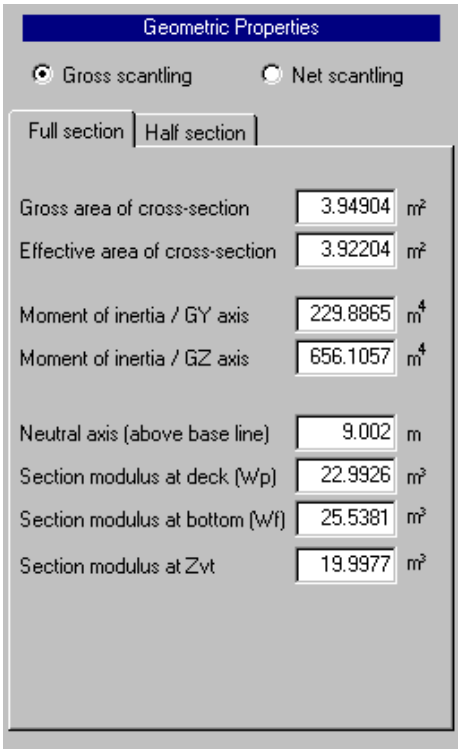


Figure 3: GEOMETRIC PROPERTIES RESULT WINDOW

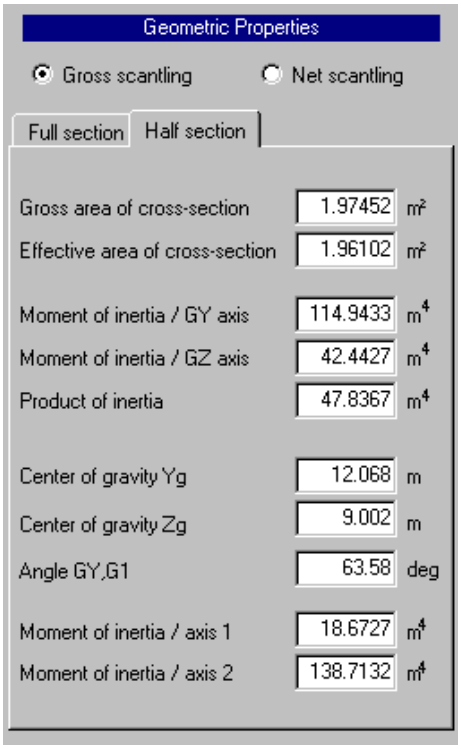


Figure 4: GEOMETRIC PROPERTIES RESULT WINDOW (HALF)

The program provides results for Gross scantling and Net scantling.

All results except gross area are effective values: it means that, compared to the calculation of gross area, the program takes into account mechanical properties as:

- Young modulus: the calculation, is provided assuming an homogeneous material with Young modulus equal to:
  206.000 N/mm2, for a ship built in steel
  70.000 N/mm2, for a ship built in aluminium alloy

  Ship built in steel:
  The program corrects the areas by multiplying by the hereunder ratio which takes into account the actual Young modulus and c coefficient of the material of the considered plating.
  Actual Young modulus / (206.000 * c)

  Ship built in aluminium alloy:
  The program corrects the areas by multiplying by the hereunder ratio which takes into account the actual Young modulus of the material of the considered plating.
  Actual Young modulus / 70.000

- Bending efficiency: when strakes and longitudinal stiffeners are defined with a bending efficiency different from 100%%, it means that they do not have a full contribution to the overall strength. To take their effective contribution into account, the area of these elements is multiplied by the actual value of the bending efficiency coefficient.

Full section tab

- Gross area of cross-section : pure geometric calculation

- Effective area of cross-section: different from gross area when strakes and longitudinal stiffeners are defined with a bending efficiency different from 100.

Section moduli are calculated:

- at deck (Wp)

- at bottom (Wf)

- at distance Vt from neutral axis

A modulus is calculated at a distance Vt from neutral axis in case where there are continuous trunks and longitudinal hatch coamings contributing to the longitudinal strength. This Vt value is evaluated in a point defined by its coordinates b (Y coordinate) and V1 (distance to neutral axis). This point is the one that leads to the maximum value of Vt.

Half section tab

The results displayed are relevant to the half section only (Fig. 14).

Moreover the program prints for the whole section a list of the different types of stiffeners existing in the section and a list of thickness with their corresponding length in the section.

Chapter 3 : GLOBAL STRENGTH CRITERIA

Clicking on the *Global strength criteria* button  or on *Global strength criteria* on the *View* menu (Figure 19), you enter the *Global strength criteria result* window

3.1 HULL GIRDER LOADS

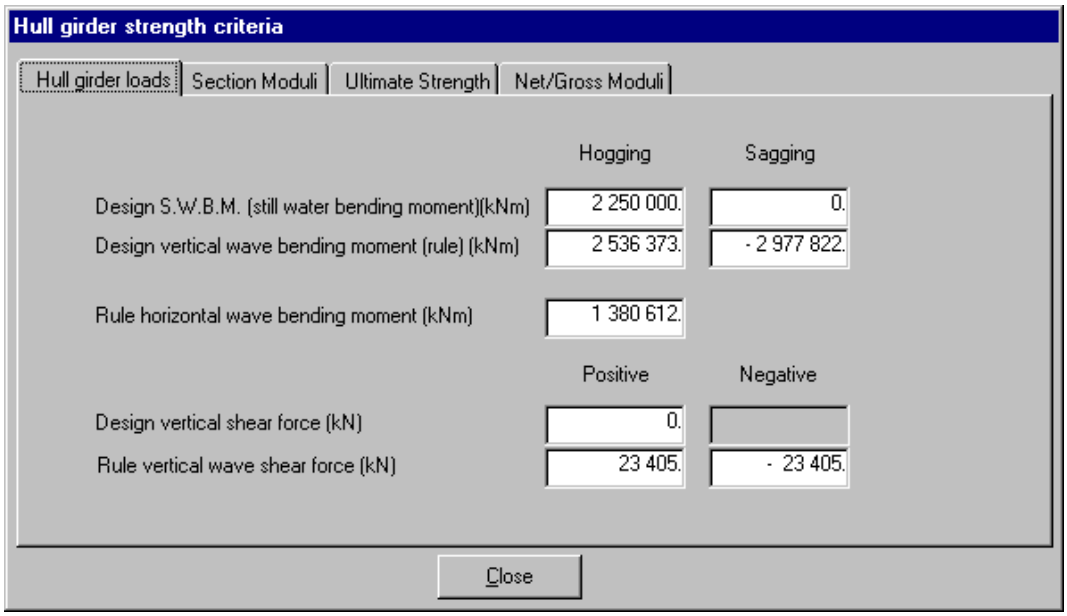


Figure 5: GLOBAL STRENGTH CRITERIA RESULT WINDOW (1)

Still water bending moment
The value of the design still water bending moments used to calculate the section are evaluated as follows:

- Equal+ to local still water moments (builder's proposal) entered in the module 'Definition of a section' for the current section, if they have been defined.

- Otherwise, equal to the values calculated at the longitudinal location of the section according to the rule distribution law. The maximum value of this distribution is equal to the still water moments (builder's proposal) entered in the module 'Basic ship data' if they have been defined. Otherwise, the program uses the rule permissible still water bending moments.

Wave bending moment
Design vertical wave bending moments are basically rule value.
However if the user has defined values in input process and if he has explicitly required non-rule calculation, these value are used and printed out.

3.2 SECTION MODULI

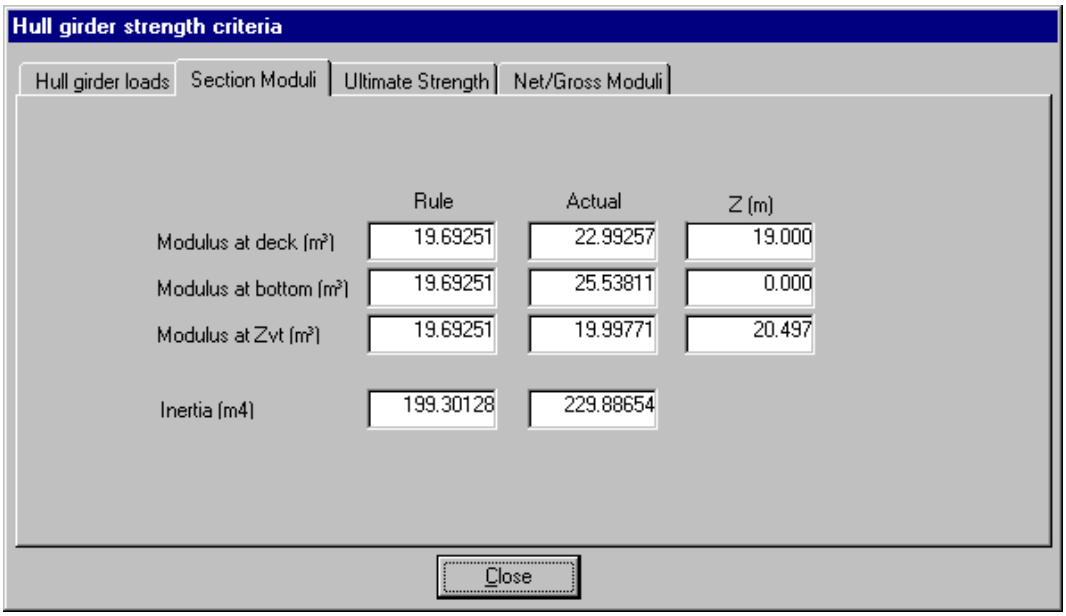


Figure 6: GLOBAL STRENGTH CRITERIA RESULT WINDOW (2)

The global characteristics of a section subject to rule criteria are:

- The moduli at bottom, at deck at side and at a distance Vt.

- Only for midship section, the moment of inertia of the section about the neutral axis.

The rule values of modulus at bottom, at deck at side and at a distance Vt are based on design bending moments.

The program displays a summary allowing comparing the rule values of moduli to the actual values. The red color means that the actual value does not comply with the rule one.

Even in the case of non-compliance with rules, the actual values are used for the calculations of the local rule scantlings.

3.3 ULTIMATE STRENGTH

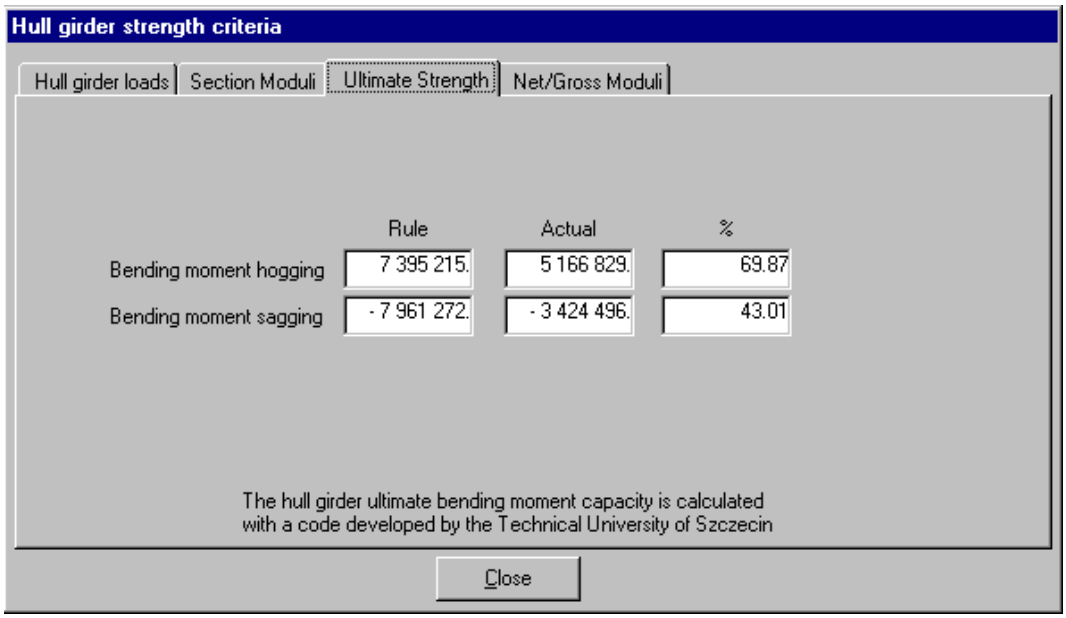


Figure 7: GLOBAL STRENGTH CRITERIA RESULT WINDOW (3)

3.4 NET/GROSS MODULI

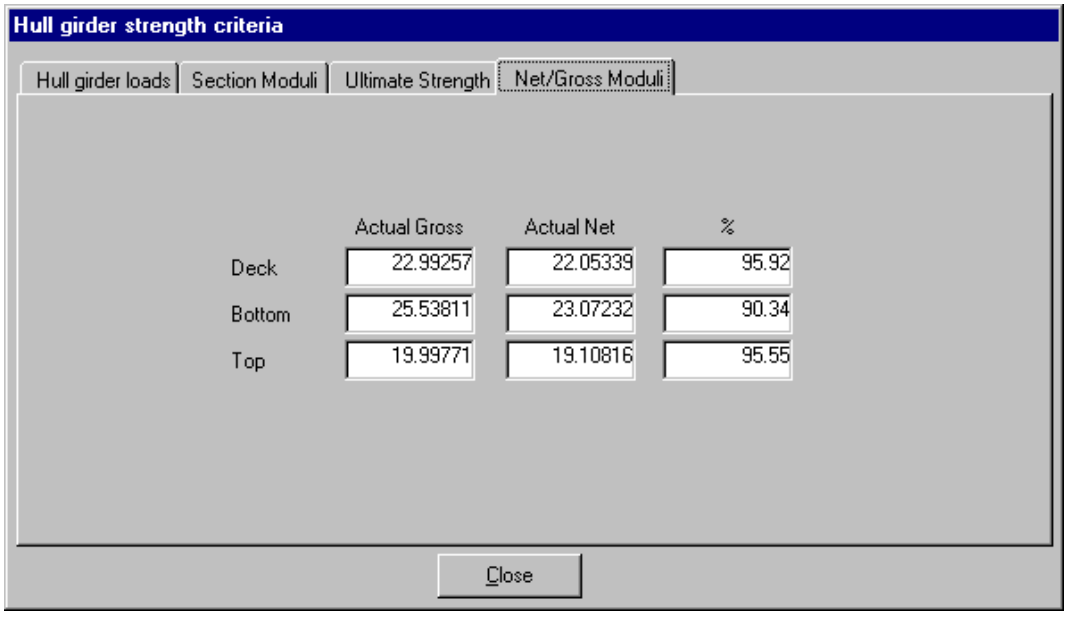


Figure 8: GLOBAL STRENGTH CRITERIA RESULT WINDOW (4)

Local stress calculations are based on the actual net scantling. But, if one of the actual net moduli is 90% less then its corresponding actual gross moduli, local stress calculations are based on 90% of the actual gross scantling.

Chapter 4 : STRAKES

Clicking on the *Strakes* button  or on *Strakes* on the *View* menu (Figure 19), you enter the *Strakes result* window:

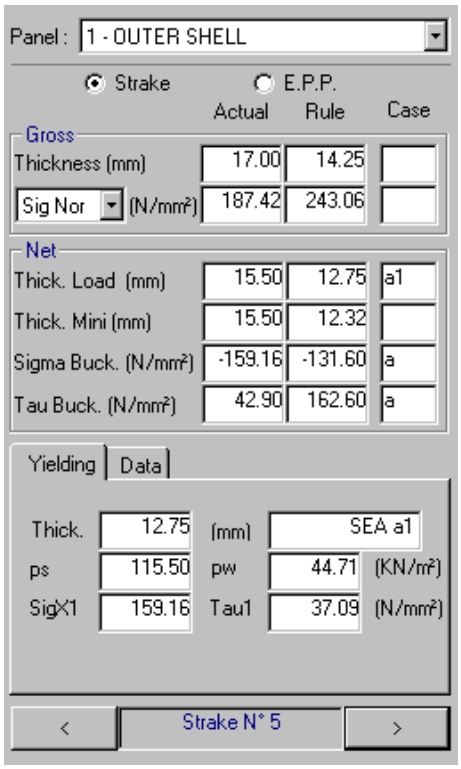


Figure 9: STRAKES RESULT WINDOW

4.1 SCANTLING CALCULATION

The purpose of this calculation is to check in a given section the actual scantling of strakes contributing to the longitudinal strength of the ship.

The window display the results at strake level or in more detailed way for each elementary plate panel forming the strake. To highlight the strake anomalies if the actual value of considered result is lower then rule value this latter become red.

4.2 SYMBOLS

Gross frame

	Actual	Rule
Thickness	Gross thickness of the strake.	Maximum of Rule thick. Load and Rule Thick. Mini added with the corrosion margin of the strake.
Sig Nor	Normal stress induced by vertical bending moments.	Rule normal stress induced by vertical bending moments.
Tau Nor	Shear stress induced by vertical bending moments.	Rule shear stress induced by vertical bending moments.
Sig Comb	Normal stress induced by torque and bending moments.	Rule normal stress induced by torque and bending moments.

Tau Comb	Shear stress induced by torque and bending moments.	Rule shear stress induced by torque and bending moments.

Net frame

	Actual	Rule
Thick. Load	Net thickness of the strake.	Thickness based on external or internal design pressure and on a stress factor depending on the overall bending stress. It is calculated on each E.P.P. considered by the program. The output value of Thick. Load is the maximum one.
Thick. Mini	Net thickness of the strake.	Minimum rule thickness. Maximum of the values calculated on each E.P.P.
Sigma Buck.	In plane hull girder compression normal stress. The output value is the one obtained on the E.P.P. where the ratio Rule/Actual is the harshest.	Critical buckling stress based on Euler stress. The output value is the one obtained on the E.P.P. where the ratio Rule/Actual is the harshest.
Tau Buck.	In plane hull girder shear stress. The output value is the one obtained on the E.P.P. where the ratio Rule/Actual is the harshest.	Critical buckling shear stress based on Euler stress. The output value is the one obtained on the E.P.P. where the ratio Rule/Actual is the harshest.

Yielding tab

This tab gives details of the Thick. Load result on the E.P.P. where it is maximized.

Thick	Thickness based on external or internal design pressure and on a stress factor depending on the overall bending stress.
Load reference	Code indicating the corresponding type of loading and load case. The possible items of loading are described hereafter.
Ps	Still water pressure.
Pw	Wave pressure.
SigX1	In-plane hull girder normal stress.
Tau1	In-plane hull girder shear stress.

Load references

SEA	Sea pressure
LIQ i	Liquid pressure
BULK i	Bulk pressure
SEALIQ I	Differential pressure due to sea and internal liquid cargo
SEABULK I	Differential pressure due to sea and internal bulk cargo
FLOOD I	Pressure in flooding condition
Slosh I	Sloshing pressure
Impact i	Impact pressure
Wheel	Wheel load
UniCarg	Dry uniform cargo pressure
AccDeck	Accomodation

For the code described hereabove the figure i mean: number of the compartment, the data of which are used in the calculation of pressure.

Chapter 5 : STIFFENERS

Clicking on the *Longitudinal stiffeners* button or on *Stiffeners* on the *View* menu (Figure 19), you enter the *Longitudinal stiffeners result* window:

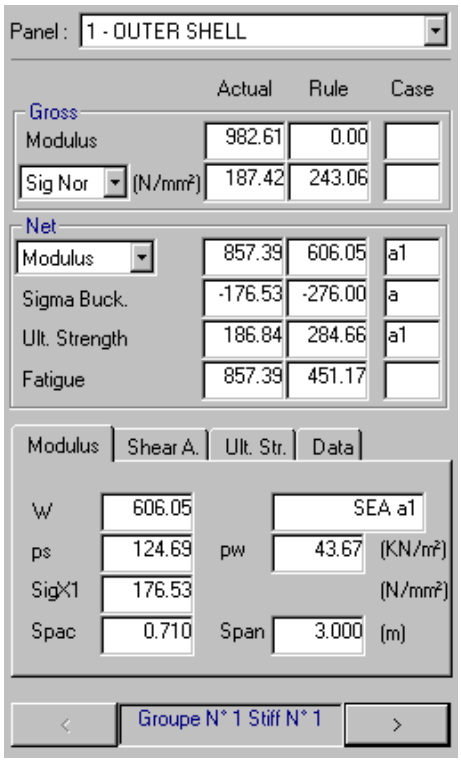


Figure 10: LONGITUDINAL STIFFENERS RESULT WINDOW

5.1 SCANTLING CALCULATION

The purpose of this calculation is to check in a given section the actual scantling of longitudinal stiffeners contributing to the longitudinal strength of the ship.

To highlight the stiffener anomalies if the actual value of considered result is lower then rule value this latter become red.

5.2 SYMBOLS

Gross frame

	Actual	Rule
Modulus	Modulus of the stiffener based on its gross scantling.	
Sig Nor	Normal stress induced by vertical bending moments.	Rule normal stress induced by vertical bending moments.
Tau Nor	Shear stress induced by vertical bending moments.	Rule shear stress induced by vertical bending moments.
Sig Comb	Normal stress induced by torque and bending moments.	Rule normal stress induced by torque and bending moments.
Tau Comb	Shear stress induced by torque and bending moments.	Rule shear stress induced by torque and bending moments.

Net frame

	Actual	Rule
Modulus	Modulus of the stiffener based on its net scantling.	Modulus based on external or internal design pressure and on a stress factor depending on the overall bending stress.
Shear Area	Shear area of the stiffener based on its net scantling.	Shear area based on external or internal design pressure and on a stress factor depending on the overall bending stress.
Thick. Mini	Net thickness of the stiffener web.	Minimum rule thickness.
Sigma Buck.	In plane hull girder compression normal stress. The output value is the one obtained for maximal ratio Rule/Actual.	Critical buckling stress based on Euler stress. The output value is the one obtained for maximal ratio Rule/Actual.
Ult. Strength	In plane hull girder compression normal stress. The output value is the one obtained for maximal ratio Rule/Actual.	Ultimate strength stress. The output value is the obtained for maximal ratio Rule/Actual.
Fat Modulus	Modulus of the stiffener based on its net scantling	Modulus based on external or internal design pressure range and on the overall bending stress range.
Fat Life		

Modulus tab

This tab gives details of the net modulus result.

W	Modulus based on external or internal design pressure and on a stress factor depending on the overall bending stress.
Load reference	Code indicating the corresponding type of loading and load case. The possible items of loading are described hereafter.
Ps	Still water pressure.
Pw	Wave pressure.
SigX1	In-plane hull girder normal stress.
Spac	Spacing of the stiffener.
Span	Span of the stiffener.

Shear A. tab

This tab gives details of the shear area result.

S Area	Shear area based on external or internal design pressure and on a stress factor depending on the overall bending stress.
Load reference	Code indicating the corresponding type of loading and load case. The possible items of loading are described hereafter.
Ps	Still water pressure.
Pw	Wave pressure.
Spac	Spacing of the stiffener.
Span	Span of the stiffener.

Ult. Str. tab

This tab gives details of the Ult. Strength result.

SigU	Ultimate strength stress. The output value is the obtained for maximal ratio Rule/Actual.
Load reference	Code indicating the corresponding type of loading and load case. The possible items of loading are described hereafter.
Ps	Still water pressure.
Pw	Wave pressure.
SigX1	In plane hull girder compression normal stress.
Spac	Spacing of the stiffener.
Span	Span of the stiffener.

Load references

SEA	Sea pressure
LIQ i	Liquid pressure

BULK i	Bulk pressure
SEALIQ I	Differential pressure due to sea and internal liquid cargo
SEABULK I	Differential pressure due to sea and internal bulk cargo
FLOOD I	Pressure in flooding condition
Slosh I	Sloshing pressure
Impact i	Impact pressure
Wheel	Wheel load
UniCarg	Dry uniform cargo pressure
AccDeck	Accomodation

For the code described hereabove the figure i mean: number of the compartment, the data of which are used in the calculation of pressure.

Chapter 6 : TRANSVERSE STIFFENERS

Clicking on the *Transverse stiffeners* button or on *Transverse stiffeners* on the *View* menu (Figure 19), you enter the *Transverse stiffeners result* window:

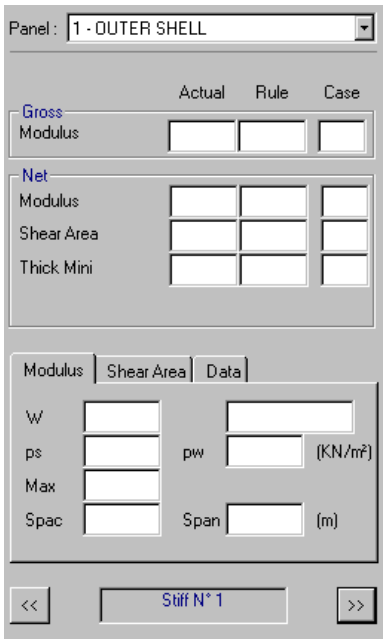


Figure 11: TRANSVERSE STIFFENERS RESULT WINDOW(1)

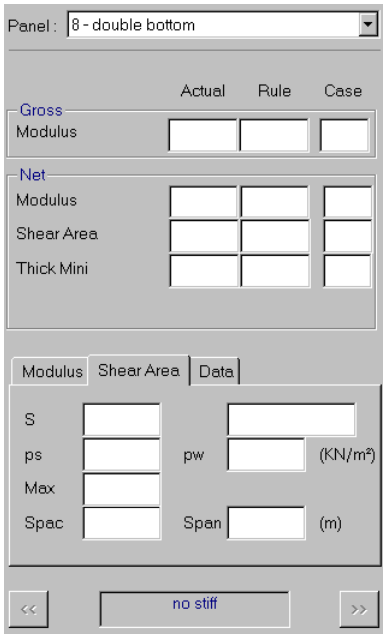


Figure 12: TRANSVERSE STIFFENERS RESULT WINDOW(2)

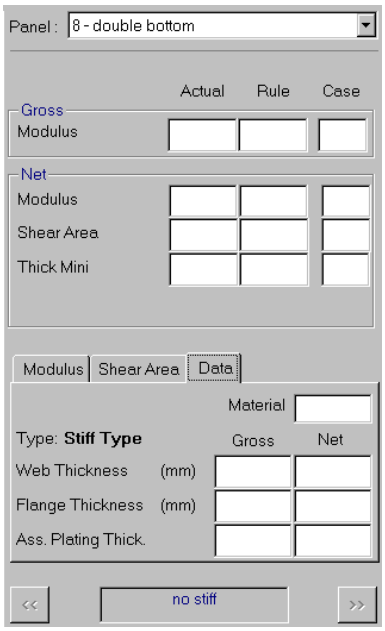


Figure 13: TRANSVERSE STIFFENERS RESULT WINDOW(3)

6.1 SYMBOLS

Gross frame

	Actual
Modulus	Modulus of the stiffener based on its gross scantling.

Net frame

	Actual	Rule
Modulus	Modulus of the stiffener based on its net scantling.	Modulus based on external or internal design pressure and on a stress factor depending on the overall bending stress.
Shear Area	Shear area of the stiffener based on its net scantling.	Shear area based on external or internal design pressure and on a stress factor depending on the overall bending stress.
Thick. Mini	Net thickness of the stiffener web.	Minimum rule thickness.

Modulus tab

This tab gives details of the net modulus result.

W	Modulus based on external or internal design pressure and on a stress factor depending on the overall bending stress.
Load reference	Code indicating the corresponding type of loading and load case. The possible items of loading are described hereafter.
Ps	Still water pressure.
Pw	Wave pressure.
Max	Code indicating the position along the stiffener where the ratio Rule/Actual is maximal. The possible items of position are described hereafter.
Spac	Spacing of the stiffener.
Span	Span of the stiffener.

Shear A. tab

This tab gives details of the shear area result.

S Area	Shear area based on external or internal design pressure and on a stress factor depending on the overall bending stress.
Load reference	Code indicating the corresponding type of loading and load case. The possible items of loading are described hereafter.
Max	Code indicating the position along the stiffener where the ratio Rule/Actual is maximal. The possible items of position are described hereafter.

Ps	Still water pressure.
Pw	Wave pressure.
Spac	Spacing of the stiffener.
Span	Span of the stiffener.

Load references

SEA	Sea pressure
LIQ i	Liquid pressure
BULK i	Bulk pressure
SEALIQ I	Differential pressure due to sea and internal liquid cargo
SEABULK I	Differential pressure due to sea and internal bulk cargo
FLOOD I	Pressure in flooding condition
Slosh I	Sloshing pressure
Impact i	Impact pressure
Wheel	Wheel load
UniCarg	Dry uniform cargo pressure
AccDeck	Accomodation

For the code described hereabove the figure i mean: number of the compartment, the data of which are used in the calculation of pressure.

Position references

Start	Start of the stiffener (panel direction)
End	End of the stiffener (panel direction)
Between	Point where the moment is maximal.

Chapter 7 : RENEWAL

7.1 STRAKES

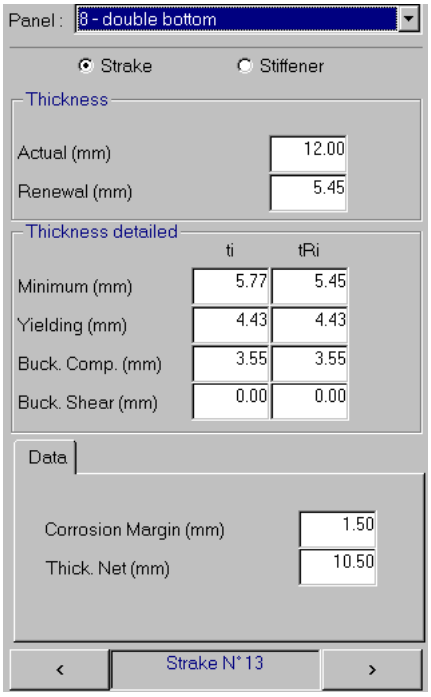


Figure 14 : STRAKE RENEWAL RESULT WINDOW

Thickness frame

Actual	Gross thickness of the strake.
Renewal	Maximum of tRi.

Thickness detailed frame

	ti	tRi
Minimum	Minimum net thickness.	Minimum renewal thickness.
Yielding	Thickness based on external or internal design pressure and on a stress factor depending on the overall bending stress.	Renewal thickness of plating subjected to lateral pressure and wheel loads.
Buck. Comp.	Compression buckling net thickness.	Compression buckling renewal thickness.
Buck. Shear	Shear buckling net thickness.	Shear buckling renewal thickness.

7.2 LONGITUDINAL STIFFENERS

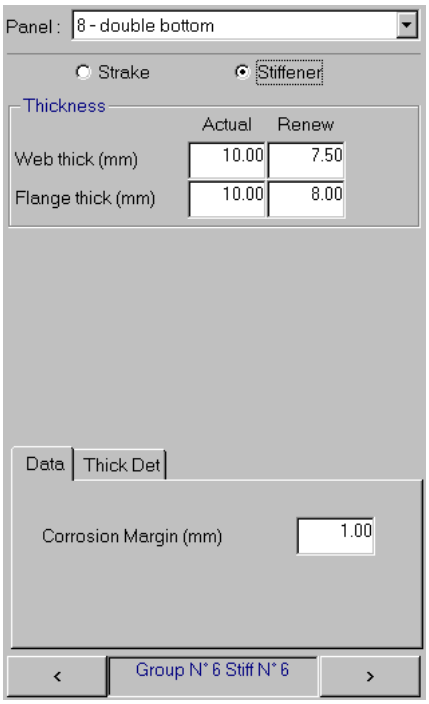


Figure 15 : STIFFENER RENEWAL RESULT WINDOW

Thickness frame

	Actual	Renew
Web thick	Actual web thickness	Renewal web thickness.
Flange thick	Actual flange thickness	Renewal flange thickness.

Thick Dat tab

	Web	Flange
%		
Buck		

Chapter 8 : STRESSES

Clicking on the *Stresses* button  or on *Stresses* on the *View* menu (Figure 19), you enter the *Stresses result window*:

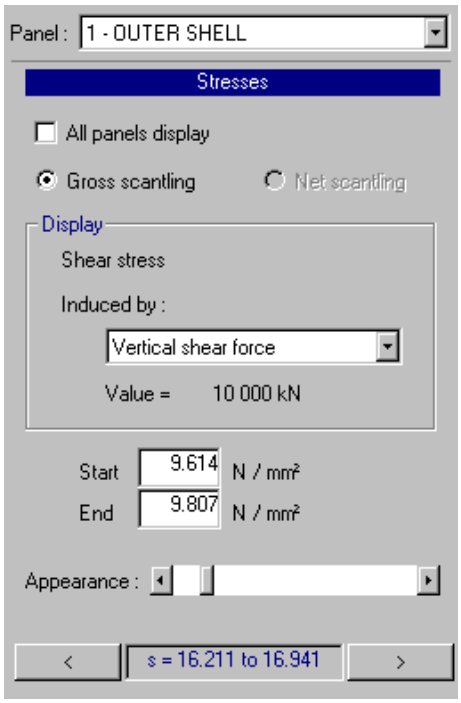


Figure 16 : STRESSES RESULT WINDOW

The distribution of stresses is calculated for unitary internal forces using the geometry of the section. The actual stresses are obtained by multiplying the unit stresses by the value of the internal forces stored for each loading conditions.

The stress display are induced by:

- Vertical Shear Force;
- Horizontal Shear Force;
- Saint Venant torque;
- Warping torque;
- Warping moment.

When you click in the section view the window shows the value for the selected segment in *Start* and *End* fields.

Chapter 9 : RATIO

Clicking on the *Ratio* button  or on *Ratio* on the *View* menu (Figure 19), you enter the *Ratio result window* :

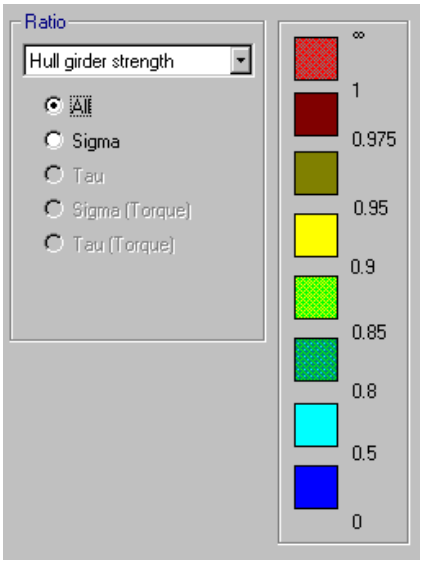


Figure 17 : RATIO RESULT WINDOW

This window allows to select one or several types of ratio from the following list :

- Hull girder strength.
- Local strength – Strakes.
- Local strength – Stiffeners.
- Local strength – Transverse stiffeners.
- Corrosion.

Hereafter the different type of ratio available for each item.

Hull girder strength

- All
- Sigma

Local strength – Strakes

- All
- Thickness Load
- Thickness Mini
- Buckling – Normal stress
- Buckling – Shear stress

Local strength – Stiffeners

- All
- Thickness Load
- Net modulus
- Net shear area
- Buckling – Normal stress
- Ultimate strength
- Thickness Mini
- Fatigue

Local strength – Transverse stiffeners

- All
- Net modulus
- Net shear area
- Thickness Mini

Corrosion

- Corrosion addition

Chapter 10 : MAIN FEATURES

10.1 MENU

File Menu
It allows to launch the calculation, to print and to quit MARSRULE.

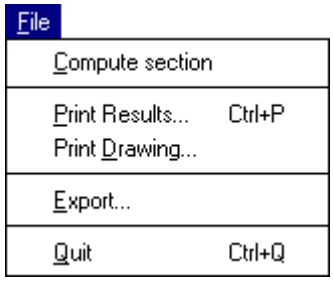


Figure 18 : FILE MENU

Item	Use	Shortcut
Compute section	allows to set calculation options (see 1.3).	
Print Results…	prints the calculation results (see 1.4.1).	Ctrl + P
Print Drawing…	Prints a drawing of a section (see 1.4.2).	
Export…		
Quit	Quits MARSRULE to return to MARSHELL.	Ctrl + Q

View menu
It gathers the results from calculation.

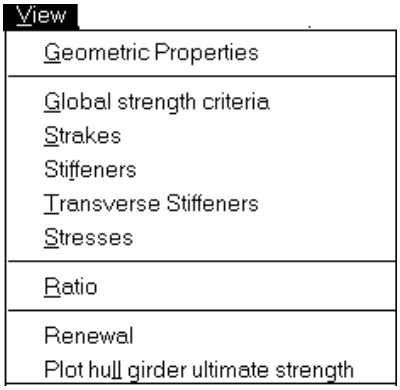


Figure 19 : VIEW MENU

Item	Use	Shortcut
Geometric properties	displays the geometric properties results window (see 2).	
Global strength criteria	displays the Hull girder strength criteria results window (see 3).	
Strakes	displays the Strake results window (see 4).	
Stiffeners	displays the Stiffener results window (see 5).	
Transverse stiffeners	displays the Transverse stiffener results window (see 6).	
Stresses	displays the Stresses results window (see 7).	
Ratio	displays the Ratio results window (see 8).	
Plot hull girder ultimate strength	displays the Hull girder ultimate strength results window (see 9).	

Check menu
It includes checking tools.

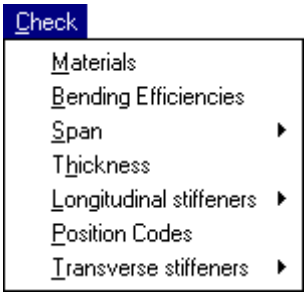


Figure 20 : CHECK MENU

Item	Use	Shortcut
Materials	displays the strakes and stiffeners with a different color for each material.	
Bending Efficiencies	displays the strakes and stiffeners with a different color for each bending efficiency.	
Span	(see Figure 21)	
Thickness	displays the strakes with a different color for each thickness.	
Longitudinal stiffeners	(see Figure 22)	
Positions Codes	displays the segments with a different color for each position code.	
Transverse stiffeners	(see Figure 23)	

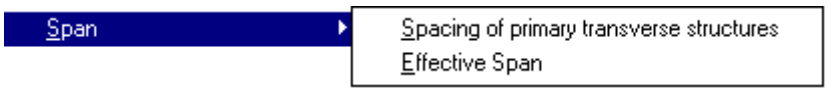


Figure 21 : SPAN SUB-MENU

Item	Use	Shortcut
Spacing of primary transverse structures	displays the strakes with a different color for each span.	
Effective Span	displays the strakes and stiffeners with a different color for each span.	

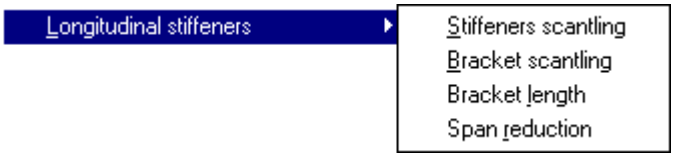


Figure 22 : LONGITUDINAL STIFFENERS SUB-MENU

Item	Use	Shortcut
Stiffeners scantling	displays the stiffeners with a different color for each stiffener scantling.	
Bracket scantling	displays the brackets with a different color for each bracket scantling.	
Bracket length	displays the brackets length.	
Span reduction	displays the span reduction.	

Figure 23 : TRANSVERSE STIFFENERS SUB-MENU

Item	Use	Shortcut
Stiffeners scantling	displays the transverse stiffening zones with a different color for each scantling of those transverse.	
Stiffeners spacing	displays the transverse stiffening zones with a different color for each spacing of those transverse.	

***Tools* menu :**

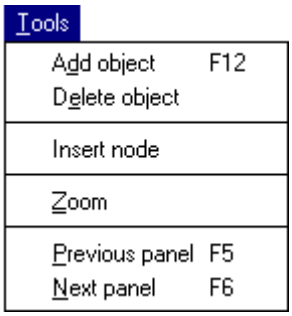


Figure 24 : TOOLS MENU

Item	Use	Shortcut
Display always…	allows to set drawing preferences.	
Preferences…	displays a set up window for the drawing preferences on the screen or a printer.	
Zoom	allows to zoom in (see 10.3).	
Refresh section	refreshes the screen in case of display anomalies.	**F9**
Previous panel	changes the current panel to the previous one.	F5
Next panel	changes the current panel to the next one.	F6
Copy to clipboard	Allows to paste the section drawing in any other application	Ctrl + C
Debug Occurs on panel…	displays occurrence attributes for one panel in the section.	
View File EPP	displays yielding values for all the Elementary Plate Panel in the section.	

View File Stiff	displays modulus values for all the stiffeners in the section.	
View Ultimate Strength	displays Ultimate strength values for all the stiffeners in the section.	
Debug EPP on panel…	displays Elementary Plate Panel attributes for one panel in the section.	

10.2 MENUS PRINTING

10.2.1 Printing data

Clicking on  or on Print Results… on the File menu (Figure 18) or pressing Ctrl + P, you enter the Print Results management window :

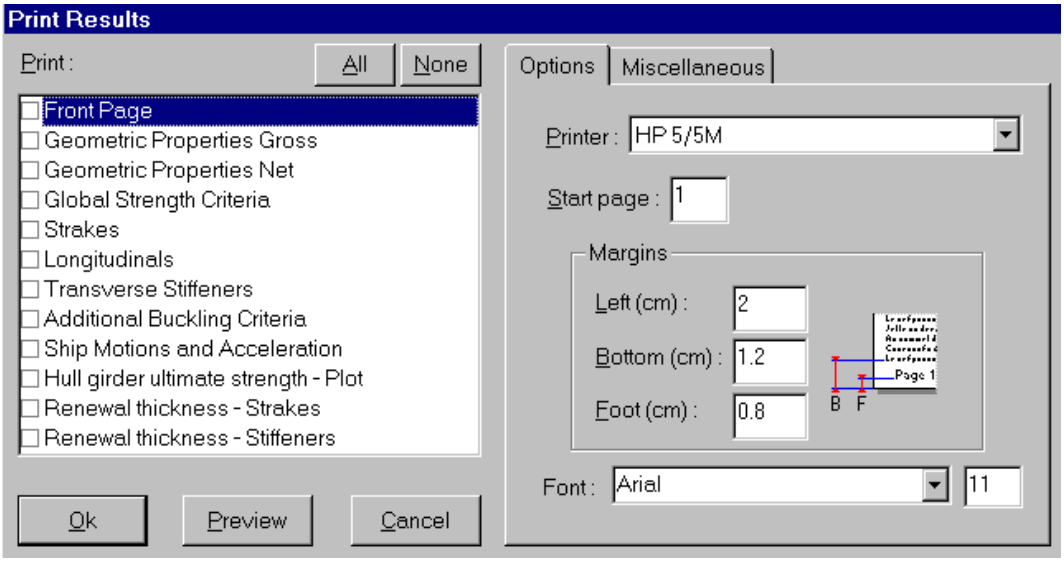


Figure 25 : PRINT RESULTS MANAGEMENT WINDOW

This window allows you to select what you want to print. The *All* (*None)* button selects (deselects) all the items of the list.

Front page produces the cover page of a report.

10.2.2 Printing drawing

Clicking on *Print Drawing…* on the *File* menu (Figure 18), you enter the *Print Drawing management* window :

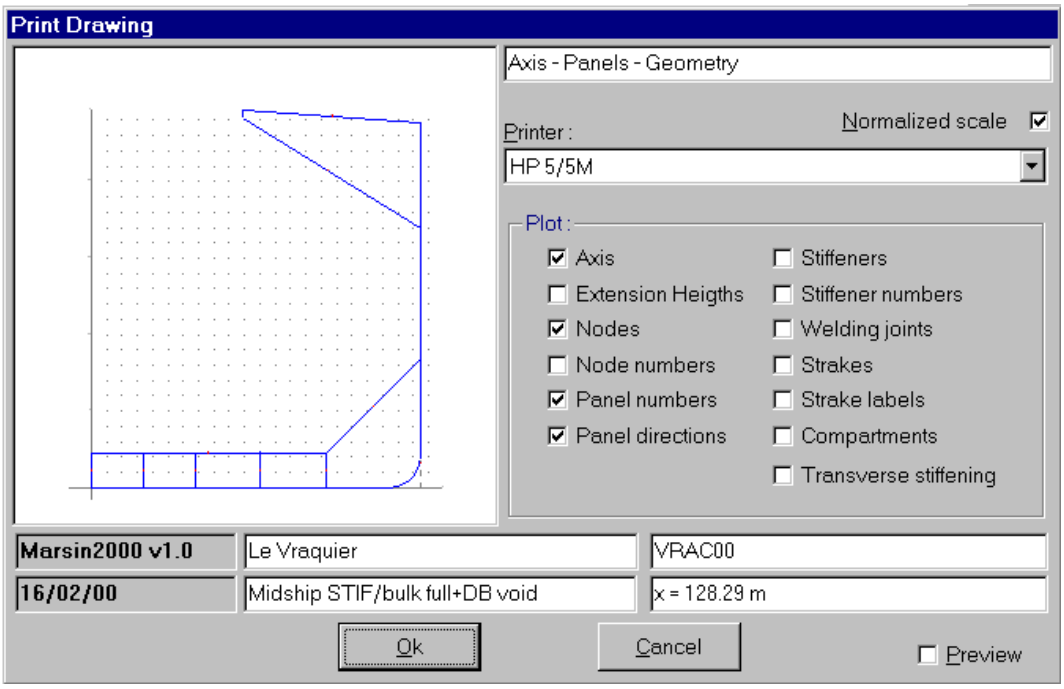


Figure 26 : PRINT DRAWING MANAGEMENT WINDOW

This window allows you to select which item will be printed on the item.

The *Normalized scale* check box will make the drawing printed with a regular scale (e.g. 1/50, 1/100,…).

10.3 ZOOM

It is possible to Zoom in on or out of the Section view thanks to the *Zoom* Toolbar:

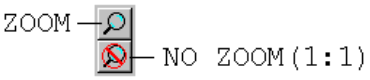


Figure 27 : ZOOM TOOLBAR

A first click on the *Zoom* button (Figure 27) or on *Zoom* on the *Tools* menu (Figure 24) changes the cursor in  and allows you to zoom in the section view by simple click on it.

A second click on the *Zoom* button (Figure 27) or on *Zoom* on the *Tools* menu (Figure 24) changes back the cursor in  and allows you to work on your zoomed section view.

To zoom out of the section view, you can :

- click on the *No Zoom (1:1)* button (Figure 27) to bring back the view to the initial size.
- right-click on the section view when the *Zoom* button is down.

Table of Figure

MARS2000
User's guide

Booklet 4

DEFINITION OF A BULKHEAD ARRANGEMENT

February 2000 /1
Dev/UG/MARS2000/04

MARINE DIVISION
DEVELOPMENT DEPARTMENT
17 bis, place des Reflets – La Défense 2
92077 Paris – La Défense Cedex
Tel.: + 33 (0)1 42 91 52 91
Fax: + 33 (0)1 42 91 52 98

TABLE OF CONTENTS

Chapter 1 : GENERAL COMMENTS

1.1 INTRODUCTION

BhaIn allows the input of any bulkhead arrangement along the ship length. A bulkhead arrangement is a set of bulkheads located at the same longitudinal position.

The bulkhead is described by:

- Its geometry and scantling.
- Primary and ordinary stiffeners.
- Forward and aftward compartments.

1.2 A GOOD WAY TO CREATE A BULKHEAD ARRANGEMENT

When creating a new bulkhead arrangement, the following window appears:

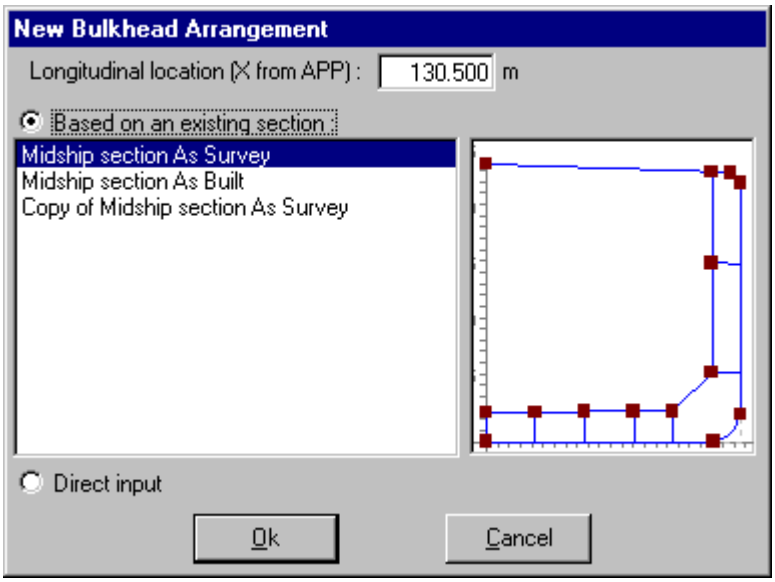


Figure 1 : BULKHEAD ARRANGEMENT CREATION

There are two ways to modelise a bulkhead arrangement:

- Direct input
- From a section

The second way (from a section) is the fastest: it automatically detects the bulkhead geometry based on the compartments. It also initializes the compartment data.

1.3 BHAIN INTERFACE

The module allowing to input the data of a bulkhead is organized around the following application:

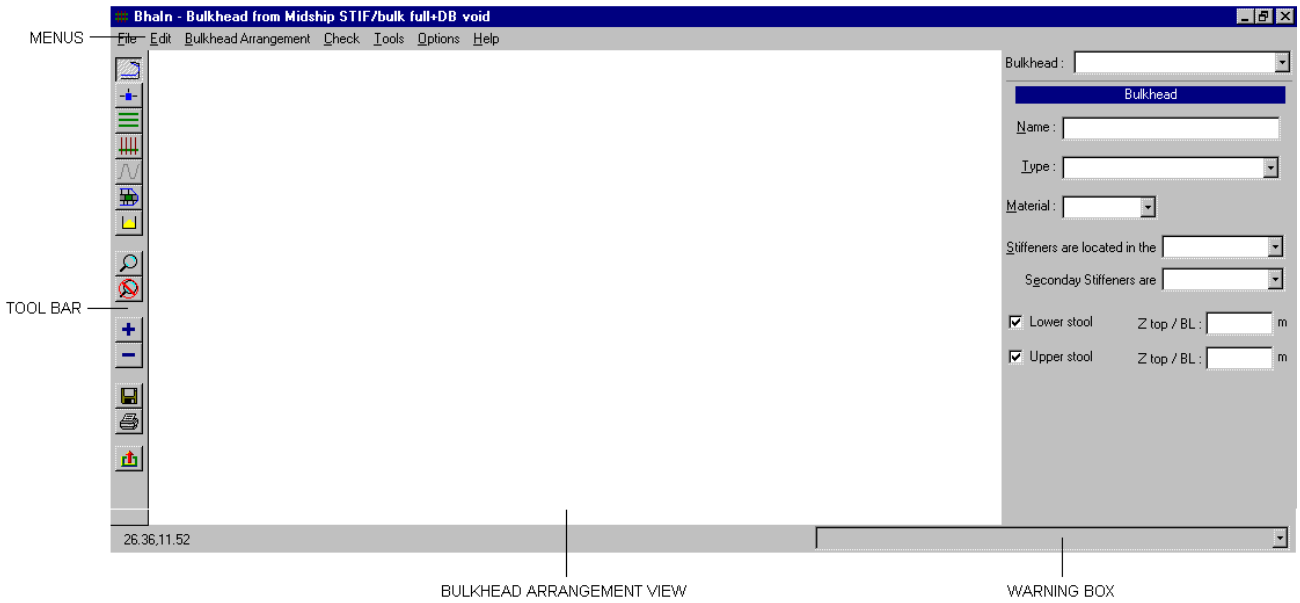


Figure 2 :BHAIN

Warning Box : displays warning message when BHAIN detects incoherence in the bulkhead definition.

Bulkhead arrangement view : displays a view of the bulkhead arangement.

Chapter 2 : MAIN SECTION DATA

On the *Bulkhead Arrangement* menu, click on *Main Data* (Figure 24) to display the Main Bulkhead Arrangement Data Window.

2.1 MAIN DATA

The first tab of the main bulkhead arrangement data window allows to define a identification of bulkhead, the X longitudinal location of the bulkhead and also the type of section used to define the bulkhead.

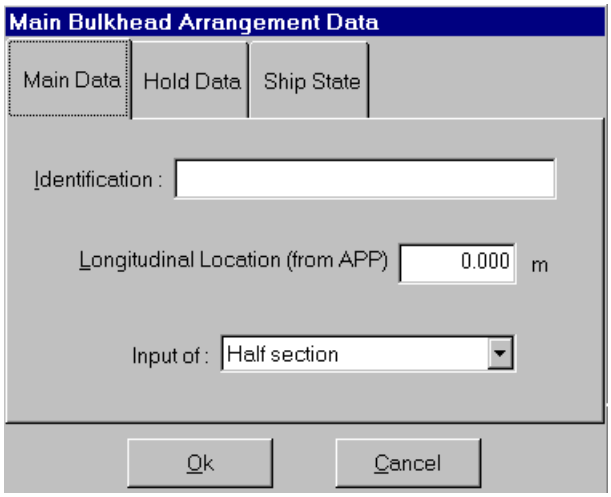


Figure 3 : MAIN DATA WINDOW

2.2 HOLD DATA

The second tab of the main bulkhead arrangement data window allows defining the hold data. These values have to be filled to calculate bulk pressure.

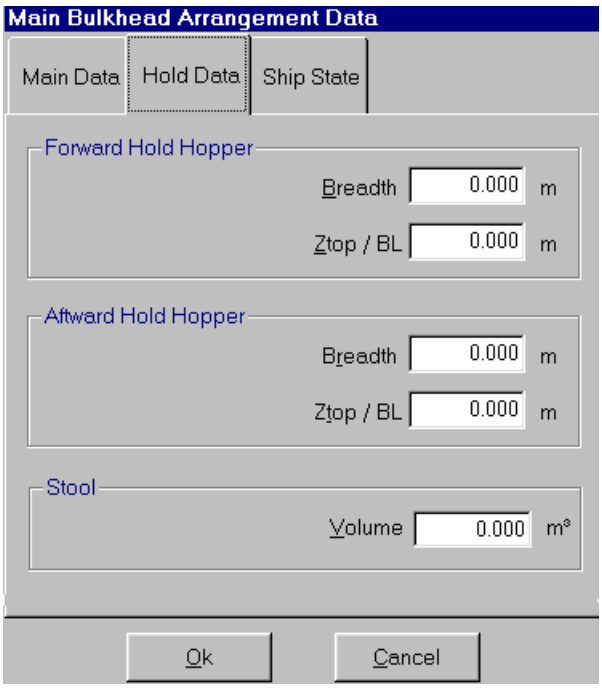


Figure 4 : HOLD DATA WINDOW

2.3 SHIP STATE

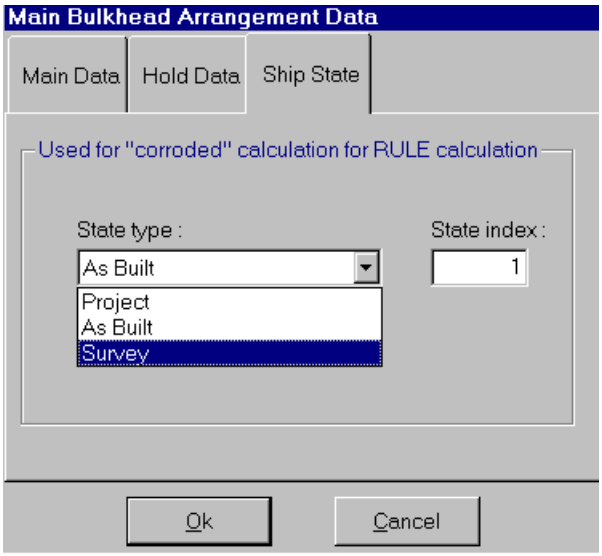


Figure 5 : SHIP STATE WINDOW

Chapter 3 : BULKHEAD GEOMETRY

3.1 INPUT SEQUENCE

There are six main sequences to define the geometry of a bulkhead:

- *Nodes*
- *Primary stiffeners*
- *Secondary stiffeners*
- *Corrugation*
- *Strakes*

These different sequences of the geometry description may be accessed by the following toolbar as follows:

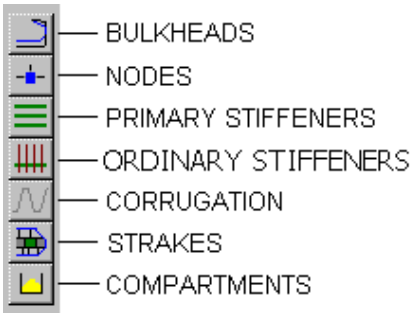


Figure 6 : BHAIN TOOLBAR

Bulkheads: allows defining the bulkhead name and characteristics.

Nodes: allows defining the geometry of the bulkhead by a succession of segments.

Primary Stiffeners: allows locating all the primary stiffeners.

Ordinary Stiffeners: allows locating all the ordinary stiffeners and to define their scantlings in case of a single or double skin bulkhead.

Corrugation: allows defining geometry of a corrugated bulkhead.

Strakes: allows locating strake bands and characteristics.

Compartments: allows defining forward and aftward compartments.

All these input sequences are bulkhead-oriented. It means that, inside an input sequence, the data are available bulkhead by bulkhead.

When an input sequence is selected in the toolbar, the program displays generally the data corresponding to the current bulkhead. The current bulkhead is the last selected bulkhead.

They are four ways to move from one bulkhead to another:

- a direct click on the desired bulkhead in the bulkhead arrangement view,
- the *Next Bulkhead* and *Previous Bulkhead* items on the *Tools* menu (Figure 24),

- the F6 key to jump to the next bulkhead or the F5 key to jump to the previous bulkhead,
- using the bulkhead list placed on the right-hand corner of BHAIN :

Figure 7 : BULKHEAD LIST

3.2 CREATION AND DELETION OF DATA

In each input sequence, you can create or delete data:

- Bulkheads : creation or deletion of a bulkhead
- Nodes : creation or deletion of a segment
- Primary stiffeners : creation or deletion of a primary stiffener
- Ordinary stiffeners : creation or deletion of a secondary stiffener
- Strakes : creation or deletion of a strake band

Each object has to be created or deleted using the following toolbar:

Figure 8 : CREATION-DELETION TOOLBAR

For example, if you want to create a new bulkhead

- click on the *bulkhead* button (Figure 6),
  the *Bulkhead management window* (Figure 9) is displayed,
- click on the *creation* button (Figure 8).

Inversely, if you want to delete an existing panel,

- click on the *bulkhead* button (Figure 6),
- select the bulkhead you want to delete,
- click on the *deletion* button (Figure 8).

To create data, it is also possible to hit the F12 key instead of clicking the *creation* button.
Another way to create or delete objects is to use *Create object* or *Delete object* on the *Tools* menu (Figure 24)

3.3 BULKHEAD DATA

Clicking on the bulkhead button  or on Bulkhead on the Bulkhead arrangement menu (Figure 24), you enter the Bulkhead management window:

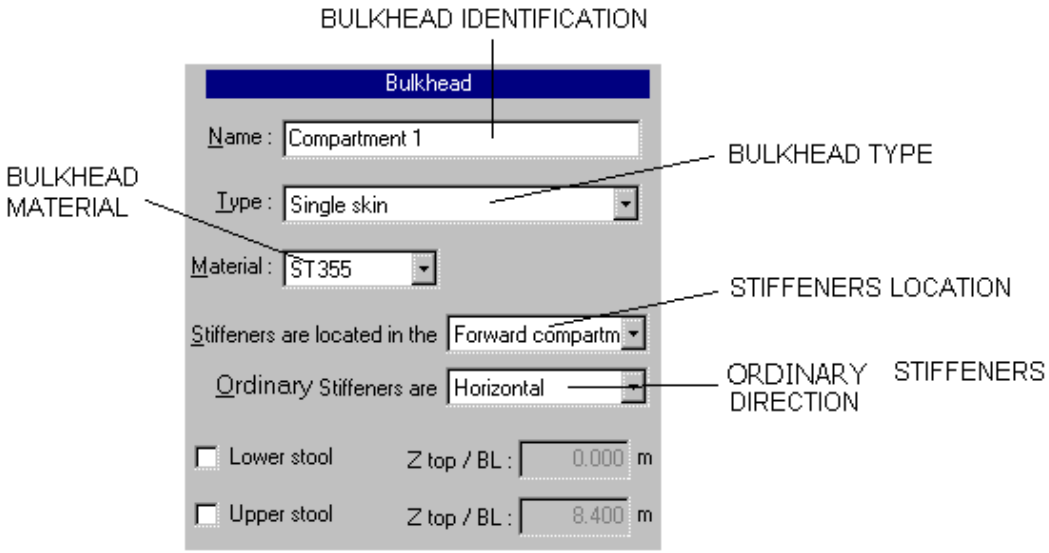


Figure 9 : BULKHEAD MANAGEMENT WINDOW

In this window, you can create or delete bulkheads using the *creation* or the *deletion* buttons (Figure 8). Identification, a type and material define each bulkhead. Moreover you can enter the stiffeners location as regards compartments near to bulkhead and the secondary stiffeners direction.

Bulkhead Identification: up to 40 characters.
The bulkhead has to be clearly identified. This identification will appear in input sequences and in output of data and results.

Type: single skin, double skin or corrugated.

Material: material of bulkhead.

Stiffeners are located in the: stiffeners location as regards compartments near to bulkhead.

Ordinary Stiffeners are: ordinary stiffeners direction.

3.4 NODE BUTTON : GEOMETRY OF THE BULKHEAD

3.4.1 Nodes and segments

A bulkhead contour is made of contiguous segments of different geometry (straight or circular line).
Each segment is described by:

- its ending node
- its type of curve.

First node of a bulkhead

Clicking on the node button  or on Nodes on the Bulkhead arrangement menu (Figure 24), you enter the Node management window where these inputs are performed:

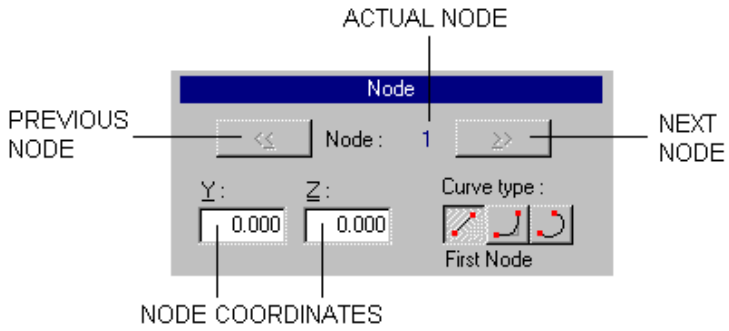


Figure 10 : NODE MANAGEMENT WINDOW

Node creation

The *creation* button  allows you to create a node extending the bulkhead contour.

The *deletion* button  is used to delete nodes.
It is also possible to insert a node between two existing nodes by clicking on the *Tools…* menu and selecting *Insert node* (Figure 24).

Next node - Previous node: Those two buttons allow to navigate node by node within a bulkhead. A direct click on the desired node in the bulkhead arrangement view is also possible.

Node characteristics
Coordinates of the node (in m): The Y and Z coordinates of the current node.

Segment characteristics
The characteristics of the segment between the current node and the previous one consist in:
Curve Type: it specifies the type of the segment.

 : straight line

 : tangent arc (see 3.4.2.1)

 : arc (see 3.4.2.2)

3.4.2 Circles

There is two ways to define circle:

- tangent arc button
- arc button

3.4.2.1 Tangent arc

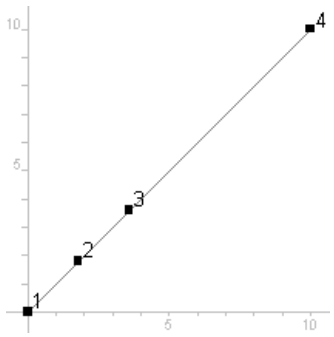
The *Tangent Arc* button is a way of defining circular segment tangent to both segments enclosing it. Therefore you have to create the nodes for the three segments (the circular one end its adjacent).

Example of input of bilge with a flat bottom

For a rise of keel, you enter:

Node	Y Coordinate	Z Coordinate	Click On
1	0.	0.	+ to create a node.
2	-	-	+ to create a node.
3	-	-	+ to create a node.
4	10.	10.	

You should obtained this kind of bulkhead:



Click on segment 2 (node 3) and hit on the *Arc Tangent* button (Figure 10).The window here after is displayed on screen :

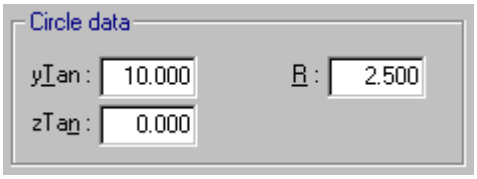
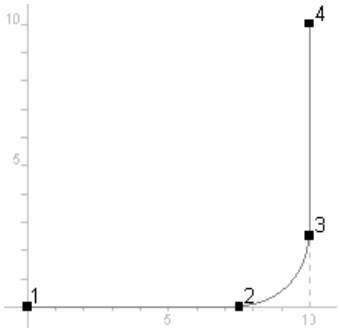


Figure 11 : TANGENT ARC DATA

R (in m): Radius of the circle

YTan and *ZTan* (in m) : Y and Z coordinates of the tangent intersection of the enclosing segments.

The coordinates of the first and the last nodes on this segment are automatically calculated and cannot be changed. The result is:

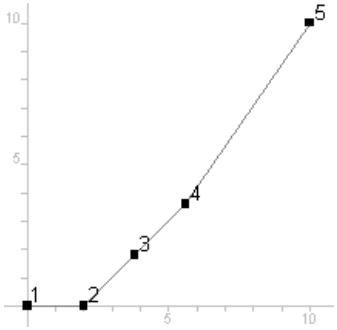


Example of input of bilge with rise of keel

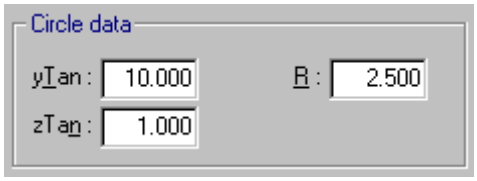
For example, for a rise of keel, you enter:

Node	Y Coordinate	Z Coordinate	Click On
1	0.	0.	+ to create a node.
2	2.	0.	+ to create a node.
3	-	-	+ to create a node.
4	-	-	+ to create a node.
5	10.	10.	

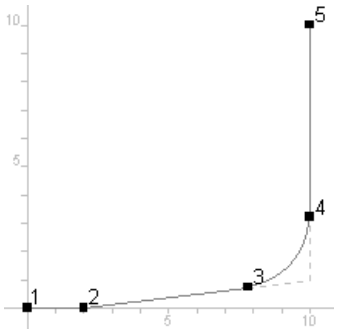
You should obtained this kind of section:



Click on segment 3 (node 4) and hit on the *Arc Tangent* button (Figure 10). The window here after is displayed on screen:



The result is:



3.4.2.2 Arc

The *Arc* button can be used to define any circular segment.
The coordinates of the first and the last nodes on this segment have to be input.
The definition of an arc is completed by means of the window here after. It is displayed on screen, hitting arc button when the current node is the last node of the circle

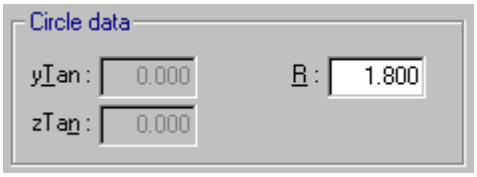
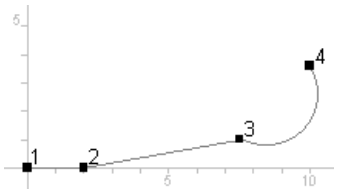


Figure 19 : ARC DATA

R (in m) : Radius of the circle.

With this method, you can define any kind of circular segment. For example:



3.5 PRIMARY STIFFENER

All the primary stiffeners existing on the bulkhead have to be defined by their position and their coordinate. They are automatically numbered, starting at number 1 for each bulkhead.

Clicking on the *Primary stiffeners* button (Figure 6) or on *Primary stiffeners* on the *Bulkhead Arrangement* menu (Figure 24), you enter the *Primary stiffener management window* where these inputs are performed:

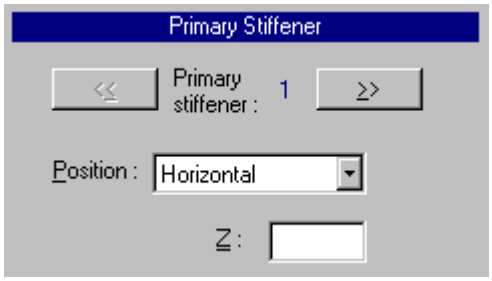


Figure 12 : PRIMARY STIFFENER MANAGEMENT WINDOW

The *creation* and the *deletion* buttons (Figure 8) allow you to create or delete primary stiffeners.

Position: horizontal or vertical.

To select a primary stiffener, click on a primary stiffener in the bulkhead arrangement view or in the previous/next button.

Z/Y (in m): coordinate of primary stiffener.

3.6 SECONDARY STIFFENER

3.6.1 Location

Clicking on the *Secondary Stiffener* button (Figure 6) or on *Secondary Stiffeners* on the *Bulkhead Arrangement* menu (Figure 24), you enter the *Secondary Stiffener management window* where these inputs are performed:

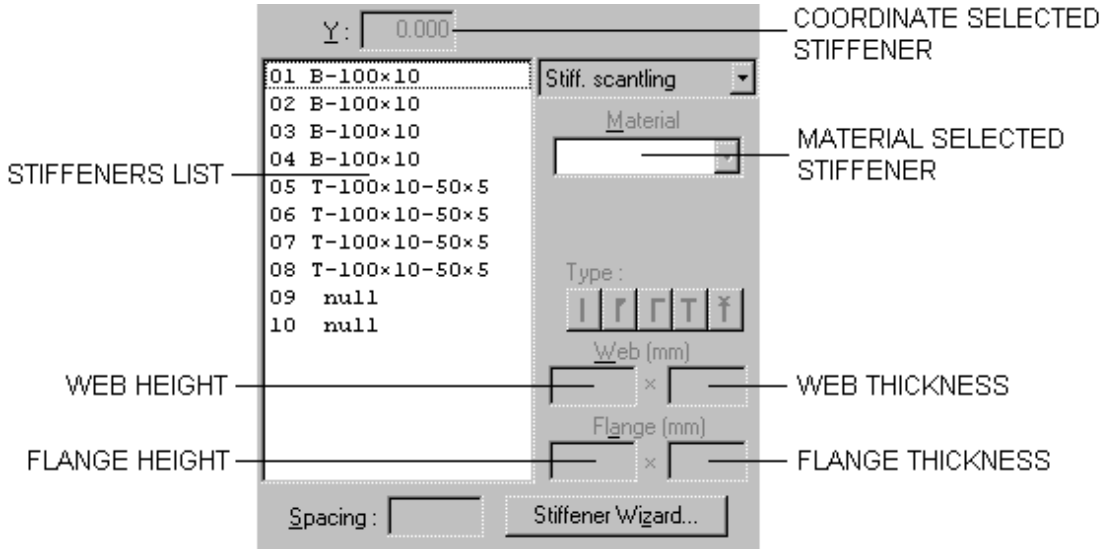


Figure 13 : SECONDARY STIFFENER MANAGEMENT WINDOW

The location is done by defining a coordinate of stiffener. The *creation* and the *deletion* buttons (Figure 6) or the Stiffener Wizard button (see 3.6.2) allow you to create or delete stiffener.

Stiffener List: Display of the stiffeners defined on the current bulkhead and of their scantling.
The list shows: the stiffener number and its scantling.
The stiffeners selected are the current stiffeners.

Selection of one stiffener
click the stiffener in the *Stiffener List* or in the Bulkhead Arrangement view.

Selection of stiffeners that are next to each other
select the first stiffener you want to select; hold down the SHIFT key and click the last stiffener you want to select.

select in the list the first stiffener you want to select, hold down the mouse button and drag the selection to the last stiffener you want.

Selection of stiffeners that are not next to each other
hold down the CTRL key, and then click each stiffener you want to select.

Material: allows changing the current stiffener material which, by default, is the supporting bulkhead one.

Spacing (in m): spacing of the selected stiffeners.

3.6.2 Stiffener Wizard

The stiffener wizard is the best way to easily define a group of stiffeners regularly spaced.

Clicking on the *Stiffener Wizard* button you enter the *Secondary stiffeners creation window* where these inputs are performed:

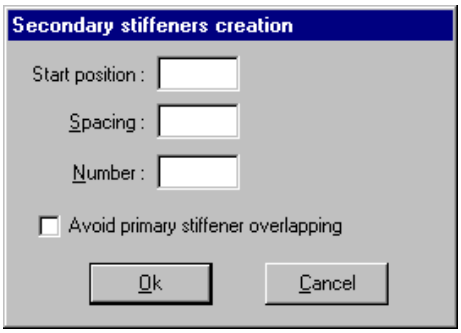


Figure 14 : SECONDARY STIFFENERS CREATION WINDOW

Start position (in m): coordinate of first stiffener.

Spacing (in m): spacing of considered stiffeners.

Number: number of considered stiffeners.

3.6.3 Scantling

Scantling tab
Stiffener Type: allows to select the stiffener type:

-  : flat stiffener
-  : bulb stiffener
-  : angle stiffener
-  : T-bar stiffener

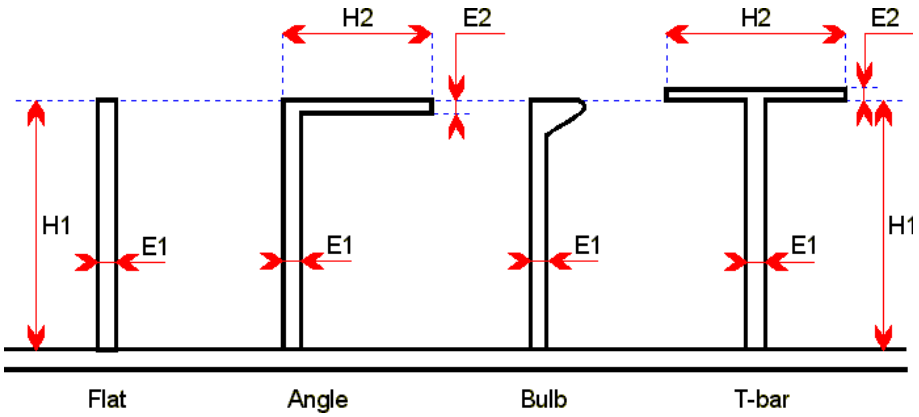
  In case of flat or bulb type, flange characteristics aren't available.
  If none type is chosen, the stiffener is considered as a NULL type which is defined in 3.6.4.

Web Height (in mm): Height of stiffener web (H1).

Web Thickness (in mm): Thickness of stiffener web (E1).

Flange Height (in mm): Width of stiffener flange (H2).

Flange Thickness (in mm): Thickness of stiffener flange (E2).



3.6.4 Null type

A stiffener with NULL type has no effect on the geometric characteristics (areas, inertia, moduli) of the bulkhead.

3.7 CORRUGATION

Choosing a type corrugated for a bulkhead the corrugation button  and *Corrugated* item on the *Bulkhead Arrangement* menu (Figure 24) become active; clicking on one of these you enter the *Corrugation management window* where these inputs are performed:

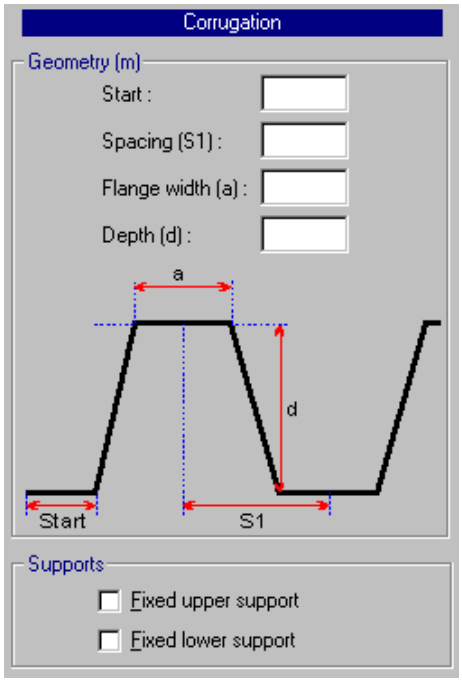


Figure 27 : CORRUGATION MANAGEMENT WINDOW

3.8 STRAKES

Clicking on the *Strakes* button (Figure 6) or on *Strakes* on the *Bulkhead Arrangement* menu (Figure 24), you enter the *Strake band management window* where these inputs are performed:

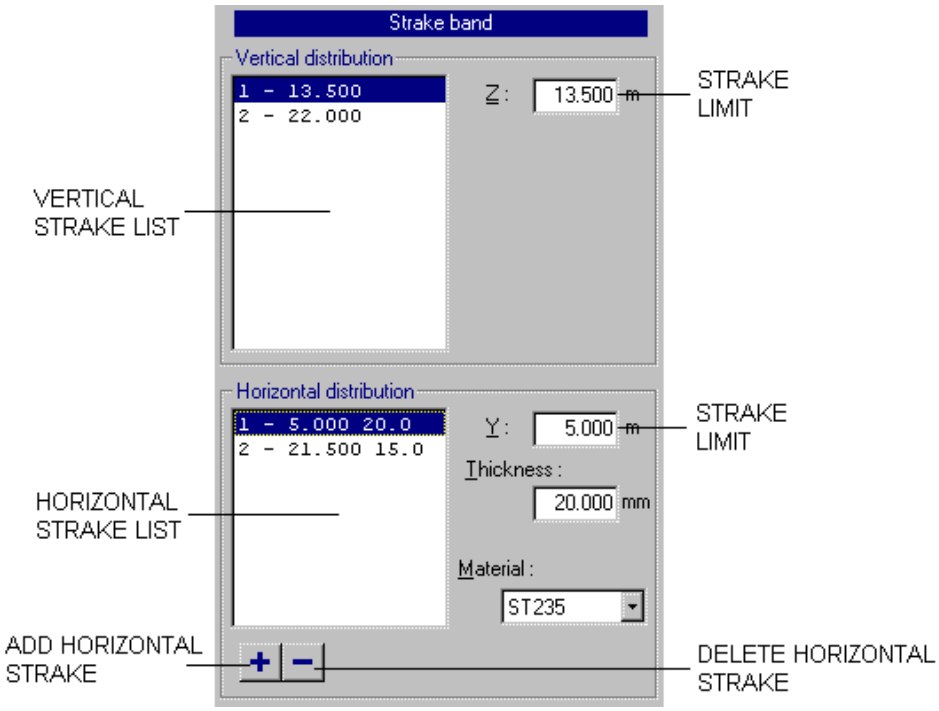


Figure 15 : STRAKE BAND MANAGEMENT WINDOW

During a first input, there is a strake automatically created on each bulkhead with width equal to the width of the bulkhead.

The *creation* and the *deletion* buttons (Figure 6) in the Tool bar allow you to create or delete vertical strakes.

To create or delete horizontal strakes you have to use the *creation* and the *deletion* buttons located in *Horizontal distribution* frame.

Creating a strake means to divide the strake selected in the strake list (Figure 29) into two strakes of same characteristics, thickness and material, and with width equal to the half width.

Vertical Strake List: display of the number and limit of the strakes defined on the current bulkhead. The strake selected is the current strake. To select a strake, click on a strake in the list or in the Bulkhead Arrangement view.

Horizontal Strake List: display of the number, limit and thickness of the strakes defined on the current bulkhead. The strake selected is the current strake

Thickness (in mm): thickness of the current strake.

Material: material of the current strake.

3.9 COMPARTMENTS

Clicking on the *compartment* button (Figure 13) or on Forward and aftward *compartments* on the *Bulkhead Arrangement* menu (Figure 24), you enter the *Compartments management window*:

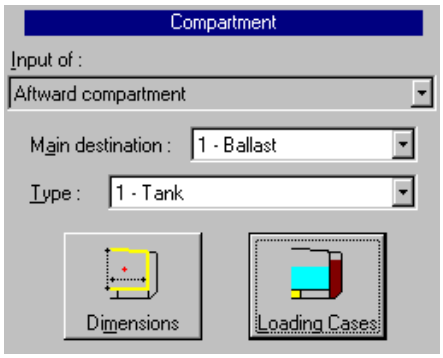


Figure 16 : COMPARTMENTS MANAGEMENT WINDOW

In this window, you can define the data for forward and aftward compartments.

Main destination: main load type.

Type: choose a compartment type in the list.
The possible types are:

1	Tank
2	Double bottom, skin
3	Dry compartment
4	Engine room
5	Boiler compartment
6	Tunnel
7	Hopper well 1
8	Hopper well 2

3.9.1 Dimensions

Clicking on the *Dimensions* button (Figure 16), the compartment-dimension window is displayed:

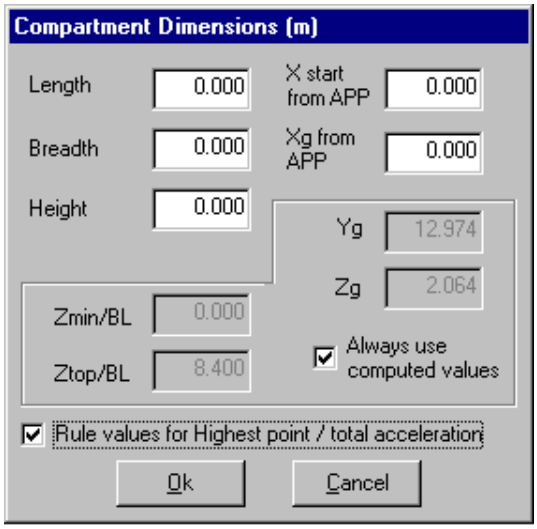


Figure 17 : COMPARTMENT-DIMENSION WINDOW

Length (in m): compartment length.

Breadth (in m): compartment breadth.

Height (in m): compartment height.

X start (in m): longitudinal location of start compartment from APP.

Xg (in m): longitudinal location of compartment center of gravity from APP.

The following data may be obtained by calculation clicking on *Always use compute values* check or by direct input:

Yg (in m): center of gravity (Center line).

Zg (in m): center of gravity above base line.

Zmin/BL (in m): min of compartment from base line.

Ztop/BL (in m): top of compartment from base line.

If you don't want use the Rule values for highest point / total acceleration click in the related check. The window become:

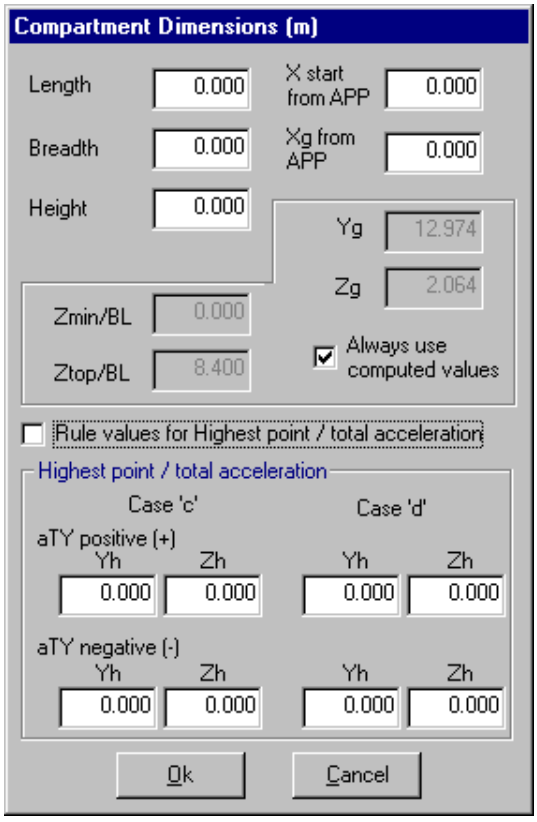


Figure 18 : COMPARTMENT-DIMENSION WINDOW (2)

3.9.2 Loading cases

Clicking on the *Loading Cases* button (Figure 16), the compartment load window is displayed:

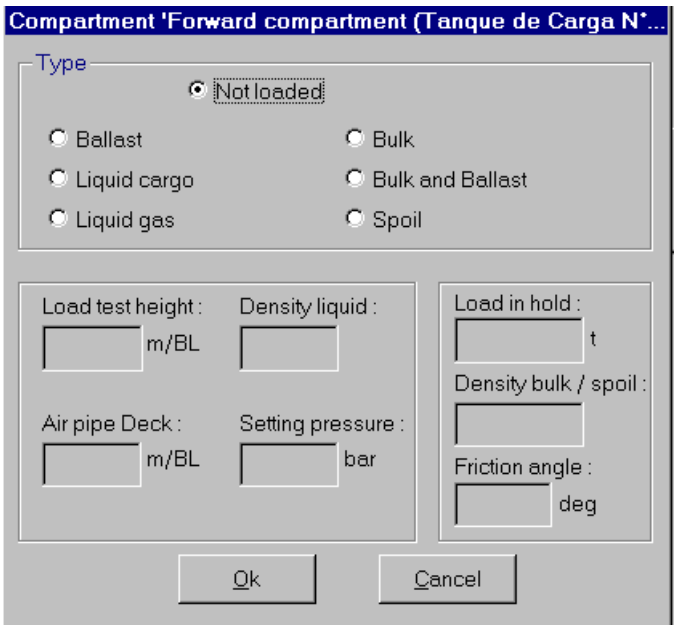


Figure 19 : COMPARTMENT LOAD WINDOW

Type: select a load type.

Liquid cargo
Load test height (in m): tank testing load height from base line.

Air Pipe (in m): distance from top of air pipe height to base line.

Density liquid: density of liquid cargo.

Setting pressure (in bar): setting pressure of safety valves.

Bulk cargo
Load in hold (in t): load in hold (bulk or spoil).

Density of bulk or spoil (in t): density of bulk or spoil.

Friction angle (in °): internal friction angle for bulk.

This input allows to define for the same compartment a liquid cargo and a bulk cargo. This facility may be used for ballastable hold in bulk carrier.

3.10 MENUS

File Menu
It allows to manage the bulkheads (save, open), to print and to quit BHAIN.

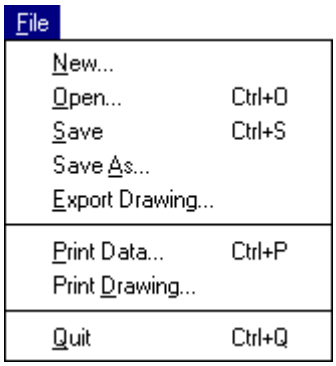


Figure 20 :FILE MENU

Item	Use	Shortcut
New…	Creates a bulkhead.	
Open..	Opens an existing bulkhead.	you can also press Ctrl + O.
Save	Saves the opened bulkhead.	you can also press Ctrl + S or
Save As…	Saves a copy of the bulkhead with another name.	
Export Drawing	Creates a bitmap file from the bulkhead drawing	
Print Data…	Prints the data of the bulkhead (see 3.12.1).	you can also press Ctrl + S or
Print Drawing…	Prints a drawing of a bulkhead (see 1.3.2).	
Quit	Quits BHAIN to return to MARSHELL.	you can also press Ctrl + Q or

Edit Menu
It allows to undo the last action or copy the section drawing to clipboard.

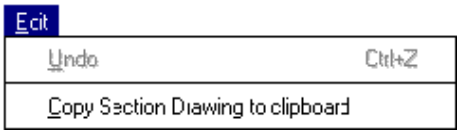


Figure 21 : EDIT MENU

Item	Use	Shortcut
Undo	cancels the last action.	Ctrl + Z
Copy Section Drawing to clipboard	allows to paste the section drawing in any other application.	

Bulkhead Arrangement menu
It gathers the entries to the input fields.

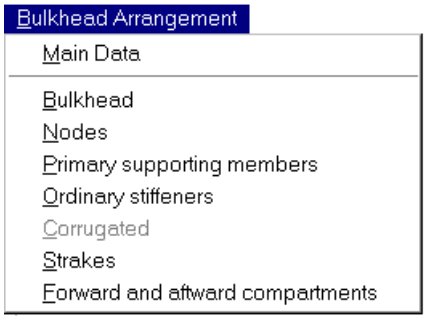


Figure 22 : BULKHEAD ARRANGEMENT MENU

Item	Use	Shortcut
Main data	displays the Main Bulkhead Arrangement Data Window (see Chapter 2 :).	
Bulkhead	displays the Bulkhead management window (see 3.3).	
Nodes	displays the Nodes management window (see 3.4).	
Primary stiffeners	displays the Primary stiffeners management window (see 3.5).	
Secondary stiffeners	displays the Secondary stiffeners management window (see 3.6).	
Corrugated	displays the Corrugation management window (see 3.7).	
Strakes	displays the Strake Band management window (see 3.8).	
Forward and aftward compartments	displays the Forward and aftward compartments management window (see 3.9).	

Check menu
It includes checking tools.

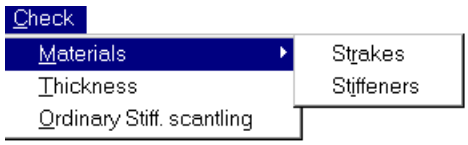


Figure 23 : CHECK MENU

Item	Use	Shortcut
Check Materials - Strakes	displays the strakes with a different color for each material.	
Check Materials - Stiffeners	displays the stiffeners with a different color for each material.	
Check Thickness	displays the strakes with a different color for each thickness.	
Check Secondary Stiffeners scantling	displays the secondary stiffeners with a different color for each stiffener scantling.	

Tools menu :

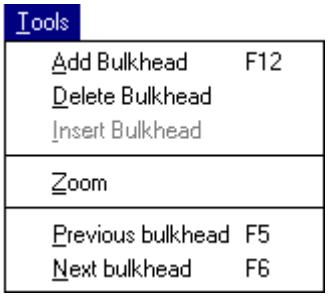


Figure 24 : TOOLS MENU

Item	Use	Shortcut
Add object	creates data (see 3.2).	**or F12**
Delete object	deletes data (see 3.2).	
Insert node…	inserts a node (see 3.4.1), this item is available only during the node input sequence.	
Zoom	allows to zoom in (see 3.13).	
Previous bulkhead	changes the current bulkhead to the previous one (see 3.1).	F5
Next bulkhead	changes the current bulkhead to the next one (see 3.1).	F6

Options menu

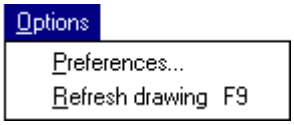


Figure 25 : OPTIONS MENU

Item	Use	Shortcut
Preferences…	displays a set up window for the section drawing on the screen or a printer.	
Refresh drawing	refreshes the screen in case of display anomalies.	F9

3.11

3.12 PRINTING

3.12.1 Printing data

Clicking on  or on Print Data… on the File menu (Figure 20) or pressing Ctrl + P, you enter the Print Data management window:

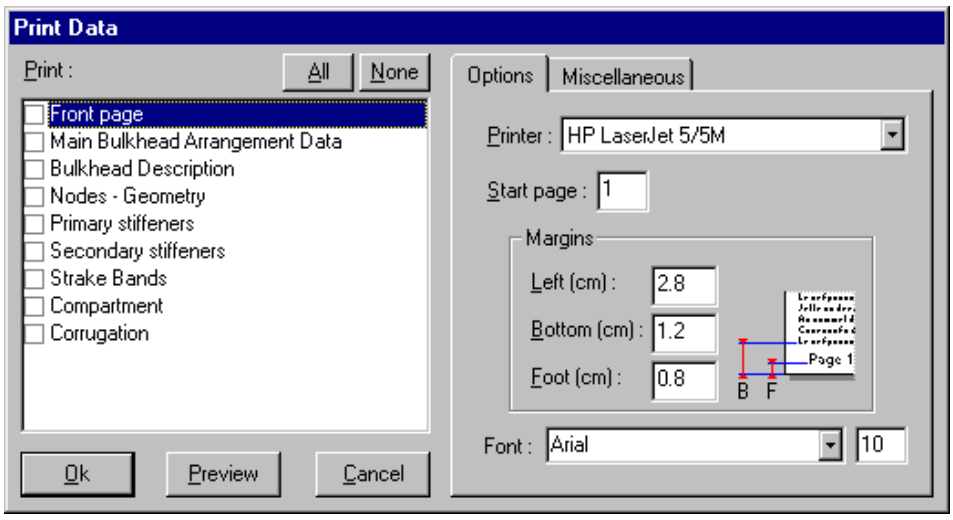


Figure 26 : PRINT DATA MANAGEMENT WINDOW

This window allows you to select what you want to print. The *All* (*None)* button selects (deselects) all the items of the list.

Front page produces the cover page of a report.

3.12.2 Printing drawing

Clicking on *Print Drawing…* on the *File* menu (Figure 20), you enter the *Print Drawing management window*:

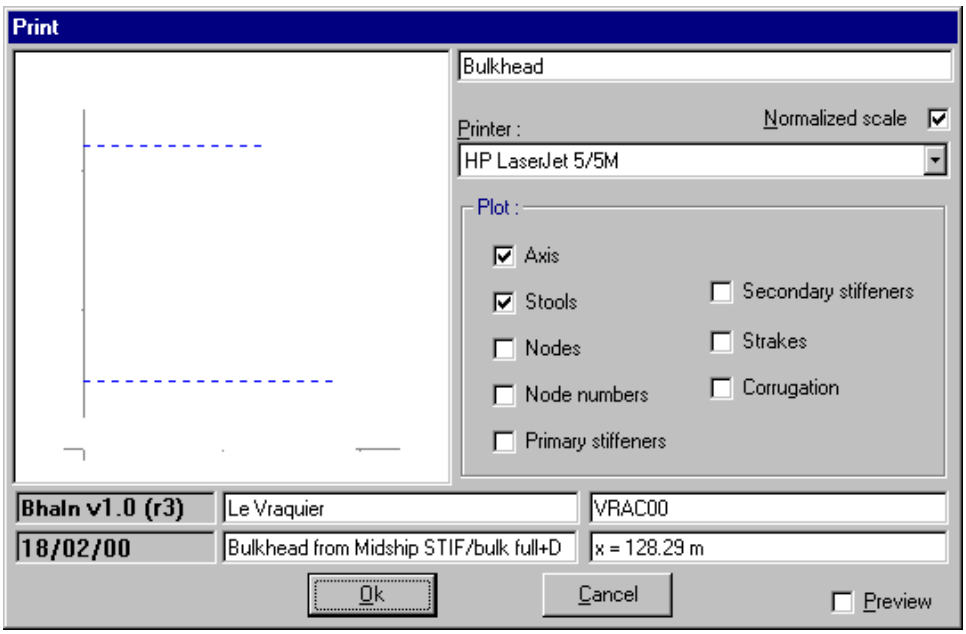


Figure 27 : PRINT DRAWING MANAGEMENT WINDOW

This window allows you to select which item will be printed on the item.

The *Normalized scale* check box will make the drawing printed with a regular scale (e.g. 1/50, 1/100, …).

3.13 ZOOM

It is possible to Zoom in on or out of the Bulkhead view thanks to the *Zoom* Toolbar:

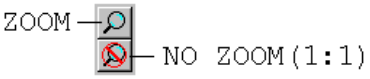


Figure 28 : ZOOM TOOLBAR

A first click on the *Zoom* button (Figure 28) or on *Zoom* on the *Tools* menu (Figure 24) changes the cursor in  and allows you to zoom in the bulkhead view by simple click on it.

A second click on the *Zoom* button (Figure 28) or on *Zoom* on the *Tools* menu (Figure 24) changes back the cursor in  and allows you to work on your zoomed bulkhead view.

To zoom out of the section view, you can:

- click on the *No Zoom (1:1)* button (Figure 28) to bring back the view to the initial size.

right-click on the section view when the *Zoom* button is down.

Table of Figure

MARS2000
User's guide

Booklet 5

CALCULATION OF A BULKHEAD ARRANGEMENT

February 2000 /1
Dev/UG/MARS2000/05

MARINE DIVISION
DEVELOPMENT DEPARTMENT
17 bis, place des Reflets – La Défense 2
92077 Paris – La Défense Cedex
Tel.: + 33 (0)1 42 91 52 91
Fax: + 33 (0)1 42 91 52 98

TABLE OF CONTENTS

Chapter 1 : GENERAL COMMENTS

1.1 CONCEPTS

BhaRule is able to perform calculations in any bulkhead all along the ship length. The bulkheads are to be defined as described in the booklet "Definition of a bulkhead arrangement".

Local calculations are based on the net scantling of the section. This net scantling is evaluated from the main destination of each compartment.

An elementary plate panel (E.P.P.) is a subdivision of plate bordered by primary stiffeners, ordinary stiffeners and bulkhead bounds.

Local calculations for plating are always performed by E.P.P. As strakes can be divided in E.P.P., syntheses of calculations are made to the strake level.

When a welding joint is located between two stiffeners, the E.P.P. belongs to both jouxting strakes. But it is calculated twice, each time with the thickness of one of the two strakes.

1.2 MAIN FEATURES

The BhaRule module performs, for plates and ordinary stiffeners located on a transverse bulkhead, rule calculations according to Jap Rules and Regulations for classification of Steel Ships.

All the calculations carried out by this module which is explained in this booklet, are relative to the bulkhead arrangement which has been selected in MarsShell.

The module allowing to perform the calculation for a given bulkhead arrangement is organized around the following application:

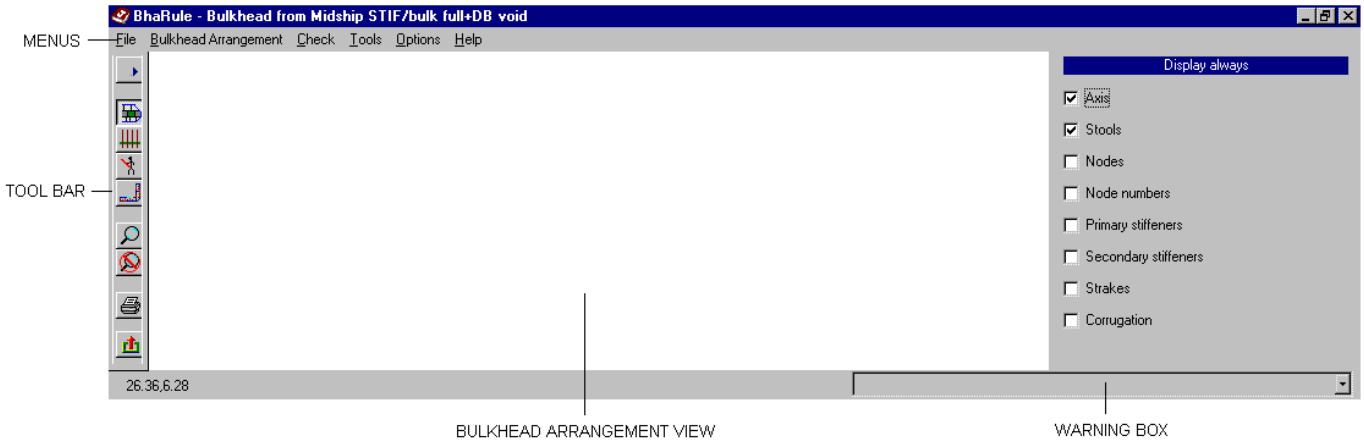


Figure 1 : BHARULE

Warning Box : displays warning message when BhaRule detects incoherence in the Bulkhead arrangement definition.

Bulkhead Arrangement view : displays a view of the bulkhead arrangement.

1.3 COMPUTE BULKHEAD ARRANGEMENT

When you launch BhaRule module or you click on the Compute bulkhead arrangement button  or on Compute bulkhead arrangement on the File menu (Figure 2), the following window is displayed:

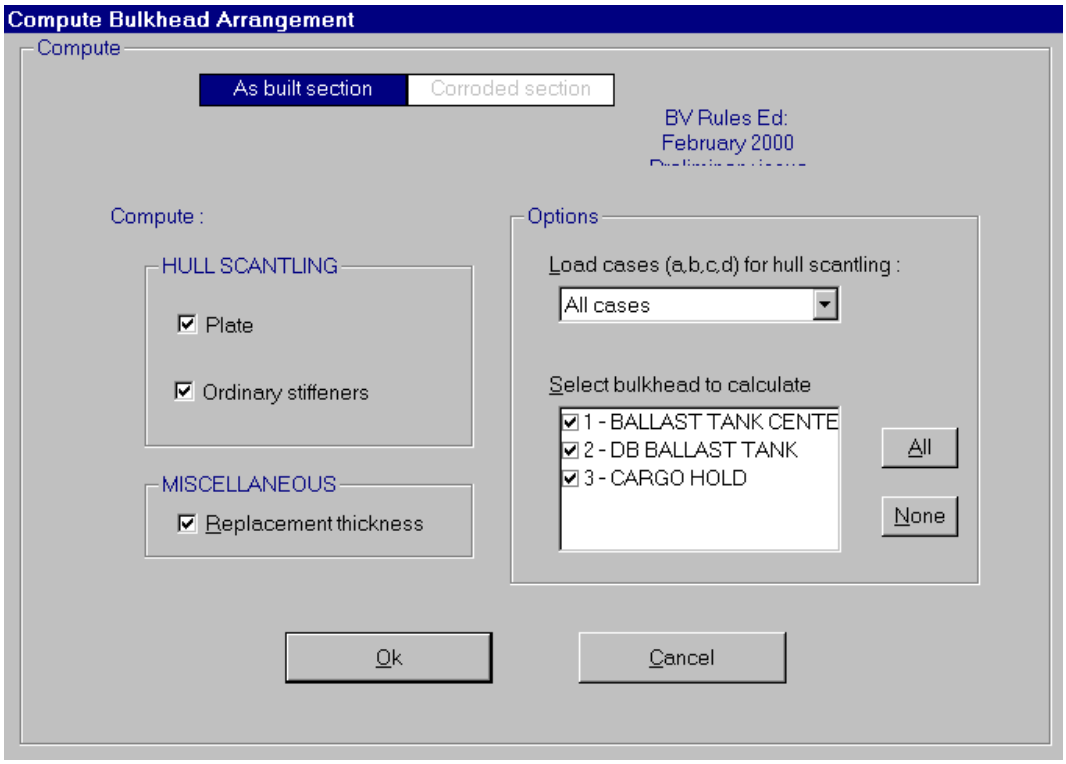


Figure 2 : COMPUTE BULKHEAD ARRANGEMENT WINDOW

This window allows carrying out the hull scantling calculation. It is possible to require calculations for only plates or only stiffeners (by default both of them).

Options
It is possible to require calculations for all the load cases (default value) or for only one of these (e.g. case a, case b,…).

Moreover, it is possible to select all the bulkheads of the bulkhead arrangement (default) or only some of them.

The *All* (*None*) button selects (deselects) all the items of the list.

Chapter 2 : STRAKE RESULTS

Clicking on Strakes button  or on Strakes on Bulkhead Arrangement menu (Figure 3), you enter the Strake results window:

Figure 3 : STRAKE RESULT WINDOW

2.1 SCANTLING CALCULATION

The purpose of this calculation is to check in a given bulkhead the actual scantling of strakes.
The window display the results at strake level or in more detailed way for each elementary plate panel forming the strake. To highlight the strake anomalies if the actual value of considered result is lower then rule value this latter become red.

2.2 SYMBOLS

Gross frame

	Actual	Rule
Thickness	Gross thickness of the strake.	Maximum of Rule thick. Load and Rule Thick. Mini added with the corrosion margin of the strake.

Net frame

	Actual	Rule
Thick. Load	Net thickness of the strake.	Thickness based on internal design pressure. It is calculated on each E.P.P. considered by the program. The output value of Thick. Load is the maximum one.
Thick. Mini	Net thickness of the strake.	Minimum rule thickness. Maximum of the values calculated

		on each E.P.P.

Yielding tab

This tab gives details of the Thick. Load result on the E.P.P. where it is maximized.

Thick	Thickness based on internal design pressure.
Load reference	Code indicating the corresponding type of loading and load case. The possible items of loading are described hereafter.
Ps	Still water pressure.
Pw	Wave pressure.

Load references

LIQ i	Liquid pressure
BULK i	Bulk pressure
Slosh I	Sloshing pressure
Impact i	Impact pressure

For the code described hereabove the figure i means: number of the compartment, the data of which are used in the calculation of pressure.

Chapter 3 : STIFFENER RESULTS

Clicking on Stiffeners button  or on Stiffeners on Bulkhead Arrangement menu (Figure 3), you enter the Stiffener results window:

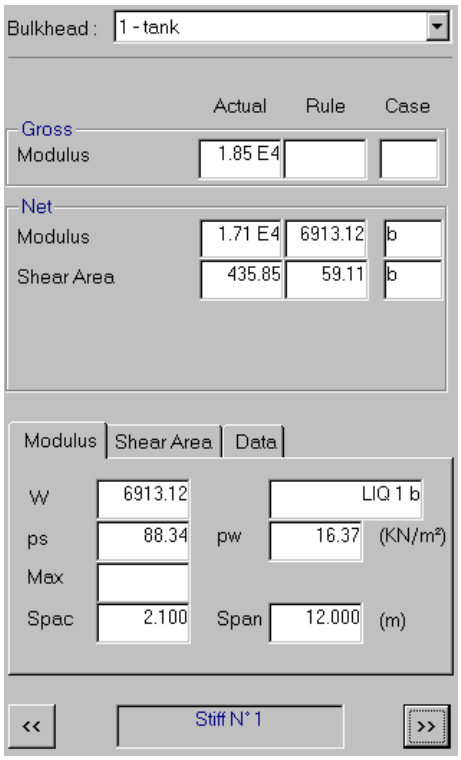


Figure 4 : STIFFENER RESULT WINDOW

3.1 SCANTLING CALCULATION

The purpose of this calculation is to check in a given bulkhead the actual scantling of Stiffeners.

To highlight the stiffener anomalies if the actual value of considered result is lower then rule value this latter become red.

3.2 SYMBOLS

Gross frame

	Actual	Rule
Modulus	Modulus of the stiffener based on its gross scantling.	

Net frame

	Actual	Rule
Modulus	Modulus of the stiffener based on its net scantling.	Modulus based on internal design pressure.
Shear Area	Shear area of the stiffener based on its net scantling.	Shear area based on internal design pressure.

Modulus tab

This tab gives details of the net modulus result.

W	Modulus based on internal design pressure.

Load reference	Code indicating the corresponding type of loading and load case. The possible items of loading are described hereafter.
Ps	Still water pressure.
Pw	Wave pressure.
Max	Code indicating the position along the stiffener where the ratio Rule/Actual is maximal. The possible items of position are described hereafter.
Spac	Spacing of the stiffener.
Span	Span of the stiffener.

Shear A. tab

This tab gives details of the shear area result.

S Area	Shear area based on internal design pressure.
Load reference	Code indicating the corresponding type of loading and load case. The possible items of loading are described hereafter.
Max	Code indicating the position along the stiffener where the ratio Rule/Actual is maximal. The possible items of position are described hereafter.
Ps	Still water pressure.
Pw	Wave pressure.
Spac	Spacing of the stiffener.
Span	Span of the stiffener.

Load references

LIQ i	Liquid pressure
BULK i	Bulk pressure
Slosh I	Sloshing pressure
Impact i	Impact pressure

For the code described hereabove the figure i mean: number of the compartment, the data of which are used in the calculation of pressure.

Position references

Start	Lower position on the span.
End	Upper position on the span.
Between	Point where the moment is maximal.

Chapter 4 : RATIO RESULTS

Clicking on Ratio button  you enter the Ratio results window:

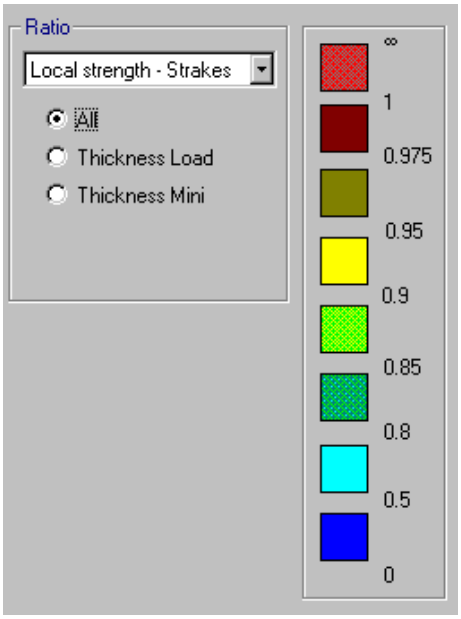


Figure 5 : RATIO RESULT WINDOW

The window allows to select one or several types of ratio from the following list:

- Local strength - Strakes.
- Local strength - Secondary.
- Hereafter the different type of ratio available for each item:

Local strength - Strakes

- All
- Thickness Load
- Thickness Mini

Local strength - Stiffeners

- All
- Net modulus
- Net shear area

Chapter 5 : RENEWAL RESULTS

Clicking on *Renewal*  button you enter the Renewal results window.

5.1 STRAKES

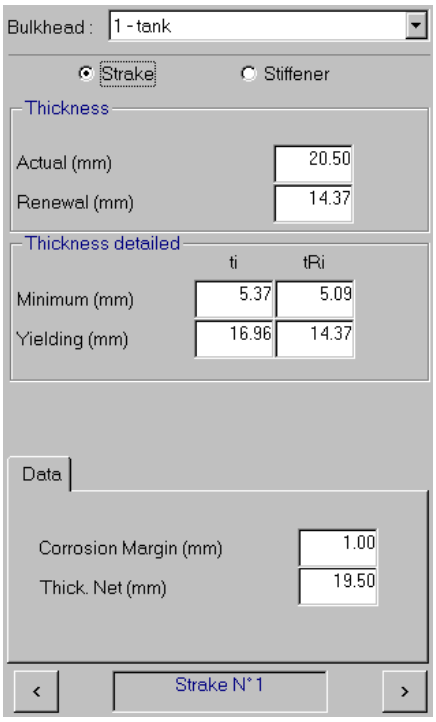


Figure 6 : RATIO RESULT WINDOW (1)

Thickness frame

Actual	Gross thickness of the strake.
Renewal	Maximum of tRi.

Thickness detailed frame

	ti	tRi
Minimum	Minimum net thickness.	Minimum renewal thickness.
Yielding	Thickness based on internal design pressure.	Renewal thickness of plating subjected to lateral pressure.

5.2 LONGITUDINAL STIFFENERS

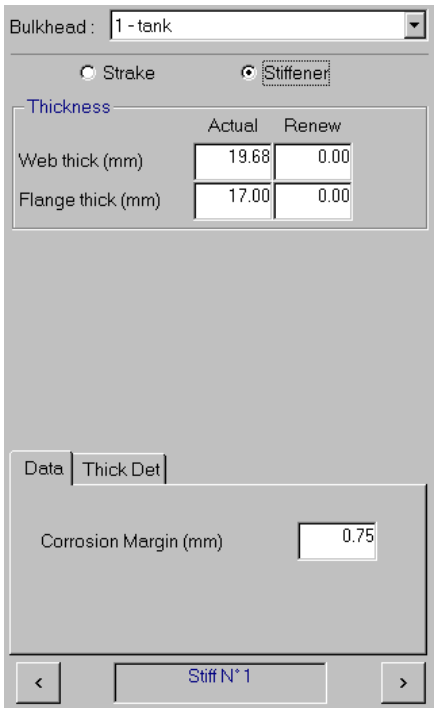


Figure 7 : RATIO RESULT WINDOW (2)

Thickness frame

	Actual	Renew
Web thick	Actual web thickness	Renewal web thickness.
Flange thick	Actual flange thickness	Renewal flange thickness.

Thick Dat tab

	Web	Flange
%		
Buck		

Chapter 6 : GENERAL FEATURES

6.1 MENUS

File Menu
It allows to manage the sections (save, open), to print and to quit MARSIN.

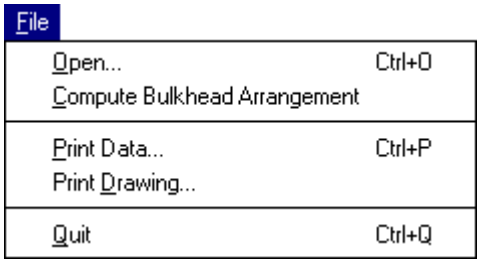


Figure 8 : FILE MENU

Item	Use	Shortcut
Open..	opens an existing bulkhead.	Ctrl + O
Compute Bulkhead Arrangement	allows to set calculation options (see 1.3).	
Print Data…	prints the data of the bulkhead (see 1.4.1).	Ctrl + P or
Print Drawing…	prints a drawing of a bulkhead (see 1.4.2).	
Quit	Quits BHARULE to return to MARSHELL.	Ctrl + Q or

Edit Menu
It gathers the results from calculation.

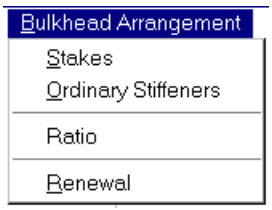


Figure 9 : BULKHEAD ARRANGEMENT MENU

Item	Use	Shortcut
Strakes	displays the Strake results window.	
Stiffeners	displays the Stiffeners results window.	

Check menu
It includes checking tools.

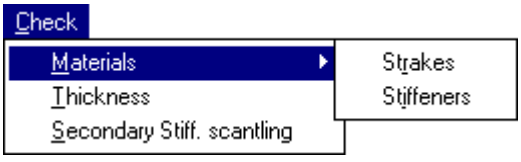


Figure 10 : CHECK MENU

Item	Use	Shortcut
Materials - Strakes	displays the strakes with a different color for each material.	
Materials - Stiffeners	displays the stiffeners with a different color for each material.	
Thickness	displays the strakes with a different color for each thickness.	
Secondary Stiff. scantling	displays the secondary stiffeners with a different color for each stiffener scantling.	

***Tools* menu :**

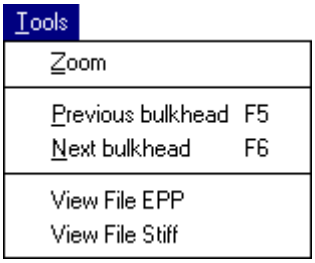


Figure 11 : TOOLS MENU

Item	Use	Shortcut
Zoom	allows to zoom in (see 6.3).	
Previous bulkhead	changes the current bulkhead to the previous one.	F5
Next bulkhead	changes the current bulkhead to the next one.	F6
View File EPP	displays yielding values for all the Elementary Plate Panel in the bulkhead.	
View File Stiff	displays modulus values for all the stiffeners in the bulkhead.	

Options menu

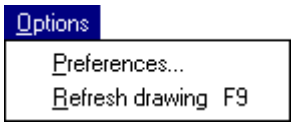


Figure 12 : OPTIONS MENU

Item	Use	Shortcut
Preferences…	displays a set up window for the drawing preferences on the screen or a printer.	
Refresh drawing	refreshes the screen in case of display anomalies.	F9

6.2 PRINTING

6.2.1 Printing data

Clicking on  or on Print Data… on the File menu (Figure 8) or pressing Ctrl + P, you enter the Print Data management window :

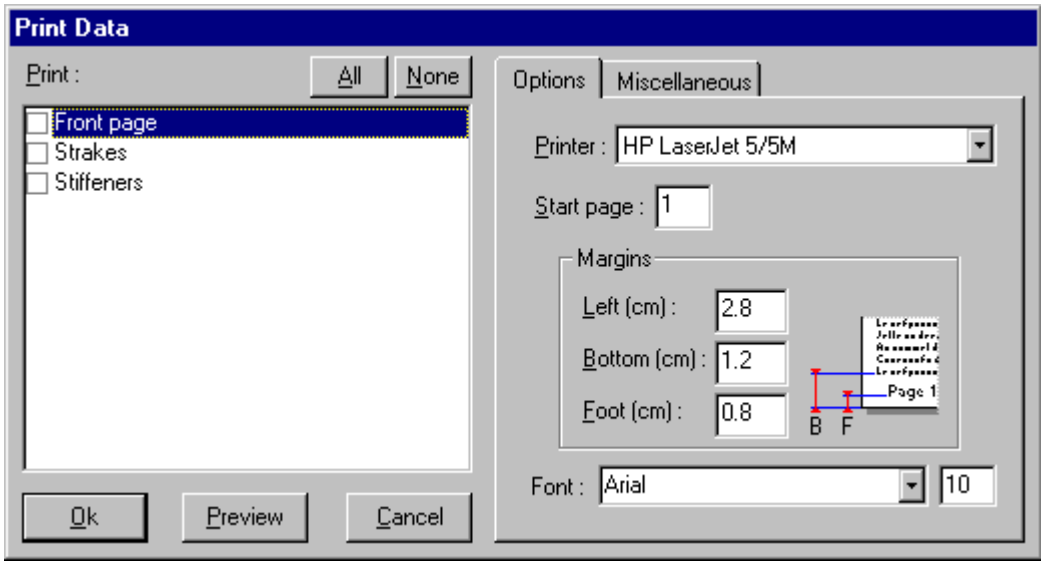


Figure 13 : PRINT DATA MANAGEMENT WINDOW

This window allows you to select what you want to print. The *All* (*None)* button selects (deselects) all the items of the list.

Front page produces the cover page of a report.

6.2.2 Printing drawing

Clicking on *Print Drawing…* on the *File* menu (Figure 8), you enter the *Print Drawing management window* :

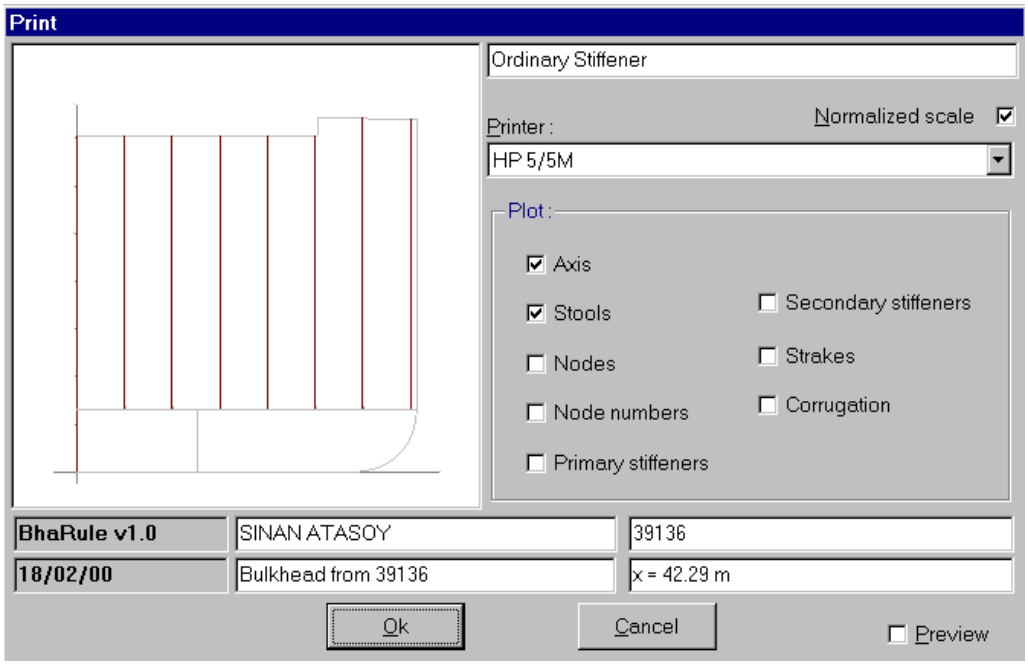


Figure 14 : PRINT DRAWING MANAGEMENT WINDOW

This window allows you to select which item will be printed on the item.

The *Normalized scale* check box will make the drawing printed with a regular scale (e.g. 1/50, 1/100, …).

6.3 ZOOM

It is possible to Zoom in on or out of the Section view thanks to the *Zoom* Toolbar :

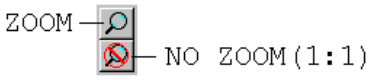


Figure 15 : ZOOM TOOLBAR

A first click on the *Zoom* button (Figure 9) or on *Zoom* on the *Tools* menu (Figure 5) changes the cursor in  and allows you to zoom in the section view by simple click on it.

A second click on the *Zoom* button (Figure 9) or on *Zoom* on the *Tools* menu (Figure 5) changes back the cursor in and allows you to work on your zoomed section view.

To zoom out of the section view, you can :

- click on the *No Zoom (1:1)* button (Figure 9) to bring back the view to the initial size.
- right-click on the section view when the *Zoom* button is down.

Table of Figure

BUREAU
VERITAS

MARS2000
User's guide

Booklet 6

DEFINITION AND CALCULATION OF A TORSION MODEL

February 2000 /1
Dev/UG/MARS2000/06

MARINE DIVISION
DEVELOPMENT DEPARTMENT
17 bis, place des Reflets – La Défense 2
92077 Paris – La Défense Cedex
Tel.: + 33 (0)1 42 91 52 91
Fax: + 33 (0)1 42 91 52 98

TABLE OF CONTENTS

Chapter 1 : GENERAL COMMENTS

1.1 INTRODUCTION

The calculation is based on a simplified model, correct for large opening ships, where the ship is made of continuous holds and each holds is defined by a section (ship beam with variable section). Each hold is described by:

- its start position (X coordinate).
- its end position (X coordinate).
- its shape (existing section in MARS file).
- The user may add on the model transverse and longitudinal beams. The transverse beam must be put at the end on considered hold (end position) and connected to the section defining the hold by an existing node in the section.
- The longitudinal beam length is not free but is always the same of the hold length where the beam is added.

1.2 MAIN FEATURES

The module allowing to input the data of a torsion model is organized around the following application:

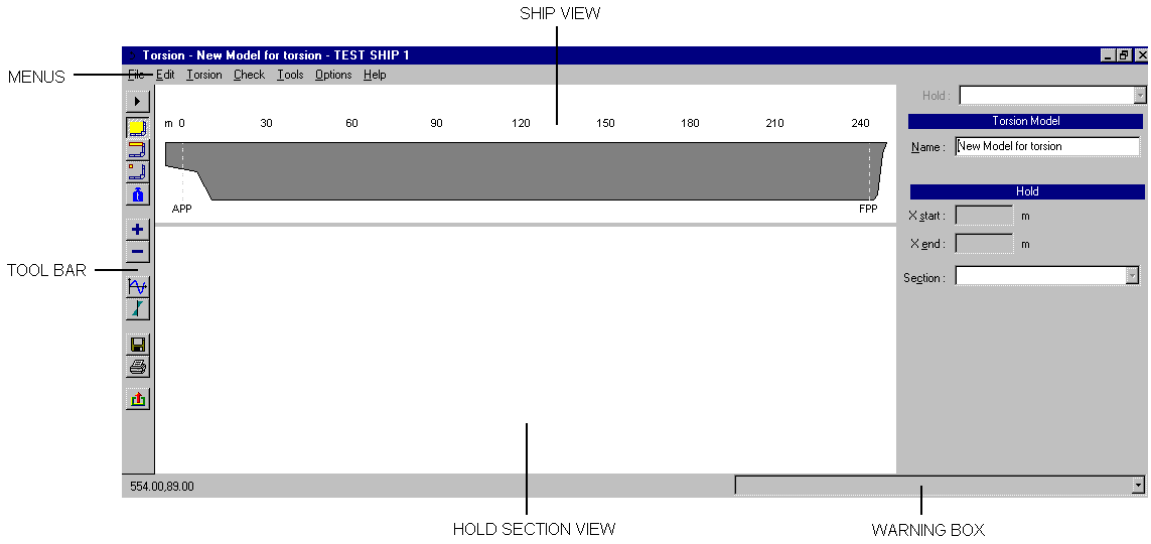


Figure 1 : TORSION

Warning Box: displays warning message when TORSION detects incoherence in the Section definition.

Ship view: displays a view of the hold limits and the position of the section defining the holds.

Section view: displays a view of the section defining the hold.

Chapter 2 : MODEL DEFINITION

2.1 INPUT SEQUENCE

There are four main sequences to define the model of torsion:

- *Holds*
- *Transverse Beams*
- *Longitudinal Beams*
- *Loads*

These different sequences of the model description may be accessed by the following toolbar as follows :

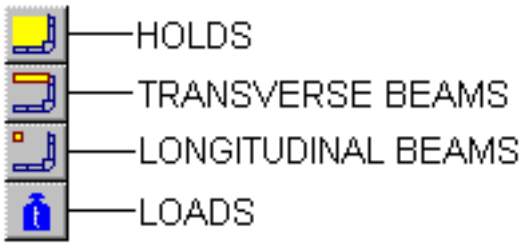


Figure 2 : TORSION TOOLBAR

Holds: allows to define the hold and his characteristics.

Transverse Beams: allows to locate and define the transverse beam characteristics.

Longitudinal Beams: allows to locate and define the longitudinal beam characteristics.

Loads: allows to set the loading condition.

They are four ways to move from one hold to another:

- a direct click on the desired hold in the ship view,
- the *Next Hold* and *Previous Hold* items on the *Tools* menu (Figure 21),
- the F6 key to jump to the next hold or the F5 key to jump to the previous hold,
- using the panel list placed on the right-hand corner of TORSION :

Figure 3 : HOLD LIST

2.2 CREATION AND DELETION OF DATA

In the following input sequence, you can create or delete data :

- Holds : creation or deletion of a hold

- Longitudinal beam : creation or deletion of a longitudinal beam

Each object has to be created or deleted using the following toolbar:

Figure 4 : CREATION-DELETION TOOLBAR

For example, if you want to create a new hold

- click on the *hold* button (Figure 2),
  the *Hold management window* (Figure 5) is displayed,

- click on the *creation* button (Figure 4).

Inversely, if you want to delete an existing hold,

- click on the *hold* button (Figure 2),

- select the hold you want to delete,

- click on the *deletion* button (Figure 4).

To create data, it is also possible to hit the F12 key instead of clicking the *creation* button.
Another way to create or delete objects is to use *Create object* or *Delete object* on the *Tools* menu (Figure 21)

2.3 HOLD DATA

Clicking on the hold button  or on Hold on the Torsion menu (Figure 4), you enter the Hold management window :

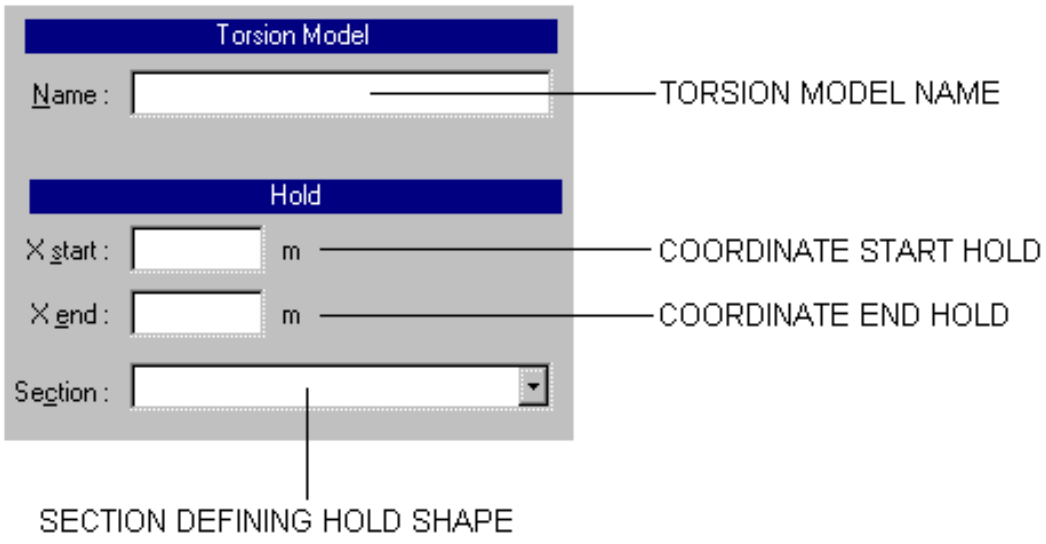


Figure 5 : HOLD MANAGEMENT WINDOW

In this window, you can create or delete holds using the *creation* or the *deletion* buttons (Figure 4).
The holds are automatically numbered, starting at number 1 and with an increasing numbering along the successive holds.

Each hold is defined by a start coordinate, a end coordinate and a section defining the shape.

Torsion Model Name: up to 255 characters.

X start (in m): X coordinate of start hold. The X start is always equal to X end of previous hold.

X end (in m): X coordinate of end hold.

Section: section defining hold shape; can be choose from the existing sections in MARS file.

When you choose a section to define a hold shape a triangle in the low part of ship view is added to indicate the position of the section along the ship.

2.4 TRANSVERSE BEAM DATA

Clicking on the transverse beam button  or on Transverse beam on the Torsion menu (Figure 4), you enter the Transverse beam management window where these inputs are performed:

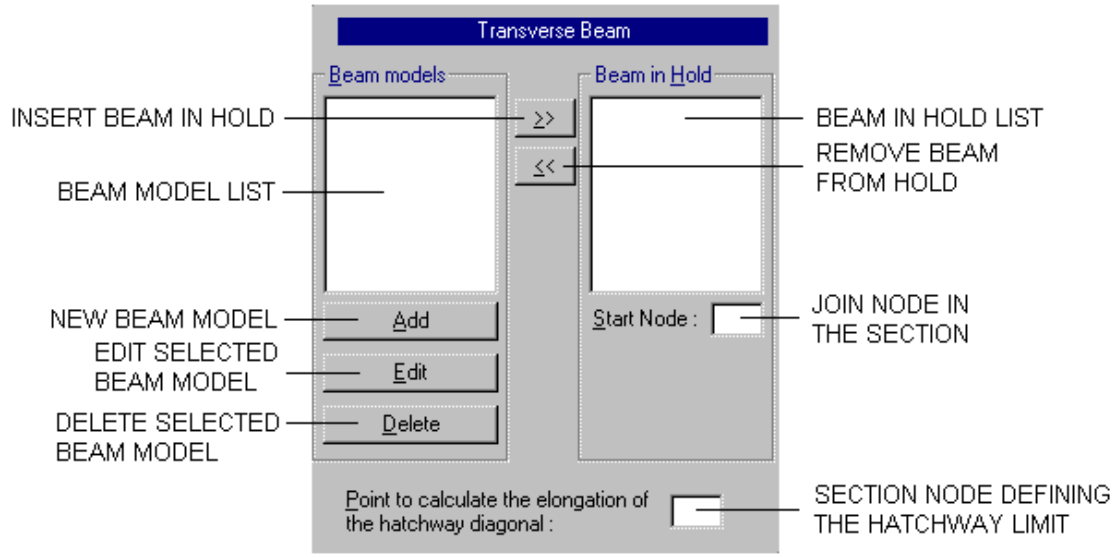


Figure 6 : TRANSVERSE BEAM MANAGEMENT WINDOW

In case of the section considered is the total one (not only the symmetric part) the interface shows also a field *End Node* that means join node for the end of transverse beam and the field *Second point to calculate the elongation* that means other node in the section that define the hatchway diagonal.

The *Add* button 

allows you to create a beam model.

They are automatically numbered, starting at number 1 and with an increasing numbering along the successive beams.

TAKE NOTE OF THE MAXIMUM NUMBER OF TRANSVERSE BEAM IN EACH HOLD IS EQUAL TO 3

The *delete* button 

is used to delete a beam model.

2.4.1 Beam Model creation

Using *Add* button and *Edit* button is displayed the Beam Model management window; this is the way to define or change the geometric characteristics of a beam model.

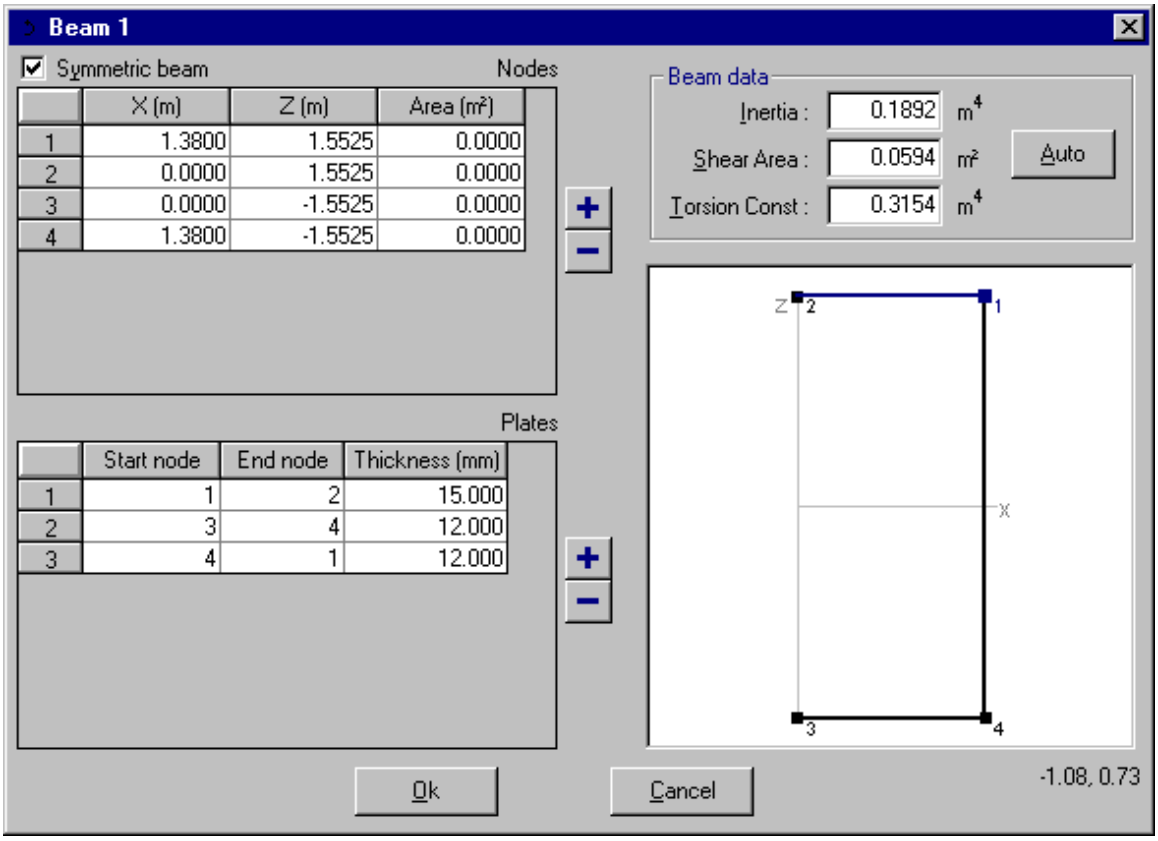


Figure 7 : BEAM MODEL MANAGEMENT WINDOW

Input data

Using the Check Box  placed on the left-hand corner you can define the type of beam.

The beam is defined as a list of nodes connected by the plates:

Nodes table

X (in m): node coordinate;

Z (in m): node coordinate;

Area (in m²): concentrated area in the node, if any (i. e. stiffener).

Plates table

Start node: start plate using a node in the node list defined above;

End node: end plate using a node in the node list defined above;

Thickness (in mm): thickness of plate.

The *creation* and the *deletion* buttons (Figure 4) allow you to create or delete nodes and plates.

Result data (Beam data) displaying when clicking Auto button

Inertia(in m^4): inertia of the beam;

Shear area (in m^2): shear area of the beam;

Torsion constant (in m^4): torsion constant of the beam.

TAKE NOTE THAT WITHOUT CLICKING AUTO BUTTON THE ABOVE VALUES ARE NOT INITIALISED

Clicking on Ok will then create the beam.

Clicking on Cancel will close the window but the beam model remains in the list.

2.5 LONGITUDINAL BEAM DATA

Clicking on the *Longitudinal beam* button  or on Longitudinal beam on the Torsion menu (Figure 4), you enter the *Longitudinal beam management window* where these inputs are performed:

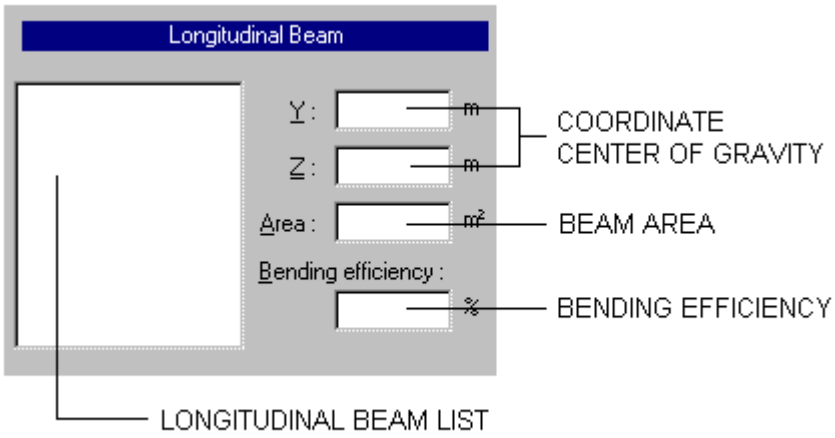


Figure 8 : LONGITUDINAL BEAM MANAGEMENT WINDOW

The *creation* and the *deletion* buttons (Figure 4) allow you to create or delete Longitudinal beams.

TAKE NOTE OF THE MAXIMUM NUMBER OF LONGITUDINAL BEAM IN EACH HOLD IS EQUAL TO 3

Each longitudinal beam is defined by the coordinates, area and bending efficiency.

Y (in m): Y coordinate of center of gravity;

Z (in m): Z coordinate of center of gravity;

Area (in m^2): longitudinal beam area;

Bending efficiency (in %): percentage of contribution to the overall longitudinal strength.

2.6 LOADS DATA

Clicking on the *Loads* button  or on Loads on the Torsion menu (Figure 4), you enter the *Loads management window* where these inputs are performed :

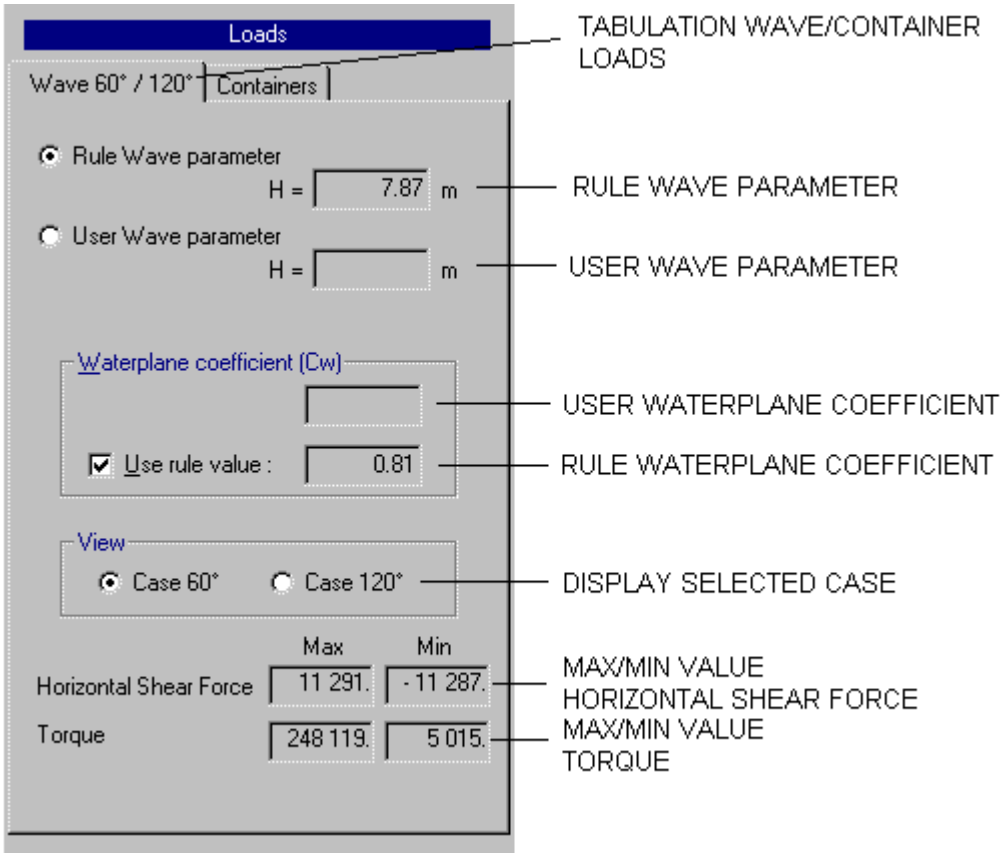


Figure 9 : LOADS MANAGEMENT WINDOW (1)

In this window you can display the load distributions applied in the TORSION calculation.

The interface shows in the section view the following distributions conforming to user choice in frame View:

- Horizontal Shear Force for ship direction forming an angle of 60° or 120° with the prevailing sea direction;
- Torque for ship direction forming an angle of 60° or 120° with the prevailing sea direction.
- The default setting for this window is rule values but you can use own values for wave parameter and waterplane coefficient.
- The window shows also the maximum and minimum values for the distributions displayed.

For non uniform distribution of cargo container case the window display the rule distributions and values (using as input container stacks and tiers) or the user distributions (using as input X coordinate of maximum distribution and maximum distribution value).

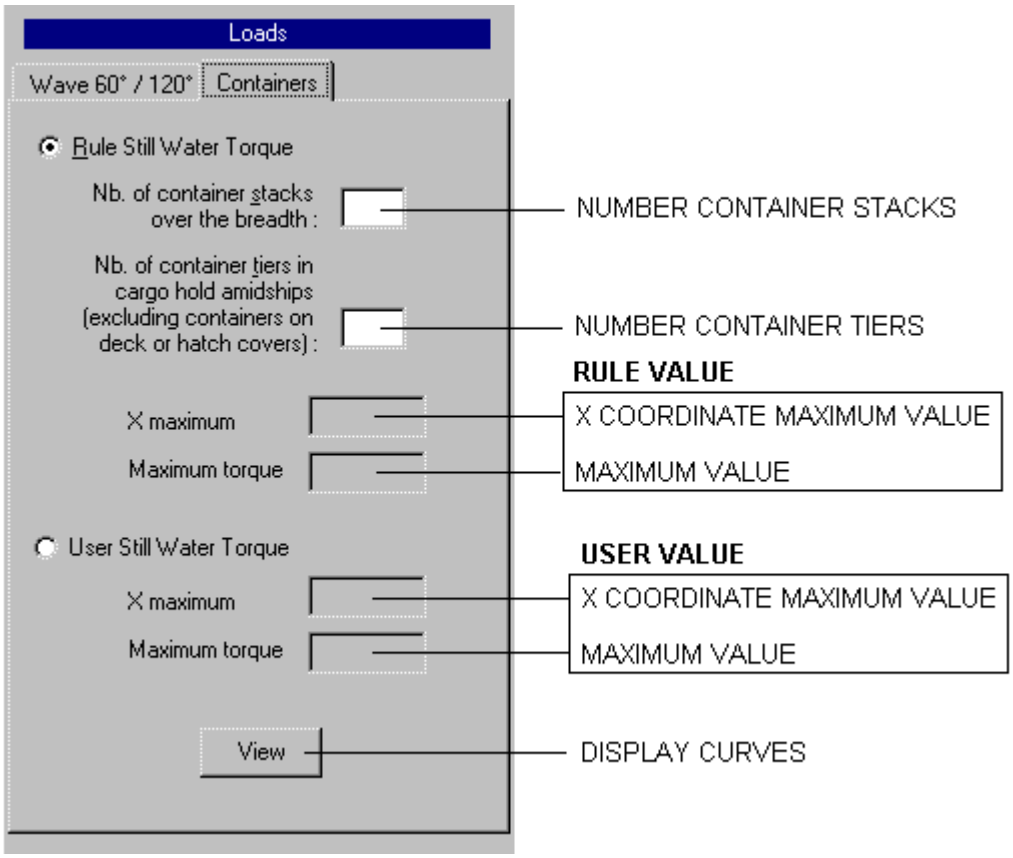


Figure 10 : LOADS MANAGEMENT WINDOW - 2

Chapter 3 : RESULT

When you have finished the data input you have to run the calculation with the Run button
At the end of calculation the torsion result button and stresses button become active.
The strength and bimoment distributions may be saved and used in MarsRule2000 module in plates and ordinary stiffeners scantling calculations.

3.1 TORSION RESULTS

The distributions displayed, calculated in fifty points among the X start of first hold and X end of last hold, are the following:

- Horizontal Shear Force;
- Torque;
- Bimoment.

Clicking on the *Torsion results* button  or on *Torsion result* on the *Torsion* menu (Figure 4), you enter the *Torsion result management window*:

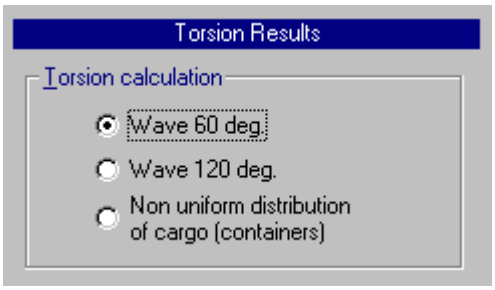


Figure 11 : TORSION RESULT MANAGEMENT WINDOW

The window allows you to select which item will be display among the three considered loading case (see 2.6.).

3.2 STRESSES

Clicking on the *Stresses* button  or on *Stresses* on the *Torsion* menu (Figure 4), you enter the *Stresses management window* :

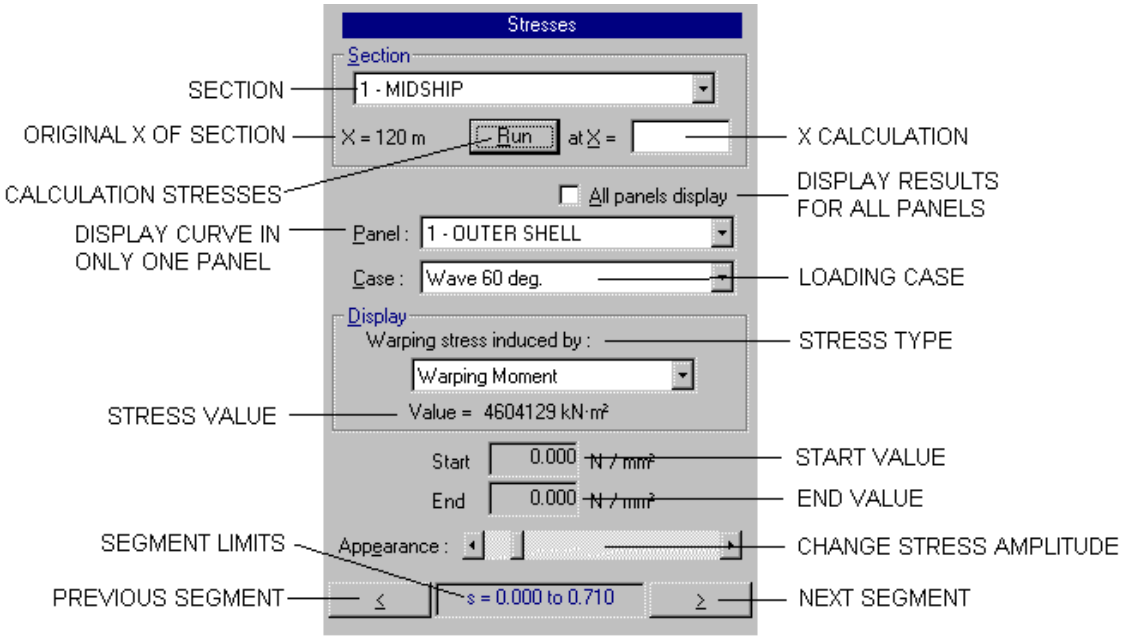


Figure 12 : STRESSES MANAGEMENT WINDOW

The user can run the stress calculation for all the sections existing in the MARS file (also for the section not utilised for the model) at whatever X coordinate.

The stress display are induced by:

- Horizontal Shear Force;
- Total internal torque;
- Warping moment.

When you click in the section view the window shows the value for the selected segment in *Start* and *End* fields.

3.3 PRINT RESULTS

The program is able to calculate the displacements of nodes defining the hatch cover limits. Clicking on the *Hatchway Results* item in the Print window the results are displayed for the three loading cases:

Nodes Displacements - Case Wave 60°

Hold	Nodes Nb	Nodes Y (m)	Nodes Z (m)	Hold aft end U (m)	Hold aft end V (m)	Hold aft end W (m)	Nodes Nb	Nodes Y (m)	Nodes Z (m)	Hold fore end U (m)	Hold fore end V (m)	Hold fore end W (m)
1	39	14.225	20.600	0.002	0.000	0.000		-14.225	20.600	-0.025	0.192	0.104
2	35	14.225	20.600	0.037	0.192	-0.104		-14.225	20.600	-0.037	0.443	0.227
3	23	14.225	20.600	0.047	0.443	-0.227		-14.225	20.600	-0.045	0.652	0.331
4	23	14.225	20.600	0.045	0.652	-0.331		-14.225	20.600	-0.043	0.829	0.422
5	35	14.225	20.600	0.035	0.829	-0.422		-14.225	20.600	-0.033	0.964	0.501
6	39	14.225	20.600	0.026	0.964	-0.501		-14.225	20.600	-0.018	0.988	0.534
7	30	10.345	19.128	0.013	0.933	-0.388		-10.345	19.128	-0.013	0.889	0.387

Nodes Displacements - Case Wave 120°

Hold	Nodes Nb	Nodes Y (m)	Nodes Z (m)	Hold aft end U (m)	Hold aft end V (m)	Hold aft end W (m)	Nodes Nb	Nodes Y (m)	Nodes Z (m)	Hold fore end U (m)	Hold fore end V (m)	Hold fore end W (m)
1	39	14.225	20.600	0.000	0.000	0.000		-14.225	20.600	-0.009	0.041	0.025
2	35	14.225	20.600	0.017	0.041	-0.025		-14.225	20.600	-0.021	0.211	0.105
3	23	14.225	20.600	0.030	0.211	-0.105		-14.225	20.600	-0.031	0.429	0.202
4	23	14.225	20.600	0.031	0.429	-0.202		-14.225	20.600	-0.034	0.681	0.311
5	35	14.225	20.600	0.023	0.681	-0.311		-14.225	20.600	-0.029	1.001	0.451
6	39	14.225	20.600	0.015	1.001	-0.451		-14.225	20.600	-0.012	1.386	0.642
7	30	10.345	19.128	0.010	1.319	-0.467		-10.345	19.128	0.011	1.504	0.543

Figure 13 : PRINT NODES DISPLACEMENTS

-
-
- Moreover the program is able to calculate the elongation of the hatchway diagonal :

Elongation of the hatchway diagonal

Hatch	Diagonal Length (m)	Loading condition Wave 60° (m)	Wave 120° (m)	Container (m)
1	39.988	0.155	0.036	0.011
2	39.988	0.229	0.146	0.021
3	39.988	0.210	0.197	0.023
4	39.988	0.180	0.222	0.023
5	39.988	0.134	0.257	0.022
6	39.988	0.034	0.277	0.018
7	34.936	0.014	0.093	0.004

Figure 14 : PRINT ELONGATION

-

- Clicking on the *Transverse Beams: Shear Force* item in Print window the shear force values applied on the section node connected to the transverse beams are displayed:

Shear Force in Transverse Beams

Hold N°	Name	X (m)	Loading condition Wave 60° (KN)	Wave 120° (KN)	Container (KN)
1	Beam 1	63.500	-2692.170	-1471.613	-233.198
2	Beam 1	91.600	-1263.030	-1045.491	-128.953
3	Beam 1	119.700	-3213.413	-3346.517	-365.431
4	Beam 1	147.800	-2887.563	-3798.175	-372.956
5	Beam 1	175.900	-836.041	-1748.202	-140.689
6	Beam 1	204.000	-394.995	-3862.289	-219.478

Figure 15 : PRINT SHEAR FORCE

-

3.4 3D View

Clicking on *3D View…* on the *Check* menu (Figure 5), you enter the *3D View management window*:

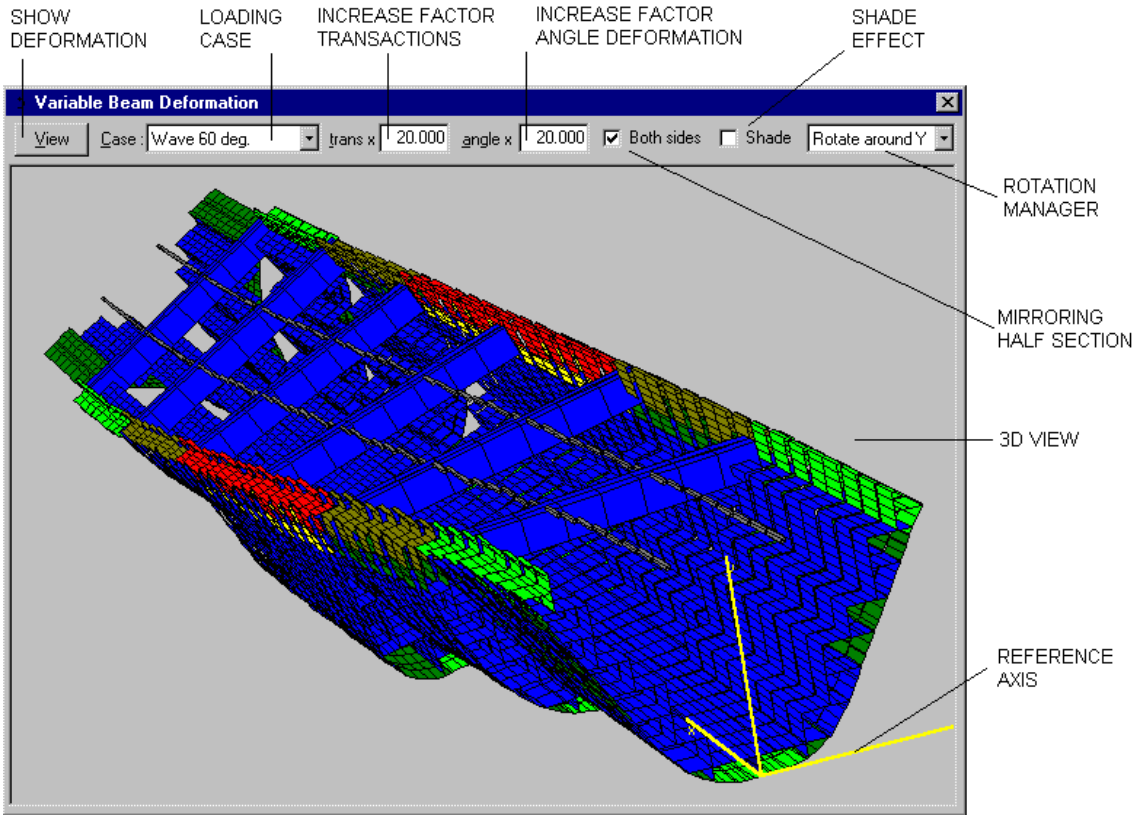


Figure 16 : 3D VIEW MANAGEMENT WINDOW

This window allows you to display the deformation in 3D view. The user can show the torsion deformation for the following loading case:

- Wave 60°;
- Wave 120°;
- Non uniform distribution of containers.

In case of the deformation values are not considerable you can use the increase factors to amplify the deformation effects. Using the *Both sides* check you can show all the ship model with the mirror part as regards the Z axis.

Using the rotation manager and after clicking in the reference axis is possible to change the 3D view as the user likes.

Chapter 4 : GENERAL FEATURES

4.1 MENUS

File Menu
It allows to manage the sections (save, open), to print and to quit MARSIN.

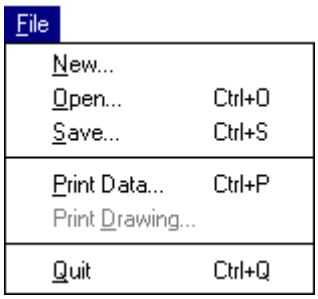


Figure 17 :FILE MENU

Item	Use	Shortcut
New…	creates a new model for torsion.	
Open..	opens an existing model for torsion.	you can also press Ctrl + O.
Save	saves the opened model for torsion.	you can also press Ctrl + S or
Print Data…	prints the data of the model for torsion (see 4.2.1).	you can also press Ctrl + S or
Quit	quits TORSION to return to MARSHELL.	you can also press Ctrl + Q or

Edit Menu
It allows to undo the last action or copy the section drawing to clipboard.

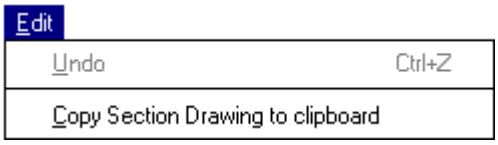


Figure 18 : EDIT MENU

Item	Use	Shortcut
Undo	cancels the last action.	Ctrl + Z
Copy Section Drawing to clipboard	allows to paste the section drawing in any other application.	

Torsion menu
It gathers the entries to the input fields.

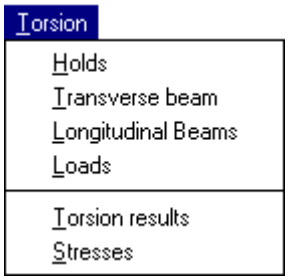


Figure 19 : TORSION MENU

Item	Use	Shortcut
Holds	displays the Hold management window (see 2.3).	
Transverse Beam	Displays the Transverse beam management window (see 2.4).	
Longitudinal Beams	Displays the Longitudinal beam management window (see 2.5).	
Loads	Displays the Load management window (see 2.6).	
Torsion results	Displays the Torsion results management window (see 3.1).	
Stresses	Displays the Stress management window (see 3.2).	

Check menu
It includes checking tools.

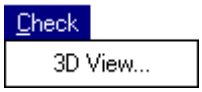


Figure 20 : CHECK MENU

Item	Use	Shortcut
3D View…	Displays 3D Beam deformation (see 3.3).	

***Tools* menu :**

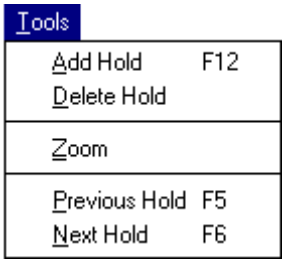


Figure 21 : TOOLS MENU

Item	Use	Shortcut
Add object	creates data (see 2.2).	+ **or F12**
Delete object	deletes data (see 2.2).	−
Previous Hold	changes the current hold to the previous one (see 2.1).	F5
Next Hold	changes the current hold to the next one (see 2.1).	F6

Options menu

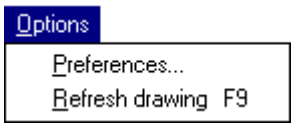


Figure 22 : OPTIONS MENU

Item	Use	Shortcut
Preferences…	displays a set up window for the section drawing.	
Refresh drawing	refreshes the screen in case of display anomalies.	F9

4.2 PRINTING

4.2.1 Printing data

Clicking on  or on Print Data… on the File menu (Figure 17) or pressing Ctrl + P, you enter the Print Data management window:

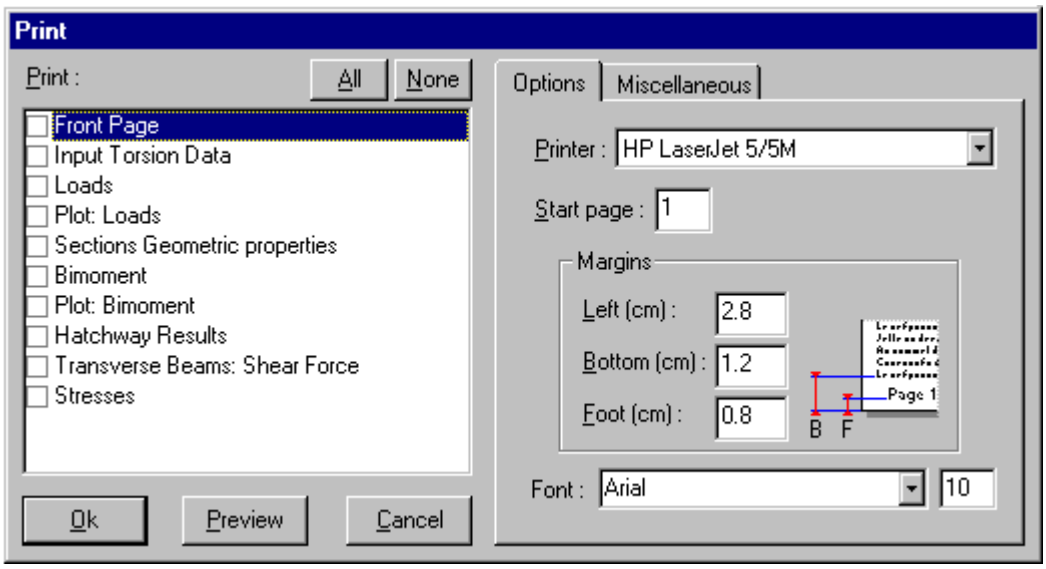


Figure 23 : PRINT DATA MANAGEMENT WINDOW

This window allows you to select what you want to print. The *All* (*None)* button selects (deselects) all the items of the list.

Front page produces the cover page of a report.

Table of Figure