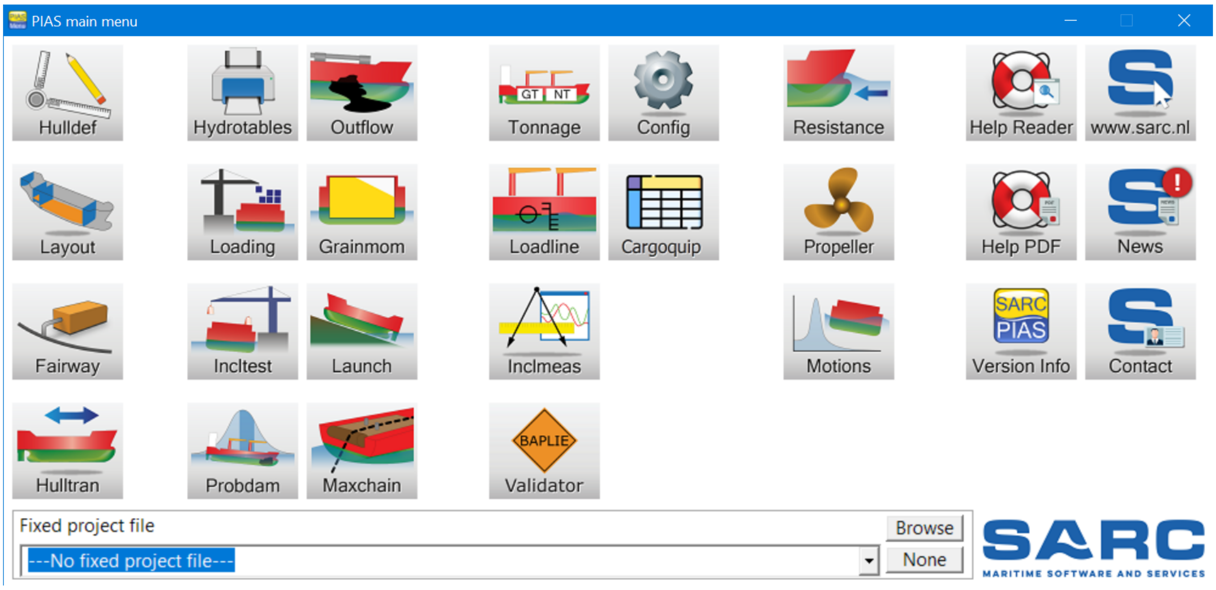
Generated on June 8, 2025

Manual of PIAS[1]

Program for the Integral Approach of Shipdesign



Scheepsbouwkundig Advies en Reken Centrum (SARC) BV
Landstraat 5
1404 JD Bussum, The Netherlands
Phone +31 85 0409040
E-mail sarc@sarc.nl
www.sarc.nl

Contents

Chapter 1

Preface

*This is the manual for the **P**rogram of **I**ntegral **A**pproach of **S**hip design, PIAS. PIAS contains modules for hull form design, fairing, definition of compartments, bulkheads and decks and a wide variety of modules for naval architectural design calculations, such as stability, (probabilistic) damage stability, longitudinal strength and a number of hydrodynamical modules. In this manual all functions and options of all modules are being discussed into detail. For more general information and (theoretical) background we refer to the internet, to www.sarc.nl/pias.*

1.1 Structure of this manual

The basic structure of this manual is a one-to-one relationship between chapters and PIAS modules. For this reason, most chapter titles start with the corresponding module naam, although there are also a number of supporting chapters where for instance installation or frequently used menus are discussed, or general instructions are given. The chapter sequence is roughly:

1. Introductory chapters and installation details.
2. Hull form design, linesplan and fairing, notably the Fairway module.
3. Definition of hullform and other items related to the external geometry, on module Hulldef.
4. Hull form transformation.
5. Everything related to internal geometry, such as bulkheads, decks, compartments, tank volume, sounding tables etc., in particular on the Layout module.
6. The production of hydrostatics or stability-related tables, notably on the Hydrotables module, but also ob e.g. Maxchain and Grainmom.
7. Loading and intact stability, on module Loading.
8. Damage stability, also probabilistic with the Probdam module.
9. Resistance and propulsion.
10. And finally, a number of auxiliary chapters on modules which play no central role in PIAS, although they may provide useful assistance.

A novice is advised to start with the introductory chapter 2 on page 3, Getting started with PIAS, while the more advanced users will find their way browsing.

1.2 Contact details

PIAS is produced by the company SARC, with the following full contact details:

Scheepsbouwkundig Advies en Reken Centrum (SARC) BV
Landstraat 5
1404 JD Bussum
The Netherlands
Tel. +31 85 04 09 040
Web www.sarc.nl
Email sarc@sarc.nl Support support@sarc.nl

SARC website (2016)

Chapter 2

Getting started with PIAS

The PIAS suite consists of many modules, each module addresses a specific area of ship design, such as hull form definition, hull form design, extended hydrostatics, intact- and damage stability calculations and resistance and propulsion estimations. In subsequent chapters each module will be elaborated. But first, in this chapter the installation of the program, the main menu, general menu options, and definitions will be addressed.

2.1 Installation of PIAS

Installation of PIAS software is started by executing the installation program *username.exe*, which can be obtained after login in with the **Login** button on `www.sarc.nl`. After starting the installation program, you need to agree with the licence agreement, which is shown on screen. Next step is to choose a location for the installation of the program and whether or not a shortcut will be created on the desktop. Hereafter, the PIAS programs are installed in the selected folder and the shortcut may be created.

2.1.1 Installation command line parameters

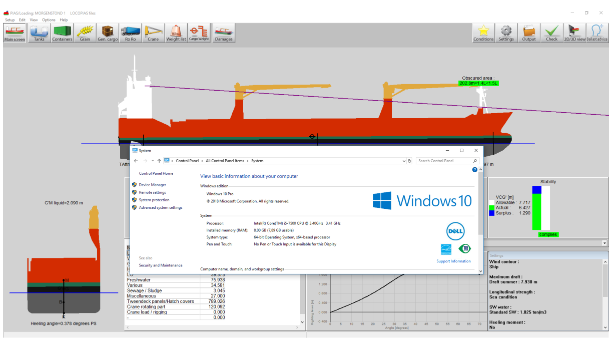
The installation program *username.exe* accepts the following command line parameters;

-s	The PIAS installation is performed without user interaction.
-r	PIAS menu is not started automatically by the installer.
-path=	PIAS will be installed in the folder specified here. The specified installation path may not contain spaces.

Example: `username.exe -s -r -path=c:\custominstallpath`

2.1.2 System requirements

For the current system requirements for PIAS, please refer to the information on our website `www.sarc.nl/system-requirements`



(LOCO-)PIAS under Windows 10, 64 bits.

The module Fairway achieves its high efficiency through intensive use of the graphics adapter. The adapter should support at least OpenGL 2.0; which is a standard from 2004 and supported by most cards these days. Although the program doesn't put further requirements on the graphics adapter, the capabilities of modern hardware are fully utilized.

There are exceptional cases in which Fairway exposes weaknesses in the driver software of the graphics adapter, which cause it to malfunction or not work at all. The cases that we know of can be resolved with specific driver settings, or by installing a different version of the driver. Details are discussed in section 6.3.8 on page 125, Troubleshooting.

2.1.3 Software protection key

PIAS is protected against unauthorized use by a security device called *Codemeter*, from *Wibu systems*.

2.1.3.1 Codemeter

The Codemeter, produced by Wibu systems, is available in two variants. One with a *hardware key*, and one working with a (registered and identifiable) computer that acts as license server. The installation- and update procedure of Codemeter is described in section 3.1 on page 21, Additional installation and update details from the Codemeter software protection. Additionally, two short clips on the installation procedure are available:

- `Installation of PIAS using Codemeter license protection`[1].
- `Configure the Codemeter protection as a network license server`[2].

2.1.4 Digitizer (tablet)

For measuring drawings (body plan or wind profile) a digitizer (tablet) can be used, for which you need to install a so-called *Wintab* driver, which is specific for each type or brand of digitizer. In general, such a driver is provided by the manufacturer of your digitizer (an overview of Wintab drivers can be found on the `Autodesk site`[3], because Wintab is also used by Autocad). The digitizer would also be equipped with function keys, for which you may refer to section 3.3 on page 27, Digitizer function keys.

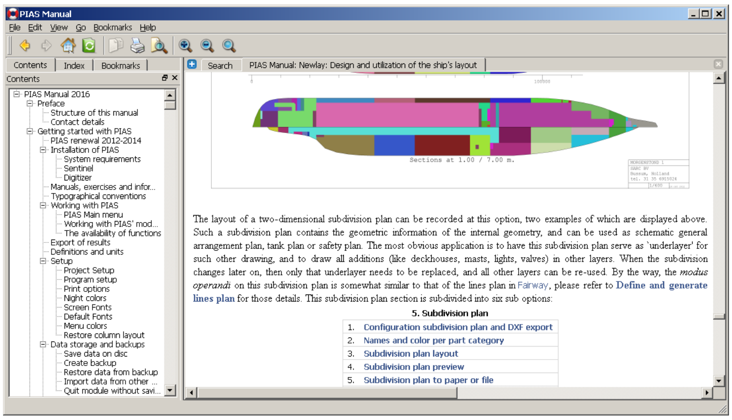
2.2 Manuals, exercises and information sources

The manuals are available in three incarnations, which are identical by content:

- One PDF file which contains all chapters, and is called PIASmanual_en.pdf for the English version. A PDF-reader is required to open this file.
- HTML pages, viewable with a web browser.
- A help reader, co-installed with PIAS, accessible from each module, which shows directly the module-specific manual section. This reader (from which an example is shown below) contains also the usual functions such as search, select on index words, print etc.

The PDF and HTML versions are available on `www.sarc.nl/downloads`. The manual is primarily organized per module, so each module has a dedicated manual chapter where the role of the module, and the several functions and tools are being discussed. The role of the manual is to provide background information and support in the use of PIAS. Although quite some design-related subjects are discussed in the manual, it cannot be considered as a course in ship design. Addtionally, this `Downloads`[4] section also contains a link to PIAS *exercises* and training material. The novice in PIAS is advised to go through these exercises.

[1] `https://youtu.be/AOS3NP6ExPM`
[2] `https://youtu.be/Oc-J3IG4mhg`
[3] `https://knowledge.autodesk.com/support/autocad/troubleshooting/caas/sfdcarticles/sfdcarticles/Obtain-latest-Wintab-digitizer-drivers.html`
[4] `https://www.sarc.nl/downloads/`



Manual in help reader.

Significant software changes or enhancements are communicated in news messages, which are distributed through different channels:

- All news about modifications and enhancements of PIAS — as well as other SARC related news such as trade fair presence — is placed on website[5] and published in Linkedin group SARC BV[6].
- Additionally, news on crucial PIAS modifications, for example those who lead to changed calculation results, is send by e-mail, for which you can (un-)subscribe with a mail to sarc@sarc.nl.
- Approximately once a year the major modifications of PIAS are summarized in a newsletter which is sent by mail to those who subscribed. These newsletters are also collected on the website.

Finally, on a number of subjects which touch PIAS some background references are available, in the shape of papers[7], as presented on a conference or published in a journal. The majority of these papers is written in English, with an occasional one in Dutch.

2.3 Typographical conventions

Text between < > symbols indicates the letter or name of a keyboard key to be pressed, e.g. <Enter>. Key combinations are typeset with a + as in <Ctrl + Q>, and a sequence of key presses is written as e.g. <Alt><C>. A menu option (from the menu bar of the window, or a push button in the window) is indicated as [Option].

2.4 Working with PIAS

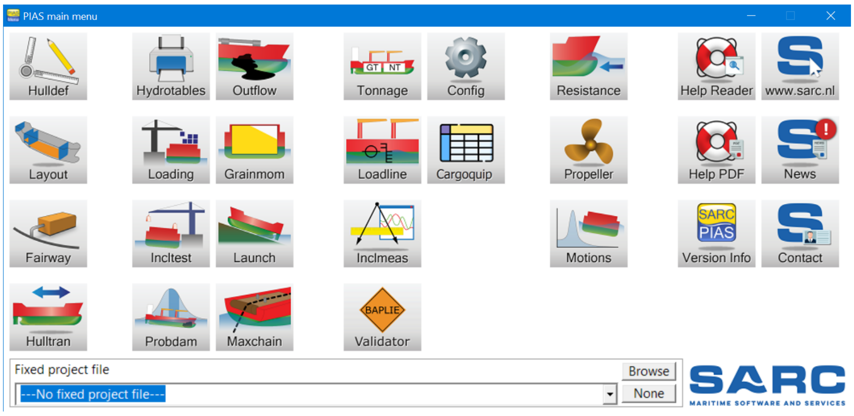
This section describes the PIAS main menu and how to start a project from there.

2.4.1 PIAS Main menu

PIASmenu shortcut.

When you start PIAS (with the PIASmenu shortcut on your desktop or, for example, by clicking **PIASmenu.exe** in Windows' Explorer), PIAS' main menu appears:

[5] https://www.sarc.nl/news
[6] https://www.linkedin.com/company/sarcbv
[7] https://www.sarc.nl/publications



PIAS main menu.

In the main menu, the following actions can be performed:

- You can start a module by double clicking the left mouse button or by hitting <enter> when the mouse pointer is above the module button. Each PIAS module opens in its own window, the standard MS-Windows commands can be applied, such as move, resize, minimize, maximize, restore etc. Also, the menus in upper bar of the module's window act in the standard MS-Windows fashion.
- By clicking the right mouse button, or <F1>, on a module or submenu, the help-reader opens the chapter corresponding to that specific module or submenu.
- At the bottom of the main menu you can specify a fixed project file, which is subsequently used by all modules. Providing a fixed file is optional, if no fixed file is specified each individual module asks for the file to be used. Next to this fixed file field, there are two more buttons: [Browse], this function can be used to browse your folders, and choose a PIAS files, and [None] which indicates you don't want to use a fixed project file.

The main menu consists of buttons which symbolize a module. Their purposes are summarized in the table below.

Hulldef
Input of hull-related items, such as an existing body plan, non-watertight openings, special points, appendages, deck line and wind contour.

Layout
Input or design of internal shape, such as bulkheads, decks and compartments. Computation of tank sounding tables etc.

Fairway
Hull form design, fairing, visualization and export.

Hulltran
Hull form transformation.

Hydrotables
Computation and output of hydrostatics-related tables, such as hydrostatics, cross-curves, maximum allowable VCG' (intact as well as damaged), deadweight tables and deadweight scale and Van der Ham's trim diagram.

Loading
Definition of loading conditions and computation of intact stability, longitudinal strength and (deterministic) damage stability.

Incltest
Inclining test or draft survey report.

Probdam
Probabilistic damage stability.

Outflow
Oil outflow according to MARPOL.

Grainmom
Maximum allowable grain heeling moments.

Launch
Longitudinal launching.

Maxchain
Maximum allowable anchor chain forces for anchor-handling vessels.

Tonnage
Calculation of gross and net tonnage.

loadline
Freeboard calculation according to the International Convention on load Lines.

Inclmeas
Registration and processing of digital inclination measurement.

Config
General project setups and configurations.

Cargoquip
Definition of cargo equipment.

Resistance
Re sistance predictions according to empirical methods.

Propeller
Propeller calculations with standard series, for B-series, ducted propellers etc.

Motions
Estimation of ship motions.

Help reader
A help reader (which is also accessible from each module by pressing <F1>), which shows directly the module-specific manual chapter, also see: section 2.2 on page 4, Manuals, exercises and information sources.

Help PDF
One PDF file which contains all chapters of the PIAS manual. A PDF-reader is required to open this file.

Version info
Print the same creation date and revision number of the current version of the software to a preview window, from which it can be printed on paper, or copied & pasted into another document, for example to a stability booklet in order to record the particulars of the software used for the calculations. Copy preferably as image, not as RTF because that can be interpreted differently by the various text editors, which might lead to small layout differences. While as image the entire content, including the rounded frame, is exactly transferred. An example is depicted below. An exclamation mark indicates there is an important update available or that the current version of the software is more than six months old. Because legislation, hardware capacities and opinions about program design the software is subject to permanent change, the PIAS software is frequently updated. Consequently, it is strongly recommended to install on a regular basis the most recent software versions, for which you can refer to section 3.2 on page 26, PIAS distribution.

www.sarc.nl
SARC website. `www.sarc.nl`

News
Newsfeed where important information is given on updates and changes in the software. An exclamation mark indicates there is an unread news bulletin. `www.sarc.nl/news`

Contact
Contact details.

These PIAS modules do neccesarily act strictly separated, the modules can be instructed to share their data in the background, in order to achieve that changes in one module, for example a hull shape modification, is directly processed in other modules, such as the stability calculation. This mechanism is baptized *local cloud* and is further discussed in section 2.11 on page 17, Local cloud: simultaneous multi-module operation on the same project.

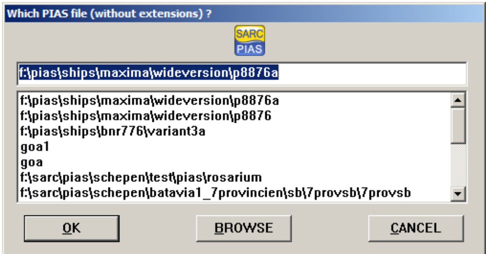
Finally, the question might be posed whether multiple PIAS modules can be used simultaneously. The answer is:

- Obviously, it is useless to invoke the same module twice. So, the PIAS main menu does not allow so.
- With the local cloud mechanism multiple modules share their data instantaneously, without the involvement of *files*. For this reason, multiple different modules for the same project can be invoked, which is exactly the reason of existence of the *cloud*.
- If the cloud is **not** in use, still multiple modules can be invoked simultaneously (which is unavoidable, because this must be facilitated for projects which do use the cloud mechanism). However, one should realize that in this case all modules read their data from file only once, at start-up, from file. So, later

changes from one module will not be processed in other modules. Errors are easily made in this case, so it is discouraged to do so.

- If it is desired to work on two different projects at the same time, the PIAS main menu can simply be started twice. The question whether this is convenient or confusing is left to the user.
- Besides, the different modules can also be started directly from Windows, after all they are simply independent executable programs. Except for very specific applications — e.g. in the context of a larger automated ship design system — this is strongly discouraged.

2.4.2 Working with PIAS' modules



File selection popup window.

When invoking a PIAS module, a *ship or project file name* must be given. For that purpose, a selection window pops up, as shown in the figure above, offering three options:

- Enter a file name in the topmost row. Here always the most recently used file name shows up as default, so that in general a simple <Enter> will suffice to continue with that file.
- Choose one of the twenty most recently used files, as depicted in the text box in the middle.
- Browse the directories to find your file.

Note

It is recommended to create a new subdirectory where PIAS stores all project files. This is not obliged, however, experience has shown this practice to be clear.

The ship or project file name which one is supposed to give is *excluding* extension and in principle *including* the name of the path (=folder =subdirectory). If no path is given at all, Windows will choose a path at own choice, which is presented to you for approval (and if you disapprove, by the way, then the module stops and can be re-invoked while including the desired path explicitly). More details on files and their management is given in section 2.10 on page 16, File conventions.

After choosing the file name the module main menu comes up. Its operation and input facilities are discussed in a separate manual chapter, which is chapter 4 on page 35, Operation of PIAS. It is recommended to take a quiet read at that chapter.

Note

If one wishes to avoid the labour of entering the filename each time a module is invoked, in the main PIAS menu a fixed file name can be specified, as discussed in section 2.4.1 on page 5, PIAS Main menu.

2.4.3 The availability of functions

By now it will be clear that PIAS is composed of several modules. Moreover, within such a module multiple functions can be accommodated, such as the longitudinal strength function which is hosted by the Loading module. For details reference is made to the `function list`[8]. Each PIAS user has selected a tailored package, which implies that only actually relevant functions have been purchased. It might be that in this manual functions will be discussed that are not part of your package, in that case the program reports "This function has not been purchased", or something similar.

[8] https://www.sarc.nl/pias/modules

2.5 Export of results

Output of PIAS can be exported to file (see section 5.1.10 on page 44, Output filetype) or to Windows' clipboard (please refer to section 2.8.3 on page 12, Print options for how to configure). The user can choose from a number of standard formats, viz:

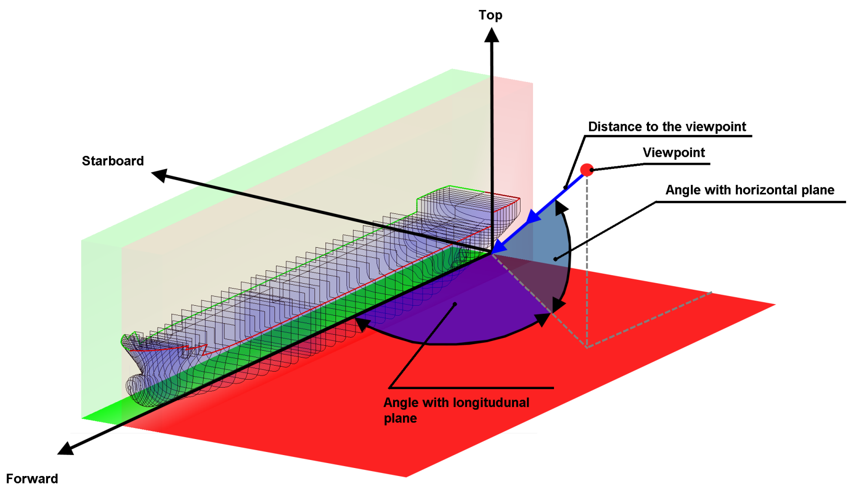
Text
This is the simplest output, just plain ASCII text, without drawings and without attributes for font types, font sizes, paper sizes etc. This format can be read by every text editor or spreadsheet program. However, many details will be lost.

Tabbed text
Almost identical to the Text format, albeit that multiple spaces have been replaced by 'Tab' characters. This enables some spreadsheet programs to separate multiple figures on one line into spreadsheet columns.

Image
With this format a graphical map of a page is generated. This map contains all font types, character attributes and of course all possible plots. However, the disadvantage of this format is that characters are not recognized by many receiving applications, so, for example, text modification with a text editor will not be possible in many cases. In this area MS Word is a little smarter than some other editors, because it recognizes the characters and enables their manipulation.

Rich Text Format
Rich Text Format[9] is a Microsoft standard, which is suitable to transfer documents between text editors, or to generate documents for text editors. RTF is supported by OpenOffice, MS-Word and Windows' Wordpad. With RTF the the output of PIAS can be sent to a word processor integrally, so, including plots and graphs.

2.6 Definitions and units

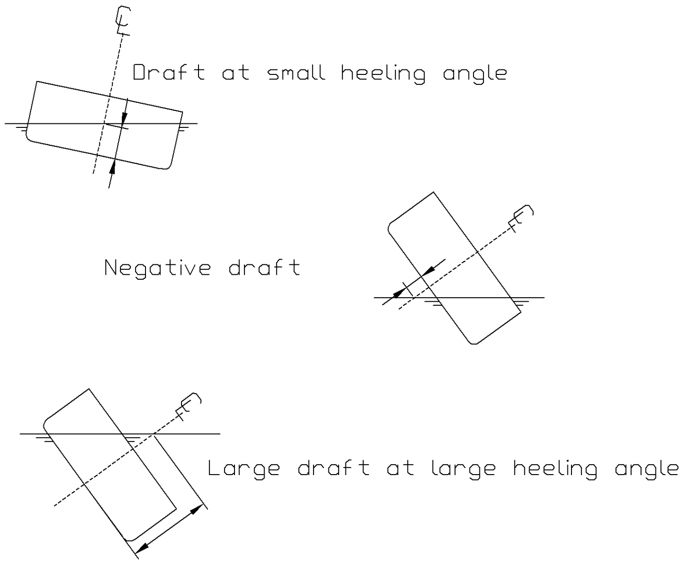


Definition of viewing angles in three-dimensional views.

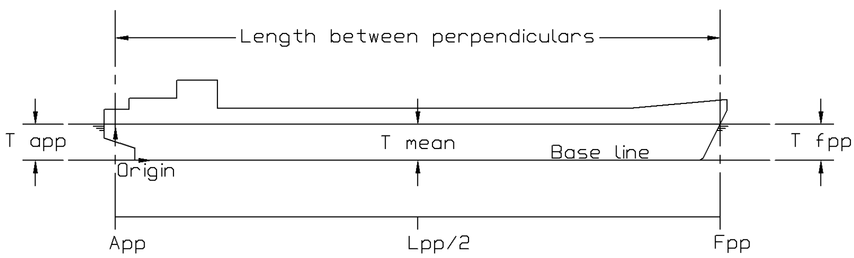
- Standard units of PIAS are metric.
- Threedimensional views are defined by means of the angles with the horizontal plane (from above is positive for angles $< 180°$) and with the longitudinal plane (from PS is positive for angles $< 180°$), as indicated in the sketch above. As a matter of feedback, in the case of the 3D hull view in Hulldef, and the 3D compartment views in Layout (there only **during** rotating) in the status line at the bottom of the window the numerical values of these angles are printed.
- Unless stated otherwise, all dimensions are in meters, volumes in m^3, weights in metric tons and densities (=specific weights) in ton/m^3.
- Breadths are measured from center line, positive to starboard, negative to portside.

[9] http://www.microsoft.com/en-us/download/details.aspx?id=7105

- Heights are measures from origin, positive upwards. The user is free to choose origin, in general it will coincide with base line, but it doesn't necessarily has to.
- Lengths are measured from a origin, which can freely be chosen by the user. Commonly, the origin will coincide with aft perpendicular, however, it might occasionally be convenient to put origin at the aft extreme of the ship. Anyway, the measurements are positive forward of the origin and negative to the aft.
- Length between perpendiculars can in principle be chosen freely, and is given in the main dimension menu in Hulldef or Fairway (see section 7.2.1.1 on page 167, Main dimensions and allowance for shell and appendages or section 6.4 on page 128, Define main dimensions and other ship parameters).
- Drafts are measured **in** center plane, as depicted in the figure below. At large angles this can lead to large positive or negative drafts, however, that is nothing special, it is just a consequence of the applied system of axes, a matter of definition.
- *Draft aft* is taken at the origin (by definition at a longitudinal position of zero meter). *Draft fore* is taken at forward perpendiculars.
- *Mean draft* is at *length between perpendiculars / 2*.
- *Trim* is the difference between *draft fore* and *draft aft*.

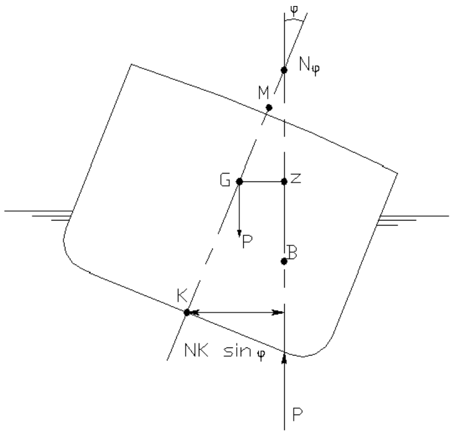


Definition of draft.



Definition of length and draft.

And finally, the stability lever. KN×sin(φ) is defined as the least distance between the vector of the buoyancy force (which intersects centerline at the false metacentrum **N**) and keel point **K**. As depicted in the figure below, where **B** is the center of buoyancy. Assuming that center of gravity **G** is located at centerline, the righting lever can be determined with the equation GZ = NG×sin(φ) = KN×sin(φ) - KG×sin(φ).

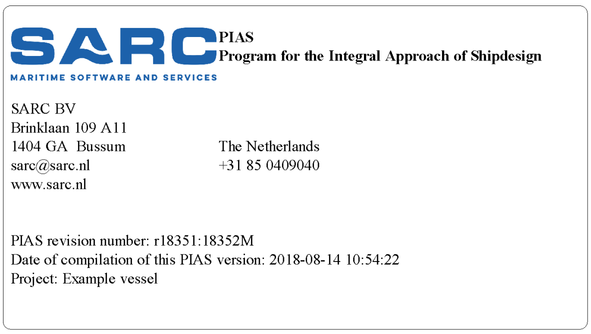


Convention of stability lever and KN.sin(φ).

2.7 Help

Each module contains the PIAS [Help] menu. This menu has five sub options:

- Open the *help reader* (can also be done by pressing function key <F1>), see also section 2.2 on page 4, Manuals, exercises and information sources).
- Open the entire manual in PDF format.
- Connect to www.sarc.nl.
- Show the creation date (compilation date) of the present module, as well as its version number (revision number). These data can be used to determine whether it is about time to download an update, please also refer to section 3.2 on page 26, PIAS distribution for that.
- Print the same creation date and revision number to a preview window, from which it can be printed on paper, or copied & pasted into another document, for example to a stability booklet in order to record the particulars of the software used for the calculations. Copy preferably as *image*, not as RTF because that can be interpreted differently by the various text editors, which might lead to small layout differences. While as *image* the entire content, including the rounded frame, is exactly transferred. An example is depicted below.


SARC MARITIME SOFTWARE AND SERVICES
PIAS
Program for the Integral Approach of Shipdesign

SARC BV
Brinklaan 109 A11
1404 GA Bussum The Netherlands
sarc@sarc.nl +31 85 0409040
www.sarc.nl

PIAS revision number: r18351:18352M
Date of compilation of this PIAS version: 2018-08-14 10:54:22
Project: Example vessel

Printout of program version and date.

2.8 Setup

Each module also contains the PIAS [Setup] menu. This menu contains the following options (by the way, not the whole lot is relevant to work properly with the software):

- Project Setup
- Program setup
- Print options

- Screen Fonts
- Default Fonts
- Menu colors
- Restore column layout

2.8.1 Project Setup

Here the setup for the specific ship or project of the contemporary opened file can be given. These options are discussed in detail in chapter 5 on page 41, Config: General project configurations.

2.8.2 Program setup

By means of menu bar option [Setup]→[Program setup] a number of program configuration choices can be set. This setup is valid for entire PIAS — not just the project at hand — and are stored and used on the computer where it is made. Options are:

Maximum number of processors to be used by PIAS.
Many of the computationally intensive tasks of PIAS are distributed across multiple processors (or *cores* that will be used as a synonym here), please refer to the details thereof at section 3.11.1 on page 32, PIAS/ES 1: dualthreading and section 3.11.2 on page 32, PIAS ES 2: octothreading. That's fine, because that reduces the calculation time. Unfortunately, MS-Windows treats the load distribution over the various processors somewhat clumsy, with the result that if all processors work hard on PIAS, occasionally, other tasks hardly can come between. That makes the computer to react a bit sluggish. The statement that this does not necessarily **has** to be so — and that such *thread scheduling* in Unix in 1985 was better arranged then under MS-Windows today — does not bring us forward. So, a practical solution was implemented where a user can set the number of processors to be simultaneously assigned to PIAS. That number can be specified here.

Icon size in toolbars.
With this selector the toolbar button format is specified. This is only applicable to icons from the Windows-managed toolbar, as located directly under the function button bar, at the top of each window. All other icons or buttons are not affected by this selector.

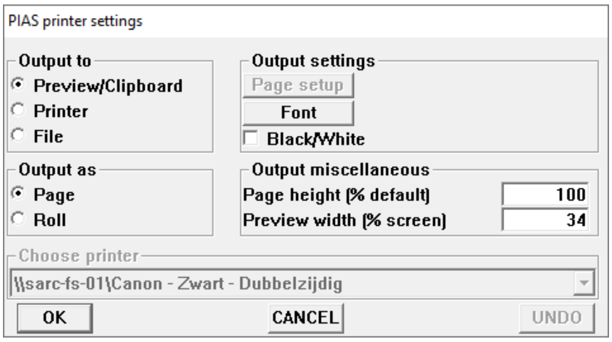
Keyboard interpretation Edit.
With this choice (which is the default) the keyboard operation is similar to that of an ordinary spreadsheet. Cell values can then directly be entered. Top bar menu commands such as [Insert] — as far as not activated with the mouse — should be given in conjunction with the <Alt> key.

Keyboard interpretation Command.
An occasional person prefers to navigate by keyboard without directly entering values in cells. For those, this setting is available, which makes that a cell value can only be entered after the <X> command (from e**X**change) has been given. The advantage of this setting is that top bar menu commands keys need not to be combined with the <Alt> key. In some cases, a follow-up action of PIAS depends on whether the first character in the cell is a letter or a digit. For those cases in which this digit-effect is desired, the combination <Shift><X> can be given, in which case directly a digit (0) is inserted. This *Command* interpretation of the keyboard is only rarely used and is not actively maintained anymore. This option is expected to disappear in due time.

Communication parameters interoperability.
These settings are experimental for the time being, and therefore not for general use.

2.8.3 Print options

Opens a popup which contains several options to modify the way output is handled within PIAS.



Print options.

Output to

Where to the output is directed:

- *Preview/clipboard*, Which offers a preview on the output and a facility to copy the output to clipboard. When using the clipboard, its menu bar contains several functions:
  - *Prev and Next*, The operation of these buttons speaks for themselves. Navigating through the pages can also be achieved via the keyboard with the <PgUp> (or left arrow) and <PgDn> (or right arrow) key respectively. Using the above mentioned keys in combination with the <Ctrl> key one can be quickly navigated to the first or last page.
  - *Go to page*, This allows for quick navigation to a specific page. Can also be invoked by pressing the <G> key on the keyboard.
  - *Copypage*, Copies the current page to clipboard. In other Windows applications, such as word processors or spreadsheets, this page can be imported with the [Paste] option. The page can be copied to clipboard in several formats, please refer to section 2.5 on page 9, Export of results for a discussion.
  - *CopyAll*, Copies all pages of the output to clipboard.
  - *Print&Quit*, Print yet all output, and close the preview window.
  - *Quit*, Close the preview window.
- *Printer*, One of the installed printers as configured by Windows *Devices and printers* option (or whatever its name will be in your Windows version).
- *File*, Sends the output to the specified file, which can be defined at section 5.1.10 on page 44, Output filetype.

Output as

For the **Output to** types *Preview/clipboard* and *Printer* the output can be printed in the following styles:

- *Page*, Where a page length of approximately A4-size is assumed, which makes a longer output to be subdivided over the sheets of paper. This is the default style within PIAS.
- *Roll*, Which does not subdivide the output because of the paper size into pages. With this option it is assumed that the printer has an endless roll of paper (where, by the way, the *application* can still decide to start on a new page, but that is because a new subject asks for it, not because of the paper format). This style is intended for exceptional situations — for example if the output should become available to other software without intermediate headers, or if one wishes to create a personal page distribution.

Choose printer

Select the printer to be used when the **Ouput to** has been set to *Printer*.

Output settings

Some general output settings:

- *Page setup*, Choose this option to open the page setup menu of the printer as selected under *Choose printer*.
- *Font*, Modify the printer font size.
- *Black/White*, Activate this option to print output in black and white.

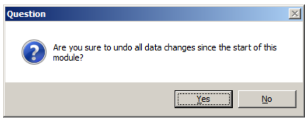
Output miscellaneous

A few miscellaneous settings:

- *Page height (% default)*, Output to printer can be formatted quite neatly, because PIAS can communicate with the printer about the measurement details, such as font size and paper size. However, output to preview/clipboard and to RTF is often pasted into a word processor, from which these details are not available to PIAS. For that reason, PIAS uses a standard page height for 'standard' preview/clipboard cases. With this option this height can be tuned, with a value that is a percentage of PIAS' standard. So, when using 100 the PIAS standard is applied.
- *Preview width (% window)*, With this option the width of the preview window can be set, relative to the active window.

2.8.4 Screen Fonts

Adjust the fonts on screen to your preferences. With this option the font type and font size of the primary windows, which are managed by PIAS, are set. However, with this option the font properties in a *popup box* are not affected, because those are being managed by Windows. That particular font size may occasionally be somewhat small, see the example just below, but can be configured in Windows. With Win 7 use *Control panel*, *Personalization* and finally *Windows color and Appearance* to do so. With other WIndows versions this setting might be in a different location or have a different name.



Popup box, for which the font properties can be set in Windows.

2.8.5 Default Fonts

Restore the fonts for printer and screen to their default settings.

2.8.6 Menu colors

Here, the colors can be set as they are used in the selection windows and input windows, such as discussed in section 4.1 on page 35, Selection window and section 4.2 on page 35, Input window. De sub options here are:

Make standard color scheme
This sets all colors to their default setting.

Lines between cells
Set the color of the lines between the cells of the selection windows and input windows.

Cell options indication
In section 4.3 on page 38, Content and options in the cells of selection windows and input windows it is explained which symbols are used to indicate the cell options. Their color can be set here.

Lines around the text cursor row
The selected cell of the moment is indicated by a color, this is the text cursor. Because rows may be long, users have expressed the desire to indicate the entire line of the text cursor. This is done by means of pronounced line around it. Its color can be set out here.

General background color
Of the empty window.

Foreground and background font colors
For a number of text categories (which are supposed to speak for themselves), foreground and background colors can be chosen here.

2.8.7 Restore column layout

The widths of columns and cells in input windows have default values which are tuned to the anticipated largest size of the cell contents. If desired, a user can adjust the widths, and the column order as well, as is discussed in section 4.2 on page 35, Input window. With this function, [Restore column layout], those manually configured column properties are being restored to their defaults, either for all menus of PIAS, or just for the present menu.

2.9 Data storage and backups

A premise of PIAS is that a user enters data with the aim to keep them, which therefore implies that when leaving a module all input data is stored on disk, which requires no separate action. Also, while being used modules will save their data on a regular basis, the advantage is that in case of unexpected failures fairly recent data is still available. In addition, some modules provide an exact data storing time interval. If the data should **not** be stored, the module can be quitted by the *File and backup management / Quit module without saving the data* option, as discussed below (section 2.9.5 on the next page, Quit module without saving the data). Additionally, (most of) the PIAS modules have a backup system which works with their specific files, but has a generic way of operation. A typical menu for this functionality is:

File and backup management

1.	Save data on disc
2.	Create backup
3.	Restore data from backup
4.	Import data from another project
5.	Quit module without saving the data

2.9.1 Save data on disc

As discussed, the PIAS modules save their data on a regular basis, and do that for certain on exit. But one can at some stage also have the need to make sure that everything is stored in between, which can be achieved with the present option.

2.9.2 Create backup

With this option a backup copy can be made from all data as managed by the particular PIAS module. With this backup the time and calendar date are also saved. Additionally, a window appears, where a description of the backup can be given, which will be saved together with the backup.

2.9.3 Restore data from backup

Here a list comes up which shows all backups that are available on the ship directory (which is the folder which contains the files of the current project). This can be used for three actions:

- In the third column the first line of the backup description is presented. With an <Enter> on that field the whole description appears, which can be modified if desired.
- With [Remove] the highlighted backup is permanently erased.
- With [restOre] the data of the highlighted text cursor are being restored. It speaks for itself that all current data of the current project are brutally overwritten. Because that is not always the users' intent, two precautionary provisions have been made:
  - When using this [restOre] first all available backups are compared with the present data. If this shows that a backup exists which is a copy of the present data, then there is no risk of losing data, because this copy can always be restored. However, if this is not the case it might be wise to make a quick backup from the present data, so, if program detects this situation it shows a popup box offering this option. By the way, please see that this comparison is very accurate, a copy should be perfect to be recognized as such. That means that for example a minor setup modification (if stored) will result in the backup not being seen as a copy.
  - In the 'Duplicate' column the user can see for each backup whether or not it is such a copy. For clarity: this means that the backup is identical to the present data, however, it still remains a standalone copy without any further link to the module data.

Furthermore, in this window also a greyish backup might be visible. That is an automatic backup as produced by the module at start-up, which only use is in combination with option Quit module without saving the data (discussed on the following page). This backup cannot be utilized for other purposes.

2.9.4 Import data from another project

The previous option is intended to restore data from the current project, with this option data from another project can be imported. After selecting this option, a file browser appears which must be used to select the desired backup. It can be that this backup contains multiple data categories (e.g. 'frames', 'openings' or 'wind contour'), in which case the user is asked which category or categories to import. By the way, with this import action all current data (of the appropriate category) are replaced, so they are not added or something alike.

2.9.5 Quit module without saving the data

This option works as it suggests; the current PIAS module is closed, and the involved files are restored to the state at the start of the module. All intermediate changes and actions are being discarded with this option, that does not only apply to 'typed in' changes, but also for restoring or importing backups, and for automatic data saves. However, an exception is made for **deliberate manual save actions** (e.g. with section 2.9.1 on the previous page, Save data on disc), which constitutes a new starting point, and makes it so that with the present [quit without save] action the files are restored to the content of the moment of this deliberate manual save.

2.10 File conventions

2.10.1 Files and extensions

PIAS distributes shipdata or projectdata over many files, where each contains a separate type of information. In the vast majority of these files the information is stored binary, so they are not readable by a human. The files start with the project file name and end with an extension chosen by PIAS, which indicates the type of information. Occasionally, users have asked for a list of file extensions, and although such an overview existed, in 2013 it was withdrawn for two reasons. In the first place many of these files are mutually associated, so data consistency might be jeopardized if files are replaced at random. And, secondly, it is no longer necessary to exchange individual files because with the *backup* facility — which has been discussed just above — all information which is managed by a particular module is combined into a single file which can be imported elsewhere. By the way, if *all* data of a project must be transferred or backed up then nothing is easier than *zipping* — compressing — into a single file. Furthermore, files are in principle compatible between all PIAS versions, although there are three exceptions:

- New PIAS versions often have more capabilities, which sometimes require an enhancement of the file format. So, the files have to be converted than, which remains unnoticed by the user because it happens automatically. So PIAS versions are *upwards compatible*. However, if files are subsequently transferred and used in an elder version then this will obviously not recognize the new format. The remedy is rather simple by replacing the elder version with an update.
- Although files of the LOCOPIAS loading software originate from PIAS, they are encrypted and consequently no longer usable in PIAS. This is deliberate, in order to ensure the file integrity of LOCOPIAS.
- PIAS versions supplied to educational institutes could also (deliberately) be made incompatible with other PIAS versions. The reason is to discourage the use of an educational license for commercial work.

In the last two cases it is yet possible to convert the files, so they can still be reused. You can contact SARC for this purpose.

2.10.2 Hull form representations

In PIAS, different hull shape representations are used for different purposes. These describe, obviously, the same geometry, but in a different way. The table below presents an overview.

Representation	Contains	Purpose	File extension
Frame model	Cross sections (frames).	All volumetric and hydrostatic computations, (damage) stability, tank sounding tables, longitudinal strength, etc.	Until 2019 .hyd, then .frames
Curved surface model, alternatively *solid model*	The closed, curved hull surface.	For Fairway.	.fr∗
Triangulated surface model	The closed hull surface, represented by many small flat triangles. Generated by Fairway.	For plots to the closed hull shape in PIAS, exceptionally for a (e.g. tank sounding) computation.	.tri
Wireframe model	3D curves from the ship hull surface.	Importing 3D geometry (e.g. as exported by other CAD software) in Fairway, please refer to paragraph 6.3.7.1.3 on page 116, Import ship hull models in SXF/CXF format for that.	.sxf and .cxf

- At first sight it may seem confusing that the second representation has two names. This is because it can be seen from two points of view: for a user, the image is what counts, and that is the curved hull surface. Internally, in Fairway, however, it is embedded in the structure of a *intrinsically closed model* which offers possibilities for unambiguously finding intersections etc. Depending on the focus of the program or manual in which we are, one or the other term is used.
- In Config it can be set whether PIAS computates on basis of the frame model, or with the triangulated surface model, please see section 5.1.5 on page 43, Preferential format of hull files.
- In section 7.1 on page 165, Hulldef's hullform definition method details and consequences of the frame model are discussed.
- Although it is not customary to name file extensions from the various PIAS data files, they are still included in the table, because some will have grown accustomed to these extensions. However, please realize that these files cannot be used separated from PIAS.

2.11 Local cloud: simultaneous multi-module operation on the same project

Ship data are saved on file, however, an additional communication system has been developed, which is baptized *local cloud*. This facilitates inter-communication between PIAS modules, without using disc files, and without the user involvement. The advantage is that the effect of modified input on a calculation result can directly be made visible. Three examples:

- When a user has the screens of the new PIAS modules form input and loading conditions open, with the last one showing the bar chart of the stability index, then a modification of, for example, the height of an opening in module form input is directly translated to another stability index in the loading conditions module. That may change, for example, from red into green.
- If the modules Fairway and Layout are simultaneously open, then it can be seen that a hull form modification in Fairway is directly processed in Layout. We have made a short video clip of this example[10], in which you can see how a hull form modification in Fairway is converted in a modification of the form of the tank top in Layout.

[10] https://youtu.be/LUfbpjprrfs

- When designing a hull form with Fairway, with the so-called 'local cloud monitor' of Hydrotables for instance volume, LCB and maximum allowable VCG' can be shown on the fly. An example is presented in section 10.7 on page 261, Activate the Local cloud monitors.

2.12 Frequently asked questions

In this manual it has been pursuit to elucidate *per module* its background and modus operandi. However, an occasional question of general nature appears to arise, which will be discussed in this section.

1. **Does PIAS also work on 64-bits Windows?**
   Yes. Please see the screen dump at section 2.1.2 on page 3, System requirements.
2. **Is a student version of PIAS available?**
   Yes and no. An educational institute can for educational purposes, under certain prerequisites, acquire a campus license under favourable conditions. Unfortunately, due to administrative and technical constraints individual student licenses of PIAS cannot be distributed.
3. **Is PIAS also available for Apple Mac?**
   PIAS is not available natively for the Mac. A Mac can be configured to emulate or run Microsoft Windows (possibly in a virtual machine), which might offer the ability to run PIAS (although PIAS will then not even be aware of the Mac basis).
4. **During computations the processor is not working at 100% performance**
   In the first place there is the possibility that other tasks or processes prevent the processor from working at full throttle. More frequent, this question is posed when working on a multi-processor (or multi-core) machine, while assuming that the computation task will automatically be distributed over the multiple processors. Unfortunately that is not the case; parallelisation — which is the name of this game — must be implemented for each and every task separately and explicitly. For PIAS various multi-threading facilities are available, as discussed in section 3.11 on page 32, Speed enhancing mechanisms in PIAS: PIAS/ES.
5. **When re-computing probabilistic damage stability, occasionally slightly different damage boundaries are found then the first time.**
   That can happen, please refer to section 22.4 on page 433, Computing the damage boundaries.
6. **In the PIAS output of the probabilistic damage stability, the grouping of probabilities per damage case (*p*, *r*, *v* and *s*) differs slightly from the regulations.**
   Correct observation, the grouping in PIAS is better arranged, as discussed in section 22.3.5.1 on page 432, Execute and print the calculation. Anyway, this is a formatting issue, the results are not affected.
7. **With the (sub-)compartment method from the probabilistic damage stability, for many damage cases the boundaries are found rather quick, while an occasional other case requires quite some time.**
   The reason is that for the determination of the damage boundaries, Probdam applies in first instance classical search algorithms, which operate fast. If, occasionally, damage boundaries are not found with these algorithms then an alternative method is used — a so-called genetic algorithm — which has the advantage that it often succeeds in difficult search tasks, however against the price of a significant computation time. Please also refer to section 22.4 on page 433, Computing the damage boundaries.
8. **During lengthy calculations the program appears to 'hang', with occasionally the Windows message that the program is not responsive anymore.**
   This message is incorrect. Windows is aimed at communication and user interaction, however, during a long computation there is no interaction between an application program and the operating system. Windows then draws the erroneous conclusion that the program is not responsive, however, in the 'task manager' it can easily be seen that the processor is quite busy computing (which is just what computers have been made for).
9. **I am not able to relocate an existing physical plane — in Layout, to define internal geometry — to every location.**
   That may very well be, the relocation possibilities of physical planes are limited in order to maintain the 'logic' of the compartment layout. Please refer to section 9.1.6.3 on page 201, Limited positioning of a physical plane for details.
10. **In 3D wireframe models of hullform or compartment, PS and SB sides seem to be swapped.**
    That will be a matter of optical delusion, against which a tool is available, see section 7.8.1 on page 190, View.
11. **Occasionally, text characters are represented improperly, or an open rectangle is drawn instead.**
    This happens probably with a character that does not belong to the familiar ASCII standard, such as a Greek

symbol, or a letter with a diacritical mark. The PIAS output may contain such output — notably with output in French or German — however, that is only represented well if the chosen font indeed contains these characters. And unfortunately not each Windows font contains each and every character. So, if these characters are not represented properly, then you will have to select another Windows font — in PIAS, or in your word processor or spreadsheet if it happens there. Selecting a screen font is discussed in section 2.8.4 on page 14, Screen Fonts, and for the printer font reference is made to section 2.8.3 on page 12, Print options.

12. Stability criteria.
Answers to a number of questions regarding stability criteria are given in section 15.6 on page 301, Answers to frequently asked questions on stability assessments.

13. The trend of waterline properties (such as area or moment of inertia) against draft, in a hydrostatic table, is a bit jerky.
That might occasionally occur with pontoon-like hulls, but in general does no harm. Explanation and remedy are discussed in section 7.2.4.1 on page 174, Number of frames.

14. I know that a body plan available in .BMP beatan format can be digitized in PIAS. Can this also be done with other formats?
Please refer to the first paragraph of paragraph 7.2.4.7.3 on page 179, Defining the section shape by digitizing a BMP file.

15. My virus scanner reports a PIAS file to contain a virus. What to do?
Some scanners do indeed wake up on an occasional PIAS file, but as far as SARC is aware this has, to date, always been a false alarm. Obviously, this does not guarantee that such a warning will always be false, but it is primarily the responsibility of your scanner supplier, who manages the algorithms and their data after all. So, if you would like to assist them, feel free to inform them. SARC cannot provide support because there are too many types and brands of scanners around. Two final comments:

- Sometimes a scanner thinks it a good idea to delete some components of PIAS, e.g. a .dll file. It goes without saying that PIAS will then no longer work properly.
- At SARC, all files, including PIAS packages intended for customers, are systematically tested for viruses and malware (with *ESET Endpoint Antivirus*).

16. PIAS refuses to start, with error message "The program can't start because MSVCR120.dll is missing from your computer. Try reinstalling the program to fix this problem.".
This is an error message from Windows, and indicates that an essential part of the Windows installation is missing or has become corrupted. This is about "Microsoft Visual C++ 2013 Redistributable" and (re)installing of this is neccessary: By following `this link`[11] the x86(32 bit) and x64(64 bit) versions can be installed.

17. A waterline or buttock in my GUI or in a plot on the output sometimes has an incorrect shape.
In general, this is not a cause for concern because these lines are not used for any calculation. section 7.1 on page 165, Hulldef's hullform definition method contains some more background information of this phenomenon.

18. PIAS works well on my computer, however, everything is so sluggish.
If your PIAS is equipped with the Codemeter software protection then this **could** be caused by the fact that the Codemeter location has not been set. In effect, this causes the system to search the entire network for it. The installation procedure that prevents this is described in section 3.1.1 on page 21, Codemeter installation.

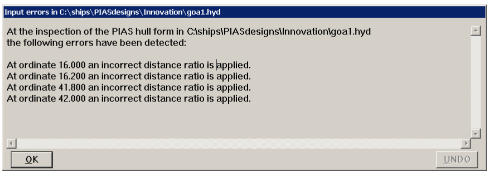
19. Calculating probabilistic damage stability may take quite some time. What can I do about that?
SARC has spent much time and efforts to optimize Probdam on lead time. Still, there are some tips that users can take to heart:

- Take advantage of the highest possible level of *multi-threading*, as available in PIAS under the name PIAS/ES, see section 3.11 on page 32, Speed enhancing mechanisms in PIAS: PIAS/ES.
- Minimize the number of *external compartments*, see section 22.2.2 on page 417, External compartments.
- With PIAS it is very easy to generate large amounts of damage cases. However, they will all have to be calculated, while from ever larger damage cases their contribution to the attained subdivision index (A) is slowly approaching zero. So, restrict yourself when generating damage cases, e.g. by setting the number of compartments per damage case (see paragraph 22.3.1.1.20 on page 424, Maximum number of damaged (sub-)compartments per damage case) high only when really necessary.
- Use the numerical integration method instead of the compartment method. See paragraph 22.3.1.1.3 on page 420, Calculation method probability of flooding for a discussion of the various methods available

[11] `https://support.microsoft.com/en-us/help/2977003/the-latest-supported-visual-c-downloads`

in PIAS for determining the probability of damage.
- Use the zonal method instead of the compartment method. Unfortunately, in terms of turnaround time, you won't gain anything because the time saved by the shorter zonal calculation time will be more than outweighed by the higher number of person-hours required for manual definition of damage boundaries — while, in addition, the zonal method in general yields a lower A than the other methods available in PIAS — but then again, the question was limited to the calculation time as such.

20. When invoking a PIAS module, with a file that already contains a ship hull model, I receive warnings about frame spacing ratios, such as in the figure below. What should I do with that?
Take such warnings to heart, because the way this hullform has been defined might decrease the computation accuracy. Please also refer to section 7.2.4.2 on page 174, Ratio of longitudinal frame distances.



Warning on frame spacing ratio, when invoking a PIAS module.

Finally, it is possible that the program gives an (error) message which is not crystal clear at a glance. In such a case, please proceed as follows:

1. Plain, English, messages are assumed to be clear, it might help to visit the manual chapter on the subject which deals with the message. A message with program codes or addresses have no external meaning, even for SARC this is only useful by inspecting the program *from within*.
2. Upon each report that a message is not clear, the program will be modified. So, there is a good chance that with the latest PIAS version of the message disappears, or is put more clearly. Therefore it is advisable in any case to install the latest update, see the next section for that.
3. If that does not solve the issue, then you can contact SARC. However, nothing can be undertaken without having the disposal of your data. So please send an email with the text of the program message, the module and all PIAS project files — compressed in a ZIP file or similar — to `support@sarc.nl`.

2.13 Closing remarks

This chapter ends with some good advices, the terms of usage including copyright notice:

- Because legislation, hardware capacities and opinions about program design are subject to permanent change, the PIAS software is frequently updated. Consequently, it is strongly recommended to install on a regular basis the most recent software versions, for which you can refer to section 3.2.1 on page 26, Distribution channel.
- It is recommended to make a backup of your project data on a regular basis.
- PIAS uses the computer system date and time to check on any changes made since the last session. It is therefore of vital importance that date and time of your computer are set correctly, see section 3.4 on page 28, Computer clock.
- **Up-to-date knowledge of naval architectural backgrounds and practices is required for a responsible use of PIAS.**
- **When PIAS and this manual refer to requirements from e.g. (inter-)national legislation or classification societies, it is not meant to replace the source texts of those criteria.**
- The software described in this manual is furnished under a licence agreement. The software may be used or copied only in accordance with the terms of that agreement. The software is protected by the copyright laws which pertain to computer software, it is illegal to make copies of the program or this manual other than for the use or backup by a legitimate user. Copyright (© 1993-2025) of software and manual is held by SARC BV. The last chapter of this manual contains the license terms of PIAS.

Chapter 3

Installation details

In this chapter additional functions and properties of PIAS are described. A common user can safely skip this chapter, however, in quest of installation details or specific functions one could benefit from reading this material.

3.1 Additional installation and update details from the Codemeter software protection

In section 2.1.3.1 on page 4, Codemeter an introduction is given to the Codemeter software protection. This chapter discusses its installation and update procedure.

3.1.1 Codemeter installation

PIAS is protected by means of a licensing mechanism using Codemeter dongles from Wibu-systems (=hardware lock). The dongle can be used directly in a local PC, or in a PC in the network/LAN (server) and the licenses are available to all computers which are part of this LAN (clients). A driver is required, to enable communication between (the installed) PIAS software and the dongle. The installation of this driver is described below:

- Download the `dongle driver (Codemeter Runtime system)`[1].
- Select *CodeMeter User Runtime for Windows*.
- Install the downloaded software on [a] the network PC acting as license server and [b] the client computers running PIAS. Please note that [a] and [b] can be physically the same PC.
- If [a] and [b] are different computers then proceed as follows:
  - After installation, insert the dongle and open the Codemeter Control Center on [a] on the right lower side of the window click Webadmin.
  - In the WebAdmin click *Configuration Server*.
  - Under section *Network Server* select Enable.
  - Click Apply. Note the IP adress or Hostname of the Server. Now open the Codemeter Control Center on [b]. Again, on the right lower side of the window click Webadmin. In the WebAdmin click *Configuration, Basic*. Under section *Server Search List* click *Add*.
  - Type the server IP address or hostname (obtained in the previous step) and click *Apply*. . If this installation is not done, the Codemeter system itself might still be functioning, although every time communication with the lock is needed the entire network is searched for it, which can make the program operation rather slow. It is therefore advisable to go through these steps carefully.
- If [a] and [b] are physically the same computer then take these steps:
  - After installation of the codemeter runtime system, insert the dongle and open the Codemeter Control Center on the PC.
  - On the right lower side of the window click Webadmin.
  - In the WebAdmin click *Configuration, Basic*.
  - Under section *Server Search List* click *Add*. Type 127.0.0.1 (or “localhost”) and click *Apply*. On client request the licensing system might be equipped with a (server-specific) licensing file, instead of a dongle. This does not change the installation procedure (apart from the fact that the dongle is lacking).

[1] `https://www.wibu.com/nl/support/user/user-software.html`

3.1.2 Codemeter license update

Licences can be remotely switched on, off or added by SARC (temporarily). This mechanism works as follows: A PIAS user first creates a *license requests file*. The user then sends this file (by email) to SARC. SARC uses this request file to create a *license update file* and returns it (by email) to the PIAS user. This file allows the user to update the licenses. The exact procedure is described below:

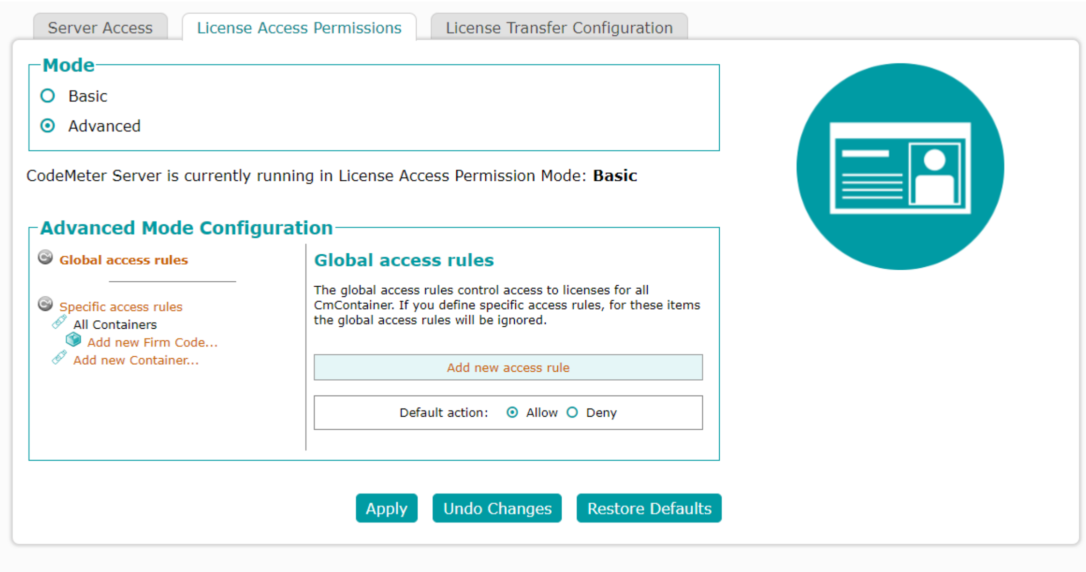
- Open CodeMeter Control Center (CCC) on the computer with the license (stick or CMact) installed.
- Select the stick or CMact to update. Now click “Update License" button. - Click “Next".
- Click “Create license request".
- Click “Next”.
- Select “Extend existing license”.
- Choose Vendor “SARC”.
- Click “Next”.
- Note/modify the file location the license request file will be stored (extension .wibuCmRaC).
- Click “Commit”.
- Send the request file to SARC.
- Based on this file SARC can create and return an update file (extension .wibuCmRaU) and returns the file to the PIAS user.
- Drag this file in the CodeMeterControl Center and the license will automatically update. After the update a dialog confirms if update of the license was successful.

3.1.3 License access management

The current set-up of licenses allows users to freely choose which modules are needed and which are not. However, if several licenses are available but only 1 license of some modules is available, it may happen that it is in use. SARC has done their best to avoid this as much as possible, but sometimes there is no other way.

When this is an issue, the license access can be managed by use of the Codemeters WebAdmin. Access to a specific license can be arranged per user, Domain group, etc. by allowing or denying access. Below it is explained how this is done. The complete manual can be found online at Manuals & Guides page[2], under *Manuals for Users* , in *CodeMeter Administrator Manual* .

1. Open CodemeterControlCenter
2. Click WebAdmin at the bottom right.
3. Go to Configuration, then Server and click on License Access Permissions.
4. Select the ‘Advanced’ mode.

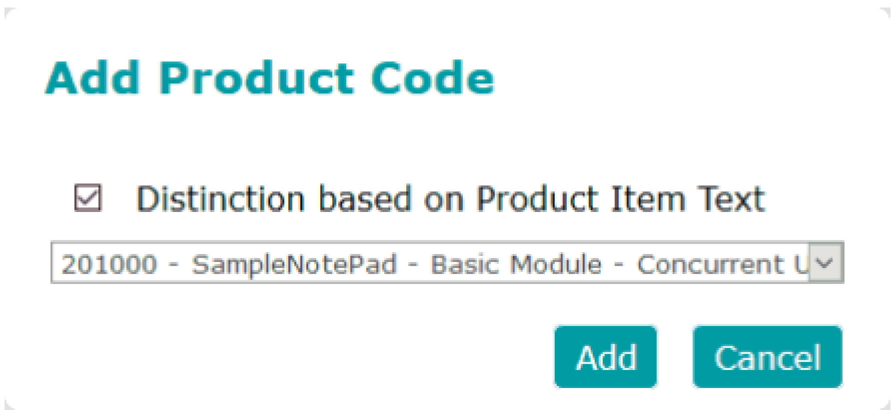


[2] https://www.wibu.com/support/manuals-guides.html

In order to create specific access rules to control license access to separate Product Codes, please proceed as follows:
Creating a Product Code-specific access rules requires a previously created Firm Code-specific access rules.

- Select the "Specific access rules" item in the left tree-view.
- Click the "Add Product Code" button for a specific module or click “Add new container” for a complete license.

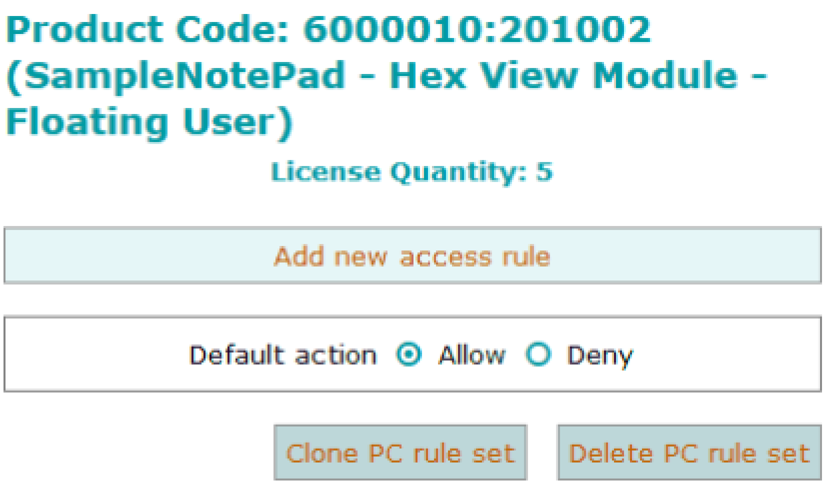
The dialog for selecting a Product Code displays.



Checkbox "Distinction based on Product Item Text".

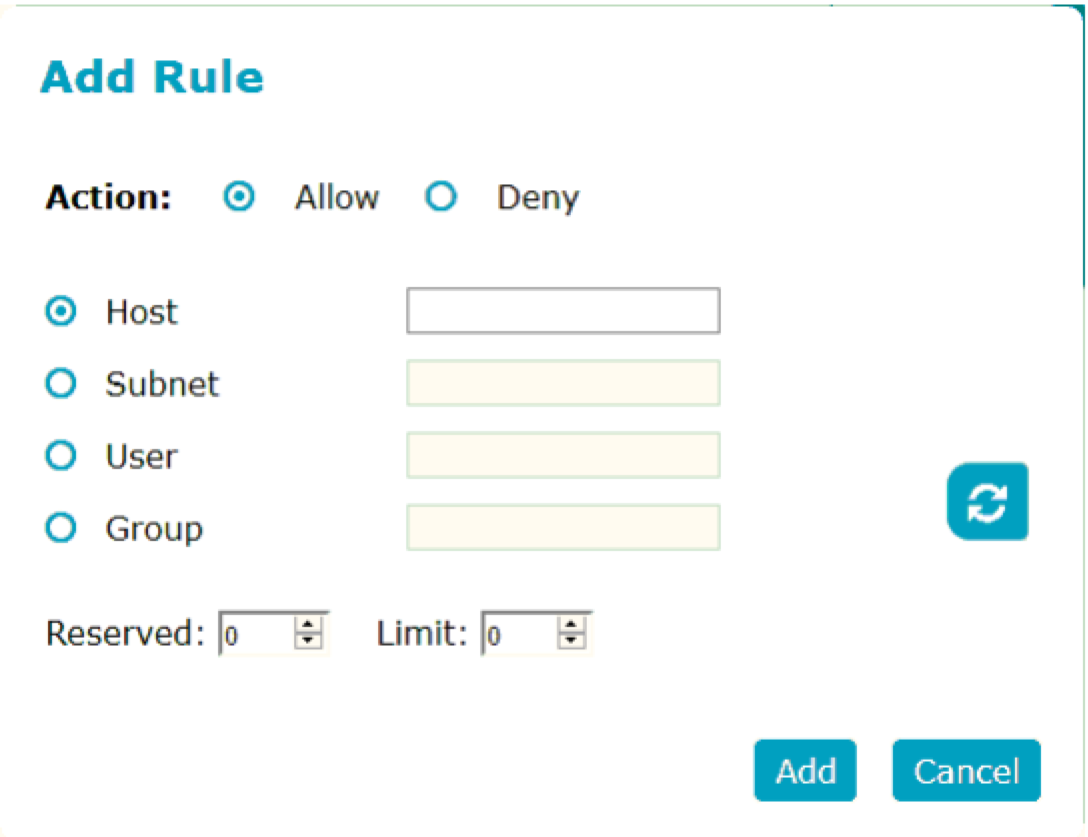
- Select the Product Codes and click the "Add" button.

A new specific access rules valid for this Product Code displays in the right rule view. At the same time, the entry displays information on the License Quantity, i.e. the number of concurrent licenses on a network. This number is not to be exceeded, if later defining limits to the number of accesses.



- Click the "Add new access rule" button.

A dialog for defining a new rule displays.

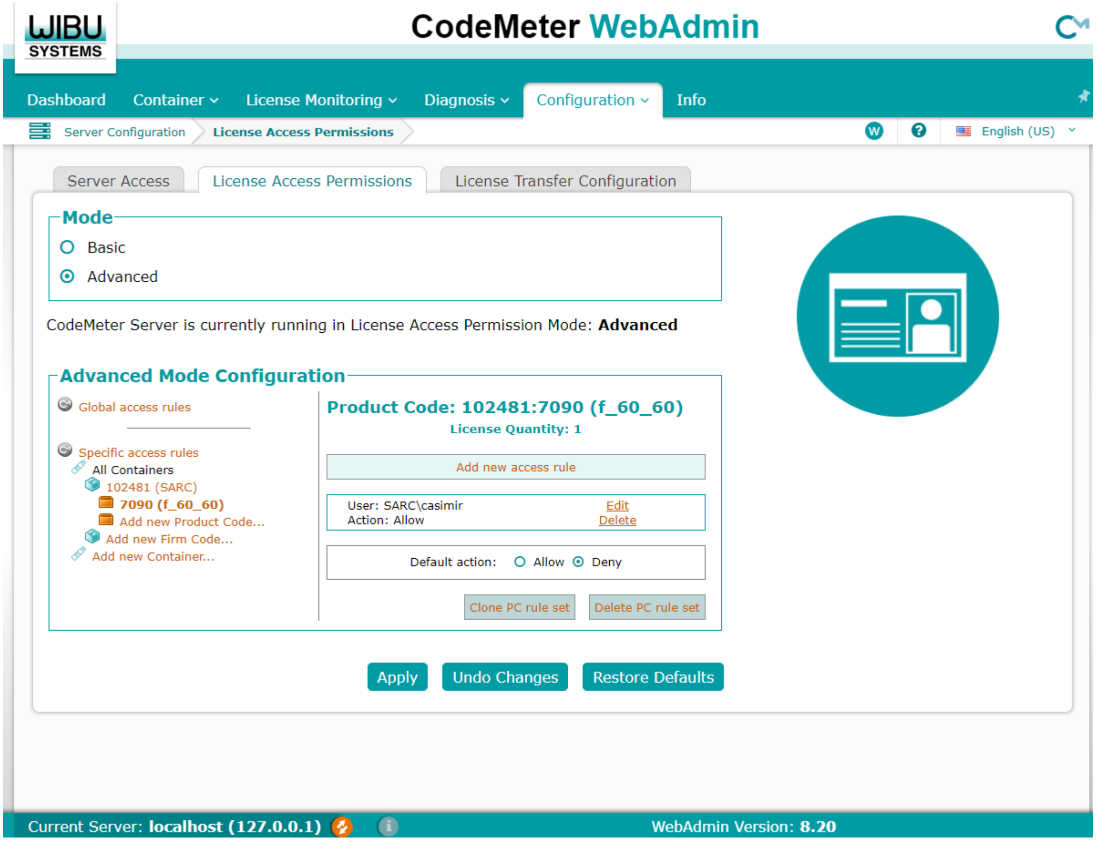


- Click the "Allow" or "Deny" radio button in the area Action to decide whether the following license access by client is to be allowed or denied. A client access can be defined by one of the following parameter: Computer name, Subnet address User or Group name.
- Specify the desired parameter in the respective field. If an active directory is connected, a separate button enables automatic filling of the User and Group fields.
- Specify the number of license accesses which can be optionally reserved for a defined client in the field Reserved. The field Limit states the allowed maximum of allocated license accesses by this client. The setting for a reserved license access always available for the client is: Reserved: 1; Limit: 1.
- Click the "Add" button to add the new rule.

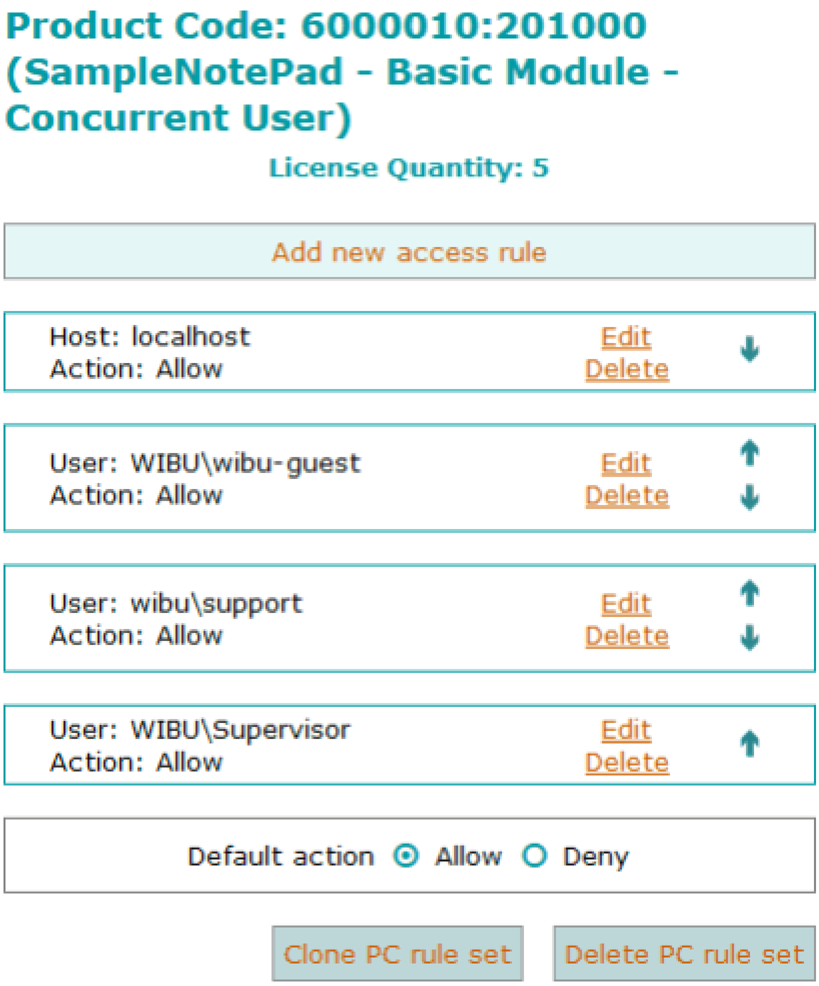
A click on the "Cancel" button cancels the process. The new rule displays in the right rule view.

If you defined several rules, you may change the rule sequence by using the arrow symbols ↑ ↓. Rules are processed top-down, which means that the order of the rules is decisive for the result. Using the "Edit" or "Delete" link allows you to modify a completely delete a rule. In order to delete a complete Product Code-specific access rules use the "Delete rule list" button.

- Define the default setting for all license accesses which are not covered by rules. You have the option to set the Default action to allow or deny license access. Click the "Allow" or "Deny" button.



The figure below shows an example of a specific access rules with exclusive access rights (Reserved: 1; Limit: 1) to the Product Code 201000 of Firm Code 10 for a guest user, the complete support department and a Supervisor. 2 license accesses of a total of 5 license accesses (license quantity) remains available and the default license access is defined as allowed.



3.2 PIAS distribution

3.2.1 Distribution channel

Updates from PIAS can be obtained in two ways:

- Login to the `download`[3] section of SARC's website. A login name and password has been or will be supplied for that.
- Order on memorystick at SARC, against cost price.

Further distribution over your own computer network should be done by yourself. That will be standard procedure, however, if PIAS must be installed on many workstations that's always a chore. PIAS is able to do that automatically. At SARC a document can be obtained in which the *modus operandi* is discussed.

3.2.2 Version number and production date

PIAS' help menu (see section 2.7 on page 11, Help) includes two functions to request software creation date (compilation date) and version number (revision number). This can be used to determine whether it is time to download and install an update. For each user, a new version of PIAS is regularly created and set ready for download. So, this has a recent date and a version number, to which the following should be added:

- The revision is a number which is incremented by 1 upon each source code change. In this respect, not every change in source code is structural, for example, there are adjustments to the manual, correction of spelling errors, addition of new PIAS and LOCOPIAS customers, experiments and functionality in development that does not affect your specific PIAS version. E.g. 50 revision numbers may pass without anything essential having been changed in your version.

[3] `http:\www.sarc.nl/download`

- The same applies to the date of creation: almost every day (often also in weekends and holidays), an up-to-date PIAS is automatically created for every user, even without any changes to your version.

When it is useful to actually install an update:

- In any case on a regular basis. PIAS is constantly being improved and expanded, and even a collection of corrected typographical errors or a new auxiliary function is nice to have. For example, once every six months.
- After significant extensions or modifications have been completed. Depending on your package size, they may come in bits and pieces for a few weeks or months, and then in a pace of two per week. You will be kept informed of such changes through the sources of information as discussed in section 2.2 on page 4, Manuals, exercises and information sources.

3.3 Digitizer function keys

It stimulates a smooth operation of a digitizer if it would be equipped with function keys (or a similar feature), however, that is only seldom the case. So, they can be emulated by sticking a function key sticker on the digitizer, as depicted in the image below. You can print this sticker, stick it in the **top right corner** of the digitizer, and use it for digitizing commands. Please make sure to print teh sticker at the designated width of 27.5 mm. This can be achieved by using the sticker from the `pdf manual`[4], instead as those from the HTML or help reader versions. Alternatively, the sticker can be placed in the **bottom left corner**, in that case an additional setting is required, for which reference is made to *Functions_digitizer_low* in section 3.10 on page 30, External variables.

[4] `http://www.sarc.nl/images/manuals/pias/PIASmanual_en.pdf`

PIAS DIGITIZER SARC BUSSUM	
Knik	
Begin opnieuw	
Nieuwe schaal	
Niet opslaan en stoppen	Opslaan en stoppen

PIAS DIGITIZER SARC BUSSUM	
Knuckle	
Restart	
New scale	
Do not save and quit	Save and quit

Digitizersticker to print

3.4 Computer clock

In order to increase computation speed, in quite some occasions PIAS only will perform a computation if the data have been changed since the last one. E.g. a tank table in Layout is recalculated only when the tank (or hull) has really been changed. In order to assess whteher a recalculation is actually required, the definition time as well as the calculation time are stored, and compared if a new calculation is requested. To this end, it is of the utmost importance that your computer clock is functioning well and indicates the correct time, because data changes may otherwise remain unresolved.

3.5 Temporary files

Just like much other contemporary software, in the background PIAS makes use of temporary files. The use of these files will be unknown to the user unless there is a problem while creating or writing to these files. For example, if PIAS tries to create a such a file in a directory where the user is not allowed to create or write files. The directory location for temporary files is handled by the operating system, where Windows applies these rules:

- The path specified by the TMP environment variable.
- The path specified by the TEMP environment variable, if TMP is not defined.
- The Windows directory, if neither TMP nor TEMP are defined.

3.6 ASCII text file

When the manual mentions ASCII text files, it means a plain text file without control codes. A text file generated with a ‘normal’ word processor (such as MS-Word) frequently contains control codes for the purpose of formatting. To create a plain text file the word processor must be configured such that control codes are not added. An alternative option is editing ASCII files with a simple *editor*, such as Notepad or Notepad++[5] (for free).

3.7 Unicode text file

An ASCII file can more or less only contain Anglo-saxon characters. In order to accommodate also characters from other scripts, internationally, the Unicode standard[6] was developed. Unicode is also supported by PIAS in the static output, which is the output text as included in the program as such. A Unicode file, can be edited with Unicode text editors. A plethora of Unicode editors is available, at SARC we use the free Babelpad[7]. Using Unicode in input data of PIAS, such as names of tanks or loading conditions, is a subject of future development.

3.8 Export to and import from XML

XML[8] is an international data exchange standard. Not primarily from drawing or product information, but in general for structured data collections (“data bases”). In a sense XML is an empty shell, which can be filled for a particular application. Such content has a meaning for the reader and writer of such a file, where the meaning is given by *terms*, which are called *markup* in XML parlance. These terms are free to choose, and must be defined somewhere, in a dictionary, preferably including meaning and units. A nicety of XML is that in general quite expressive terms can be chosen, e.g. `longitudinal_centre_of_flotation_from_aft_perpendicular`, which does not really require a dictionary to grasp its meaning.

For data exchange, XML is the preferred standard within PIAS, so that it will gradually be implemented in ever more places. There is already a large list of keywords for this which typically relate to ship design. These are summarised in the document “SARC XML dictionary on ship design data and commands” which is available on request from SARC (including a number of sample XML files).

It should be borne in mind that not every input or output parameter of PIAS has an XML counterpart. It would be a serious task to implement that, and who is going to use it all? That’s why PIAS is currently equipped with XML support that was most in demand, such as XML output from Loading and Hydrotables, and two-way traffic (input and output) in Layout and Hulldef. Furthermore, XML support is added on demand, i.e. that specific XML functionality can be included in PIAS at the specific request of a user, against payment of the development costs, or sometimes a part thereof.

At the moment, a PIAS basic rule for *reading* an XML file is that it will replace any existing data of that ship or project in its entirety. Reading such a file is really focused on the once-only import of all ship data, and not on incremental or interactive import. For such applications, something could be suggested or conceptualized, but there will first have to be context and scenarios before this could be put into practice in a generally usable way. At the time of writing (2018), however, a project is under development which focuses on interactive cooperation between various computer programs, notably between CADMATIC and PIAS. This system also works on the basis of XML, but that is communicated directly over the network, so that there is no question of an XML **file**. Anyway, this is beyond the XML file functionality available for the standard version of PIAS.

[5] https://notepad-plus-plus.org/
[6] https://en.wikipedia.org/wiki/Unicode
[7] http://www.babelstone.co.uk/Software/BabelPad.html
[8] https://en.wikipedia.org/wiki/XML

3.9 Output in different languages

The output PIAS can be done in different languages. All these output texts are included in separate text files, often named programname.txt (eg. Hydrotables.**txt** with the Hydrotables module). Such a file is a Unicode text file, please refer to the Unicode discussion just above. Each such a language file contains many text blocks, with in each block one line of PIAS output in multiple languages. At the file start a counter is included which represents the number of supported languages for that file. Subsequently, a block with language coding, e.g. *nl* for Dutch and *en* for English, according the ISO-639[9] standard.

In cooperation with SARC, each PIAS user is in principle free to add languages using these files, according to this procedure:

- Contact SARC, because the software itself will also require a small extension to accommodate the new language.
- Ensure to use indeed the most recent language files of PIAS.
- Create a **complete** translation per language file (which in general corresponds with a single PIAS module or function). So that the lines which remain unused at a project or some output will not remain untranslated.
- Please notify SARC which language files are to be extended. Then SARC is aware not to include new texts in those language files.
- Use a limited amount of time per language file, and forward each finalized language file directly to SARC.
- Accept that the extended language files will be distributed to all PIAS users. it is not possible to manage language file for individual users.
- In the future from time to time verify whether new phrases have been added to the language files, add a suitable translation for those lines, and forward it to SARC so that the whole system remains up-to-date. New texts can be recognized by a series of question marks.

Attention

Standard PIAS supports output in Dutch and English, which languages are maintained by SARC. For many modules also other languages are available, such as German, Chinese or Russian. These languages are not actively supported by SARC. If you encounter a “????????” in the PIAS output, it means that the output text for the selected language is not available. If a consistent output in the chosen language is of interest to you, then the corresponding language files can be modified using the procedure above.

3.10 External variables

With external variables PIAS can be configured. Either for a particular option, or for an exotic switch that found to place in a regular menu. It must be considered that the application of external variables is an exception, only intended for specific options or limited to a small number of users. A regular user should not specifically need to apply them. An external variable consists of an argument (= the variable name) and a value. There are four alternative ways to specify an external variable:

- As an *environment variable* of the operating system, in the form variable=value, e.g. PIASmailserver=xyz. Under Windows an environment variable can be defined with a *set* command in a CMD-window (a.k.a. “DOS-box”) or in the *System Properties Panel*. For more details please consult the Windows documentation.
- In an ASCII text file named PIAS.CFG, which must be located in the directory which also contains the PIAS programs themselves. This file contains a number of lines, with on each line one variable and the corresponding value (without the “=” symbol).
- As an additional argument (command line parameter) when invoking Piasmenu. E.g. `Piasmenu PIASmailserver=xyz`.
- Defining it in the Windows registry, in entry HKEY_CURRENT_USER\Software\Sarc\General. For more information about the registry, reference is made to Windows manuals.

3.10.1 List of external variables

piasname=XXX

To specify a fixed file name.

[9] https://nl.wikipedia.org/wiki/Lijst_van_ISO_639-codes

pias_page_height=XXX
Applicable to the output to preview/clipboard, where it can be used to specify the target paper height (in mm).

pias_preview_character_hb_ratio=XXX
In order to specify, at the output to preview/clipboard, that PIAS should apply a character height/breadth ratio of XXX, when composing the lay-out of a page. When the variable is set as follows: *pias_preview_↩ character_hb_ratio=standard*, PIAS sets the ratio to 1.80.

Functions_digitizer_low=1
Functions_digitizer_low=1 to specify that on the digitizing tablet the sticker with function keys is placed on the lower-left corner (instead of the default upper-right corner).

Australian_livestock=yes
Specifies that in the calculation of intact stability, instead of the default heeling grain moments, the heeling moments according to the Australian (AMSA) requirements for livestock transport will be used. For a complete output according to AMSA intact stability criteria for the carriage of cattle this variable must be set; only applying the AMSA stability criteria is not sufficient.

Frame_interpolate=1
Offers at the module for hull form definition (see section 7.2.4 on page 173, Frames (frame positions and frame shapes)) the option to interpolate intermediate frames. Here the same question could be raised as with the previous setting, and the answer would be similar: this interpolation option is rather limited, in the first place there is the requirement that the frames at both sides of the new frames to be created have the same number of points. And then all corresponding points are connected and new points are linearly interpolated. It could not be simpler, if a more advanced method of interpolation is required, Fairway is recommended, this module has been designed for these kinds of operations. And also, by using this setting here you agree to accept this simple method.

PIASmailserver=xxx
Specifies the address of the e-mailserver to be used by PIAS. This server should be configured in such a way that the workstation on which PIAS runs is allowed to send e-mails with the SMTP-protocol. Moreover, these email settings can be specified per project as well, See also: E-mail settings

PIASemailsender=xxx
The e-mail address of the sender.

PIASemailrecipient=xxx
The e-mail address of the recipient.

PIAS_TIME_RECORD_FILE=XXX\ YYY
A file YYY will be written/updated in the directory XXX. A record of time of usage of PIAS software is kept in this file. Every PIAS module writes the following information per session: Elapsed seconds between start time and end time [sec], start time [dd/mm/yy hh:mm], end time [dd/mm/yy hh:mm], username, program name, project directory+filename. It is also possible to define a project description with the external variabele PIAS_PROJECT_NAME, that one will also be included in this file. Convenient for automated hour registration!

ANSIcharset=1
By mid-2011 PIAS switched to the use of the `Unicode`[10] character set for the representation of international characters. With Unicode much more characters are available, which benefits non-western languages in particular. Some PIAS users applied non-standard characters for e.g. the names of tanks or loading conditions. These characters were coded (by Windows) in ANSI or OEM, which is unfortunately incompatible with Unicode. In order to avoid too much inconvenience, PIAS can be instructed with this external variable to treat special characters in the same fashion as before (although 100% correctness can not be guaranteed). However, please be advised that the ANSI/OEM standard is basically incompatible with Unicode, so that it is better to avoid this method from now on. In due time an integrated facility will be created in PIAS to use special characters according to the Unicode standard.

No_multithreading=1
To switch off PIAS' multithreading facility (which is active by default, if purchased).

[10] `http://nl.wikipedia.org/wiki/Unicode`

3.11 Speed enhancing mechanisms in PIAS: PIAS/ES

A characteristic of our naval architectural profession is that we often encounter intensive calculation tasks. Although the computer serves us well in this area for decades already, the processing time may still be a bottleneck, also because man has adapted himself to the increased processing power, and demands more extended calculations than without the computer would have been the case. This mechanism also manifests itself with PIAS, so it is worthwhile to strive for an optimized calculation process. Processors of modern PC's offer more and more cores. In order to squeeze as much power as possible out of the computer, PIAS and LOCOPIAS are increasingly distributing their tasks into multiple threads. For that purpose, PIAS has been equipped with three speed-enhancing packages, named PIAS /ES, where ES is an acronym of *Enhanced Speed*, Dualthreading, Octothreading and Viginitythreading.

3.11.1 PIAS/ES 1: dualthreading

This option, which was implemented in PIAS around 2005, uses the processor technology that became generally available in that period. For a long time past a PC generally had one processor, while this processor contains one core. That implies that the computer can process one task at a time (although the operating system may fool you, and give the impression that multiple tasks are processed simultaneously). However, there is a tendency where a computer is equipped with multiple real or virtual processors (which are multi-processor and multi-core machines respectively). So, this technology enables a program to execute tasks parallel, but the software will have to be adapted for that facility, where tasks which are suitable for simultaneous processing are explicitly offered to the processor for parallel processing. That implies that for every function of a software package it must be considered whether or not it is suitable for parallel processing, and it must be adapted accordingly, if appropriate. Limiting ourselves to PIAS, many tasks can be recognized which can be processed parallel, such as calculating damage stability for multiple angles of inclination, or drawing hull lines with Fairway. On the other hand, there are also jobs which are not suitable, such as the calculation of intermediate stages of flooding, where at first the final stage must be determined, whereupon the water level which corresponds to the filling percentage can be calculated. The following subjects have been implemented:

- At all intact and damage stability calculations: the computation of stability, simultaneously for all angles of inclination (except for the first angle).
- At probabilistic damage stability: the computation of the probability of damage by means of numerical integration (by applying multiple integration threads simultaneously).
- At Layout the computation of the several intersections between bulkheads and/or compartment boundaries.

3.11.2 PIAS ES 2: octothreading

In the previous paragraph it has been discussed why and how PIAS was enhanced to use two processors (or *cores*) simultaneously for her computation-intensive tasks. In subsequent years, computers were provided with an increasing number of cores, so it seems obvious to apply more than two of them concurrently. Unfortunately, the 2005 solution was not scalable; Microsoft has come to the conclusion that the original Windows *multithreading* facilities themselves had a significant overhead (which is correct, we also observed a serious performance penalty), and rather than improve their efficiency, in good-old Microsoft tradition they replaced these with something different, the so-called `thread pools`[11]. Around 2015 PIAS has been extended to use such *pools*, so that more than two cores can now be used in parallel. A further discussion on this subject is available in this `white paper`[12]. In theory that could be quite a number of cores, however, in practice all kinds of considerations and choice must be made, which depend upon the calculation task and the overhead time of starting a separated thread (which is still not zero unfortunately, even when using *pools*). Therefore, the maximum number of concurrent threads has been set to eight, hence the name **octothreading**.

Octothreading is applied in a number of core algorithms in PIAS, which will be increased in the course of time. Incidentally, multithreading can only be applied in algorithms that are inherently capable to be *parallelized*, from which PIAS contains quite a number. However, some tasks are *sequential* in nature and thus cannot be multithreaded. This means that the implementation in PIAS must be considered case by case, there is no magic stick which will distribute the tasks of a program such as PIAS over all processors.

[11] https://en.wikipedia.org/wiki/Thread_pool
[12] https://www.sarc.nl/wp-content/uploads/2017/06/Acceleration-by-hardware-support.pdf

3.11.3 PIAS ES 3: vigintithreading

As described in the previous paragraph, PIAS was adapted around 2015 so that more than two calculation cores can be used in parallel. In view of the overhead time involved in starting up a separate calculation task, a maximum of 8 simultaneous calculation tasks was chosen at the time. Meanwhile, multithreading has been implemented in even more places within the PIAS for lengthy and compute-intensive tasks such as:

- At probabilistic damage stability: the optimization of the damage boundaries.
- At probabilistic damage stability: the generation of damage cases.

The last few years multi-core computers have become widely available and there are now PC‘s available for the common user which have ten or more cores, where hyperthreading allows twenty or more threads to run simultaneously. Especially with time consuming calculation tasks, such as the ones mentioned above, the overhead time hardly plays a role and one can gain a considerable amount of time by calculating with more than 8 threads. Therefore, the maximum number of parallel processes within PIAS has been increased from eight (octothreading) to twenty. We have called this **vigintithreading**.

3.11.4 PIAS ES 2 / PIAS ES 3: AVX

While developing octothreading, every option was used to maximize the computing speed of PIAS. Contemporary processors have facilities for simultaneously running multiple floating point operations — which are relatively time-consuming, so where it would be advantageous to do a number of them at the same time if the computation algorithm allows. Intel baptized these facilities AVX2[13]. Both the octothreading and vigintithreading version of PIAS utilizes deep in her computation core AVX2 as much as possible.

3.11.5 Limit number of processors to be used

Finally, there is an additional feature for when you just do not want all processors to be taken into use by PIAS, for example, if one wants to use the computer simultaneously for other processor-intensive tasks. This can be set with ‘Maximum number of PIAS processors’ as discussed in section 2.8.2 on page 12, Program setup.

3.11.6 Multithreading task monitor

To give the user a clear insight in the use of multiple threads, especially in time consuming calculations and in combination with *vigintithreading*, PIAS has been extended with a *thread monitoring interface*. This interface is visible during the calculations and shows relevant information per thread, such as: information about the status, i.e. a description of the current task of the thread, start time and elapsed time. However this thread monitoring interface was developed together with the implementation of *vigintithreading*, it is now available where implemented in P↩ IAS, also for *single-, dual-* and *octothreading*.

When a thread is finished with a calculation task, the application assigns it a new calculation task. Until there are no more calculation tasks. The image shows an example of a calculation in its last minutes, with some threads already "finished". So there are no more calculation tasks available, the calculation is coming to an end.

[13] https://en.wikipedia.org/wiki/Advanced_Vector_Extensions

Calculation of damage boundaries

Thread ID	Damage case	Start time	Elapsed time (sec)	Thread status
1	5.5	12-06-2020 13:24:55	47	Calculating
2	-	-	-	Finished
3	-	-	-	Finished
4	5.16	12-06-2020 13:24:16	86	Calculating
5	6.15	12-06-2020 13:23:28	134	Calculating
6	6.9	12-06-2020 13:23:29	133	Calculating
7	5.17	12-06-2020 13:24:12	90	Calculating
8	5.1	12-06-2020 13:24:57	45	Calculating
9	-	-	-	Finished
10	5.11	12-06-2020 13:24:50	52	Calculating
11	-	-	-	Finished
12	5.13	12-06-2020 13:24:49	53	Calculating
13	-	-	-	Finished
14	5.15	12-06-2020 13:24:33	69	Calculating
15	-	-	-	Finished
16	6.21	12-06-2020 13:23:28	134	Calculating
17	4.41	12-06-2020 13:24:57	45	Calculating
18	-	-	-	Finished
19	6.13	12-06-2020 13:23:28	134	Calculating
20	-	-	-	Finished

Calculation of damage boundaries

Number of calculated damage cases : 73 / 85

Multithreading task monitor

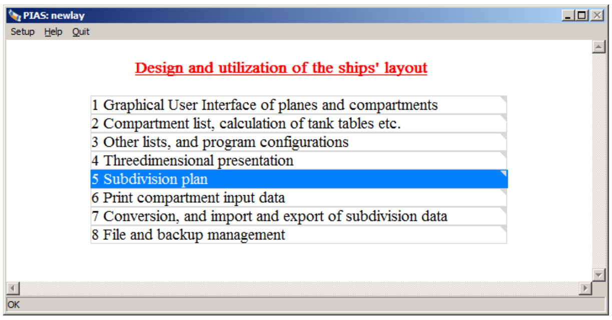
Chapter 4

Operation of PIAS

In section 2.4.1 on page 5, PIAS Main menu a discussion is included on PIAS' main menu, which is used to switch between the different modules. Those modules make intensive use of selection windows and input windows, for which the operation and control are generally the same everywhere. Functions specific to a particular module will be discussed in the corresponding module-specific chapter, but the general process and options are covered here.

4.1 Selection window

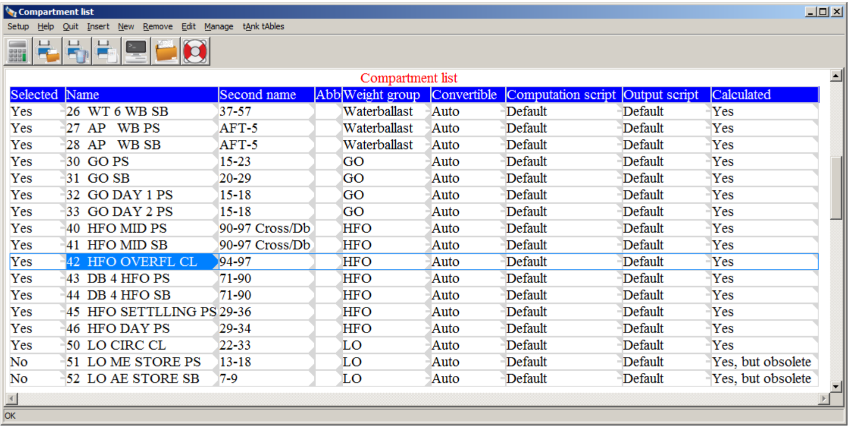
After the module is invoked and the project file has been chosen, with the majority of modules a selection window appears. Selection windows are used all over PIAS and present a list of options or functions which can be selected by the user. As shown in the figure below, one of the options is highlighted, this is the 'text cursor'. The text cursor can be navigated with mouse (select cell by a click of the left mouse button) or the keyboard's cursor keys. An option can be selected with <Enter> or a <Double click left mouse button>.



A selection window.

4.2 Input window

An input window is somewhat similar to a selection window, however, in general it will contain more cells. The prime difference between the two is that an input window also facilitates the input of data or options. An example is depicted below.



Compartment list

Setup Help Quit Insert New Remove Edit Manage tAnk tAbles

Selected	Name	Second name	Abb	Weight group	Convertible	Computation script	Output script	Calculated
Yes	26 WT 6 WB SB	37-57		Waterballast	Auto	Default	Default	Yes
Yes	27 AP WB PS	AFT-5		Waterballast	Auto	Default	Default	Yes
Yes	28 AP WB SB	AFT-5		Waterballast	Auto	Default	Default	Yes
Yes	30 GO PS	15-23		GO	Auto	Default	Default	Yes
Yes	31 GO SB	20-29		GO	Auto	Default	Default	Yes
Yes	32 GO DAY 1 PS	15-18		GO	Auto	Default	Default	Yes
Yes	33 GO DAY 2 PS	15-18		GO	Auto	Default	Default	Yes
Yes	40 HFO MID PS	90-97 Cross/Db		HFO	Auto	Default	Default	Yes
Yes	41 HFO MID SB	90-97 Cross/Db		HFO	Auto	Default	Default	Yes
Yes	42 HFO OVERFL CL	94-97		HFO	Auto	Default	Default	Yes
Yes	43 DB 4 HFO PS	71-90		HFO	Auto	Default	Default	Yes
Yes	44 DB 4 HFO SB	71-90		HFO	Auto	Default	Default	Yes
Yes	45 HFO SETTLLING PS	29-36		HFO	Auto	Default	Default	Yes
Yes	46 HFO DAY PS	29-34		HFO	Auto	Default	Default	Yes
Yes	50 LO CIRC CL	22-33		LO	Auto	Default	Default	Yes
No	51 LO ME STORE PS	13-18		LO	Auto	Default	Default	Yes, but obsolete
No	52 LO AE STORE SB	7-9		LO	Auto	Default	Default	Yes, but obsolete

OK

An input window.

The selection windows and input windows offer the following navigation options:

Keyboard navigation

PIAS has been designed to work swiftly with mouse as well as keyboard. For the latter, the following keys can be used:

- The arrow keys for moving the text cursor. In combination with <Ctrl> the cursor keeps its position, and it will be the whole text block that moves (so the block will scroll).
- <Page Up> and <Page Down> to move the text cursor one page.
- <Home> and <End> to jump to the first or last row. In combination with <Ctrl> to the first or last row of the entire text block.

<Enter>

Just as in a selection window, to select a cell, in other words to go to the window or menu one level deeper into an option or cell where the text cursor is situated.

<Esc>

Also just as in a selection window, to return to the previous window, in other words to go one level back. <Esc> performs the same function as [Quit]. When <Esc> is pressed in the main menu of a module, PIAS returns to PIAS' main menu.

Mouse

By default, the mouse buttons have the following functions in PIAS:

- The <left mouse button> for selecting the cell below the mouse pointer.
- <Double click left mouse button> is equivalent to <Enter>.
- The <middle button> to choose from predefined values, as discussed in section 4.3 on page 38, Content and options in the cells of selection windows and input windows.
- The <right mouse button> is equivalent to <Esc>.
- The mouse wheel to steer the text cursor up or down.

Mouse buttons may be assigned a different function in Windows, which will make the mouse to behave unexpectedly. This can be solved by reconfiguring Windows. Specific PIAS modules may assign specific functions to mouse buttons, notably within the context of a GUI. If so, this will be discussed in the chapter on that module.

Keyboard input

Names and numbers etc. can simple be typed into a cell, as is custom in e.g. spreadsheets. The specific input and selection facilities will be discussed in section 4.3 on page 38, Content and options in the cells of selection windows and input windows.

Furthermore, the menu bar may display the following functions:

Help

See section 2.7 on page 11, Help.

Setup
: See section 2.8 on page 11, Setup.

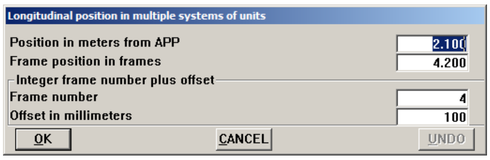
Quit
: Return to the previous menu, identical to <Esc> just above..

Insert
: Add a line above the current line.

New
: Add a line below the current line.

Remove
: Removes the current line or all selected lines of a selection.

Edit
: Contains a number of editing tools - such as copy, paste and undo - which are discussed at section 4.4 on page 39, Copy, paste etc..



Longitudinal position in multiple systems of units, activated by F4.

And, finally, the following options facilitate in the use of input window further:

<F1>
: Opens the context-sensitive help reader, of which section 2.2 on page 4, Manuals, exercises and information sources contains an example.

Function key F2
: In order to edit the text in a cell, the <F2> can be used. The existing text will then become modifiable, with the following operations:

- <Left mouse button> to move the edit cursor. This can also be done with <End>, <Home>, <arrows left and right>, as usual.
- <Backspace> and <Delete> to remove a character.
- <Crtl><C>, <Crtl><V> and <Crtl><X> for the usual copy and paste with Windows' clipboard.
- <Crtl><arrow left> and <Crtl><arrow right> to jump a word to the left or to the right.
- <Enter> or <Double click left mouse button> to stop editing.
- Common characters are inserted at the cursor location.
- <F3>, see just below.

Function key F3
: To edit a longitudinal position, <F3> enables the conversion from frames to meters. After <F3> the program will display 'Fr.?', inviting you to enter a frame position, which will directly be converted to meters. A non-integer frame position can be entered in two ways; the first is simply with the decimal point, for instance frame 3 3/4 can be given as <3.75>, and the second contains an offset in millimeters, e.g. 150 mm before frame 14 can be typed as <14+150>. This conversion mechanism is only available when frame spacings have actually been defined in in the menu of the ship's general particulars, see section 7.2.1.3 on page 167, Frame spacings for a discussion.

Function key F4
: <F4> is an extended version of <F3>. It offers the same frame position conversion options, however, now shown in a popup window, which shows also the frame position and offset, see the picture above for an example. Please be informed that the *meter* values are leading, and that those are converted each time to frame positions. Frame positions as such are never stored.

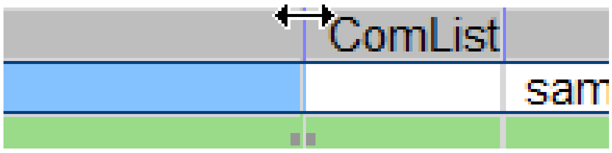
Function key F5
: For certain positions/dimensions, it may be convenient to define them referentially. <F5> opens the necessary pop-up window to make them referential, if applicable. To use this functionality outside of Layout, reference planes must be defined, see section 9.1.6 on page 199, Menu with properties of planes.

Set column width and column order of an input window

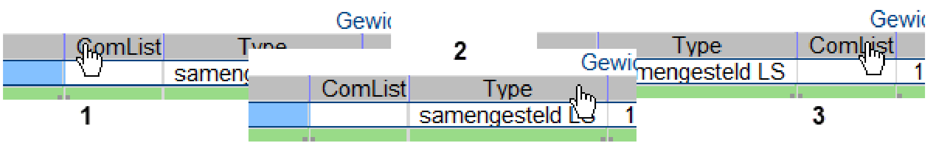
An input window usually contains a header with titles at the top of each column. At the edges of those headers vertical separation lines are drawn, and when hovering the mouse pointer above such a line a horizontal arrow appears. By pressing the left mouse button, the column width can be adjusted, as depicted in the figure below. If a header is not present, individual cells can be adjusted in the same way.

In a similar way, the column *order* can be customized: direct the mouse pointer to a header text, press the left mouse button, and then go to the separation between two other headers. Upon releasing the mouse button the designated column is relocated to this place. See figure below for an example. This feature has not been implemented in each and every menu of PIAS; only where useful — in case of wide input windows — it is included. If this feature is present, the indication thereof is, the light blue lines on the edges of the column headers.

Both settings, column widths and order, can be brought back to their defaults with [Setup]→[Restore column layout], see section 2.8.7 on page 14, Restore column layout, either for all menus of PIAS, or for just the menu visible at that moment.



Adjust column width with horizontal arrow.



Rearrange column order.

Besides these standard options, most modules contain specific functions, which are discussed in the chapters of those modules.

4.3 Content and options in the cells of selection windows and input windows

With respect to the cells of an input window, distinction can be made in **three** methods of interaction;

- Select. I.e. go to the underlying window or menu, with <Enter> or <double click left mouse button>.
- Enter a content. This can be further subdivided in the following two options, viz:
  - Enter a free value or name, such as the vertical center of gravity for a weight item, or the name of a compartment. That value or this name can simply be typed on the keyboard.
  - Choose from a limited number of predefined values, such as the side of a compartment which can either be ‘SB’ or ‘PS’ or ‘double (=SB and PS)’. With such a choice a popup window comes up where the selection can can be made. An exception is if there is only a binary choice — a choice between two (such as yes/no or selected/deselected). In this case it is pointless to present that choice in a popup window because it is evident that one whishes to choose the single alternative. In order to increase the speed of working with PIAS, the value will therefore be immediately switched to the other value without showing such a popup window. So, although a choice of two is presented differently than a choice of multiple options, they are essentially the same, and therefore also the same in terms of operating. Making a choice of predefined types is simply also a way of data input, just like the entry of a name or a number, and is therefore also invoked by a convenient key on the keyboard, which can be any letter or number. Many persons use for this purpose an easily accessible key such as <Spacebar>,

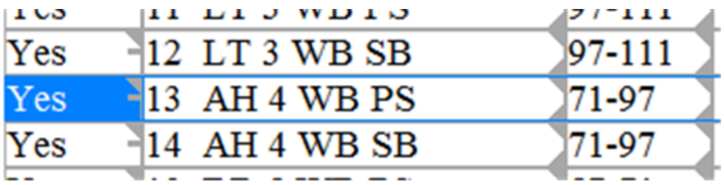
<+> or <-> on the numeric keypad, however, if one prefers a key that leave no trace — which can be nice if you accidentally press the key in cells which *do* accept textual input — then <F5> can be used for this purpose. Working with the mouse, the choice of such a predefined type is initiated by the <middle mouse button>. A third way to invoke the selection of predefined types is described in the bold text just below.

Attention

Please note that <F5> also has another purpose, see section 4.2 on page 35, Input window.

In order to indicate which of these three actions apply in a particular cell, symbols are located on the side of the cell with the most free space, that is to say, on the left if the text in the cell is right aligned, and on the right if the text is placed left. There may be, moreover, also combinations possible of the three actions, such as that at a loading condition its name can be changed by typing **and**, by pressing <Enter>, that this loading condition can be accessed in order to enter tank fillings and weights. These symbols are as follows:

- Select with <Enter>: a small triangle at the top of the cell.
- To choose from predefined values: a rectangle in the middle of the cell. For completeness, this rectangle is not only a passive indication that this cell contains predefined types, but also an **active switch which will pop up the selection window when doubly clicked with the left mouse button**.
- Typing text: a small triangle at the bottom of the cell.



Symbolic indications at the edges of the cells.

4.4 Copy, paste etc.

Quite some numerical input in PIAS is done in an input window, which justifies to extend this facility with some supporting functions. The availability of these functions is determined for each module individually, so some variations may occur, however, the majority of input windows is equipped with the [Edit] function, which contains the following sub-functions:

Undo (or ctrl-Z)
Undoes the last modification.

Redo (or ctrl-Y)
Redoes the last modification.

Copy cell (of ctrl-C)
Copies the content of a cell to Windows' clipboard. Except the content of a single cell, also the content of a selection of multiple cells can be copied. For this purpose first such a selection should be made by holding the <Shift> or <Ctrl> key while extending the selected area by moving the text cursor with the cursor keys or the mouse via clicking or dragging. The <Ctrl> key can be used to make multiple selections that are not adjacent to one another. Alternatively, all cells of the input screen can be selected with the [Select all] function. With the text cursor *in* the selected area, the entire selection will be copied, otherwise only the single cell on which the text cursor resides will be copied. If several selections have been made, it may not be possible to copy them to the clipboard, in which case a message will be displayed.

Paste cell (or ctrl-V)
With this function the clipboard content will be pasted into this input window. This function has two variants, the first one is actually the exception, which occurs if the clipboard contains only the content of a single cell which is pasted into a selection of multiple cells. In this case the clipboard content will be copied into every cell of the selection. In all other cases the clipboard content, irrespective whether it contains one or more cells, is copied starting on the location of the text cursor.

Copy row
Which makes the entire row where the text cursor is located to be copied into internal PIAS format (so, not to clipboard). The difference with the previous [Copy] is that under the visible row much more information can be available, which is also copied in this fashion. Examples are frames (which includes points of the frames 'below' the frame itself) and loading conditions (which contain weight items 'below'), which can be copied integrally in the input window with this function.

Paste row
If a row has been copied with the previous option, with this [Paste] function a copy will be made into the line of the text cursor. In the case of pasting to a selection, all the rows of the selection are overwritten.

Info
In a particular input window paste or undo may not be supported for each column. In this case, with this function one can enquire for which columns this applies.

Select all (or ctrl-A)
Which selects all cells of the input window.

Select column(s) (or ctrl-S)
Selects one or more columns, i.e. the column the text cursor is on or all columns of the selection.

Select row(s) (or ctrl-D)
Selects one or more rows, i.e. the row the text cursor is on or all rows of the selection.

Cell functions
The function keys <F2> to <F5>, which are discussed in section 4.2 on page 35, Input window, but now as menu options. It should be noted that the availability of these options may vary from cell to cell.

On these supporting functions a few remarks still can be made:

- A selection is almost always cleared after performing an action/command or moving the text cursor without holding down the <Shift> or <Ctrl> key.
- Selected areas consisting of multiple cells can be used for copy and paste, as discussed above, and for concurrent input as well. For certain actions, selected areas have significance, e.g. when deleting multiple rows, in which case this will be described in the manual for the relevant action.
- Undo and redo information remains available per session of an input window. So it is not stored permanently.
- The exchange format between a selection and the clipboard is according to Excel convention, which means that columns are separated by a Tab, and rows are terminated by a CR (*Carriage Return*) LF (*Line Feed*) combination.
- The GUI of Fairway has its own integrated *undo/redo* facilities, which have nothing to do with those in the input windows.

4.5 Error messages and well-intentioned warnings

A program can, for several reasons, generate error messages aswell as well-intentioned warnings. This, for example, can happen due to user input being outside of accepted values or because a calculation cannot find a solution. This menu facilitates in the proper handling of these messages and warnings.

The menu displays a description of the error message or well-intentioned warning. The colour depicts if it is an error message or a well-intentioned warning — these colours can be set to your personal liking —. The description cannot always display all the information available, if so then the description column can be *entered* to show all the information, this can also be discerned from the 'Extra info' column when it is set to 'Yes'. The information presented, when entering the description column, can be a more in-depth explanation of the description or contain references to items that are at fault. The column 'No. of times given' gives a quick overview of the number of times that error message or well-intentioned warning has been given.

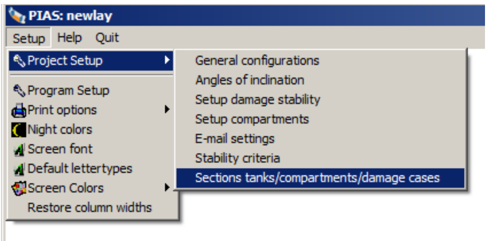
There are also several menubar option which are explained below:

- The option [Settings] gives the possibility to turn off well-intentioned warnings and gives control over the colours used for *Error messages* and *Well-intentioned warnings*.
- If the option [Link to manual] is available then the PIAS manual will be opened to a relevant section with information for resolving or elaborating on the error message or well-intentioned warning.
- The option [Open log] opens a written text file containing all the error messages and well-intentioned warnings. This can come in handy when there are several items that need to be resolved. The text file is always generated when this menu pops-up, so there might be a need to save the text file to a new location in order keep the error messages and well-intentioned warnings.

Chapter 5

Config: General project configurations

With this module the choices and parameters for calculations, calculation variants and output can be set. This module can be activated as independent PIAS module from PIAS&apos main menu, although most options are also available through the Project setup function, at the left-hand side in the upper (menu) bar of each other PIAS module, as depicted below.



Project setup function in each PIAS module.

In this module *general program settings* can be defined. These are not:

- *General ship particulars*, which are, after all, defined in the ship hull design or definition modules Fairway or Hulldef, as discussed in section 7.2.1 on page 167, Define main dimensions and further ship parameters.
- Program setting which are specific for one single module. These settings are given at that particular module.

Defined here are program settings — occasionally a computation method — which are applicable to more than one single module.Config starts by showing its main menu, which contains the distinct setup categories:

General project configurations

1.	Calculation methods and output preferences
2.	Angles of inclination for stability calculations
3.	Settings for compartments and tank sounding tables
4.	General settings damage stability
5.	Sections tanks/compartments/damage cases
6.	Stability criteria
7.	Wave settings (for stability)
8.	Settings page heading
9.	E-mail settings

5.1 Calculation methods and output preferences

At this option the desired output format can be selected, such as output language and file format, as well as some general settings which are applicable to the hydrostatic calculations, such as calculation methods, wave particulars and the density of sea water.

Calculation methods and output preference

5.1.1 Output language

Here the language can be specified as used for the output of calculations and the like. Choices are e.g. Dutch, English, German and Chinese, but only the first two are available in **all** PIAS modules. If a language is selected that is not available in a particular module or output, then by default English will be used. Incidentally, each user is able to expand the supported languages, see section 3.9 on page 30, Output in different languages.

Attention

If the output contains a row of question marks instead of plain text, then PIAS has encountered an unavailable translation. In the link stated just above the procedure to handle such anomalies is discussed.

5.1.2 Apply Local cloud

With this option set to 'yes' for this project the *local cloud*, a concept which is discussed at section 2.11 on page 17, Local cloud: simultaneous multi-module operation on the same project, will be activated. However, for the time being this option is still experimental, so it is not yet released for general use.

The module Loading always prints an item in the loading conditions which represents the total light ship weight. This total weight represents the combined weight of items from the 'common list&rsqauo. The name of this*light ship weight* can be specified here.

5.1.3 Stability calculation method

With this setting the calculation method of intact and damage stability can be chosen, with a choice from four options:

- With fixed trim. At zero inclination the ship has its initial trim, depending on the difference between LCB and LCG. This trim is kept constant for all angles of inclination, and used during the computation of stability. This 'fixed trim' option is applicable to intact stability.
- With free-to-trim effect (=constant LCB). Also, in this case the ship has its initial trim. When the vessel heels *for each heeling angle* the real trim is calculated so that the longitudinal center of gravity coincides with the longitudinal center of buoyancy. This is also known as then 'constant LCB method'. It will be obvious that in general the trims will be larger with a vessel of a longitudinally asymmetric nature, such as a supply vessel at larger heeling angles.
- With free-to-trim effect, including the effect of VCG on trim. With only 'free-to-trim', when calculating intact or damaged stability, PIAS determines longitudinal equilibrium (and consequently trim) by coinciding longitudinal center of buoyancy with the longitudinal center of gravity, which means that they are on the same longitudinal location. The implicit assumption behind this method is that the forces act *perpendicular to the base line* (which is the longitudinal axis), however, with a trimmed vessel this is obviously not the case. This simplification results in a small trim inaccuracy, which is larger at larger trims and higher VCGs, but which is sufficiently small in normal cases. However, it might be desirable to make a somewhat more accurate computation by taking the forces *perpendicular on the waterline*, which is done with this setting at 'yes'. With this setting one should also realize the other effects, such as:

  - Stability calculation results do not necessarily have to agree with results of computations based on tables of maximum allowable VCG', because there the effect of VCG' on trim is not included (which would be a bit difficult to do, because the actual VCG' of the loading condition was unknown on forehand, when composing the VCG' tables.
  - Stability calculation results might differ from manual calculations as composed with the aid of cross curves, because it would not be possible to compose them for the actual VCG' of the loading condition, as well as for all other VCGs of all other loading conditions. In theory multiple cross curves for multiple trims could be composed, but nobody does and nobody requires so.
- Heeling around the weakest axis. With the previous three methods, heeling occurs around, more or less, the longitudinal axis of the vessel. With this method automatically the axis of weakest stability is determined. Therefore, this method is the most realistic of the four. This setting is only implemented in PIAS in calculations where it is relevant and consistent, so e.g. for the calculations of intact and damage stability, but not for e.g. cross curves and maximum allowable grain moments. The background of this method is described in a separate document *Stability around the weakest axis in PIAS; explanatory notes*, which is available at SARC on request.

5.1.4 (Damage-) stability including the shift of COGs of liquid

The conventional way to take free surfaces into account is by correcting the vertical center of gravity by a virtual increase due to the free surface(s) at heeling angle zero. This virtual increase of the actual VCG is taken constant at all heeling angles. However, in reality the free surface effects change due to heel and trim.

- If this option is answered with 'No', the (damage-) stability is calculated in the traditional way with a VCG correction which is constant for all heeling angles.
- If this option is set to 'Yes', and if this option has been purchased, then for every compartment containing a liquid the actual center of gravity is calculated at the actual heeling angle and trim at the intact stability and damage stability calculations (with the Loading module).

This option set to 'Yes' has the following consequences:

- Tables of maximum allowable VCG' are no longer valid while the virtual center of gravity G' has become meaningless.
- Centers of gravity and free surface moments printed in loading conditions may differ from the input data. Due to heel and trim the centers of gravity may have shifted and the surface of the liquid may have a different shape than at zero heel and trim.
- All weight items (in Loading) which contain liquid cargo should be of types 'tank' or 'floodable tank'. Weight items from different types which yet have a non-zero value for Free Surafce Moment are not allowed.

5.1.5 Preferential format of hull files

PIAS can perform hydrostatic calculations and (damage) stability calculations on the basis of two of the representations as discussed in section 2.10.2 on page 16, Hull form representations :

- The *frame model*, which essentially consists of cross sections (ordinates), and which may be defined by Hulldef or generated by Fairway.
- The *triangulated surface model*, which is essentially a representation of the hull surface (including differentiating inside and outside), which can be produced by Fairway.

In principle, a surface model can be more precise, particularly when determining the shape of small compartments. However, it has a major objection, and that is that, in order to be sufficiently precise, the number of triangles of the hull surface must be rather large, leading to correspondingly long computation time. Therefore, in practice, the surface model is not used for ordinary calculations. PIAS also has a third option, which is 'calculate with frame model, plot surface model', with which all calculations are based on the frames, while in the graphical presentation — e.g. as in the GUI of Loading — the full surface model of the hull is shown. That setting offers a nice compromise.

5.1.6 Density outside water

Enter here the density (the specific weight of the outside water (sea water, in ton/m^3) to apply for all hydrostatic, stability etc. computations. As a rule, for sea water 1.025 is taken.

5.1.7 Calculate intact stability etc. with a heeling to

For the sake of all stability-related calculations, angles of inclination can be given (see section 5.2 on the next page, Angles of inclination for stability calculations). That leaves the question of which side (PS, SB or both) these angles should be applied. Which is controlled here, by a choice from four options:

1. Portside.
2. Starboard.
3. The side of the heel. With this setting, the side of the worst stability is estimated with this method: if the statical angle of inclination (the heel) is to PS then the calculation is made to PS, otherwise to SB. With zero heel the calculation is (therefore) to SB. This estimation will often be correct — in the sense that that is indeed the side of worst stability — and sometimes not; e.g. when the heel is to SB, but openings on PS are submerged at a much smaller angle than on SB. If one does not want to rely on an estimation in determining the side of worst stability than the fourth option can be used.
4. Portside and starboard. With this setting there will be no *a priori* assumption on the "worst side", instead the stability will be calculated to PS as well as SB, while both sides are fully taken into account in the stability assessment. If stability criteria have been defined (and selected, see section 15.1 on page 276, Manipulating and selecting sets of stability criteria) then also a maximum allowable VCG' can be determined, which will be the minimum of **all** stability criteria and **both** sides.

The third option is the default, but that does not mean that the user should not rethink the applicable choice for the vessel under consideration. In any case, one must realize that the more asymmetrical the ship is (in terms of hull shape, openings or compartments) the less accurate the first three options can be. Obviously, the fourth option will require more computations, and hence more calculation time.

Please take in this respect also take note of the opening remark of section 15.3.3 on page 288, Types of basic criteria.

5.1.8 Calculate damage stability with a heeling to

This option is similar to the previous, albeit applicable for damage stability instead of intact stability.

5.1.9 Backward compatibility in calculations

Rarely, one of the central computation algorithms from PIAS is modified. If feasible in such a case, a switch may then be added which makes the program still to apply the original algorithm. In general, it is discouraged to use such a *backward compatibility mode*, because the whole reason for the modification will be that the new algorithm will be better (e.g. faster, more accurate or more robust) than the original. However, if you need results to be compatible with previous PIAS versions — for example for existing designs or elder projects — you can set that here. This option is only available in Config, not from the *Project setup* in the menu bar of other modules. At present only a single switch exists, namely:

5.1.9.1 Volumetric computations with the method from before December 2016

For the implementation of octothreading, in December 2016 some core volumetric integration procedures of PIAS have been redesigned. In that process the **accuracy** of one particular algorithm was increased a bit. That algorithm existed for more than 25 years, but the steadily increased computer power allowed a refinement to be incorporated now. Due to this modification, hydrostatics and stability-related results of PIAS might differ from earlier versions. If this box in the popup window is ticked **on**, the pre-December 2016 algorithm will remain to be used for this project.

5.1.10 Output filetype

If the output has been redirected to *File*, see section 2.8.3 on page 12, Print options, then a file will be generated of the selected output filetype and with the file name as given on the next line. Supported file formats are (please also refer to the somewhat more detailed discussion in section 2.5 on page 9, Export of results):

- ASCII. With this option only alphanumerical output (tables and text) are send to file, graphics, layout and formatting are lost.
- Rich Text Format. Contains the complete output, including all formatting and graphs, in RTF, which can be imported into a word processor such as MS-Word or OpenOffice.

- Drawing Exchange Format (DXF). Drawings and graphs are stored in a file according to the Autodesk DXF specification, which can be imported into e.g. AutoCAD or Rhino.
- Postscript, which will save drawings in vector format. This has the advantage of being resolution-independent, which results in much more sharpness in large or highly zoomed drawings.

5.2 Angles of inclination for stability calculations

With this option the angles of inclination which will be used at the intact and damage stability calculations can be defined, with a maximum of 100. The angles may be larger than 90°, however, not larger than 180°. Angles between 85°and 95°are not allowed. On the angles as givenhere two expections may be applicable:

- For probabilistic damage stability calculations it can be set that PIAS should use a default setting for angles of inclination, as discussed in section 5.4.13 on page 50, Compute probabilistic damage stability on basis of.
- Using *Consecutive Flooding*, trough pipe connections and internal flow over thresholds, discontinuities may occur in the stability curve. In order to model these neatly, with such calculations internally many more angles will be used, as described in section 21.4.1.3 on page 413, Basis of larger angle stability (GZ-curve).

5.3 Settings for compartments and tank sounding tables

This menu contains settings which are applicable to tanks and compartments:

- Tables with everywhere the maximum free surface moment. By default, the free surface moments as printed in the tank tables are for the actual filling level of the tank. If this option is set to 'yes', then first the maximum free surface moments will be computed for the present tank, after which for each height (in between completely filled and completely empty) instead of the actual moments these maximums will be inserted. After toggling this setting, the tank tables in Layout should be recalculated before this setting exerts its effect. Moreover, any previously calculated table should be discarded first.
- Difference internal/external geometry including external hullforms. This setting is applicable to the comparison of internal and external geometry, such as has been discussed in section 9.10.3 on page 241, Difference between internal and external geometry. If set to 'no' this comparison does not include added hull forms (as discussed in section 7.2.2 on page 172, Hullforms. If set to 'yes' it does take such hull forms into account.
- Direct calculation of tank data. If this option is selected, the tank data in the loading conditions is no longer determined by interpolation on the pre-calculated tank tables, but by a direct calculation of the current volume or liquid level. The tank data is also calculated immediately when the tank tables are printed in the Layout module. There are no pre-calculated tables available in Layout. The tables can be checked by printing them.

5.4 General settings damage stability

General setup for damage stability calculations

1.	Calculation damage stability according to the method of
2.	Calculation Consecutive Flooding according
3.	Time domain calculation time step
4.	Time domain maximum number of time steps
5.	GZ calculation interval in seconds
6.	Minimum weight difference for a GZ calculation
7.	Maximum allowable equalization time
8.	Percentage of transferred liquid at which equalization is considered ready
9.	Minimum sectional area for instantaneous fluid passage
10.	Permeability of connected compartments
11.	Righting levers denominator
12.	Intermediate stages with global equal liquid level
13.	Compute probabilistic damage stability on basis of
14.	Significant wave height for SOLAS STAB90+50 (RoRo)
15.	Damage stability with correction 0.05' x cos(phi)
16.	Automatic propagation of damage case

5.4.1 Calculation damage stability according to the method of

On this line, one can specify which method should be used for compartment connections and (internal) openings and so on. Two systems exist for this purpose, viz. 'Complex stages' and 'Consecutive Flooding'. The first has been in development from ±1990 to 2022, supporting virtual connections between compartments, including so-called 'critical points' with which thresholds and internal openings can be modelled. The second is available from 2023, and supports the entire topology and geometry of internal connections, pipelines, valves, check valves etc. Both systems are discussed in extenso in section 21.1 on page 402, Background from tools for ship-internal connections in PIAS.

To be elaborated: include the list as present in the Dutch manual version.

5.4.2 Calculation Consecutive Flooding according

Consecutive Flooding currently has two variants to choose between here:

- More or less conventional intermediate stages of flooding, genaamd 'Fractional", see section 21.2.1 on page 403, With conventional intermediate stages of flooding ("Fractional").
- A more realistic flooding scenario based on small time steps. Named &;squo;Time Domain', see section 21.2.2 on page 406, Damage stability in time domain.

5.4.3 Time domain calculation time step

As explained in section 21.2.2 on page 406, Damage stability in time domain, this method essentially simulates the continuous flooding process with a series of very small steps in time. The duration of such a step, in seconds, is specified here. Incidentally, this is a global setting belonging to the vessel as such. At the detail level a more specific time step can be specified, see the discussion at section 9.6.3.2 on page 230, Properties of piping networks. Todo: and in Loading.

5.4.4 Time domain maximum number of time steps

The previous setting manages the calculation process, but carries the risk that using a time step that is (in hindsight) too short will result in extreme computation times, and corresponding output. To limit this, the number of time steps can be capped here. Please also consult the discussion at section 21.4.1.2 on page 412, Basis of damage stability in time domain.

5.4.5 GZ calculation interval in seconds

At each time step of a time domain calculation, fluid flow, draft, trim and heel will be calculated. In addition the full stability (a GZ curve over all heeling angles) could also be calculated. However, if that is done for each time step it could lead to a large amount of output that is by no means all relevant.

To mitigate that, this setting, together with the next, manages on which steps a full stability calculation will be made. For example, if 15 is specified here, then every 15th step the GZ will be calculated (and verified against the stability criteria). Incidentally, around the first and last time steps the GZ will always be calculated, because that is supposed to be interesting anyhow, regardsless of the setting here.

5.4.6 Minimum weight difference for a GZ calculation

This setting pairs with the previous. At the beginning of a time-domain process, in general the fluid flows are quite strong, but towards the end they become less and less so. This means that the differences per time step towards the end are also small, so that even when limiting stability output using the previous parameter still quite a lot of, not always relevant, output may be produced. One can further reduce that additionally with this parameter; here one specifies what the minimum difference in total weight (in ton) with the previous full stability calculation should be. So if you enter 22.45 here, the stability is only calculated at that step where at least 22.45 ton of fluid has been displaced with respect to the previous full stability calculation.

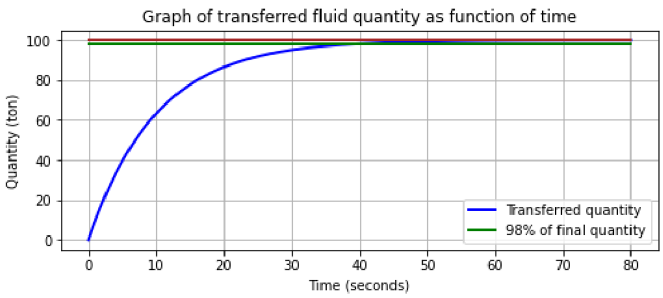
5.4.7 Maximum allowable equalization time

For the purpose of time-domain calculations: the maximum time for the ship to come to rest, after damage and water ingress. This paremeter is used to determine in which cases final-stage stability criteria are applied, and when those for intermediate stages, see discussion in paragraph 21.2.3.2.1 on page 407, Choice of stability criteria with the time domain method.

5.4.8 Percentage of transferred liquid at which equalization is considered ready

The whole idea of a time domain calculation is to calculate the time in which the fluids equalizes. That is finished when the whole system of vessel and fluids have come to rest. However, towards the end of the process the fluids start to flow slower and slower; after all, the level differences get smaller, and hence the pressures, and hence, courtesy of Bernoulli, the flow velocities and flow rates. In essence, it is an asymptotical process where after, so to speak, many hours, milliliters are still flowing through the pipelines and openings. Indeed, in theory, equalization time will always be infinite. In practice, there is a certain tolerance in PIAS, so that if e.g. the difference in draftt between two consecutive time steps is less than a mm or 1/10 mm that is considered as ‘rest’. However, that is an arbitrary tolerance that unintentionally has a large outcome on the final answer; at 1/10 mm, the equalisation time can easily be twice as long as at 1 mm.

One might think of implementing in a practical limit; after all, we are interested in the tons flowing through the system in the early time, and not so much in the millilitres in the last seconds. With that idea, a criterion can be formulated that is related to the transferred weight. For example ‘if 98% of the total, final, fluid weight has flown through then I consider the system at rest’. The default percentage used in PIAS is indeed 98%, but this Config setting allows the users to adjust it as they see fit. The effect of this setting is visualized in the figure below.



Transferred liquid as function of time.

5.4.9 Minimum sectional area for instantaneous fluid passage

If the cross-sectional area of a pipe or connection is larger than the area specified here (in m²) then the water will flow freely through it during heel, and otherwise not. Background to this is discussed in section 21.4.1.3 on page 413, Basis of larger angle stability (GZ-curve).

Attention

Please acknowledge that the minimum area is given here in m^2, while pipe' cross-sectional areas in Layout ar egiven in cm^2.

5.4.10 Permeability of connected compartments

The good old art of naval architecture has brought our community differences in permeabilities (μ) to be applied, e.g. 98% for an intact compartment and 95% when it is damaged. For the same compartment! Although this convention is nonsensical, PIAS is adapted to it anyway, given e.g.&the different types of μs that can be assigned to each compartment, see paragraph 9.5.1.3.2 on page 219, Permeabilities. However, that is not the end of the story, take for example:

- A damaged compartment, connected to another non-damaged space, by a tiny water pipe. This other space 'feels' somehow intact, so should the intact μ then consequently be applied?
- The same damaged compartent, now connected to another space by a large duct. This 'feels' like being part of one global damaged space, so the damaged μ should now be applied? Or not? This configuration is in principle not different than the previous.

So, there is no intrinsic logic which guides us here. But perhaps the regulations provide a clue? Let's have a look at two of those:

- SOLAS 2020, chapter II-1, part B (probabilistic damage stability) reg. 7-3: "For the purpose of subdivision and damage stability calculation of the regulations, the μ *of each compartment* shall be as follows: (followed by a table of damaged μs)". This line implies that the damaged μs are applicable to **all** compartments, so the damaged as well as the connected (i.e. flooded but not damaged) compartments.
- IBC Code 2016, §2.7.2: "The permeabilities of spaces assumed to be damaged shall be as follows: (followed by a table of damaged μs)". This implies that the μs stated here are applicable only to the damaged comps, so connected non-damaged compartments should be assigned a μ as used for intact condition.

More rules could be investigated, however, to cover these two opposite expressions will already require a PIAS setting, which regulates the μ to be used for non-damaged, connected compartments. Because a user will choose this setting based upon the applicable regulation, this single global setting will suffice. With the choice between 'Use tank permeability' and 'Use permeability damaged'

Attention

Please do realize that when selecting 'Use tank permeability' for all potentially connected compartments in Layout a realistic value for 'permeability as tank' must be specified, also for the dry compartments. The default of 98% for e.g. an engine room will probably not be considered realistic.....

5.4.11 Righting levers denominator

Stability standards are in general related to righting levers, and derived parameters such as GM and area under the righting lever curve. However, in the essence, during a stability calculation righting (and heeling) **moments** are computed, instead of **levers**. Converting from moments (ton.meter) to levers (meter) is conveniently done by dividing by the vessel's displacement. For intact stability this displacement (the denominator in the division) is unambiguously the one and only factual displacement for the particular loading condition. However, for damage stability choosing the denominator is not that obvious, should for example the displacement be corrected for the liquid cargo loss, and/or the weight of the ingressed sea water? The relevant regulations provide two alternatives for this choice, which have both been implemented in PIAS:

- Constant displacement. With this setting the denominator is simply the intact displacement. This is the conventional choice, e.g. applicable to:
  - Code for the construction and equipment of ships carrying dangerous chemicals in bulk (1980 edition), guideline for the uniform interpretation, reg. 3.2: "The GM, GZ and KG for judging the final survival conditions should be calculated by the **constant displacement method**".

   - MSC.1/Circ.1461, Guidelines for verification of damage stability requirements for tankers (applicable to IGC and IBC 2016), reg. 3.3.3, as well as IACS 110 Guideline for Scope of Damage Stability Verification on new oil tankers, chemical tankers and gas carriers (2010), reg. 3.3: “When determining the righting lever (GZ) of the residual stability curve, the **constant displacement method** of calculation should be used”.
   - SOLAS 2009 probabilistic damage stability (part B.1), reg. 3 “When determining the positive righting lever (GZ) of the residual stability curve, the displacement used should be that of the intact condition. That is, the **constant displacement method** of calculation should be used”.

  Before PIAS was extended with this setting *Righting levers denominator* (in July 2018) this was the standard choice for the denominator.
- Intact displacement minus liquid cargo loss. This alternative is offered by MSC.1/Circ.1461, guidelines for verification of damage stability requirements for tankers (applicable to IGC and IBC 2016), reg. 9.3.4, as well as IACS 110 Guideline for Scope of Damage Stability Verification on new oil tankers, chemical tankers and gas carriers (2010), reg. 9.3: “Noting that calculation of stability in the final damage condition assumes both the liquid cargo and the buoyancy of the damaged spaces to be lost, it is therefore considered both reasonable and consistent to base the residual GZ curve at each intermediate stage on the **intact displacement minus total liquid cargo loss** at each stage”.

Note

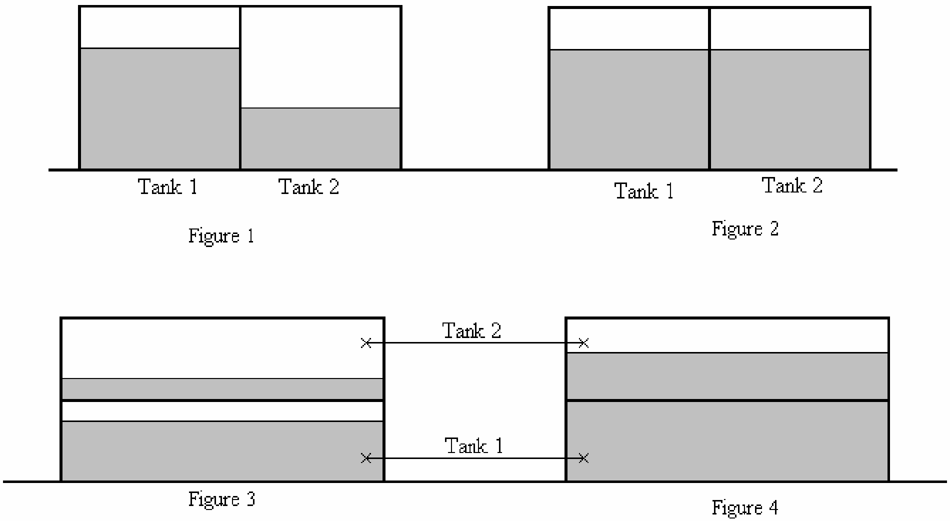
Now and then, this choice of denominator is confused with the methods of *lost buoyancy* vs. *added weight*. However, these are distinct concepts: *lost buoyancy* vs. *added weight* refers to the iteration method used to find equilibrium between weights and buoyancy. In the pre-computer era this issue carried some relevance because it determined the computational efficiency, but with abundant computer power it has become irrelevant. After all, both methods lead to the same observable parameters (draft and trim) which is quite obvious because otherwise one of the two would be outright false, and could easily be identified as such on the basis of a physical (model) test. *Constant displacement* vs. *Intact displacement minus liquid cargo loss* is not a method, it is just a single number to be divided by, which only exerts its influence on the GZ (and hence derived parameters). While GZ is not a primary physical quantity, it is a derived parameter, valid only within a particular reference framework. Apparently, there are at present two of such frameworks around, which may lead to different GZs, without a physically-based referee to judge on their correctness. So, guidance for the denominator to use can only be found in the rule of man-made law (as well as in the stance of consistency).

5.4.12 Intermediate stages with global equal liquid level

This option defines the regime of intermediate stages of flooding. For the damage stability calculation, the vessel is subdivided in multiple compartments, which can be damage simultaneously. For the final stage of flooding the presence of multiple damaged compartments gives no ambiguity, however, with intermediate stages itis the question how the ingressed water is distributed over the compartments. Suppose two tanks become damaged, and the flooding stage is 50%, then there are two options:

- Compartments with an *unequal* water level. Every compartment has half of its weight in the final stage (the 100% stage of flooding). In this case all tanks are treated separately, the water levels of tanks 1 and 2 differ, and there are two free surface moments. This is depicted in situations 1 and 3 of the figure ‘Tank fillings’.
- All compartments with an *equal* water level. All damaged compartments combined are half the weight of the total weight in the final stage of flooding. So, all compartments are treated as one with one single water level and a single free surface moment, as depicted in situations 2 and 4 of the figure.

The appropriate choice of the method of calculation depends on the configuration of the compartments which are damaged. If these compartments are separated by vertical bulkheads as in situations 1 and 2 then the first choice would be the most realistic. If, on the other hand, the compartments are separated by horizontal bulkheads as in situations 3 and 4 then the second choice would be the most appropriate.



Tank filling

Attention

This feature has long been redundant, as the same effect can be achieved with the systems for managing intermediate stages, as discussed in chapter 21 on page 402, Internal flooding in case of damage, through pipe lines and compartment connections. Therefore, it will be removed from PIAS in the course of 2025.

5.4.13 Compute probabilistic damage stability on basis of

Here it can be set for the computations of probabilistic damage stability, as well as maximum allowable VCG in damaged condition, which angles of inclination to apply:

- User-defined angles. With this choice the angles of inclination as specified at section 5.2 on page 45, Angles of inclination for stability calculations will be used. The advantage of this setting is that this is the same calculation basis as employed at the Loading module, where the GZ curve is also computed on the specified angles. So, this choice results in the same results for both computations.
- Default angles. With this setting the angles are chosen automatically, guided by the applicable stability criteria. The advantage of this setting is that the user is relieved from choosing the angles, the range for example will always be sufficient to match the stability criteria. The disadvantage may be that these angles may be different from the ones as employed in Loading.

This is, by the way, exactly the same setting as *calculate maximum VCG on basis of* as discussed in section 10.2.8 on page 255, Maximum VCG' intact tables.

5.4.14 Significant wave height for SOLAS STAB90+50 (RoRo)

For RoRo ferries with water on deck (a.k.a. as the 'Stockholm agreement' or 'STAB90+50') the wave height to be used can be given here. Please also refer to section 21.3.2.2 on page 410, Water on deck.

5.4.15 Damage stability with correction 0.05' x cos(phi)

According to the US DDS-079 damage stability criteria, in the computation of damage stability the transverse center of gravity should be corrected with 0.05 feet x cos(φ). That can be specified here.

5.4.16 Automatic propagation of damage case

When evaluating the damage stability results, it might be concluded that a calculation does not comply with the damage stability criteria because an opening of an intact compartment is submerged. One might wonder what the conclusion would be if the flooding would be extended through that opening. The evaluation of such *progressive flooding* requires some flooding scenario assumptions, and is in general still uncharted territory. But for one particular case in PIAS a provision has been included, with the following details:

- Works with the stability criterion ‘Distance to special points’ (see section 15.3.3 on page 288, Types of basic criteria).
- If a particular damage case does not meet this criterion — because the distance from the waterline to such an opening is less than the minimum required — then the conclusion is drawn “It is yet undetermined whether this damage case complies with the criteria”, and an additional damage case is created where the compartment connected to this opening will also be flooded.
- From these additional damage cases also the intermediate stages of flooding are computed, starting with a filling percentage of 1% for the newly added compartments. This reflects the fact that these are just about to be flooded, but also tests whether the original damage case meets the other stability criteria.
- Since the flooding through such an opening may take a long time, it is not certain that in all cases assessment against the stability criteria for intermediate stages is allowed. Therefore, in this case the criteria for the final stage of flooding are applied.
- This mechanism repeats itself, so, if such a newly generated damage case also does not comply because an *other* opening has a too small distance to the waterline, then a further additional damage case will be created, etc. etc. Until it is demonstrated that it will comply in this case of progressive flooding (in which case the original damage case complies), or until the ship no longer satisfies another stability criterion (in which case the damage case does not comply).

This facility — which has been discussed and agreed with Lloyd’s Register of Shipping in April 2016 — is aimed at inland waterway tankers which must meet the ADN criteria. But anyone with access to this facility is, obviously, free to switch it on in this menu for any application.

5.5 Sections tanks/compartments/damage cases

Here the sections and other properties of sketches of compartments and damage cases can be given, details are discussed in section 20.2 on page 396, Sketches of tanks, compartments and damage cases.

5.6 Stability criteria

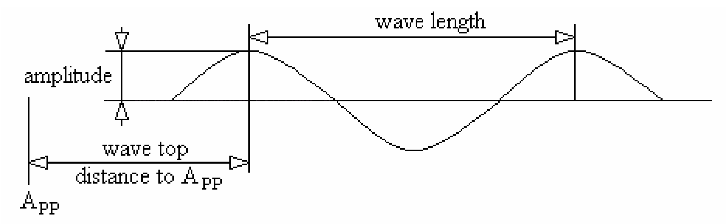
The stability criteria definition system has so many options and possibilities that a dedicated chapter is included, please refer to chapter 15 on page 275, Stability criteria for intact stability and damage stability.

5.7 Wave settings (for stability)

The hydrostatic and (damage) stability calculations can also be executed for the ship in a static wave (a.k.a. frozen wave). You then specify the wave amplitude, the position of the wave top and the length of the wave, according to the sketch below. The default wave amplitude is zero, then the calculations are made for the ship in calm water. This static wave has its effect on the hydrostatic-related calculations, such as:

- Hydrostatics, cross curves and Bonjean tables.
- Intact stability, and all related computations, such as grain stability.
- Damage stability (where the wave does not extent **inside** the damaged compartments).
- Longitudinal strength (shear force and bending moments).

The static wave given here has no effect on the seakeeping calculations of Motions, as they do not take a single, static wave into account, but an entire distribution of waves (a wave spectrum) instead.



Wave parameters

Wave settings

5.7.1 Wave amplitude

5.7.2 Location of the top of the wave

5.7.3 Wave length

5.7.4 Wave direction

As a rule, a wave as used for the calculation of longitudinal strength or stability is taken to be *longitudinally*. By exception, it can be required to take an oblique wave, in which case here the angle of the wave direction (in degrees, relative to the centerline) should be specified at this option. Besides, the wave effect in transverse direction is linearized, which implies that the intersection of the ordinates with the wave surface is approximated by a straight line.

5.7.5 Wave type

Two types can be chosen, sinusoidal (the default) or trochoidal.

5.8 Settings page heading

This option gives the user the ability to modify the PIAS page heading for all generated output, i.e. to personalize the headings.

Note

> There are no automatic checks performed to verify if an object is overlapped by another object, but a visual check can be performed using [Print preview].

In this menu you can add and remove page headings and give it an identifiable name. Each new page heading will be the PIAS default. You can select the application of the page heading.

If no page headings are defined or all defined headings are disabled, no page heading appears in the output.

If 1 page heading is defined, and it is not disabled, then it applies to all output pages.

If multiple page headings are defined, then it can be set to use 1 page heading for the first page output and another page heading for all remaining pages output.

Page headings can be exported and imported using their respective options [Export] and [Import] under the option [File], thus a company standard may be defined and exported to be imported elsewhere.

5.8.1 Page heading objects

A page heading is constructed with the help of objects. Each object has to be positioned in with the parameters **Line number** and **Tab position**. Different types of objects have different properties. The differences are explained below.

Object type
: Properties for different types are listed below.

  Normal text
  : A user-defined text.

  Title
  : The pre-programmed title for that specific output.

  Project name
  : The project name which is defined in Hulldef or Fairway.

Date and time
The date and time when creating the output.

Date
The date, without the time, when creating the output.

Page number
Start value for page numbering.

Chapter name
The chapter name of this generated output.

PIAS version
The PIAS version of the program with which the output was generated.

PC and username
Which computer and user generated the output.

Text
Based on the **Object type** this column contains a predefined text. The object types that can be defined further are listed below:

Normal text
Any text which is not covered by a standard type.

Page number
The start value for the page numbering. If this text is left undefined then this value should be defined in the popup message when printing. If the **W** character is present in the text and the copied 'Rich Text' output is pasted into a 'Rich Text' compatible word processor, then the page numbering is automatically updated to the page numbering of the word processor.

Chapter name
Then a popup is posed at the start of a print job, as mentioned in the *Page number* object, to allow the overriding of the predefined chapter name.

Font size
The height of a line of text.

Line number
The line on which this object has to printed, i.e. the vertical positioning of an object. This is dependent on the text heights.

Tab type
The alignment of the object relative to its *Tab position*, thus a left tab has its alignment to the leftmost part of the object and a right tab to the rightmost part of an object.

Tab position
The horizontal position of an object in millimetres.

Line number 1 This is a left tab
Line number 2 This is a centre tab
Line number N This is a right tab
All the above tabs have a tab position of 110 mm > The greatest tab position is 184 mm >

Overview object positioning

With the option [Print preview] one can check if all objects, in the current page heading, are correctly positioned and not overlapping.

5.9 E-mail settings

It is possible to let PIAS send an e-mail after a calculation or print task is finished. The idea is that is sometimes occurs that a computer is busy with a lengthy calculation, and it is inconvenient time after time to look personally whether this task has already finished. In such cases it can be handy if one receives an e-mail, notifying the operator that the job is done. If it concerns a print task with output to RTF file or text file, that file will be send as attachment, so the print results are directly available. This facility has the following settings:

- Send e-mail, with three sub options:

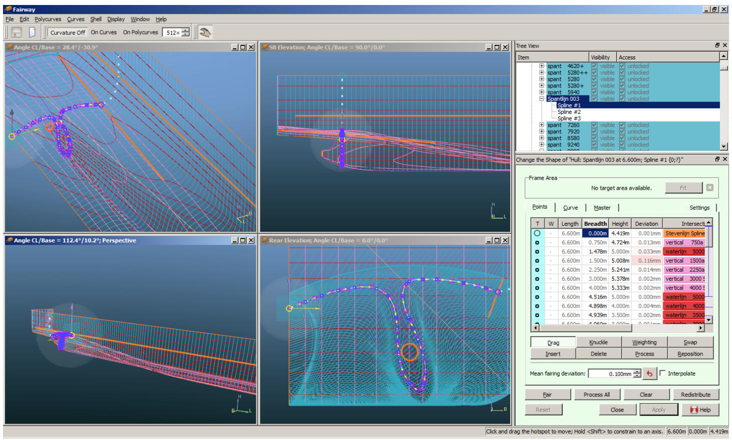
  - Never.
  - After each print task (and damage stability calculation task).
  - After print or calculation of a damage stability module.
- Senders address, recipients address and mail server. As a rule, these will always be the same for a particular computer and user, in which case it is appropriate to set those by means of external variables (see section 3.10 on page 30, External variables where this mechanism is explained). In other cases, settings in this menu overrule the external variables, and are stored and used per project. So, either nothing is filled in (a blank line), and then the external variable setting will be used, or something is filled in, and then used. By the way, both the computer and the sender must have the right to approach the mail server with e-mail commands according to the SMTP protocol. Furthermore, sending email messages according the SMTP protocol can be blocked by anti-virus software.
- Minimum time required to send e-mail. In general, it will not be desirable to receive mails from each short print command. Therefore, it can be set here how long a print job or calculation task must take, for an e-mail to actually be sent. If this time is for instance set at ten minutes, then you will only receive a mail from really time-consuming tasks. If set to zero an email will always be sent.

Chapter 6

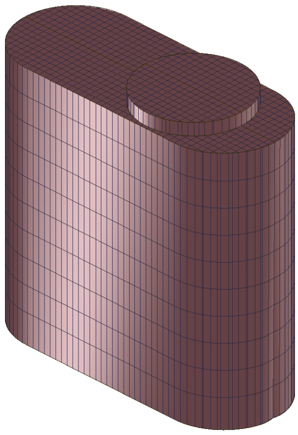
Fairway: hull shape design

Fairway is the hullform modelling module of the PIAS/Fairway suite of naval architectural software. Fairway can be used for any activity with the hullform, such as:

- *Hullform generation, both ab initio design and based on a pre-existing shape, in which developable surfaces and doubly curved surfaces may be mixed.*
- *Design modifications during preliminary design and final design.*
- *Hullform transformation.*
- *Fairing with user-controllable accuracy, up to, and beyond production tolerances.*
- *Generation of shell plate expansions.*
- *Generation of linesplans and tactile scale models (Rapid Prototyping, 3D printing).*
- *Import or digitization of hullform data, either complete or partial.*
- *Perform simple hydrostatic analyses, and farm out complex analyses to the PIAS suite.*
- *Addition of extra, user defined curves, for example extra frames.*
- *Export of hullforms, for example to general CAD systems such as AutoCAD (DXF) or Rhinoceros (IGES), to CAE systems such as NUPAS, or to Finite Element or Computational Fluid Dynamics software.*



Fairway GUI showing a complex RoRo aft body, with angled skegs.



Simple model in Fairway: the trunk of an oil tank.

6.1 Introduction

Fairway is a so-called solid modeller, which is based on a closed 3-D surface model, and wherein the user manipulates the shape of the hull by means of 3D lines, which coincide with the hull surface. As a rule, waterlines, ordinates, buttocks and 3D chines will be used for this purpose, however, the user is completely free to choose those lines which are considered necessary or handy. This introduction starts with a description of some basic concepts, followed by a set of definitions as used in this manual.

6.1.1 Basics of Fairway

1. A line consists of one or more concatenated curves. In Fairway this is called a polycurve. The user specifies the nature of the connections between the curves (fair, tangential or with a knuckle).
2. Curves are defined as NURBS, and the user can specify the curve type as:
   (a) Spline
   (b) Straight line
   (c) Circular arc
   (d) Parabolic, hyperbolic or elliptical arc.
3. Points are defined along the length of a curve. This can be intersection points with other curves, and so-called internal points used to anchor the shape of the curve. An important objective is to keep the distance between points and curves below a certain tolerance.
4. The fairness of a curve can be inspected by means of the curvature, plotted perpendicularly to the curve. Curves can be faired through its points automatically with a local scheme, where the user specifies a mean deviation between the original points and the faired curve. The user can also specify the relative weight of each individual point, in three grades: neutral, inactive and heavy. The mean deviation is analogous to the stiffness of the physical spline (the larger the deviation, the stiffer the spline), while the relative weight can be considered as a model for the weights of the so-called ducks.
5. Polycurves are connected to each other through the intersection points of point 3, and thereby form a network that describes the hull surface. Contrary to NURBS surfaces, which only exist over a regular network, this network is very shapeable. This is because polycurves need not extend over the full length of the ship, but may be defined where they are really needed.
6. Polycurves must start and end at another curve, curve ends cannot dangle freely in space.
7. Internally, the network is represented unambiguously with appropriate techniques. Without the use of these techniques a set of curves is ambivalent. Fairway, however, knows about the logical coherence between the points, curves and surfaces, so Fairway does have an unambiguous and correct picture of the object. Together with the methods from points 10 and 11, a solid shape representation is obtained.
8. When the program is used for hullform generation, the unambiguous representation is present from the start. When a digitized linesplan is used or a hullshape is imported, the representation will be created

automatically. In both cases new curves can be added by intersection with arbitrary planes and other means, they can be manipulated and removed.

9. A network of polycurves is termed “consistent” when all polycurves run through their points within the tolerance mentioned in item 3.
10. With special techniques, Fairway constructs surfaces over the meshes of the network, based on the shape of neighbouring curves. Areas are automatically detected where it is appropriate to use one surface. The surfaces have curvature in two directions, unless the user explicitly specifies that a surface must be developable.
11. Individual surfaces are connected to form a contiguous shell with tangential continuity, unless a curve is defined as a chine.
12. In this way a complete, unambiguous surface description is made, on which the following actions can be performed:
    (a) Adding new curves by means of all kinds of intersections.
    (b) Showing threedimensional views, with or without hidden line or hidden surface removal.
    (c) Calculating intersections with other surfaces, or perform boolean operations with other objects.
13. The surface is defined by the curves from item 1. If a surface is shaped insatisfactorily, then the network of curves should be adjusted.
14. If the tangent continuity from item 11 is not sufficient at some locations, extra curves should be added across that area, which *de facto* makes the continuity shift to fair.

6.1.2 Geometrical notions

This section deals with a few geometric concepts that are important for using Fairway. No mathematical definitions and backgrounds are involved, just a simple explanation, if necessary illustrated with some graphical examples. Firstly some geometric definitions regarding lines are given, followed by definitions regarding planes.

6.1.2.1 Lines

Curve

A curve is a line segment, straight or curved, without knuckles or cusps.

Polycurve

A polycurve is a concatenation of one or more curves. Initially, curves are independent from each other, so there will be a knuckle in the polycurve where two curves meet. By defining boundary conditions one can achieve various forms of transition between curves, which creates a dependency of shape at start- and end-points of adjacent curves. There are six types of polycurves:

- Frames (or ordinates), waterlines and buttocks. Thoese will speak for themselves.
- Diagonals, always with an inclination of 45°.
- Lines in an arbitrary plane. These are fixed in some plane, although not a plane of one of the previous types. With this type also diagonals under non-45°angles can be made.
- General 3D-lines are polycurves which are not fixed in some plane. For example a deck line of a ship with sheer.

The division of polycurves in these six categories is made for the comfort and overview of the user. For the program itself all polycurves are equivalent.

Knuckle

A knuckle is a point between two adjacent curves of a polycurve. These two curves are initially independent from each other.

Chine

A chine is a polycurve on which crossing polycurves have a knuckle. It is recommended to connect knuckles with a chine. In the Fairway GUI chines are often visualized thicker than other polycurves.

Spline

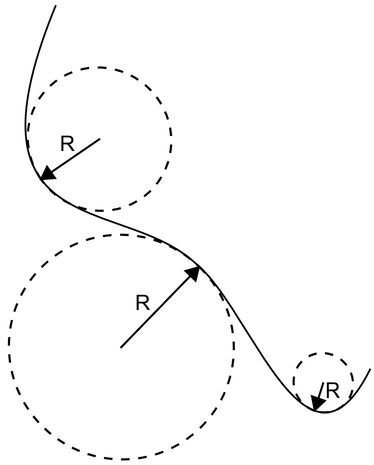
A Spline is a curve which is defined by several angular points called vertices (singular: vertex). The vertices are sometimes called control points, and together they form the so-called control polygon. The curve, as a matter of speaking, is attracted by the control polygon. You might say that the spline is a fair approximation of the control polygon. By changing the vertices the shape of the spline can be manipulated.

Line direction, left and right

In Fairway a polycurve has a certain direction. For example, the possible directions of a waterline are from

'stern to bow' or from 'bow to stern'. Fairway visualizes the direction of selected polycurves by means of an animation that reminds of waterdroplets that run along the line in the direction of the polycurve.
In relation to this, left and right are defined in Fairway as follows: imagine yourself walking on the outside of the ship, perpendicular to the shell, on the line from the beginning to the end of the line. From this position Fairway's left is at your left hand, and right at your right hand.

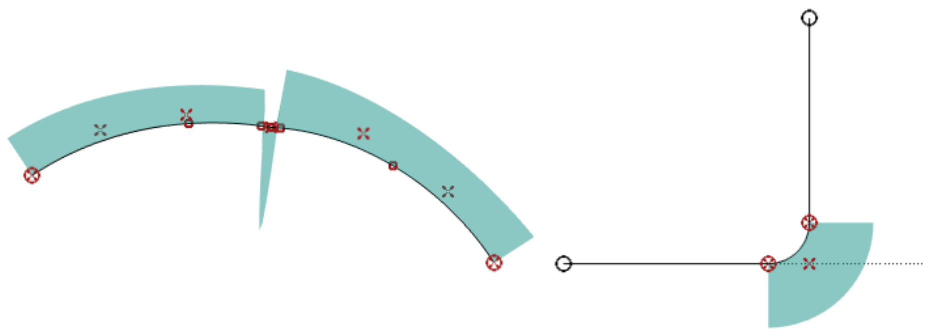
Radius of curvature
For each point of a curved line it is possible to imagine a circle which coincides with the line in the considered point. The radius of this 'fitting' circle is called the radius of curvature. In the figure the radius of curvature (R) is illustrated.



Radius of curvature R.

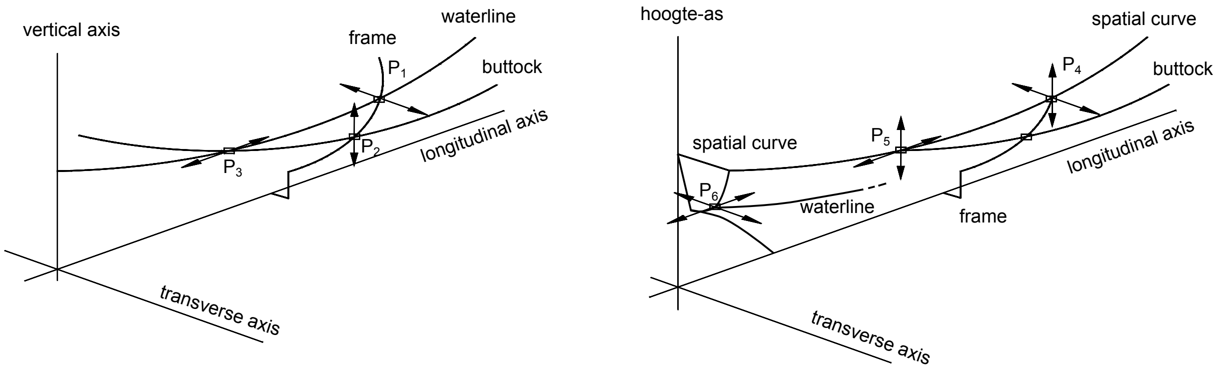
Curvature
The curvature of a curve in a considered point is defined as the reciprocal of the radius of curvature, 1/R.

Curvature plot
In Fairway the curvature is used as a tool for fairing curves. For each point of a line the curvature can be plotted perpendicularly to the considered curve, the curvature plot. A curve can be considered fair if the curvature plot is without unexpected jumps. Two examples are given below. The wild sagging in the left plot is unintended and indicates that the curve is not fair at that location. On the right the plot is discontinuous as is to be expected, between the straight lines (no curvature) and the circular arc (constant curvature).



Curvature plot.

Moving points
A point can only be moved in the plane in which the polycurve of the point is defined. A point on a frame can be moved both in vertical and transverse direction. A point on a spatial polycurve can be moved in all directions. For points in which polycurves of different type intersect the following degrees of freedom arise.

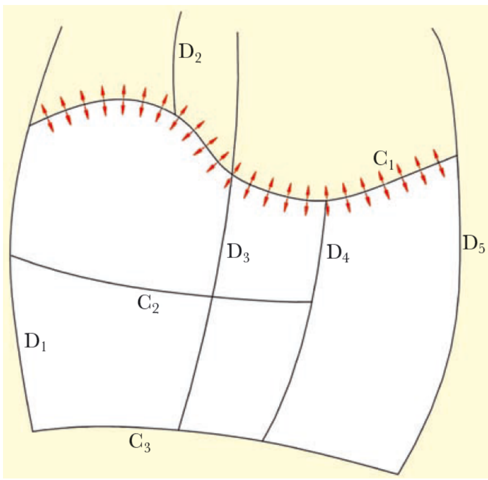


Degrees of freedom.

P_1: only transverse motion possible	P_4: transverse and vertical motion possible
P_2: only vertical motion possible	P_5: longitudinal and vertical motion possible
P_3: only longitudinal motion possible	P_6: transverse and longitudinal motion possible

Fairway manages the degrees of freedom and will offer you the available directions of motion.

6.1.2.2 Surfaces



A tangent ribbon (in red), constructed from the shape of all curves C and D at their intersections.

Face / surface
The manual mentions faces and surfaces. A face is the smallest area **generated** by intersecting lines in 3D space; faces are the meshes of the network of curves. A surface is an area **defined by the user**, bounded by intersecting lines. A surface can have certain properties, like, for example, developability. No lines can exist within a face. They can exist within a surface, as the surface may consist of several faces.

Curved surface
As discussed in the Basics of Fairway, Fairway has the option to create *curved surfaces*. These derive their shape from the shape of the neighbouring curves. So, there are no handles or functions to manipulate the surfaces, they simply arise between the curves, and the way to influence their shape is by re-shaping the curves in the vicinity. However, there is a single setting for this curved surface creation algorithm, which is discussed just below.

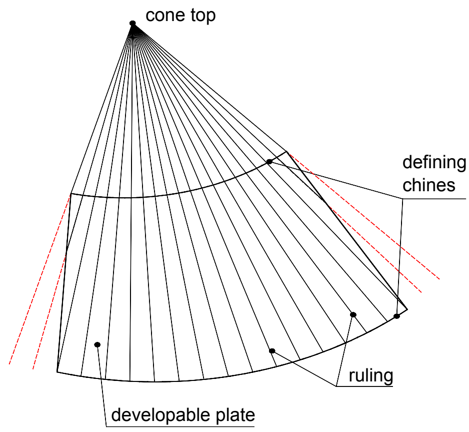
Tangent ribbon
As foundation of a curved surface, ribbons of cross-boundary tangents are interpolated. These are the so-called tangent ribbons, which ensure the continuity of the curved shape over the entire surface. These are

being constructed on basis of the shape of the curves at all mutual intersections between curves, an example is given in the figure above where the red arrows depict one such a tangent ribbon. This interpolation comes in two flavours, ‘smooth’ or ‘articulated’ at each intersection. In general, smooth tangent ribbons produce a smoother surface, but they can also cause unwanted undulations. In that case articulated tangent ribbons can be selected. Articulated tangent ribbons can be appropriate very early in the design when few curves are present that define the shape, and at the final stage of modelling when many curves are present for construction that give a high number of intersections. The type of tangent ribbon can be set for each solid individually in the menu [Object management] (discussed on page 155).

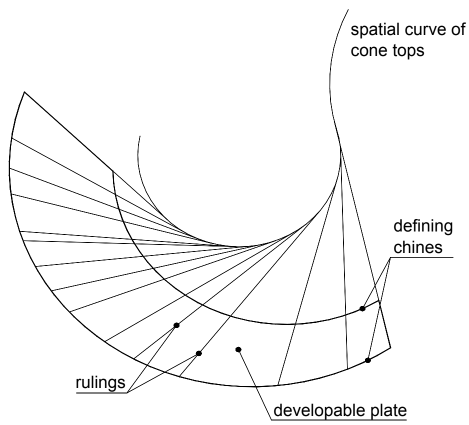
Developable surfaces

Developable surfaces are surfaces that are curved in one direction only. They have the advantage that shell plates can be formed without stretch or shrinkage. Conic surfaces are the only developable surfaces. This includes cylindrical surfaces, as these can ge seen as cones with a top at infinity. Two kinds of developable surfaces can be distinguished: single-top cones and multi-top cones. The cones may have any base.

A **single-top cone** is generated by moving a straight line about a single point in 3D space. This single point is the top of the cone. The straight line is called the **ruling**.



A **multi-top** cone can be described as a cone with a shifting top. This top moves along a curved line in 3D space. Each ruling of the cone is a tangent of this curved line at the corresponding top. If a linesplan with developable surfaces is made by hand, the use of multi-top cones is often too complicated. By using multi-top cones it is possible to create complex developable surfaces. When working with Fairway, the user need not to worry about the details of cones and tops, but only indicates the curves that bound the developable surface. Fairway calculates and displays the result, see the figure just below.



A condition for a surface to be developable is that all rulings must exist. A ruling connects a point on one defining line (see below) with a point on the other defining line in which the tangents are coplanar. The rulings are not allowed to cross and no ‘holes’ between the rulings are allowed. This is an indication that the shifting cone top has moved inside the surface boundaries, which is not physically possible. Crossing rulings and holes are illustrated in the figures below.

Forbidden constellation of rulings.

It is not necessary to watch for these defects all the time, Fairway will validate developable surfaces on its own.

Defining lines of a developable surface

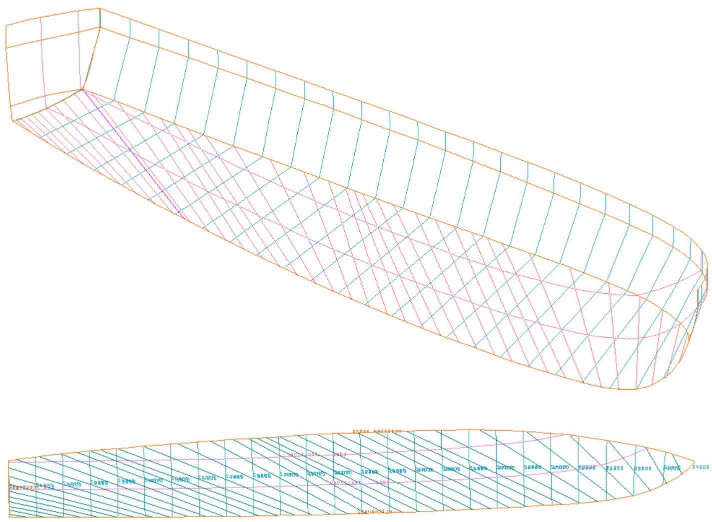
The defining lines of a surface are the two boundary lines inbetween which the rulings run. These lines are called defining lines because only these lines determine the shape of a developable surface. The defining lines must always be chines. No knuckles are allowed on the chine.

If a developable surface is defined by the rulings from a single fixed cone top or if a cylindrical developable surface is defined, only one defining boundary line can be specified.

Region

A shell region is a domain on the hull surface, bounded by a closed contour, that may have specific properties. For example, in case of developable surfaces the property is *developability*. The user can define a region by defining its corners on bounding polycurves. Shell regions are used for e.g. developable surfaces, shell plates — which can be used for shell plate expansions and templates — or to define a phantom face. The *modus operandi* with shell regions is discussded in section 6.3.5.24 on page 101, Define Shell Region.

The figures below illustrate a developable surface. The first figure gives a 3D view of a completely developable hull shape. The bottom plate has been selected for processing. The chine and the stem contour are the defining borderlines. You can see the rulings on the bottom plate. The lower figure shows the developed bottom plate.



6.1.2.3 Solids

When designing the shape of a ship hull it may be convenient to consider the vessel to be composed by multiple building blocks. Such a block can be imagined as a closed and solid bdy, so it will be denoted by the word *solid* hereafter. In principle each solid is completely independent, and contains its own points, curves and surfaces. Solids can also be imported or exported independently. Also functions are available to glue solids together — e.g. a bulbous bow which is intersected with the hull — however, these are still experimental.

6.1.3 Definitions and concepts

6.1.3.1 Phantom face

It is important to realize that in Fairway the network of polycurves defines basically a closed surface. This implies that without special provisions the hull will also be closed over deck and over center plane. However, for the sake of clarity polycurves over center plane are by default not generated. This is the result of the center plane and deck region being defined as a *phantom plane*, which is a face (please refer to section 6.1.2.2 on page 59, Surfaces for its definition) in which newly generated polycurves do **not** extend. A phantom face can be toggled on or off at any desired moment, seesection 6.3.5.23 on page 100, Phantom Faces. However, when toggling the network remains unchanged, so at that moment no polycurves will be added or removed, it only will have its effect on future actions.

There is rarely a need to change the phantom face setting. Only when it is explicitly desired that polycurves do also extend over the center plane, the phantom face has to be switched *off*. Another case is in the modeling of a deck with one or more hatch openings, then it can be practical to define the openings as phantom faces.

6.1.3.2 Polycurve visibility

This attribuut nidicates whether a polycurve is visiible in the GUI. The visibility will occasionally also be used at export to other file formats, such as DXF or IGES (when it is asked whether only the visible polycurves should be included in the export). The visibility can be set in the GUI (see paragraph 6.3.2.2.3 on page 66, Polycurves).

6.1.3.3 Polycurve locked

A polycurve can be ‘locked’, which comes in two variants, namely ‘lock against removal’ and ‘lock against removal and modification’. Locking against removal especially comes in handy when working with section 6.1.3.6 on this page, Polycurve positions sets. For example, when a group of polycurves is added, of which some coincide with the already existing polycurves. When afterwards this group is erased, the already existing will also be removed, which obviously will not be the intention. Locking the original polycurves before adding the groups will prevent this. Locking against modification can be used for extra safety. Especially during fairing, it might prove to be useful to lock certain polycurves which are absolutely not allowed to change. The polycurves can be locked in the GUI (see paragraph 6.3.2.2.3 on page 66, Polycurves).

6.1.3.4 Construction Water Line (CWL)

The attribute ‘CWL’ is applicable to polycurves of the type waterline, and indicates which waterline is the ‘construction water line (CWL)’. The CWL is only applied in Fairway, in graphical presentations (such as those of section 6.7 on page 135, Show (rendered and colored) surfaces) to show the waterplane and/or to apply differents colors below and above the water surface. The CWL attribute can be set in the GUI (see section 6.3.5.12 on page 84, Properties of polycurves).

6.1.3.5 Deck at side

Indicates whether this line is a ‘deck at side’. This is only relevant for the conversion of the hull to PIAS’ frame model (please refer to section 6.8.1 on page 137, Convert this Fairway model to PIAS model for that function), because there the frames from the frame model will be cut off at the level of that deck at side. This attribute can be specified in section 6.3.5.12 on page 84, Properties of polycurves.

6.1.3.6 Polycurve positions sets

A polycurve position set (prior to 2012 known as a group of line places) is a set of systematic locations for frames, waterlines or buttocks, which can be shown or added at those locations. This mechanism offers the option of some kind of *preview*, showing polycurves without them actually being added to the model (see section 6.3.5.18 on page 88, Show Indicative Intersections). These groups of locations are managed in a menu which can be called from paragraph 6.3.5.8.2 on page 81, Position sets. A description of the menu follows below.

Attention

> A position-set is nothing more than a set of declared positions, irrespective of whether matching polylcurves exist or not. Selecting or deselecting a certain set will not add or remove polycurves at the positions of that set.

Selected
: This value can be toggled to either ‘yes’ or ‘no’, and indicates whether this set is active in the GUI (corresponds to the check mark in the list of sets in the GUI).

Name
: This value contains the name of the group. Specifying a clear name may prevent against unwanted adding or removing of lines. A useful name may be something like a short description of the group of lines.

Line type
: Defines the type of the entire group of lines, the following types can be chosen: frames, buttocks and waterlines.

minL, minB, minH, maxL, maxB, maxH
: Define the region in which the lines of the concerned group are being added. By means of this function, frames, waterlines and buttocks that do not cover the entire hull shape, can be added. The lines are being cut off on the nearest intersecting line outside the defined domain.

Further definition of a set can be done by clicking it, or by <Enter>. Then a window appears where the following properties can be set:

Multiple
: This value can be toggled to either 'yes' or 'no'. This specifies whether this definition is appropriate for one line or more than one.

Beginning cq. location
: This is the first location of the group of lines, or, alternatively, the only location in case *Multiple* is toggled to 'no'. The value is the length, breadth or heigth of the first line. Recall that length, breadth or heigth depends on the chosen line type (frame, buttock or waterline).

End
: The last location of the group of lines.

Increment
: Defines the distance between each line. This value can be filled in directly, or, alternatively, indirectly by the value *number of intervals*.

Number of intervals
: The value is the number of lines which fit between the *'beginning'* and the *'end'* with the given increment. Therefore if this value is modified, the increment value is modified automatically, since the beginning and the end are constant during this operation.

Finally, there is a supporting function: if frame spacings have been defined in Config (as discussed in section 7.2.1.3 on page 167, Frame spacings) and a set of **frame** positions is being edited, then the option [Config-import] appears in the menu bar. This option imports the frame spacings as defined in Config into the current set.

6.2 Start and main menu

After starting Fairway, the filename of the project is asked. When you start a new Fairway project, type in the name (and path) for the project. Because Fairway saves a variety of different files, it is a good idea to start in an empty directory. Next the following menu will appear:

New Fairway project (file name)
Start with new hull design (minimal hull)
Start with new hull design (rectangular barge)
Start with new hull design (horizontal cylinder with R=B/2)
Import hull form from existing PIAS file
Import hull form from CXF/SXF file
Import wireframe from IGES file
Import wireframe from DXF file
Start with empty model (advanced)

The first three options provide the best starting point for a new design, the next steps of which is described in the paragraph below. The fourth option produces a Fairway model from an existing PIAS frame model (in section 2.10.2 on page 16, Hull form representations the different hull shape models are introduced). Of such a model the frames are precisely defined, so they will all be included in Fairway. Stem/stern contours and deck line(s) need not be specified in PIAS, so their explicit information is not available. Attempts are made to generate these as completely as possible, but this requires sufficient cohesion in the frames in the vicinity. If stem/stern contour or deck line have an undesired shape then it will need manual adjustment in Fairway (or, alternatively, it can be re-imported with a PIAS form which has been given more coherence by means of more frames).

The fifth option allows the import of a wireframe model, closed hull from a pair of SXF/CXF, as discussed in paragraph 6.3.7.1.3 on page 116, Import ship hull models in SXF/CXF format and onwards. The import from IGES and DXF produces a set of unconnected curves, as a wireframe. Wireframes can be manipulated within Fairway and converted to solids as described in section 6.3.7 on page 113, Wireframes. The detais regarding IGES and DXF are discussed respectively in paragraph 6.3.7.1.2 on page 115, Import 3D lines from IGES format and in paragraph 6.3.7.1.1 on page 114, Importing 3D lines from DXF format. When starting with an empty model (last option) the next step would be to create an object in [Object management] (discussed on page 155), as well as defining the design main dimensions, see section 6.4.1 on page 129, Main dimensions (design) & hull coefficients.

If "Start with new hull design..." is selected, the following menu will appear:

Design main dimensions

Project name	
Length PP	
Moulded breadth	
Moulded draft	
Moulded depth	
Blok coefficient	(optional target value)
Center of Buoyancy (% of Lpp from Lpp/2)	(optional target value)
Midship coefficient	(optional target value)

After entering the main dimensions in this menu, if the first option was chosen, an initial model will be generated by Fairway (with the specified main dimensions), containing one deck line, one stem/stern contour and one frame. With the second option a rectangular barge of the correct main dimensions is created, with the third option a cylinder.

This model is the base for subsequent actions. Values for the block coefficient, LCB and midship coefficient are used as target values for the sectional area curves. These values are not necessarily equal to the final hydrostatic particulars, it is up to the user to achieve those values, with the aid of the controlling mechanisms that Fairway offers.

After the hull is read into Fairway the following main menu appears:

Hull shape design

1. Graphical User Interface (GUI)
2. Define main dimensions and other ship parameters
3. Hullform transformation
4. Settings and auxiliary tools
5. Show (rendered and colored) surfaces
6. Export of hullform
7. Define and generate lines plan
8. Shell plate expansions and templates
9. File and object management

The upper bar of this main menu contains the [Setup] option, which allows general Fairway settings to be specified. Details are discussed in chapter 5 on page 41, Config: General project configurations. This chapter ends with a set of appendices in Appendices.

6.3 Graphical User Interface (GUI)

This section starts with an introduction to the general structure of the grafical user interface of Fairway, followed by ways to change the view on the model. Next the dragger is introduced, which is a graphical entity for interactive manipulation of 3D positions, specifically designed for Fairway. The section continues by documenting the various common modelling actions by which the model can be changed, and supporting functionality. Details of working *with wireframes is presented in a dedicated section, including the process of the conversion into solids. Finally there is a section that you can consult if you encounter problems.*

6.3.1 Start up

At start up a progress bar in the status bar indicates how solids are read into the GUI. This process is completed when it reads “Ready” in the status bar, and the GUI is responsive to the mouse and keyboard. Curves are represented by a course polyline initially in order to get ready for user interaction quickly, and the system continues preparing for final display accuracy and the curvature information while the user is working with the model. Care is taken that curves that are under the mouse pointer are prioritised, so that these are always displayed at high accuracy. Because of this task, CPU load can be high for the first moments after loading a large model, but it will normalize eventually. For configuration of the display accuracy see section 6.6.1.3 on page 134, Configuration GUI.

6.3.2 GUI Structure

The GUI consists of a central modelling area, around which various control- and information panels can be positioned, according to the preferences of the user. The menu bar along the top and the status bar along the bottom are the only static elements in the main window, everything else can be repositioned, detached and embedded by means of drag-and-drop. To show or hide a particular window, click on the corresponding item in the [Window] menu.

6.3.2.1 Modelling Views

The modelling area can be filled with one or more modelling views in various layouts. A new modelling view can be opened with [Window]→[new]. When there are several views open in the modelling area, the one under the mouse pointer is automatically activated and raised to the front in case of overlapping views. A view can be prevented from being occluded by the active view by selecting “Stay on Top” from the drop-down menu under the top left icon of the view window. View windows may be layed out automatically filling the modelling area by selecting [Window]→[Tile].

As Fairway has excellent controls for changing the view angle and zoom level in a view window (see section 6.3.3 on page 68, Navigation: Pan, Zoom and Rotate) many people prefer working in a single maximized window, which you get by clicking on the corresponding button in the top right corner of the view window frame. Alternatively, several large views may be stacked on top of each other by selecting [Window]→[Stack]; which gives each window a tab pane along the top for switching between views.

The projection of the view can be toggled between parallel projection and perspective projection from the context menu, by clicking the right mouse button over the view window in question. Since manipulation of curves in various planes is very well handled in Fairway and independent from the view angle or projection, you may even prefer to model and manipulate in perspective, as it helps with spatial orientation and to differentiate curves in the foreground from the rest of the model; it is a depth cue.

When the GUI is closed, the view window layout and view angles are stored and restored when the GUI is reopened at a later time.

6.3.2.2 Tree view

The tree view contains a hierarchical list of the elements of a model. It can be shown and hidden from the menu [Objects]→[Tree View], and be repositioned to another location — even another screen if you have one — by dragging its title bar. A double-click on the title bar toggles between floating above the main window and embedded within.

The tree can be expanded and collapsed by clicking the small triangles to the left of the items, or by double clicking the item itself. To expand an item and all its sub-items, select the item and press <*>. Every solid has a sub-item for visualisation of its shell surface, followed by sub-items for every kind of polycurve.

6.3.2.2.1 Solids

Solids appear as the top-level elements in the tree view. A solid can be hidden or shown by toggling the check box in the Visibility column, or changed between active and inactive by toggling the check box in the Access column. An inactive solid cannot be manipulated, which is indicated graphically with a uniform colour. When a solid is hidden it is automatically made inactive, and when a solid is activated it is automatically made visible. These changes are recorded in the action history and can be undone, see paragraph 6.3.5.1.1 on page 74, Undo and redo.

The order of solids in the tree view kan be changed by bringing up the context menu with a click on the right mouse button and the option [Relocate in tree]. This change is saved with the model, but since it has no implication on the action history it cannot be undone.

Whether or not the solid is buoyant can also be changed using the context menu. This change can be undone.

6.3.2.2.2 Shell

The visualisation of the shell can be toggled for each solid individually with the check box behind the [Shell] item. To reveal or hide the shells of all solids at once, the [Show All Shells] and [Hide All Shells] buttons in the [Display]→[Rendering...] window can be used. That window also contains the options with which the visualisation of the shell can be adjusted.

The [Shell] item can be expanded to reveal the [Material] properties. Both the color and transparency of the surface can be changed here with a double-click on the corresponding value. However, the shell of inactive solids is rendered uniformly as set in [File,Preferences...,Curves,Solids,Inactive color].

Changes to the shell visualisation settings in the tree view are saved with the model, but they are not recorded in the action history as they do not affect modeling actions. The settings in the [Display]→[Rendering...] window are user preferences and are thus shared across projects.

6.3.2.2.3 Polycurves

When a polycurve is selected graphically, the tree is automatically expanded and scrolled to bring the selection into view. Selections can also be made in the tree view with the left mouse button <LMB>. A composed selection is made with <Ctrl+LMB> and a range is selected with <Shift+LMB> or holding <LMB> while dragging over the list. It is also possible to select all expanded items with <Ctrl+A>. A click with the right mouse button on a polycurve in the tree view brings up the context menu from which the polycurve can be renamed, and actions on that particular polycurve can be started.

There are two additional columns in the tree view containing check boxes for visibility (see section 6.1.3.2 on page 62, Polycurve visibility) and access (see section 6.1.3.3 on page 62, Polycurve locked). If a particular polycurve is hidden, the visibility check box of its parent item as well as of its parent solid is filled to indicate that not all polycurves are visible. Clicking this box in the parent solid will unhide all polycurves in the solid. Clicking this box in the parent item unhides all child polycurves if there are hidden polycurves, otherwise it hides all of them. The menu option [Display]→[Unhide All] will unhide all hidden items in the entire model at once.

The Access column controls the active/inactive state of solids, and the lock status of polycurves. Inactive solids change to a uniform colour and cannot be modified. Polycurves can be in one of three lock states:

1. Unlocked
2. Irremovable
3. Fully locked

An irremovable polycurve can still be modified, but not be deleted. The polycurve lock state is cycled with a click on its check box.

State changes are recorded in the action history and can be undone and redone, see paragraph 6.3.5.1.1 on page 74, Undo and redo.

6.3.2.2.4 Curves

Polycurves always have at least one child curve. Its name indicates the type of the curve and a running number, followed whether it has a master curve and the number of slave curves, if any. The context menu, brought up by a click with the right mouse button, offers actions that can be started on that particular curve.

6.3.2.3 Levels of information and control

The Fairway GUI is designed to present both information and control at different levels of interaction, at the right time, without the user needing to ask for them or to search for them in the menu's. Controls that are irrelevant to the task at hand are therefore abscent and cannot cause confusion or distraction.

6.3.2.3.1 Unselected polycurves

Unselected polycurves of active solids are colour coded according to the plane in which they are defined. Chines are displayed with an increased line width. The colours and widths can be configured according to your preferences from the menu [File]→[Preferences...]→[Curves]. Polycurves that are part of inactive solids are uniformly displayed in the *inactive colour*. In short, the following information is available at first sight:

- Whether a polycurve is part of an active solid.

- Whether a polycurve is a frame, waterline, buttock, diagonal, planar or spatial polycurve.
- Whether it is a chine or regular polycurve.

6.3.2.3.2 Prelit polycurves

When the mouse pointer is moved over the model, polycurves under it will light up in a distinct colour, the *prelight colour* (yellow by default). This is done for two reasons.

Firstly, it aids the user in making selections. It hints which curve or polycurve will be selected if the left mouse button would be pressed. In case of an ambiguity, when there are more than one polycurves under the cursor, all will light up and at mouse press a pop-up will differentiate the items. If selection of the prelit polycurve is prohibited in the current modelling context, e.g., when attempting to delete a locked polycurve, then it will be prelit using the *prohibited colour* (red by default). The reason why selection is prohibited is given in the status bar.

Secondly, prelighting reveals more information about the polycurve:

- Knuckles are displayed with a small circle, showing the subdivision into curves.
- The existence of boundary conditions at the knuckles are indicated, by displaying tangents with dotted lines.
- A curvature plot indicates the fairness of the polycurve, if switched on. [Display]→[Curvature Plots]→[On Polycurves] produces a plot along the full length of the polycurve, while [Display]→[Curvature Plots]→[On Curves] only shows the curvature of the curve that is directly under the mouse cursor or that is being manipulated (see paragraph 6.3.5.19.8.2 on page 96, Show Curvature Plot). Similar switches are found on the tool bar [Window]→[Curvature Plots]. The plot is constructed by setting out the curvature value perpendicularly to the curve. The scale of the plot can be adjusted with the <Up> and <Down> arrow keys.
- The single curve directly under the mouse cursor is further accented with additional information:
  - The vertices that define the shape of the spline.
  - The points through which the spline is faired.
    - Inactive points are marked with an outlined arrow pointing upwards.
    - Heavy points are marked with a solid arrow pointing downwards.
- In the status bar the currently prelit curve is identified with
  - The name of the solid.
  - The name of the polycurve, as well as its position (when applicable).
  - The type of the curve and its running number.
  - If the curve has a defined master curve, that curve is identified between braces.

6.3.2.3.3 Selected Polycurves

Polycurves, consisting of curves, can be selected on two levels. Firstly the polycurve as a whole can be selected, and secondly a curve as an individual can be selected. If a curve is selected, its parent polycurve is always selected as well. Selections can be made in the modelling area as well as in the Tree view (discussed on page 65). A compound selection can be made by holding the <Ctrl> key, or by dragging over items in the tree view.

The current modelling context may limit the freedom to make selections. For example, when deleting polycurves it is not possible to select curves, and when manipulating a curve then a compound selection is not possible.

A selected curve or polycurve is highlighted using an animation reminding of a string of droplets running along the line or of marching ants. The speed of this animation can be configured with the value of *Dash animation speed* in [File]→[Preferences...]→[Curves]. Apart from giving a visual distinction, the animation serves an important functional purpose:

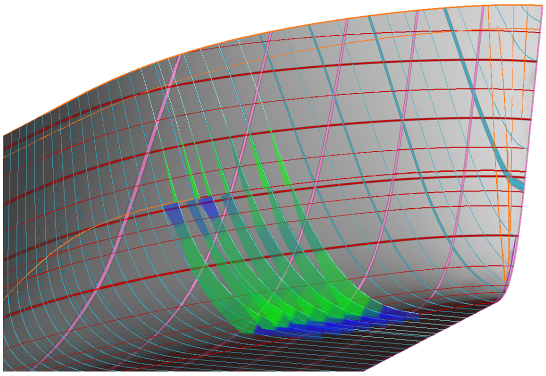
- The *direction* of the polycurve is indicated (for the definition of polycurve direction see section 6.1.2.1 on page 57, Lines).

Moving dashes are also practical, as they do not permanently obstruct the view on details of the drawing.

There is a second set of defined colours, associated with the curve type: Spline, straight line, circular arc and (other) conic arc. A selected curve is displayed in this colour, with ants in the *prelight colour*. Other parts of a selected polycurve are coloured normally, but ants are coloured according to the type of the underlying curve.

Furthermore, points and vertices of selected curves become “hot”, meaning that they themselves light up when the mouse pointer is positioned over them. When clicked, a Change the shape of a curve (discussed on page 89) action is started, allowing points and vertices to be dragged instantly.

If switched on, curvature plots also show on selected polycurves. If a set of consecutive polycurves is selected in parallel planes, which is easiest done from the tree view (discussed on page 65), a plot appears on each of them. This visualises curvature transitions across curves, which can be a valuable insight. You way want to enable colour coding of the curvature plot, so overlapping plots can still be read, using [File]→[Preferences...]→[Curves]→[Curvature].



Curvature plots on a sequence of selected polycurves, coloured according to curvature gradient.

6.3.2.4 Keyboard operation

At SARC we have a focus on keeping mouse travel distances low, and the previous Fairway GUI was known for its keyboard shortcuts and the swift operation they allowed. Even though the *current* GUI has more buttons and dials than its predecessor, we have maintained our commitment and the keyboard is still a fully supported control device.

The menu bar is activated through the <Alt> key like you are used to, after which mnemonics are displayed for the keys that activate the menu items. Some items like [File]→[Save], [Undo]→[Undo] and [Undo]→[Redo] can be activated without opening the menu, by pressing the shortcut key combination that is displayed to the right of the item in the menu.

As discussed in section 6.3.5 on page 73, Modelling actions, many modelling actions bring up a dedicated panel with relevant controls and information. This *action panel* stands out visually with a distinct background colour, so it is easily located on the screen. Most controls on the action panel have a corresponding item in the context menu, which can be brought up with a click on the right mouse button in a modelling view. Whenever the context menu is up, hotkeys are displayed over the controls on the action panel, so they are easily memorised while using the program. These keys are “hot” whenever the control is visible, unless an input field has keyboard focus. Pressing a hotkey simulates activation of the associated control on the action panel.

6.3.2.4.1 Numerical input

Fields for numerical input always show the unit of the number, unless it is dimensionless. If digits are selected in the field, then these are replaced instantly upon the first key-press. Double-clicking selects the digits before or after the decimal mark, and a third click selects the entire number.

Often, real numbers are displayed in reduced precision to save space on screen, but when they are being edited the field may expand to reveal more digits. This allows the number to be inspected and edited at a higher precision.

If small arrow buttons are shown next to the field then these may be clicked (and held) to change the value in predefined small steps. The same can be accomplished by pressing the <Up> and <Down> arrow keys. Bigger steps are made with the <Page Up> and <Page Down> keys. A third way of adjusting numbers is by rolling the mouse wheel over an input field. If you do that while holding the <Ctrl> key, then bigger steps are used. For this it is not necessary to click on the field first, it suffices to place the mouse pointer over the field and start rolling.

6.3.3 Navigation: Pan, Zoom and Rotate

Panning, zooming and rotating is collectively known under the term *navigation*. Fairway provides several interfaces for navigation, all of which are designed to not interfere with actions for selection and manipulation. That is, no operation must be interrupted or cancelled for navigation. When used in isolation, the following always applies:

- The **left mouse button** <LMB> is used for selection and manipulation.

<table>
<tr><th>Pan</th><th colspan="2">Zoom</th><th>Rotate</th></tr>
<tr><td rowspan="2"><MMB></td><td><Ctrl + MMB></td><td rowspan="3">Wheel</td><td><Shift + MMB></td></tr>
<tr><td><MMB + RMB></td><td><MMB + LMB></td></tr>
<tr><td><F4><LMB><F4></td><td><F4><Ctrl + LMB><F4></td><td><F4><Shift + LMB><F4></td></tr>
<tr><td colspan="4">3Dconnexion navigation device</td></tr>
</table>

Navigation Controls.

- The **middle mouse button** <MMB> is used for navigation.
- The **right mouse button** <RMB> brings up the context menu.

If you don't have a mouse with a middle button you can use the Navigation mode (discussed on the next page). If the middle mouse button does not work then try to troubleshoot it (discussed on page 127).

Table 6.1 lists ways to control general pan, zoom and rotate functionalities. Various special navigation instructions are given below, such as *zoom all*, *reorient on curve*, *spin*, *fly through* and others.

6.3.3.1 Current orientation

The current view direction is always displayed in the title bar of the view as two angles: one is the view direction relative to the center plane and the other relative to the base plane. If the view direction happens to be perpendicular to one of the main planes, then this is also indicated in words.

The orientation of the model relative to the viewer is indicated by the set of orientation axes displayed in the lower right corner of the view window. The positive length, breadth and height directions are each represented by an indicidually coloured arrow. The orientation axes can be switched on and off and colours can be customized in [File]→[Preferences...]→[General]→[Orientation axes].

For changing the orientation please consult section 6.3.3.4 on this page, Rotating.

6.3.3.2 Panning

Panning brings different parts of the model into view. Panning is possible in these ways:

- Press <MMB> and drag.
- Click and release the <MMB> to pan the clicked location to the middle of the modelling area.

6.3.3.3 Zooming

Zooming happens in the direction underneith the mouse cursor. Zooming is possible in these ways:

- Press and hold <Ctrl> and <MMB>, then drag up or down.
- Press and hold <MMB> first, followed by <RMB>, then drag up or down.
- Wheel up or down.
- Zoom in on a curve with just a click on the curve (<Ctrl + MMB> or <MMB + RMB>), without dragging.
- Zoom all with a click in the background (<Ctrl + MMB> or <MMB + RMB>), without dragging. If you rotate after having done this, the zoom level is adjusted on the fly to make the model fill the view, until you pan or zoom explicitly.

6.3.3.4 Rotating

Rotating the camera around the model is possible in these ways:

- Press and hold <Shift> and <MMB>, then drag.
- Press and hold <MMB> first, followed by <LMB>, then drag.
- View a planar polycurve in-plane with just a click on the polycurve (<Shift + MMB> or <MMB + LMB>). Click the same polycurve once more to rotate 180 degrees. This is sometimes called *to reorient on a curve*.

The center of rotation is set to the center of the visible part of the bounding box of the model. When dragging the mouse from left to right the camera rotates around a vertical axis. When dragging away or towards you the camera tilts around its horizontal axis, whilst keeping the center of rotation in view.

6.3.3.4.1 Spinning

If, while rotating, the <MMB> is released in the middle of a dragging motion, then the camera will continue to orbit around the rotation center, until the <MMB> is pressed again.

6.3.3.5 Perspective views

The view projection can be toggled between orthographic and perspective using the context menu, or with <Ctrl + Shift + P>. In perspective views, panning rotates the camera around its own center, synonimous with the concept of panning in photography. But zooming is replaced with dollying, meaning that the camera position moves forward or backward. By combining pan and dolly it is thereby possible to “walk” or “fly” through the model in perspective projection. And because dollying happens in the direction under the mouse pointer (as is zooming) it is possible to translate the camera sideways without changing its orientation by dollying in and out in different directions.

6.3.3.6 3Dconnexion navigation device

Fairway has built-in support for the 6-degree of freedom navigation devices from 3Dconnexion, making it possible to pan, zoom and rotate simultaneously in one smooth motion. The SpaceNavigator devices have two buttons. The left button resets the view to view all of the model, the right button brings up the device configuration panel.

If you have trouble using the navigation device then try to troubleshoot it (discussed on page 127).

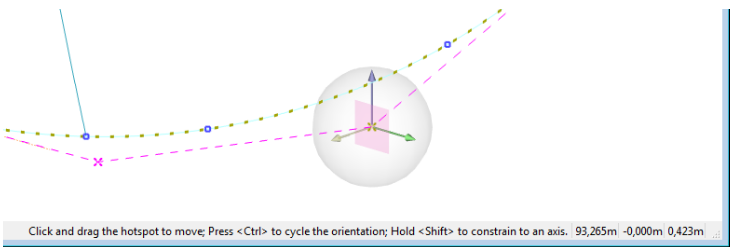
6.3.3.7 Navigation mode

In navigation mode, the left mouse button acts as the middle mouse button, so that it can be used for navigation. Toggle the navigation mode on and off in either of the following ways:

- Press <F4>
- Select the menu [Display]→[Navigation Mode]
- Click the “hand” icon in the Navigation tool bar. By default, that tool bar is hidden, but it can be shown from the General tab in the Preferences dialog, [File]→[Preferences...]→[General]→[Tool Bars].

6.3.4 The dragger: interactive graphical positioning

Whenever a position can be manipulated graphically, a so-called dragger appears in the modelling views. It consists of one or more arrows contained in a translucent sphere that loosely resembles a soap bubble. This sphere marks the dragger “hotspot”: to interact with the dragger you need not aim precisely at the arrows, it suffices to click on the hotspot and start dragging.



Dragger with status information.

The dragger attaches to a movable entity such as a point or vertex, and thereby gives a handle on its position. If there are more than one movable entities in the view, then the dragger jumps to the one that is nearest to the mouse pointer, so it is easy to switch focus between them.

Whenever a dragger is present, its three-dimensional coordinates are displayed in the right end of the status bar, along with a concise usage instruction.

6.3.4.1 Freedom of motion

The arrows of the dragger indicate the positive directions of the freedom of motion. If there is just one arrow then it can only be dragged linearly along a single axis. If there are two arrows then dragging is possible in the plane that contains the arrows, which is also indicated by a small square at the center of the dragger. If the dragger is attached to an entity that is free to be positioned in three dimensions, then this is indicated by a third orthogonal arrow. The third arrow is transparent to indicate that it is inactive. By holding the mouse pointer over the hotspot and pressing <Ctrl> you switch to a different pair of arrows, changing the plane of motion. The square in the middle marks the current plane.

If a dragger has more than one arrows, you can constrain motion linearly along one of the arrows by holding <Shift> while pressing the mouse button. This selects the arrow that is closest to the mouse pointer, as indicated by a dash-dotted line.

Sometimes an entity's freedom of motion does not coinside with the main axes, as is the case for points on a curve that is defined in an oblique plane. In that case, arrows are displayed that each are contained in the oblique plane *and* in one of the main planes. These are then coloured according to the respective main plane. So if, for example, you would hold <Shift> and drag the waterline-coloured arrow (red by default) you would move the dragger horizontally. Draggers of this type *can* have three arrows, but they are all coplanar. Also, the plane in the middle of the dragger is cropped by a cube, making it possibly six-sided, so its edges still run parallel to the main plane, helping you to interpret its orientation.

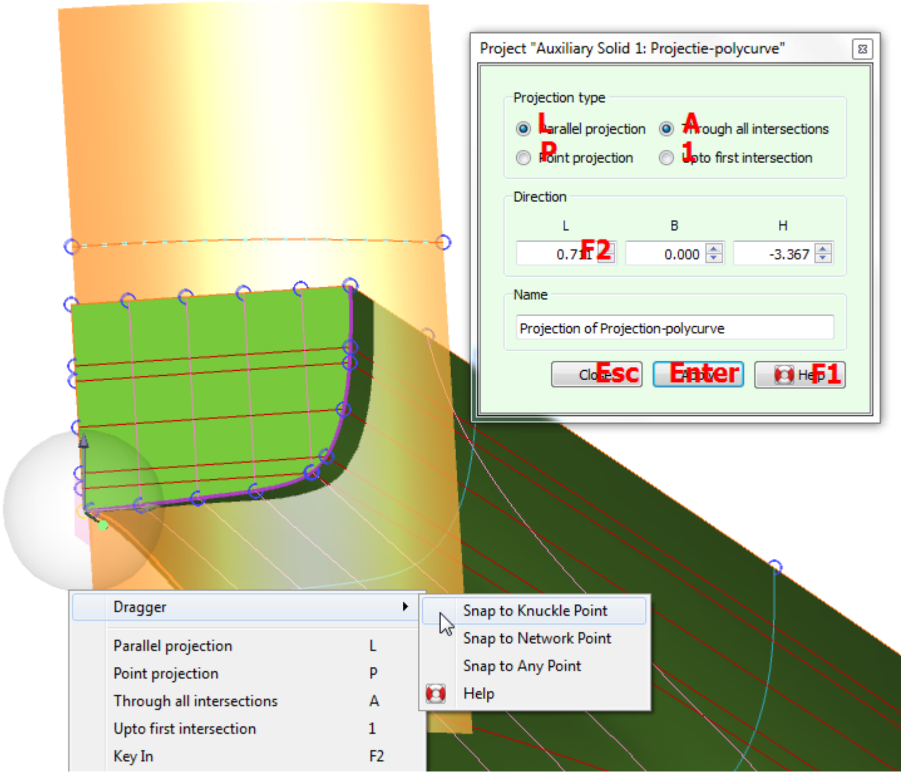
6.3.4.2 View point induced constraints

So the direction of motion is independent from the view direction; it is solely under the control of the dragger. There is a chance however, that one arrow is close to parallel to the view direction, or that the view direction is close to being collinear with the current plane of the dragger. This could cause excessive translation in the view direction, and to prevent that, motion is automatically constrained to the other arrow if there is one, or fully constrained otherwise. An arrow that is made inactive this way will be transparent to indicate that freedom of motion does exist in that direction, but that the view direction must be changed before it can be moved accordingly. Also, whenever a dragger is being dragged while there is a view point induced constraint, an attention notice is given in the status bar.

6.3.4.3 Snapping to other points in the model

Sometimes a position needs to exactly coinside with or be aligned with another point that is already present in the model. A faster alternative to keying in the coordinates manually is to bring up the context menu by clicking the right mouse button *over the dragger*. You will see a sub-menu called [Dragger], containing three options:

- [Snap to Knuckle Point] will light up all knuckle and end points in active solids, from which one may be selected to drag to.
- [Snap to Network Point] will do the same as above, but will include all points that define an intersection between curves.
- [Snap to Any Point] will offer all points in active solids, including internal ones.



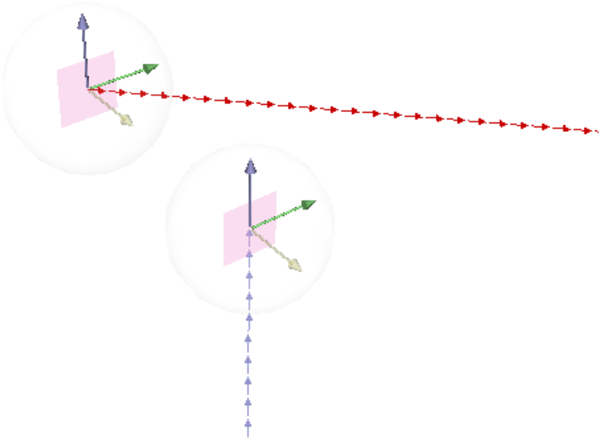
Using dragger snap to define the direction of projection.

When a point is highlighted, a dash-dotted line is drawn from the current position to where the dragger wil travel. If the dragger is not free to translate in any direction, this might not coinside with that point. This makes it possible to align a point in say, a frame, with another point in another frame, in height and breadth. This line of travel is color-coded according to its direction, using the colors associated with the main planes and axes. When the line is not parallel to any of these, then it will be colored like a curve in an oblique plane.

The coordinates displayed in the right end of the status bar normally indicate the current position of the dragger. But when snapping to other points they indicate the position that the dragger would translate to, for the currently highlighted point.

6.3.4.4 Dragging a direction vector

Besides the manipulation of positions, in some actions Fairway allows the manipulation of a direction represented by a vector. Instead of requiring that the vector is always defined at a fixed origin, Fairway defines the direction as the difference between two positions that can be manipulated independently. The vector is displayed as a string of small arrows such that the direction is clear, and if it coinsides with one of the main axes or main planes it adopts the corresponding color. An ordinary dragger jumps to the start or end of the vector, whichever is closest to the mouse pointer, by which respectively the whole vector can be moved or its direction can be changed. By snapping the dragger to existing points in the model as explained above, it is easy to measure directions and angles in the model.



Examples of defining a direction by dragging the “from” and “to” positions.

6.3.4.5 Dragger customization

It is possible to customize the dragger. For exampple, when producing illustrations it may be desirable to adjust the dragger to a white background. Technically confident users are referred to section 6.A.5 on page 162, Customizing the dragger appearance (advanced) for instructions and examples.

6.3.5 Modelling actions

When we at SARC designed the Fairway GUI we made an inventory of the ways in which a model can be changed, and clustered them into a smaller number of modelling actions. This has resulted in a clean user interface that is to a high degree self-explaining, and it has enabled us to implement some very powerful features, some of which are quite unique.

This section opens with an introduction to the common functionalities of actions, and continues with a documentation of each individual action:

- Actions regarding solids
  - Move Objects
  - Scale Objects
  - Rotate Objects
  - Shift Frames (Lackenby)
  - Inflate/Deflate Frames
  - Increase/Decrease Parallel Section
- Actions regarding polycurves
  - New Planar Polycurve by Intersection
  - New Polycurve by Projection
  - Move polycurve
  - Remove Polycurve
  - Properties of polycurves
  - Systemize polycurve names
  - Join polycurves
  - Split polycurve
  - Connect Points
  - Generate Fillet Points
  - Show Indicative Intersections
- Actions regarding curves
  - Change the shape of a curve
  - Curve Properties
  - Change the shape of the SAC

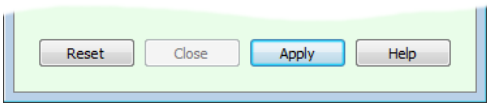
  - Bulk Change of All Curves
- Actions regarding shell surfaces
  - Phantom Faces
  - Define Shell Region
  - Remove Shell Region
  - Seams and Butts
- Actions for working with wireframes
  - See section 6.3.7.5 on page 120, Actions for wireframe manipulation.

6.3.5.1 Common functionality

When you start an action from the menu bar, an action-specific panel comes up, easily identified with its distinct background colour. Many actions require a selection of one or more items to act upon, and the panel will tell you to make a selection when needed. Selections can be made graphically or from the tree view (discussed on page 65). You can also make a selection first and then start the action, which will skip the display of instructions.

When the prerequisites are met, the action enters the configuration stage. In the configuration stage you are able to adjust properties and make changes, but none of these are final. The action will show you interactively how the model will change, but the original, unaltered model will shine through. This way it is easy to see the impact of a change before and after.

Every action has the following four buttons at the bottom of the action panel:



The [Help] button, also operated with <F1>, brings up the help reader at the section that documents the current action. If the configuration stage of the action has been changed, you can either [Reset] the changes (key <Esc>) or [Apply] them (key <Enter>). When reset, the action reverses to its initial state and the model remains untouched. When applied, the model is actually modified. Either way, the action panel stays open, ready for a new change of the same kind. The action panel can be closed with the corresponding button (also key <Esc>) or by starting another action.

Apart from this manual and the context-sensitive help mentioned above, most buttons and options show a tool-tip with a short explanation when the mouse pointer is held still over it for a second or two. These tips may be all you need to refresh your memory, and the manual can stay on the shelf most of the time.

6.3.5.1.1 Undo and redo

An added advantage of this staged way of working on a model is that its evolution is subdivided into well-defined units of change, perfectly suited to be recorded and played back and forth at will. That is how undo and redo work in Fairway. Single steps can be undone and redone in the conventional way of pressing <Ctrl+Z> and <Ctrl+Y> but the complete string of changes can also be inspected by selecting [Undo]→[Navigate action history]. This brings up a list of performed actions, which can also be embedded inside the main window just like the tree view, and by clicking on the items in that list you can undo and redo multiple items at once.

Attention

> Undo is an exclusive feature of the GUI. The action history will be cleared whenever you leave the GUI or use any of

- The option [Legacy Interface] of Properties of polycurves (discussed on page 84)
- Any changes made through the menu option [Solids]→[Object Management...]

6.3.5.2 Move Objects

All active solids and wireframes can be moved with the action started from [Solids]→[Move Objects] or with the keys <Alt><O><M>.

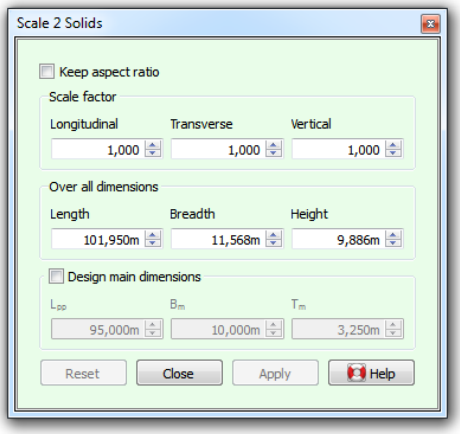
The translation can be keyed in on the action panel, or defined interactively by dragging the dragger away from the origin. Objects can be activated and deactivated at any time by toggeling their [Access] checkbox in the tree view.

Note

The names of polycurves remain unchanged, you may want to update them to reflect their new position using [Systemize polycurve names] (discussed on page 85).

6.3.5.3 Scale Objects

All active solids and wireframes can be scaled with the action started from [Solids]→[Scale Objects] or with the keys <Alt><O><C>. Objects can be activated and deactivated at any time by toggling their [Access] checkbox in the tree view.



Configuration of the scaling action.

The objects can be scaled by changing the scale factors or by changing the over all size of the active objects combined (assuming transverse symmetry). If the [Keep aspect ratio] check box is marked, then objects are scaled with equal factors in all three dimensions.

If the scaling is performed on the object that the design main dimensions apply to (see section 6.4.1 on page 129, Main dimensions (design) & hull coefficients), then you are encouraged to mark the [Design main dimensions] check box. This will cause the design main dimensions to be scaled as well, and allow you to define the scaling by changing the main dimensions.

Note

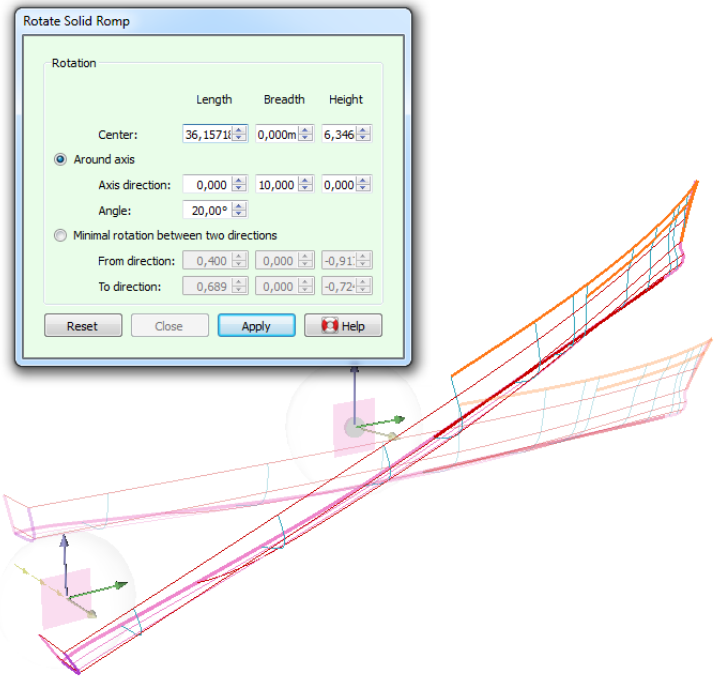
The design main dimensions do not necessarily agree with the current geometric state of the hull. For example, if your design moulded breadth is 10m, but the hull is actually broader, scaling to a moulded breadth of 12m does not produce an actual beam of 12m; it will be more than that. In this case it would be appropriate to first scale the over all breadth to 10m while leaving the [Design main dimensions] box unchecked, to obtain agreement between the model and the design moulded breadth.

The names of polycurves remain unchanged, you may want to update them to reflect their new position using [Systemize polycurve names] (discussed on page 85).

6.3.5.4 Rotate Objects

All active solids and wireframes can be rotated simultaneously with the action started from [Solids]→[Rotate Objects] or with the keys <Alt><O><R>. Objects can be activated and deactivated at any time by toggling their [Access] checkbox in the tree view.

This action displays several draggers at the same time: There is one dragger for positioning a small white sphere which defines the center of rotation. In addition, depending on how the rotation is defined, there are one or two draggers for defining directions. Depending on the zoom level these may overlap each other, zooming in will help identify them.



Two ways to define a rotation.

The axis of rotation obviously passes through the center of rotation, in a given direction. The axis direction, together with the angle of rotation, can be defined in two ways.

By default the axis direction and angle of rotation are defined explicitly, with the option [Around axis]. The angle can be manipulated interactively on the action panel (see paragraph 6.3.2.4.1 on page 68, Numerical input), the axis direction can in addition be manipulated graphically using a directional dragger (see section 6.3.4.4 on page 72, Dragging a direction vector). Note that the directional vector does not need to have its origin coincide with the center of rotation (see figure) which allows these values to be measured from the model by applying dragger snap (section 6.3.4.3 on page 71, Snapping to other points in the model).

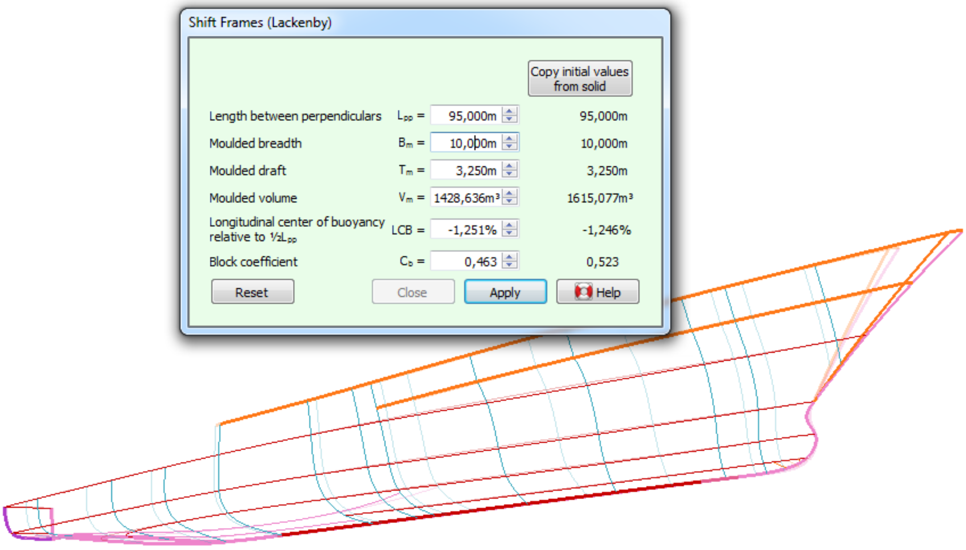
Alternatively, the option [Minimal rotation between two directions] allows the axis direction and angle of rotation to be determined implicity. Given a vector pointing in a given direction it finds the rotation that makes it point in a second given direction. The axis of rotation is always perpendicular to these two vectors. Again, you are free to drag or snap the origin of these vectors independently, which makes it easy to extract directions from between existing points in the model.

Note

> After rotation, planar polycurves are likely in a different plane and thereby of a different type. The names of polycurves remain unchanged, you may want to update them to reflect their new type using [Systemize polycurve names] (discussed on page 85).

6.3.5.5 Shift Frames (Lackenby)

This action, available from [Hydrostatics]→[Shift Frames (Lackenby)] or with the keys <Alt><H><S>, performs a Lackenby transformation on a single solid: by scaling frames in breadth and height and shifting them forward or backward, the solid is transformed to meet specified main particulars. This transformation is also available from the Fairway main menu, see section 6.5.3.2 on page 131, Shift frames (Lackenby). Please also confer section 6.5.4 on page 131, Hints for and backgrounds of the transformation process.



Transforming a hull by shifting its frames.

In the right-most column of the action panel the design main dimensions are listed as configered in [Main dimensions (design) & hull coefficients] (discussed on page 129) and other particulars derived from the initial shape of the frames. Left of it is the column of editable target values; a change in any of these causes the solid to be transformed accordingly. For convenience, the values from the right-most column can be copied into the fields of target values by pressing the button [Copy initial values from solid].

Note

> If exact target values are not available, it is practical to "scroll" towards a viable solution using the mouse wheel (discussed on page 68) and visual feedback.

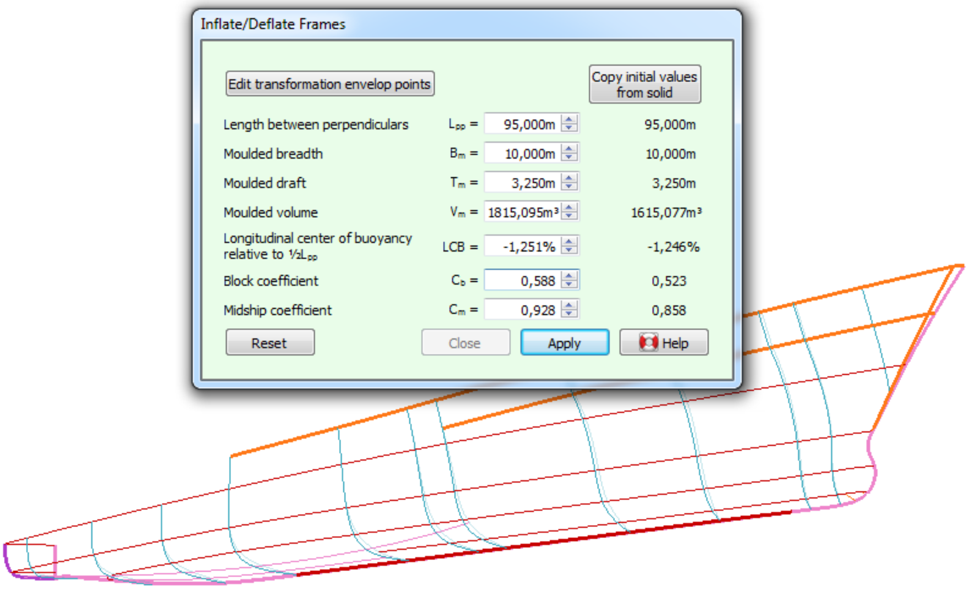
Attention

> Because this transformation shifts frames to new positions, you may want to apply [Systemize polycurve names] (discussed on page 85) to update the names of frames according to their new position.

If the project is configured to make use of a target sectional area curve or SAC (see section 6.6.1.1 on page 133, General Fairway settings) then this SAC is transformed as well, and some additional options become available in the action panel, as discussed in paragraph 6.3.5.6.1 on the next page, When designing with a target SAC.

6.3.5.6 Inflate/Deflate Frames

This action, available from [Hydrostatics]→[Inflate/Deflate Frames] or with the keys <Alt><H><I>, performs a transformation on a single solid by moving points on frames inward or outward to satisfy specified main particulars. This transformation is also available from the Fairway main menu, see section 6.5.3.3 on page 131, Inflate/deflate frames. Please also confer section 6.5.4 on page 131, Hints for and backgrounds of the transformation process.



Transforming a hull by shifting its frames.

The columns of target values and initial values are identical to the ones in [Shift Frames (Lackenby)], with the addition of the midschip coefficient C_{m}.

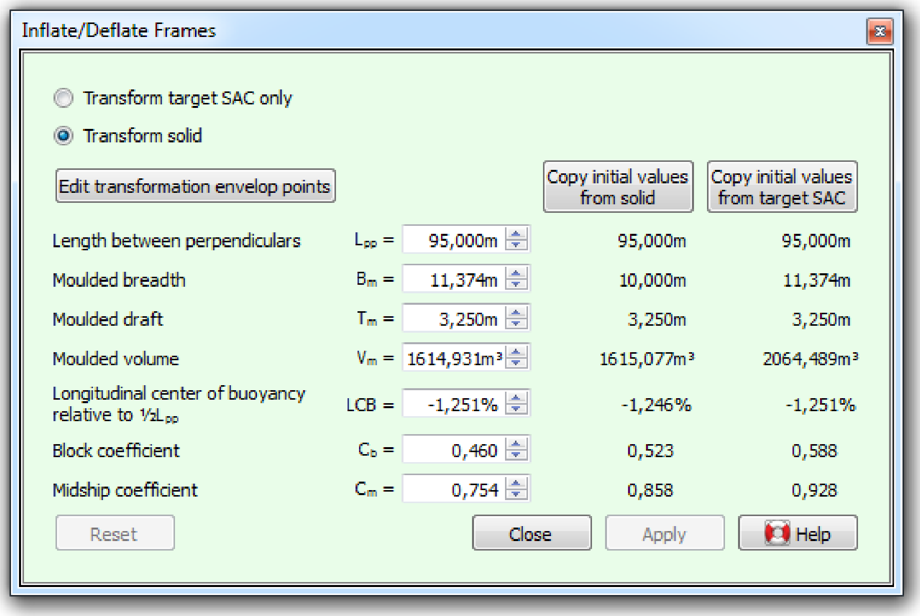
Note

> If exact target values are not available, it is practical to "scroll" towards a viable solution using the mouse wheel (discussed on page 68) and visual feedback.

The inflation of frames can be constrained to not protrude a given set of longitudinal planes, the so-called transformation envelop. The planes pass through any defined envelop points successively, which can be edited by pressing the [Edit transformation envelop points] button. Transformation envelop points are discussed further in section 6.5.2 on page 130, Specify envelop lines midship section.

6.3.5.6.1 When designing with a target SAC

If the project is configured to make use of a target sectional area curve or SAC (see section 6.6.1.1 on page 133, General Fairway settings) then this SAC is transformed as well, and some additional options become available in the action panel (this applies to [Inflate/Deflate Frames] as well as [Shift Frames (Lackenby)]):



Additional options regarding a target sectional area curve.

When [Transform solid] is checked (the default) then the target SAC is transformed as well as the solid. When [Transform target SAC only] is checked, the transformation is only applied to the target sectional area curve, the solid itself remains untouched. Its use is discussed below in paragraph 6.3.5.6.1.1 on the current page, Transformation of the target SAC only; the shape of the SAC can be inspected using [Hydrostatics]→[Sectional Area Curve (SAC) Window].

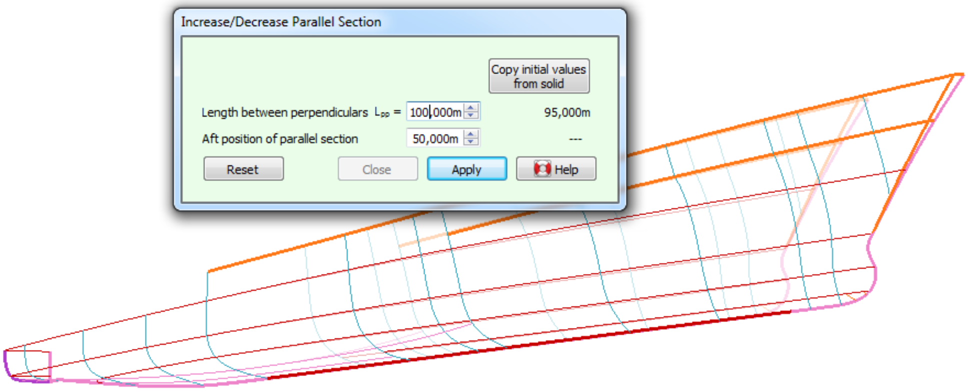
In addition, an additional column shows the main particulars according to the initial state of the target SAC, with an option to copy these to the target values for the transformation.

6.3.5.6.1.1 Transformation of the target SAC only

With the option [Transform target SAC only] or the button [Fit] (discussed on page 98) in the action [Change the shape of the SAC] the target SAC can be transformed separately. Why should one do so? With the total hullform transformation, with a single command an entire ship hull is transformed, which goes pretty quick. The disadvantage is, however, that this transformation is acting globally, so no control of local details is possible. Fortunately, Fairway offers also tools to adapt frame shapes to the desired area, as given by the target SAC. This process is more laborious, but offers much more local control. More details can be found in paragraph 6.3.5.19.8.1 on page 96, Show Target Frame Area.

6.3.5.7 Increase/Decrease Parallel Section

This action, available from [Solids]→[Increase/Decrease Parallel Section] or the keys <Alt><O><P>, allows to quickly change the length of the parallel mid-section of a solid. It works by moving all points forward of the given [Aft position of parallel section] to satisfy the given [Length between perpendiculars]. This transformation is also available from the Fairway main menu, see section 6.5.3.4 on page 131, Increase/decrease parallel midbody.



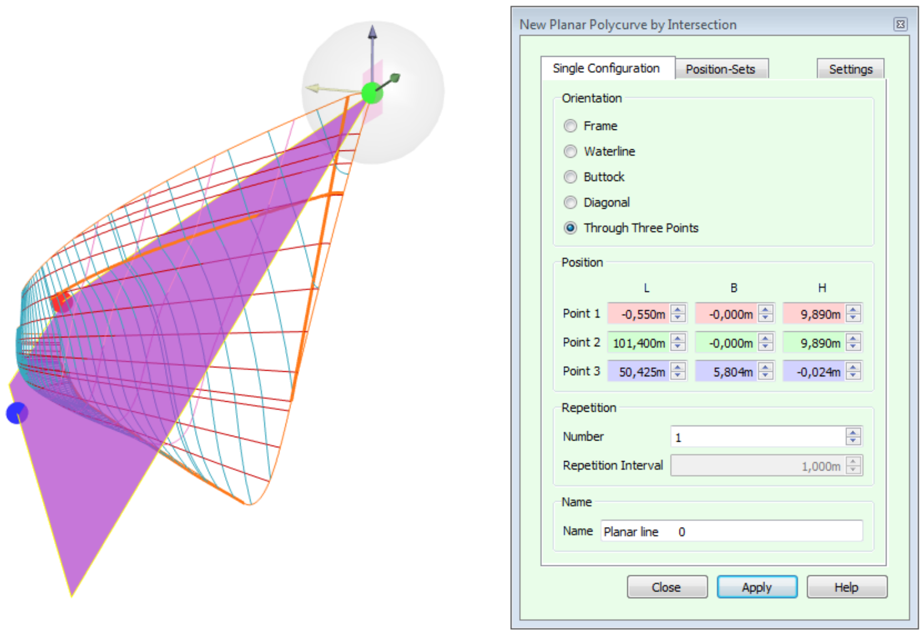
Lengthening a hull by adding a parallel section.

Attention

Because this transformation shifts frames to new positions, you may want to apply [Systemize polycurve names] (discussed on page 85) to update the names of frames according to their new position.

6.3.5.8 New Planar Polycurve by Intersection

This action enables the user to quickly add planar polycurves to active solids by intersecting them with a plane, or a set of parallel planes. It is started from [Polycurves]→[New Planar Polycurve by Intersection] or with the keys <Alt><P><N>. These can then be used to manipulate the hull shape in higher detail and be exported to construction software. Polycurves also add visual detail, but if that is your only objective then you way want to see them just temporarily, which may be easiest with [Show Indicative Intersections] (discussed on page 88). Another way of adding polycurves is offered by the action [Connect Points] (discussed on page 86).



Adding a new polycurve in a plane through three points.

Planar polycurves can be added in two ways: either in manually configured positions and orientations or in stored configurations, the so-called position-sets. Each of these have their own tab in the action panel.

6.3.5.8.1 Single configuration

The [Single Configuration] is pretty straightforward, consisting of a choice of [Orientation] and accompanying [Position]. Most orientations are determined by just a single value, which in the case of frames may be given in the form of a construction frame number plus an optional offset in millimeters, if construction frames have been defined and are enabled (see section 6.4.2 on page 129, Frame spacings). Planar polycurves in arbitrary orientations are defined in a plane through three points. The points are colour-coded in graphics and their corresponding input fields. The points can be moved by means of the dragger and by keying in values, and the plane through them is clipped to the maximum dimensions of the model. The planes in other orientations also have a dragger that they can be moved with.

There are two more groups on this tab and both of them are optional: [Repetition] and [Name]. With [Repetition] one can add several polycurves in parallel planes at once, by increasing the [Number] and specifying the [Repetition Interval]. The direction of repetition can be reverted by inverting the sign of the interval.

The [Name] field shows how the polycurve will be called when it is generated. You have the opportunity to change the name here, unless the repetition number is higher than one.

6.3.5.8.2 Position sets

The [Position-Sets] tab shows a list of existing position sets (see section 6.1.3.6 on page 62, Polycurve positions sets) with the option to check and uncheck them. Preview-planes will be shown for checked sets. The button [Edit Position-Sets...] allows you to add and remove sets from the list and change the positions in them.

6.3.5.8.3 Settings

The [Settings] tab in the upper right corner of the action panel automatically expands if the mouse pointer is moved over it. This tab contains settings with which the user can configure the accuracy in which new curves will match the existing curved surface. This is a model-wide setting, and is saved with the model.

Not with curved surface
: The curved surface is completely ignored, new polycurves are faired through intersecting polycurves only. This can be appropriate in dense networks, where new polycurves do not need internal points.

Surface based on smooth tangent ribbons
: Internal points will be generated on a high quality surface. Appropriate in sparse networks.

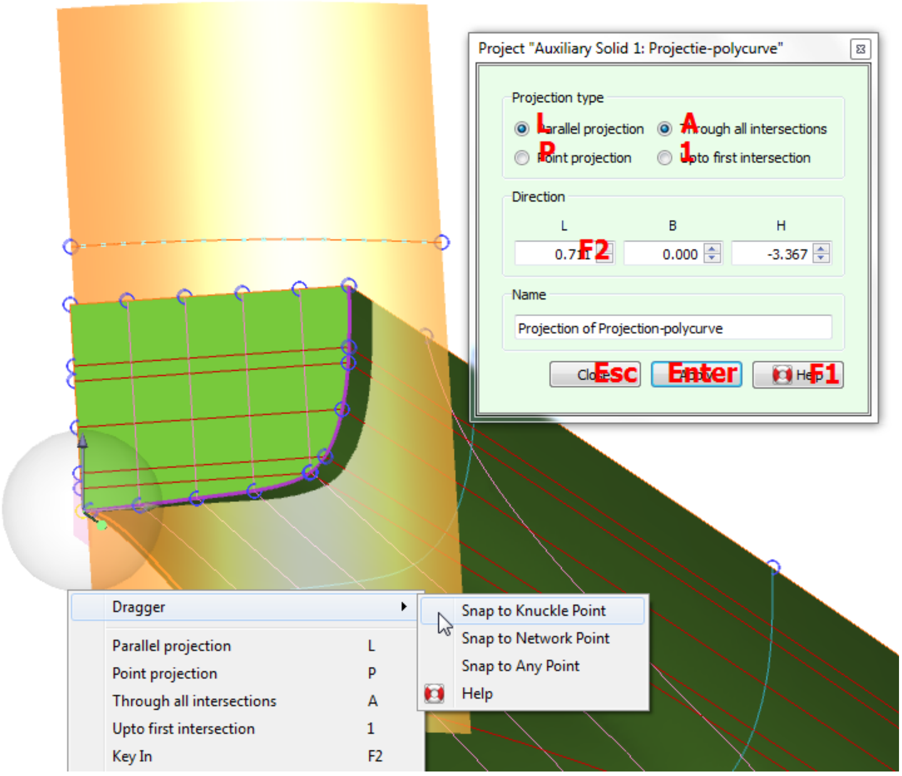
Surface based on linear tangent ribbons
: Internal points will be generated, based on surfaces with reduced smoothness requirements. Appropriate in moderately dense networks, where smooth tangent ribbons could oscillate and cause unwanted inflections.

Approximate distance between points on curved surface
: This is the target distance between internal points. A smaller distance will make the curves match the surface at a higher accuracy. However, the aim is often not to require more points than necessary but still enough to produce a desirable shape.

Minimum number of points per face
: By setting this to >1, internal points will be generated even if a polycurves' entry and exit points of a face are closer together than the above value.

6.3.5.9 New Polycurve by Projection

This action, started with the keys <Alt><P><E>, enables the user to project a polycurve of an active solid or wireframe onto other active solids. This can also be viewed as intersecting a solid with a (possibly) curved surface, defined by an extrusion of a polycurve in a defined direction. The projection can be parallel or a point-projection.



Polycurve projection with dragger snap.

This action does not make use of curved surfaces, the new polycurve will only have points at intersections with existing curves in the model. Therefore, it may be desirable to add planar polycurves prior to projection to achieve sufficient accuracy, see [New Planar Polycurve by Intersection] (discussed on page 80).

Note that if there is just one solid present in the model, no polycurve can be selected because projections of polycurves onto its own solid are not supported. You can however create and manipulate an auxiliary polycurve and project that.

6.3.5.9.1 Auxiliary curves

If the polycurve to be projected is not already present in another solid or wireframe, it may be desirable to create a new polycurve for the occasion, independent from any solid. An auxiliary polycurve can be created from within this action by pressing the button [Manipulate New Auxiliary Polycurve]. Polycurves created this way will appear in a wireframe with the name "Projection Polycurves". The new polycurve will have an initial shape of a straight line running diagonally through the model. Most likely this is not the desired shape and therefore the current action is interrupted and intermediate manipulation of the auxiliary curve is automatically started; see [Change the shape of a curve] (discussed on page 89). As soon as that action is closed, the projection action is continued for further configuration.

6.3.5.9.2 Parallel projection

While configuring a parallel projection of the polycurve, a translucent preview surface is shown through the selected polycurve in a given direction, intersecting the solids in the model. This surface shares the color of spatial polycurves, as the resulting projection will be one or more spatial polycurves running along the intersection between the preview surface and active solids.

The direction vector can be keyed in on the action panel directly, and is visualized by a dotted line in one of the ends of the selected polycurve. A dragger allows graphical manipulation of this vector: by dragging the end of the dotted line the direction is changed, and by dragging the start the vector is simply translated without changing its direction (see section 6.3.4.4 on page 72, Dragging a direction vector).

6.3.5.9.2.1 Aligning with existing curves

In some cases a straight line exists in the model that represents the direction in which the projection should be applied. Alignment like this is easily accomplished using the dragger snap functionality discussed in section 6.3.4.3

on page 71, Snapping to other points in the model, by translating the direction vector to the start of the straight line and selecting the end point as its direction.

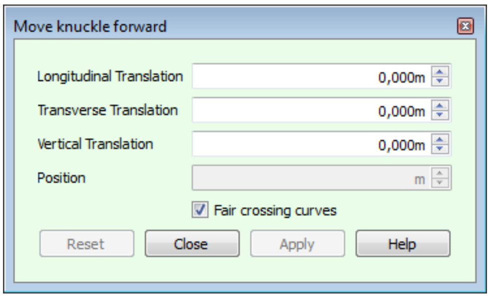
6.3.5.9.3 Point-Projection

When point-projection is selected, the preview surface converges through a single point. Again, this through-point can be keyed in manually, or translated graphically using the dragger.

Both the direction vector and the through-point are persistent across applications of the action, so that multiple polycurves can be projected successively using the same projection.

6.3.5.10 Move polycurve

Polycurves can be translated with the action started from [Polycurves]→[Move Polycurve] or with the keys <Alt><P><M>, as well as from the context menu after clicking the right mouse button on a polycurve in the tree view.



Moving polycurves

The translation is either keyed in relatively in the first three fields on the panel, or, if just a single planar polycurve is selected, to an absolute position in the fourth field. If a single frame is selected, the position can also be given in terms of a construction frame number plus an optional offset in millimeters, if construction frames have been defined and are enabled (see section 6.4.2 on page 129, Frame spacings).

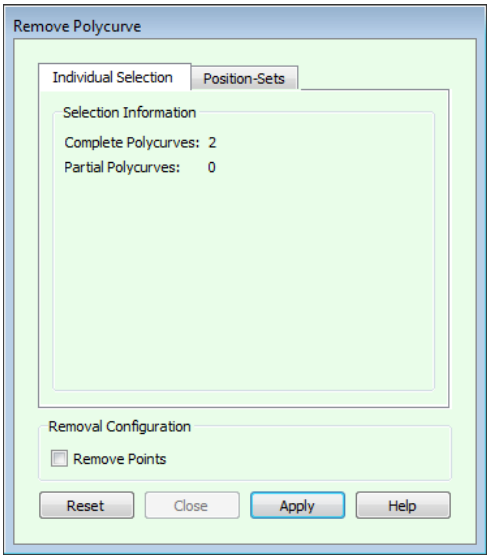
Polycurves can be selected in the usual ways, either before or after starting the action. If you need to deselect a single polycurve by holding the <Ctrl> key, be sure to click the original polycurve, not its shifted image. It is possible to move an entire object by selecting all polycurves in it, but [Move Objects] (discussed on page 74) will work quicker.

If the option [Fair crossing curves] is checked, crossing curves will be adapted to the translation. However, if a system of mutually intersecting polycurves is being moved then you may want to leave this option off.

The names of polycurves remain unchanged, you may want to update them to reflect their new position using [Systemize polycurve names] (discussed on page 85).

6.3.5.11 Remove Polycurve

This action allows the user to remove one or more polycurves from solids. It is started from [Polycurves]→[Remove Polycurve] or the keys <Alt><P><N>, as well as from the context menu after clicking the right mouse button on a polycurve in the tree view. Polycurves may be selected before or after starting the action, which are then rendered transparently to show that they will be removed when the action is applied. As usual, polycurves can be added or removed from the selection by holding the <Ctrl> key. A sequence of polycurves is easiest selected from the tree view (discussed on page 65), by holding the <Shift> key or by dragging across items. If the polycurve cannot be removed completely, because another polycurve starts or ends on it, then the remaining part will be rendered in white. The action panel shows how many polycurves will be removed completely and how many partially. A remaining part can be removed completely by adding the polycurves that end on it to the selection.



Removing polycurves

6.3.5.11.1 Remove points

If the option [Remove Points] is checked, then the points on selected polycurves will be removed from intersecting polycurves as well; knuckles excluded. If unchecked, then no points will be removed, which prevents crossing curves from changing shape.

6.3.5.11.2 Position-sets

Analogous to paragraph 6.3.5.8.2 on page 81, Position sets, the [Position-Sets] tab in the [Remove Polycurve] action panel allows all polycurves in active solids at positions in checked sets to be selected at once. Position-sets can be edited by pressing the [Edit Position-Sets...] button, as described in section 6.1.3.6 on page 62, Polycurve positions sets.

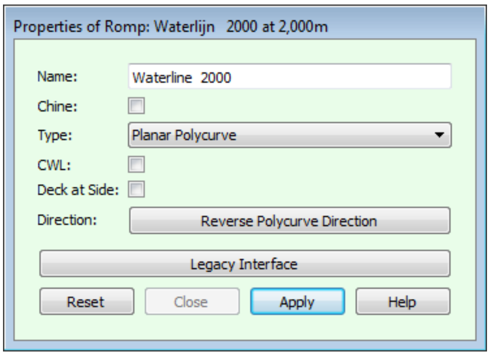
You may switch back to the [Individual Selection] tab see the selection information, and optionally add or remove polycurves to and from the selection.

Attention

> Assume frames exist at 0.0m, 30.0m and 60.0m. Next, frames are added at a position-set with frames between 0.0m and 60.0m at an interval of 0.50m. Now if this set is used to remove polycurves, then the original frames at 0.0m, 30.0m and 60.0m will be removed as well. This can be prevented by deselecting specific polycurves before applying the action, or by locking the polycurves in the Tree view (discussed on page 65) before checking the position set.

6.3.5.12 Properties of polycurves

Properties of one or more polycurves can be changed with the [Polycurves]→[Properties of Polycurves] action, which can be started with the keys <Alt><P><O> as well as from the context menu after clicking the right mouse button on a polycurve in the tree view.



Changing polycurve properties.

These are the properties that can be changed:

Name
If a single polycurve is selected, its name can be edited here. This field is unavailable when several polycurves are selected.

Chine
The chine property of polycurves can be turned on or off. Chines are often drawn with a thicker line width, and are generally used to connect knuckles in crossing polycurves. Any new polycurves that are added to the solid will get a knuckle where they cross a chine.

Type
Polycurves can be changed from planar polycurves into spatial polycurves and *vice versa*, or be explicitly constrained to one of the main orthogonal planes, turning them into frames, waterlines or buttocks. When a spatial polycurve is turned into a planar polycurve, a plane is found that best fits the spatial polycurve, in which the polycurve and its points are projected. If, for example, all points of a spatial polycurve lie in a horizontal plane, then the polycurve will turn into a waterline.

CWL
This option is only available if a single waterline, planar polycurve or spatial polycurve is selected. It will mark the polycurve that will be used in rendered output, see section 6.7 on page 135, Show (rendered and colored) surfaces, to distinguish the parts above and below the waterline. It can be used to render the water surface as well.

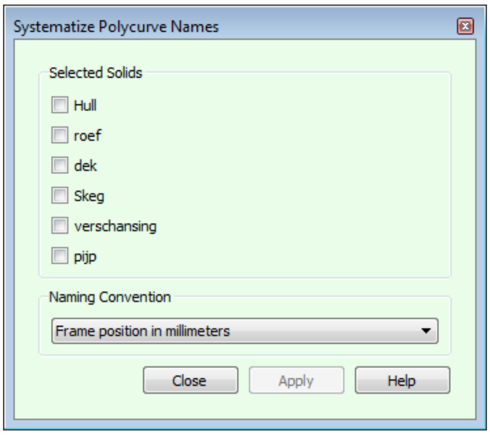
Deck at Side
This option is only applicable for longitudinals, and indicates that the polycurve is to be considered as (highest watertight) deck at side. When converting to a PIAS frame model (see section 6.8.1 on page 137, Convert this Fairway model to PIAS model) the frames will extend to this polycurve, not further. It is possible to assign this attribute to multiple polycurves, so that a deck-edge jump can be modelled; but only a complete polycurve can be deck at side, not a partial one. However, it is allowed that a frame is crossed by multiple 'deck at side' polycurves, in that case the highest is ruling. Finally, it is advised that if the deck at side mechanism is applied, each polycurve where a frame should end is assigned this attribute, even if it is the highest polycurve in the hull model already.

Reverse Polycurve Direction
This button will reverse the direction of selected polycurves, as defined in section 6.1.2.1 on page 57, Lines. You may want to do this to align the direction of a slave curve with its master.

To change the position of a polycurve, users are referred to [Move polycurve] (discussed on page 83). Complete objects can be moved with [Move Objects] (discussed on page 74).

6.3.5.13 Systemize polycurve names

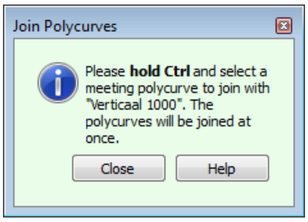
This action, started from [Polycurves]→[Systemize Polycurve Names] or the keys <Alt><P><Y>, renames all polycurves in selected solids according to a specified convention.



Systematizing polycurve names.

6.3.5.14 Join polycurves

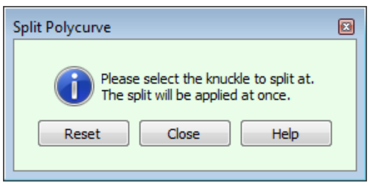
This action, started from the menu by [Polycurves]→[Join Polycurves], the keys <Alt><P><J> or from the context menu after clicking the right mouse button on a polycurve in the tree view, is very minimalistic; its panel lacks an [Apply] button and contains instructions only. The fastest way to join two polycurves that share an end point is to select both of them and start the action. The polycurves are joined instantly into a single polycurve, with a knuckle at the joint. Alternatively, you can start the action first and it will inform you what to do.



Joining polycurves.

6.3.5.15 Split polycurve

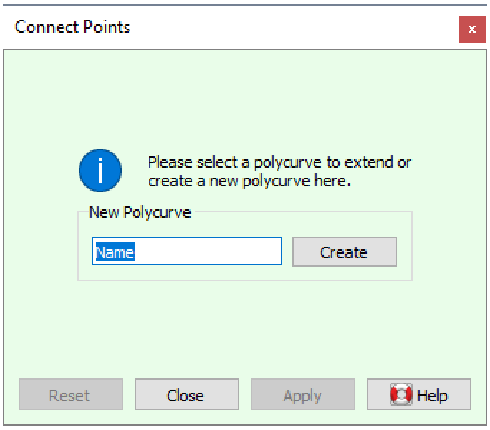
Polycurves can be split at a knuckle, by starting the action from the menu [Polycurves]→[Split Polycurve] or with the keys <Alt><P><S>, or from the context menu after clicking the right mouse button on a polycurve in the tree view. When a polycurve is selected, the user is asked to select the knuckle to split at. The polycurve is plit instantly.



Splitting a polycurve.

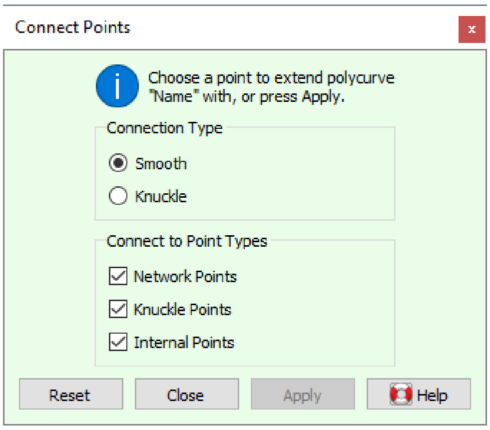
6.3.5.16 Connect Points

One way to add polycurves to a network is the use of [New Planar Polycurve by Intersection] (discussed on page 80). Alternatively, you can weave a new polycurve through existing points with the action initiated from the menu at [Polycurves]→[Connect Points] or with the keys <Alt><P><C>. This action can also be used to extend an existing polycurve; for which the action can be started from the contect menu in the tree view.



Optionally create a new polycurve first.

If you wish to create a new polycurve, then fill out a suitable name and press the [Create] button. Next you will be able to pick the start point of the new polycurve, followed by successive points that the curve is to pass through.



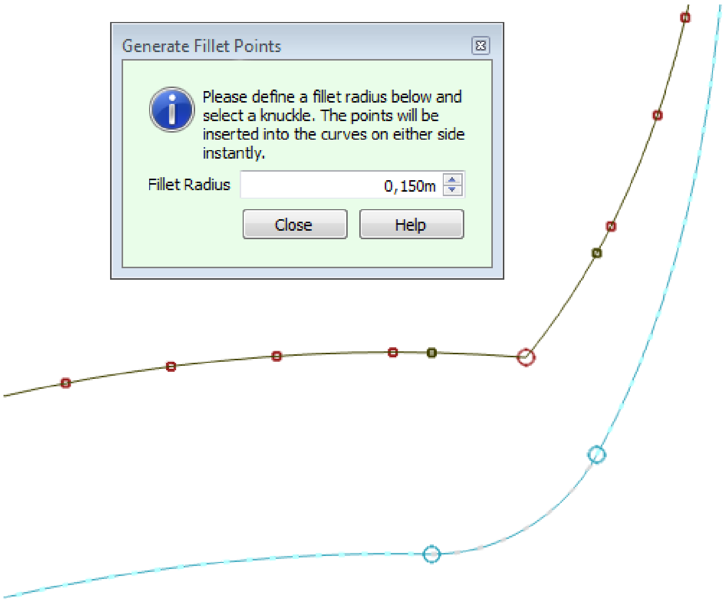
Setting the connection type and the point type filter.

The resulting curve can be chosen to run smoothly through the next point, or producing a knuckle. Points can be filtered out by type, by unchecking one or more of the checkboxes. These settings can be changed individually from connection to connection. Finish by pressing [Apply] or <Enter>.

This action is equipped with its own undo/redo functionality, so that an erroneously connected point in a sequence of connections can simply be undone and corrected without having to start over. This functionality becomes available after at least two points have been connected. For as long as this action is open, the global undo functionality is unavailable.

6.3.5.17 Generate Fillet Points

A chine can be reconstructed into a fillet by rounding the knuckles in crossing curves. For this it is necessary to find the fillet points: where a circle of a given radius touches the curves on either side of a knuckle. This action, initiated from [Polycurves]→[Generate Fillet Points] or the keys <Alt><P><F>, will find these points. Afterwards, the fillet points can be turned into knuckles and the intermediate section turned into a circular arc, using [Change the shape of a curve] (discussed on page 89).

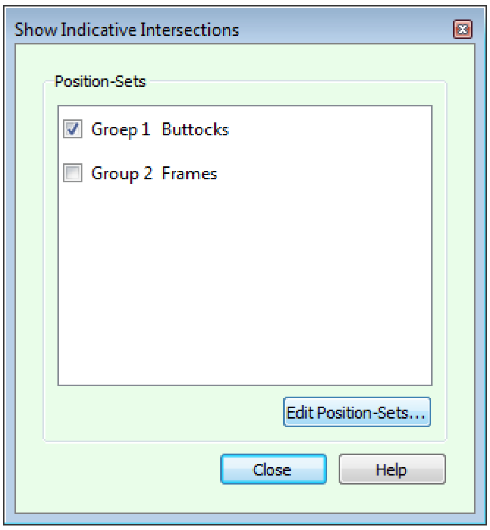


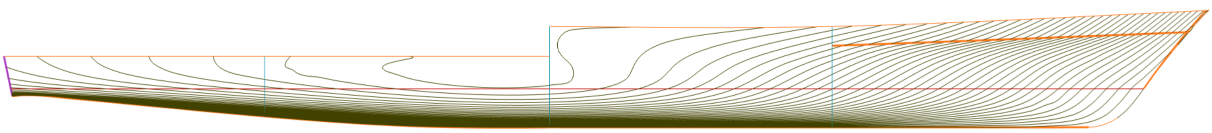
Generation of fillet points on either side of a knuckle (upper curve) and after fitting a circular arc (lower curve).

Points are inserted at the instant of a mouse click, so one can just trace the chine, there is no need to select any curves. After the fillet points have been turned into knuckles, these themselves should be connected with new chines, using [Connect Points] (discussed on page 86) and [Properties of polycurves] (discussed on page 84).

6.3.5.18 Show Indicative Intersections

This action does not change the model, but can temporarily generate intersection curves at selected position sets without adding them to the network.





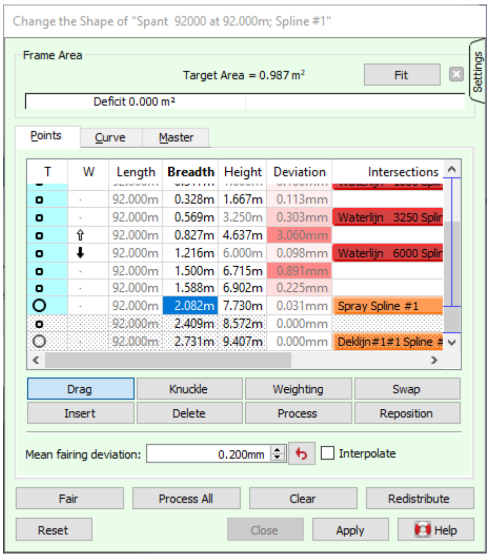
Indicative intersections.

This can be used to visually verify the definitions of position sets, but also to visualise the surface shape. The intersection curves disappear as soon as the action panel is closed. Position sets can be edited as described in section 6.1.3.6 on page 62, Polycurve positions sets.

6.3.5.19 Change the shape of a curve

This action is at the core of Fairway modelling, and packed with functionality. It is started from [Curves]→[Change the shape of the curve] (keys <Alt><C><S>), or by selecting a curve and clicking on one of its points or vertices, or from the context menu after clicking the right mouse button on a curve in the tree view.

There are three main ways to change the shape of a curve. Firstly, it is possible to change the points of the curve, and let Fairway fair the curve through the points. Secondly, it is possible to change the curve directly, by changing the vertices of the control polygon or setting the curve type or specifying boundary conditions. The third way is to define a master/slave relation between curves, in which the curve shape is derived from the shape of a master curve.



Point manipulation.

The action panel provides three tabs to work in either of these three ways: the [Points] (key <P>), [Curve] (key <C>) and [Master] tab (key <M>). The tab in the top right corner of the action panel provides optrefen{fwy_↩ action_curve_shape,fwy_action_curve_shape_settings} with which the behaviour of the action can be adjusted.

Below the tabbed section there are four general purpose buttons: [Fair], [Process All], [Clear] and [Redistribute]. These are discussed below.

6.3.5.19.1 Process All

When [Process All] is pressed (keys <Shift+P>), all the points of the curve are shifted onto their closest position on the curve, within their freedom of motion. This is an important part of the effort to maintain a consistent network, as defined in the introduction (discussed on page 57). By default, this task is automated with the option [Points Follow Curve] (discussed on page 96).

6.3.5.19.2 Fair

When [Fair] is pressed (key <F>), a new curve shape is computed. Unless the curve type in the [Curve] tab is different from [Spline] and/or a master curve is defined in the [Master] tab, the new shape passes closely through the points. Exactly how close can be seen in the [Deviation] column of the coordinate table on the [Points] tab. The accuracy of the fit depends on the [Mean fairing deviation] given underneath (the smaller that value the closer the curve) and whether points have been given a non-neutral weight factor in the [W] column. Larger deviations, as compared to the mean fairing deviation, are given a redish background in the table to allow for a quick optical quality check.

After fairing, the deviations can be eliminated by processing the points, if this isn't done automatically already.

Alternatively, the curve can be fitted exactly through the points (omitting the ones that are marked inactive in the [W] column) by checking the [Interpolate] option. This will also change the text on the [Fair] button. Care should be taken in using this option, because an interpolating curve may easily oscillate between points if one of them is slightly misplaced, and inflections can spread like ripples. The ability to allow for acceptably small deviations between points and curves is actually one of the strengths of Fairway, and helps in producing fair lines for production.

By default, fairing (or interpolation) is done instantly and automatically with the option [Curve Follows Points] (discussed on page 96) whenever that is appropriate. Nevertheless, that option does not make this button obsolete, as you will see below.

Fairing, in the naval architectural sense of smoothing up and working out unwanted inflections, can often be accomplished by alternatingly pressing [Fair] and [Process All]. Because [Fair] allows for small deviations and [Process All] eliminates them, both curve and points subtly converge to a higher fairness with each iteration. Obviously, if [Points Follow Curve] (discussed on page 96) is on, points are processed immediately, and just pressing <F> a few times will quickly improve the fairness. This can easily be visualised by the curvature plot (discussed on page 96). The rate of convergence may be increased by increasing the [Mean fairing deviation], but when done it is wise to reset that to the default value by means of the red arrow button behind the input field, as defined in [General Fairway settings] (besproken on page 133).

Because crossing curves are connected through their shared intersection points, fairing one curve may reduce the fairness of other curves. It is easy to switch to a crossing curve and repeat the process there: crossing curves are listed by the coloured buttons in the [Intersections] column of the coordinate table, and pressing one of these will implicitly apply the changes to the current curve and start manipulation of the crossing curve. By repeatedly visiting and fairing the curves in a problematic area, imperfections can be eliminated, producing a fair and consistent network of curves.

Another way to switch to another curve is by selecting a curve from the Tree view (discussed on page 65), which will also implicitly apply changes to the current curve.

6.3.5.19.3 Clear

Pressing [Clear] (keys <Shift+Delete>) removes any and all internal points from the current curve. Of course the intersection points with other curves remain.

6.3.5.19.4 Redistribute

When [Redistribute] is pressed (keys <Shift+R>), two things happen. First the internal points are removed, like [Clear], and then new internal points are inserted, based on the vertex locations of the spline control polygon. This is mainly useful after vertex manipulation, to help fixate the curve shape during future curve fairings. You may want to [Fair] the curve after redistributing points to verify that the shape is fixated properly.

6.3.5.19.5 Points

The coordinate table forms a central part of the [Points] tab. It lists all points of the entire polycurve. The subdivision into individual curves is seen in the first column [T] (for "Type") which differentiates knuckle points from ordinary points. The background of points on the current curve in that column are filled with the colour associated with the curve type. Rows for other curves are marked with a dotted background to indicate that these are not directly manipulated. If there are more points than fit in the table, a scroll bar appears on the right with horizontal marks to indicate the location of knuckles. The current curve is marked with a vertical line on the scroll bar. The cells in the coordinate table can be edited much like a spreadsheet. If cells are edited outside the current curve, changes to the current curve are applied and a new session is started on the other curve.

Cells and columns can be selected with the mouse, and the key combination <Ctrl+C> copies the selected values to the system clipboard, much like a spreadsheet does. In the same manner, values from the clipboard can be pasted into selected cells of the coordinate table with the key combination <Ctrl+V>. Only editable cells of the current polycurve segment will be overwritten.

Directly underneath the coordinate table are the mode buttons, from which there is always one depressed↩ : [Drag], [Knuckle], [Insert], [Delete], [Weighting], [Swap], [Process] and [Reposition]. The mode determines how the mouse works in the modelling view.

6.3.5.19.5.1 Drag

The [Drag] mode (key <D>) on the [Points] tab simply allows for interactive manipulation of the positions of points by means of a dragger, introduced in section 6.3.4 on page 70, The dragger: interactive graphical positioning. If [Curve Follows Points] (discussed on page 96) is on, dragging points is an effective tool for manipulating the shape of the curve.

If the position of a particular point needs to be keyed in exactly, make sure the dragger is snapped onto that point and press <F2>. This will teleport the mouse pointer over to the corresponding row in the coordinate table and open the first editable cell for editing. You may switch between cells with <Tab> and <Shift+Tab>. See also paragraph 6.3.2.4.1 on page 68, Numerical input.

6.3.5.19.5.2 Knuckle

The [Knuckle] mode (key <K>) lets you toggle the type of a point between knuckle and ordinary. This effectively splits or joins curves in the polycurve. Because the [Change the shape of a curve] action can only work on one curve at a time, any changes to the current curve are applied right before a knuckle is toggled on or off. The point nearest to the mouse pointer will light up and a message in the status bar shows what will happen to it if the left mouse button is clicked.

It is also possible to toggle knuckles in the left-most column of the coordinate table by means of a double click, <F2> or <Space>.

6.3.5.19.5.3 Insert

The [Insert] mode (key <I>) on the [Points] tab simply allows you to insert extra internal points on the curve. The position at which the point will be inserted is shown dynamically in the prelight colour, and you can preview its exact coordinates on a temporary row in the coordinate table. Click to insert a point.

6.3.5.19.5.4 Delete

The [Delete] mode (key <Delete>) on the [Points] tab is for deleting internal points on the curve. If a certain point cannot be deleted, the reason why is displayed on the status bar.

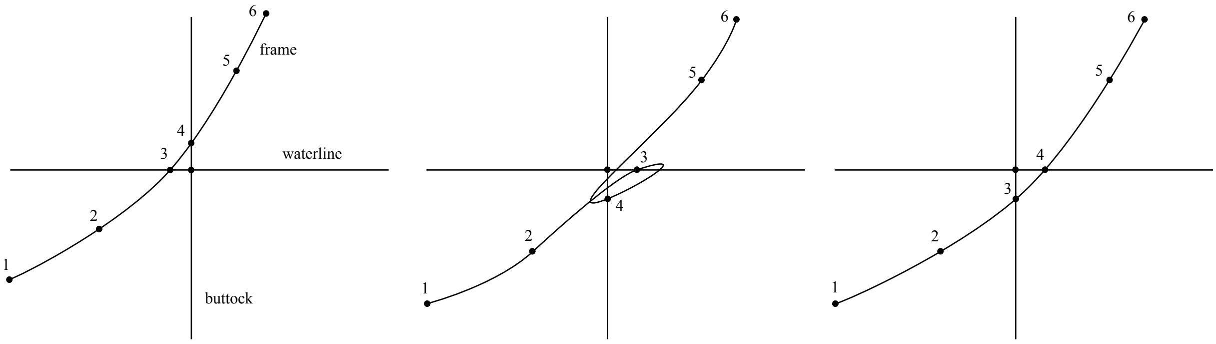
6.3.5.19.5.5 Weighting

In the [Weighting] mode (key <W>), the relative importance of points can be changed between neutral, inactive and heavy. Instructions on how to do that appear in the status bar. An inactive point is marked with an outlined arrow pointing upwards, a point with a heavy weight is marked with a filled arrow pointing downwards.

As with knuckles, it is also possible to change weights in the [W] column of the coordinate table by means of a double click, <F2> or <Space>.

6.3.5.19.5.6 Swap

In the figure below a situation is sketched in which it may be necessary to swap two points on the curve.



Use-case for swapping two points. Left: Initial situation. Middle: Frame shifted. Right: Swapped points.

On the left a frame is shown with points numbered in sequence. Points 3 and 4 mark the intersection with a waterline and buttock respectively. Together the three curves form the sides of a triangular face. Then a manipulation is performed (middle figure) that causes the frame to pass the intersection between waterline and buttock on the other side. Geometrically the triangle has been flipped (look at its corners) but the connections remain unchanged, so in effect the triangle is oriented inside-out: Points 3 and 4 remain on their respective curves, so now the points are ordered out of sequence, producing a kink in the curves. By swapping points 3 and 4 the order can be restored and the kinks untangled (right figure).

The [Swap] mode is designed to swap the corners of a triangle, flipping it topologically. The following requirements and limitations apply:

1. Only points that form the side of a trianglar face can be swapped, as shown in the figure.
2. The curves must pass smoothly through the corners of the triangle, they cannot end on a corner or have a knuckle at a corner.
3. No other curves can be connected to the corners of the triangle.

Any pair of points nearest to the mouse pointer that satisfy this requirement will light up, and be swapped upon a click of the mouse.

Sequencing problems around faces with a higher number of sides or of a higher complexity than the above cannot easily be repaired. Often the fastest solution is to delete the problematic polycurve entirely and reinsert it. Another solution is to split it up, remove the kink and reconstruct the missing part by connecting points. See section 6.3.5.15 on page 86, Split polycurve, section 6.3.5.11 on page 83, Remove Polycurve and section 6.3.5.16 on page 86, Connect Points.

6.3.5.19.5.7 Process

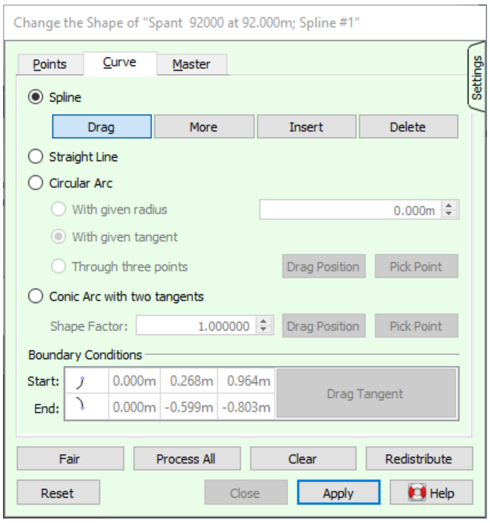
In the [Process] mode, individual points can be shifted onto the curve, within their freedom of motion.

6.3.5.19.5.8 Reposition

In the [Reposition] mode, individual points may be shifted along the curve, if their freedom of motion allows this. If not, an explanation will appear in the status bar. Care should be taken not to shift points past neighbouring points, which would bring them out of sequence, causing a kink in the curve when it is faired.

6.3.5.19.6 Curve

The [Curve] tab is divided into two sections. In the upper section, the curve type can be specified, which also enables controls that are relevant to the type. In the lower section, boundary conditions can be set and manipulated to the start and end points of the spline.



Curve manipulation.

6.3.5.19.6.1 Spline

As explained in section 6.1.2.1 on page 57, Lines, a spline is a freely malleable curve, both by vertex manipulation and by the fairing algorithm. All other curve types produce a fixed shape. The default colour associated with splines is light blue. Splines may have specified boundary conditions at both ends. There are four modes for vertex manipulation.

6.3.5.19.6.2 Drag

In the [Drag] mode on the [Curve] tab vertices can be moved by means of a dragger. It is often easier to design good curves with a low number of vertices, therefore you may want to [Delete] vertices first.

6.3.5.19.6.3 More

In the [More] mode on the [Curve] tab, more vertices may be added locally to give more shape control. Existing vertices will be shifted to preserve the current shape of the curve. Highlighted vertices will be shown to indicate where vertices will be positioned if the mouse button is pressed.

6.3.5.19.6.4 Insert

In the [Insert] mode on the [Curve] tab, additional vertices may be inserted into the control polygon. This will change the shape of the curve. It is possible to quickly “sketch” a shape with a string of vertices by giving a sequence of clicks in the graphical view.

6.3.5.19.6.5 Delete

The [Delete] mode on the [Curve] tab simply allows vertices to be deleted from the control polygon.

6.3.5.19.6.6 Straight Line

Straight lines remain straight and are unaffected by fairing operations. Curves in flat sections of the shell can advantageously be defined as straight lines. When a curve is turned into a straight line, any existing boundary conditions are removed. The colour associated with straight lines is green by default.

6.3.5.19.6.7 Circular Arc with given radius

The default colour for circular arcs is light grey. Arcs with a given radius can only exist in polycurves that are defined in a plane, not in spatial polycurves. The arc can be flipped to the other side by inverting the sign of the radius.

6.3.5.19.6.8 Circular Arc with given tangent

Arcs with a given tangent have always one tangent defined. The tangent can be changed in the lower section of the [Curve] tab. You are free to define a tangent at the other end, which will remove the existing boundary condition.

6.3.5.19.6.9 Circular Arc through three points

This type enables two modes. The [Drag Position] mode displays a dragger with which an arbitrary position can be set that the arc will pass through. Using the [Pick Point] mode the arc can be made to pass through an existing point on the curve.

6.3.5.19.6.10 Conic arc with two tangents

Straight lines and circles are conic sections as well, but this type allows the remaining conic sections to be defined: parabolic, hyperbolic and elliptical arcs. These are defined by a two-edged control polygon, the middle vertex of which is given by the intersection of the two tangents. Consequently, the tangents should be coplanar; if they are not, the curve will find a plane in the middle, but not adhere to the given tangents strictly.

The type of conic section depends on the [Shape Factor]: a higher shape factor will pull the curve tighter to the middle vertex, a factor of 0 will give a straight line. A parabolic is produced with a factor 1, higher factors produce a hyperbolic and lower factors an elliptical arc.

The shape factor can also be determined automatically by specifying a third point that the arc should pass through, analogous to [Circular Arc through three points].

6.3.5.19.6.11 Boundary Conditions

In the lower section of the [Curve] tab there is a table listing the boundary conditions at the start and end of the current curve (for an explanation of curve direction see section 6.1.2.1 on page 57, Lines). In the left column an icon indicates the current condition at each end, as follows:

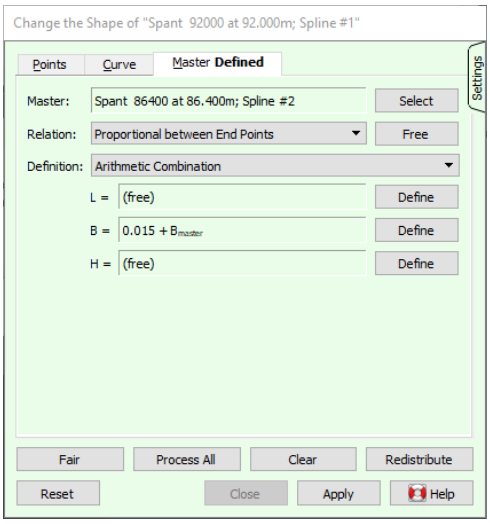
- No boundary conditions
- Manually specified tangent
- Tangent derived from adjacent curve
- Manually specified tangent, straight end
- Tangent derived from adjacent curve, straight end
- Tangent and curvature derived from adjacent curve

The condition can be changed with a double click on the icon, which brings up a pull-down menu with available conditions. Not all conditions may be available, for instance if there is no adjacent curve at that end. If so, the reason will be given in a tool-tip when the mouse pointer is kept still for a few seconds.

Next to the icons are the coordinates of the current tangent vector. These can be edited in the table or manipulated interactively in the [Drag Tangent] mode, activated with the corresponding button in the table.

6.3.5.19.7 Master

On the [Master] tab a master/slave relation can be defined, in which the shape of the current curve, the slave, can be made dependent on another curve in the model, the master curve. Whenever the master curve changes, the slave curve will change also, according to the defined dependency. This is exemplified at the end of this section with the construction of deck camber and sheer strake. A cascade of depedencies can exist, as master curves themselves can be a slave of other master curves, and several curves can depend on the same master curve.



Configuration of shape dependency

The first step in establishing a master/slave relation is to select the master curve, using the [Select] button. You are free to select a master from any active solid; solids can be toggled active/inactive in the [Access] column of the [tree view] before starting the selection.

An existing relation can be annulled with the [Free] button.

Next steps are to select the proper relation and the definition of the dependency.

6.3.5.19.7.1 Relation

The relation defines how points on the current curve are mapped to points on the master curve and *vice versa*. Choose between these four relations:

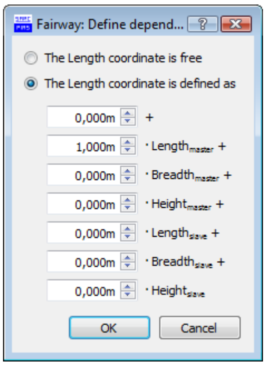
Proportional between End Points
: The start and end points of the current curve will be related to the start and end points of the master curve. Inbetween these, points will be related proportionally to the lengths of the curves. It may be necessary to reverse the direction of one of the polycurves using [Properties of polycurves] (discussed on page 84).

Longitudinal/Transverse/Vertical Position
: Points on the current curve will relate to points on the master curve that are on the same longitudinal, transverse or vertical position respectively. You will need to decide which direction is appropriate for the situation; mostly this will be the direction in which lines can be drawn that cross the master and slave curves at angles closest to 90°.

6.3.5.19.7.2 Definition

The last step in the configuration of a master/slave relation is how points on the current curve are defined, based on points on the master curve.

Arithmetic Combination
: The length, breadth and height coordinates on the current curve can be defined individually as a linear combination of a constant value, the coordinates on the master curve and the coordinates on the unaltered slave curve.



Linear combination of coordinates.

Offset

The current curve will be shaped similary to the master curve, offset from the master curve in a direction and distance that may vary over the length of the curve. The direction is perpendicular to the projection of the master curve onto a certain projection plane. This plane can be visualized by pressing the [Drag Plane] button, which will also reveal a rotational dragger with which the orientation of the plane can be manipulated interactively. You may also key in the components of the vector manually, and the plane can be made to fit the master curve using [Master Plane], or the offset can be defined in the current view-plane of the active camera by pressing [View-Plane].

The offset distance can be specified for the start and end point individually, which will produce a linear transition over the length of the curve. These can also be dragged interactively using the [Drag Offset] mode, which will show a linear dragger snapped to the curve end nearest to the mouse pointer.

6.3.5.19.7.3 Example: Deck camber

Let's say you want to construct a deck with constant camber of 2/100 of the local breadth of the deck. To do that we start manipulating the deck profile line at the plane of symmetry. Then we select the deck line in the side as the master curve, whose transverse coordinates mark half the local breadth. We will set the relation to [Longitudinal Position], because both curves generally run longitudinaly. We want breadth to be unaffected (free) as well as length, but the height should equal the height of the deck in the side plus 4/100 of its breadth. So we define an arithmetic combination of $H = 0.04 \cdot B_{\text{master}} + H_{\text{master}}$.

6.3.5.19.7.4 Example: Sheer strake

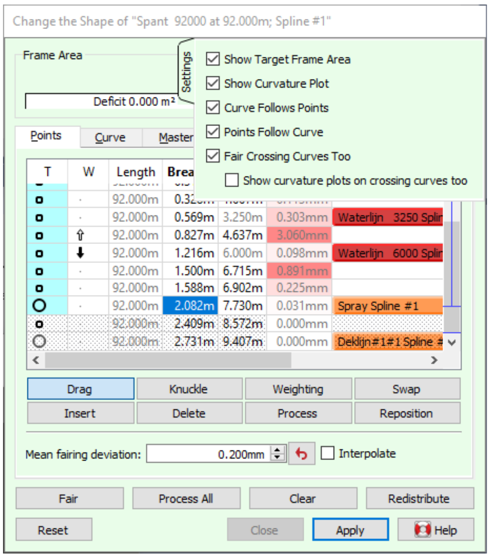
If you need to construct a sheer strake then this is one way to do it: Start manipulating the curve that marks the lower seam of the sheer strake and select the upper seam as master curve. The relation can be set to 'Proportional between End Points' or 'Longitudinal Position'; which is the better option depends on the hull shape and you may try both to select the appropriate one.

Now set the definition to 'Offset', and define an offset plane parallel to the center plane, with normal (0.0, 1.0, 0.0). Finally the height of the sheer strake can be specified at the front and end of the seam.

This produces a sheer strake of constant width as seen from the side.

6.3.5.19.8 Settings

The [Settings] tab in the upper right corner of the action panel automatically expands if the mouse pointer is moved over it. This tab contains some options with which the behaviour of the action can be adjusted.



Action settings and frame area

6.3.5.19.8.1 Show Target Frame Area

If a target sectional area curve (target SAC) has been constructed, see section 6.3.5.21 on the next page, Change the shape of the SAC, you may want to compare the current frame area (below the construction waterline) with the corresponding point on the target SAC. The option [Show Target Frame Area] toggles an additional [Frame Area] section at the top of the action panel, as shown above; provided the current curve is part of a frame. It can be removed at any time with the cross button in the top right.

Shown is the target area according to the SAC, and a bar graph indicating the surplus or deficit area of the current frame, as compared to the target. The [Fit] button will transform the current frame so that its area matches the target SAC, while its shape will be preserved as much as possible. Further details are found in section 6.5 on page 129, Hullform transformation. If no target area is available, then probably no target SAC was generated, or it does not extend to the current frame position.

Together with the [Frame Area] section in the action panel, appears in the modelling views a transparent rectangular plane. The area of this plane equals the target frame area, while the height is equal to the design depth. This may help optically in designing the frame shape, as the area under the frame within the rectangle needs to match the area from the frame up to the construction waterline outside the rectangle.

6.3.5.19.8.2 Show Curvature Plot

With the [Show Curvature Plot] option on the [Settings] tab the curvature plot can be switched on during curve manipulation, even if curvature plots have been switched off in general; see paragraph 6.3.2.3.2 on page 67, Prelit polycurves. The plot is also shown for any curves in the polycurve that are directly preceding or succeeding the curve under manipulation, as these may change shape when their common knuckle point is dragged or when they follow a boundary condition with the current curve.

For reasons of consistency this option is automatically switched on when the general curvature plot is on or is being switched on. Conversely, the general plot is automatically switched off when this option is being unchecked.

6.3.5.19.8.3 Curve Follows Points

If the option [Curve Follows Points] is checked, then the curve is instantly and automatically faired upon any change in the information on which the fairing is based. Most notably, when this is on, the curve can be interactively shaped by dragging points and tangents. In many cases this can be an effective alternative to shaping the spline through its vertices, and has the added bonus that the shape remains fixed during future curve fairings. For vertex manipulation, on the contrary, generally requires the shape to be explicitly fixated (discussed on page 90), which still is no guarantee against subtle changes during future curve fairings.

6.3.5.19.8.4 Points Follow Curve

If the option [Points Follow Curve] is checked, then all points will be processed automatically (shifted onto the curve) whenever the shape of the curve changes. Together with [Fair Crossing Curves Too] this is a great help in keeping the network consistent at all times.

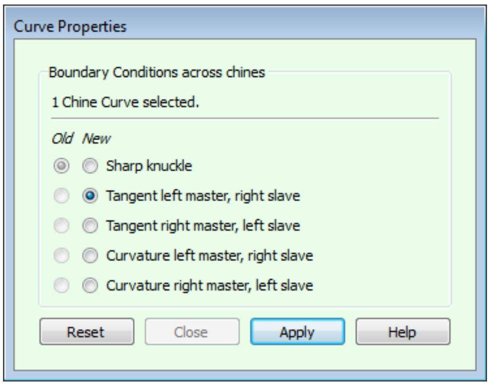
If you switch this off, you will most probaly want to [Process] (discussed on page 92) individual points or [Process All] (discussed on page 89) before you apply the shape changes.

6.3.5.19.8.5 Fair Crossing Curves Too

Check this if you like all curves that pass through the points of the current curve to adapt dynamically to any changes in their position. This is a great help in keeping the network consistent.

6.3.5.20 Curve Properties

Most properties of curves relate to their shape, which is the domain of [Change the shape of a curve] (discussed on page 89). Other properties can be adjusted with this action, found in the menu at [Curves]→[Curve Properties]. This action can also be started with the keystrokes <Alt><C><P>, as well as from the context menu after clicking the right mouse button on a curve in the tree view. Currently, this only concerns chines, which cause crossing polycurves that are added to the model successively to get a knuckle at that point. Chines can be defined with [Properties of polycurves] (discussed on page 84) and are displayed with a greater line thickness by default. With the options here it is possible to specify boundary conditions between the curves on either side of the knuckles, when polycurves are added.



Setting curve properties.

Several chines can be selected, and their current properties are indicated by the radio buttons in the *Old* column. They can collectively be given a new property value in the *New* column. Here the notions “left” and “right” are used as defined in section 6.1.2.1 on page 57, Lines, and the term “left master, right slave” denotes that the curve to the right of the chine has a boundary condition dependency on the curve to the left of the chine. Curvature continuity implies tangential continuity, see also paragraph 6.3.5.19.6.11 on page 93, Boundary Conditions.

6.3.5.21 Change the shape of the SAC

Attention

> This functionality is turned off by default. To enable it, configure the project for design using a target SAC in [File]→[Preferences...]→[Fairway Project Settings...], see section 6.6.1.1 on page 133, General Fairway settings.

If main dimensions have been set, including target values for block coefficient (C_b) and longitudinal center of buoyancy (LCB), see section 6.4.1 on page 129, Main dimensions (design) & hull coefficients, Fairway enables you to design towards these targets by means of the target sectional area curve (target SAC). This action is for the construction and manipulation of a target SAC, and is started from [Hydrostatics]→[Generate/Modify Target SAC], keys <Alt><C><A> or a click on one of the points on the target SAC. The SAC can be viewed in a dedicated modelling view by selecting [Hydrostatics]→[Sectional Area Curve (SAC) Window], which also shows the hull lines as seen from below and from the side, for reference, shown in the figure below. The view also shows the actual SAC in gray, which cannot be edited and represents all buoyant solids and forms the basis for hydrostatics information; it can be updated from the hydrostatics window (see section 6.3.6.3 on page 112, Hydrostatic Data). The SAC view is automatically brought forward when this action is started.

Once a target SAC has been created, it can be used to compare the submerged frame area during frame manipulation (see paragraph 6.3.5.19.8.1 on the preceding page, Show Target Frame Area). The target SAC, if present, is also used to base automated hullform transformation on, see [Shift Frames (Lackenby)] (discussed on page 76) and [Inflate/Deflate Frames] (discussed on page 77).

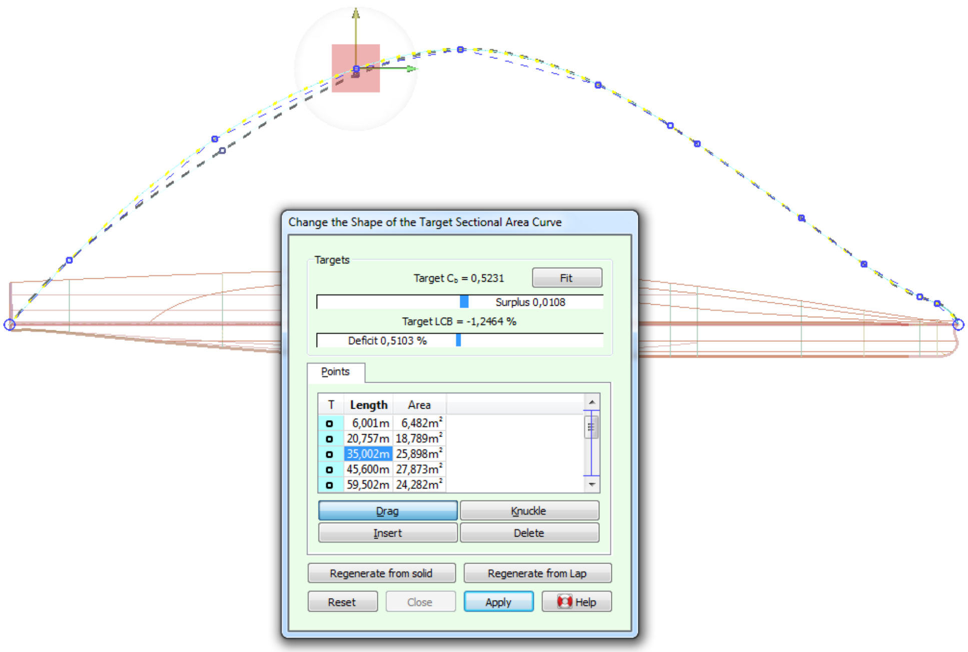
Note

> The target SAC only applies to a *single buoyant solid*. Therefore, if more than one solid is marked as buoyant at startup, the user is asked to identify one particular solid and others are made non-buoyant automatically. Buoyancy can be managed by hand in Object management (discussed on page 155).

If a target SAC is not present yet when the action is started, the user is asked to create one by pressing one of two buttons:

- [Create target SAC from current solid] derives the SAC from the current shape of existing frames in the buoyant solid.
- [Create target SAC from Lap diagrams] generates the SAC based on Lap's diagrams, which are discussed in some more detail in paragraph 6.3.5.21.2 on this page, The use of Lap diagrams. When selecting this option you have to specify additionally whether the vessel is single or twin propeller, because different diagrams exist for these respective types.

The above functionality is also available during manipulation of an existing target SAC, with the buttons [Regenerate from solid] and [Regenerate from Lap] seen in the figure below.



Manipulating the target sectional area curve.

The target SAC is represented by a polycurve fitted through a number of given area values, and can thus be manipulated by changing these values; much in the same way as points can be changed on ordinary polycurves. There are four manipulation modes: [Drag], [Knuckle], [Insert] and [Delete]; their operation is completely analogous to the point manipulation modes (discussed on page 90).

6.3.5.21.1 Fit

During manipulation of the target SAC, two gauge bars in the action panel show how well the SAC matches the target values as specified in [Main dimensions (design) & hull coefficients] (discussed on page 129). The [Fit] button here will transform the target SAC so it matches the target values, using the same algorithm as in [When designing with a target SAC] (discussed on page 78).

6.3.5.21.2 The use of Lap diagrams

When commencing a ship design from scratch, with a target displacement or LCB, an initial SAC might be a proper tool to speed up this process. Unfortunately, not many methods are available to generate a SAC, although

Lap (NSP, Wageningen, The Netherlands) did publish one. Those are the so-called Lap-diagrams, applicable on bulbless vessels with a conventional cruiser stern (so, no pram-type of aftship). So, at the ship extremes (for and aft) a conflict might arise between the sectional area according to these diagrams, and the desired stem or stern profile. Such a conflict may be solved iteratively, *for instance* in the following fashion:

1. Start with the SAC generated from Lap diagrams.
2. Design the hullform, making certain that in the ‘mid region’ (that is, not near the stem and stern) the frame areas correspond to this SAC, see paragraph 6.3.5.19.8.1 on page 96, Show Target Frame Area.
3. Generate the target SAC from this hullform.
4. This SAC will now have incorrect values of C_b and LCB. So it must be transformed towards the correct parameter values.
5. Adapt the hullform to this SAC.
6. When necessary, repeat the process from step two.

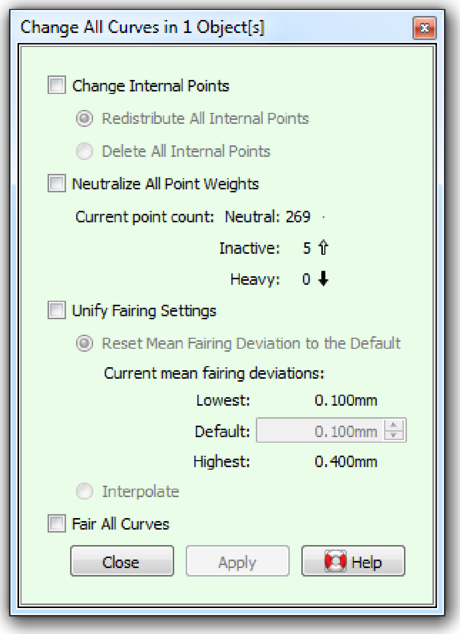
Note

> Lap diagrams offer a facility to ease the design process, which may prove to be useful for the ship designer. However, its use is certainly not obligatory, and it is not the core of Fairway at all.

The Lap diagrams are numerically contained in a separate file, `kvslap.txt`, which is modifyable so a user can use alternatives, if available. section 6.A.3 on page 160, File format of diagrams for generation of a sectional area curve contains more information on the file format.

6.3.5.22 Bulk Change of All Curves

This action, started from [Objects]→[Bulk Change of All Curves] (keys <Alt><O><B>), allows select manipulations that can be done on individual curves using Change the shape of a curve to be done in bulk on all curves in active solids and wireframes simultaneously.



Bulk curve manipulation.

Four different kinds of changes can be made. The first option makes changes to internal points. This can be practical when curves are defined with densely spaced points, which can occur in some imported wireframes. There are two sub-options that either redistribute points based on the number of spline control points, similar to [Redistribute] (discussed on page 90), or delete altogether, similar to [Clear] (discussed on page 90). Note that all unconnected points in a wireframe are considered internal, so before deleting internal points you should consider connecting curves using [Wireframe connections] (discussed on page 122).

The second option is available when points exist with non-neutral weights, and neutralizes these. This is only applicable to solids. See section 6.1.1 on page 56, Basics of Fairway on the influence of point weights.

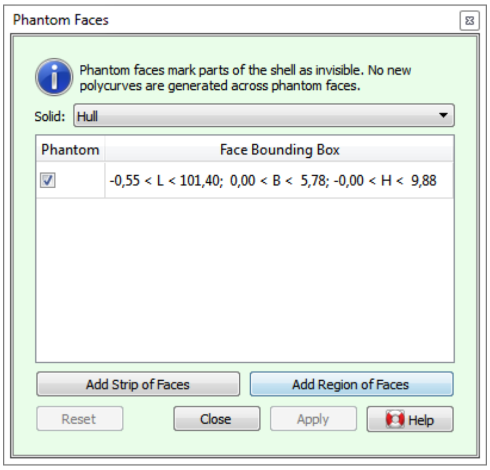
The third option allows the mean fairing deviation to be changed on all curves. Alternatively, all curves can be made to interpolate points exactly. Consider using the latter on an imported wireframe where curves are known to pass through points exactly, to be able to connect curves with a low tolerance. The input field for the default mean fairing deviation is identical in function to [General Fairway settings] (discussed on page 133), so any future curves will get an initial mean fairing deviation with this value.

Finally, the last option fairs all curves anew taking the preceding options into account. This option is selected by default. Prior to fairing, this option performs two additional services that should normally go unnoticed: first it rounds the position of planar polycurves to within 0.01mm, then it makes sure that points lie in constraining planes.

6.3.5.23 Phantom Faces

The shell of a solid is internally defined as a closed surface. Parts of that surface can be hidden from the eye and omitted during the creation of new polycurves. These parts are called phantom faces. Most hulls that are modeled on just one side of the center plane have a phantom face to hide the *imaginary* part of the surface that connects the keel line to the deck line inside the hull. See also section 6.1.3.1 on page 62, Phantom face.

Phantom faces can be defined and removed by starting the action from the menu [Shell]→[Edit Phantom Faces], or using the shortcut <Alt><S><P>.



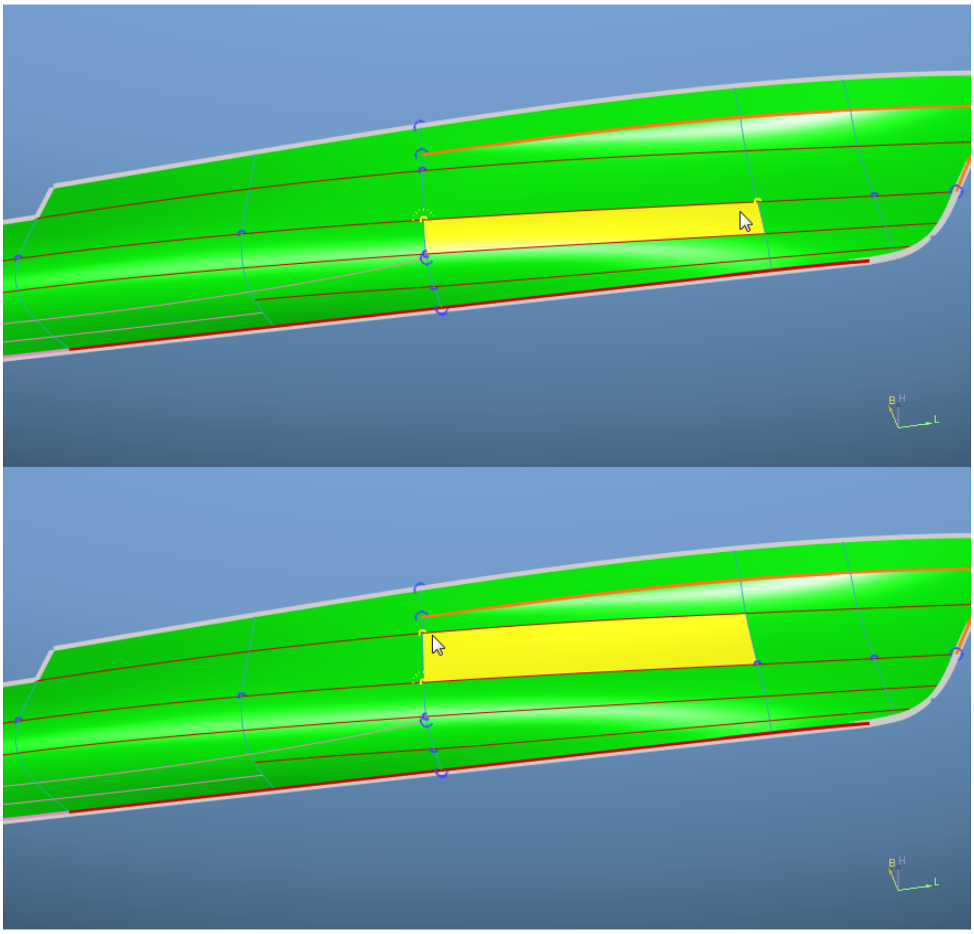
Editing phantom faces.

If there are multiple active solids in the model, then the solid of which the phantom faces are to be edited can be selected from the [Solid] pulldown menu at the top of the action panel.

Existing phantom faces of the selected solid are shown in the table underneath, and marked graphically with thick borders in the modeling area. A phantom face can be removed (turned into an ordinary part of the shell) by unchecking its checkbox in the table. The second column of the table contains the coordinates of the bounding boxes of phantom faces, to differentiate them in case there is more than one. Selecting a row in the table will highlight the borders graphically, so the right face can easily be identified.

There are two means of adding a new phantom face. A single face can easiest be added by pressing the [Add Strip of Faces] button. This will make the network points of the solid selectable. After selecting a corner of the face in question, only network points on polycurves running through the selected corner become selectable. Hovering over these will prelight faces to the left or the right of the polycurve, allowing you to select the second corner of the same face. This identifies the face unambiguously, and it is marked as a new phantom face. Obviously, this mode can also be used to mark a strip of contiguous faces in one go.

The rule that determines on which side of the polycurve faces are highlighted is as follows: "walking on the outside of the shell along the polycurve from corner to corner in successive order selects the faces to your right." But there is no real need to memorize this because of the visual feedback.



Visual feedback during face selection. The first (selected) point has emphasis, the mouse pointer hovers above the second point. (Thick grey curves mark the contour of the phantom face through the center plane.)

Secondly, faces can be marked in bulk by defining a region of faces, using the button [Add Region of Faces]. This lets you mark the corner points bordering the region in clockwise direction, which should be closed eventually by returning to the start point. For an illustration of this process see the sequence of figures in paragraph 6.3.5.24.2 on the following page, Definition of a region contour.

Pressing [Apply] will make the specified changes. Note that the action is implicitly reset when another solid is selected from the list, or whenever a solid changes its state of activity in the tree view.

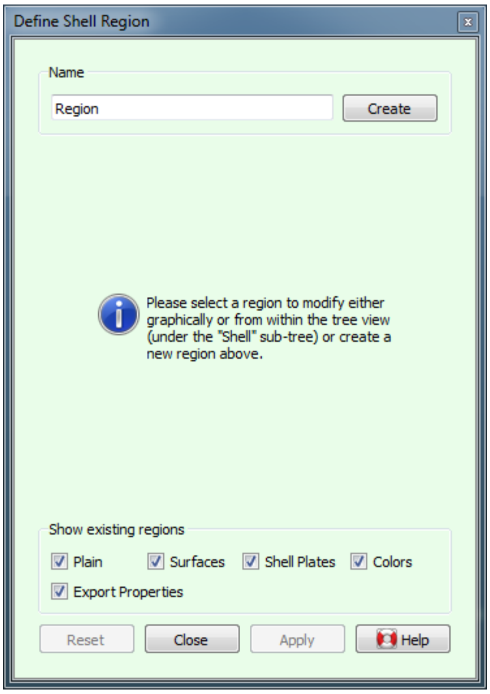
6.3.5.24 Define Shell Region

The shell of a solid can be partitioned in regions for various purposes. Shell regions are bounded by a sequence of polycurves or sections thereof that form a closed border. The action for defining and modifying regions is started from [Shell]→[Define a Shell Region] or the keys <Alt><S><D>. Regions can be defined for the following purposes:

1. Naming a specific area on the shell.
2. Setting surface shape properties in a region such as developability or so-called slave surfaces.
3. Defining shell plates.
4. Setting a deviating color and transparency for visualization purposes.
5. Defining specific export properties, where values should deviate from the default.

Most of these purposes can be combined in a single definition, like a shell plate that is also developable, but because plates have a distinct color in order to differentiate them from adjacent plates, plate regions and colored regions are mutually exclusive. Also, export options cannot be combined with shell plates or surface regions — but this is no limitation because in general regions are allowed to overlap. However, overlapping plates are usually undesired and regions with a surface shape property must not overlap.

All regions carry a name.



Initial state of the Define Shell Region action panel.

6.3.5.24.1 Visibility of existing regions

As long as the action is active, existing regions can be visualized by their borders (in bold light gray) and color if applicable. Because regions can serve various purposes and it can be distracting to see regions with a purpose that is not of interest, it is possible to switch their visibility on and off based on purpose with the checkboxes at the bottom of the action panel.

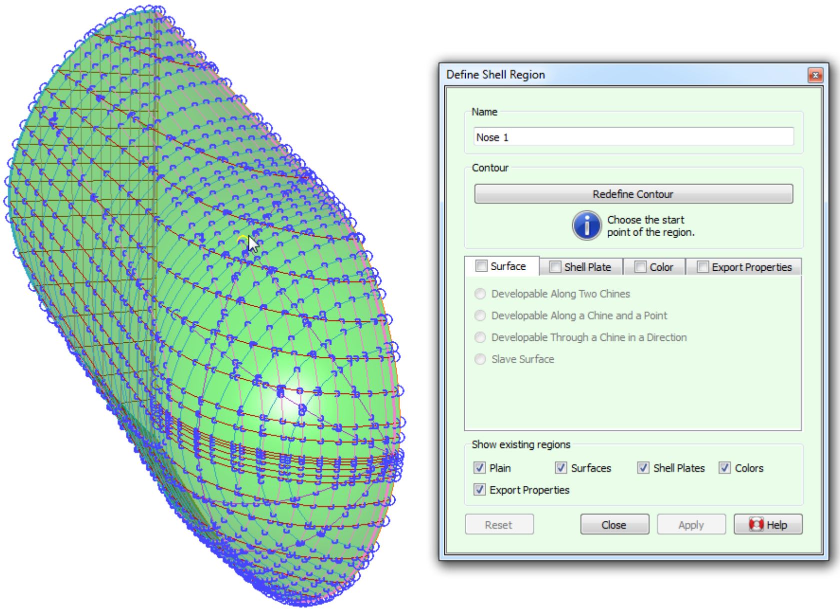
1. The [Plain] checkbox toggles visibility of regions that neither define a surface, plate nor specific color. Regions of this type may be defined to assign properties at a later point, or just to name a particular area of interest.
2. The [Surfaces] checkbox toggles the visibility of developable surfaces and slave surfaces. However, the describing lines of developable surfaces (lines that remain straight before and after bending of a plate) are always visible, even when the action is closed.
3. The [Shell Plates] checkbox toggles the visibility of defined shell plates. When visible, the ordinary shell is made semi-transparent to make it easier to identify the parts of the shell where no plates are defined yet, while existing plates are shown with their border and individual color. Also, any defined seams and butts are shown hashed, to make it easier to define plates that are to override the pattern of seams and butts.
4. The [Colors] checkbox toggles the visibility of borders of regions that solely define a deviating color and/or transparency. Naturally, the region itself will be colored whatever the state of the checkbox.
5. The [Export Properties] checkbox toggles the visibility of regions with specific export properties.

If a region serves more than one purpose, it will be visible as long as any of the corresponding checkboxes is checked.

6.3.5.24.2 Definition of a region contour

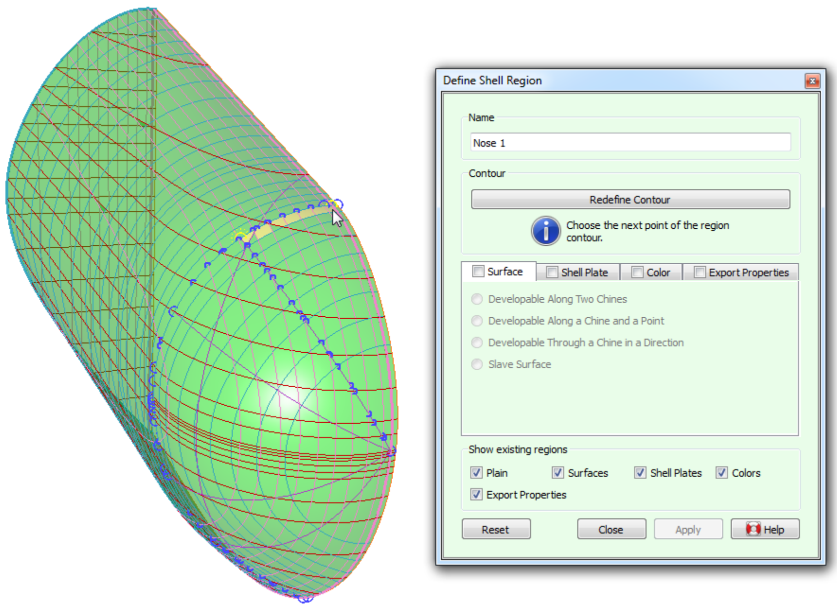
Regions are defined by a closed contour along (parts of) polycurves that border the region. This contour is defined interactively by clicking in succession on corners of the region in a clockwise direction when looking from outside at the shell.

The process of defining the contour is started directly after the [Create] button is pressed, and can be restarted at any time by pressing the [Redefine Contour] button. The first step is to define the fist corner of the contour, and for this any of the network points in active solids can be clicked.



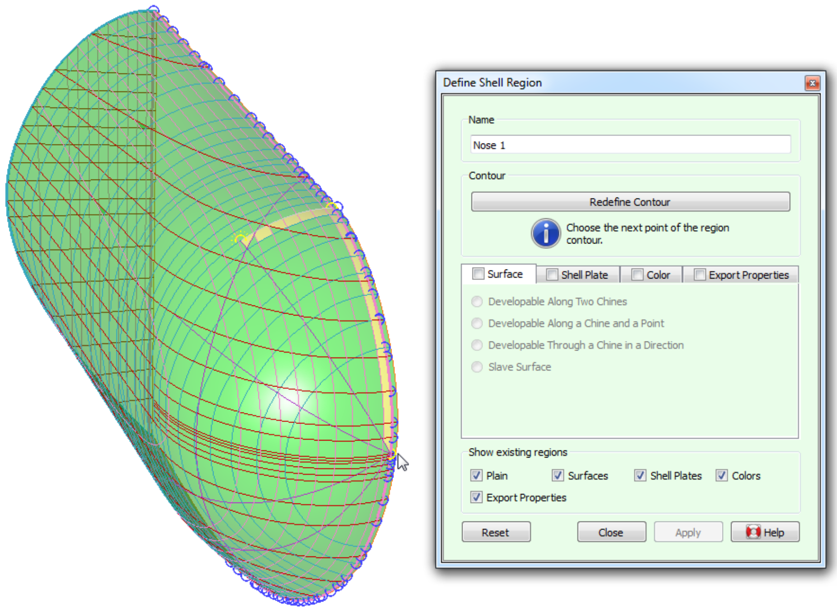
Selecting the first corner of the region contour.

Upon the first click, the start point is marked as such with a halo around it. From this moment on, the only network points that can be selected are the ones that can be reached over the polycurves that cross through the latest added corner. When moving the mouse pointer to the next corner, faces to the right of the path are highlighted, as a visual feedback for the inside/outside of the contour; these faces will form the periphery of the region.



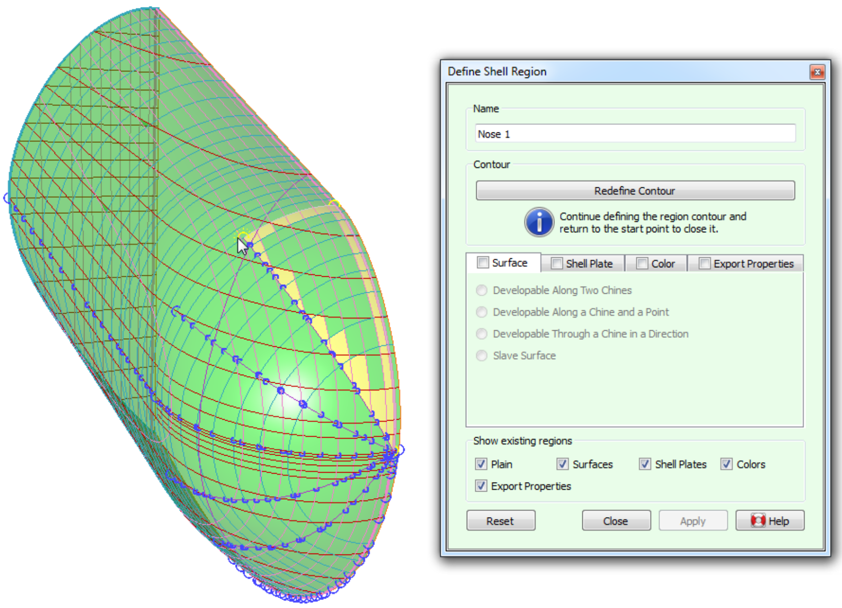
Selecting the second corner of the region contour.

This process is repeated over all intermediate corners. Contours will most commonly be convex, but non-convex contours are supported as well. When points are packed tightly on screen then looking at the last highlighted face will help to determine whether the corner is correctly identified. If at any time a mistake is made and the wrong point is clicked, the process can be restarted by pressing the [Redefine Contour] button.



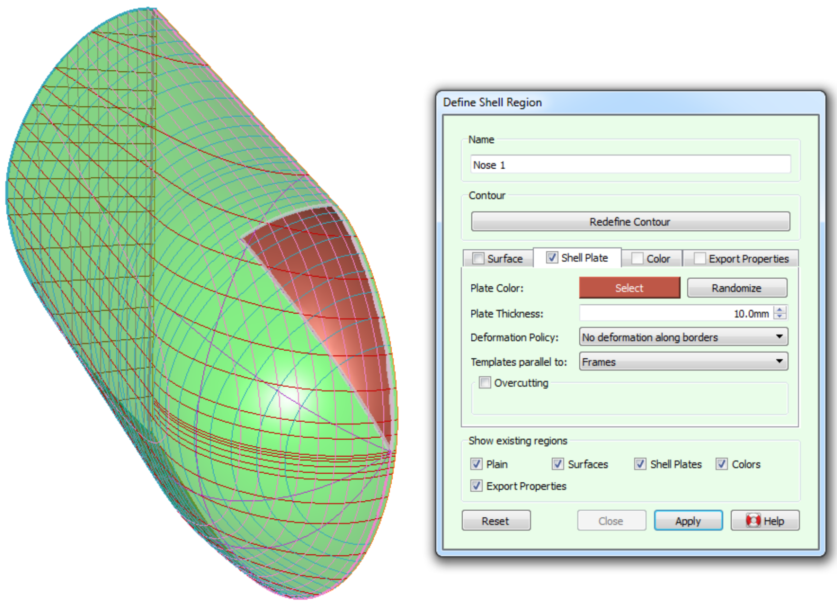
Selecting intermediate corners of the region contour.

The process is completed by selecting the start point as the last corner, which produces a closed contour.



The contour is closed by selecting the start point.

A finished contour is displayed in bold gray curves. At this point the region can be given properties by checking one or more checkboxes in the property tabs.



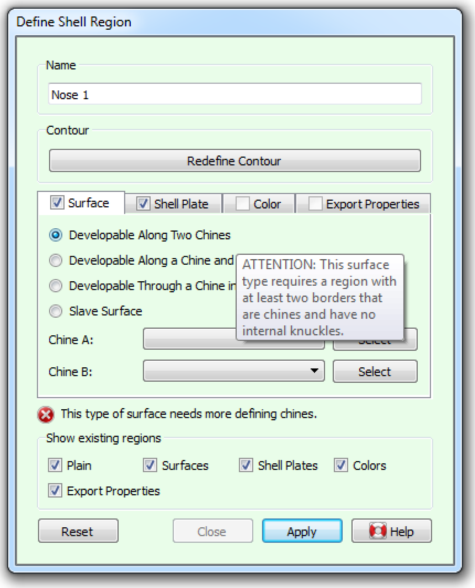
Defining a shell plate.

6.3.5.24.3 Surface-shaping regions

A region can impose certain shape features onto the curves that it covers, most notably surface developability. Unlinke doubly curved surfaces, developable surfaces (discussed on page 60) have local curvature in one direction only, which allows plates to be formed without stretching or shrinking. The [Surface] tab supports the following types of shape-imposing properties:

1. [Developable Along Two Chines] is for the construction of a developable surface with a moving top.
2. [Developable Along a Chine and a Top] produces a conic surface with fixed top coordinates.
3. [Developable Through a Chine in a Direction] places the top at infinity, producing a cylindrical surface. See section 6.3.4.4 on page 72, Dragging a direction vector for graphical manipulation of the direction.
4. A [Slave Surface] copies shape characteristics such as curve type, end-conditions, radius etc. of some master curve to all curves in parallel planes within the region.

Depending on the type of developable surface, its construction requires one or two defining borders, which must be chines and have no internal knuckles. Fairway analyzes the borders and presents the ones that meet these requirements in a pull-down from which they can be selected. Often there is just a single valid option and it will automatically be picked without the need for user interaction. The defining border will be drawn in the developable color (green by default). A defining border can also be selected graphically after pressing the corresponding [Select] button. If the border under consideration does not meet some of the requirements then it will be prelit in the prohibited color (red by default) with a message in the status bar explaining its deficiencies.



Surface definition with clearly indicated deficiencies.

If a developable surface can be constructed, straight rulings will be shown (finely dotted, in green by default) indicating the direction in which the surface has zero curvature. These will remain visible after the action is closed. However, it is not always possible to construct a developable surface for a given configuration, and regions in general can be invalidated as well. When this occurs, a warning message appears in the action panel, describing the problem. Nevertheless, the action will allow any changes to be applied, so that deficiencies may be resolved later. Region validity is discussed in paragraph 6.3.5.24.8 on page 108, Region Validity.

The region will impose its shape upon affected curves after [Apply] is pressed, as well as whenever a defining border or master curve is changed.

6.3.5.24.4 Definition of a shell plate

Fairway can generate shell plate expansions and information for the production of shell plating, see section 6.10 on page 150, Shell plate expansions and templates. A region can easily be marked as a plate by checking the [Shell Plate] tab. (If the checkbox is disabled then the region has probably already a [Color] property. Uncheck the [Color] property, or define a second version of the region.)

The [Plate Color] serves only visualization purposes, to differentiate adjacent plates. If the current plate color does not contrast enough with an existing adjacent plate then a different color can quickly be generated with a click on the [Randomize] button, or selected manually by pressing the colored [Select] button.

The [Plate Thickness] and [Deformation Policy] are relevant for the expansion of doubly curved plates.

1. [No deformation along borders] ensures that adjacent plates always fit during assembly. Within these constraints the expansion is optimized for minimal deformation.
2. [Only stretch; no shrinkage] optimizes for minimal stretch and prevents that the plate needs to be shrinked.
3. [Minimal deformation] minimizes the amount that the plate needs to be deformed, without additional constraints.

In case the production of shell plates requires templates (see section 6.10.4 on page 153, Production of templates) the orientation of the templates can be specified using the [Templates parallel to] pull-down.

If the plate needs to be cut somewhat bigger or smaller than the plate contour, then check the [Overcutting] checkbox. This reveals a table in which the overcutting can be specified for each bordering curve individually. A positive value will cause the plate to overlap the adjacent plate (in support of joggling, for example) and a negative value produces an undercutting (to obtain a root opening for welding).

Plate expansions and templates can be generated from [Shell plate expansions and templates] in the main menu (discussed on page 64).

6.3.5.24.5 Painting the shell

The [Color] property serves purely optical and presentation purposes. It allows to show regions of the shell with a specific color and transparency, deviating from the overall shell material settings (see paragraph 6.3.2.2.2 on page 66, Shell). This is the only region that remains visible (without its borders that is) outside region-oriented actions.

Examples for usage of this property include using a different color for the underwaterbody, painting the sheer stripe, fitting transparent windows and windshields, and painting the chimney in company colors.

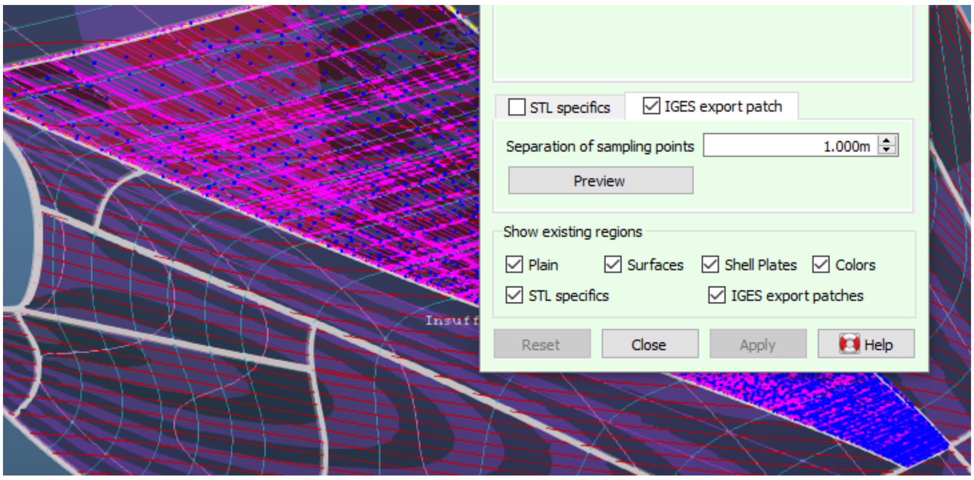
6.3.5.24.6 STL export specifics

The [STL Specifics] tab offers two options that are relevant when the hull shape is exported in STL format (see section 6.8.8 on page 141, Stereolithography file (.STL) for CFD or 3D printing), in particular when the STL file is intended for CFD analysis. If this tab is unchecked, the global values defined for the export apply to the underlaying shell surface. If the tab is checked, it becomes possible to omit the surface covered by this region from the export, or to specify a [Specific desired triangle size] that will make the underlying surface to be triangularized in the STL export with a triangle size that is different from the global values defined for the export.

6.3.5.24.7 IGES export patch

The [IGES export patch] tab offers the posibility to fit a single NURBS surface onto a quadrilateral region, if the checkbox in the tab is ticked and the region satisfies a number of prerequisites. All of these regions can then be exported to IGES format using [IGES NURBS regions (2018)] (discussed on page 139). Connecting borders of incident patches are mathematically equivalent, so that a watertight patchwork can be constructed across the entire shell. The following conditions must be met:

1. The region must have exactly four corners.
2. There may be no knuckles within a border (as these will count as additional corners).
3. The surface within the region should not vary too wildly, which would confuse the fitting algorithm. More formally:
   (a) it should be possible to reach any point within the region from any other point within the region by walking in the direction of its shortest spatial distance, and
   (b) the normals through any two points within the region should not cross before they would intersect a simple bilinearly blended Coons patch through the region borders.



The IGES export patch tab and generated NURBS surface in the background.

The fitting of the NURBS surface is based on sampling points of the shell surface, the separation distance of which is specified in the input field of the action panel shown above. As a minimum, samples are always taken at every curve intersection within the region as well as in the middle of every face within the region. Pressing the [Preview] button will start the fitting process and show the sampling points in blue, the NURBS control network in magenta (or however the vertex color is configured) and the NURBS patch itself in red. If any sampling points show up in yellow then that indicates that there is a problem with these, which can happen if the third condition above is not met. These points are ignored during fitting and thus a satisfactory result is still possible, but proceed with caution and preferably check the consistency of the curve network and try an alternative region contour.

The time it takes for the fitting algorithm to complete, grows roughly quadratically with decreasing sampling point separation. This means that, for example, fitting with a separation of 0.01m can take 100 times longer than fitting with a separation of 0.1m. Therefore, it is recommended to start out with a large separation, like a tenth or fifth of the longest side of the region which will give a minimal sampling count as explained above, then assess the result and decrease the separation only when necessary. The largest separation that still produces a good fit depends of course on the internal shape of the region: a completely planar surface produces a perfect fit with a minimum of sampling points. The [Preview] button needs to be pressed again to refresh the preview with the new separation distance.

Along the borders of the region, the generated NURBS patch will match the underlaying polycurves exactly, except for a very small affine transformation that cancels out any deviation between the intersecting polycurves at the corners of the patch. This ensures a water tight connection between neighbouring patches. But the internal shape of the patch is an approximation of the sampling points. To obtain high surface quality it is of course important that the curves within the region are well faired and the network has no large deviations (see [Check Solid] (discussed on page 111)) and the finely rendered surface (see [Display]→[Rendering...]) does not show unintended ripples and bulges. But even when all these prerequisites are fulfilled, it can happen that the fitting algorithm produces a patch with erratic undulations over smaller or larger areas of the patch. This is likely caused by an unfortunate distribution of sampling points, which can be resolved by changing the separation of sampling points slightly up or down. In other cases, it helps to adjust the contour of the region, for example by splitting the region in half. For this reason, a careful inspection of the preview is important, with a focus on the NURBS control net (magenta).

Although this functionality can produce IGES export surfaces of much higher quality than the automated output of pre-2018 (see [All faces to IGES NURBS patches] (discussed on page 139)) it involves the manual partition of the shell surface into four-sided regions as required by the IGES standard, while also taking care not to violate the remaining two conditions presented above. This may not always be a straightforward task. When necessary, it may be possible to cover a three-sided area by placing one corner within one of the sides, producing a so-called degenerate corner. This works best where the side has highest curvature and the surface is flat. For example: If a transom stern has one side along the deck line, one along the center buttock and one along the aftmost frame producing three corners, the fourth corner of the region may be placed in the kim of the frame. Othertimes one may need to cover a five-sided area, for which one may choose to add an extra polycurve connecting one of the corners with a point on the opposite side, producing two four-sided regions (see [New Planar Polycurve by Intersection] (discussed on page 80)). Even areas that do have four corners can benefit from being split into smaller regions if opposite sides have very different lengths, in order to prevent NURBS control points from being very tighly packed in the narrow end which can reduce smoothness due to limited floating point accuracy.

6.3.5.24.8 Region Validity

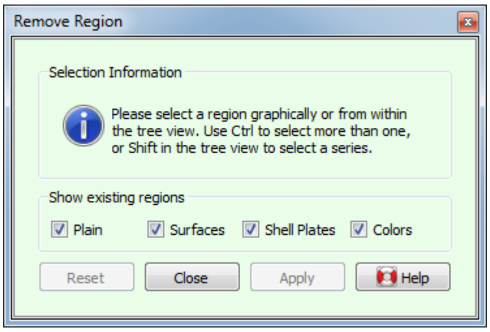
Clearly, the definition of regions depends on a particular state of the underlaying curve network. Instead of prohibiting relevant changes in the model, Fairway retains complete freedom and validates the correctness of regions at appropriate times. If, for example, the removal of a polycurve removes the support for one of its borders, the region will be marked as invalid with an icon in the tree view, and a tool-tip will explain what the problem is. In this example the problem can be resolved at any time by opening [Define Shell Region] and redefining the contour with the correct borders.

Changes in curve shape also have the potential to invalidate developable surfaces along two chines; as well as the ability to heal them. As explained in section 6.1.2.2 on page 59, Surfaces, rulings connect two points on opposite defining borders where the tangent vectors are co-planar. If there is too much twist among the defining borders it may happen that no rulings exist that satisfy these conditions. If this is the case, Fairway will complain with a message like "Could not construct a developable surface that contains the current defining chines [...]". The solution is to change the shape of the defining curves and reduce twist.

Another problem is when corresponding points do exist on opposite defining borders but their order is not synchronous. In this case, rulings are generated but they cross or split, and it actually means that the moving top has moved inside te region contour — a situation that is not physically possible. Fairway detects this condition, marks the region as invalid and shows the rulings in the prohibited color (red by default). In practice, this is likely caused by one or more bending points in one of the defining chines that is not sufficiently reflected in the other.

6.3.5.25 Remove Shell Region

This action allows the user to remove one or more defined regions (see section 6.3.5.24 on page 101, Define Shell Region) on the shell of solids. It is started from [Shell]→[Remove a Shell Region] or the keys <Alt><S><R>.

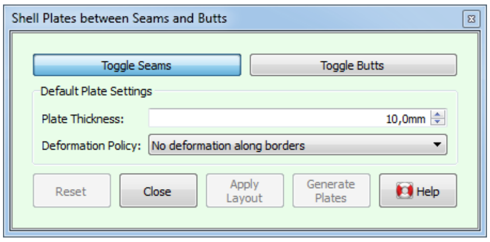


Removing regions

Note that regions may serve several purposes at once, and the visibility of regions can be filtered using the checkboxes at the bottom, as discussed in paragraph 6.3.5.24.1 on page 102, Visibility of existing regions.

6.3.5.26 Seams and Butts

Shell plating can be defined piece by piece as described in paragraph 6.3.5.24.4 on page 106, Definition of a shell plate, but for large vessels and simple geometry there is a faster way. This action works by defining the seams and butts where the plates should ajoin, in between which plates can be generated in one batch. The action is started from [Shell]→[Seams and Butts] or using the keys <Alt><S><S>.



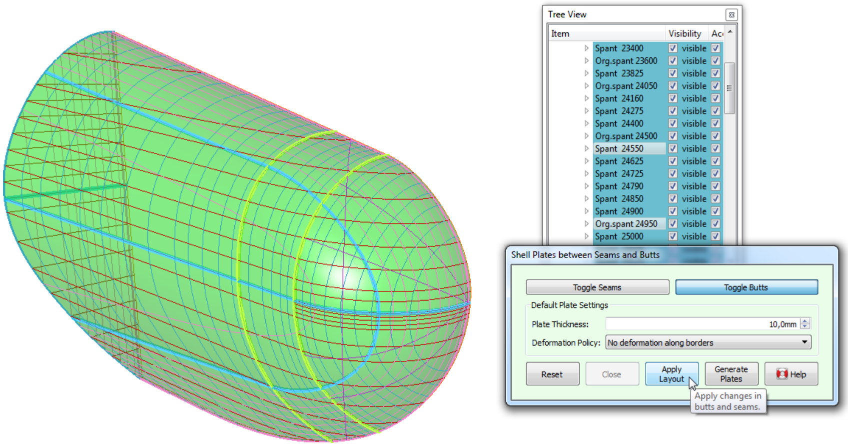
Defining seams and butts for batch generation of shell plates.

Depending on whether the [Toggle Seams] button or the [Toggle Butts] button is depressed, a click on a polycurve will select it as a seam or butt respectively; a second click will deselect it. Traditionally, where plates ajoin along their longer side are called *seams*, *butts* are where plates meet with their shorter side. Fairway however treats seams and butts equally, it does not care what is used. The only difference is that seams and butts are shown in different colors, which may be usitlized for optical clarity.

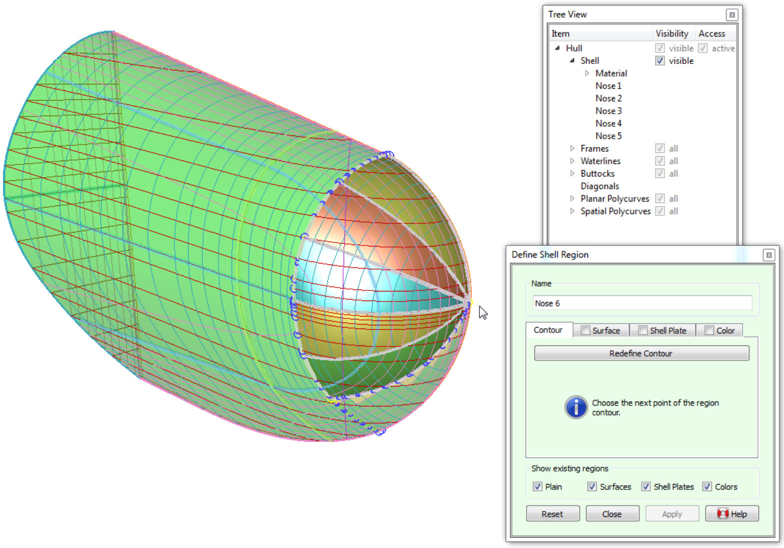
When seams and butts are layed out properly, a click on [Generate Plates] will automatically generate plate regions inbetween the seams and butts in all active solids at once, with a thickness and deformation policy as given in [Default Plate Settings]. These values (as well as the plate color) can be changed individually afterwards using [Definition of a shell plate] (discussed on page 106). In some cases, a generated plate may not be valid, so it is wise to check whether there are any regions marked invalid in the tree view after plates have been generated. This can happen, for example, when a seam does not end on another seam or butt. Plate expansions and templates can be generated from [Shell plate expansions and templates] in the main menu (discussed on page 64).

There are two details that are worth noting: Firstly, Fairway takes care not to generate plates where plates already exist. And Secondly, if the time is not yet ripe for the generation of all plates, the layout can be retained until later by pressing [Apply Layout]. This allows for exceptions on the regular layout of seams and butts, using the following workflow illustrated by the sequence of figures below:

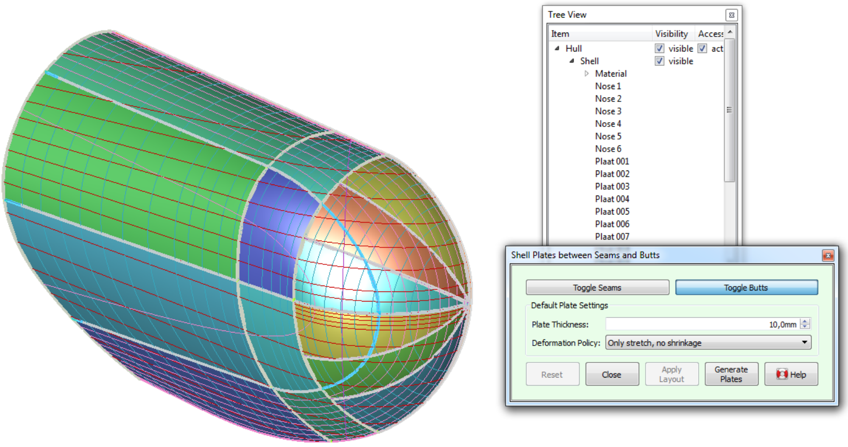
1. Define the general plate layout using [Seams and Butts], then press [Apply Layout].
2. Define exceptions on the general layout as individual plates, using [Define Shell Region].
3. Generate remaining plates in batch by pressing [Generate Plates] in [Seams and Butts].



Applying the general layout of seams and butts.



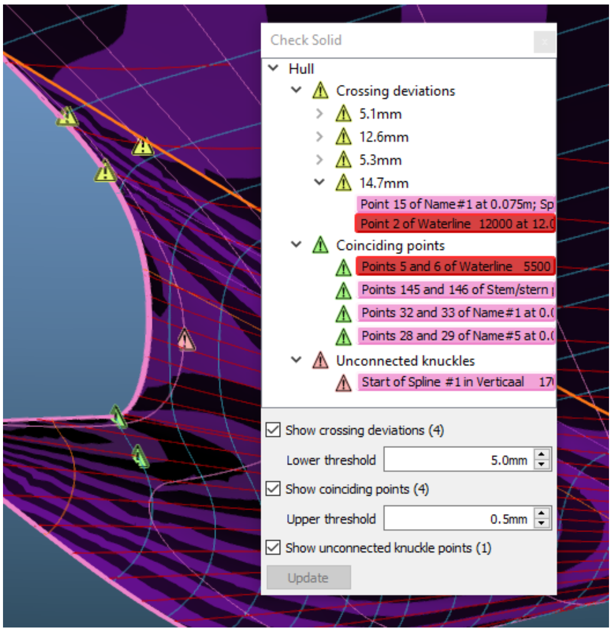
Defining exceptions as individual plates. Seams and butts are hatched.



Generating plates in batch, save existing plates.

6.3.6 Supporting functionality

6.3.6.1 Check Solid



The Check Solid docking window with markers in the model.

There are several kinds of conditions that can exist in a Fairway model that may be unwanted, although they are allowed and thus no errors. With the menu option [Objects]→[Check Solid] active solids are scanned for the following three conditions:

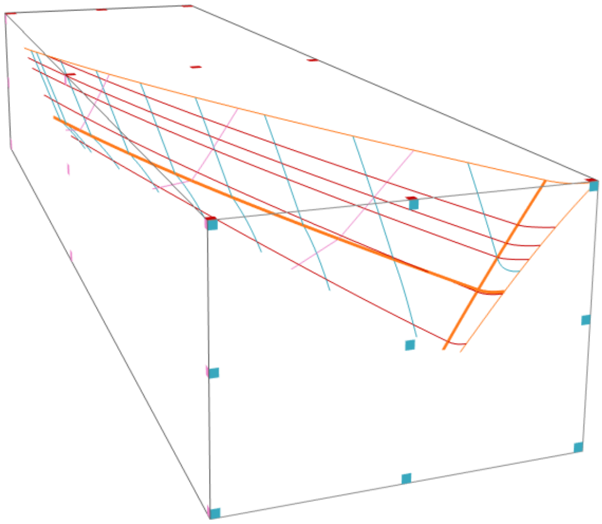
- **Crossing deviations**. Fairway allows crossing polycurves to deviate from their common intersection point to a configurable degree, which is a valuable functionality in the process of incremental hull fairing. However, larger deviations can cause deficiencies in surface generation, and a production-ready model should not contain deviations that exceed tight bounds. If the corresponding checkbox is ticked, points are indicated where the distance between intersecting curves exceed the given threshold. The actual distance is shown in the tree view of the docking window, and by expanding an item either of the two corresponding curves can be selected to resolve the deviation using [Change the shape of a curve].

- **Coinciding points**. Points that have identical coordinates or that are unproportionally closely spaced can be confusing, and they can cause complications for fairing and degrade surface rendering. Where possible, one may want to eliminate such coinciding points. By ticking the corresponding checkbox all points that fall within the given threshold are marked. The tree view in the docking window shows the respective curve.

- **Unconnected knuckle points**. Usually, one will want corresponding knuckles in successive polycurves to be connected by another polycurve or chine, to express that surface feature across the shell surface. To easily detect any unconnected knuckles, the corresponding checkbox can be ticked.

6.3.6.2 Clipping

To prevent parts of the model from occluding an area of interest, the menu option [Display]→[Clipping]→[Clip to Box] can be used to hide the parts of the model that fall outside a resizable box.



Clip to box.

The coloured facelets can be picked and dragged to resize the box (along the edges and in the corners) or to translate the box (in the middle of each face). The box can be resized to contain the whole model with the menu option [Display]→[Clipping]→[Clip Box Contain All]. The option [Display]→[Clipping]→[Hide Box] will hide the box and its facelets, but the clipping will remain active.

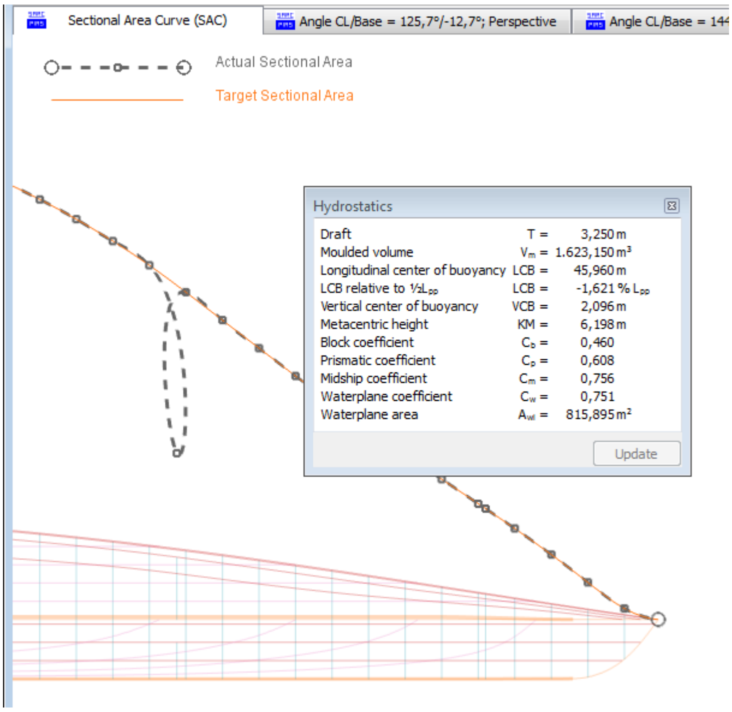
When a curve is being manipulated that partly falls outside the clipping box, it will not be clipped but drawn in its entirity.

6.3.6.3 Hydrostatic Data

The menu option [Hydrostatics]→[Hydrostatic Data] will bring up a window with hydrostatic information of the vessel. The window can either be floating separated from the main window, or be embedded in it somewhere around the modelling area. The hydrostatics are dependant on the frame shapes, so when a frame shape changes, the [Update] button in the hydrostatics window will be enabled, which allows the information to be recalculated.

Attention

> Hydrostatics are computed on the basis of the areas of the submerged parts of the frames. This mechanism might as such yield an incorrect result in case the frame set contains a frame not extending over the full draft, as depicted in the figure below. For this reason a mechanism is included to detect and ignore such incomplete frames. However, in order to verify the correctness of the result it is advised to inspect the SAC (Sectional Area Curve) with [Hydrostatics]→[Sectional Area Curve (SAC) Window].



Hydrostatics window and a defective sectional area curve due to a (not ignored) incomplete frame.

Attention

Note that the same information can also be obtained from the local cloud in real time, see section 2.11 on page 17, Local cloud: simultaneous multi-module operation on the same project. Through the cloud not only the upright hydrostatics are available, but (potentially) all PIAS results, such as full intact stability — loading conditions, GZ-curve and verification against the stability criteria — resistance and propulsion, tanks capacities etc. Not only this wider range of computation options is a reason that the local cloud is advised above this Fairway 'hydrostatics' function, another advantage is the availability of tools for graphical inspection of frames as they are actually used in PIAS' computation processes.

6.3.7 Wireframes

The main purpose of the wireframe functionality in Fairway is the import of 3D curves from a different source than Fairway — e.g. from a general-purpose CAD-system such as Autocad or Rhino — and their conversion to a Fairway model. The global procedure of this process will be discussed in section 6.3.7.2 on page 116, The entire procedure to import DXF or IGES files.

Having this functionality in Fairway also serves a secondary purpose, which is that a design which starts with a 'blank sheet' (a so-called *ab initio* design) can be set up sketch-wise with initially unconnected curves. At some point this wireframe will need to be converted into a solid though, which comes with its own set of challenges and requires an advanced understanding of topology and geometry.

Warning

Therefore we do not recommend this approach to the novice, while the following sections should be considered obligatory reading...

Note

The easiest way to do *ab initio* design is to start out with an initial shape that is already represented as a solid, by selecting any of the upper options in the start menu (discussed on page 63).

Finally, a third application of loose points and curves is in the support of the design of another solid, such as the marking of important positions, construction lines and as the master in master/slave relations.

6.3.7.1 Drawing exchange formats

IGES and DXF are popular file formats, and support for these formats is an important feature. However, DXF and IGES are not particularly suitable for the exchange of ship hull forms. One must keep in mind that these two formats are essentially intended for *drawing exchange*, and not for the exchange of product model data, an aspect which is confirmed by the full names of the acronyms (DXF: **D**rawing e**X**change **F**ormat, IGES: **I**nitial **G**raphics **E**xchange **S**pecification). Unfortunately, this background inhibits a guaranteed error-free, automatic import of hullforms as proper Fairway solids for every case. That is why IGES and DXF files are imported as a wireframe, after which they can be made suitable for conversion to solid within Fairway.

6.3.7.1.1 Importing 3D lines from DXF format

Attention

Import produces a wireframe only. Conversion to solid requires knowledge of the complete process, starting with section Wireframes (discussed on the preceding page).

Import of lines from a 3-D DXF is started with the corresponding option in the start menu (discussed on page 63). The following menu appears:

Correlation between ship and import file

1. Name import file (∗.dxf)	
2. The X-axis of the import file corresponds with the vessel's	Longitudinal axis
3. The Y-axis of the import file corresponds with the vessel's	Transverse axis
4. The Z-axis of the import file corresponds with the vessel's	Vertical axis
5. Merge succeeding curves	Yes
6. Multiplication factor on the import coordinates	1.0000
7. Minimum angle to recognize a knuckle in a polyline	10.0
8. Ratio of segment lengths above which a knuckle is inserted	500
9. Maximum length (m) of a polyline segment	500.0000

A 3-D DXF file contains dimensionless coordinates in an XYZ system of axes. So before the import can start the correspondence between DXF's XYZ coordinate system and the longitudinal, transverse and vertical axes of the ship must be specified. In addition, a multiplication factor must be given that will be applied on all DXF coordinates (for instance 0.001 when the DXF data are in millimetres).

Options 5 and 7–9 — from which the backgrounds are discussed in paragraph 6.3.7.1.1.1 on the following page, Intermezzo on polylines — are palliatives to deal with badly constructed DXF files. If possible, it can be a better idea to instruct the exporting computer program to structure its output in a more appropriate way. DXF supports geometric elements of quite a number of different types, of which Fairway supports the following:

- `POLYLINE` and `LWPOLYLINE`. With these types a curved line is approximated by a chain of many small straight lines. A property of this representation is that knuckle points are not defined explicitly (because theoretically each point is a knuckle). So knuckles have to be added artificially. This can be done automatically during the import according to the criteria of options 7–9, or manually in Fairway. Polylines are converted into polycurves that are faired through all points of the polyline, which can result in an unnecessarily large number of points per curve.
- `LINE`. This is simply a straight line between two points. Sometimes this type occurs inappropriately often in a DXF file, where another type would should have been used. An example is when all elements of what actually is a polyline are transferred as `LINE` elements. With option 5 switched on Fairway will merge successive lines (within a tolerance of 0.1 mm) to larger polylines regardless. If this option is off, import of a large number of lines will result in Fairway in a great number of unconnected short polycurves that tediously must be merged into longer curves and polycurves.
- `ARC`. A circular ARC.
- `SPLINE`. Actually this is a NURBS curve. A property of this representation is that multiple segments that together represent a continuous polycurve, are not connected in DXF and therefore most be connected in Fairway later on. Again, option 5 will do its best to merge successive curves automatically. Another aspect of this type of elements is that, although the geometry is transferred exactly, no internal points exist. Consequentially, if these curves are faired in Fairway prior to the insertion of a sufficient number of points, a lot of detail can get lost.

6.3.7.1.1.1 Intermezzo on polylines

Attention

> By its nature a polyline cannot distinguish between a knuckle and a non-knuckle point, and thus Fairway considers a polyline as a continuously curved curve, which is smoothed right through all polyline points. If a line contains one or more knuckles, it must be split into separate curves (which must be represented by distinct polylines) which meet at the knuckle points. If one continuous polyline is used for a knuckled line, the knuckles must be designated artificially, which may be a cumbersome job on a line containing hundreds of points.

It is a sad experience that more often than not only a model without explicit information about the knuckles is available. For those occasions the import procedure is equipped with auxiliary functions, which may help to create knuckle points automatically. However, **it must be emphasized that those functions are only palliatives, and are in no respect a replacement for proper definition of knuckles in the input data**. These functions come in four flavours:

1. With option [Merge succeeding curves] all imported curves are scanned to see if any of them can be merged into contiguous polycurves, joined by a knuckle. The criteria for this are that both curves must be either spatial or defined in the same plane, and that end-points coincide (with a tolerance of 0.1 mm).
2. A user can specify an angle (in degrees, option [Minimum angle to recognize a knuckle in a polyline]) between two successive lines of a polyline, above which the common point of those two lines is treated as a knuckle.
3. A critical ratio between the lengths of two successive lines of a polyline may be specified. When an actual ratio exceeds this critical one, the common point is treated as a knuckle.
4. A maximum length for a line of a polyline may be specified. If an actual length exceeds this maximum, additional points are inserted in that region (by linear interpolation). Those additional polyline points may give the interpolating line a broader support basis.

Regardless of the data carrier (DXF or IGES), NURBS curves are to be preferred above polylines. The reason is that Fairway converts polylines to NURBS anyway, and that conversion may always lead to a reduced accuracy.

6.3.7.1.2 Import 3D lines from IGES format

Attention

> Import produces a wireframe only. Conversion to solid requires knowledge of the complete process, starting with section Wireframes (discussed on page 113).

Import of lines from a 3-D IGES file is started with the corresponding option in the start menu (discussed on page 63). The following form appears, described below.

Correlation between ship and import file

1. Name import file (∗.iges)	
2. The X-axis of the import file corresponds with the vessel's	Longitudinal axis
3. The Y-axis of the import file corresponds with the vessel's	Transverse axis
4. The Z-axis of the import file corresponds with the vessel's	Vertical axis
5. Merge succeeding curves	Yes

A 3-D IGES file contains dimensionless coordinates in an XYZ system of axes. So before the import can start, the correspondence between IGES' XYZ coordinate system and the longitudinal, transverse and vertical axes of the ship must be specified. Option [Merge succeeding curves] has been discussed as first auxiliary function in the *intermezzo* just above.

IGES supports a large number of geometric entities. Fairway recognizes these entities:

- Entity type 110, the straight line.
- Entity type 126, the Rational B-Spline curve. Actually this is a NURBS curve.
- Entity type 128, the Rational B-Spline surface (a NURBS surface). Importing a set of NURBS can only be successfully done if the edges of adjacent surfaces coincide sufficiently.
- Entity types 142 and 144 ('Curve on parametric surface' and 'Trimmed surface'). These entities allow some exotic constructions which are not all supported. For example, an internal *trimming* is not processed, but that is not necessary because it indicates a hole, which should be avoided in the intrinsically closed hull of Fairway. Common forms of trimmed surfaces are normally processed.

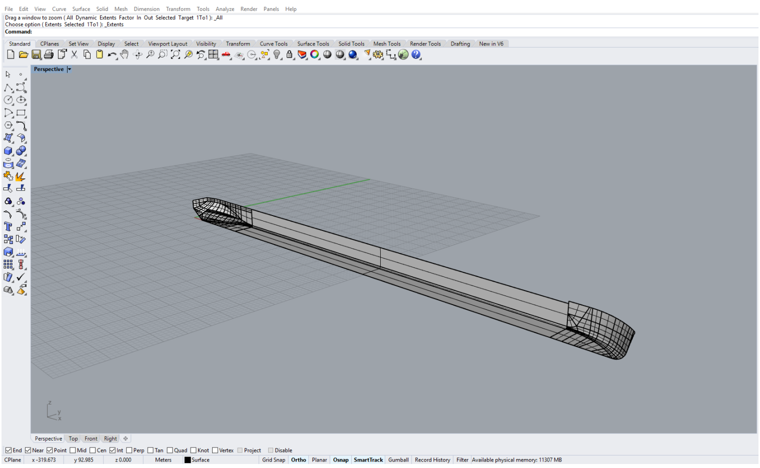
- Entity type 314, color definition, is ignored.
- Entity type 406, forms 3, 7 and 15. This concerns a *level* and the name, which is being recognized and skipped.

6.3.7.1.3 Import ship hull models in SXF/CXF format

SARC has defined an open file format that supports Fairway's polycurves and solid definitions to enable third parties to exchange models without the dependence on assumptions and heuristics. A model in this format consists of two files: the curve and polycurve definitions are in a text file with extension `.cxf` and the solid definition is in a text file with extension `.sxf`. The format of these files is discussed in the appendix, see section 6.A.2 on page 158, File CXF and SXF file formats.

For applications that must frequently export their hull forms to Fairway, it is recommended to develop an interface routine that writes CXF and SXF files directly. With this file combination there is no need for Fairway to reconstruct the shape of the solid, thus saving time and avoiding possible reconstruction anomalies. In this way, also external applications — e.g. for the generation of parametric objects, such as cilinders, gas tanks, NACA profiles, keels and rudders — can store their shapes in a fashion that enables smooth import into Fairway.

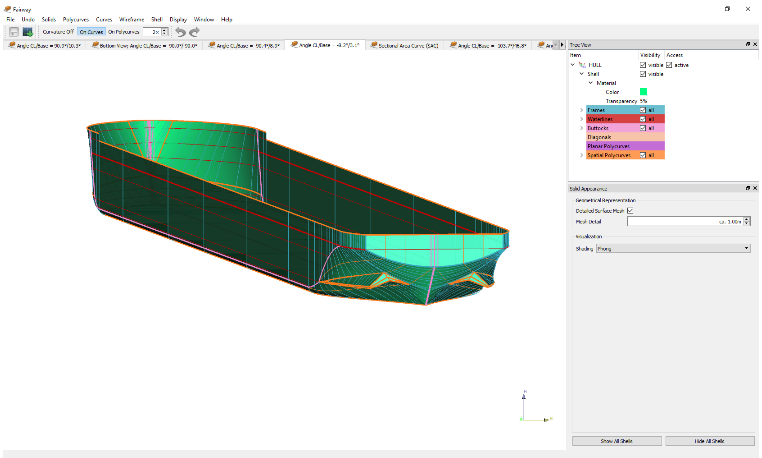
6.3.7.2 The entire procedure to import DXF or IGES files



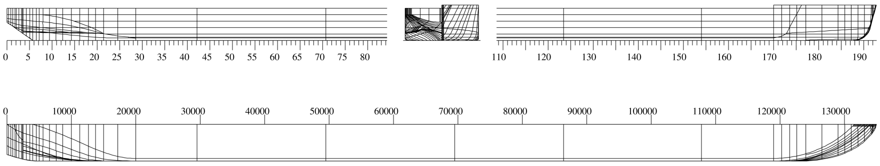
Ship hull, represented by IGES NURBS surfaces, in Rhino.

The procedure for importing an external file, and the conversion of its content to Fairway consists of some clearly identifiable steps (which are illustrated by the figures in this section, from which the hull form has been published in Mucha, P., el Moctar, O., Dettmann, T., Tenzer, M. *Inland waterway ship test case for resistance and propulsion prediction in shallow water.* Ship Technology Research 64(2):106-113, 2017):

1. Importing the 3D lines from DXF or IGES, as discussed in paragraph 6.3.7.1.1 on page 114, Importing 3D lines from DXF format and paragraph 6.3.7.1.2 on the previous page, Import 3D lines from IGES format.
2. If after import it appears that lines that are in reality continuous, in the source file do consist of multiple parts or separated line segments, then for a proper overview it is advised to merge them. That task could be performed in the source program (eg. Autocad), but also Fairway contains tools for this task. The easiest way is to switch option '5. Merge succeeding curves' to 'Yes', than all lines will directly be merged at import. Alternatively, lines can be merged manually with the function [Join polycurves] (discussed on page 86).
3. Generating a wireframe, with assistance of the so-called *wireframe points*, as discussed in section 6.3.7.6 on page 120, Wireframe points and section 6.3.7.7 on page 122, Wireframe connections.
4. Converting this wireframe to a Fairway solid, as discussed in section 6.3.7.8 on page 123, Convert Wireframe to Solid. For this conversion, a correct wireframe is a prerequisite, so first a test will be performed. If this is not successfully completed then the conversion will be aborted.



The imported model in Fairway.



Fairway-generated lines plan from the imported model.

6.3.7.3 Manual construction of a wireframe

Attention

> The recommended procedure for doing *ab initio* design is not described here, in general it is easier and faster to start out with a solid instead of a wireframe.

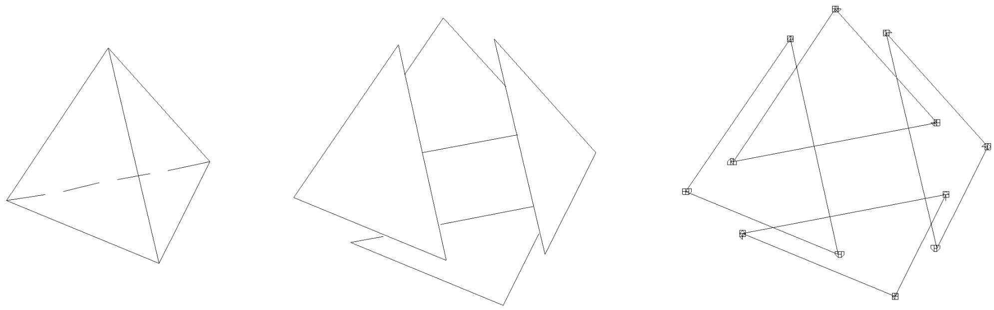
Normally, functionality for working with wireframes is disabled in the user interface of Fairway, in order not to confuse the user with options that would not be applicable. Therefore, if one wants to set up a wireframe by hand, this functionality needs to be enabled in [Fairway project settings] (discussed on page 133), option [Enable wireframe modeling]. When a wireframe is imported then this option will be enabled automatically.

Hereafter it is possible to add a new empty wireframe in [Object management] (discussed on page 155). In case a project needs no solids at all, the last option in the start menu (discussed on page 63) is appropriate: [Start with empty model (advanced)].

New free polycurves can be added to a wireframe by connecting points using the action [Connect Points]. Points can be added using the action [Wireframe points]. For further wireframe manipulations consult section 6.3.7.5 on page 120, Actions for wireframe manipulation.

6.3.7.4 A brief introduction to topology and connectivity of solids

A wireframe model is an open approximation of a solid, constituted by edges and vertices ('vertices' is the plural of 'vertex') on the boundary of the solid. For example, the object in the figure below contains 4 vertices and 6 edges. However, because the wireframe model does not describe the closed object it is ambiguous. A proper unambiguous way to describe a solid object is the method of boundary modelling, where explicit information about the faces is included in a so-called boundary representation, or B-rep. Our example below contains four faces.



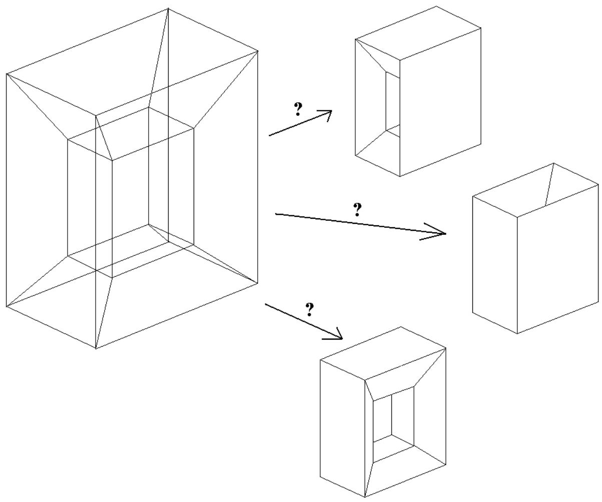
Solid, faces and vertices and edges.

There is a well-known relationship between the number of vertices (V), edges (E) and faces (F); for solids without trough-holes this so-called Euler relation is V-E+F=2. It can be verified easily that this relation is indeed applicable on the figure above.

Attention

> The Fairway user interface denotes vertices as *wireframe points*, mainly in order to avoid confusion with spline control points, which are also called vertices.

The necessity of explicit face information to describe a solid unambiguously can be demonstrated with the object in the figure below. With wireframe information only (vertices and edges) the actual shape of the object cannot be determined.

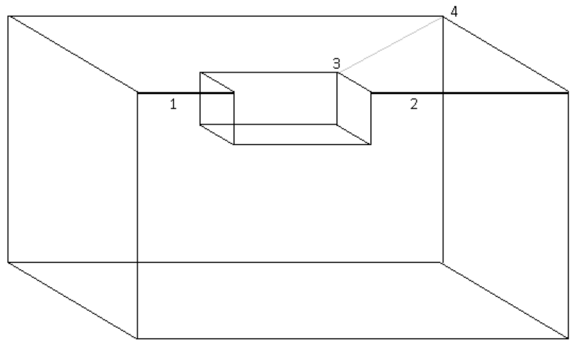


Wireframe only is ambiguous.

So an important task in the conversion of a wireframe to a solid is the recognition of the faces between the edges. In general, this problem is unsolvable, but under some practical constraints iterative methods are available. One of those methods is implemented in Fairway. This method implies the following constraints:

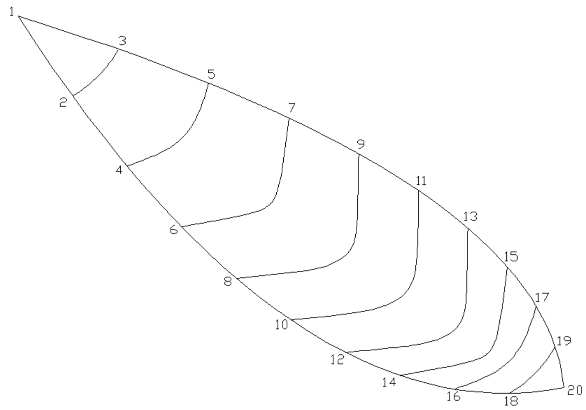
- The solid must be closed, without through-holes.
- The solid may not be 2-connected.

A 2-connected solid is a solid where the removal of 2 vertices separates the solid in two parts. For example, the object in the figure below is 2-connected, because the removal of vertices 1 and 2 leaves the small inner part unconnected from the larger, outer part. By the way, one additional edge between the vertices 3 and 4 would make this object not 2-connected anymore.



A 2-connected object.

One might perhaps expect at first sight that this freak object is unlikely te be encountered when importing ship's lines. However, consider a very simple initial body plan in the figure below. This one is highly 2-connected, because many combinations of edges exist which leaves the object separated at deletion (edges 2-4 & 3-5, 4-6 & 5-7, 6-8 & 7-9 etc.).



This simple bodyplan is 2-connected.

A few more remarks can be made about this figure:

- Apart from the theoretical aspects of 2-connectivity, one can also imagine that this wireframe cannot be converted into a solid automatically, because explicit information about inside and outside is missing.
- Extra waterlines or buttocks, but even the inclusion of one additional edge, between vertices 1 and 20, would make this object not 2-connected anymore.

It would lead us too far to discuss all theoretical aspects, but fortunately there are some exceptions where 2-connectivity is allowed:

- Some areas of 2-connectivity are acceptable, in the case when an unconnected part of the wireframe forms one face with the vertices which makes the object 2-connected. In the simple body plan above this is, for example, the case with vertices 2 and 3, whose removal leaves vertex 1 unconnected from the rest, which does no harm because vertices 1, 2 and 3 are all part of one valid face.
- For 'open' objects, such as the simple body plan above, 2-connectivity may be allowed.

All these considerations have led to a two-stage face recognition procedure in Fairway:

- Initially an object is assumed to be not 2-connected, and faces are generated accordingly. When a valid combination is found the generation process ends, because it is a theoretically valid solution.
- When no valid combination of faces is found, the wireframe is annotated graphically with pairs of vertices that cause the wireframe to be 2-connected. Finally the faces are constructed under the assumption that the object is 2-connected. However, this second stage may fail to find a proper solution.

6.3.7.5 Actions for wireframe manipulation

If wireframe modeling is enabled (see [Fairway project settings] (discussed on page 133)) then the user has access to the following dedicated actions:

- Wireframe points
- Wireframe connections
- Convert Wireframe to Solid
- Convert Solid to Wireframe

Further, the following generic actions also work on wireframe curves:

- Move Objects
- Scale Objects
- Rotate Objects
- Move polycurve
- Remove Polycurve
- Properties of polycurves
- Systemize polycurve names
- Join polycurves
- Split polycurve
- Connect Points
- Change the shape of a curve
- Bulk Change of All Curves

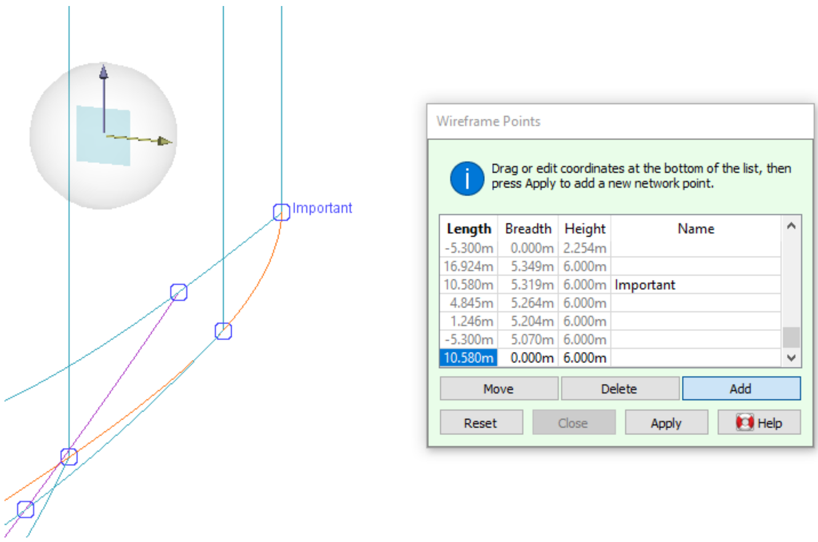
6.3.7.6 Wireframe points

Wireframe points represent the connecting vertices in a boundary representation, see section 6.3.7.4 on page 117, A brief introduction to topology and connectivity of solids. This action, which is started from the menu [Objects]→[Wireframe Points] or with the keys <Alt><O><P>, makes it possible to

- Move wireframe points,
- Label wireframe points,
- Delete wireframe points, and
- Add new freestanding wireframe points.

Since wireframe points can also be created implicitly where curves meet or intersect with section 6.3.7.7 on page 122, Wireframe connections, the purpose of this panel is mostly sketching, marking and labeling particular coordinates, and removing wireframe points (and their connections) that are not desired.

Contrary to ordinary points in Fairway, which are features of curves, wireframe points are features of a wireframe. Wireframe points are differentiated optically from curve internal points and knuckle points with a bigger marker, so that they do not obvuscate ordinary points when they are displayed at the same time.



Manipulation of wireframe points (vertices).

Note that it is possible for a wireframe to exist without wireframe points — a newly imported DXF or IGES wireframe will not have wireframe points, which means that curves are unconnected and without structure. Curves can only connect through wireframe points. Wireframe points are essential in the conversion from a wireframe to a solid: they define the topology of the wireframe, giving information on which curves are connected where. Strictly speaking, a solid also has B-rep vertices or wireframe points, but in a solid their connections are permanent and are therefore hidden from the user.

A wireframe can also exist without curves and purely consist of wireframe points. This can occur after the import of a point cloud or manual definition of a point set, with the intention of constructing wireframe curves through the points afterwards. Another application of a wireframe in Fairway is just to group freestanding points and curves in support of ordinary solid modeling, to mark important positions and shape features, without the need to be part of a boundary representation. Such a wireframe would just be for marking geometric information, be it temporarily or permanently, and never be converted to a solid. The ability to label wireframe points can be a practical feature in this use case.

6.3.7.6.1 Move wireframe points

When the [Move] button is depressed, the position of wireframe points of active wireframes can be changed, either by dragging or with the keyboard after pressing <F2>. If there are any curves connected to the wireframe point, these curves are faired through the new position, and the available axes of motion are constrained to the planes in which connected polycurves are defined.

6.3.7.6.2 Label wireframe points

As an added feature, wireframe points can be labelled with a name. The names will be added as a graphical annotation to the wireframe points after the action is applied.

If there are many points present in a wireframe, the list can be long and unordered. It may therefore be most practical to depress the [Move] button and identify the point graphically, which will bring the corresponding row in the list into focus. Then press <F2> to start editing, and press <Tab> to proceed to the [Name] collumn.

6.3.7.6.3 Delete wireframe points

Wireframe points can be deleted when the [Delete] button is depressed. Obviously, this will disconnect any curves that were connected to it. Pressing <Ctrl+A> will remove all wireframe points in all active wireframes.

6.3.7.6.4 Add new freestanding wireframe points

If curves are present, one would most frequently be interested in adding wireframe points where curves should connect. For that, another action is more appropriate, see section 6.3.7.7 on the next page, Wireframe connections. But when a new wireframe point must be added that is freestanding and unrelated to curve intersections, this can be accomplished with this action when the [Add] button is depressed. The position of the prospective wireframe point can be dragged with the dragger, or typed in as the last item in the list. The point will be added when the action is applied.

6.3.7.7 Wireframe connections

This action manages the connections between curves in a wireframe, which is a vital step in the conversion to a solid. Wireframe points represent the vertices in the boundary representation (B-rep) of the solid. The action is started from the menu [Objects]→[Wireframe Connections] or with the keys <Alt><O><C>.

The main purpose of this action is to determine which curves intersect and where they should be connected with a shared wireframe point. The most important criterion in this assessment is the value in the [Tolerance] field.

Attention

> Tolerance should not be confused with accuracy. In particular, *you should not specify a low tolerance with the intention to achieve high accuracy; doing so will most likely give you adverse results.* See also paragraph 6.3.7.7.4 on the following page, Why does a low tolerance result in duplicate points?

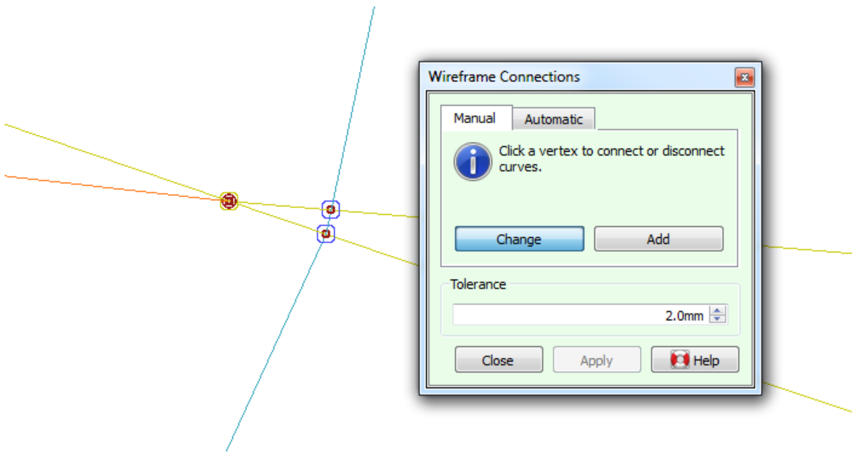
The process of searching for curve intersections is as follows.

1. The distances between curves are evaluated.
2. Where the distance between two curves falls below the given tolerance, they are classified as intersecting.
3. For each intersection, positions on the relevant curves are found that minimize their distance.
4. If no wireframe point exists within the tolerance of these positions, a new wireframe point is created in the middle. So, in case the curves intersect exactly, the new wireframe point will be placed exactly at the intersection. If curves do not intersect exactly (due to, for example, fairing with a mean fairing deviation) then the wireframe point is positioned where it yields the highest accuracy.
5. The relevant curves are scanned for points (internal points and knuckle points) that are within the tolerance of the wireframe point. If no point is found a new internal point is inserted.
6. The respective points on the curves are given a reference to the wireframe point to mark their connectivity.

Note

> As a rule of thumb, the tolerance can be specified quite high, but not higher than the closest distance between curves that should not be connected. Nor should it be higher than the distance between intersections that must be distinct. If no internal points are present in curves where they intersect, the tolerance should also not be higher than the distance between the intersection and unrelated points on the curve.

It should be clear that there is no one-size-fits-all tolerance. Therefore it may be necessary to experiment a bit, and use [Undo] if results are not as intended. It may well be that most connections can be made automatically, but that a few cases remain that require manual attention.



Inspection and manipulation of wireframe curve connections.

6.3.7.7.1 Change existing connections

If existing wireframe points are present and the [Change] button on the [Manual] tab is depressed, wireframe connections to these points can be inspected, made and broken. Initially, moving the mouse pointer over wireframe points will highlight the curves that are connected to it. This way it is easy to scan for missing or incorrect connections, see the figure.

After clicking on a specific wireframe point, connected curves are selected. By clicking on a selected curve it will be disconnected from the wireframe point, and selecting an unselected curve will connect it *if it runs by the wireframe point within the tolerance*. So if a curve cannot be connected, try again after increasing the value in the [Tolerance] field. It is not necessary to hold <Ctrl> while selecting and deselecting curves.

6.3.7.7.2 Add a new connection manually

A single new connection between two curves can be added by hand when the [Add] button on the [Manual] tab is depressed. Holding <Ctrl>, two curves should be selected that approach eachother within the given tolerance. A new wireframe point will be added to which the selected curves are connected. If more than two curves should intersect at the same point, the additional curves can be added afterwards as explained in paragraph 6.3.7.7.1 on the previous page, Change existing connections.

6.3.7.7.3 Add new connections automatically

Switching to the [Automatic] tab will initiate a search in which every curve in the wireframe is compared against every other curve. It is most practical to adjust the tolerance before switching the tab as changing the tolerance will restart the search. New wireframe points will be added for every curve intersection that is found for the given tolerance.

Before pressing [Apply], the generated connections may be inspected by changing back to the [Manual] tab, pressing the [Change] button and hovering over the generated wireframe points as described in paragraph 6.3.7.7.1 on the preceding page, Change existing connections.

If [Apply] is pressed while the [Automatic] tab is active, the action will immediately start a new search. This is harmless and the action can be closed at any time.

Attention

> After connections have been generated and the action is applied, repeating the action with a different tolerance will not automatically remove connections already present. Use [Undo] to remove the connections made in the previous run. When necessary, all existing connections can be removed in one go, see paragraph 6.3.7.6.3 on page 121, Delete wireframe points.

6.3.7.7.4 Why does a low tolerance result in duplicate points?

After connections have been generated, a validity check of the wireframe (paragraph 6.3.7.8.1 on the following page, Checking the validity of wireframes) can report a high number of duplicate points. Two points on the same curve are considered duplicate if their separation is not more than 1 mm, which is a problem in the conversion to solid. Duplicate points can of course be resolved by deleting individual points from curves using [Delete] (discussed on page 91) from the [Change the shape of a curve] action, but the problem can likely be avoided by connecting with a more appropriate tolerance, or by reducing the mean fairing deviation of all curves using [Bulk Change of All Curves] (discussed on page 99). The reason why too low a tolerance can produce double points, even though polylines in the input file run through identical coordinates, is the following.

If polylines have been converted to curves by the import procedure (see paragraph 6.3.7.1.1.1 on page 115, Intermezzo on polylines) then a non-zero mean fairing deviation will allow a deviation between the coordinates of the imported polyline and the faired curve, see paragraph 6.3.5.19.2 on page 89, Fair. Consequently, the search in steps 3. and 4. (discussed on the previous page) may yield an intersection point that deviates from the closest polyline coordinate. Since a connection between a wireframe point and a wireframe curve requires a point on the curve within the tolerance to the wireframe point, in step 5 a new point will be inserted on the curve if the distance to the existing point exceeds the tolerance. Likely, the new point is within 1 mm of the existing point, which qualifies them as double points. With a larger tolerance the existing point would have been within reach of the wireframe point, and *it* would have been connected, without the introduction of double points.

6.3.7.8 Convert Wireframe to Solid

This action attempts the conversion from wireframes to solids, and gives diagnostic feedback in case the conversion fails. It is started from the menu [Objects]→[Convert Wireframe to Solid] or using the keys <Alt><O><S>.

The process consists of two stages. The first stage is an initial validation of active wireframes, described in paragraph 6.3.7.8.1 on the following page, Checking the validity of wireframes. The second stage consists of an iterative search for a complete boundary representation as described in section 6.3.7.4 on page 117, A brief introduction to topology and connectivity of solids. If this search is successful then the action can be applied after which the solid is a fact. If the search fails, a list of deficiencies is displayed and graphically annotated, which should aid in the investigation why the wireframe cannot represent a solid, described in paragraph 6.3.7.8.2 on this page, Wireframe conversion feedback. The annotations of the check and the feedback can be hidden and recalled at any time, also during the use of other actions, so one can easily focus one's attention to the problematic areas of the wireframe and fix its shortcomings.

6.3.7.8.1 Checking the validity of wireframes

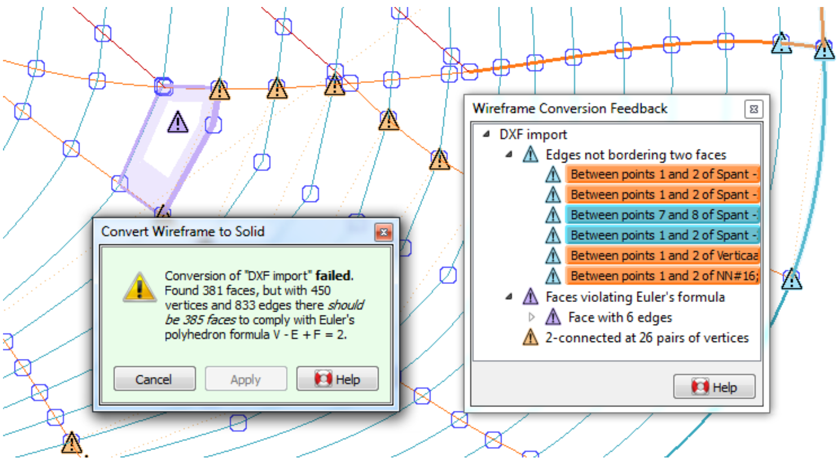
The validity check is automatically started when the [Convert Wireframe to Solid] action is initiated, but can also be started independently. The results of the check are listed in a separate tree view window, which can be shown explicitly from the menu [Objects]→[Check Wireframe]. This window can float on top of the GUI, or be docked along any of the sides of the main window, just like the main treeview (discussed on page 65). The individual issues are annotated with colorcoded hazard triangles and the items in the list can be clicked to directly select the corresponding curve.

There are three conditions that the wireframe is checked against:

- ⚠ Loose ends. Loose ends are where wireframe curves have ends that are not connected to other wireframe curves. Loose ends can be resolved using [Wireframe connections] (discussed on page 122) or by trimming the curves using [Change the shape of a curve] (discussed on page 89) or by extending the curves using [Connect Points] (discussed on page 86).
- ⚠ Duplicate points. Points on curves (internal points as well as knuckle points) must be separated by at least 1 mm. Duplicate points can be removed with [Delete] (discussed on page 91) and possibly avoided as described in paragraph 6.3.7.7.4 on the preceding page, Why does a low tolerance result in duplicate points?
- ⚠ Duplicate segments. Duplicate segments are connections between wireframe points that are topologically identical, meaning that they describe indistinguishable paths between two wireframe points. Although duplicate segments may be geometrically distinct, the order in which you would cross them while walking over the imaginary surface is not clearly defined. Duplicate segments can be resolved by constructing a crossing curve that connects the segments with at least one other wireframe point in a way that makes the order unambiguous. Note that the shape of this curve is subordinate, it can be removed from the solid after the wireframe has been converted successfully.

6.3.7.8.2 Wireframe conversion feedback

If the wireframe conversion fails, the action pane will display the mismatch in Euler's relation, and the feedback may show issues of three different types in a similar way as the wireframe check (discussed on the current page). This feedback can be hidden by closing the issue tree view window, and shown by restoring the window from [Objects]→[Wireframe Conversion Feedback].



Feedback on a failed conversion to solid.

The issues indicate where the search algorithm got stuck. This need not indicate the exact problem, but often the problem can be found in the neigbourhood.

- Edges not bordering two faces. In a valid boundary representation of a solid, every edge should connect exactly two vertices and border exactly two faces. Issues of this type typically indicate that one of the incident faces to an edge could not be unambiguously identified. The hazard triangles identify the wireframe points that are connected by this edge, and the corresponding curve section is marked with the same color.
- Faces violating Euler's formula. Euler's relation was discussed in section 6.3.7.4 on page 117, A brief introduction to topology and connectivity of solids, and the indicated faces are where deviation from Euler's relation was detected. As Euler's relation considers the wireframe as a whole, the wireframe deficiencies need not occur at the indicated faces, but often they can be found in their vicinity.
- 2-connected at vertex pairs indicates the pairs of vertices that make the wireframe 2-connected, meaning that removal of any of these pairs and their incident edges would partition the wireframe into two distinct sets. As indicated in section 6.3.7.4 on page 117, A brief introduction to topology and connectivity of solids, in many cases 2-connected wireframes can be converted successfully regardless, if no other deficiencies are present. If Euler's relation is not complied with, then there are probably other deficiencies that need your attention first.

It can be a challenge to spot why the wireframe cannot describe a solid unambiguously. These are some of the things that are worth checking against:

The wireframe represents a solid with one or more through-holes.
Make sure to remove all curves that define the hole so that a continuous surface can be found across the hole openings.

Curves connect in an impossibe order, comparable to the work of artist M.C. Escher.
Remove the offending curve parts by splitting and/or deleting end points. Optionally, proper connections can be constructed by connecting points.

Missing connections / intersection points.
Use the prelight function on mouse-over from [Change existing connections] (discussed on page 122) to look for missing or incorrect connections.

6.3.7.9 Convert Solid to Wireframe

This action converts all active solids into wireframes. It is started from the menu [Objects]→[Convert Solid to Wireframe] or using the keys <Alt><O><W>.

As opposed to solids, wireframes do not have the ability to generate a surface or compute a volume. On the other hand, wireframes do not impose strict connectivity requirements like solids do, and polycurves in a wireframe can be freely connected to and disconnected from each other, using [Wireframe connections]. Wireframes can be converted back into solids by using [Convert Wireframe to Solid], unless it has become ambiguous as a solid representation.

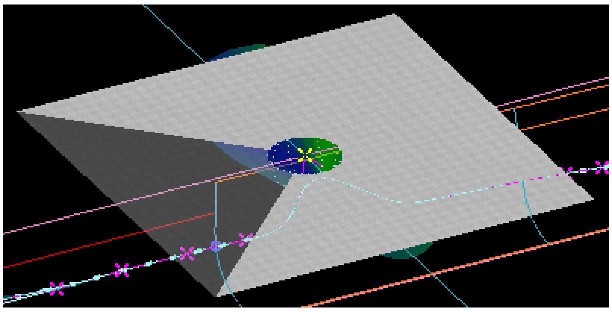
6.3.8 Troubleshooting

This section contains some known problems with their solutions.

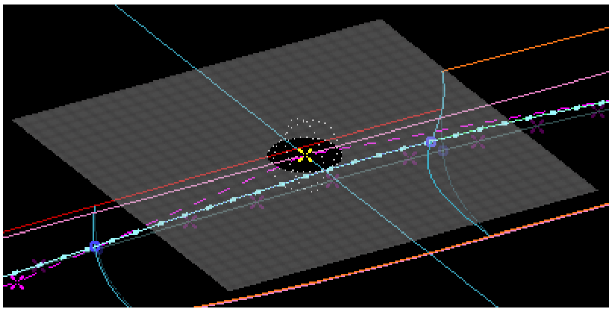
Problem	**Likely cause**	**Solution**
The drop-down for boundary conditions, point weights and knuckles opens at double click, but closes immediately.	This is a problem that occurs for a certain Windows configuration option.	The drop-down will probably stay open for as long as you keep the mouse button pressed as part of the second click. You can then make a selection by releasing the button over the desired item. In Windows versions prior to Windows 10, the drop-down will also stay open after the double click by checking the “Slide open combo boxes” option in Control Panel -> System and Maintenance -> System -> Advanced system settings -> Performance Settings -> Visual Effects.
Upon entering the graphical user interface (GUI) Fairway hangs and freezes just before or during the initial rendering of the model, and remains unresponsive. The operating system may report that Fairway does not respond anymore, with the option to terminate the program.	This is a known problem on Nvidia Quadro graphics adapters; at least the Quadro 1000M and M1000M, Quadro M2000M, Quadro K420, Quadro K2000, Quadro K3000M, Quadro K3100M and Quadro P600 exhibit the problem with driver versions 320.00–391.58 and possibly other types and versions. Nvidia GeForce adapters are not affected and work very well.	There are three possible solutions for this problem; if one does not work, another probably will. • Open the Nvidia control panel by right-clicking on the background of the Windows Desktop. Navigate to “3D Settings -> Manage 3D settings” and switch off the feature “Threaded optimization”. It is possible to change this setting for `fairway.exe` specifically by adding a new profile. Fairway is multi-threaded already, and needs no help from the Nvidia driver. • Download and install a driver update from the nVidia website. At least one case is known where a P600 (released 2017) was shipped with a driver where the above had no effect. • When using Windows 10: In the file explorer, right-click on `fairway.exe`, choose “Properties”, select the “Compatibility” tab and check “Run this program in compatibility mode for Windows 8”.

Problem	**Likely cause**	**Solution**
Upon entering the graphical user interface (GUI) Fairway crashes, possibly with an appcrash message in `aticfx32.dll`.	This is a known problem in the ATi driver, specifically version 12.104.0.0 released on 19-04-2013 in combination with the AMD Radeon HD 6570 graphics adapter, and possibly other types and versions.	The problem has been fixed in version 14.100.0.0 of the driver, distributed as part of the AMD Software Suite 14.4, released on April 25, 2014. Version 8.850.0.0 of the driver is also known to work.
Curves of the same type show with different intensity.	The anti-aliasing setting is on, and you don't like it. Anti-aliasing is a technique to reduce distortion when representing high precision graphics on lower resolution devices such as a raster display. Without anti-aliasing, curves have a jagged appearance because pixels are either turned fully on or fully off. Anti-aliasing results in graded pixel values, making curves look smoother and more precise, albeit a bit woollier. When the position of a vertical (or horizontal) line coinsides exactly with the pixel raster, it is displayed with a width of 1 pixel at full intensity, whether anti-aliased or not. When the line position falls in between two pixel positions, it may be displayed by either of the two pixels without anti-aliasing, at a visual inaccuracy of half a pixel. With anti-aliasing, both pixels are turned on at a reduced intensity, giving the perception of higher accuracy.	On some monitors the difference in intensity due to anti-aliasing is more noticeable than on others. You can turn off anti-aliasing by choosing [File]→[Preferences...]→[OpenGL] and turning off "Hardware-based smoothing" and "Multi-sampling". Alternatively, you can reduce the difference in itensity by increasing the line width in [File]→[Preferences...]→[Curves].
The middle mouse button does not work.	Your mouse driver might be configured to assign a task to the middle button.	Exit or uninstall the mouse driver, or configure the driver. If you are using Logitech Set↩ Point™: 1. Open SetPoint 2. Select the "My Mouse" pane 3. Select the third button 4. In "Select Task", mark "Other" 5. Choose "Middle Button" If nothing helps, you can use the Navigation mode (discussed on page 70) instead.
My 3Dconnexion SpaceNavigator does not work.	There is no graphics area that has input focus.	Give a mouse click in a graphics area.

Problem	Likely cause	Solution
	The driver has not been started while the device was plugged in.	Make sure that the device is plugged in. Then start the driver from the Windows start menu, folder “3Dconnexion”, subfolder “3DxWare”, item “Start Driver”.
	The driver may not be responding.	Start the Windows Task Manager <Ctrl+Alt+Del> and end the 3dxsrv.exe process. Then follow the points above.
The appearance of some objects on screen seems wrong. Figure fig. 6.67, e.g., shows how the ATi RADEON X300 SE display adapter partly fails to render transparency.	There may be a bug in the driver for your display adapter.	Visit the website of your adapter manufacturer and locate the latest driver that matches your graphics card. Follow the instructions for installation.



Render defect with old display adapter driver.



Proper transparency with updated driver.

6.4 Define main dimensions and other ship parameters

This menu, accessible from the Fairway main menu, is also found in Hulldef. For the operation of Fairway the first option is of primary interest, and possibly the fourth, discussed below. All other ship hull parameters that can be specified in this menu are only used for successive calculations with PIAS. These parameters are discussed in section 7.2.1 on page 167, Define main dimensions and further ship parameters.

Define main dimensions and other ship parameters

1.	Main dimensions (design) & hull coefficients
2.	Main dimensions and allowance for shell and appendages
3.	Roll data (for Intact Stability Code weather criterion)
4.	Frame spacings
5.	Maximum drafts and minimum freeboards
6.	Allowable maximum trims
7.	Particulars of export to Poseidon
8.	Yacht particulars
9.	Particulars for SOLAS chapter 2, part B1
10.	Anchor handler particulars
11.	Towing hook and bollard pull
12.	Particulars for inland waterway container vessels
13.	Line-of-sight and air draft points

6.4.1 Main dimensions (design) & hull coefficients

Here main dimensions for the hull design can be specified, and, in addition, also a number of hull form coefficients that are used as target values in the design. With these coefficients (block coefficient, LCB and midship coefficient) a SAC can be generated, which can be utilized in two ways:

- As basis for a global hull form transformation (see section 6.5 on this page, Hullform transformation).
- As guide when designing the frame shapes zie paragraph 6.3.5.19.8.1 on page 96, Show Target Frame Area.

Also the SAC itself can be generated in two ways, either automatically, see section 6.5 on this page, Hullform transformation, or by interactive graphical manipulation, see section 6.3.5.21 on page 97, Change the shape of the SAC.

6.4.2 Frame spacings

If construction frame spacings have been defined in this menu, then length positions of frames in the actions [New Planar Polycurve by Intersection] (discussed on page 80) and [Move polycurve] (discussed on page 83) can be inspected and modified in terms of construction frame numbers and an optional offset in millimeters. The settings for this option are discussed in section 7.2.1.3 on page 167, Frame spacings.

6.5 Hullform transformation

The GUI allows you to make very precise local changes in hull shape, or to design a hull ab initio. However, it may sometimes be easy to deform or rescale an existing shape globally. This is called hull form transformation — or hull form variation — and Fairway offers also functions for that, which are discussed here. Incidentally, the two methods can also be mixed: first a global transformation, then for example a local modification at a chine or a bulb, and finally again a global transformation to bring the block coefficient or the longitudinal center of buoyancy exactly the desired value.

Attention

> The hullform transformation is applied on and with the solid that is *single selected* in menu [Object management] (discussed on page 155).

The hullform transformation functionality is, by the way, also available within the GUI, please see section 6.3.5.5 on page 76, Shift Frames (Lackenby) and subsequent paragraphs. After the transformation option has been selected here, from the Fairway main menu, a selection menu appears with only three options:

Hullform transformation

6.5.1 Transformation parameter menu

This menu contains the following parameters:

- Length between perpendiculars (L_{pp}), as given in [Define main dimensions and other ship parameters] (discussed on page 128).
- Moulded breadth (B_m), ditto.
- Draft (T), ditto.
- Block coefficient (C_b) ($C_b = \Lambda/(L_{pp} \cdot B_m \cdot T)$).
- Moulded volume (Λ).
- LCB (% of L_{pp} from $L_{pp}/2$).
- Midship coefficient (C_m) ($C_m =$ Largest ordinate area$/(B_m \cdot T)$).
- Transformation type, zoals dat besproken wordt in section 6.5.3 on this page, Transformation types and their properties.

Depending on the chosen transformation type all or only some parameters will be included. If for instance the *linear scaling* transformation type is selected the hull coefficients will not appear, because they are not modifyable with this transformation type. This menu contains two columns. The first column shows the desired transformation parameter values, as entered by the user. The second column contains the actual values from the solid to be transformed — the single selected solid. Furthermore, the following commands are available:

- [Copy], which copies the parameter values of the 'solid' to the 'desired value' column.
- [Transform], which transforms a global hullform transformation.

6.5.2 Specify envelop lines midship section

These 'envelop lines' represent the hull limits as applied by the 'inflate/deflate' transformation method (as discussed in section 6.5.3.3 on the next page, Inflate/deflate frames). For other transformation types these lines do **not** have to be given. By specifying these lines, the frames are forced to stay within this envelop. A maximum of ten points can be given, so there is ample space to accomodate knuckles, deadrise etc.

6.5.3 Transformation types and their properties

The following transformation types are supported by Fairway.

- Linear scaling
- Shift frames (Lackenby)
- Inflate/deflate frames
- Increase/decrease parallel midbody
- Shift complete vessel
- Perpendicular to the shell

6.5.3.1 Linear scaling

All transverse, vertical or longitudinal coordinates are multiplied by a factor. The modifyable parameters are L_{PP}, B_m and T, the coefficients will not change.

This transformation is also available as [Scale Objects] (discussed on page 75), which has the advantage of being undoable, and applies to wireframes as well.

6.5.3.2 Shift frames (Lackenby)

The principle of this Lackenby transformation type[1] is that the frames are shifted in longitudinal direction while the frame area and frame shape remain unchanged. This is done in such a way that the desired parameters are obtained. In Fairway this principle has been extended with an optional scaling of the frames, by which variations in breadth and draft are also supported. **All** points on the hull are shifted when using this option, contrary to the [Inflate/deflate frames] (discussed on the current page) method.

This transformation is also available within the GUI, which has the advantage of graphical feedback and the availability of undo/redo, see section 6.3.5.5 on page 76, Shift Frames (Lackenby).

6.5.3.3 Inflate/deflate frames

With this transformation type the desired values of the parameters are obtained by 'inflating and deflating' the frame shapes. The points of the frames are shifted perpendicular to the frame shape outwards or inwards. Care is taken to preserve the frame shape as much as possible, without exceeding the extreme hull limits (as represented by the lines as defined in section 6.5.2 on the previous page, Specify envelop lines midship section). With this transformation type it is possible to change all parameters (only this type can also change midship coefficient (C_{m})). With this type of transformation **only points on the frames are relocated**, all other points in the network, such as points located on waterlines only, remain unchanged.

This transformation is also available within the GUI, which has the advantage of graphical feedback and the availability of undo/redo, see section 6.3.5.6 on page 77, Inflate/Deflate Frames.

By the way, this transformation type is also used in the hullform transformation module which is applicable on non-Fairway hulls in PIAS, Hulltran.

6.5.3.4 Increase/decrease parallel midbody

When selecting this type, on the first line the desired new length between perpendiculars should be given. The second row the location of the aft side of the parallel midbody is entered. The additional parallel body (in case of lengthening) starts at this point and has a constant section equal to the section at this point.

This transformation is also available within the GUI, which has the advantage of graphical feedback and the availability of undo/redo, see section 6.3.5.7 on page 79, Increase/Decrease Parallel Section.

6.5.3.5 Shift complete vessel

When using this transformation type the ship is shifted as a whole. With this option you can simply shift, for example, the base, aft perpendiculars etc. After selecting this option, three input fields will appear: longitudinal shift, transverse shift and vertical shift.

This transformation is also available as [Move Objects] (discussed on page 74), which has the advantage of being undoable, and applies to wireframes as well.

6.5.3.5.1 Perpendicular to the shell

With this scheme points of the hull are shifted normal to the hull, with a user-specified positive (outwards) or negative (inwards) offset. The normal-direction can only be determined at the intersection point of two lines. This implies that internal points must be absent for this option and they will be removed by the program automatically. Note that the normal-direction is undefined at knuckles; the program will take the average of the normals around the knuckle. It is unavoidable that undulations in the vicinity of knuckles may occur, particularly with negative offsets (inward).

6.5.4 Hints for and backgrounds of the transformation process

6.5.4.1 Which transformation type to apply?

When performing a 'real' transformation (so, not something simple as scaling) the question might arise which transformation method to use: 'inflate/deflate' or the frame shifting method of Lackenby. The answer is up to the user, however, the following properties can be mentioned for the two methods:

- With 'inflate/deflate' frames remain on their original location, while with Lackenby they will be shifted. That is an advantage of 'inflate/deflate'.

[1] According to H. Lackenby (1950) 'On the systematic geomatrical variation of ship forms', Trans. INA, Vol.92, pp. 289–316.

- Lackenby allows in general somewhat larger transformations than 'inflate/deflate'. If the ship has a parallel midbody then Lackenby will expand or shrink it as needed. If a ship has no parallel midbody, and it should be become much fuller, then Lackenby will not insert the parallel body. In that particular case it is advised to add a parallel body separately, and then to apply the Lackenby transformation.
- With 'inflate/deflate' only the frame shape is modified, points not located on a frame (for example only located in a waterline) are not midified. That is a major disadvantage, which can be relieved to some extent by removing all 'internal points' (which are points not situated on the intersection with an other curve), see section 6.6.6 on page 134, Remove all internal points from all curves.
- With 'inflate/deflate' also the midship coefficient can be modified, which is not possible with Lackenby.

Overseeing this list, for significant transformations **Lackenby is to be preferred** above 'inflate/deflate', except in those cases where the midship coefficient is to be modified. Limits for changes in parameter, which still lead to decent hullforms, cannot be given, those depend on the particulars of the hullform. For example, the block coefficient modification limit of a slender ship will be higher than that of a full ship. That is because the slender vessel has more room available in the middle, and particularly in the ends, which facilitates an even transformation. While with the full vessel there is only limited space to expand the hull form. For this reason no crisp transformation limits can be given, although in practice the following guidelines have emerged:

- maximum change of block coefficient ± 0.05,
- maximum change of LCB from $L_{pp}/2$: $\pm 4\%$ of L_{pp},
- maximum change of C_m: ± 0.02.

It is useless to try to circumvent these limits by re-applying a transformation. For example, two transformations with a block coefficient increase of 0.05 yields the same as a single transformation with a 0.10 increase. These limits are, by the way, not a computer program limitation as such, instead they arise from the combination of hull form particulars and transformation method.

6.5.4.2 Parent hulls

Given a collection of parent forms, with the hullform transformation method a hull shape for a new design can be obtained within a couple of minutes. In order to stimulate this design method a library of about twenty parent hulls is available at SARC for general use. These hullforms, from which the majority was created at Delft University of Technology, can be obtained at `http://www.sarc.nl/fairway/parenthulls`.

6.5.5 General rotation and scaling

The hullform transformation methods of options [Transformation parameter menu] have arisen in the naval architectural tradition, and have a specific ship design background. Under the current option [General rotation and scaling] the general object transformation methods are collected. This option here are rather rudimentary, and entirely alphanumerical. Currently, work is ongoing on similar functionality in the GUI. For the transformations here apply:

- With function [Transform] the transformation will be executed.
- The transformation is performed on all selected solids (contrary to the conventional transformation as discussed earlier, which is only applied on the single selected solid).

This method is rather simple; for each of the three directions longitudinal, transverse and vertical a factor is given with which all coordinates will be multiplied. There is no fundamental difference between this option and the earlier [Linear scaling] (discussed on page 130), albeit the latter is more naval architecturally oriented, because there target values for length between perpendiculars, moulded breadth and draft are applied, while the current option works with multiplication factors (which are applied at each transformation).

This transformation is also available as [Scale Objects] (discussed on page 75), which has the advantage of being undoable, and applies to wireframes as well.

Here, must be given:

- An axis around which the object is rotated, which can be specified in two ways; either by specifying two points of the axis, or by the combination of one point and a direction.
- The rotation angle, clockwise (seen from the first point in the direction of the second, respectively in the direction of the axis) is positive.

This transformation is also available as [Rotate Objects] (discussed on page 75), which has the advantage of being undoable, and applies to wireframes as well.

6.6 Settings and auxiliary tools

When this option is selected the next submenu appears:

Settings and auxiliary modelling tools

1.	Fairway project settings
2.	Uniform weight factors
3.	Uniform mean deviations
4.	Verification of network and curves
5.	Make all curves consistent
6.	Remove all internal points from all curves
7.	Close vessel at deck

6.6.1 Fairway project settings

The project settings for Fairway can be entered in a property sheet which pops up when choosing this option. In the same sheet also GUI settings can be done; from the Graphical User Interface (GUI) (discussed on page 64) this sheet is accessible from the menu [File]→[Preferences...]→[Fairway Project Settings...]. The settings are ordered in several tabs, which are discussed below.

6.6.1.1 General Fairway settings

Time interval automatic save

This option lets you specify the approximate time interval (in minutes) between two automatic save actions of the program. With a time interval of zero the automatic save functionality is disabled. The user can control in which of the saved design variants the hull shape is saved automatically, see section 6.11.1 on page 155, File history.

Standard mean deviation

This is the mean deviation as used during the fairing of new curves. This value can be adjusted for individual curves, as described in paragraph 6.3.5.19.2 on page 89, Fair. See also the option [Uniform mean deviations] (discussed on the following page).

Naming convention cross sections

This is the naming convention used for new frames, and can also be set in [Systemize polycurve names] (discussed on page 85).

Design using a target SAC

Switching this option on enables the user of Fairway to design on the basis of a sectional area curve (S↩ AC), so that frames can be shaped towards a given volume and longitudinal center of buoyancy (LCB) (paragraph 6.3.5.19.8.1 on page 96, Show Target Frame Area). This target SAC can be generated (paragraph 6.3.5.21.2 on page 98, The use of Lap diagrams) and manually manipulated (section 6.3.5.21 on page 97, Change the shape of the SAC). Because Fairway also offers excellent tools for shape transformation (section 6.5 on page 129, Hullform transformation) by which the volume and LCB of an existing hull can be adjusted at any time, working with a target SAC is not necessary. Therefore this option can be left unchecked (the default) which simplifies the user interface of Fairway.

Enable wireframe modeling

Switching this option on enables functionality for working with wireframes and loose curves and points. This is a special feature of Fairway. Because the conversion of a wireframe into a solid can be challenging, we do not recommend novice and intermediate users to start a new design based on a wireframe. In any case, the user should be familiar with the contents of section 6.3.7 on page 113, Wireframes. Normally this option is off, to reduce the complexity of the user interface of Fairway. The option is turned on automatically after importing a wireframe from DXF or IGES.

6.6.1.2 With curved surfaces

The settings here are identical to the settings described in paragraph 6.3.5.8.3 on page 81, Settings.

6.6.1.3 Configuration GUI

This tab is only accessible when activated from the Graphical User Interface (GUI) menu [File]→[Preferences...]→[Common Settings...].

Maximum plotting inaccuracy
To draw a line on the screen the line has to be divided into very many tiny straight line pieces. Dividing a curved curve into more straight line pieces increases the draw accuracy, at the expense of processing time. With this option the maximum allowed deviation, in millimetres, between the mid of the line piece and the curved curve can be given.

Maximum angle of two adjacent linelets
Complementary to the previous option the maximum angle between two successive line pieces can be entered. The same rule can be applied here: decreasing this angle increases plotting accuracy and computation time.

The user preferences that control the appearance of the GUI are accessed from the GUI menu bar using options [File]→[Preferences...]. Changes in these preferences are persistent across sessions for each user individually. It is possible to manage different sets of preferences and to copy them to other installations or share them with co-workers, as explained in section 6.A.4 on page 162, Managing various sets of user preferences.

6.6.2 Uniform weight factors

Note

This function is also available from within the GUI as [Bulk Change of All Curves] (discussed on page 99).

This will give all points in the model a neutral weight factor. For a discussion of point weights see section 6.1.1 on page 56, Basics of Fairway.

6.6.3 Uniform mean deviations

Note

This function is also available from within the GUI as [Bulk Change of All Curves] (discussed on page 99).

While fairing a curve, see paragraph 6.3.5.19.2 on page 89, Fair, it is possible to adjust the mean deviation between the curve and its points. This option will reset the mean deviation for all curves in the model to the default value specified in section 6.6.1.1 on the previous page, General Fairway settings.

6.6.4 Verification of network and curves

This functionality has been moved to the graphical user interface, see section 6.3.6.1 on page 111, Check Solid.

6.6.5 Make all curves consistent

When this option is selected, new splines will be calculated for all curves (with a constant mean deviation of 0.↩ 0001 mm). This is done in such a way that the consistency between the spline representations of all curves and the points of the network is guaranteed. This can result in a model that deviates from the desired model. When dealing with a curves in a plane (for example a frame) all points and splines will be pressed in that plane. This function is also available from within the GUI as [Bulk Change of All Curves] (discussed on page 99).

6.6.6 Remove all internal points from all curves

With this option it is possible to remove all internal points of the curves in the network. The definition of internal and external points within Fairway has been given in section 6.1.1 on page 56, Basics of Fairway. For example, this option can be used before sending (converting) the hullform to the hydrodynamic program Dawson (MARIN). This function is also available from within the GUI as [Bulk Change of All Curves] (discussed on page 99).

6.6.7 Close vessel at deck

For some options it may be necessary to close the vessel at the top. This option connects the highest point of each frame with the centerline plane.

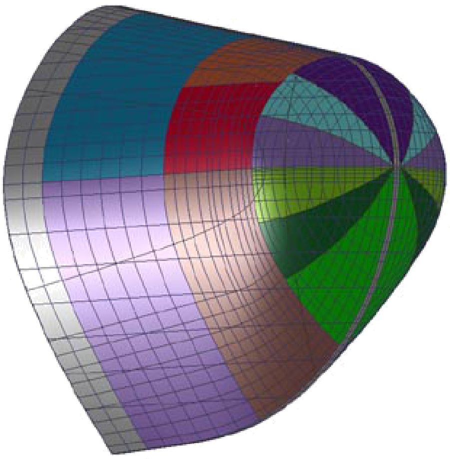
6.7 Show (rendered and colored) surfaces

This option is for the visualisation of the model. Sometimes it may be hard to interpret the model from viewing at the wireframe model, it might also happen that surfaces are defined in another way then the user had meant by line modelling. This option is of course also very valuable for making presentations or flyers for the company or the customer. Because of the interactive nature of this menu option, it is less thoroughly described than other menus in this manual.

Unlike the other main menu options, after hitting the <Escape> key the software does not return to the main menu, but generates a rendered image of the model. To return to the main menu without generating a rendered model, use the [Abort] function from the menu bar.

Drawing type

One can choose between 'Normal', which means the colors of the area under and above the waterlines have different colors, and 'Shell plate layout', which means every defined shell plate has a different color (see the figure below).



Rendering shell plate layout

Use "curved surfaces"

This option defines whether curved surfaces must be used that are derived from the shape of the curves, according to the settings in the right-most column of the menu [Object management] (discussed on page 155). If this is the case then the surface is triangulated during rendering, up to the detail level given in the setting below. Otherwise coarse triangles are generated between the intersection points in the network of curves.

Target dimension of planar elements

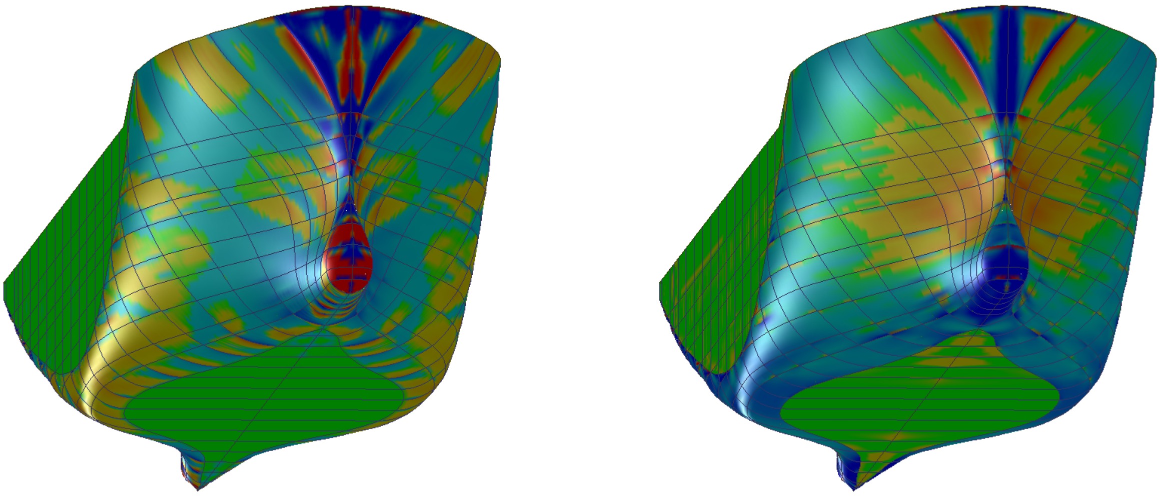
This value is an indication of the level of triangularization. A large value may cause the model to look a little rough, while a very small value may cause the computation to take a long time.

Representation of surface curvature

In the graphical user interface of Fairway, the curvature of a line can be visualised. However, a curved surface is curved in two directions. To visualise the curvature, the curvature in both directions have to be combined to a single curvature parameter. In general, the curvature in the direction of the largest curvature (K1) and the curvature in the direction of the smallest curvature (K2) are combined in one of the following ways:

- Gausse curvature = K1 x K2
- Mean curvature = (K1 + K2) / 2
- Absolute curvature = abs(K1) + abs(K2)

Fairway can determine these curvatures, and present them by means of a color distribution. The examples below show the Gaussian and Mean curvature:



Gaussian curvature (left) and mean curvature (right)

6.7.1 Option in the render window

The window with the rendered view contains in the menu bar a number of functies for setup, printing and export (e.g. to bitmap or VRML format) which are discussed in section 7.8 on page 189, Rendered views.

6.8 Export of hullform

With this option the hullform can be converted and exported to a format suitable for use in other software. The most commonly used formats are included in the menu that pops up first, the more exotic varieties are accessible in a sub menu via the last menu option.

***How** the conversion is done will be discussed below, and **what** will be converted depends on the setting per solid in the menu [Object management]* (discussed on page 155), as follows:

- Aimed at the PIAS calculation modules (which apply PIAS' frame model and occasionally the triangulated surface model) the solids and wireframes will be exported which are selected in the PIAS column.
- For NURBS and STL export those solids will be used that are selected in the Export column. It is foreseen that in due time **all** export is governed by this column, but that is not yet implemented everywhere.
- On export to DXF the user can choose whether only the 'single selected solid' will be exported, or all vor Export selected solids. This choice will disappear one of these days.
- With the disused options only the 'single selected' solid is exported (what used to be the standard in times past).

Export of hullform

1.	Convert this Fairway model to PIAS model
2.	Offsets to ASCII-file
3.	All lines to AutoCAD DXF format in three 2D views
4.	All lines in 3D to AutoCAD DXF-polyline format
5.	All lines in 3D to AutoCAD DXF-NURBS format
6.	All lines as NURBS to IGES
7.	All faces to IGES NURBS patches
8.	Stereolithography file (.STL) for CFD or 3D printing
9.	Enable hullform to be used as Hull Server shape data base
10.	Disused export formats

Attention

It is recommended to read section 6.8.11 on page 145, On production fairing before using any of the options from this menu.

6.8.1 Convert this Fairway model to PIAS model

From each object which is selected for PIAS in the menu [Object management] (discussed on page 155) all frames are converted to PIAS' frame model — please refer to section 2.10.2 on page 16, Hull form representations for a discussion of the different hull shape models. The converted hullform can be used for all functions and calculations in PIAS. At this conversion, the following mechanisms are applicable:

- In the solid properties menu, three columns are important for this conversion. The column PIAS indicates that an object is converted, the column 'main hull' indicates whether the object is part of the main hull form. If that is not the case then the column 'buoyant' at 'yes' indicates that it is an added form, and at 'no' that it is an extra body. All of this in accordance with the categories in Hulldef, see section 7.2.2 on page 172, Hullforms and section 7.2.3 on page 173, Extra bodies.
- More than one object can be classified as 'main hullrsquo;. In this case, those are sorted in longitudinal direction, and forwarded to PIAS one after the other. This feature is strictly intended for a hull shape consisting of several independent objects that do not overlap in a longitudinally — such as stern - nave - foreship. If objects nevertheless overlap, Fairway will produce a wrning on that, however, it is up to the user to correct.
- A PIAS frame model must meet a number of conditions, such as those listed in section 7.2.4 on page 173, Frames (frame positions and frame shapes). It is the responsibility of the user to observe these rules, Fairway does not check for them.
- To pias' frame model, those lines (=polycurves) are converted which are classified as 'frames' in Fairway. So, it is not sufficient for a line *de facto* to be a frame — because its coordinates share their longitudinal positions — it actually needs explicitly to be from this type.
- Once upon a time **all** the frames present in a Fairway solid were converted to frames model. Occasionally, that proved to be somewhat inconvenient with frames which did not extend over the entire hull body. Such as an auxiliary 'frame' which only did exist in the bilge area, or a tiny 'frame' just existing far above the waterline. For PIAS is based on longitudinal integration of ordinate areas, and as such tiny 'frames' have little to no area, they induce a fault in the correct sequence of ordinate areas, and to correspondingly incorrect volumes and stability results. But that's all past; frames are only converted to PIAS only if they extend over the entire hull surface. This 'entire hull surface' is interpreted as the surface which is bounded at the edges by a phantom face (see section 6.1.3.1 on page 62, Phantom face for an introduction to this concept). It is good to be aquainted with this background, for if Fairway frames are missing in the PIAS model, then it is advisable to check whether in Fairway these are at the bottom and the top indeed connected to a phantom face (which will by default be the case).
- Parts of frames adjacent to a phantom face are not being converted to PIAS. In order to avoid that lines over CL or at the ship's extremes along the deck are included in PIAS' frame model.
- If longitudinal polycurves, such as waterlines or 3D lines, are present which have the property 'deck at side' then the PIAS frames will be cut of at the level of this line or these lines. Please also refer to section 6.1.3.5 on page 62, Deck at side.
- It is advised to verify the resulting frame model thoroughly with Hulldef.

If you use the *local cloud* (for which reference is made to section 2.11 on page 17, Local cloud: simultaneous multi-module operation on the same project), the frames do not neccessarily have to be converted to a *file*, because at any moment the PIAS equivalent of the Fairway hull form is available through the *cloud*.

If in the solid properties menu it was specified that a solid should also be converted to PIAS' triangulated surface model, then such a file is also created with this menu option.

6.8.2 Offsets to ASCII-file

With this option an ASCII file (∗.off) is generated, which contains the coordinates of all lines of the hull form. After selecting this option the number of desired decimals is asked. This file contains all solids which are selected for export, and has the following structure:

- Name of the line.
- Number of points on the line.

- Longitudinal, transverse and vertical coordinates of every point (relative to to the intersection point of the aft perpendicular and the base line) in metres.
- Whether a point is a knuckle or not.
- Whether a point is an intersection or not. When the point is an intersection point, the intersecting line is given as well.

6.8.3 All lines to AutoCAD DXF format in three 2D views

This option produces three ∗.dxf files. Each file contains one view (front, side and top view). For each view a translation can be given, which allows to fix the layout of the drawing to some extent.

Warning

> On one hand this option offers the possibility to prepare simple layout operations, on the other hand the possibilities are way too limited to generate an attractive lines plan immediately. The purpose of this option is to help the user **to some extent** in composing a drawings, because that task is sometimes rather laborious with general CAD systems. This option is not intended to compose a detailed linesplan **here**, the Fairway 'linesplan' facility (see section 6.9 on page 145, Define and generate lines plan) is much more suitable for that job, especially because such a lines plan can also be exported to a 2D DXF file (as discussed in section 5.1.10 on page 44, Output filetype).

6.8.4 All lines in 3D to AutoCAD DXF-polyline format

With this option it is possible to export the 3D network to AutoCAD. After selecting the option you can enter whether you want to convert the waterlines, frames and buttocks to real 3D lines or to 2D lines. Both options result in the same 'picture' in AutoCAD. But there is a difference. When you select the option '3D lines', the lines are real 3D lines.

When 2D lines are selected, the model in AutoCAD will for the most part be composed of 2D lines, while the plane in which the lines lie will be rotated in such a manner that a 3D view appears. The difference will become clear when using editing modes in AutoCAD. For example, when using the AutoCAD command [Offset]. This command cannot easily be applied to real 3D lines.

After selecting the line type, you are asked to enter the maximum length of a line segment. In AutoCAD every curve is represented by a so-called 'polyline' that is composed of small line segments. The lower the segment length the finer and more accurate the curve representation.

At last you are asked to enter if you want the (non-internal) network points to be exported. When entering [Yes], points will also be generated at curve intersections in the AutoCAD model.

The result is a .dxf file, which contains a 'polyline' of every line. Fairway tries its best to generate 'polylines composed of circle segments', which allow the curvature of the curve to be approximated. Unfortunately, the DXF format and AutoCAD itself cannot handle 3D polylines containing circle segments (i.e. lines of the Fairway type 'Spatial polycurves'). Therefore, 3D curves will be composed of straight line segments. As a consequence, 'polylines composed of circle segments' can only be exported when the above-mentioned option '2D lines' is used. This shows that the 2D history of AutoCAD is still shining through.

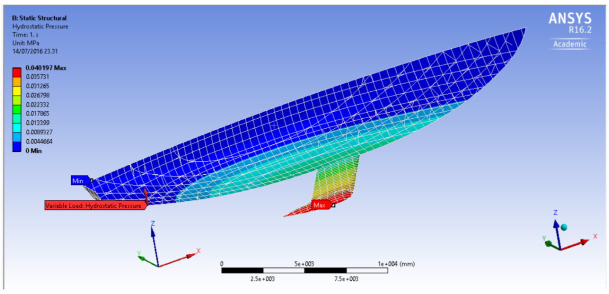
6.8.5 All lines in 3D to AutoCAD DXF-NURBS format

This option generates a DXF-file that contains the Fairway lines in the AutoCAD spline format (DXF group code 100). Actually, this is a 3D NURBS curve, which is the same type as Fairway is using. So this DXF option is not using an approximation like the two mentioned above, but translates the form coefficients of Fairway to DXF, so that with a minimum of information transfer a maximum precision is attained.

6.8.6 All lines as NURBS to IGES

IGES is an abbreviation of Initial Graphics Exchange Specifications, and contains international agreements regarding the file format which is used for the exchange of information between different CAD systems. Fairway uses the IGES type *126; Rational B-spline* for this conversion, to a file with extension .igs.

6.8.7 All faces to IGES NURBS patches



Faces of Fairway, through IGES imported in Ansys.

With this option an IGES file (∗.igs) will be created, which contains the facets — so, a surface representation — of the hull shape. In IGES, (curved) facets can only be quadrilateral, in Fairway, they can also be polygonal (eg. five- or six-sided). For that reason, the Fairway facets may be subdivided into multiple, smaller ones. One must realize that the conversion of Fairway to IGES has a finite accuracy, for the following reasons:

- In IGES, many types of representations can be used, but that of Fairway is not among them. This requires **conversion** which commonly leads to loss of accuracy.
- The most common representation is the NURBS patch. If two such patches are adjacent to each other, and have an unequal number of *vertices* along their common edge, then there will generally be a gap between the two edges. The gap might be very, very small, but when heavily zoomed in it still can be visible. This has nothing to do with Fairway, this is an intrinsic property of the NURBS.

It can be concluded that IGES and/or NURBS are not capable the represent the Fairway hull shape precisely. However, often there is no other choice, one will have to be practical. As said, IGES accommodates many representations, for the facets Fairway support two of them:

- IGES number 114, the ‘Parametric Spline Surface’.
- IGES number 128, the ‘Rational B-Spline Surface (NURBS)’. This type is the most common, and is recommended.

After choosing the conversion of facets to IGES, the user can choose from two types viz. the most recent, from 2014, and the original, from 1998. It goes without saying that the recent is recommended, the one from 1998 is only present for backward compatibility.

6.8.7.1 IGES NURBS regions (2018)

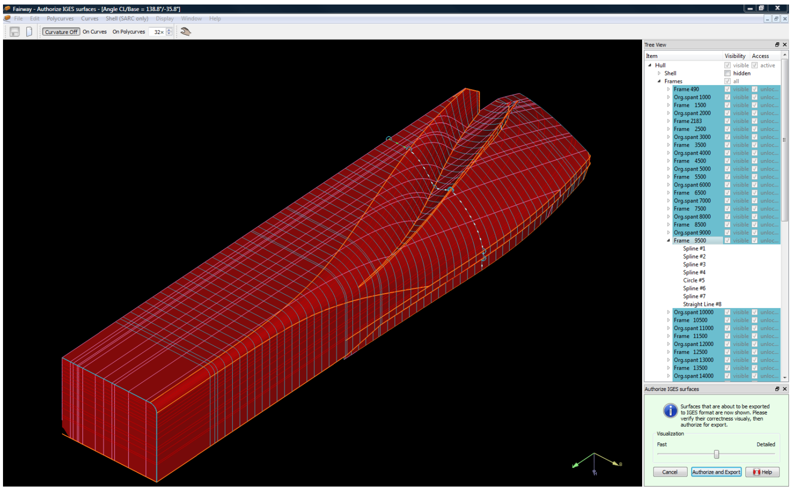
This option exports defined shell regions that have the "IGES export patch" option, see [IGES export patch] (discussed on page 107)

6.8.7.2 IGES faces (2014)

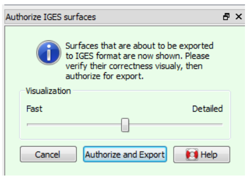
This option creates a parametric or NURBS surface with adjustable accuracy (and corresponding file size). In order of increasing accuracy the options are:

- Without curved surfaces. The IGES facet is only based on the form of the vertices, and therefore, a rough approximation.
- With curved surfaces, with the most simple facets. The facets that are used here are indeed curved, but at the same time of the simplest shape, in case of a NURBS of 16 (4x4) vertices.
- With curved surfaces, with adequate accuracy. Here the number of coefficients or vertices are determined in a way than the mean deviation will be less than 0.1 mm. Please note that this is the **average**, for individual points the difference between Fairway shape and IGES facet may be more (but that will be compensated by other items where the difference is less).
- With curved surfaces, with high accuracy. Here the number of coefficients or vertices are determined determined such that the mean deviation will be less than 0.01 mm.

Regardless of what accuracy is used, after conversion a window will appear with the converted hull shape. Here, the facets are displayed as they will be written in the IGES file. The user is expected to approve this first, and then click the ‘Authorize and export’ function. Bottom right also a slider is included, which allows the visualization accuracy to be set. This only applies to the drawing accuracy of this window, it has **no effect whatsoever on the contents of the IGES file** .



Preview on the IGES file.



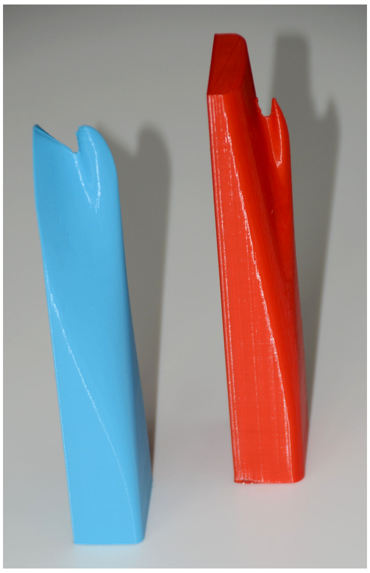
User authorization for export.

6.8.7.3 IGES faces with raw shape (1998)

This option originates from the late ’90s and creates an IGES file which represents the Fairway hull shape approximatively. Here a number of choices have to be made:

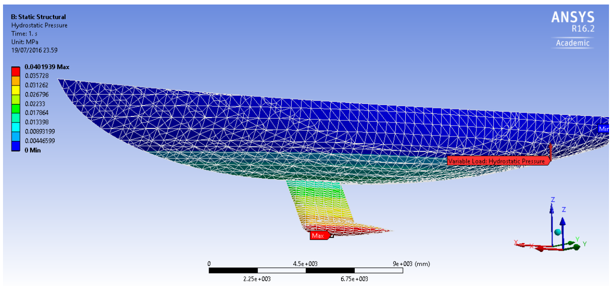
- The type of IGES surface, 114 or 128, as discussed above.
- Use the ‘curved surfaces’ capability, at least, when this option is available. In general if use is made of this option a smoother surface will be obtained. By the way, if the ’curved surfaces’ option is not used the system will have to remove internal points. These points will be deleted permanently, so it is to be recommended not to save the hull model on disc anymore.
- Quadruple the number of IGES patches per face. Without quadrupling, in general one IGES surface per face in Fairway is generated. The alternative is four surfaces per face, which may give a better fit between neighbouring IGES surfaces. This option will produce a less ‘raw’ IGES file, however, it is a) a rough remedy, because the number of patches is bluntly quadrupled, and b) not a panacae because quadrupling might still not be sufficient for an accurate result.
- With flattened corners. This option is rather specific; by nature the surfaces will be slightly twisted in the vicinity of their corners, but with this option this twist is artificially flattened. In mathematical parlance this is called *zero-twist*. The practical relevance is that with hull models which fit or fair badly, the common method may lead to wild fluctuations in the IGES surface. In these cases ‘flattened corners’ could be chosen.
- Only faces which are bordered by visible lines. In this case only faces which are bounded by visible lines at all sides will be converted.

6.8.8 Stereolithography file (.STL) for CFD or 3D printing



Model designed with Fairway, produced on an Ultimaker 3D printer.

With this option, a `.STL file`[2] can be produced, which is suitable to produce a scale model of the ship hull with a milling machine or a 3D printer. On the latter has been reported in the `SWZ maritime`[3] journal, please refer to `www.sarc.nl/images/publications/appendix_swz2012.pdf` for further reading. Because STL might also be used convert the ship shape to CFD software (for instance to `v-Shallo`[4]), for that purpose provisions are available in this conversion.



Fairway model, through STL imported in Ansys.

Before generating the STL file, a menu appears where the parameters can be given as discussed below. If this menu is quitted in the regular way then the STL file wil be generated, with [Abort] the conversion will stop without making the STL file.

Purpose

The purpose for which the STL file is produced, choice of '3D printing of a model' and 'CFD (Computational Fluid Dynamics)'. Differences between the two are:

- For 3D printing the STL file must contain a closed *solid model*, for CFD that is not required. This implies that for printing, the hull will be closed at deck level (if not already the case), for CFD not. Similarly, when printing a demi model the CL plane will be closed, for CFD that is not done.
- For printing the STL file is in millimeters, for CFD in meters.
- For printing a scale can be specified, for CFD not; a CFD model is always actual size.
- For printing a ship can be segmented (subdivided), for CFD that would be useless.

[2] https://en.wikipedia.org/wiki/STL_(file_format)
[3] http://www.swzonline.nl/news/2708/3d-printing-ship-models-extras
[4] http://www.hsva.de/our-services/software/v-shallo.html

Format for output file(s)
Choice between the ASCII or the binary variant of STL. Also the TLOM (*Thick Layered Object Manufacturing*) is available, however, this is experimental and not generally available.

Desired maximum triangle size
Essentially, the STL file format describes a long list of triangles. Here the desired maximum size of each triangle can be entered. It will be obvious that with a smaller triangle size the model will become more accurate, and the number of triangles (and thus the STL file size) will be larger. The triangle size specified here refers to the actual size vessel, and is therefore independant from the model scale. In the subdivsion proces from the surface into small triangles the 'curved surfaces' functionality of Fairway is applied, so it only works if that option has been purchased. Concerning the triangle size there are still a few differences between STL for 3D printing and for CFD:

- If a triangle is already fully flat, for 3D printing is not further subdivided, that would be useless. For CFD a different consideration applies, there it will be divided further.
- For CFD it may be useful to define areas of a coarser or finer triangle distributions. This can be assigned to a *shell region*, please see paragraph 6.3.5.24.6 on page 107, STL export specifics. There it can also be set that a particular region is fully omitted from conversion to STL.

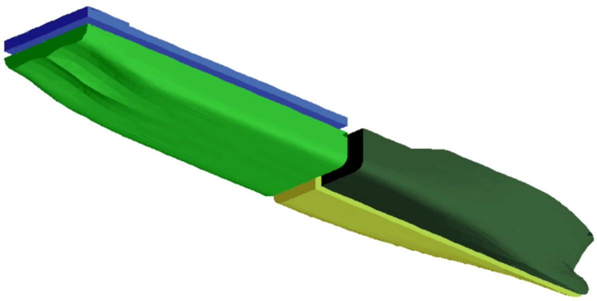
Ship side
If the purpose is CFD, this configures what sides of the ship should be exported.

Model scale
Define the reciprocal of the model scale, so X in scale 1/X.

Subdivide the ship
Here the choice is offered between:

- 'Yes, automatically'. In order to take into account the physical limitations of the manufacturing device, ship models are subdivided into producable blocks, see the example in the figure below. This option does that automatically. However, this is still an experimental algorithm, so this feature has not been released for general use.
- 'No, make demi model', which will not subdivide the ship, but produce two separate halves, SB ans PS.
- 'No, make full model', which will also not subdivide. The STL file will simply contain the entire ship, SB and PS (that is, if the solids have such properties).

Manufacturing technology and maximum segment dimensions
These parameters are not discussed further since they are only of interest to 'automatic subdivision', but that feature is not generally available.

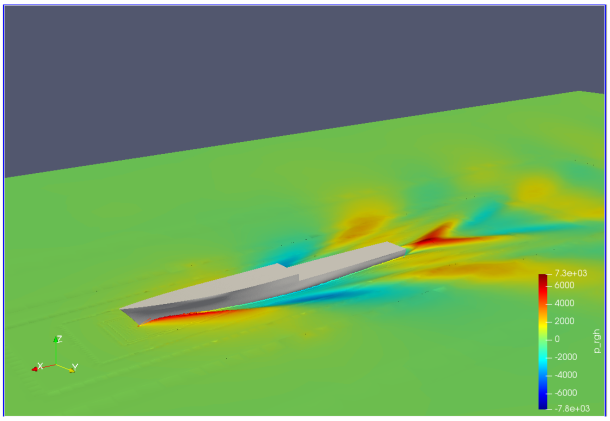


Ship model segmented for 3-axis milling.

Attention

- De specified triangle size is the **maximum** size, not the desired size. In other words, faces are being subdivided into smaller triangles, but not merged into larger ones. That would be impossible, because if two triangles are being merged, than usually a quadrangle is created, which does not fit into STL.

- All solids flagged for 'export' in the menu [Object management] (discussed on page 155) will be converted to STL. In this respect, one should be aware that the conversion is per individual solid, so the solids will not be merged together in the STL file. If that would be desirable, external STL tools should be used for that purpose.
- Each phantom face (see section 6.1.3.1 on page 62, Phantom face) is omitted from the conversion to S↩ TL. Furthermore, it is for the manufacturing of a demi model assumed that the ship is modelled in the conventional Fairway way. I.e. with a phantom face on CL.
- A solid of the type 'complete' (so, one which extends over portside and starboard) is not automatically split in two when making a demi model. This must be done manually (with [Centerplane]→[Split] from the solid menu).



Fairway hullform exported to STL, imported in and computed with OpenFOAM.

6.8.9 Enable hullform to be used as Hull Server shape data base

The background of the *Hullserver* is that CAD/CAM software (the client) can communicate directly with Fairway, without the use of files. This imnplies that the CAD system and Fairway are running simultaneously on the same computer, with the CAD system repeatedly requesting shape data from the ship model. The Hull Server determines the requested shapes and returns them to the CAD system. This communication is dynamic and is not limited to the set of lines as available in Fairway. For instance, if a client asks for a specific line which does not yet exist in the model, the Hull Server will simply generate that new line and returns its coordinates to the client. Detailled information on the Hull Server is available in a separate document (fwserver.pdf) which is available upon request at SARC. Hullserver clients are available in NUPAS and Mastership engineering software.

This conversion option here in the Fairway export menu option is nothing more but a test which indicates whether the model is sufficiently defined for its use in a production environment. If this is the case, Fairway will mark the model accordingly, so it is fit for application in the Hull Server environment. This check verifies (to some extent) that only proper data are being transferred between both programs. Please note that the model may still contain errors, for example a line may inadvertently be designated as a knuckle (or not), the shape itself may not be satisfactory, etc. This type of errors are caused by (poor) modeling, not by the (interfacing of) software.

6.8.10 Disused export formats

Disused export formats

6.8.10.1 All lines to NUPAS import format

Attention

> In June 2006 it is agreed with NUPAS' manufacturer NCG that this file interface to NUPAS will be depreciated, because Fairway' *hull server* mechanism, where Fairway acts as shape database for NUPAS, offers a much more secure and complete conversion. Technically, it is still possible to generate a NUPAS interface file, but it is no longer supported.

If you apply this option, five files (with extension .pnu) are generated, which can be used as 3D lines in early versions of NUPAS. By the way, with option option Enable hullform to be used as Hull Server shape data base (discussed on the preceding page) a Fairway hull is prepared to be usable for the *hull server*.

6.8.10.2 All lines to Eagle format

A file (∗.eag) is generated that can be used in Eagle. After selection of this option one is asked to enter the number of coordinates placed at a line in the ASCII-file. Furthermore, the maximum number of points of a line can be entered. When using the <Enter> key the number is not restricted.

6.8.10.3 Relevant lines to Tribon (Stearbear) format

A file (∗.stb) is generated, which can be used in Tribon (which was called Stearbear, in days long past).

6.8.10.4 Relevant lines to Schiffko format

With this option two files are generated (QS001.DQS and LL0001.DLL) which can be used in Schiffko, are generated. Two methods of conversion are possible: by the network or by the exact frames.

6.8.10.5 Create finite element model

With this option a ASCII-file, with extension .fem, is generated. This file contains two parts. The first part is a list of all faces with numbers for each point of the faces. These numbers refer to a list of coordinates at the end of the file, where this number corresponds with the coordinates of this point. These data can be used in a finite elements program. It should be noticed that this option exports the faces 'accidentally' at hand. Optimization of face dimensions and face location does not occur. In other words, it is not a mesh generator.

6.8.10.6 Create Dawson-model (MARIN)

With this option the hull form is converted to a ∗.pnl file, in a format suitable for the potential flow CFD program, as developed by MARIN in the '90s. Before generating a Dawson model, the 'internal' network points on the lines have to be removed. The definition of 'internal' network points has been given in section 6.1.1 on page 56, Basics of Fairway. When removing the 'internal' network points, the hullform is not changed. If you want to keep the original model, you have to make a backup. When creating a DAWSON model, you are asked whether all 'internal' network points have to be removed automatically. When entering <No>, no Dawson model is created. When entering <Yes>, the following appears: 'For Dawson the hullform upper boundary must be a (possibly trimmed) waterline. Give the waterline level or the name of that waterline'. Only the hullform under the waterline is important in the Dawson program. Existing waterlines can be entered by giving the height of this line in metres or the name of the line. It is also possible that you want to export the hullform with a certain trim. Then you can define an angled waterline and enter this with its name. An angled waterline can be defined as described in section 6.3.5.8 on page 80, New Planar Polycurve by Intersection.

6.8.10.7 Frames to Poseidon (DNV•GL)

Poseidon is a scantling-program of DNV•GL. With this option, all defined frames in Fairway are written to a Poseidon readable format. This export option is outdated, because it only concerns the hull form. With Layout both hull form and internal geometry can be converted to Poseidon, see be found in section 9.11.6 on page 243, Export to Poseidon (DNV•GL) for more information.

6.8.10.8 Frames to Castor (ASC)

Castor is a steelweight-estimation program developed by ASC. With this option, all defined frames in Fairway are written to a Castor readable format.

6.8.10.9 Relevant lines to ShipConstructor

This option, from about 2005, exists and produces files which sould be consumable by ShipConstructor. However, SARC has no knowledge of any test with these files in that program.

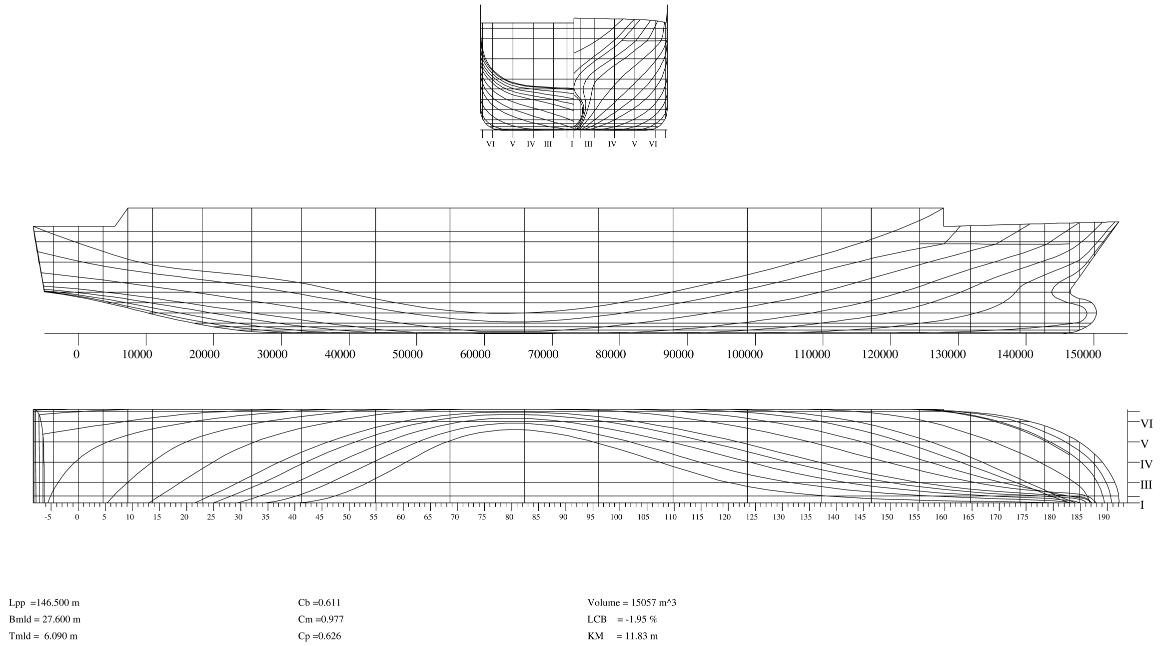
6.8.11 On production fairing

In thi schapter the export facilities of Fairway have been discussed. The results can and will be applied in other (CAD and CAE) systems, often also for production purposes. In the latter case the Fairway hull form requires a level of consistency and accuracy which is sufficiently high for production. Fairway offers the user tools to fair for production. However, the actual fairness is dependent on the assessment by the user and the time that has been invested into the fairing process. The use of Fairway as a sole fact is by no means a guarantee that a hull shape is actually fair for production.

6.9 Define and generate lines plan

With Fairway it is possible to generate a lines plan from the threedimensional shape. The sections to draw, and their locations are configurable, and additional texts and size markers can be generated. An example of such a linesplan is depicted below. By the way, the mechanism behind this lines plan generation is quite similar to that for the subdivision plan from Layout. Details will differ, however, for a better understanding the subdivision pages might be visited on *section 9.9 on page 238, Subdivision plan.* *Although in this manual the line plan mechanisms will be discussed thoroughly, experience shows that a few experiments may also be instructive. And please be assured that although with an 'erroneous' lines plan definition the result may be unexpected, it is not possible to ruin the underlying ship hull model.*

If a hull consists of multiple solids, it may be desirable to include some in the lines plan, and to leave out others. That can be specified in the menu [Object management] (discussed on page 155).



Example of linesplan, as generated by Fairway.

Define and generate lines plan

1.	Definition of the layout of the lines plan
2.	Enter drawing head texts
3.	Draw and extent linesplan, on screen
4.	Draw selected linesplan on paper

6.9.1 Definition of the layout of the lines plan

After selection this option an input window appears where a maximum of four linesplans can be composed, and where one of those can be selected for actual output. The purpose of the columns in this menu is deiscussed in the table just below. Furthermore, when you move the text cursor to a drawing and press <Enter> the menu [Views of [linesplan name]] (discussed on this page) appears, where the views of the drawing can be defined.

Slct Here you can select the lines plan to be actually generated.

Description
Just a simple textual description, just for your own orientation.

Margin
Additional margin at the circumference of the paper, in millimetres. The default margin is 10 mm.

Frame
Whether or not a framework should be drawn.

Drawing head
Whether or not to include a drawing head. The sheet border and drawing head are not displayed when drawing on screen. The drawing head text can be adapted in section 6.9.2 on page 150, Enter drawing head texts. .

Coefficient
Whether or not the most important hydrostatic coefficients (such as the block coeffcient) should be niclude in the lines plan.

6.9.1.1 Views of [linesplan name]

In this menu the views (with a maximum of 16) of this lines plan can be specified. With <Enter> in the first column one goes one level deeper into each view - where the viewing parameters are given into detail as discussed in the paragraphs below - however, first the main properties of the views have to be defined in this menu. The purpose of the columns here is:

Description
A description of this view.

Active for text
Texts can be added to a lines plan, this is discussed in paragraph 6.9.1.1.1.6 on page 150, Define additional legends and in section 6.9.3 on page 150, Draw and extent linesplan, on screen. These text belong to the view which is 'active for text', as defined in this column. This implies that if a view is removed, then the belonging texts are removed as well.

View
The type of view can be chosen here, avialable types are:

In standard directions
These are conventional views, on the planes of waterlines, frames, buttocks or diagonals (under 45°) The applicable parameters will be discussed in paragraph 6.9.1.1.1 on this page, Definition of views in standard directions.

3-Dimensional
Threedimensional view on hullform, under a user-defined viewing angle. For teh parameters please refer to paragraph 6.9.1.1.1.2 on the next page, Definition of threedimensional views.

Plate layout
A three-dimensional view of the shiphull. The view shows the plate borders of the selected plates with addition of the name of the plate. The parameters are identical to thos of the threedimensional view, just above

Only SAC
The Sectional Area Curve (only of the single solid as used for this lines plan, so the contribution of possible other buoyant solids is not included). See paragraph 6.9.1.1.1.3 on the following page, Definition of a view on the Sectional Area Curve for details.

6.9.1.1.1 Definition of views in standard directions

With this option an intermediate menu opens up, with the following options:

View: [description]

1.	Define view
2.	Properties first axis
3.	Properties second axis
4.	Define additional legends

6.9.1.1.1.1 Definition of views in the standard directions

You can specify which view should be drawn, here it must be placed and how it is formatted. An important concept in this respect is the 'selection box'. That is a thee dimensional box with user-defined boundaries, with the meaning that all hull lines inside that box will be inlcluded in this particular view, and the remainder (consequently) not. If one does want to include for example only the aft ship in a view of the lines plan, then for that purpose a box with aft limit -∞ to forward limit L/2 can be taken (and because in practice ∞ is quite large, e.g. 1000 can be used instead).

Longitudinal lower limit selection box [m]
The longitudinal location, measured from APP, *from which* the lines should be included in this view.

Transverse and vertical lower limit selection box [m]
Similar for the other two dimensions.

Longitudinal upper limit selection box [m]
The longitudinal location *to where* the lines should be included in this view.

Transverse and vertical upper limit selection box [m]
Similar for the other two dimensions.

Transverse shift on paper [m]
Here you can specify, in real world coordinates, the amount of transverse shift on paper or on screen. A positive shift is to the right, a negative to the left. The purpose is to be able to position the different views relatively to each other The shift of the view on paper or screen in transverse direction. A positive value

means a shift to the right and a negative value means a shift to the left. The purpose is to position the different views relatively to each other. The program will always ensure that all views are visible. So this shift is not connected to issues of scale or dimsonsions of screen or paper. Because the shifts are relative to each other, there is no firmly defined zeroshift; simply take an arbitrary view with zero shift, and give the shifts of the other views relative from that one.

Vertical shift on paper [m]
Similar to the transverse shift. Positive is a shift upwards.

Mirror about CL
The view is mirrored about the center line plane. For example to be used for aftship frames in the body plan.

Draw only visible lines
In the Graphical User Interface (GUI) it is possible to hide individual polycurves. When selecting 'yes' for this option, only the visible polycurves will be drawn in the lines plan. With 'no', all visible and all invisible curves will be drawn.

Type of measurements
With the option [Define and generate lines plan,fwy_linesplan_screen] lines can be measured - which means that automatically legends can be allocated to lines, where the location of the line determines the value of the legend. With the present option you can choose the naming system, with choices between 'frame number', 'ordinate number', 'name of line', 'number in arabic', 'number in roman', 'automatic meter', 'automatic millimeter'. Their properties will be discussed at 'Types and legends of size markers' of paragraph 6.9.1.1.1.4 on the current page, Properties first axis.

Font height of measurements
The font height of the measurements, in millimetres.

Projection plane
here the type of view can be spcified, with the choice from frames (a front view), waterlines (top view), buttocks (side view) or diagonals (under 45°).

6.9.1.1.1.2 Definition of threedimensional views

Defining threedimensional views is quite similar to defining a view in the standard directions, as discussed just above. The differences are:

- The option 'projection surface parallel to...' is not available.
- There are two additional cells; the 'angle between viewing axis and CL' and 'angle between viewing axis and base line'. These are the angles, in degrees, under which the vessel is viewed, please refer to section 2.6 on page 9, Definitions and units for the definitions.
- There is an additional cell labelled 'perspective', which indicates whether this is a perspective view. If this is set to 'yes' then four additional cells will appear with perspective parameters, amongst which the 'distance from the eye to the objectpoint'. That object point, which is the point where the eye is looking at, is defined by means of its three coordinates in the cells just below.

6.9.1.1.1.3 Definition of a view on the Sectional Area Curve

With this choice the actual Sectional Area Curve, that is the SAC of the active solid, will be plotted. Such as SAC is only possible in a private drawing, so not in combination with other hull views. However, it still is possible to define additional legends (consisting of fixed texts) for the SAC, this will be discussed in paragraph 6.9.1.1.1.6 on page 150, Define additional legends.

6.9.1.1.1.4 Properties first axis

In each view two axes can be defined, which are in general used as x-axis or y-axis. With this option the orientation and system of measurements of the first axis can be given. The axis method has two important properties:

- An axis is essentially three dimensional, and is defined by means of its start and end points, so by six figures in total. In the lines plan views the axis will be projected in the same manner as the hull lines.
- Legends with the axis are placed at the *right side* of the axis, seens from startg point to end point.

Draw this axis
'Yes' if the axis should actually be included in the plot, 'no' if not.

Longitudinal coordinate start axis
The distance from aft perpendicular to start point of the axis, in longitudinal direction, in meters.

Transverse coordinate start axis
The distance from centerline to start point of the axis, in transverse direction, in meters.

Vertical coordinate start axis
The distance from baseline to start point of the axis, in vertical direction, in meters.

Coordinates end axis
Similar to the previous three coordinates, for the end point of the axis.

Types and legends of size markers
With this option it is possible to determine whether size markers should be placed and to which they refer. As noticed earlier, the size markers are always drawn at the rightmost side, seen from start to end, of the axis. Nine types of size markers exist:

Automatic meter
Measurement of the axis in meters. The size markers and texts are automatically generated by the program

Automatisch millimeter
As just above, albeit in millimeters.

Frame number
With frame number. These whould have been specified in Hulldef, as discussed in section 7.2.1.3 on page 167, Frame spacings.

Ordinate number
Measurement of the axis with ordinates. Ordinate 0 is the aft perpendicular and ordinate 20 is the fore perpendicular. Obviously, the length between the perpendiculars has to be defined properly for this purpose, see section 6.4 on page 128, Define main dimensions and other ship parameters for that.

Name of line
The name of the line - simple the name as defined by the user - is used with this choice.

Ordinal numbering arabic
Numbers is our common western numbering system (1,2,3,4....). The numbers are the ordinal numbers of the actually drawn lines. From 'aft to front', 'bottom to top' and 'inside to outside' the numbers are increasing.

Numbering in Roman
Just as above, albeit in the Roman system (I, II, III, IV....).

Nothing
No size markers or legends.

Only size markers
Only size ticks will be drawn, without legends.

Line type for numbering
If the legend type is either 'name of line', 'number arabic' or 'number roman', then the type of line to follow has to be entered. Possible line types are frames, waterlines and buttocks. If the legend type is 'number arabic' or 'number roman' then a fourth choice is present, which is 'fixed increment, given on line below'. This type can be used if size ticks are required on fixed distances (for example every 5 meter), while these tick should subsequently be numbered.

Fixed value for numbering
If the line type for numbering as given the line above is 'fixed increment' then that increment, in meters, can be given here.

Numbering about how many frames
If the legend type is 'frame number', then here can be specified about how many frames should be numbered. With 1 each frame is numbered, with 5 each fifth frame.

Dimension size marker
The desired height of the size markers in millimetres.

Angle text legend
The angle of the text legends (which are the figures or number printed at a size marker) with the axis. With 0 the text is parallel to the axis, with 90 the text is rotated 90° counterclockwise.

Size of text legend
The font height of the text legend, in millimeters.

6.9.1.1.1.5 Properties second axis

Completely similar to the first axis.

6.9.1.1.1.6 Define additional legends

Here additional, fixed, texts van be specified which will be included in the lines plan. The 'breadth' and 'height' coordinates are measured from the origin of the view which is 'active for texts', in meter, on real-world scale, in the same logic as the mutual shift of the different views is given. The dimension of the text is the font height, in millimeters. The texts which are generated with the interactive measurement, as discussed in section 6.9.3 on this page, Draw and extent linesplan, on screen, will also be included in this list.

6.9.2 Enter drawing head texts

Here the text lines can be given for the drawing head, as it can be included in a lines plan. The name of the ship or project, the date and the used scale are always included in the drawing head.

6.9.3 Draw and extent linesplan, on screen

With this option a window pops up with a preview of the lines plan. However, there is an additional purpose of this window, which is dynamically add measurements of lines in the plane which is viewed perpendicular in a certain view. The issue is that measurements alongside axes can easily be automated, because they are written in empty space on paper. However, right *within* a drawing puttingthe texts properly is much more cumbersome, because it is there not only the nitention to putthe text near the intended line, but also just **not near other lines**. And for it is still desired to measure such lines, for instance numbering frames in the body plan, it can be done interactively. The prime idea is that the user clicks a line, which make the measurement legend to be attached to the cursor, while the user puts this text on the appropriate location. In more detail, these functions are:

Zoom in
: Assigns the 'zoom' function to the left mouse button. If this is pressed once a zoom rectangle is created which can be used to indicate the zoom area. With a second mouseclick this zoom is executed.

Zoom back
: Reverts to the previous zoom level.

Measure line
: Contains the core of the measurement system. Also this fnuction is assigned to the left mouse button. With the mouse a line can be indicated and with a mouseclick the measurement legend is determined (according to the setting of 'type of measurements', see paragraph 6.9.1.1.1.1 on page 147, Definition of views in the standard directions for discussion) and attached to the cursor. This text can now be moved to the proper place, and with a secon dmouse click the legend is placed permanently.

Font height
: Is used to set the font height (in millimeters) of the texts to be placed.

Insert text
: Is used to insert a free text on a user-defined position.

Remove text
: If this function is assigned to the left mouse button, then already placed texts can be indicated, and removed with a click on the left mouse button.

6.9.4 Draw selected linesplan on paper

With this option a selected lines plan can eb prnited on paper. As always, the output can be redirected to file. With a lines plan a high resolution might be desired, and the best method to obtain that is to use a file in vector format, such as DXF or PostScript. This can be set in Config, please see section 5.1.10 on page 44, Output filetype.

6.10 Shell plate expansions and templates

This menu lists all plates that are currently present in the model. Shell plates are defined in the GUI as 'shell region' — a notion that is discussed in section 6.1.2.2 on page 59, Surfaces — see for guidelines section 6.3.5.24 on page 101, Define Shell Region. The first column of this menu indicates whether the plate is selected or not, the second column contains the solid and plate name, the last column shows whether the region is valid or not. Valid plates may be selected for generating plate expansions and template information.

Warning

Before generating plate expansions, make sure the model is defined accurately enough. The distance between points and curves must be less than 1 mm. See [Make all curves consistent] (discussed on page 134).

6.10.1 Processing the current plate

Whether a plate is selected or not, if the text cursor is on a valid plate then that plate can be processed using the following menu items:

- [Plate expansion]→[Current plate]→[On Paper and/or file] brings op the dialog for output of tables and drawings of the expansion of the current plate, see section 6.10.3 on this page, Production of plate expansions.
- [Plate expansion]→[Current plate]→[On Screen] shows a preview of the expansion of the current plate on screen.
- [Templates]→[Current plate] brings up the dialog for output of tables and drawings for the production of templates for the forming of the current plate, see section 6.10.4 on page 153, Production of templates.

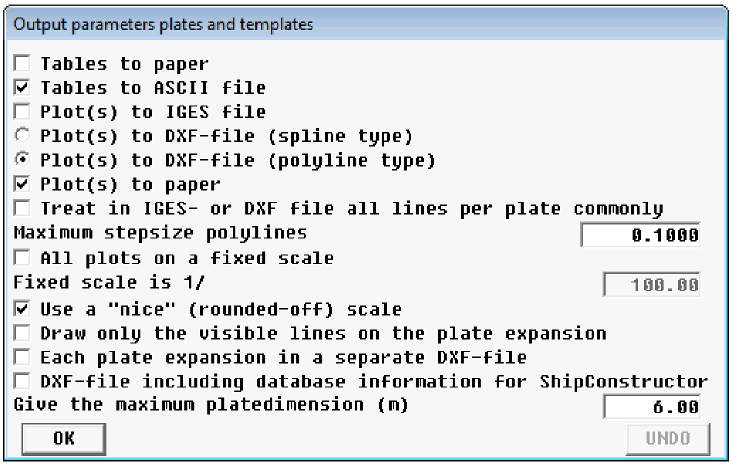
6.10.2 Processing selected plates

Depending on the state in the leftmost column, selected plates can be processed using these menu items:

- [Plate expansion]→[Selected plates on paper and/or file] brings op the dialog for output of tables and drawings of the expansions of selected plates, see section 6.10.3 on this page, Production of plate expansions.
- [Templates]→[Selected plates] brings up the dialog for output of tables and drawings for the production of templates for selected plates, see section 6.10.4 on page 153, Production of templates.

6.10.3 Production of plate expansions

After selecting any of the above menu items for plate expansion, the dialog below configures the output.



Configuration of output of plate expansions.

- Check [Tables to paper] to print out the tables with expansion information on the selected print device (discussed on page 12). This output also contains any warnings and error messages, see section 6.10.3.1 on the next page, Warnings and error messages.
- Check [Tables to ASCII file] to write out the tables to a textfile with extension .XTB. This file will also contain the plate weight, center of gravity and the approximate cutting length.
- Check [Plot(s) to IGES file] to generate a file with extension .IGS containing the expanded plate with hull lines in the format of IGES type 126 (NURBS curve).
- Check [Plot(s) to DXF-file (spline type)] to generate a DXF file containing the expanded plate with hull curves in NURBS representation. This file can be imported in Autocad version 14, but unfortunately version 2000 (and subsequent versions) contains a serious bug. Its consequences have been discussed in section 6.8.5 on page 138, All lines in 3D to AutoCAD DXF-NURBS format.

- Check [Plot(s) to DXF-file (polyline type)] to generate a DXF file with curves in DXF's polyline format, which is essentially a chain of short straight line segments.
- Check [Plot(s) to paper] for a hardcopy of the shell plate expansions on the selected print device (discussed on page 12).
- When [Treat in IGES- or DXF file all lines on the plate commonly] is checked, the individual curves of a shell plate expansion are grouped together. Both in IGES as well as in DXF parlance such a structure is called a 'block', and the advantage can be that when shifting the plate expansion the whole plate is picked up, and not only loose lines.
- The [Maximum stepsize polylines] is, by approximation, the greatest length of the straight line segments of a polyline in meters at scale 1:1. Note that adjacent line segments may be situated exactly on a straight line, in which case these segments will be merged and the final segment length will consequently be greater than the value specified at this option.
- With [All plots on a fixed scale] the scale at which all expansions will be drawn on paper can be specified. If this option is deselected each plate will be drawn at an individual scale.
- If the above option is not checked, checking [Use a "nice" (rounded-off) scale] will result in the use of a scale that is practical for measuring, e.g. 1:10 or 1:25. When unchecked each plate fill be printed page-filling.
- If [Draw only the visible lines on the plate expansion] is checked then hidden polycurves will not be drawn on the expansion plot. Otherwise all polycurves in an expansion will be plotted.
- If [Each plate expansion in a separate DXF-file] is checked then a separate file will be created for each shell plate. Otherwise all expansions will be collected in the same file.
- The option [DXF-file including database information for ShipConstructor] adds additional information to the DXF-file with the expanded shell plate, such as plate area and COG's. Also, with this option an additional DXF-file will be created that contains the (approximate) three-dimensional shape of each shell plate.
- With [Maximum plate dimension] the maximum size (in meters) of an expanded shell plate can be specified. If the option [Each plate expansion in a separate DXF-file] is not selected, this information is used to determine the mutual distance of the expanded shell plates in the resulting IGES or DXF file.

6.10.3.1 Warnings and error messages

The output of tables with expansion information should be checked for the occurence of any of the following messages:

The maximum number of edges for the expansion is 2000. This plate has ⋯.
This message indicates that the maximum number of face edges is exceeded. This shouldn't be confused with the number of border curves in the contour of the plate. In this context the number of edges refers to the total number of edges of all faces which appear in the plate under consideration.

The maximum number of points for the expansion is 2000. This plate has ⋯.
This message indicates that the maximum number of points in a plate is exceeded.

Two points of the plate coincide. No expansion can be made.
This message is considered to be self-explanatory.

In line ⋯ a deviation between point and line of more than 1 mm occurs.
This message indicates low accuracy, but the expansion process does continue. However, the results must be suspected!

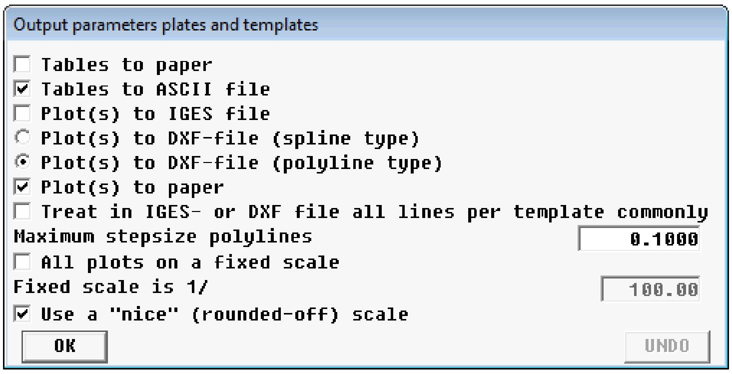
In line ⋯ between the points ⋯ and ⋯ the line is not a geodetic. The length difference amounts ⋯ %. It is advised to give the line more support with an additional line.
A part of the expansion process is the subdivision of the plate in triangles, with geodetic curves as sides. A geodetic curve is the shortest line, over a curved surface, between two points. When a low number of points is present in the plate, it can be that the edges of the triangles are not geodetic. You can improve that situation by the inclusion of one or more lines in the neighborhood of the coordinates which are printed in this warning. After this warning the expansion process continues, but the results should be treated with suspect.

The number of points on the plate is ⋯. That is more than the maximum of 1000. The plate expansion is produced, albeit *not optimized for minimal deformation.*
This message indicates that although the number of points in a plate is smaller than the absolute maximum, it is still too large for minimization of deformation.

6.10.4 Production of templates

Templates aid in the forming of shell plates to get the shape right. Templates run parallel to one of the main planes, configured when the plate was defined, see paragraph 6.3.5.24.4 on page 106, Definition of a shell plate. An additional longitudinal template connects the others at a given angle.

The options for output of template information are *mutatis mutandis* equal to the ones which apply to the shell plate expansion discussed above, section 6.10.3 on page 151, Production of plate expansions.



Configuration of output of templates.

6.10.4.1 Position and shape of templates

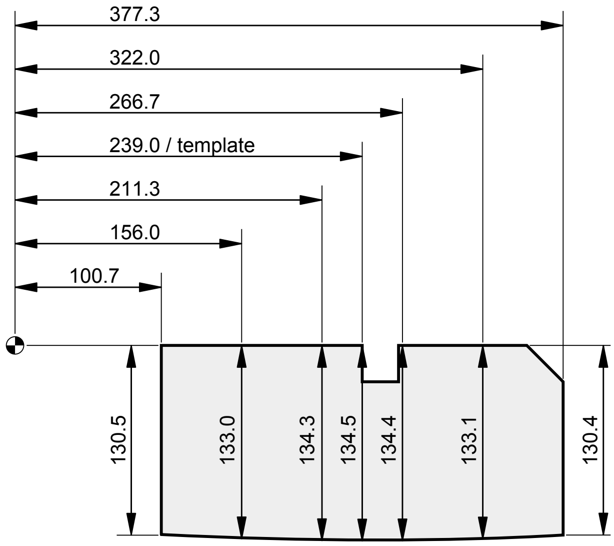
Each template has a name identical to the name of a polycurve on the shell plate expansion, which fixes the position. To aid the orientation of the template relative to the shell plate, as well as the orientation of the shell plate relative to the vessel, each template features a cut-away and a chamfer. The cut-away of 25 $\times$ 25 mm at the upper side is for aligning the templates, and to provide support for the longitudinal template. The chamfer at one side of the template is helps orienting the template and shell plate in relation to the vessel: If the templates run parallel to frames then the chamfer is located furthest from the center plane. Otherwise (templates are parallel to waterlines or buttocks) the chamfer is oriented in the direction of the fore ship.

The upper sides of all templates for the same plate are coplanar. That plane is chosen such that it minimizes the area of the templates, providing a minimal height of 100 mm, and therefore is seldomly co-planar with one of the main planes.

If the option [Tables to ASCII-file] has been checked, coordinates of the templates will be written to a `.tpl` file, in mm. All templates of the same plate are defined in the same coordinate system. The height is measured from the top of template and the breadth is measured from a certain reference line. A fragment of a `.tpl` file can look like this:

```
Line : Frames     400   Location : 0.400 m
          X        Height       Angle
       100.7        130.5          89
       156.0        133.0
       211.3        134.3
       266.7        134.4
       322.0        133.1
       377.3        130.4          89
       239.0        134.5             (Longitudinal template)
```

These coordinates can be set out to yield the dimensions of the template as in the figure below.

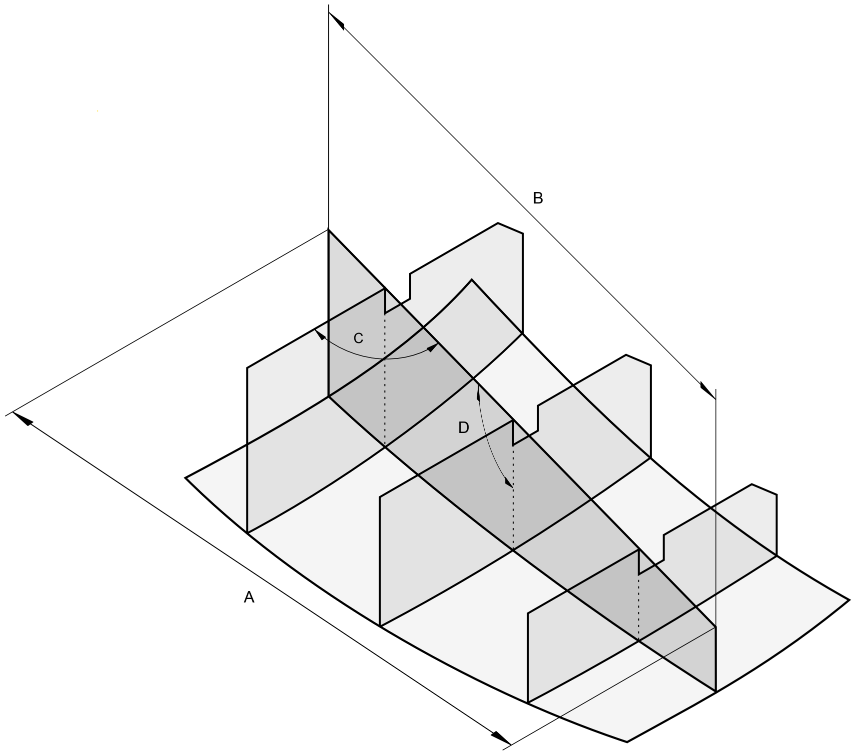


Coordinates listed in the tables define the dimensions of the template.

The angles listed in the tables define the angle (in degrees) between the plane of the template and the formed shell at both ends of the template. In case templates run parallel to the frames, this angle is set out aft of the template; If the templates run parallel to the waterlines then the angle is set out underneith the template, and if they run parallel to buttocks then the angle is set out on the inside of the template.

The tables are concluded with four additional values, illustrated in the following figure:

1. The length of the assembled framework of templates(A)
2. The length of the longitudinal template (B)
3. The angle between the longitudinal template and the other templates, measured in the plane through the upper sides of all templates (C)
4. The angle between templates and the plane through the upper sides of all templates, measured in the plane of the longitudinal template (D)



Framework of assembled templates.

6.11 File and object management

File and object management

1.	File history
2.	Save current design
3.	Object management
5.	Quit the program without saving

6.11.1 File history

This option is meant to save a set of designs or design stages. It is also possible to use it as a backup option. Maximally fifteen designs can be saved. Apart from the standard editing functions, the option [Select] is also available. With this button you can select a design to work with. When defining a new design with [New] or [Insert], the currently selected design is copied to a new file, for which you must specify a file name. The new design is a copy of the selected design at that moment.

In the menu with design variants one column is marked [Automat. Save]. In this column you can specify which design of the available set is used to save the hullform when the program performs an automatic save action. If the automatic save option is used without a marked design variant, by default the selected design is used for that purpose. The time interval for automatic save is documented in section 6.6.1.1 on page 133, General Fairway settings.

6.11.2 Save current design

The current hull shape is saved without leaving the program.

6.11.3 Object management

Note

This menu is also available from within the GUI, by selecting [Solids]→[Object Management...] in the menu bar.

When designing the shape of a ship hull it may be convenient to consider the vessel to be composed by multiple building blocks. These are the so-called *solids* — from which the purpose and background is discussed in section 6.1.2.3 on page 61, Solids — or *wireframes*, see section 6.3.7 on page 113, Wireframes. These objects are managed in this menu, which contains information as discussed below.

- The first column is used for the selection of solids to take part in Boolean operations, zee below.
- Name. The name of the object.
- Side. The side where the solid resides. The four possibilities are:
  - SB. The solid is a half hull, situated at SB.
  - PS. The solid is a half hull, situated at PS.
  - SB & PS. The vessel is symmetrical over center plane, while the solid is situated at SB, and also models the PS mirrored part.
  - Complete. The solid represents a complete hull with a part at SB as well as at PS.
- Active. This cell indicates which solids are active in the GUI.
- Single, a cell which indicates that his solid is *single selected* for subsequent operations which only act on a single solid, such as hullform transformation.
- Visible. Indicates whether the object is visible in the GUI.
- Locked. If this cell is set to 'yes', the solid is protected against any modification.
- Buoyant. If this cell is set to 'yes', and if this object is a solid, then it is included in Fairway's hydrostatic calculations. On export to PIAS this switch determines whether this object — solid as well as or a wireframe — will be exported either as added hullform or as extra body.
- PIAS, which indicates whether this object is included in conversion to PIAS' frame model. If a user has the option available, then it can also be specified here that additionally a PIAS triangulated surface file must be created. Details and mechanisms of this conversion are discussed at section 6.8.1 on page 137, Convert this Fairway model to PIAS model.
- Main hull. This field is only applicable for objects which have been assigned for export to PIAS. All 'main hull' objects are glued one after the other and together form the main hull, the same as is depicted by 'main hullform' in Hulldef.
- Linesplan, which determines whether this solid is included in the lines plan, as discussed in section 6.9 on page 145, Define and generate lines plan. - Export, which determines whether this solid is included in conversion to other (CAD) file formats — such as STL, DXF or IGES — as discussed in section 6.8 on page 136, Export of hullform.
- Type. For data management reasons some objects are also modelled as 'solids'. This field indicates the type of objects, possible values are:
  - The ship hull, or part of the hull, represented by a *solid model*
  - The ship hull, or part of the hull, represented by a *wireframe model*
  - The Sectional Area Curve.
  - A projection line.
  - Empty.
- Curved surfaces. Fairway can derive the shape of the surfaces inbetween the curves from the shape of the curves. For this task an interpolation method on the 'tangent ribbons' can be set here, please refer to section 6.1.2.2 on page 59, Surfaces for a discussion of this issue.

Attention

The columns **PIAS** and **Export** govern the export of Fairway's solids and wireframes. The actual execution of such a conversion, and details of their operation are discussed in section 6.8 on page 136, Export of hullform.

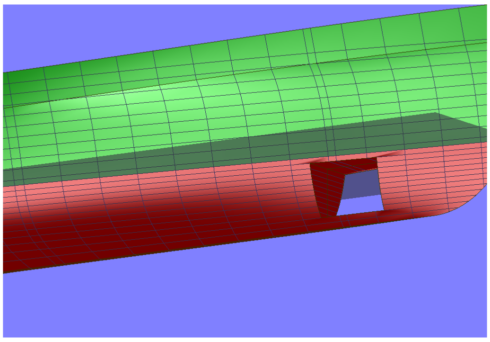
Apart from the general menu bar functions, which amongst other things allow copy and paste of a solid, the menu bar in this window contains a number of additional functions:

- File IO operations of solids and wireframes:

- Export an object. When this option is chosen the highlighted solid or wireframe will be exported to a file, with a user-defined name. The file which is created can be imported into another Fairway project, but it can also serve as a stand-alone hull shape. Note that if master/slave relations were present between curves in this object and other objects, these relations will be removed.
- Import a single object in Fairway format. After specifying the desired file name, the solid or wireframe is imported into a new object. Make sure that the Fairway file to be imported contains only a single object.
- Generation of objects of simple shape. At this moment two shapes are available, a 'minimal ship', (which is an object containing a deck line, a stem/stern contour and one straight ordinate) and a 1x1x1 m cube. In due course scaling options and rotation options will be included, in order to give the generated objects the desired size, location and orientation. While these options are still lacking, one can use the 'hullform transformation' (see section 6.5 on page 129, Hullform transformation) to resize and translate objects.

- Functions to combine or divide a hull shape over center plane, ([Centerplane]):
  - [Merge]. This function will merge two solids at the opposite sides of center plane. In order to use this function, three conditions have to be met:
    * Both solids to merge should contain distinct PS and SB half hulls.
    * The two solids must match exactly at center plane. This requirement implies that the vessel must be closed at deck. If that is not the case, the vessel can be closed automatically with a straight deck, see section 6.6.7 on page 134, Close vessel at deck.
    * Both solids should be selected, and others not. Otherwise the program does not 'know' which solids to use.
  - [Split]; split a complete hull, which contains both PS and SB, into separate PS and SB half hulls.
- Functions to merge an aft ship with a separate fore ship, or divide a single solid into a distinct aft part and forward part ([Aft+fore]):
  - The [Merge] option, which involves merging aft ship and fore ship, which do not necessarily have to be strictly adjacent to each other; if both parts are separated, they will be extended towards the middle, and there connected to each other. The [merge] process is reasonably automated, but it should **exactly** meet the following conditions (which are as such not very special):
    * Exactly two solids should be declared as 'active', not intersecting each other, and of which the one entirely located forward of the other.
    * On the terminal frames of the two solids — where they are connected to each other, and which we here for brevity baptize 'midship' — the solids should have real and complete polycurves of type *frame*. This frame should extend up to the deck edge, so the lines across the deck and centerplane should not belong to a frame.
    * The parts to be merged should each be a half hull, with a phantom face at centerplane, located at the same side (PS or SB).
    * The 'midship's must not have non-convex faces. Thus, the 'midship' as a whole may be non-convex, but must then be divided, for example with waterlines and buttocks into convex faces.

    This [Merge] operation results in one new solid where aft and fore are combined. As a rule, this result will have to be postprocessed a bit — for example toggling knuckles or chines, or deleting newly created intersection lines on those locations where the two 'midship's had a slightly different shappe — but that can be done with the regular tools of the GUI. In any case, the resulting solid will have a deck line on centerplane, which may be required because the two constituent parts may have a different height at their 'midship' sections, so a transverse facet should be placed there in order to bridge this difference. Anyway, if this deck line is redundant it can be removed, that is up to the user.
  - [Split], here a longitudinal position can be specified where the single selected solid is to be split into an aft part and a forward part. There is no need for a frame to exist at that position, if this is lacking it will be created. If the assignment of the splitting lines to polycurves is not satisfactory, you can change that in the GUI.
- Boolean operations make it possible to combine two solids in various ways. These solids are marked with the letters A and B in the first column, after which it is possible to perform the following Boolean operations: *union* $A \cup B$, *intersection* $A \cap B$ and *difference* $A - B$ (in the menu represented with A+B, A^B en A-B). This way it is possible, for example, to merge a keel with a hull, or construct an open bow thruster tunnel by subtracting a cylinder from the hull, as can be seen in the illustration below. Although the Boolean operations are fully implemented, they are not generally available.



Result of a Boolean ‘difference’ operation.

6.11.4 Quit the program without saving

With this option it is possible to leave the program without saving the design. It deals only with changes in design. Special points, definitions of linesplans, main dimensions and other parameters not related to the form will be saved. To prevent mistakes when using this option, the program asks: “Are you sure¿‘.

6.A Appendices

6.A.1 File extensions

Fairway stores model data in various files. The file names have a common stem, represented here by ∗, but different extensions.

`∗.fwy`	This is the main file that references the others.
`∗.fw1`	Contains topological information consisting of points, edges and faces.
`∗.fw2`	Contains general information about curves.
`∗.fw3`	Contains geometric information in the form of NURBS vertices.
`∗.fw4`	Contains special points.
`∗.fw5`	Contains the name, main dimensions and coefficients, as well as the position sets and other settings.
`∗.fw6`	Contains data for any defined linesplans.
`∗.fw8`	Contains data for geometric dependencies between curves (master-slave).
`∗.fw9`	Contains shell regions.
`∗.cf`	Contains groups of polycurves that are not part of a solid.
`∗.sf`	Contains connections between polycurves in the `∗.cf` file.

Warning

The files `∗.fw1`, `∗.fw2`, `∗.fw3`, `∗.fw8` and `∗.fw9`, as well as `∗.cf` and `∗.sf` depend on each other. Do not separate these when copying files.

6.A.2 File CXF and SXF file formats

These file formats are intended to import curves — in the CXF, Curve eXchange Format — and surfaces — in the SXF, Solid eXcange Format — into Fairway. For backround and context reference is made to paragraph 6.3.7.1.3 on page 116, Import ship hull models in SXF/CXF format.

6.A.2.1 Syntax of Curve eXchange Format

A CXF file is a plain ASCI file, with an even number of lines. Each pair of lines consists of a code (the first line) and an argument (the second line). The code defines the meaning of the argument. Behind the code a # may be placed, which precedes a comment. Lines that aren’t immediately preceded by a code and start with a # are recognized as comment, and ignored. All units are in metres, the sequence of vectors is Length, Breadth, Height. SB = + and PS = -. Currently defined codes and arguments are:

```
  10     #File type (must be the letters 'CXF')
CXF
  20     #File version (Must be 1)
   1
  30     #Creator (Description of program or person who created this file)
Creator A
  40     #Project name
Project ABCDEFG
  50     #Project version number
   N
  60     #Date (Year / Month / Day)
yyyy   mm   dd
  70     #Time (Hour / Minute / Second)
  hh   mm   ss
1000     #New solid (Currently only one solid is supported)
Solid name
1005     #Solid identification number (optional for single solid)
   N
1500     #Solid attributes (internal to Fairway)
   N
2000     #New line
Line name
2010     #Chine property (1=chine, 0=ordinary)
   N
2020     #Plane type (0=frame 1=wl 2=buttock 3=diagonal 4=arbitrary plane 5=3D line)
   N
2030     #Normal vector of plane (L, B and H components of normal vector)
   L.lllll     B.bbbbb     H.hhhhh
2040     #Location of plane (metres from origin)
   P.ppppp
2500     #Line attributes (internal to Fairway)
   N
3000     #New segment
Segment name
3020     #Basic geometry type (1=polyline 2=NURBS)
   N
3100     #Coordinates of polyline point (Length, Breadth and Height of a point)
   L.lllll     B.bbbbb     H.hhhhh
3110     #Polyline point, specified as reference to a (unique) vertex number of the SXF file,
         #followed by the coordinates (Length, Breadth and Height) of this point.
   Vertex number       L.lllll     B.bbbbb     H.hhhhh
3200     #NURBS Vertex (Length, Breadth, Height and Weight of a NURBS vertex)
   L.lllll     B.bbbbb     H.hhhhh     W.wwwww
3300     #NURBS knot
   K.kkkkk
3400     #NURBS order (order=degree+1)
   K
3500     #Spline/segment attributes (internal to Fairway)
   N
3505     #Spline identification number (internal to Fairway)
   N
9999     #End of CXF file
   Optional CRC checksum, otherwise 0
```

Note

- Currently the codes 30, 50, 60 and 70 are not used in Fairway, but in the future they may be used.
- From a line either a polyline or NURBS representation has to be given. The preprocessor determines both representations and writes them in the CXF file.
- Concerning the NURBS, please remember that number of vertices + order = number of knots.
- Polyline points may be specified directly (code 3100) or as reference to a solid vertex (code 3110). For application in combination with the SXF file, finally, only references must be used.
- The chine property (code 2010) may be omitted. The default value is 'ordinary'.

6.A.2.2 Syntax of Solid eXchange Format

Similar to the CXF file, an SXF file is a plain ASCI file, with an even number of lines. Each pair of lines consists of a code (the first line) and an argument (the second line). The code defines the meaning of the argument. Behind the code a # may be placed, which precedes a comment. Lines that aren't immediately preceded by a code and start with a # are recognized as comment, and ignored. Vertex locations are in metres, the sequence is Length, Breadth, Height. All faces must be oriented clockwise (seen from the outside). Currently defined codes and arguments are:

```
   10     #File type (must be the letters 'SXF')
SXF
   20     #File version (Must be 1)
    1
   30     #Creator (Description of program or person who created this file)
Creator A
   40     #Project name
Project ABCDEFG
   50     #Project version number
    N
   60     #Date (Year / Month / Day)
yyyy   mm   dd
   70     #Time (Hour / Minute / Second)
  hh   mm   ss
 1000     #New solid (Currently only one solid is supported)
    Solid name
 1005     #Solid identification number (optional for single solid)
    N
 1500     #Solid attributes (internal to Fairway)
    N
 2000     #Vertex number, plus coordinates of that vertex
    Vertexnumber    L.lllll     B.bbbbb     H.hhhhh
 2010     #Vertex number, plus name of that vertex
    Vertexnumber    name
 3000     #Edge number, plus the numbers of the two vertices bounding this edge
    Edgenumber       Number_of_vertex1    Number_of_vertex2
 4000     #Indicates start of face, face number
    Facenumber
 4010    #Edge of face: Reference to edge number & orientation (+1 or -1)
    Edgenumber      Orientation
 9999     #End of SXF file
    Optional CRC checksum, otherwise 0
```

Note

- Currently the codes 30, 50, 60 and 70 are not used in Fairway, but in the future they may be used.
- An edge orientation (as used in the face definition) of +1 means the edge is used for that face according to the sequence of definition of that edge. An orientation of -1 means it is used for that face in the opposite direction.

6.A.3 File format of diagrams for generation of a sectional area curve

The file `kvslap.txt` in the PIAS installation directory contains the numerical representation of the diagrams of Lap, which are used to generate a target sectional area curve (target SAC) based on main dimensions, see [Change the shape of the SAC] (discussed on page 97). This is a text file, see section 3.6 on page 29, ASCII text file. The diagrams give a frame area at various ordinates (longitudinal positions) through which the SAC can be fitted. The file format is explained here, followed by the represented diagrams. The information from this appendix allows you to adjust these diagrams.

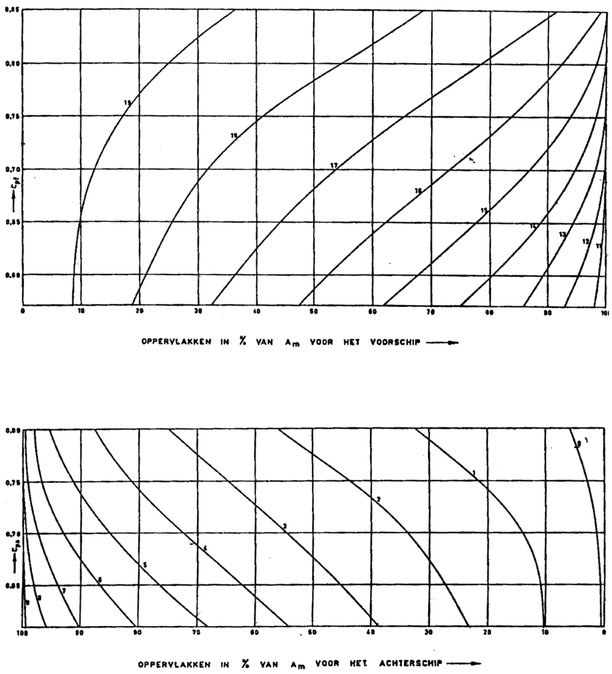
The first half of the file is for single-propeller ships. After the row with the word `DUBBELSCHROEF` follows the information for twin-propeller ships. Each of these parts consists of a diagram for the aft ship and a diagram for the fore ship.

Each diagram starts with a row with two numbers. The first number is the number of prismatic coefficients in the table, the second is the number of ordinates in the table. Next follow the prismatic coefficients, each on a separate row. Finally follows the table of percent values of the midship area, each ordinate on a row of its own. The first column gives the ordinate number, followed by a column for each prismatic coefficient.

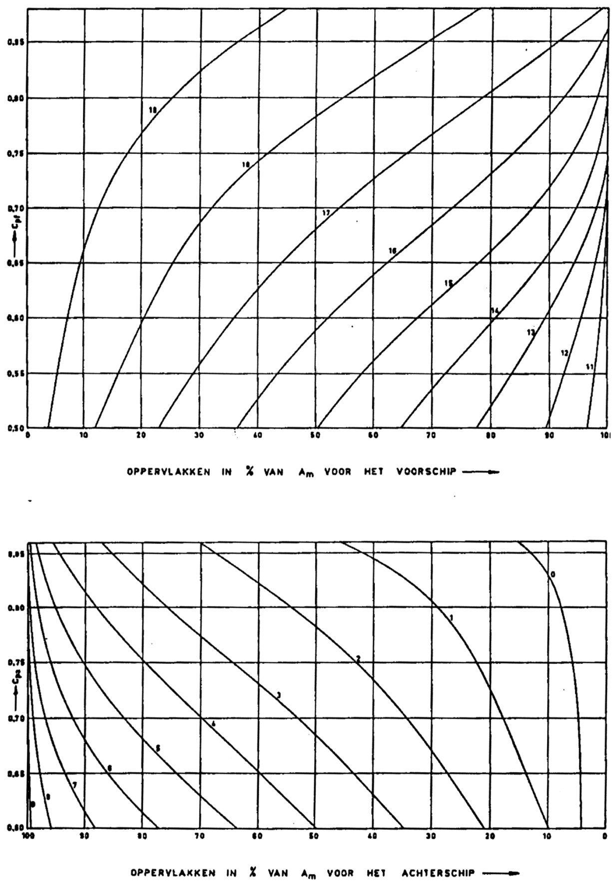
As an example you will find the representation of the diagram for the fore ship of single-propeller hulls below.

```
8        11
0.55
0.60
0.65
0.70
0.75
```

0.80								
0.85								
0.9								
10	100	100	100	100	100	100	100	100
11	96.9	98.7	99.6	100	100	100	100	100
12	91.2	95.1	98.1	99.8	100	100	100	100
13	82.8	89.6	94.2	98.1	99.8	100	100	100
14	70.5	80.4	88.5	94.5	98.1	99.9	100	100
15	57.1	67.4	77.4	86.8	93.7	98.4	100	100
16	43.9	52.2	62.1	72.8	83.4	92.5	98.9	100
17	29.1	36.4	43.9	53.6	65.2	78.5	91.5	99.1
18	16.9	21.1	25.4	31.4	41.1	54.7	68.7	79.6
19	8.2	8.7	9.7	12.1	17.2	25.2	36.1	45.2
20	0	0	0	0	0	0	0	0



Frame areas for single-propeller hulls.



Frame areas for twin-propeller hulls.

6.A.4 Managing various sets of user preferences

Changes in the user preferences of the GUI, [File]→[Preferences...], are saved in a text file in a directory defined by the operating system. By copying this file it is possible to manually manage several variations of user preferences, including the transfer to other installations. For example: for a demonstration on a projector or while recording a screencast it can be beneficial to temporarily work with thicker lines and colors of higher contrast; and for illustrations on paper a white background and darker curves work better (see also section 6.A.5.4 on page 164, Adjusting hotspot appearance). Configurations for these occasions are easily saved and restored by hand.

The file containing the customized preferences (as far as they divert from the default) carries the name `Fairway.ini` and resides in a directory called `SARC`. The location of that directory depends on the Windows version and is typically configured in the `%APPDATA%` environment variable. Examples are `C:\Users\`*`User Name`*`\AppData\Roaming\SARC\Fairway.ini` and `C:\Documents and Settings\`*`User Name`*`\Application Data\SARC\Fairway.ini`.

6.A.5 Customizing the dragger appearance (advanced)

The appearance of the draggers, with which the position of points and other elements can be manipulated, can be adapted to personal preferences. This is done by editing one or more geometry files.

The default geometry of the draggers is compiled into the program, but an image thereof is contained in the `*.iv` files in the installation directory. These are not used by the program as it is, but it is recommended to leave them untouched, for reference of the default geometry.

- `arrowedTranslate1Dragger.iv` contains the geometry for the linear dragger.
- `arrowedTranslate2Dragger.iv` contains the geometry for the planar dragger.
- `arrowedTranslate3Dragger.iv` contains the geometry for the spatial dragger.

You should know that the linear dragger is used twice inside the planar dragger, and the planar dragger is used three times inside the spatial dragger. So, in order to maintain uniformity, if one of these files is changed then a similar change may be needed in the other files.

To use custom versions of these files, perform the following steps.

1. Copy the *.iv files from the installation directory to a directory of your choice. Do not rename the files.
2. Edit the files with a pure text editor, e.g. notepad.exe (see section 3.6 on page 29, ASCII text file).
3. Define the environment variable SO_DRAGGER_DIR to point to the directory with the modified files.
4. Restart the program. Note that, if you start Fairway from the command prompt, then the command prompt window needs to be restarted as well before the changes take effect.

After a short introduction to the file format, we will give a few examples to experiment with.

6.A.5.1 File format

The format of the geometry files follow the Open Inventor File Format. You don't need to know this format in detail in order to make simple changes, the contents of the files are quite comprehensible. But if the format is violated then the file will fail to load and the dragger will have no geometry at all. If you cannot get it to work, just delete the file completely and the system will fall back to its compiled-in version.

We follow the convention that identifiers in all-capital letters are only used within the file in question. Mixed case identifiers are referenced by the program to construct the final geometry of the draggers.

If you want to know more about the format, here are some pointers:

- MIT has a collection of files discussing the Open Inventor file format[5]. This material is dated and has not been updated for a long time, but may still be relevant in many if not all respects.
- Chapter 11 of the *Inventor Mentor* (Josie Wernecke, 1994) discusses the file format from a programmer's perspective. This book can be found on-line in HTML[6] and PDF[7].
- Most of the scene objects that are available to you, and their fields and accepted values, can be read from the programmer's documentation[8]. Look for sections called "FILE FORMAT/DEFAULTS", as for instance in the class documentation of SoDrawStyle[9].

6.A.5.2 Increasing the dragger size

The on-screen size of the draggers is held approximately constant, irrespective of the camera zoom level and distance. This is accomplished by the use of a SoConstantSize node and its value field projectedSize. Let's say that arrowedTranslate1Dragger.iv contains the line

```
DEF ARROWED_TRANSLATE1_CONSTANT_SIZE SoConstantSize { projectedSize 50 }
```

This defines the identifier ARROWED_TRANSLATE1_CONSTANT_SIZE. Whenever this identifier is used in the file, 1 unit size in the geometry following after it (within the same Separator) will approximately be 50 pixels on screen.

So, in order to increase the size of the dragger, it suffices to change the above line into

```
DEF ARROWED_TRANSLATE1_CONSTANT_SIZE SoConstantSize { projectedSize 60 }
```

You will want to repeat this exercise in arrowedTranslate2Dragger.iv and arrowedTranslate3↩ Dragger.iv.

6.A.5.3 Changing the arrow head

Dragger axes are represented by arrows. The arrow head is constructed by means of a Cone node, with fields for height (arrow head length) and bottomRadius (arrow head width). You could make the arrow more articulate by increasing bottomRadius, for example.

Again, you will want to repeat this exercise in arrowedTranslate2Dragger.iv and arrowedTranslate3↩ Dragger.iv.

[5] http://web.mit.edu/ivlib/www/iv.html
[6] http://www-evasion.imag.fr/Membres/Francois.Faure/doc/inventorMentor/sgi_html/
[7] http://www.ee.technion.ac.il/~cgcourse/InventorMentor/The%20Inventor%20Mentor.pdf
[8] http://coin3d.bitbucket.org/Coin/group__nodes.html
[9] http://coin3d.bitbucket.org/Coin/classSoDrawStyle.html#_details

6.A.5.4 Adjusting hotspot appearance

The “hotspot” of a dragger is the transparent sphere around the arrows that reacts to mouse clicks, which eases picking of the dragger. Depending on your monitor, you may find that the rendering is too weak or too strong. This can be corrected by adjusting the `transparency` field in the `ARROWED_TRANSLATE?_HOTSPOT_MATERIAL` nodes in `arrowedTranslate1Dragger.iv` and `arrowedTranslate2Dragger.iv`. If you prefer to not see the hotspot at all, you can set `transparency` to 1.

If you have configured a white modelling view background, instead of the default, you way want to increase the contrast by setting the red, green and blue values of the colour fields of the hotspot material to

```
diffuseColor 1.0 1.0 1.0
emissiveColor 0.0 0.0 0.0
specularColor 1.0 1.0 1.0
transparency 0.85
shininess 1.0
```

6.A.5.5 Switching off the feedback plane

When translating in a plane, the plane is rendered transparently in the corresponding color. If you do not like this, the plane can simply be disabled by editing `arrowedTranslate2Dragger.iv` as follows. Comment-out all lines between the opening and closing curly-braces of the identifiers `arrowedTranslate2FeedbackOrthogonal↩ Active` and `arrowedTranslate2FeedbackArbitraryActive`, by putting a “#” in front of each of these lines.

Chapter 7

Hulldef: hullform definition and output

This is the PIAS module to define, and manage the hullform, an to apply it for output and export. The notion of 'hullform' is interpreted rather wide in this module: it also contains related matters, such as openings, deck line and wind contour. The main menu of Hulldef is depicted below, but before we get into the details of the program will we explain below first in general the hull form definition method.

Hull geometry data

1.	Input, edit and view general particulars and hull geometry data
2.	Output of hull geometry data
3.	Export of hull shape data to a number of specific file formats
4.	Import frames from (a number of specific formats of) a text file
5.	Generate cylindrical shapes
6.	File and backup management

7.1 Hulldef's hullform definition method

PIAS contains two modules for hull form definition. One is Fairway, which is designed for hull shape design and fairing, and which generates a closed hull surface. In addition, there is this module, Hulldef, where cross sections are entered. Although Fairway gives a much more complete and better result, working with Hulldef is in general faster. Because of the difference in objective, there is also a difference in hull shape representation method. section 2.10.2 on page 16, Hull form representations contains a list of the representations used in PIAS, and there we read that Hulldef uses the **frame model**. Its main characteristics are:

More in detail the characteristics of Hulldef are:

- A hull form is defined on cross sections (or frames. The words cross sections, ordinates and frames are being used alternately in the manual they are considered to be synonymous) only. Buttocks and waterlines are not used.
- The advantage of this definition method is that it is very simple: basically, a body plan is just digitized. One drawback is in between the frames nothing exists, so no intermediate frames can be interpolated. However, in the calculations this plays no role, PIAS is fully optimized for this definition method.
- The number of frames is up to the user, although there are some guidelines, which are discussed in section 7.2.4.1 on page 174, Number of frames. Frame distances are basically free, although there is a limit on very uneven spacing, see section 7.2.4.2 on page 174, Ratio of longitudinal frame distances for details.
- Longitudinal discontinuities are defined by using a two coinciding frames at that location. Examples of longitudinal discontinuities are the transition from to forecastle and the forward and aft sides of deck house, moonpool or hatch coaming, see section 7.2.4.3 on page 175, Double frames for examples.
- If no special settings have been made, only a **demi ship** needs to be defined, which is assumed to be symmetrical, so that the other half is identical, but mirrored. That half the ship is located to starboard, so transverse coordinates are positive. Of course, there are facilities for asymmetric ships and so on, but those are the exceptions.
- The frame shape is defined counter clockwise. So, in general starting at the bottom, at centreline, and then in outward/upward in the direction of the deck edge. Basically, this continues until the frame is closed, at centreline at the top. But in practice it is often more convenient to stop at the deck edge, and to let the

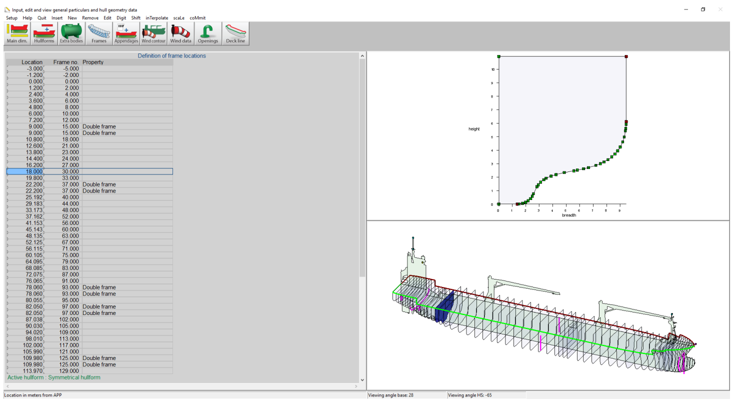
upper part (with the deck, or perhaps a hatchway or superstructure) be generated automatically using an appendage. Refer to section 7.2.5 on page 180, Appendages for that.

- More detailed tips are presented in section 7.2.4 on page 173, Frames (frame positions and frame shapes).
- **All hydrostatical and stability calculations (and the like) in PIAS are based on this frame model**.

Attention

Although the frame model is a sufficient basis for the calculations, it does not contain any information about the shape of the hull in between the defined frames. As a result, on output the shape of a longitudinal line or an intermediate frame might seldomly not be drawn accurately. However, this is no reason for concern, as these lines do not play any role in the calculations.

7.2 Input, edit and view general particulars and hull geometry data



Input window for main particulars and hull geometry.

With this option an input window appears, from which an example is depicted above. This window consists out of three sub-windows, from which the leftmost is used for alphanumerical data entry, the window top right for the graphical representation of one particular type of data and the window at the bottom right to show all data types as managed by Hulldef simultaneously, graphically and in 3D. For options in the 3D window reference is made to the last section of this chapter, notably the orientation box which provides a tool to assist in the proper viewing direction, see section 7.8.1 on page 190, View. With the top bar icons, the appropriate input subject can be selected, which can alternatively be achieved with a function keys-control combination, where <ctrl><F1> equals the activation of the leftmost icon, etcetera, until <ctrl><F9> for the rightmost icon. Anyway, the different subjects are:

1. Define main dimensions and further ship parameters
2. Hullforms
3. Extra bodies
4. Frames (frame positions and frame shapes)
5. Appendages
6. Wind contour
7. Wind data sets
8. Openings
9. Deck line

The menu bar contains, besides the common function, an additional [Visible] function, which can be used to specify which of the data elements as managed by this module (such as wind contour, openings and deck line) will be visible in the right-under three-dimensional representation window.

7.2.1 Define main dimensions and further ship parameters

In this menu the general particulars of the ship can be given. These belong to the ship data definition, therefore this menu is included in this hull definition module Hulldef, and also in the hull form design and fairing module Fairway, see section 6.4 on page 128, Define main dimensions and other ship parameters. Here only *ship data* are included, *program settings*, on the other hand, can be defined in Config. This module is subdivided into sub menus for the different categories:

Define main dimensions and other ship parameters

1.	Main dimensions and allowance for shell and appendages
2.	Roll data (for Intact Stability Code weather criterion)
3.	Frame spacings
4.	Draft marks and allowable maximum and minimum drafts
5.	Maximum drafts and minimum freeboards
6.	Allowable maximum trims
7.	Particulars of export to Poseidon
8.	Yacht particulars
9.	Particulars for SOLAS chapter 2, part B1
10.	Anchor handler particulars
11.	Towing hook and bollard pull
12.	Particulars for inland waterway container vessels
13.	Line-of-sight and air draft points

7.2.1.1 Main dimensions and allowance for shell and appendages

- Ship name, the project name, or the name of the vessel. This name will appear on all printed output.
- Length between perpendiculars. The trim is based on this length.
- Moulded beam, on the construction waterline.
- Draft, the moulded draft at $L_{PP}/2$ on the construction waterline. As a rule, for sea-going vessels>24 meter the summer draft according to the Load Lines Convention.
- Dredging draft, the draft with reduced freeboard, as could be applicable for hopper dredgers. for details please refer to daarover chapter 18 on page 384, Stability of open hopper vessels.
- Moulded depth, the minimum depth at side above base.
- Appendage coefficient, a multiplication factor for shell and appendages. In general, this factor has a value between 1.000 and 1.010. Please be advised that the common values for the appendage coefficient are for the effect of appendages *and* shell plate thickness. If a shell plate thickness is defined separately, the appendage coefficient may not include this shell effect.
- Mean shell plate thickness, which is used to determine the shell volume explicitly, and should be given in **metres**. The shell plate volume is determined offsetting the frame contours with the shell plate thickness, a method which leads to two effects. The first one is that the thickness of a transom or a flat and vertical stem is not taken into account. And the second effect is that the shell plate volume, contrary to the common shipbuilding practice, is added directly to the moulded volume.

7.2.1.2 Roll data (for Intact Stability Code weather criterion)

- Type of midship section: Round or sharp bilge.
- Projected area of (bilge-)keels: Total area in m^2.

These parameters are used to determine the angle of roll in accordance with the Intact Stability Code 2008. Table 2.3.4-3, where for the factor A_K the area entered here is substituted.

7.2.1.3 Frame spacings

This menu facilitates the definition of the (construction) frame numbers and also the frame numbers where the frame spacing changes. In each menu of PIAS where longitudinal distances are required it is an alternative to press function key <F3> and enter the frame number, which is converted into meters instantaneously. Please see that although <F3> will prove to be very convenient, it is only a local aid to convert frames to meters, all output and input of PIAS will still remain in *meters* or related units. There is also a somewhat more extended frame conversion utility, activated with <F4> — section 4.2 on page 35, Input window contains more details on the use of <F3> and <F4>. With the data entered in these menus an entire frame table can also be printed, please see section 7.3.5 on page 187, Frame location table for a discussion. The frame definition parameters as such are given the following submenu:

Definition of frame spacings

1. Define the frames where the frame spacing changes
2. Define frames spaces
3. Define remaining frame spacing data.

7.2.1.3.1 Define the frames where the frame spacing changes

Here the frame numbers (with a maximum of 150) can be entered where the frame spacing changes. The actual frame spacing can be defined at the next option, however, these 'frames where frame spacing changes' should be given first.

7.2.1.3.2 Define frames spaces

Here the frame spacings (in meters) can be specified, for each of the regions with a distinct frame spacing.

7.2.1.3.3 Define remaining frame spacing data.

The location of (construction) frame zero should be specified here, in meters, in PIAS' system of axes (see section 2.6 on page 9, Definitions and units. With the aftmost and foremost frames, which should also be specified in this menu, the frame range is fixed, which is important when printing a frame location. Finally, it can be indicated whether or not the frame spacing information should be used.

7.2.1.4 Draft marks and allowable maximum and minimum drafts

In this menu the location of the draft marks — and other locations where the drafts must be calculated and checked — can be given, which may be applied at the calculation of loading conditions with Loading. A maximum of twenty locations can be defined, and for each location must be given:

Name
A free, textual description which is printed on the output for identification.

Draft mark.
Indicates whether this location is a draft mark. If this is not the case, it is any other location where the draft has to be checked.

Check
Indicates whether the draft at this location must be checked to a minimum or maximum allowable draft. If so, this can be further detailed with:

- *Tmax global*, where the **mean** draft is checked against the maximum as set in Loading. This Loading maximum is a *type* rather than a number, e.g. Summer Draft or Winter North Atlantic. This can be set with option [Settings] in the loading condition as discussed in paragraph 16.2.1.3.2 on page 311, Draft.
- *Tmax local*, where the draft at this mark is checked against the local maximum, as given for this mark.
- *Tmin local*, *mutatis mutandis* for the local minimum.

Tmin and Tmax
If with the previous option is set that the draft should be verified against a local minimum or maximum, then those can be given here.

T ps/sb mean
which indicates if the mean draft over PS/SB has to be calculated, checked and printed.

Print paper
'Yes' or 'no', which indicates whether this mark should be included in output of Loading. In the majority of cases that will be desirable, however, occasionally one might wish to switch the mark 'off'.

Print screen
Which, similar to the previous option, indicates whether this mark should be printed on screen in the GUI of Loading.

Plot Just as in the previous two options, concerning the plots of the mark line in the GUI of Loading.

If <Enter> is pressed in the first column, the menu for the definition of line segments appears. Several line segments can be defined, for example a segment on the transom and another before the propeller. If both segments have an intersection with the waterline plane, the maximum draft will be taken. For each line segment should be given:

- *Name*, a free, textual description which is used in the user-interface.
- *Reference height*, This will be in general the underside of the keel, in which case the keel plate thickness should be given, negative (because in general the keel extents below the base line) and in meters (because that happens to be the standard in PIAS.
- *Side*, which indicates whether the mark is situated on PS, SB or on both sides (double).

If <Enter> is pressed in the first column, the menu for the definition of a table of coordinates pops up. It is not necessary that the line segment starts at the reference height.

7.2.1.5 Maximum drafts and minimum freeboards

Here the maximum drafts or the minimal freeboard can be given, as defined by the load line convention, so for example summer draft or WNA freeboard. One can choose either to give the freeboards, or the drafts; one is automatically converted into the other. For this purpose, in the last two lines of this menu the deck plate thickness according to the load line convention should be given, as well as the moulded depth (which might also have been given at section 7.2.1.1 on page 167, Main dimensions and allowance for shell and appendages, however, for convenience this parameter is also included here). The drafts or freeboards defined in this menu are amongst others applied when producing a deadweight scale (see section 10.2.6 on page 254, Deadweight scale) and at the check on maximum drafts in Loading, see the figure in the previous paragraph.

7.2.1.6 Allowable maximum trims

If trim limitations — a limit on the trim by the bow or the stern — are applicable, then such can be specified here, so that during the calculation of loading conditions, in Loading, they can be assessed for compliance. This facility might come at hand with probabilistic damage stability, where according to the explanatory notes the computation results might only be valid for a limited trim range (which might be dependent on the draft). In PIAS, trim by the stern is negative, so a maximum allowable trim by the stern of X meter would be given as a minimum trim of -X meter.

7.2.1.7 Particulars of export to Poseidon

In the main dimension menu, see section 7.2.1.1 on page 167, Main dimensions and allowance for shell and appendages, the real main particulars are being given, which are (also) for PIAS of prime importance. Additionally, some auxiliary dimensions or parameters exist which don't play a governing role in PIAS, or which are only being used for export to other software. These are:

- Class notation.
- Deadweight. This parameter is implicitly already fixed in PIAS by the combination of light ship weight (as given in Loading) and summer draft, but nevertheless the deadweight is explicitly given here as **design parameter**, also because SARC does not want to be involved in semantic discussions on which weight components would be included in or excluded from the deadweight.
- Scantling length and scantling draft, according to classification societies rules.

These parameters are for the time being only used in export of the PIAS model to DNV•GL Poseidon (see section 9.11.6 on page 243, Export to Poseidon (DNV•GL)).

7.2.1.8 Yacht particulars

- Area of sails As, according to ISO 8666.
- Beam of the waterline.
- Center of area As, according to ISO.
- Height of waterline hLP, according to ISO.
- Allowance 'delta' on STIX, according to ISO.

7.2.1.9 Particulars for SOLAS chapter 2, part B1

- Subdivision length.
- Position 'Aft terminal'.
- The lightest service draft as well as the subdivision draft, both as defined in the SOLAS 2009 & 2020 damage stability regulations. These values are relevant to determine the draft-dependant permeability, as applied at the calculations according to IMO res. A.265 and SOLAS 2009&2020. Compartments with such a varying permeability should be declared to belong to a specific type of 'space prob.damage stab. SOLAS0920', see paragraph 9.5.1.3.2 on page 219, Permeabilities for more details. These drafts are the same as the lightest and subdivision draft as given in Probdam.
- Permeability damaged compartments according. A draft dependent permeability can be chosen here according to various rules. This choice only applies to Hydrotables and Loading. In Probdam, the setting selected there is always adhered to.
- Number of persons with lifeboats provided.
- Number of persons without lifeboats provided. Note: The Solas 2020 no longer distinguishes between number of persons for whom lifeboats are provided and for whom not. Here, the total number of persons (N = N1 + N2) will be used.

7.2.1.10 Anchor handler particulars

Particulars to be used at the calculation of maximum anchor handling forces. For the details we refer to the chapter on the module for the calculation of tables of maximum anchor forces, Maxchain: calculation of maximum allowable anchor handling chain forces.

7.2.1.11 Towing hook and bollard pull

These particulars will be used to compute the heeling moment of the towing force, according to the stability criteria settings as discussed in section 15.4.5 on page 293, Bollard pull .

- Maximum bollard pull, in ton.
- Height of the towing hook above baseline, in meter.
- Breadth of the towing hook from center line, in meter.
- The length of the towing hook from the aft perpendicular, in meter, this based on **loadline length of the tug**.
- The loadline length of the tug, in meter, can be used to apply a deviating loadline length which is used by *the length of the towing hook from the aft perpendicular*. This only applies to IS Code 2020, tow tripping, and in particular for calculating the C1 coefficient. If this value is less than zero then the "normal" length between perpendiculars will be used, like defined by section 7.2.1.1 on page 167, Main dimensions and allowance for shell and appendages.
- A correction factor on the heeling lever. This is a dimensionless multiplication factor on the lever. For the IS Code 2020 tow tripping criteria this correction factor is not used.
- The virtual (keel) point above base line (in meter). At the definition of the tug stability criterion, also the lever to apply on the bollard force, in order to obtain the heeling moment, can be specified. For two of those option this point has to be defined:
  - **From towing hook to midway between draft and keel point**, for this the keel point needs to be defined. For vessels where the lower edge of the lateral area does not coincide with base line, the actual vertical location of that lower edge (or the *average* location of the lower edge) should be entered here as *keel point above base*.
  - **The distance from towing hook to a certain height above base**, that 'certain height' is defined via this virtual (keel) point.

In both of these options the height of this virtual (keel) point should be given in meters above base line and if this virtual (keel) point is protruding below the base line then this value should be negative.

7.2.1.12 Particulars for inland waterway container vessels

The following data is exclusively meant for the calculation of the maximum allowable VCG' for European inland waterway container vessels. Via section 15.2.1.2 on page 280, European inland navigation one can add the appropriate standard criteria and then perform the necessary calculations with Hydrotables. The following main dimensions need to be defined, length waterline, moulded breadth, moulded depth, appendage coefficient and maximum speed in knots or km/h in menu section 7.2.1.1 on page 167, Main dimensions and allowance for shell and appendages. The calculation, see section 10.2.8 on page 255, Maximum VCG' intact tables, can be done on the basis of the geometry of the PIAS hullform, but in the event that such a form is lacking a paragraph 7.2.1.12.3 on this page, Drafts-displacements table — as found in a Tonnage certificate — can be used as an alternative.

Particulars for inland waterway container vessels

1. Calculation of maximum allowable VCG values based on
2. Dimensions cargo hold and ballast tanks
3. Drafts-displacements table
4. Dimensions superstructures (only relevant for secured containers)
5. Print data of the expired module 'Rhine'
6. Import as much data as possible from the expired 'Rhine' module

7.2.1.12.1 Calculation of maximum allowable VCG values based on

This first option determines the applied hydrostatic determination method. There are three possible options:

- *Frame model*, Hydrostatic calculation is based on section 7.2.2 on the next page, Hullforms.
- *Ship (drafts-disp. table)*, Hydrostatic calculation is interpolated on the given paragraph 7.2.1.12.3 on this page, Drafts-displacements table. To be used for ship shapes other than a pontoon.
- *Pontoon (drafts-disp. table)*, Similar to *Ship* but another approximation formula is used for determining the KM and the waterline moment of inertia. Only to be used for pontoon-like ship shapes.

7.2.1.12.2 Dimensions cargo hold and ballast tanks

In the 'Description' column a name can be given. The tanks that need to be defined are the container hold(s) (which are obligatory according to the regulations) and ballast tanks which contain residual water.

7.2.1.12.3 Drafts-displacements table

The drafts with their associated displacements are in meters from the base line and in metric tons. Take care that the step size between the drafts is regular. The ratio between two successive step sizes must lie between 1/4 and 4. Start at the light draft and end at maximum draft. If a *Frame model* is used then this table of drafts and displacements will not be used.

7.2.1.12.4 Dimensions superstructures (only relevant for secured containers)

The rule of volumes that are not taken into account within a range of 0.05L from the extremities of the vessel is applied automatically when defining the aft and forward boundary of a superstructure volume. When defining the height of a superstructure the rule of, up to a maximum height of 1 meter above the defined depth or up to the lowest aperture in the volume under consideration, is automatically taken into account. In addition to the total volume, the total moment of inertia of the superstructures is also determined.

7.2.1.12.5 Print data of the expired module 'Rhine'

Here the input data is printed of the expired module 'Rhine'.

7.2.1.12.6 Import as much data as possible from the expired 'Rhine' module

The functionality of the expired module 'Rhine' has been incorporated into the modules Hulldef and Hydrotables from the beginning of 2020. With this option, several data entries can be imported from the 'Rhine', if applicable. When entries are imported, a question is posed if the main dimensions from 'Rhine', need to be imported as well. The main dimensions that will be imported are; project name, length waterline, moulded breadth, moulded depth, appendage coefficient and maximum speed in km/h and knots. These will overwrite the original main dimensions.

7.2.1.13 Line-of-sight and air draft points

For the assessment of the ship's draft and trim in a certain loading condition, as is computed by Loading, it might be convenient to let the draft/trim combination be verified against requirements on line-of-sight and air draft. The related parameters can be given in this menu and consist of:

- The *name* of the point, this is a textual description which is printed on the output.
- Its L, B and H coordinates.
- The type of point, which can be:

– The *conning position*, which is the location of the eye of the person who should have a particular line of sight.
– *Visibility obstruction*, which are fixed points of the ship which obstruct the vision, such as points of the bulwark or of a crane pedestal. Please realize that cargo items which are managed with specific cargo modules of Loading, such as containers, are always included integrally in the determination of line of sight, so visibility obstructions by such cargo items do not have to be entered in this menu.
– *Air draft points*, which is the highest fixed point of the vessel, and as such determines the air draft. Although always a single specific point will govern the air draft, through the effects of heel and trim it is not always certain on forehand which point that will be. For that reason, it might be convenient to give multiple air draft points here.

- *Fixed*, which applies only to *Air draft points*, indicates that the point is or is not switchable in LOCOPIAS.

The data as entered in this menu are being applied in the output of Loading, particularly in its GUI, see section 17.1.1 on page 324, Main window layout. In Loading the line of sight can also be checked against a visibility criterion, where one can choose between *IMO A.708(17)*, *Panama canal ballast* and *Panama canal full load*. Because different loading conditions might be subject to different criteria, the choice of the criterion is done per loading condition.

7.2.2 Hullforms

A hull form resides in a file which has been created by Hulldef or Fairway. Such a file may contain all hull shape data of an entire ship, however, it can be useful to define different parts in different files, and make a composition of them all. For example to be able to re-use certain parts in the future — such as a rudder or a coaming — or in case of variable buoyant parts, such as a deck cargo of wood. Therefore, at this option multiple hull shapes, with a maximum of 75, can be composed to one buoyant assembly. This menu contains one line per added hullform, and this line contains the following elements:

- The *description* where as a reminder a textual description can be given, which has no further relevance for the calculations. By the way, for the 'root form' – which is the form which is always present, simply because its filename equals the project filename — no further description otherwise than the standard can be given,
- The shifts in longitudinal, transverse and vertical directions, *L-dis*, *B-dis* and *H-dis*. L-dis(placement) is the distance between APP of the form to add, and the APP of the root form. And by analogy the H-dis(placement) is the distance between the base lines, and the B-dis(placement) the distance between the center lines.
- The *perm(eability)* defines the multiplication factor, with a typical value between -1 and +1, on the volume of the added form. With a permeability of 1.00 the added form contributes for the full 100% of its volume, with 0.00 its volume is neglected and a permeability of -1.00 the total volume of this added form is subtracted from the assembly.
- The *side* defines which side of this added form had to be added to the assembly. Choices are PS, SB or both.
- The *file name*, which can be given in two fashions. Either by just typing the name, or by pressing <Enter>, which opens the familiar Windows' file browser window where the intended file can be selected. Apart from giving the full path/file name, PIAS offers also the provision to specify that both the root file and the added file reside in the same folder, without specifying this folder explicitly. This is labelled "specifying the file name relatively", and is achieved by precursing the file name by the *ampersand*, the **&** symbol. **Moreover, it is even encouraged to apply this facility because it prevents that the folders names of all added hullforms have to be modified when you place the project in another folder or on another computer. The same applies when using hull forms as so-called 'external subcompartments' in Layout.**
- A column *print* (yes/no) which can be used to indicate whether the hullform should be included in the output, as is discussed in section 7.3 on page 186, Output of hull geometry data.

If a specific form has been selected by placement of the text cursor, then this one is active in the remainder of Hulldef, so the frames and appendages you inspect or edit are from this selected form. Finally, the upper menu bar contains three specific menu buttons:

- [Copy], which makes simply a copy of the entire row. In other menus this task is performed by the generic edit/undo/copy facility, however, because this menu contains two *paste* variants an exception had to be made here.
- [Paste], which pastes the copied row into the present row. This function has two variants, the first one is the regular, where simply the entire row content is pasted. The second one, [Paste Copy], makes a copy of the hull form file, where you are prompted for the file name. The first variant is used if the copied hull form is

applied multiple times, on different locations. The second is used is the pasted hull form shape is going to be modified independently from the original.

- [(A)symmetrical] which can be used to toggle between a symmetrical and asymmetrical hull form. In case of asymmetry the SB demi hull will obviously be represented in another file than the PS demi hull, so you should choose the appropriate file for each side. Anyway, the first time that a hullform is switched to asymmetrical, the programs ask whether the symmetrical hull form file should be copied. This copy might provide a quick start for the new demi hull.

Attention

This menu shows that the hullforms to add are stored in their own file, so essentially it is another ship or project, and can also be applied as such. It is even possible to define the additional hullforms simply as another project under its own filename, instead of using this *hullforms* option in the main project. Moreover, in the pre-2014 PIAS version that was the only method.

7.2.3 Extra bodies

This option is somewhat analogous to the previous one, here one can choose and select other hull forms, however, they will not be added to the buoyant assembly. Consequently, the columns L-dis(placement) etc. are not present here. One can wonder why one should choose another form, while **not** adding it to the assembly? That is because in some occasions a full form is used as an *external subcompartment*, as is discussed in paragraph 9.5.1.3.7 on page 220, Shape definition external subcompartments, and it would be somewhat silly if all *added* hull forms would be treated here in Hulldef integrally, and related forms which are not added, but used in the project at a later stage, not. Because these extra bodies do not constitute the buoyant assembly they are not included in the aggregated 3D view on the hull — the lower right window.

Attention

The *attention* paragraph of the previous section (Hullforms) also applies here.

7.2.4 Frames (frame positions and frame shapes)

This option serves in the first place to specify all frame locations (=frame positions), and secondly to define breadth-height coordinates for each frame. The maximum number of frames is five hundred. The longitudinal distance of the frame is measured from the aft perpendicular, and it is required that all longitudinal are strictly increasing. This menu contains the following auxiliary functions:

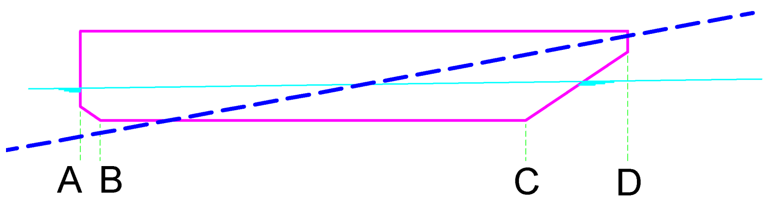
- [Digit], which enables to define the section shapes by digitizing, see paragraph 7.2.4.7.2 on page 178, Defining the section shape by means of a digitizer (tablet) and paragraph 7.2.4.7.3 on page 179, Defining the section shape by digitizing a BMP file for more details.
- [Shift], which you can use to shift a frame in vertical or transverse directions. To activate place the text cursor on the frame to shift, and press [Shift], which pops up a menu where you can enter the translations in both directions. After translation negative transverse values may occur, these have to be corrected.
- [inteRpolate] can be applied to interpolate additional frames in between two existing frames. This function is rather limited, for this kind of lines design it is much better to use the Fairway module. As acknowledgement of this limited functionality, the user should set the external variable *Frame_interpolate* to activate this function, see section 3.10.1 on page 30, List of external variables for details on that.
- [scaLe] can be used to scale the cross section, in the upper right window. The options here are 'maximum dimensions frame', in which case the frame under consideration is window-filling (with the X-axis drawn through the underside of the frame) or 'maximum dimensions hullform', with the X and Y-axes always going through the origin.

In the introduction to Hulldef a general description of the way of hull form definition has been given, see section 7.1 on page 165, Hulldef's hullform definition method. More detailed elucidation and tips are given below, on successively:

7.2.4.1 Number of frames

The maximum number of frames is with five hundred quite large. There is no strict minimum, but in general it is recommended for the hull to use at least twenty and preferably more frames. Even if the vessel has a simple shape, because also simple shapes can turn into complex submerged geometries under large heel and trim. In particular when there are operational conditions where under trim the deck is submerged in the aft or forward regions it is advised to use sufficient frames in those areas. These consideration does not necessarily apply to composed forms, which may be so short that two frames will suffice. It will be evident that using more frames will be beneficial for the computation accuracy (and detrimental for processing speed).

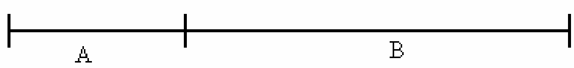
The frames constitute the basis of allPIAS' hydrostatic, stability and tank volume computations on basis of its wireframe model. In the essence, from and with these frames, intersections, areas and moments are computed, which by 'Simpson' are integrated volumetrically. Characteristics of waterlines, such as area and moments of inertia, are not computed directly from the hull shape, but are derived from the volumetric properties instead. In case of a limited number of frames, in combination with a large gradient in hull shape, this process may lead a bit jerky tendency of these waterline characteristics versus draft, The most important statement that can be made in this respect is that as such this does no harm, because in PIAS these waterline characteristics are never used in the calculations; they are just output, printed because sometimes people like to see these figures. Please see, as elucidation, the sketch below. Where for the determination of the waterplane area the frames between B and C. After all, these are intersected by the waterline, and A and D not. For the determination of volume, a correction is applied for the presence of A and D, however, this correction is not found back in the derived waterline characteristics. Only when the draft is sufficiently large to submerge the transoms at A and D, they will also be involved in the determination of waterline characteristics. So, if a fluent tendency of waterline characteristics is desirable, then a sufficient number of frames should be inserted between A-B and C-D. At large trims (the dashed waterline in the figure), the same effect will occur between B and C.



Side view of a pontoon.

7.2.4.2 Ratio of longitudinal frame distances

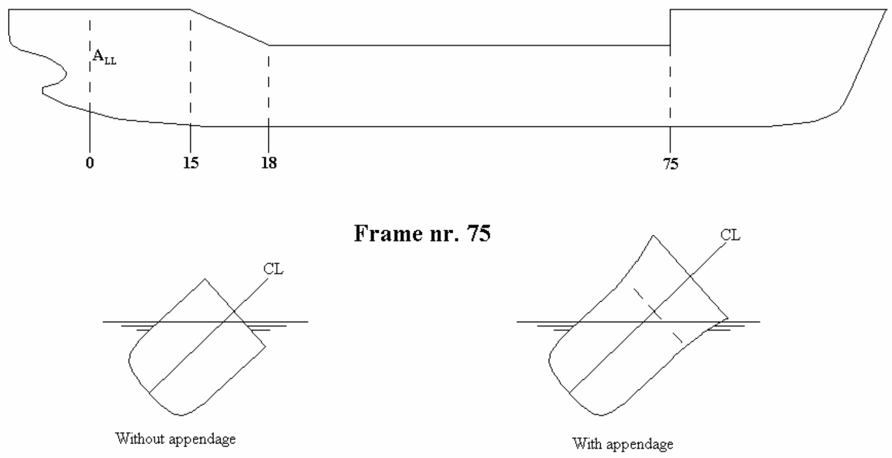
The distances between three successive points may not be smaller than 1:4, and not be larger than 4. If this condition is not met, the hullform can be saved anyway — it would be frustrating if that would be denied — however, each time when saving or reading the hull file a warning about this will be displayed. Take that warning at heart! If this condition is not met the accuracy of calculations might be impaired.



Frame distance ratio $\frac{1}{4} \leq$ A/B ≤ 4.

7.2.4.3 Double frames

As mentioned before, it is very important to define discontinuities in longitudinal direction by placing two frames on the same longitudinal position, (a double frame, two coinciding frames). The figure below gives an example of a ship with one discontinuity and two knuckles. This indicates we need three double frames at the positions 15, 18 and 75 metres from App. The two frames at 75 m are not alike to indicate a discontinuity there and the frames at 15 and 18 metres are alike to indicate a knuckle in the longitudinal direction.



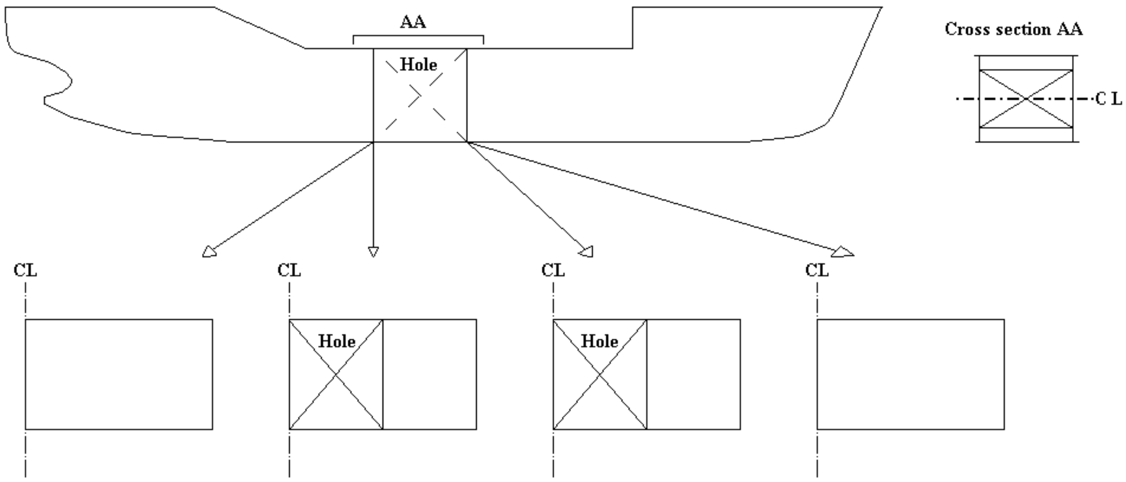
Frame shapes.

7.2.4.4 Deckhouses as appendage

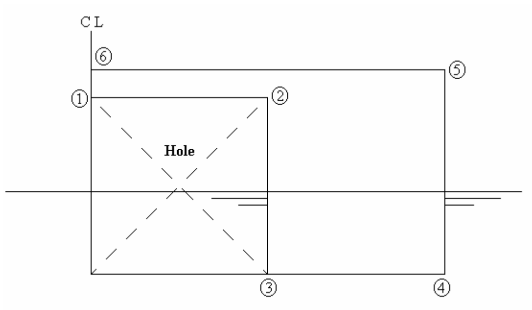
In general, the hullform is defined without superstructure. Deckhouses can be added later on with the option [appendages], see section 7.2.5 on page 180, Appendages. In order to define a deckhouse properly, their fore side and aft side must be situated on a double frame. In order to achieve this, (identical) double frames have to be defined on forehand, at the locations where later on with [appendage] the extremes of the deckhouse(s) will be situated. [Appendage] will extend one of these frames with the deckhouse shape, and the other not.

7.2.4.5 A hole in the hullform

The position of a hole (such as a moonpool) has to be defined by placing a double frame at the aft and forward sides of the hole. In this case a double frame consists of one frame without the hole (the regular frame) and one frame at the same position including the hole, see the figure below. The frames in the hole area are defined according the example in the second figure. The points 1 to 6 define the frame. The height-coordinates of points 1,2,5 and 6 are equal but drawn differently for clarity. The water level in the hole always equals the outside water level.



Hole at centerline.



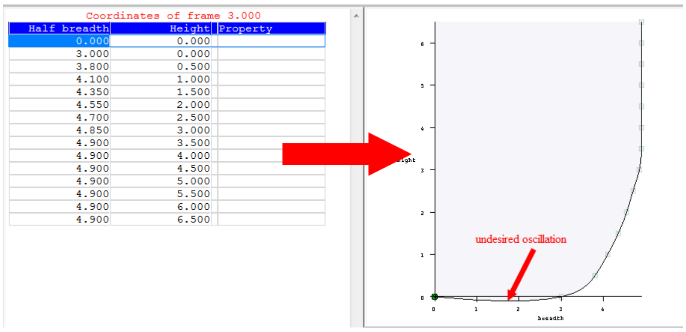
Frames in the hole region.

7.2.4.6 How to define the frame shape correctly

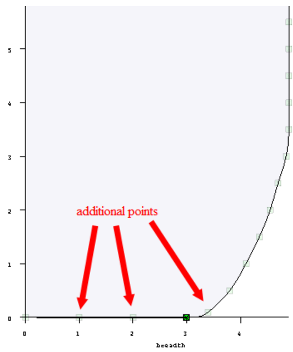
General tips are:

- The points of a frame should be defined in a counter-clockwise order (in the convention that the frame is situated at the right of centerline).
- The first point should be on the centreline. The last point on the deck in the side. Deck camber, deckhouses etc. are usually added afterwards as appendage, see section 7.2.5 on page 180, Appendages, however, they can also simply be incorporated in the frame shape by a continuation of defined points beyond the deck at side, extending to centerline.
- A minimum of two points must be defined. As a rough guideline, twenty points will suffice to define a relatively simple frame correctly.
- The maximum number of points lies in the hundreds, however, the application of much more points than required to define the frame shape properly is discouraged. For the reason that the more frames are used, and the more points per frame, the longer subsequent computations will take.
- Frames with a tunnel do not need special treatment; simply follow the frame line counter-clock wise, regardless whether it is ascending or descending. This also applies to catamaran or trimaran types of hull shapes. See e.g. the figure just above, which illustrates how to define frames over a hole at center line, which is likewise applicable to a catamaran frame.
- Through ordinary (=non-knuckle) points a *curved* curve is drawn, at a knuckle a new curve starts. This notion 'curved curve' should be taken literally; if for example a point is defined twice, the frame curve crosses that point indeed twice, resulting in a nice loop in that area. And a triple point will give two loops. It might be evident that multiple, coinciding, points should be avoided!

- At a discontinuity of curvature in the line it is advised to apply a knuckle point, even if this point is no knuckle in the strict sense of the word.
- In regions of high curvature, it is advised to apply more points, where the curvature is low less points will suffice, in general.



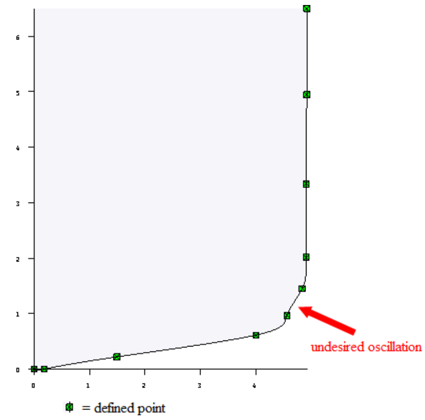
Unwanted undulation caused by too few points on baseline.



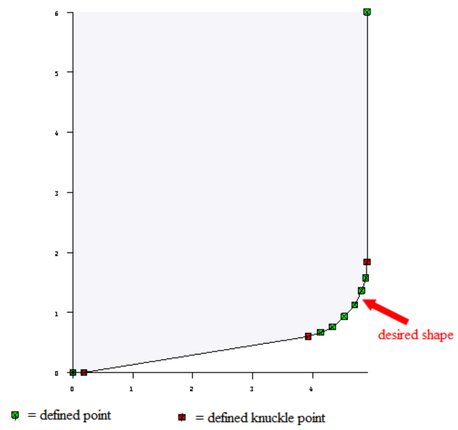
Proper curve shape with additional points on baseline.

In addition to these general rules, two pitfalls will be discussed here. The first one is the definition of a simple midship-like section, anyway, a section with a significant flat bottom part. If this is done by manually typing in the coordinates (re-typing a table of offsets) one might 'naturally' start with a point on baseline, followed by points on each subsequent waterline. In this case the effect might be that the frame will overshoot beneath the baseline, as illustrated in Unwanted undulation caused by too few points on baseline.. This can be prevented by inserting additional points on baseline, as depicted in Proper curve shape with additional points on baseline.. Alternatively, the point in the transition between bilge and baseline is defined as knuckle, in that case no additional points on baseline need to be given because the line between CL and bilge is between two knuckle and consequently strictly straight.

A second example is dedicated to the definition of the frame shape by digitizing. One might be inclined to tip a few points, more or less evenly distributed along the frame, however in this fashion regions of high curvature might be covered with too few points, leading to an unwanted undulation, as depicted in Unwanted undulations by too few points in region ofhigh curvature.". A solution is to digitize more points in highly curved regions, in our example the bilge area, although it is even better to apply two knuckles at the begin and at the end of this region. In that case the straight parts in bottom and side need no additional support, as illustrated in Proper curve shape by knuckles at both ends of the highlycurved region.".



Unwanted undulations by too few points in region of high curvature.



Proper curve shape by knuckles at both ends of the highly curved region.

7.2.4.7 Defining the shape of the frames

By placing the textcursor on a frame location, and pressing <Enter>, you reach the program part where the sectional shape (the shape of the frame) can be defined. For this task there are three alternatives, in the first place points on the frame can be defined by simply typing in their vertical and transverse coordinates, secondly the frame shapes can be digitized by an external digitizer, and thirdly a BMP file of the sections can be digitized from screen. Those three methods will be elaborated below.

7.2.4.7.1 Entering the breadth and height coordinates by keyboard

An input screen appears which displays two columns. The left column will contain the half breadth from centreline and the right column the height from the baseline. A fair curve will be determined through the given points, unless a point has been designated to be a knuckle, which can be done by placing the text cursor on that point and activating the [Knuckle] function. A knuckle is indicated in the coordinate table by the word ‘Knuckle’ after the coordinates. Ships are always fitted with an appendage (even if it is only a straight deck), at the starting point the word ‘Start appendage’ is displayed on the right; which indicates that the upper appendage begins at that point. The points of the appendage follow underneath and cannot be changed.

7.2.4.7.2 Defining the section shape by means of a digitizer (tablet)

The digitizer can only be used if it is switched on and connected to the computer from boot time. After the activation of the digitizing function a dialog enrols in a *Console Window*, which is temporarily located on top of the common PIAS window. It is important to steadily step through this dialog, without side steps to other activities.

In the first place the drawing should be taped on the digitizer, this does not necessarily have to be completely aligned, because the program corrects for rotation. Then the scale must be defined, for which the program asks:

- To digitize the origin (the intersection between centreline and baseline).
- To digitize a point on the baseline. It is advised to digitize a point as far as possible from the centreline for determining an accurate scaling factor.
- To enter the breadth and height coordinates of a single reference point. For advices on this point see the remarks on this subject in the paragraph below, on digitizing a bitmap file.
- To digitize this reference point.

Now that the scale is defined, the points of the frame are digitized by tipping of the digitizer stylus. On the digitizer five function keys are available which play a role in the digitizing process:

- <Kuckle>, which will make the next frame coordinate a knuckle point.
- <restart> to delete all coordinates of this frame and start again.
- <New scale> to define new scaling factors.
- <Do not save and quit>, which will stop the digitising process without saving the already digitized points.
- <Save and quit> to stop digitizing this frame and store the points

These function keys are part of a sticker that should be sticked to the digitizer, please refer to section 3.3 on page 27, Digitizer function keys for that.

7.2.4.7.3 Defining the section shape by digitizing a BMP file

Attention

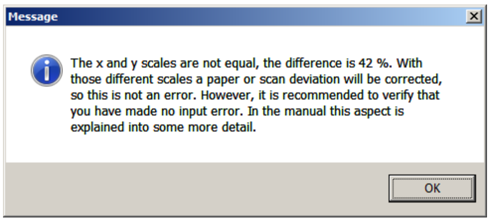
> It happens regularly that there is no file available in BMP format, but in another graphic format, such as JPEG, PNG or PDF. This is not directly usable in PIAS, and will therefore have to be converted. Each tool can be used for this purpose, here are numerous viewers and converters that allow you to do so -—. e.g. with Windows Paint, XnConvert[1] or XnView[2] — googling on “Convert to BMP” results in 494.000 *hits*.

In the first place the BMP file[3]) has to be selected by option [Window]→[Open]. Then the system of axes and the scale have to be set up, which can be done with this sequence:

- Digitize the origin, which is the intersection between base line and center line, with [Origin].
- Digitize a point on the baseline with [Baseline].
- Enter the breadth and height coordinates of a reference point with [Coordinates refpoint]). These coordinates are used to determine the scaling factors in breadth and height (which can differ, so there are two scaling factors). It is advised to choose a point as far as possible from the origin. The breadth of the scaling point must be given from CL, in general this breadth will equal **half the ship’s breadth**. If the frames to be digitized are situated left of CL (as in general will be the case with the aft ship) then also a scaling point at the left side should be used, however, the given breadth coordinate should still be positive. When changing from frames left of CL to frames at the right (so, from aftship to foreship) a new scale should be defined. These coordinates are used to determine the scaling factors in breadth and height.
- Digitize this reference point, by option [Refpoint].

With these parameters the scale of the BMP drawing can be determined. In principle the scale should be the same for all directions, however, in practice it may happen that the scale in X-direction differs slightly from the Y-scale. That may have been caused when copying the body plan, where one direction is stretched a bit and the other not. Another source can lie in the scanner which may treat the two directions differently. Anyway, by means of these different scales PIAS will nicely correct such a distorted BMP file. In case of different scales an informative message will be given, similar to the picture below. It is advised to verify the scale differences; a small difference could have been caused by a stretched BMP picture. However, in case of a large difference, such as the 42% in the graph below, it is likely that the digitized points (which are used for the scale determination) are not correct.

[1] https://www.xnview.com/en/xnconvert/
[2] https://www.xnview.com/en/xnview/
[3] http://en.wikipedia.org/wiki/BMP_file_format



Information message on different scales in X and Y directions.

Modus operandi and options are further:

- Digitizing is performed by subsequently touching points on a frame with the cursor, and then pressing the left mouse button for each point of which the coordinates should be sampled.
- Digitizing a knuckle (at the intersection with a chine) is similar, except that first the function [Knuckle] must be activated. This function *per knuckle*, which implies that when digitizing three knuckles subsequently, this [Knuckle] function should be chosen three times. In that case it might be more convenient to indicate the knuckle with a double-click of the left mouse button, for that is another option.
- With [Markers] the colors and types of the markers (which indicate different kind of points, such as origin or knuckle) can be set.
- With [Re-digitize] all digitized points of the frame are deleted, and the digitizing process restarts.
- [Stop without save] and [Stop] are assumed to speak for themselves. They apply to the frame under consideration.
- With the function [Frames] the aft and forward boundary can be defined which will be used for drawing the already digitized frames. Might be convenient for the context, and to indicate the digitizing progress.
- The function listed above are included in the top function bar, however, they can also pop up in a floating menu by pressing the right mouse button.
- With key <Delete> or function [Delete point] the digitized point closest to the cursor will be deleted.
- With [Window] and [Arrange] other bitmap files can be opened (it can for example be convenient to have separate bitmapfiles of aftship and foreship both available) or closed, windows can be re-arranged etc.
- Finally, this window offers the facility to zoom with the mouse wheel, and to pan by pressing the middle mouse button (or the mouse wheel) while moving the mouse.

7.2.5 Appendages

This option enables you to define **upper** appendages, with a maximum of 30. The appendages are automatically added to the frames for the stability calculations, but do not form a fixed part of it. In the frame definition menu, the points of the hull form and the points of the appendages are visible, but only the points of the hull form can be changed. When the frames are printed, both the frame points and the points of the appendages, if the frame is within its boundaries, are printed.

Please be aware that an appendage, such as a deck house, can be the cause of a longitudinal discontinuity, in which case in the hull form definition a double frame will be justified, see section 7.2.4.3 on page 175, Double frames for more details.

Attention

Appendages are always symmetrical around centerline. Asymmetrical appendages can be obtained with the asymmetrical hull form option, see section 7.2.2 on page 172, Hullforms for more details.

When entering this appendage definition menu, a list of all present appendages is presented, which shows their most important properties. The possible action here is:

- [Remove from hullform] : In PIAS from before 2020, the points of the appendages were added to the specified frame points. If this is still the case with an old model, this option can be used to remove those appendage points. The defined appendages are then added to the shape of the ship in the new way. The appendages of such an old model can only be changed by first removing the appendages from the frame shape.
- By pressing <Enter> a new menu appears where all appendage particulars can be given, similar to the menu depicted below. In this menu the first line shows the type of appendage, and the second line the 'description', which is just a textual line for your own reference. The other lines depend on the type of appendage, of which five types are available:

Type appendage	Rectangular appendage
Description	tankdome1
Aft location appendage	37.000
Forward location appendage	41.000
Half breadth	2.500
Height	14.950

Appendage definition menu.

- Deck camber.
- Deck slope.
- Rectangular upper appendage.
- Trapezoidal upper appendage.
- Upper appendage parallel to deck at side.

If no upper appendages are defined the vessel will be closed with a flat deck. For each appendage a few particulars have te be given, which are discussed below per type of appendage:

7.2.5.1 Deck camber

- Aft location of the camber. If the camber stretches itself over the entire vessel a large negative value, e.g. -100, should be given.
- Forward location of the camber. In case of a camber over the entire vessel, simply a large value should be given.
- The deck camber is given as divisor on the local breadth at deck, in other words as X for a deck camber of breadth/X. So for a camber of 1/50th of the vessel's breadth the value '50' should be given.

7.2.5.2 Deck slope

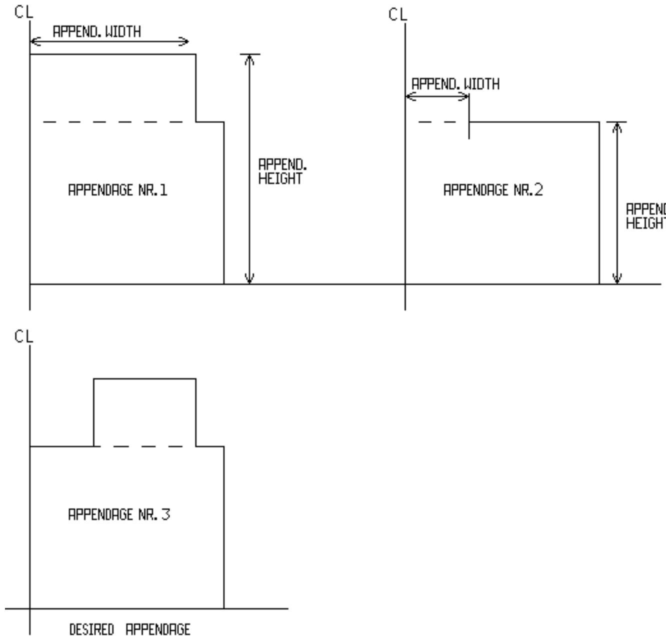
- Aft location of the deck slope.
- Forward location of the deck slope.
- The deck slope is defined as X in deck slope=breadth/X, similar as with the deck camber.

7.2.5.3 Rectangular upper appendage

- Aft location of the appendage.
- Forward location of the appendage.
- Half breadth (from centerline) of the rectangle.
- Height (from base) of the rectangle.

It is also possible to place two deckhouses on top of each other. For example, a deckhouse not extending beyond centerline can be defined as follows:

- First define the deckhouse over the entire breadth.
- Then define a deckhouse that is exactly the part to be removed with a height equal to the deck. See figure underneath. Appendage 1 is the deckhouse over the entire breadth. Appendage 2 is the appendage which defines the part to be removed. Appendage 3 is the resulting one.



Two deckhouses on top of each other.

7.2.5.4 Trapezoidal upper appendage

The trapezoidal upper appendage is similar to the rectangular type with the exception that the forward and aft locations can have different dimensions.

7.2.5.5 Upper appendage parallel to deck at side

This type of appendage is used to define a deckhouse with a constant distance to the deck at side.

- Aft location of the deckhouse.
- Forward location of the deckhouse.
- Distance from deck at side to the deckhouse.
- Height aft (from base).
- Height fore (from base).

7.2.6 Wind contour

With this option the shape of wind contours (=windage areas) can be given, with a maximum of twenty contours. The reason for defining **multiple** contours may be that a particular loading, such as container or a deck cargo of wood, might increase the windage area. For all those contours different wind moments can be computed, which e.g. also might lead to different values of maximum allowable VCG per wind contour. By the way, the Loading module offers specific submodules for specific types of cargo, such as for containers, where the **actual** loading is taken into account at the wind heeling moment computations. If this facility is used, there is no necessity to define standard contours for that cargo, here in Hulldef. Anyway, if the *wind contour* option is selected here, a menu similar as the one presented below appears:

Collection of wind contours

Selected	In 3D view	Name wind contour
Yes	No	Without containers
Yes	No	With 1 layer of containers
Yes	Yes	With 2 layers of containers

The 'selected' column is only significant for the output of the contour in Hulldef. In subsequent computations, such as the maximum allowable VCG in intact or damaged condition, for which the stability regulations may contain a criterion for the effects of wind or weather, contours can be selected separately. The 'in 3D view' defines which of the contours is included in the three-dimensional view, at the right bottom window as well as on paper. In this menu two additional functions are available:

- [shifT], which facilitates in shifting a contour in length and/or height.
- [Merge], which facilitates in merging two contours. The selected contour will be merged with the copied contour.

Attention

This option is rather limited; the subcontours of the two merging contours are simply stacked after each other, and that is it. SARC also realizes that there are many ways to add 2 contours together, but they are not implemented here.

With the <Enter> key the program goes one level deeper, 'into' the list of subcontours, a menu similar as the one presented below appears:

Subcontours of wind contour With 1 layer of containers

Name subcontour	Resistance coefficient
Ship without deckload	1
1 layer of containers	1

For each subcontour a resistance coefficient can be specified that is used in the calculation of the wind moment. Overlapping subcontours are also double charged!

In this menu two additional functions are available:

- [Digit], in order to digitize a subcontour from bitmap file or by digitizer. This is analoguous to digitizing ordinates, for which reference is made to paragraph 7.2.4.7.2 on page 178, Defining the section shape by means of a digitizer (tablet) and paragraph 7.2.4.7.3 on page 179, Defining the section shape by digitizing a BMP file.
- [shifT], which facilitates in shifting a subcontour in length and/or height.

With the <Enter> key the program goes one level deeper, 'into' the list of longitudinal and vertical coordinates of the subcontour. This list can be edited or extended. The complete contour should be defined **including the underwater body**. The last point should coincide with the first, so that a closed contour is defined.

In this menu the following function is available:

- [File], with which a subcontour can be exported or imported. The default name of the file is the name of the subcontour, but can be changed manually. The extension of the file is automatically '.sh'. The file format is as follows:
  - The total number of points of the contour.
  - Contour point column headings: 'Distance ALL Distance base'
  - And then of all the points the length and height coordinates.

7.2.7 Wind data sets

For an integral discussion on wind moments reference is made to chapter 14 on page 273, Wind heeling moments.

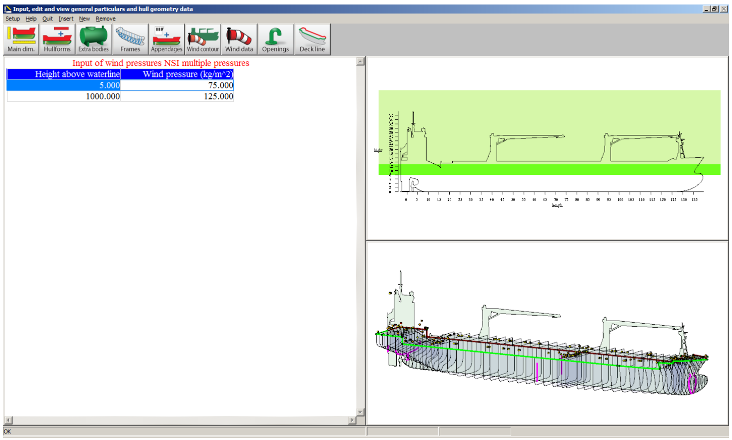
With this option wind data can be specified, such as wind pressures. These particulars are not specifically linked to one contour, therefore they are given here 'loosely', so that for all combinations of contours and wind data sets computations can be made of wind heeling moments. First, at this option a list comes up which contains the available sets of wind data, up to a maximum of twelve:

Collection of wind heeling moments

Selected	Name wind data
Yes	IMO intact stability
Yes	For damage stability, coastal zone 3a

The <Enter> key brings the user 'into' the set of wind data on which the text cursor rests at that moment, where the following menu is shown:

- *Windlever must be calculated about*, where the choices are *half draft*, *C.O.G. of the underwater body* and a *fixed height*. Commonly, the wind moment is calculated about the C.O.G. of the lateral area of the underwater body, however, occasionally it is required to calculate the wind moment about draft/2 or about some specific height (for example the centre of thrusters in a dynamic positioning system).
- If on the first line the type *fixed height* has been chosen, on the second line, at *height whereabout the moments must be determined*, that height can be given.
- At *input/edit wind pressures* the wind pressures (in kg/m^2) up to a certain height **above the waterline** should be specified. The upper height of the highest wind pressure region should exceed above the topmost part of the contour. The safest way to achieve this is to specify a thousand meter as upper limit of the highest pressure region. The figure below gives an example of the use of two wind pressure regions, and it shows that the different wind pressure regions are indicated by different intensities of green.
- The goal of this [wind data] function in Hulldef is to specify the parameters for which the wind moment calculations can be executed. However, by exception the moments or levers may already be available from another source, for example from CFD simulations or wind tunnel experiments. In that case the wind levers (in meter) can be entered at the option *user-defined wind heeling levers*, where, after pressing the <Enter> key, an input menu appears where for each wind contour a table of heights/levers can be entered.



Two regions with different wind pressures.

7.2.8 Openings

An input window appears, containing a maximum of 255 special points, such as non-watertight openings, where the columns have the following usage:

- Name: an identification for this point.
- Length: the longitudinal distance from App of this point.
- Breadth: the transverse distance from centreline, where PS is negative and SB is positive.
- Height: the vertical distance from base.
- Type, which depicts the specific meaning of the point:
  - Watertight: this is not an opening in the true sense of the word.
  - Weathertight: this type of opening can withstand adverse weather, however, not permanent submersion.
  - Margin line: a point of the margin line, which may play a role in damage stability regulations. Please consider that for accurate modelling many points of such a margin line should be defined.
  - Open: a non-watertight opening.
  - Point of a horizontal evacuation route: this type of point is applicable with probabilistic damage stability calculations, according to SOLAS 2009 & 2020, for passenger vessels. For details reference is made to reg. 7-2, par. 5.2.2 of chapter II-1 of SOLAS 2009 & 2020.
  - Non-watertight opening (V-line): some naval stability criteria, notably DDS-079, apply the concept of *V-lines*, so when working with a PIAS version equipped with these criteria, a point can be declared to be of the 'V-line type'. These V-line points are available in two variants, 'non-watertight' and 'watertight door'. For the difference and background of these concepts in detail reference is made to the regulations from which they originate.

  - Watertight door: some naval stability criteria, notably DDS-079, apply the concept of *V-lines*, so when working with a PIAS version equipped with these criteria, a point can be declared to be of the 'V-line type'. These V-line points are available in two variants, 'non-watertight' and 'watertight door'. For the difference and background of these concepts in detail reference is made to the regulations from which they originate.
  - Emergency exit: see section 15.5.1 on page 294, Types of parameters, **Emergency exits to be included**.
  - Vertical escape hatch: this type of point is applicable with probabilistic damage stability calculations, according to SOLAS 2009 & 2020. For details reference is made to reg. 7-2, par. 5.3.1 of chapter II-1 of SOLAS 2009 & 2020.
  - Buoyancy point: This point is used to calculate the freeboard to this point. The point with the smallest value then determines the 'Buoyancy index'. This index is used in some naval stability criteria.

  Ventilation openings may be protected with so-called tank vent check valves (a.k.a. ventilation caps). The categorization (i.e. watertight, weathertight, open) of a particular type or brand of such a device depends on its construction, certification and interpretation of the particular authority. So, unfortunately, no general guidance can be given in this respect.
- Compartment: each opening can be connected to a compartment. In general, the rules for flooding of a vessel, when an opening is submerged, apply, however, if an opening is connected to an already flooded compartment it is, obviously, neglected. Openings connected to compartments can only be managed in Layout, (as discussed in section 9.7.1 on page 233, List of openings and other special points) since they *are part of* their compartment. Such Layout-administered openings are also shown in this menu, however, in order to avoid inconsistencies they are not editable here — and are as indication of this nature printed in grey.

Furthermore, this menu contains two auxiliary functions:

- [+/-], which toggles the sign of the transverse coordinate (so the point switches from PS to SB or vv., does unfortunately not work for an entire selected column).
- [Sort], in order to sort the openings, which can be done in two sequences, either on type of opening, or on longitudinal position.

7.2.9 Deck line

For the calculation of intact and damage stability, and to verify against the intact or damage stability criteria, the program sometimes needs to know the deck line, which is by default not defined as such in PIAS. The first time this module is used on a ship, the deck line is derived from the hullform, at least, if a hullform model is available in conventional PIAS format. The deck line generated in this way could be erroneous, because it is assumed to be simple the connection line between the last points of all frames. Therefore, it is advised to check the deck line thoroughly. With this option it is possible to check or edit the coordinates of the deck line. A menu appears with longitudinal, transverse and vertical coordinates of the deck line. Besides the usual editing facilities, a couple of additional functions are available:

- [Derive] to (re)generate the deck line based on the PIAS frame model. Please realize that this **may** be an option to quickly derive the deck line from the hull shape, but it is certainly not a must to do it this way; after all, if that would be the case this action would have been automated. Nor is it at all necessary to give the deck line as many points as the ship has frames, e.g. for a ship with a curved hull shape and a rectangular deck without sheer or deck line jumps it is quite possible to define the ship shape with a hundred frames, and to define the deck line with only two points (i.e. aft and forward).
- [Switch side (PS/SB)] to toggle between the SB and PS deck line. With an asymmetrical vessel these could differ.
- [Mirror] to copy all points of the deck line from this side to the other, so from SB to PS, or from PS to SB.

Attention

The deck line, as defined here, only plays a role in the assessment against a number of (damage-) stability criteria. It has no effect on hydrostatics and stability etc. as such.

7.3 Output of hull geometry data

With this option all defined hull-related data can be outputted to printer or text editor file (e.g. to be imported into Word), with a preview to screen. Each option of the menu below produces specific output for a single data type, with the exception of wind data (as discussed in section 7.2.7 on page 183, Wind data sets). Printing these wind data would be a bit overdone, because these data are already printed at the wind lever calculations where they are applied. The last option is not aimed at a specific data type, it can be used to produce combined output for multiple data types instead.

Output of hull geometry data

1.	Main dimensions
2.	Coordinates of all frames
3.	Two dimensional output of hullform
4.	Three dimensional output ship
5.	Frame location table
6.	Openings and margin line points
7.	Portside and starboard deck line points
8.	Selected wind contours
9.	Combined output

7.3.1 Main dimensions

This option prints a table which contains: main dimensions, added hullforms (as specified in section 7.2.2 on page 172, Hullforms) and appendages.

7.3.2 Coordinates of all frames

With this option all defined points of all frames are printed. A **K** with a coordinate indicates this being a knuckle point. With this option the coordinates of the frames of other hullforms — from which the input is discussed in section 7.2.2 on page 172, Hullforms and section 7.2.3 on page 173, Extra bodies — can be included as well. If desired, this can be achieved by setting the 'Print' column to 'yes'.

7.3.3 Two dimensional output of hullform

With this option several two-dimensional hull shape plots can be produced, such as:

- Body plan of foreship and aft ship, separately.
- Body plan of foreship and aft ship combined.
- A schematic lines plan. This 'lines plan' is rather schematic indeed, for example waterlines and buttocks are interpolated as good as possible, but because the sectional model of PIAS holds little information about the extremes of these longitudinal lines — which are not required at all for accurate computations — these extremes are omitted in the lines plan. Another simplification is configurations and colors are not configurable by the user. For a flexible definition and production of a high-quality linesplan the Fairway module is recommended, see section 6.9 on page 145, Define and generate lines plan.

With the first sub-option of this option a number of plot parameters can be given. The first four (on frame numbering, upper appendages and the division between aftship and foreship) are applicable on all 2D plot variants, all other parameters, on waterlines and buttocks, are only applicable to the schematic linesplan.

7.3.4 Three dimensional output ship

Here the three dimensional output of all entities defined in Hulldef can be configured and produced. This output comes in two flavours:

- As lines model. This output is intended to be included in for example a stability booklet, and for that reason the output parameters are explicitly configurable here, and stored to produce exactly the same output format a next time — for example in case of a sister vessel or a design modification. The output parameters include:

  - Whether the plot should be made including openings, deck line, wind contour etc. If set to 'no' then only the frame lines are included in the plot.
  - The viewing direction of the ship. These are the viewing angles according to the convention of section 2.6 on page 9, Definitions and units, which can, incidentally, be read at the bottom right in the window of the rendered model, as discussed just below.
  - Whether the projection is in perspective, and, if so, the distance between the eye and the origin.
- As *rendered model*, please refer to the example and discussion of section 7.8 on page 189, Rendered views.

7.3.5 Frame location table

This option prints a table of frame positions (in meter from APP), which contains the range between the aftmost and foremost frame, as specified at section 7.2.1.3 on page 167, Frame spacings.

7.3.6 Combined output

With the other options of this menu individual data types can be printed. However, it might be handy if multiple data types can be specified, and printed in a single action. In particular, in case of a design change the whole bunch of data can be printed with a single command, without bothering about the output sequence and chapter numbers. This background is the reason of existence of this option, which allows in multiple rows to specify the data type chapter name and page number. With menu option [Print] the output in actually produced.

7.4 Export of hull shape data to a number of specific file formats

With this option the *frames*, as defined in PIAS, can be exported. In this respect one should realize that the exported data is in principle limited to the available data, which are, in PIAS, frames. with an accuracy (more than) sufficient to perform naval architectural computations. However, they do not necessarily on construction tolerances. Earlier PIAS versions, from the 1980s and 1990s, also contained frame conversion facilities to CAD software such as Autocad, however these functions have been scrapped with the rise of Fairway, which allows a far more accurate and above all more complete (e.g. including waterlines, stem and stern contours and surface models) definition and export. For this reason, the export from the Hulldef hull form is now limited to:

- PIAS-standard ASCII text file of sections, from the format as described in section 7.5.1 on the following page, The "PIAS standard" text file format.
- Poseidon, a DNV•GL rules program. PIAS also contains an option to export the *internal* geometry to Poeison, more information on that can be found in section 9.11.6 on page 243, Export to Poseidon (DNV•GL).
- Castor[4], the steel weight estimation program by ASC.
- Frames to Eagle, as in use by Conoship.
- Frames to Fredyn, Shipmo and Precal, for ship motion analyses, from MARIN, www.marin.nl. The Shipmo conversion is aimed at 2006-2016 Shipmo versions.
- Wireframe model to Fredyn. Fredyn imports two representations, the first is a frame (sectional) model, as generated with the previous export option. The second is a wireframe model, from this particular option. The wireframe is more complete (because additional longotudinal connections are constructed), however, the hullform should satisfy a number of constraints in order to be able to use this function.
- Seaway, a ship motion program by Amarcon, www.amarcon.com.

7.5 Import frames from (a number of specific formats of) a text file

With this option cross sections (ordinates) from a text file (ASCII file) can be converted to PIAS format. On this process the following comments can be made:

- The recommended definition method is through the modules Hulldef (for existing body plans) and Fairway (for new designs). Nevertheless, this facility might prove to be rather useful for importing body plans from another source.
- Here, no CAD files can be imported here. Such an option — notably from DXF and IGES formats — is indeed available in PIAS, however, located in the Fairway module (see section 6.3.7.2 on page 116, The entire procedure to import DXF or IGES files).

[4] http://asmussolution.nl/EN/castor.pdf

- This conversion routine does not perform any validity check. **So the user should verify that the information in the input file is complete and correct**, on two levels:
  - Syntactically, which means that the numbers in the text file have the correct format (such as a decimal point instead of a comma) and are placed on the right position in the file.
  - On content. The definition of a PIAS hullform must comply to a number of requirements, which are listed in section 7.2.4 on page 173, Frames (frame positions and frame shapes), such as the frame order (from aft to fore), the maximum number of cross sections and the maximum number of coordinates foreach cross section, the sectional distance ratio and possibly the 'double frame' requirement.

After starting the import function, a short input menu with 4 options appears:

1. Import file name. The file name as specified here is not retained. That is deemed unnecessary because, as a rule, an import action will only be undertaken once per project.
2. File format import file. Only the formats 'PIAS standard' and 'XML' are actively supported. The other formats have once been developed for specific users or applications, however, they are not documented — anyway, not in the context of PIAS.
3. Hullform type. Here you can select what kind of hullform should be added; a symmetrical main hullform, added hullform, etc.
4. Import as polylines. If this option is set to 'yes', the frames are imported as polylines (= chain of straight line segments between frame points) and cannot be modified in PIAS. However, the shape can be used for further definition and calculations. The advantage of the polyline is that its number of points per frame is unlimited, whereas in the standard way a limitation applies (incidentally, in general to more than 100 points per frame, so not that tight). The [Import] function actually reads the file and adds it to the list of shapes set at hullform type (one line higher in this menu).

7.5.1 The "PIAS standard" text file format

A file of this format essentially consists of a number of cross sections, with for each cross section a number of points on the hull surface for that cross section. So, this information is completely equivalent to that as is defined with Hulldef. The file should have this content:

- Number of cross sections (frames).
- For each cross section:
  - Location from Aft Perpendicular (APP).
  - Number of coordinate pairs.
- For each coordinate pair the breadth and height of this coordinate, and a **1** if this coordinate is a knuckle, or a **0** to indicate a smooth curve through this point.

An 'PIAS standard' file for a rectangular barge with main dimensions 100 x 20 x 10 m, with two cross sections will for example have the following content:

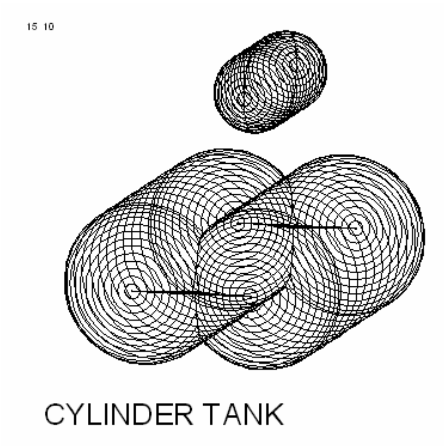
```
2
0.000
3
0.000   0.000   0
10.000  0.000   1
10.000  10.000  0
100.000
3
0.000   0.000   0
10.000  0.000   1
10.000  10.000  0
```

7.6 Generate cylindrical shapes

With this option cylindrical shapes, notably a longitudinal gas tank with a more or less circular sectional shape, can be defined parametrically, and converted into a PIAS sectional representation. The main cylinder parameter is its orientation: longitudinal, transverse or vertical. For a longitudinal cylinder the other parameters are:

- Outer radius, the cylinder radius.
- Cylinder extremities: the aft and forward location.
- Cylinder axis above base: height of the center of the cylinder above baseline.
- Cylinder axis from CL: the transverse distance of the center of the cylinder from center line.
- Type of tank head aft and forward, for which three types are available:
  - Circle head.
  - Korbogen head (R= 0.8 D).
  - Deep dish head (R= 0.714 D).
- File name of PIAS frames: name (and path) of the PIAS hull shape file name.

For transverse and vertical cylinders, the parameters are similar, with the exceptions that these types can be hollow, while the specific types of tank heads are not applicable to these types. With the defined parameters a hull shape in standard PIAS sectional representation can be generated, which can e.g. be applied as 'external compartment shape' at the compartment definition module Layout. Below an example of such a representation is depicted.



Cylindrical tanks.

7.7 File and backup management

Backups of the hull-related data can be made and restored here. Here is also the option 'Stop without saving'. See for the details section 2.9 on page 15, Data storage and backups.

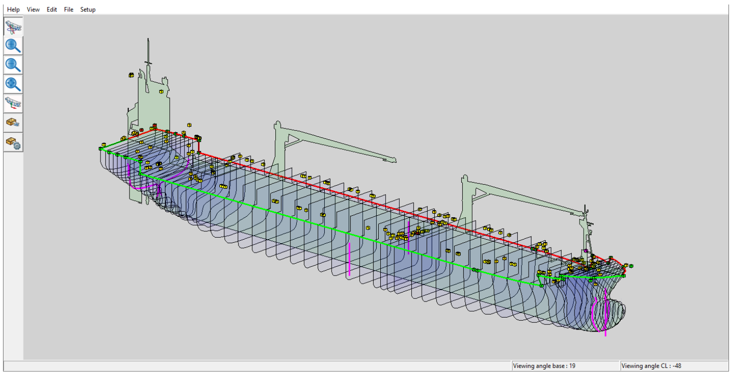
Attention

These file and backup management options are applicable to data as administrated under the current project and which are consequently saved under the project file name. However, other hullforms or bodies, as discussed in section 7.2.2 on page 172, Hullforms en section 7.2.3 on page 173, Extra bodies, are essentially independent projects with its own file name. These are consequently not included in this file and backup management system.

7.8 Rendered views

In a number of places in PIAS a system is used where the three dimensional shape of curves and/or surfaces is plotted in a *rendered* fashion, which means that hidden surfaces are not shown, that lighting can be applied etc.

This option is here in Hulldef available with the output, please see section 7.3.4 on page 186, Three dimensional output ship, but is also used other modules, particularly in Layout and Fairway. For that reason, the options of this system is discussed here in a separate section. From the *rendered* view in Hulldef an example is depicted below, which shows section curves, openings, deck line etc., exactly those elements which have been specified to be included in plots with option [View] of the input menu of this module, see section 7.2 on page 166, Input, edit and view general particulars and hull geometry data.



Three-dimensional rendered, output.

At the left side in each three-dimensional subwindow is a number of buttons that are specifically related to that subwindow:

- Rotate: assigns the rotation function to the right mouse button. This is the standard.
- Zoom+ and Zoom-: zoom in and out. It is also possible to zoom dynamically with the mouse wheel.
- Extend: zoom to full-screen.
- Pan: assigns the dragging function to the right mouse button. By pressing the mouse wheel button permanently one can also drag without using this function.
- Clip: one can *clip* the 3D subwindow i.e. that one can define a hexagonal box, where only the contents of that box is visible. With this *clip* function one switches the *clipping* on and off.
- Setclip: six clip boundaries can be set with this function. When this function is active, the hexagonal box appears, a bit transparent, and by standing on one side and by pressing the right mouse button permanently one can drag that side.

Actually, these buttons are shortcuts to functions of the upper menu bar, which will be discussed in the next paragraphs. By the way, not every render window is equipped with those lefthand buttons, because in some occasions functions have been left out because of irrelevance in the context of that module. However, **if** the functions are present they always act the same.

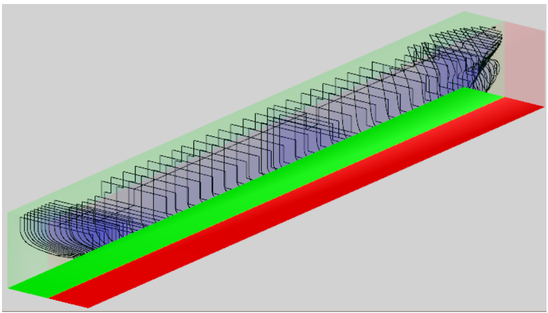
7.8.1 View

Most of the functions here, such as zoom and rotate, have been discussed right before. Additionally, this [View] menu contains an option [(In)Visible] which can be used to make certain parts visible or invisible, but that will speak for itself. More can be said about the dedicated function [Orientation box], which finds its background in the fact that although for a rendered view a viewing direction can be chosen, an orthographic (= non-perspective) projection of a lines model can be ambiguous; the viewing direction has been set; however, it is not evident from which side of the viewing axis the object is viewed. So, such a projection can always be seen from two directions, somewhat similar to the ‘optical illusion’ which can include totally different images into one picture, as illustrated with a well-known example below.

Optical illusion: young or old woman?

Although this is a principal phenomenon, PIAS has two tools which may assist in finding the right orientation. In the first place the viewing angles are often shown, according to the convention of section 2.6 on page 9, Definitions and units. And secondly, the *orientation box* can be switched on, which gives a PS part of the object a red envelop, and the SB part a green envelop. The top and side planes are transparent, but the bottom plane is opaque, so if this plane covers a part of the object it will be clear that the viewing direction is from below.

These tools do not **guarantee** a correct mental image of the picture — the viewer should also be **prepared** to set his mind's eye properly — but experience has learned that they are a good aid.



Orientation box.

7.8.2 Edit

With this function the position and intensity of external light sources can be set. One can also change the colors (as well as reflection characteristics and transparency); when the mouse pointer is standing on a part of a ship then its color can be adjusted; when the pointer is standing on nothing at all, then the background color is adjusted.

7.8.3 File

7.8.3.1 Save image in file

With this option the screen image will (two dimensionally) be saved in bitmap format. here a reduction factor can be given from which the background is discussed in section 7.8.3.3 on this page, Print image.

7.8.3.2 Copy to clipboard

With this option a copy of the image is made to Windows' clipboard, which can subsequently be pasted into documents of other applications.

7.8.3.3 Print image

After the selection of this option an input box pops up, where a reduction factor can be given. This is a bit of a technical parameter, which had to be included. For, when a printer is configured to print at a high resolution, a very dense picture will be produced. The file size of these pictures can be fairly large, which should pose no problems as such, however the sad experience has learned that Windows is not always able to cope, which may lead to a program crash. The use of this reduction factor reduces the picture size as well as the probability of failure significantly.

7.8.3.4 Generate VRML file

In a VRML (Virtual Reality Modelling Language)-file a 3D representation of the model is captured. This representation can be visualized using a VRML-viewer. Various VRML-viewers are available as shareware on the Internet. The options [File]→[Generate VRML 1.0 file] and [File]→[Generate VRML97 file] respectively produce a file according to the original VRML 1.0 specification, and the more recent, and more popular, VRML97 format.

7.8.4 Setup

7.8.4.1 Select Nearest

With this option *on*, while selecting an object always the thing closest to the eye (of the viewer) is taken. However, occasionally a deeper object might be intended to be selected. With this setting *off* a list pops up containing all objects on the cursor location (regardless of their depth) from which the user can chose.

7.8.4.2 Auto apply

Some options from the [Edit] menu have an [Apply] button, which will make modified visualisation settings to be processed. However, when [Setup]→[Auto apply] is set, each modification will immediately be processed. This gives a much more responsive character, however, at rare occasions a computer may be too slow for *auto apply*

Chapter 8

Hulltran: hullform transformation

With this module a transformation is applied on a hullform which is already available in PIAS format. This original hull is called the parent hull, and the transformed one the daughter. With this transformation the following shape parameters can be changed:

- *Length, breadth and draft. This is simply a matter of multiplication by linear scale factors.*
- *The length of the parallel body.*
- *Block coefficient.*
- *Longitudinal centre of buoyancy.*
- *Midship coefficient.*

The parent only serves as source and remains unchanged. The daughter form lives under its own filename and is after transformation not connected to the parent in any way. Apart from this main task, this module offers the option to combine distinct aft ship and fore ship hulls. However, this is a bit of a side issue.

8.1 Main menu

Hullform transformation

1.	Transform hullform
2.	Change length of parallel midbody
3.	Combine two ship hulls (aft ship and fore ship)

8.1.1 Transform hullform

This option is used for the full blown transformation, where the hull as such is distorded. The applied transformation method is the same as 'inflate/deflate' in Fairway, see section 6.5.3.3 on page 131, Inflate/deflate frames for an introduction. With this option a sub menu appears with just two choices:

Hullform transformation

1.	Enter main dimensions and coefficients of the transformed vessel
2.	Perform the transformation

8.1.1.1 Enter main dimensions and coefficients of the transformed vessel

Here the following parameters can be given:

- The filename of the transformed form. Here the filename of the daughter form should be given (which must, obviously, differ from that of the parent. Also here the '&'-character can be applied, which makes the daughter to be written in the same directory as the parent (identical to the '&'-facility of composed hullforms, as discussed in section 7.2.2 on page 172, Hullforms).
- Name. This is simply a textual description attached to the daughter form.
- Length.
- Breadth.
- Draft.

- Block coefficient, with a maximum modification of ±0.05.
- Longitudinal centre of buoyancy in % van L_{PP}, with a maximum modification of ±4%.
- Midship coefficient, with a maximum modification of ±0.02.

8.1.1.2 Perform the transformation

Which make the transformation to be applied. In order to memorize the applied parameters a single page with the existing and the new hull form parameters is printed.

8.1.2 Change length of parallel midbody

Change length of parallel midbody

1.	Enter parallel midbody particulars
2.	Perform midbody modification

8.1.2.1 Enter parallel midbody particulars

The change of the parallel body will be executed *forward* of the last frame in the aftship (For the division between aftship and foreship reference is made to [Aftship] in section 7.2.4 on page 173, Frames (frame positions and frame shapes). In thi smenu should be given:

- For 'filename' and 'name' we refer to the discussion in section 8.1.1.1 on the preceding page, Enter main dimensions and coefficients of the transformed vessel.
- The length to add. There are no principal limitations on increasing the length of the parallel body. A decrease of the length of the parallel body is defined by entering a negative length. This decrease is limited to half the length of the parallel body.
- Whether the vessel has a sloped keelline. In that case on APP and FPP the heights above base of the (moulded) keelline should be given. The program will then shift the ordinates vertically, in order to to match the slope of the keelline.
- Shift baseline to intersection keelline - half length. If 'yes' is given then the baseline is shifted in a fashion that, if the baseline intesects the keel line at $L_{PP}/2$, with the parent, then this will also be the case with the daughter. Whith 'no' the baseline will keep its position with respect to the *aftship*.

8.1.2.2 Perform midbody modification

With this option the daughter form is actually generated (a page with parameters changes will be printed here too).

8.1.3 Combine two ship hulls (aft ship and fore ship)

With this option two hull forms can be combined, in the sense that the aft ship of one hull form, and the fore ship of another will be glued together in a single new file which contains the combined hull. However, this a bit of a fringe, a single time this function may prove to be convenient, however, those are the exceptions. For normal use PIAS also offers facilities to use multiple hull forms. Those are not aggregated into a single file, instead they are treated in stability calculations (and the like) as a rigid combination. This mechanism, which offers more flexibility than explicitly combininghull forms, is discussed in section 7.2.2 on page 172, Hullforms.

The operation of this option is assumed to be evident.

Chapter 9

Layout: Design and utilization of the ship's layout

Layout is the PIAS module with which the internal geometry of the ship is recorded, managed and used. It goes without saying that that internal geometry can consist of bulkheads, decks, compartments and other spaces, but it may also contain additional data, like the weight group of the volume of a specific compartment, or the sounding pipe geometry. A description of the background of Layout can among others be read in the `Compit'11 paper`[1]. *Briefly, Layout offers the following modelling possibilities:*

- *Defining compartments through compartment boundaries.*
- *Defining continuous bulkheads and decks, between which the compartments are formed.*
- *Support by means of reference planes, to which compartment coordinates as well as bulkheads and decks may refer.*

The first two methods are mutually convertible, which means that one is able to convert from bulkheads/decks to compartments as well as vv. Briefly, Layout offers, furthermore, the following functions in the field of internal geometry:

- *Calculation of tank tables, trim correction tables etc., in a variety of formats.*
- *Output of a schematic tank plan and 3D views of compartments.*
- *Definition of the layout of a 2D subdivision plan, and its output to paper, bitmap or DXF file. This subdivision plan may function as basis of the general arrangement plan.*
- *Function as server of internal geometry, which is able to respond to requests of other software applications. So, for example, the shape of a deck or of a compartment can be made available to other (CAD)software upon request.*
- *Import and export of the internal geometry in XML format.*

9.1 Definitions and basic concepts

9.1.1 Definitions

Plane
: A plane is endless, and can have any position in the space, can therefore also be angled. But every cross-section of a plane is straight, so it cannot be curved or twisted.

Physical plane
: A physical plane is a plane which can be limited, and can be the separation between subcompartments. As a rule, physical planes are used to model bulkheads and decks.

Reference plane
: A reference plane is a plane to which the sizes of other entities can be normalized. The use of reference planes can be useful for later design modifications, but its use is not obligatory.

Orthogonal plane
: A plane which is oriented in one of the three main directions, so an ordinate plane, waterline plane or buttock plane. Planes with a different orientation are labelled 'angled planes'.

[1] `http://www.sarc.nl/images/publications/background_newlay.pdf`

Compartment
A compartment is a closed, watertight space in the ship; as a result, one can pour water in a compartment and the water will not get outside of it. As for the manner of modelling, there is no distinction between a wet compartment, a dry compartment, a hold, an engine room or a closed quarter deck. In short, anything that is watertight is a compartment for PIAS. A compartment is built from one or more subcompartments.

Subcompartment
A subcompartment is a 'logical' building block of a compartment. A subcompartment has no physical meaning, the concept has only been introduced to make it a bit orderly for people to define a complex compartment. A subcompartment can be positive or negative, in the first case the shape of the subcompartment is added to the others, in the second case it is deducted. A subcompartment can be one of three different types, which will be explained below:

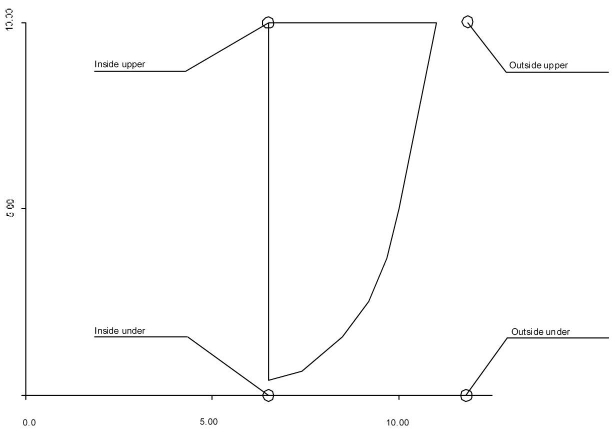
With coordinates
A subcompartment of the type 'with coordinates' is simply limited by typed coordinates (which may refer to a reference plane). The user is free to define subcompartments of this type overlappingly, or to let holes exist between them. An example of a subcompartment of this type is depicted in the figure below, where with four coordinate pairs a part of the ship is 'carved out', which constitutes the subcompartment. Please see that the coordinates may very well fall outside the ship hull boundary, if that is the case the ship boundary is simply taken also as subcompartment boundary (by the way, if shell or deck are indeed a subcompartment boundary, than it is even beter to use ∞ as boundary than to exactly use the shell breadth or deck height. See also section 9.1.5 on page 198, Intersection with the hull shape).

Space generated between planes
A subcompartment of the type 'space generated between planes' coincides with the space generated between physical planes. This type of compartments is unique, and cannot overlap between themselves.

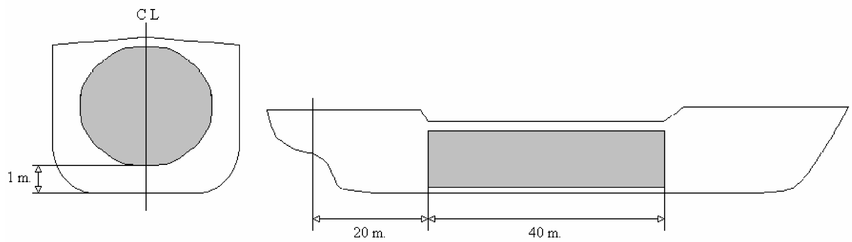
External PIAS hull form
A subcompartment of the type 'external PIAS hull form' (a.k.a. *external subcompartment*) is meant for a subcompartment being too complex for one of the other types, for example because its limits are curved, such as those of a gas tank. Such a subcompartment can be defined with a PIAS shape design or definition module, as if it is a regular ship shape. Subsequently, that 'hull form' is indicated as subcompartment, after which it is integrally included in all following steps and calculations. See the example below, where a forty meter gas tank is defined as ordinary PIAS hull form — with Hulldef or Fairway — and subsequently positioned in the actual vessel, with a longitudinal shift of twenty meters and a vertical shift of one meter, which brings the tank to its real position.

Attention

From these definitions follows an important difference between a subcompartment of the type 'space generated between planes' and the other types. For those other types have their own shape definitions, that form one whole with non-geometric characteristics, such as their names and permeabilities. But a space generated between physical planes always has a shape of its own, also without a subcompartment being assigned to it. Normally, a subcompartment of the type 'space generated between planes' is assigned to such a space, but that is not necessarily so. When no subcompartment has been assigned to such a space, that is reported by the program as **non-assigned space**. Functions are available in the GUI to assign such a space to or to unlink it from a subcompartment. The last is simply done by removing the subcompartment from the *tree view*, the space between the physical planes will remain, however.



Subcompartment type ‘with coordinates’.



Subcompartment type ‘external PIAS hull form’.

9.1.2 Use of different types of subcompartments

There are three types of subcompartments, as defined above. They can be used interchangeably at random, the use of different types of subcompartments within one compartment is also allowed. Although the user is entirely free to choose the type, there are still a few directions to be given:

- The use of physical planes is practical, firstly because the nomenclature can be made much faster with them, and secondly because the bulkheads and decks are known explicitly with them, which may be useful in the event of subsequent work or computer applications. The subcompartments that are genenerated between the planes are of the type ‘space generated between planes’, the word speaks for itself. Although this type of subcompartment in principle can be applied anywhere, it could be practical to limit its application to the larger spaces that are bounded by the primary physical planes. Suppose one would like to define, for example, a fuel oil day tank in this manner, then that would very well be possible, but then one would end with six physical planes. And in the event of a multiple of such tanks the number of physical planes will be very large, that large that one can easily lose track of the situation. Such a tank could perhaps better be defined as ‘with coordinates’, if neccessary using reference planes so that later design modifications can be processed faster.
- The type ‘with coordinates’ can be used anywhere where the subcompartment boundary consists of the hull shape, combined with (maximally twelve) boundary points. This definition is conceptually simple, overlapping subcompartments can also be defined with this, by the way, which can be an advantage or a disadvantage, that is not relevant right now, but one should be well aware of this.
- The type ‘external PIAS hull form’ is meant for subcompartments with non-flat boundaries. Subcompartments with flat boundaries (which may very well be angled) can be defined in a more practical manner with another type. By the way, subcompartments of the type ‘space generated between planes’ and ‘with coordinates’ are always trimmed by the ship shape, save that of the type ‘external PIAS hull form’.

9.1.3 Naming convention for compartments etc.

Names of (sub)compartments, reference planes and physical planes can be 50 characters long, while all visible characters are allowed. Compartment names must be unique, which is not a basic requirement in itself, but in order to keep a compartment collection orderly it has been decided upon to require this. Names of physical planes need not be unique, it might occur that there are planes with different shapes, but that they are still at the same position, so one can give them the same name. It is doubtful whether this is practical, but that is up to the user. Reference planes have infinite dimensions, so there is no need to have planes at the same position, and it may therefore be required that their names are unique. Subcompartment names only matter within one compartment, so it is not necessary that they have a unique name. When copying a (sub)compartment or reference plane, the copy gets the name of the original with the addition '(copy)'. At least, if there is place left for that and when that name is not yet in use, otherwise the copy keeps its original name.

9.1.4 Links to subcompartments

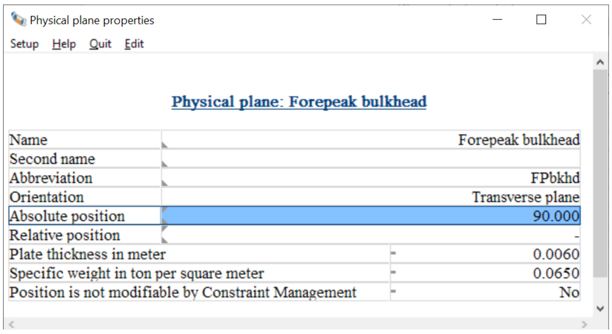
As mentioned at the definition of subcompartments, these can be positive or negative. It is not necessary, but a positive and a negative subcompartment are often used to model exactly the same space. For example, a fuel oil day tank in the ER as positive subcompartment, and exactly the same space as negative subcompartment which is deducted from the ER. It may be practical in such cases to define the shape of that subcompartment not twice, but only once, and to make a link to the second one. The advantage of this is that a geometry modification in one subcompartment is directly applied to the second one.

Such a link only applies to the shape and the name, not to the permeabilities (μ). There are sound reasons for that, because permeabilities may vary, like in our example where when the μ of the MK is 85%, that of the subcompartment to be deducted must also be 85%, because there would be more volume deducted than added. But the μ of the fuel oil day tank is of course 98% (or any other permability chosen by the user).

9.1.5 Intersection with the hull shape

Subcompartments of the type 'with coordinates' and 'space generated between planes' can be defined beyond the hull shape, typically to plus or minus ∞ (which can be defined by typing a <I> resp. <-><I> instead of a number, from **i**nfinite). In that case, the intersections between subcompartment and hull shape are determined automatically. The hull shape itself can be defined through frames (with module Hulldef) or as *solid model* (with Fairway). In the latter case, an entire planes model of the hull is available with which any subcompartment intersection can be made. But in the event of a frame model, there are only frames, there is nothing in between. That does not matter at all, PIAS has (traditionally) sufficient methods to arithmetically find an adequate solution, but for **drawing** the program must be driven back on interpolation of a subcompartment plane with those frames. In case of a longitudinal plane, such as a deck or longitudinal bulkhead, there will be in general sufficient intersections between that plane and the frames, so that a sufficiently accurate intersection line can be drawn. But in the event of angled bulkheads it is very well possible that there are only a few intersections with the frames. In theory, the intersection line can be drawn on the basis of these intersections, but as their number is small, its accuracy can be low. There are two options here, the first one is to give a shrug, because we are only dealing with a picture and not with the calculation results, and the second one is a more complete definition by means of more frames, that can be quickly generated, for example, with Fairway.

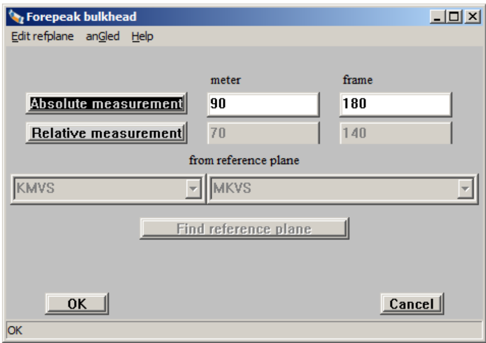
9.1.6 Menu with properties of planes



Menu with plane properties.

On multiple locations in Layout properties of a reference plane or a physical plane can be given in a text menu, from which an example is depicted above. Its parameters are the same as discussed in section 9.7.2 on page 233, List of physical planes. If the <F5> function key is pressed on the cell ‘Absolute position’ or ‘Relative position’ then an extended menu with the plane geometry pops up, as discussed in the next paragraph.

9.1.6.1 Popup menu for geometry of points or planes



Popup menu with parameters of orientation and position of a plane.

This menu is used to define the orientation, which is the direction and position, of a plane. Not only of a physical plane, but also of a reference plane. Here one can fill in:

- The position in metres of the plane (for definition and conditions reference is made to section 9.7.2 on page 233, List of physical planes}.
- The position in frames (in case of a transverse bulkhead or transverse plane).
- Possibly the relative position, which is the position in relation to a reference plane.
- When the plane has been entered referentially, then the reference plan in question can be chosen in such expendable row, either by its abbreviation (left field) or by its entire name (right field).
- As alternative for *browsing* through the reference planes list with those openklapbare rows the function [Find] can be used. For that one types the plane abbreviation in the left row (or in the right one its entire name) and presses the [Find] button.

Furthermore, this window consists of two functions in the upper bar:

- [Edit refvlak], to jump to the reference plane menu, in order to add or adjust a reference plane.
- [anGled] to change an orthogonal bulkhead or plane into an angled plane. See section 9.1.6.2 on the next page, Angled planes for more details. This function is not always available though. For example, when one changes a reference plane this is lacking, since one would be able then to convert a reference plane, for example, from transverse to angled, as a result of which all references of other transverse planes to this reference plane would be senseless.

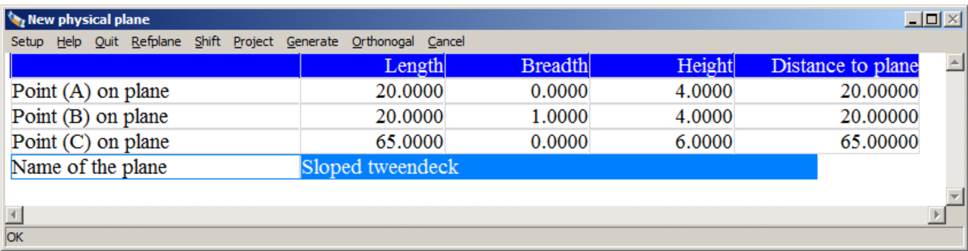
- [Help reader], opens the familiar F1 help reader.
- [Help]→[Keys], with which the list of shortcut keys appears that is applicable in this window:

Enter	Next input field
Tab	Next input field
Shift Tab	Previous input field
Ctrl-A	Select everything in box
Ctrl-Del	Remove everything from box
Alt-O	OK
Alt-C	Cancel
Alt-R	Relative
Alt-A	Absolute
Alt-Z	Find reference plane
Alt-T	Transverse bulkhead (only applicable to a new plane)
Alt-L	Longitudinal plane (only applicable to a new plane)
Alt-D	Deck (only applicable to a new plane)

So, this menu can be used to define the orientation of a plane, or to specify that a plane lies on a certain, fixed, distance from a reference plane. More or less the same menu is used to define **points** with respect to a reference plane. This popup menu is invoked by pressing <F5> in the cell with a coordinate of the point. Two more remarks ons reference planes:

- Those can be defined in the GUI, as well as in the alphanumeric table as discussed in section 9.7.3 on page 234, List of reference planes.
- If a coordinate is given relative, then in all input windows, in the status bar at the bottom, th erelative as well as the absolute value is presented from the cell where the text cursor resides.

9.1.6.2 Angled planes



Definition menu angled plane.

The orientation of an orthogonal plane, that is a transverse plane, longitudinal plane or horizontal plane, is entirely recorded with its type and one digit. In order to record an angled plane, however, more data are required. There are innumerable manners in which an angled plane can be recorded, but in Layout has been opted for doing this by means of three points in the space, because this is considered most intuitively. Those three points are in a menu, an example of which is given above. This menu essentially includes for any of the tree points a row, with in the columns the length, breadth and height coordinates of that point. As a matter of fact, these points are unrelated to any other point of the ship or its subdivsion — although the are fit to be linked to a reference plane. The last column contains the distance from that point to the angled plane. For the plane is not directly adjusted to the position of the three points, but for reasons of safety it has been chosen that the function [Generate] must be called upon there later on. Other functions that are available in the upper bar are:

- [Shift]: shift this point to the plane, along the axis of this point. So, for example, if the text cursor is located on the 'height' coordinate this will result in a shift in vertical direction.
- [Project]: project this point onto the angled plane, perpendicular to this plane.
- [Orthogonal]: reshape this plane to an orthogonal plane that is most similar to this angled plane.
- [Cancel]: leave this menu without saving the modification.

9.1.6.3 Limited positioning of a physical plane

The constellation of physical planes also fixes the nature and shape — or, put another way, the topology and geometry — of the enclosed spaces, the compartments. That's nice, but it also introduces a limitation because planes cannot be passed by their immediate neighbors; that would, after all, compromise the entire 'logic' of the subdivision. That can extent further than one might think at first glance, because planes can also indirectly (by means of reference planes) get a location. Thats all internally monitored in Layout, and if one would place a plane beyond its topological limit it is pushed back to its extreme limit. No warning is given on that — that might in fact lead to rather long warning lists, from which nobody benefits — but at least you now know the reason if a plane does not pass a certain location.

9.1.7 Compatibilitity with the former compartment module of PIAS

Around 1985 the PIAS module Compart was developed, for the definition of compartmentsm, calculations of tank tables etc. The current module Layout, which was developed in the years 2010-2012, serves the same purposes, but is much more extensive, for example because of the support with explicit decks and planes and a GUI. Compart files can be read in Layout, but this has to be done manually, see section 9.11.3 on page 243, Import PIAS compartments from pre-2012 format. If, on the basis of these compartments, subsequently physical planes are being generated it may be useful to clean the compartments first, see section 9.11.4 on page 243, Clean pre-2012 PIAS compartments.

9.2 Constraint Management

Attention

> Constaint management makes extended use of the bulkheads and decks module (40.100.0) for Layout, and will only work as intended if your model exclusively makes use of this module to define its compartments.
> Constraint management will attempt to satisfy the defined constraints by changing the position of any physical plane not locked via the 'Position is not modifiable by Constraint Management' option in section 9.7.2 on page 233, List of physical planes. It is advised to make a backup of your model before evaluating the constraints.

Layout is equipped with a constraint management module. Using the constraint editor constraints can be created, edited and deleted. Constraints can be linked to compartments and physical planes in Layout. Then, using the constraints balace, the importance of the various constraints relative to each other can be set, and the module will propose a design change which complies to the active constraints.

More information on the purpose and operation of constraint management can be found here[2] and here[3].

9.2.1 Constraint definition

Constraint Management is restricted to address the constraint problem as far as it can be included in a processing time that allows interactivity. Currently only constraints that deal with plane positions, areas and compartment volumes are considered.

Constraints can be categorized by two distinct properties, that we call *groups* and *types*. The *group* the constraint belongs to says something about how the constraints relates to the geometry of the vessel. Four different groups are identified:

- Amount (e.g. number of bulkheads)
- Position (e.g. tanktop position)
- Area (e.g. engine room area)
- Volume (e.g. cargo space volume)

It is important to note that the group *amount* is present, but is purely a 'bookkeeping' tool. This is because the Constraint Management feature works by changing the position of the physical planes of the model. It can not actually create new geometry features.

The second property of a constraint is its *type*. This type conveys the meaning of the numerical value of the constraint. Four types are identified:

- Minimum (at least..)

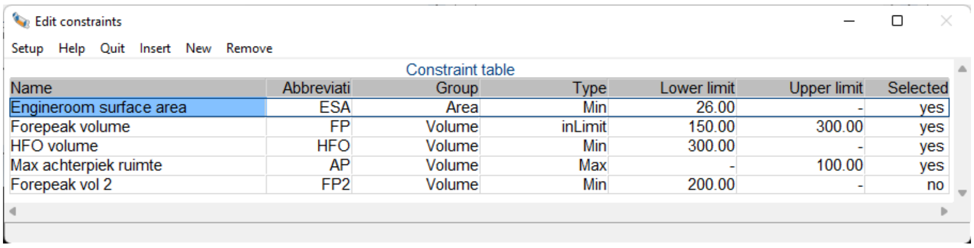
[2] https://www.sarc.nl/wp-content/uploads/2022/08/Constraint-Management-paper.pdf
[3] http://www.sarc.nl/images/publications/background_newlay.pdf

- Maximum (at most...)
- inLimit (in between limits)
- outLimit (outside of limits)

9.2.2 Creating and editing constraints

The input table for the constraints can be opened via the menu bar in the graphical user interface: cOnstraints -> edit constraints. A constraint is defined by a name and abbreviation, its group, type and the associated upper and/or lower limits. Lastly the constraint can be set to inactive to disregard it in the constraint evaluation step.

An example of a constraint table is shown below.



Name	Abbreviati	Group	Type	Lower limit	Upper limit	Selected
Engineroom surface area	ESA	Area	Min	26.00	-	yes
Forepeak volume	FP	Volume	inLimit	150.00	300.00	yes
HFO volume	HFO	Volume	Min	300.00	-	yes
Max achterpiek ruimte	AP	Volume	Max	-	100.00	yes
Forepeak vol 2	FP2	Volume	Min	200.00	-	no

Constraint table

9.2.3 Linking constraints to the model

After a constraint has been created it needs to be coupled to its compartment or physical plane. This can be done by opening the compartment properties (see paragraph 9.5.1.2.18 on page 219, Number of constraints) or the physical plane properties (see section 9.7.2 on page 233, List of physical planes), and then setting the number of constraints value to the desired amount. Then, for that number of constraints, a constraint from the constraint table can be selected.

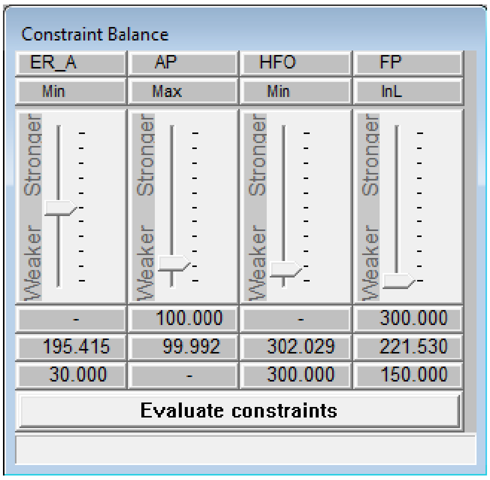
9.2.4 Constraint evaluation

An evaluation can be performed from the menu bar via cOnstraints->evaluate. Constraint management will attempt to find a solution that satisfies all active constraints.

As the various constraints probably require different things from the vessel, finding the right design compromise that satisfies all constraints can be a challenge. Often, not all the constraints are equally important, or they influence the design changes unequally and should thusly be weighted differently. Since the relative importance of the constraint is dependent not only on the specific constraint itself, but is also relative to the other active constraints in the model, the assigning of the relative importance to the active constraints is left to the user, in order to push the design in the direction that they desire.

By opening the constraint balance window the weights of the various active constraints that are linked to at least one compartment of physical plane can be set relative to each other. This can be done by the position of the slider in the vertical trackbar. Each trackbar is titled with the abbreviation of its constraint, the constraint type, and below are in order the set upper boundary value, the current value, and the lower boundary value. If the current value does not comply with the constraint it is printed in red for easy identification.

Below is an example of the constraint balance window:



Constraint balace

After the sliders of the constraints are set to the desired positions, the 'Evaluate constraints' button at the bottom of the constraint balance can be pushed to attempts to find a solution that satisfies all the constraints. This is done by moving all the physical planes in the model, except those that are set to not be modifiable by constraint manager. The solution is optimized for the smallest repositioning of the physical planes, so as to stay as close as possible to the designers intent when setting up the model.

9.3 Main menu

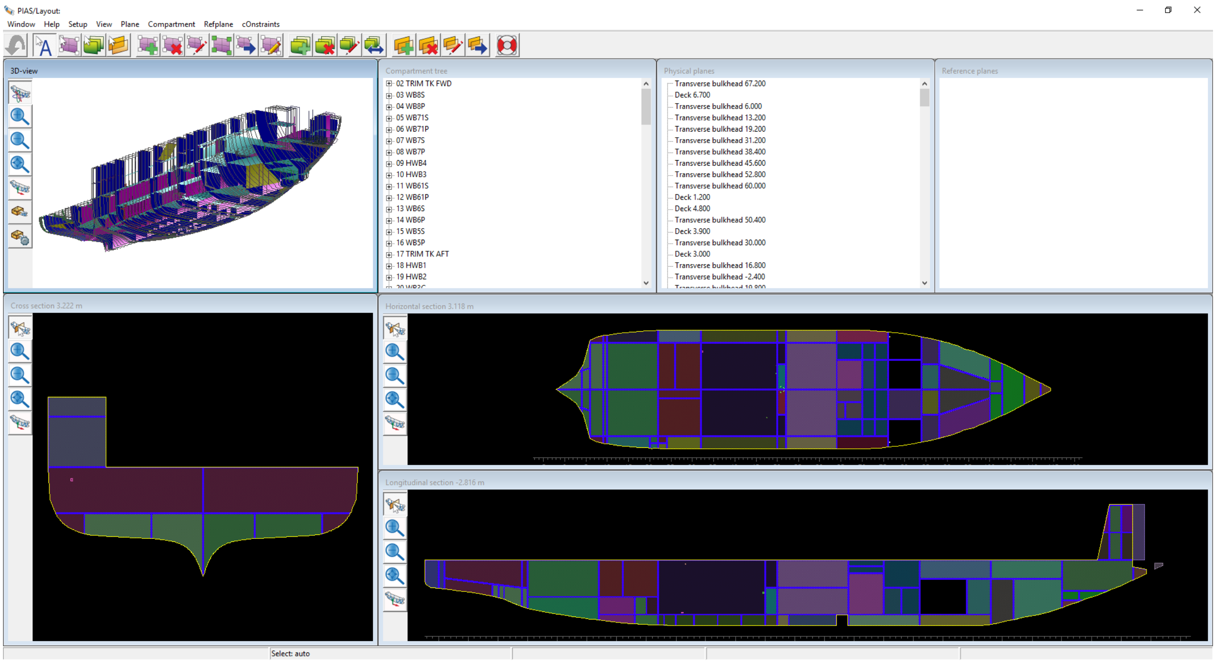
Having started up Layout, one enters the main menu, the various options of which are explained in more detail in the following sections.

Design and utilization of the ship's layout

1.	Graphical User Interface of planes and compartments
2.	Compartment list, calculation and printing of tank tables
3.	Pipe lines and piping systems
4.	Other lists, and program settings
5.	Threedimensional presentation
6.	Subdivision plan
7.	Print compartment input data
8.	Conversion, and import and export of subdivision data
9.	File and backup management

9.4 Graphical User Interface of planes and compartments

9.4.1 GUI components



GUI.

An example of the GUI (Graphical User Interface) is shown above. The GUI may contain nine sub windows (from which some can be switched on or of with functions as discussed in section 9.4.3.2 on page 208, View):

- Three orthogonal cross sections, namely a transverse, longitudinal and horizontal.
- A 3D (rendered) view.
- A *tree view* window with a tree of compartments and subcompartments.
- A *tree view* window with physical planes.
- A *tree view* window with reference planes.
- A text windowlet showing the total volume of the selected (or, the pointed) compartment.
- A *constraint management window* in which design objectives are shown, as well as the discrepancy between objective and reality. For the time being, this *constraint management* facility — from which purpose and operation is discussed `here`[4] — remains undiscussed in this manual.

Attention

The two orthogonal longitudinal sections here in the GUI are in the end derived from the ship hull shape, which as a rule will be a frame model representation (please refer to section 2.10.2 on page 16, Hull form representations for an overview of the different ship hull models applied by PIAS). Other sections can theoretically not be created with guaranteed correctness, on basis of such a frame model, actually a *curved surface model* is required for this task. In order to create these kind of sketches, PIAS contains extensive algorithms which anticipates on quite some special situations. The result is that only by rare exceptions the longitudinal sections are not properly drawn. In such an occasion there is no reason for suspicion, for it is only a visual affair, the computation results are not affected.

Right at the bottom of the GUI window a status line is displayed, subdivided in five boxes:

- The first box contains a short explanation of the function of the menu bar, when the mouse pointer stands on it.
- The second box displays the selection mode (see section 9.4.2.2 on the following page, Left mouse button and modus).

[4] `http://www.sarc.nl/images/publications/background_newlay.pdf`

- The third box dynamically displays the coordinate (L, B and H) of the pointer position in the orthogonal views.
- The fourth box dynamically displays the name of the physical or reference plane that is closest to the mouse pointer.
- The fifth box dynamically displays the name of the compartment and/or subcompartment where the mouse pointer stands above.

Furthermore, in the upper bar the GUI has a number of functions that have been subdivided in subfunctions. Those functions can either be carried out directly, or can be 'hanged' to the mouse button, which mechanism is discussed in section 9.4.2.3 on the next page, How long stays a function assigned to a mouse button?. The function bars under [Compart], [Refplane] en [Plane] are subdivided by a horizontal dividing line. The functions **above** that line are only related to the *tree view* window in question, the functions **under** the line are generally applicable.

9.4.2 General operations and modus

9.4.2.1 Mouse buttons

The mouse buttons are used as follows:

- The left button can be used for two things, namely a) the selection of compartments, physical planes and reference planes, or b) performing functions with it.
- Pressing the right button and subsequently moving the mouse is for display. In the three orthogonal views that is choosing the intersection locations (unless one has opted for pan at the tool bar at the left side of that window). And in the 3D view that is default rotation (unless one has opted for another display function at the tool bar at the left side of that window, for example, pan or clip).
- Shortly clicking the right button in the 3D view brings up a specific menu with which colors, translucency and lighting can be set, or a screen print or 3D model (in VRML-format) can be stored in a file, see section 9.4.2.4 on the following page, Operation in the 3D subwindows for a more detailed explanation of the possibilities in the 3D view.
- Keep on pressing the middle button and then moving the mouse is panning.
- The mouse wheel is zooming, as well in the 3D view as in the orthogonal views.

Furthermore, one can carry out the (for MS Windows) usual actions in the *tree view* windows such as dragging of compartments, subcompartments, physical planes and reference planes. With function button <F2> one can change a name into such a *tree view* window.

9.4.2.2 Left mouse button and modus

The left mouse button is meant default for indicating or selecting of ship's items — compartments, physical planes and reference planes — but it is also possible to assign a function to it, which is carried out later when such ship's item has been indicated. When such a function (for example function [Plane], subfunction [Edit], with which data of physical planes can be modified) has been activated, it is shown in the second block of the bottom status row. When the box displays 'Select', this means that the left mouse button is in the default position: select. What is exactly selected depends on the selection modus, which can have four positions:

Auto
: This is the most extensive position; herewith the nearest item is selected, which may be a compartment, physical plane or reference plane. At a physical plane, the <Alt> key can be used, see the discussion at 'planes' below.

Compartments
: With which only compartments are selected (see note below).

Planes
: With 'Planes' only physical planes are selected. It may happen that physical planes are very close to each other and visually indistinguishable, in which case the left mouse button in combination with <Alt> can be used. Then a box will pop up with the (maximally four) nearest planes, from which one can be chosen.

Reference planes
: Only reference planes are selected herewith.

Piping
: In which the selection is restricted to piping systems. If subsequently a pipe line is double-clicked, that pipe is opened in the piping definition GUI, which is discussed in section 9.6 on page 224, Pipe lines and piping

systems. The piping is deliberately not included in the 'Auto' setting (from a few lines above); because the pipes cross everything else, they should be explicitly selected.

Attention

Instead of selecting a compartment, it could occasionally be desirable to select a subcompartment. However, in general that would be somewhat difficult, because in the 2D views compartments are shown instead of subcompartments. For this reason, in due time, the [view] option (see section 9.4.3.2 on page 208, View) will be extended with the 'subcompartment' setting as an alternative to the current 'compartment' setting. The two will be mutually exclusive, so either the compartments are shown, or the subcompartments. And the object of selection at this point in the manual follows this view- setting. This mechanism may, however, induce a side effect: a single specific action will imply more or less a certain selection, for example if in the compartment tree a compartment is selected, then it might be obvious to select a compartment as well in the 2D views, *mutatis mutandis* for a subcompartment. However, such 'logic' might be in conflict with the present *view compartment/subcompartment* setting, and for that reason in such a case the program will switch automatically to the view setting which matches this action 'logically'.

9.4.2.3 How long stays a function assigned to a mouse button?

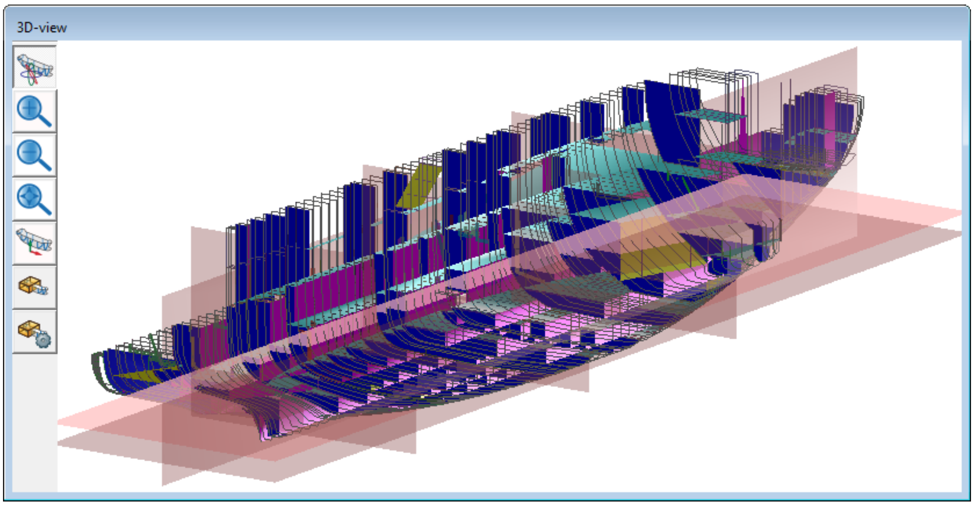
This is no principal matter, it is a choice, Layout can be made thus that it is assigned once, or permanently, or otherwise, in principle this does not matter. But users may have different wishes, and that's why that can be set, in section 9.7.5 on page 235, Layout project settings and function colors is explained how this works. There are three options:

Never
Then the mouse function always remains attached to the left button (until one chooses another).

Cancel structural commands after use
With this setting, commands that cause an important modification in the arrangement structure (such as adding and removing of planes) are removed as mouse function after single use. This prevents that planes or compartments are unwantedly added or removed in the event of fast clicking.

Cancel all commands after use.
Herewith any function is removed as mouse function after single use, and therefore one has to assign any command to the mouse button repeatedly. Apart from that, at all times the user can detach the function from the mouse button with the key <F12>

9.4.2.4 Operation in the 3D subwindows



Three-dimensional sub window.

At the left side in each three-dimensional subwindow is a number of buttons that are specifically related to that subwindow. When the right mouse button is pressed permanently, the [rotate] or [pan] function is carried out, depending on what has been set. By pressing the right mouse button shortly, a popup menu appears with which one can carry out non-modelling operations witih the ship subdivision model. These are available in four groups, are being discussed in much more extent in section 7.8 on page 189, Rendered views, but summarized their functions are:

- [View]: herewith one can carry out the same operations as with the buttons at the left side, which have been discussed above. Besides, there is still the function [(in)visible], with which one can set which individual parts of a ship are (in)visible.
- [Edit]: with this function the position and intensity of external light sources can be set. One can also change the colors, reflection characteristics and transparency of objects or background
- [File]: with this function one can save the **present picture** to file (in VRML or BMP format), print with the printer or copy to *clipboard*. This function only regards the picture, it has nothing to do with the file saving of Layout.
- [Setup] contains two obsolete configuration options.

Attention

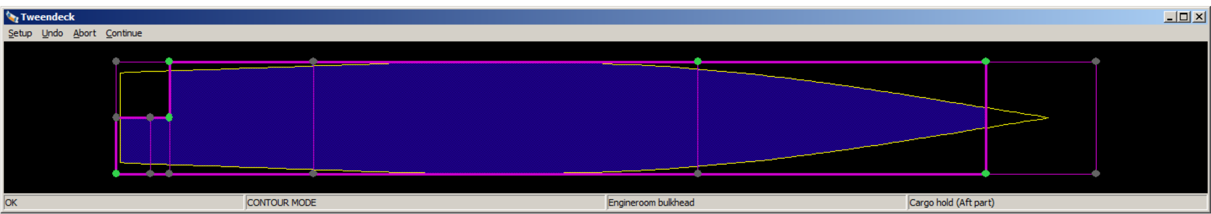
With emphasis a tools is recomended which can assist to determine from which side an object is viewed. This is the *orientation box*, from which purpose and operation is discussed in section 7.8.1 on page 190, View.

9.4.2.5 Shortcut keys

In order to speed up work it can be practical to use shortcut keys. The following are available for this:

- In the *tree view* windows the <Insert> and <Delete> keys for resp. adding or removing of a (sub)compartment, reference plane or physical plane. After <Delete> at (sub)compartment, the (sub)compartment can be sticked in again (possibly at another position), so this button rather has the meaning of ‘cutting’ than of ‘removing’.
- In the *tree view* windows the <Home>, <End>, <Page Up> and <Page Down> keys in order to jump resp. to the top of the list, to its bottom, to the upper line of the window and to the lower line.
- In the *tree view* windows the <F2> to alter the name.
- <F12> to detach a function from left mouse button (see section 9.4.2.3 on the preceding page, How long stays a function assigned to a mouse button?).
- As side-effect of Windows, any function in the upper bar can be called with the key combination <Alt><function letter>.
- In due course, other <F> function keys will be assigned to the functions that are mostly used.

9.4.2.6 The shape of a plane (the green dots)



Defining the shape of a physical plane with the green dots.

An important function of Layout is the addition of planes. These not necessarily need to extend over the entire ship, but can also be in a part of it. This shape is entered by means of the plane contour, which is controlled by the ‘green dots’, as they are called in the manual. Its background is discussed here in more detail. This all takes place in a popup window as displayed in the above figure, where you can see that the shape of the plane has been recorded with only three green dots.

What is shown there is the cross-section of the plane, with the chosen contour indicated in purple (at least, that is the default color, the user himself can choose another color at the *Setup* menu, see also section 9.7.5 on page 235, Layout project settings and function colors). The contour can stop at the intersection with other planes, so one does not enter coordinates here, one chooses to which other, already present, planes the contour extends. A topological definition has been obtained in that manner, from which results, for example, that when the position of another plane changes, this contour also changes. The main idea here is that a user can enter the desired contour by indicating points of these other planes, through which that contour has to go. By the way, one need not indicate all points, also in the event of only a few points the program itself chooses the most evident contour, see the example from the figure where the contour has been recorded with only four indicated points (the green dots). More precisely, indicating occurs as follows:

- When one stands with the mouse pointer on or near a point, then the dot can be switched on or off with the left mouse button as 'wanted' (green).
- When one stands with the mouse pointer near a connecting line between two points, then that piece of line can be switched on or off with the left mouse button as 'wanted' (green).
- When one stands with the mouse pointer on or near a point, then the dot can be switched on or off with the right mouse button as 'unwanted' (red).
- Idem for unwanted connecting lines (red).

In the event of a new plane, one can directly start to switch on/off. In the event of an already existing plane, there is protection against accidental modification, which is called the 'contour modus'. That contour modus is initially 'off' (that is also reported in the status line at the bottom of the window) so nothing can be changed. With menu option [Setup], suboptie [Contourmodus] one can switch this on. Further options from the upper bar menu are:

- Undo, undo the changes, and put the original contour back.
- Abort, abort this action and stop with this contour changing window.
- Continue, stop with this contour changing window and process the change in the ship's model. When one presses on the right upper cross of the window then it is clear that the user wants to stop with this window, but it is not clear whether the changes have to be included in the ship's model. When there actually are changes, that question is asked again.

9.4.3 GUI functions

The purpose and the operation of various functions to be chosen from the upper bar are discussed below. There are two types of functions, namely those with a direct effect (since nothing else has to be indicated) and those that are assigned to the left mouse button, because something has to be indicated later on to which this function is applied. At any function below is mentioned which type it is.

9.4.3.1 Setup

9.4.3.1.1 Clear action

The action that is attached to the left mouse button at that moment is removed from it with this. Type of function: direct.

9.4.3.1.2 Selection mode

Herewith one can choose one of the four selection modi, as explained in section 9.4.2.2 on page 205, Left mouse button and modus.

9.4.3.1.3 Setup

Herewith one calls up the menu with program settings, which is discussed in more detail in section 9.7.5 on page 235, Layout project settings and function colors.

9.4.3.1.4 Colors

Herewith one calls up the menu with which the colors of the various ship's components can be set. This is a limited version of a more general menu for setting ship's components, which is discussed in more detail in section 9.7.6 on page 236, Names and color per part category.

9.4.3.2 View

First of all, one can indicate here which things you would like to be presented in the GUI. You may select from:

- Planes, these are the physical planes and. When these have been made invisible, then the physical planes *tree view* window also disappears, because this has become useless. Likewise, functions related to physical planes cannot be activated then. Also a third choice exists, between on and off, which is *separating planes on*. With this choice only those parts of the physical planes will be drawn which constitute real separations between compartments (or, to be more precise, between subcompartments of type 'space generated between planes'). This gives a more realistic picture, however, please bear in mind that this is a drawing switch only; in the underlying model the physical planes still extend, also if the separate parts of the same compartment. In 'outside' output, such as to the subdivision plan, always the *separating planes on* method is used — regardless the switch setting here in the GUI — because this is most genuine.
- Reference planes, the reference planes. When these have been made invisible, then the reference planes *tree view* window also disappears, and functions related to reference planes cannot be activated.
- Hull, the hull lines (or planes), only applies to the 3D window.
- Compartments, applies to the 2D windows as well as the 3D window.
- Piping, all pipe lines, from which the definition method is discussed in section 9.6 on page 224, Pipe lines and piping systems, applies to the 2D as well as the 3D windows.
- Compartment Colors, where the system of compartment coloring can be chosen; the options are:
  - Uniform, where all compartments get the same color. There may be a difference in color after specific program actions, such as in the event of a just cut or generated compartment (these colors can be set at section 9.7.5 on page 235, Layout project settings and function colors).
  - Individual, where any compartment gets its own coulor (automatically determined by the program).
  - Per weight group, where a compartment is colored in conformity with the color that applies for the weight group assigned to the compartment. These colors can be set as discussed in section 9.7.7 on page 236, Define weight groups.
  - Compartment Overlap, here the program conducts an overlap test between compartments, where one can deduce from the color whether the compartments have been defined uniquely and non-overlapping, as it should be:
    - Green: good.
    - Background color: this piece of a ship is not covered by a compartment, the compartment definition is therefore not complete.
    - Red: two or more compartments overlap here.
- Compartment Volume, which makes the volume of the selected (or, alternatively, pointed) compartment to be shown in a text windowlet.

9.4.3.3 Plane

Here the menu options **above** the horizontal dividing line also apply to the *tree view*, and those **under** the line to the graphical windows. For the time being, the first group consists of only one function:

9.4.3.3.1 Sort

With this command the compartments in the *tree view* are sorted. This can be done on four criteria, namely on name, position, type and abbreviation. The sorting can be undone again with Undo. Type of function: direct.

9.4.3.3.2 Draw

With this function one draws a plane interactively. The operation is as follows:

- Choose this function.
- Go to the orthogonal view where the plane must be perpendicular to.
- Go to an endpoint of the plane and press the left mouse button. There will appear a cross-hair.
- Go to the other endpoint, and press the left mouse button again. There will appear a second cross-hair, with a connecting line.
- The bulkhead will be generated perpendicular to the view, through that line.
- In general, the line will not fall in an orthogonal plane accurately, whereas that was perhaps intended. That's why the program offers the opportunity of fine-tuning. There one can choose from: - Consider the bulkhead to be orthogonal (through the mean location of the line)
  - Idem, but with the possibility to adjust the location exactly, by typing a size.
  - As drawn (possibly angled).

- Afterwards appears a pop-up window with the ‘green dots’ (see section 9.4.2.6 on page 207, The shape of a plane (the green dots)) so positioned that an as reasonable as possible part of the line is covered by the bulkhead. Is this not satisfactory, then one can still adjust the level of extension of the bulkhead by means of the yellow dots. Type of function: left mouse button, because the location and direction of the plane have to be entered later on by means of drawing.

9.4.3.3.3 New

With this function the popupbox from section 9.1.6.1 on page 199, Popup menu for geometry of points or planes comes up, which is used to add a plane that extends over the entire ship (from stern to bow, or from bottom to top) at the beginning. Afterwards can be indicated through the ‘green dots’ (see section 9.4.2.6 on page 207, The shape of a plane (the green dots)) that the plane extends over a more limited part. Type of function: direct.

9.4.3.3.4 Insert

With this function one adds a plane in one indicated compartment. Afterwards, one can still indicate through the ‘green dots’ that the plane extends over a larger part. Type of function: left mouse button, because the compartment where the plane will appear has to be indicated later on.

9.4.3.3.5 Remove

A plane is removed with this function. After the removal of the plane, excess subcompartments may remain. These are removed according to the order of the (sub)compartment list, i.e. when several subcompartments of the type ‘space generated between planes’ refer to the same space, then the first ones are removed and the last of all remains. Type of function: left mouse button, because the plane to be removed has to be indicated later on.

Attention

> Plane A can be a boundary in another plane, plane B. If plane A is removed then B will become larger as a result, because it will lose its boundary. This change in B may in turn introduce other changes in other planes, for which B was the boundary. Etc. etc., that can result in a chain reaction of changes. It may be that the result of changes is unexpected, or even undesired. In that case one should manually adapt the changed layout to the human insights.

9.4.3.3.6 Edit

The features of a physical plane can be changed with this function, see section 9.1.6 on page 199, Menu with properties of planes for details. Type of function: left mouse button, because the plane to be changed still has to be indicated.

9.4.3.3.7 Geometry

With this function the contour (and therefore the shape) of a plane is changed. Type of function: left mouse button, because the plane to be change still has to be indicated. After having indicated the plane, a window pops up with the shape of the plane, where one can change the contour by means of the ‘green dots’ (see section 9.4.2.6 on page 207, The shape of a plane (the green dots)).

Attention

> If a planeis modified in such a way that parts are **discarded**, then the remark given just above at ‘Remove’ is also applicable here.

9.4.3.3.8 Copy

Herewith one can copy a plane. Type of function: left mouse button, because the plane to be copied still has to be indicated. The operation is:

- Choose this function.
- Point at the plane to be copied.
- A pop-up window of the copied plane appears, already filled with the copied parameters. Change the name and position in that window (NB the orientation (position of the plane) cannot be changed, so one is not able to copy a transverse bulkhead to a deck).
- Press the **OK** button, and the copied plane is added to the model.

9.4.3.4 Compartment

These menu options have been subdivided in two groups, those **above** the horizontal dividing line regard the compartments *tree view*, those **under** the line are applicable in the graphical windows. We start with the first group:

9.4.3.4.1 Compartments Tree view

The compartment tree contains the compartments in the main branches, and under each compartment the subcompartments. With this command one can collapse and expand all branches at once. Apart from that, one can of course also collapse or expand an individual branch with the + for each compartment. Type of function: direct.

9.4.3.4.2 Sort

With this command the compartments are sorted in the *tree view*. This is possible on two criteria, namely on compartment name, and on location (where the compartments are sorted in length, breadth and height direction). The sorting can be undone with Undo. Type of function: direct.

9.4.3.4.3 Newcompart

Herewith a new, and empty, compartment is added in the tree, just below the compartment that was selected at that moment. In order to control at which location in the tree the compartment is exectly added, this command can only be given from the compartment tree window. Type of function: direct.

9.4.3.4.4 NewSubcompart

Herewith a new subcompartment is added under the compartment that was selected at that moment. The subcompartment only has a default shape and type, which has no meaning, nor any connection with something else. Type of function: direct.

9.4.3.4.5 Cut

Cut a compartment or subcompartment. The type of function depends on the sub window from where the function was activated; in a compartment treeview the function has direct working, from a 2D window the function is assigned to the mouse button. By the way, the <Delete> key does exactly the same.

9.4.3.4.6 Paste

Paste a compartment or subcompartment. That object is then placed after the then selected compartment or subcompartment. Type of function: direct.

9.4.3.4.7 Undocut

Undoes the cutting of a (sub)compartment. Type of function: direct.

9.4.3.4.8 Remove eMpty

Removes all empty compartments (those compartments that have no subcompartments). This function can be practically used after a number of compartments no longer has subcompartments after dragging (graphically, or in the compartment tree). Those can easily be removed in this manner. Type of function: direct.

9.4.3.4.9 Edit

This is the first function of the list that is applicable in the graphical windows, and therefore not in the *tree view*. With this function one enters the detail window of a compartment, which is discussed in more detail in section 9.5.1 on page 213, Compartment definition window. Type of function: assign to left mouse button.

9.4.3.4.10 Assign

Compartments and spaces as they are generated between planes have been linked. This link is as much as possible maintained, so when, for example, a new plane is added then additional compartments will be generated for that, the name and other features of which can be adjusted later on by the user. But if one has removed a compartment with, for example, [Cut] or the <Delete> key the space in question still exists, but it is no longer linked to a compartment. With this function, [Assign], a new compartment is added that is linked to the space. That new compartment still has default parameters, such as name and specific gravity, but these can simply be changed later on. Type of function: assign to left mouse button, because the space to which a new compartment must be assigned has to be indicated afterwards in one of the orthogonal cross-sections.

9.4.3.4.11 Swap

When a plane is added that runs through a subcompartment, that subcompartment is divided in two parts, while the features of the original subcompartment are assigned to one space, and a new subcompartment is made for the second space, the features of which have to be filled in in more detail (except for its shape, of course). This choice is arbitrary, and it might very well be the intention of the designer that the original subcompartment is assigned to that second space. When this is the case, one can turn this assignment with this function, [Swap], again (and also turn it back again when one is mistaken). Type of function: left mouse button, since the space to be *swapped* still has to be indicated.

9.4.3.4.12 Recombine

Subcompartments are hanging under compartments, and its organisation is completely up to the user. Particularly after the event of adding new planes, new spaces are made which are each assigned to a new subcompartment that is hanging under a new compartment. When one wants to change that subdivision, one can do that by means of dragging in the compartment *tree view* window. With this function, [Recombine], one can do the same in one of the 2D windows. So one can point to a subcompartment, press the mouse button, and drag to another subcompartment. When one releases the mouse button, and after confirmation, the subcompartment no longer resides under the original compartment, but under the newly indicated compartment instead. Empty compartments (i.e. compartments that have no subcompartments) can be arise in this manner, which is no problem in itself, but for overview purposes it may be practical to remove these, either manually, or with the [Remove eMpty] function, see paragraph 9.4.3.4.8 on the preceding page, Remove eMpty.

9.4.3.5 Refplane

Here the menu options **above** the horizontal dividing line also apply to the *tree view*, and those **under** the line to the graphical windows. For the time being, the first group consists of only one function:

9.4.3.5.1 Sort

The compartments in the *tree view* are sorted with this command. This can be done on four criteria, namely on type, abbreviation, name, and location. The sorting can be undone again with Undo. Type of function: direct.

9.4.3.5.2 New

A new reference plane is added herewith. A menu pops up where the position and other data can be entered, see section 9.1.6.1 on page 199, Popup menu for geometry of points or planes for more details. Type of function: direct.

9.4.3.5.3 Remove

With this function a reference plane is removed. Type of function: left mouse button, because the reference plane to be removed still has to be indicated.

9.4.3.5.4 Edit

The characteristics of a reference plane can be changed with this function, see section 9.1.6 on page 199, Menu with properties of planes for the details. Type of function: left mouse button, because the reference plane to be changed still has to be indicated.

9.5 Compartment list, calculation and printing of tank tables

A list of compartments turns up here, with several columns — for which the order can be set by the user, see section 4.2 on page 35, Input window — which are explained in section 9.5.1.2 on page 216, Compartment data.

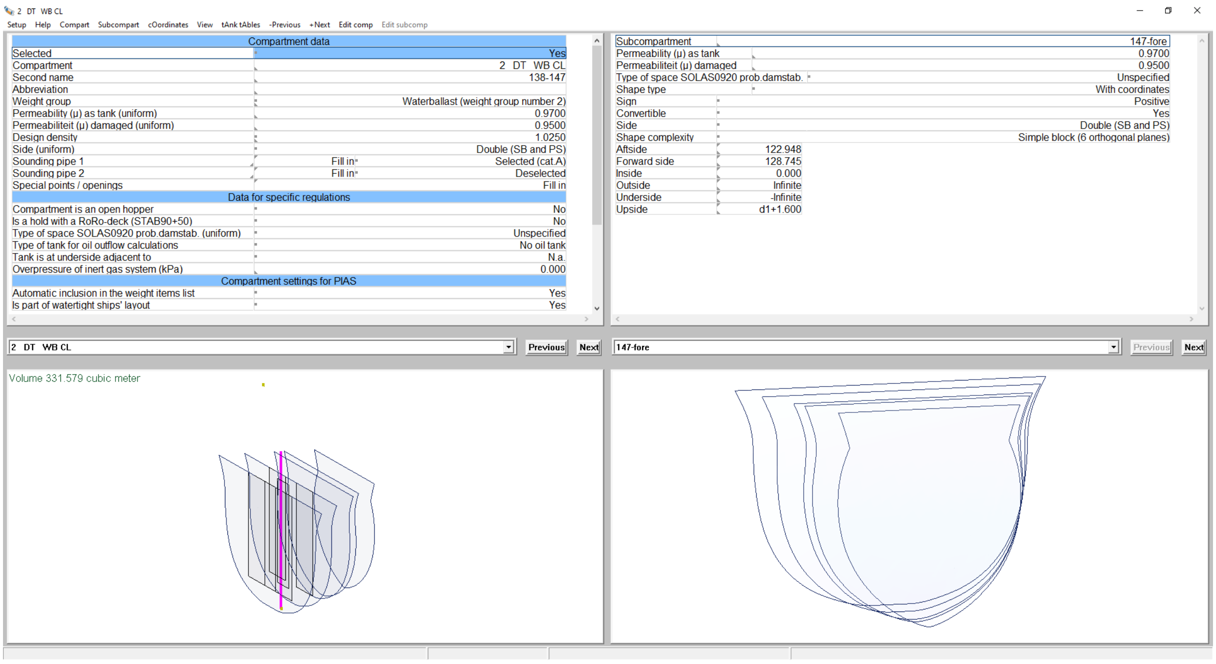
The upper bar contains some specific functions:

- [Manage], with the following sub-options:
  - [Copy] and [Paste] will speak for itself. One might wonder why these options are not included in the general copy/paste under [Edit], as common in PIAS. The reason is that over here also a [Paste Link] option exists, which is not facilitated in the regular [Edit]. In this respect Layout is an exception. Perhaps u.nnecessarily: the [Edit] options — as discussed in section 4.4 on page 39, Copy, paste etc. — only exert on the visual cell values, which can e.g. be exchanged with a spreadsheet, while the copy/paste here under [Manage] concern **all** compartment data, including all underlying subcompartments.
    If the compartment contains subcompartments of the type ‘Space generated between planes’, at pasting, these are converted into type ‘with coordinates’. This is inevitable, because the **space** between the planes exists once and cannot be duplicated. If this conversion is not possible (e.g. because the compartment shape cannot be represented with only coordinates aft and forward) then that subcompartment will be excluded from the copy.
  - [Paste Link], see the explanation at paragraph 9.5.1.1.3 on page 215, Subcompartment functions.
  - [Move], which can be used to move a compartiment up or down in the compartment list.
  - [Sort], herewith the compartments can be sorted according to column (i.e. in the order of the data of the column on which the text cursor is standing), to position and to time of definition. The selection can be undone again with [Undo last sort].

- [tAnk tAbles], with which one can calculate and print tank tables, see section 9.5.2 on page 220, Calculate and print tank tables.
- A row of icons, which may be used to invoke some of the other upper bar manu functions. These icons are presumed to be self-explanatory, although additional assistance is given by hovering over the picture. When one has entered into a tank table, by means of <Enter> in the rightmost column, these icons are also present although they will obviously not be functioning here.

With <Enter> in any other column one enters the compartment definition screen, which will be discussed later on.

9.5.1 Compartment definition window



Compartment definition window.

9.5.1.1 Design of the compartment definition window

The compartment definition window consists of the following items:

- Top left a list of compartment characteristics, such as name or sounding pipe data. These are explained in detail in section 9.5.1.2 on page 216, Compartment data.
- Bottom left a 3D view of the compartment. In this window one can call up a number of functions with the right mouse button as they have been discussed at section 9.4.2.4 on page 206, Operation in the 3D subwindows. By the way, mouse wheel is zooming in-out.
- Left of centre an drop-down list of compartments, from which one can choose another compartment (one can also type a name here, but nothing happens then). Choosing the previous and next compartment can also be done with the two buttons at the right of this list.
- At the right three subscreens related to **sub**compartments, and which have the same purpose as the three subwindows discussed above.
- At the bottom a status row, with explanations and/or sizes related to the cell where the text cursor is standing on.

Changing between compartments and subcompartments can be done through indication with the mouse pointer, but also with the <Tab> key. Subsequently, there is still a number of functions to be called up in the upper bar, which are discussed below. The upper bar also consists of the + and - functions, with which one can jump to the next or previous compartment (when the text cursor is standing on the left window) or subcompartment (when the textcursor is standing on the right window). These functions have been included so one can quickly go through the (sub)compartments with <Alt><+> and <Alt><->.

Reference planes can be used for all coordinates of compartments and subcompartments, so not only for the compartment boundaries but for example also for openings or the sounding pipe. A reference plan ecan be selected by pressing <F5> in that particular cell. This activates a popup window that is further discussed in section 9.1.6.1 on page 199, Popup menu for geometry of points or planes.

Attention

Copy/paste functions, as discussed in section 4.4 on page 39, Copy, paste etc., are not implemented here. These input windows have such a variable structure that this will hardly fit.

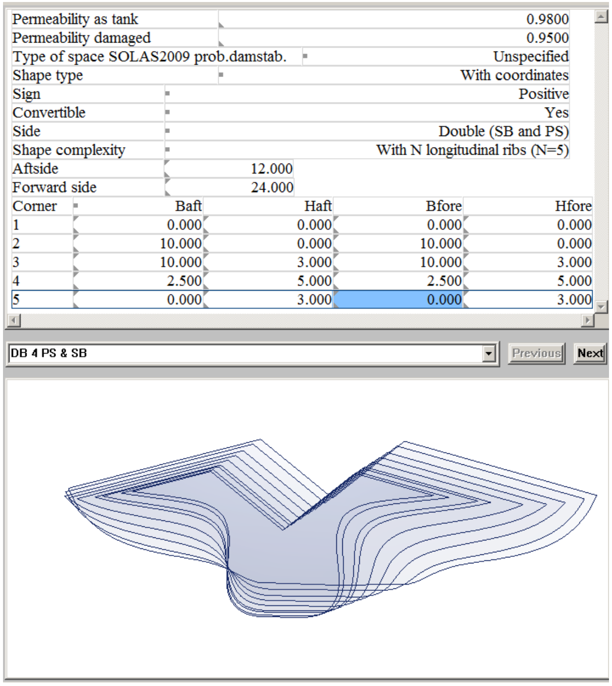
9.5.1.1.1 Types of subcompartments "with coordinates"

A subcompartment of the type 'with coordinates' is always defined with an aft and a forward boundary, and in each of it a number of points which represent the horizontal and vertical subcompartment boundary. In general, this is a flexible definition, enabling considerable freedom of shape, but since the major part of the subcompartments does not need this flexibility, a number of subtypes has been defined in order to increase userfriendliness:

Simple block
A 'simple block' is a limited interpretation of the general subcompartment definition, namely with straight horizontal and vertical boundaries. This type can be recorded with six numbers (aft, fore, inside, outside, upper and bottom).

With four longitudinal ribs
This is a slightly extended interpretation, where the number of ribs N=4, but where the longitudinal boundaries are not limited to purely horizontal or vertical.

With N longitudinal ribs
This type is even more extensive, so three-sided, five-sided or multilateral subcompartments can be defined with this setting. Note that the direction of rotation according to the vertex order must be **left turning** (counter clockwise). The number of longitudinal ribs 'N 'can be given via [cOordinates]. An example of what the subcompartment definition screen contains for a five-sided compartment is shown below:



Attention

As coordinate also ∞ can be entered, in which case the outer hull is taken as subcompartment boundary. That is quite handy, however, please realize that such an ∞ should be used for (the corresponding coordinate of) both aft boundary and forward boundary. The reason is that interpolation between finite and infinite is undefined. With such invalid input the program prints the warning 'corresponding coordinates of aft and forward boundary should both be finite or infinite', and leaves it to the user to correct the input coordinates.

9.5.1.1.2 Compartment functions

Adding and removing will be obvious; With [Insert] a compartment is added that is included in the list of compartments **before** the present compartment, and with [New] after it. [Copy] copies the compartment data, including all subcompartments to an internal clipboard. With [Paste] the compartment data, including all subcompartments, are copied from that internal clipboard to the present compartment. All existing compartment data (including subcompartments) are transcribed thereby. The difference between [Paste] and [Paste Link] is explained in the following paragraph.

9.5.1.1.3 Subcompartment functions

The functions [Insert] through [Remove] are entirely analogous to those discussed at the compartments, we refer to the previous paragraph. The [Paste Link] is related to references of subcompartments, as explained in section 9.1.4 on page 198, Links to subcompartments. With [Paste] the subcompartment data are copied to the present compartment, with [Paste Link] a reference is made from this subcompartment to the shape of the copied subcompartment.

9.5.1.1.4 Coordinates functions

These functions are related to a subcompartment of the type ‘with coordinates’. With subtype ‘other than four longitudinal ribs’, as discussed in paragraph 9.5.1.1.1 on the previous page, Types of subcompartments "with coordinates", there should be a facility to add or remove rows, in order to change the number of ribs. The first three functions ([Insertrow], [Newrow] and [removerow]) are meant for that. Because most subcompartments are prismatic (i.e. that the aft and front boundaries have the same shape), for practical purposes there is a [Copyaft] function, which will copy all coordinates from the aft boundary to the forward boundary. This function is also relevant for type ‘four longitudinal ribs’

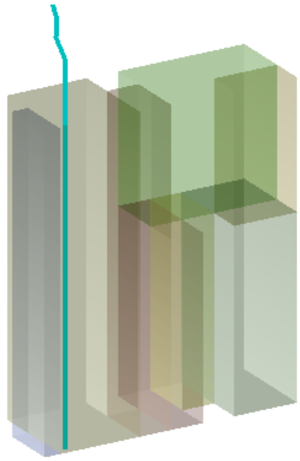
9.5.1.1.5 View functions

Without a special setting, all subcompartments of a compartment are drawn interchangeably in the left bottom subwindow, irrespective whether they are positive or negative. Mutual connections of two (positive) compartments are drawn then, although one could argue that they do not exist in a physical sense, and could therefore be omitted. With the [visually composed] function active subcompartments are actually composed graphically, so that they render a more realistic image.

Attention

> At this ‘visually composing’ of subcompartments, these are neatly cut off when they overlap. The shape of negative subcompartments is also deducted from those of the positive ones. This offers a good insight, but do remember that the **calculation** is based on the ‘bare’ subcompartment shape, so overlappings are calculated double, and a too large negative subcompartment may result in a negative compartment volume.

When the [Surfacemodel] function has been activeated, the (sub)compartments are not drawn as frame model, but as surface model instead. All this on the condition that either a surface model of the hull is available (see section 2.10.2 on page 16, Hull form representations) or the (sub)compartment is not at all cut through the hull. If one visualizes a (sub)compartment with a surface in this manner, one can also switch on the [Transparent] function, with which the surfaces become partially transparent, so that also the sounding pipe remains visible. That renders such images:



Semi-transparent compartment with sounding pipe.

9.5.1.1.6 Tank tables

This menu bar option only contains one option, [Calculate], for calculating the tank tables of the current compartment. For further information see section 9.5.2.2 on page 222, Calculate: compute the tank tables.

9.5.1.2 Compartment data

9.5.1.2.1 Selected

Indicates whether this compartment has been selected for further actions, such as calculations and output.

9.5.1.2.2 Compartment

The (unique) name of the compartment. Although fifty characters have been reserved for the names in Layout, the input here is for the time being limited to 28, for the reason that this 28 still is the maximum in subsequent modules, such as Loading. When also those modules have been upgraded to the higher maximum, the input limitation will be removed.

9.5.1.2.3 Second name and abbreviation

These are supportive names that can be included in some output for cosmetic or explanational purposes. For example, the second name at tank tables, and the abbreviation, of maximally eight characters, at the tank plan (since too long names will soon mix up there through several tanks). One can also choose to have this second name generated permanently, please refer for that to the settings as discussed in section 9.7.5 on page 235, Layout project settings and function colors.

9.5.1.2.4 Design weight group

Indicates to which weight group the volume of the compartment belongs. This value is used as *default* in loading conditions. If such a value is not known or desired, one may also opt for ‘undefined’. The purpose of weight groups and its defining has been discussed in section 9.7.7 on page 236, Define weight groups.

9.5.1.2.5 Uniform permeabilities

Permeabilities can be defined here for all subcompartments in one action. See paragraph 9.5.1.3.2 on page 219, Permeabilities for further merits.

9.5.1.2.6 Design density

Here the density (specific weight, in ton/m^3) of the liquid for which this tank is intended can be given. This value is used as *default* in loading conditions. If such a value is not known or desired, one may also opt for ‘not specified’, then there is no default.

9.5.1.2.7 Uniform subcompartment sides

One action can record the side for all subcompartments here, see paragraph 9.5.1.3.6 on page 219, Side for the options.

9.5.1.2.8 Sounding pipe

Two sounding pipes per compartment can be defined. Each of them takes up one row, where can be defined:

- The name of the pipe. This is initially a default name, depending on the selected output language at the moment the compartment is created, but these names can be changed manually.
- With 'selected/deselected' in the right column one indicates whether this pipe has been selected for output, such as drawings and tank sounding tables. Besides being selected each pipe also belongs to an exclusive category ('A' to 'J'), which enables the output script to identify the desired sounding pipes in the sounding tables (see paragraph 9.5.2.1.2 on page 221, Output scripts for details).
- With [enter] a window pops up in which one can define the coordinates of the pipe, which can also be referred (via <F5>) to the reference planes, a feature that might be useful in the event of future design modifications. The maximum number of coordinates is fifty, so that also curved pipes can be properly modelled. Furthermore, for verification purposes in the status line, right-under in the window, the total pipe length is represented. Finally, this window als has a [Selected] function which does exactly the same as the 'selected/deselected' of the previous line. As the rule a pipe will consist of two or more points, however, it is also possible to define a "pipe" with only a single point. Its effect will be discussed in paragraph 9.5.2.1.2 on page 221, Output scripts.

9.5.1.2.9 Special points / openings

Characteristics can be defined here of specific items that belong to the compartment. There are several predefined types of such items:

Open opening
: This is an open opening to the outside, which is connected with the compartment, for example, an unprotected vent.

Weathertight opening
: An opening to the outside which is connected with this compartment and protected such that it can be considered to be weathertight. Some authorities consider a vent cap to be sufficient protection to that end, others not.

Watertight 'opening'
: Obviously, this type has no effect. it is included for convenience, for example to be able to toggle an opening 'off', or to indicate that one has not forgotten or ignored a closable opening, but that it really securely closed.

Emergency exit
: See section 15.5.1 on page 294, Types of parameters, **Emergency exits to be included**.

Hopper edge or overflow
: Are solely intended for open hopper compartments, from which cargo can pour out. Their details are discussed in section 18.3 on page 385, Specify additional hopper properties.

Alarm sensor
: In order to be able to process its effect in tank tables and maximum tank filling.

Pressure gauge
: Its location is important for the calculation of pressure tables (i.e. the tables that indicate which tank filling belongs to a specific sensor pressure), and in order to be able to determine the corresponding volume at loading conditions and/or loading software at a known sensor pressure.

At [enter] a window pops up in which can be defined:

- Whether the point has been selected is defined in the last column. Only selected points result in an action. When a point has not been selected, it is just as if it does not exist at all. So selection is intended to 'throw away' something as it were, while it can be restored later on. Some points can also be selected from a certain exclusive category, that serves the same purpose as with sounding pipes, as discussed just earlier.
- The name.
- Length, breadth and height coordinates of the special point.
- The type of point, according to the definition list just above.
- The piping network to which a point belongs.

Apart from that, ventilation openings are traditionally defined in PIAS in a separate list, managed by the module Hulldef. This list will remain because it also contains other types of points, such as those of the margin line. In order to prevent inconsistencies Layout completes this list again and again with (selected) openings of compartiments, but marks them such that they cannot be modified or removed in Hulldef. If one should wish to manage all openings of the vessel in a single overview list, the Layout option as discussed in section 9.7.1 on page 233, List of openings and other special points is recommended.

9.5.1.2.10 Compartment is an open hopper

PIAS can compute the stability of hopper dredgers, which is discussed in chapter 18 on page 384, Stability of open hopper vessels. Anyway, it must be specified which compartment(s) should be considered as open hopper for this purpose. That can be done by switching this cell to 'yes'

9.5.1.2.11 Is a hold with a RoRo-deck (STAB90+50)

This option is currently under development, and not yet been released for general use.

9.5.1.2.12 Type of space SOLAS0920 prob.damstab. (uniform)

Space types can be defined here for all subcompartments in one action. See paragraph 9.5.1.3.2 on the following page, Permeabilities for further merits.

9.5.1.2.13 Oil outflow parameters

- Type of tank for oil outflow calculations: for the benefit of probabilistic outflow calculations (with Outflow) it must be known wheter a specific compartment is a fuel oil tank or a cargo oil tank (at least, within the meaning of the regulations involved). That can be defined here.
- Tank adjacent to bottom: for that same outflow calculation it may be important whether a tank borders at the bottom on the plane, or on a non-oil tank. That can be defined here.
- Overpressure of inert gas system: cargo oil tanks can be provided with inert gas systems. If such is the case, then it is of importance for the outflow calculations to define its overpressure here (in kiloPascal).

9.5.1.2.14 Automatic inclusion in loading condition

Here it can be specified whether this compartment should automatically be included in the weight items list as a tank, or hopper if applicable, when creating a new loading condition in Loading. Equally, when in Loading the function 'Add missing tanks' (in the weight item menu, see section 16.2.1 on page 307, Define/edit weight items) is used, then only those missing tanks are added whose property as discussed here is set to 'Yes'.

Note

> In the overview menu of compartments, section 9.5 on page 212, Compartment list, calculation and printing of tank tables, the column header 'Automatic inclusion in loading condition' would be a but spatious, so there the shorthand 'In loading condition' is used.

9.5.1.2.15 Is part of watertight ships' layout

Here it can be specified whether the compartment is part of the ship's watertight layout and should be included when defining or generating the damage cases. Sometimes additional compartments are defined that are used in certain calculations, e.g. grain holds for the calculation of grain moment tables. However, they may also be temporary compartments that are in use during the design phase of the ship. If a compartment is not a part of the watertight layout, it is not included when generating the damage cases in deterministic and probabilistic damage stability, and the compartment cannot be selected when manually defining a damage case.

9.5.1.2.16 Computation script, Output script and Calculated

- Computation script: see paragraph 9.5.2.1.1 on page 221, Computation scripts.
- Output script: see paragraph 9.5.2.1.2 on page 221, Output scripts.
- Calculated: where it is indicated per compartment whether the tank tables have already been calculated. Besides 'yes' and 'no' it might also be indicated that the table is available, albeit outdated. That will be the case if the compartment shapes has been modified after the most recent tank table calculation. With <Enter> a sheet is opened with all calculated values. It is possible to modify these values, but please keep in mind that those will be lost on re-computation. The table can also be modificied by the addition or deletion of lines, however, no more than the original amount of lines will be accepted. If tables for multiple trims are computed the menu bar functions [Nexttable] and [Prevtable] can be used to traverse these tables.
- Max. vol.: the maximum volume based on the compartment definition.

9.5.1.2.17 Convertible

Here one can indicate whether a subcompartment at the executing of option section 9.11.1 on page 242, Generate physical planes from the totality of convertible subcompartments must be included in the automatic conversion of the type 'with coordinates' to 'space generated between planes'. This effects the 'convertibility' of all subcompartments of the compartment. Here are four possibilities:

Automatically at conversion
At the conversion is firstly considered whether the compartment overlaps another one (and complies with other requirements, such as the completene flatness of the boundary planes). If so, it will not be converted; if not, it will be converted.

Non-convertible
Will be obvious

Is convertible
Idem

Define per subcompartment
Which is used when it cannot be recorded for the compartment as a whole whether it must be converted, but this has to be defined at the more detailed level of subcompartments, as described in paragraph 9.5.1.3.5 on this page, Convertible.

9.5.1.2.18 Number of constraints

Here the number of constraints that apply to the compartment can be inputted, with a maximum value of 20. Each constraint can then be linked to a constraint from the constraint table via a popup-window.

9.5.1.3 Subcompartment data

9.5.1.3.1 Subcompartment

The name of the subcompartment, which should be unique within the compartment.

9.5.1.3.2 Permeabilities

A compartment has a permeability, which is commonly depicted by a μ. Here in PIAS there are **two**, namely the permeability where is calculated at the tank volume calculation, and the one where is calculated at the damage calculation. Physically, such a distinction can of course not be maintained, but naval architectural practice has shown that for tank volume calculations often a permeability of 0.98 to 0.995 is used, while the rules for damage calculations often prescribe a value of 0.95. The calculation of the actual grain heeling moments, as determined by module Grainmom, uses the 'permeability as tank'. The permeability has been defined as the total volume, taking into account construction parts divided by the total volume without taking into account construction parts, and therefore as a rule has a value between 0 and 1.

Some rules, in particular the probabilistic damage stability according to SOLAS 2009 and 2020, use a permeability that varies with the draft, and with the type of space. So one can choose at the option 'type of space prob.damage stab. SOLAS0920' from the types of spaces of SOLAS.

Apart from that, it will only occur seldomly that permeabilities or types of spaces differ among subcompartments of the same compartment. When they are all equal, it is more practical to define these data at the **compartment**, since the permeabilities for all subcompartments are entered then by one action.

9.5.1.3.3 Shape type

The type of subcompartment. There are three types, as introduced in section 9.1.1 on page 195, Definitions, namely 'with coordinates', 'space generated between planes' and 'external PIAS hullform'.

9.5.1.3.4 Sign

Positive or negative, resp. whether this subcompartment has to be added to the subcompartment or whether it has to be deducted from it. This sign can not be filled in at subcompartments of the type 'space generated between planes', since they are always positive.

9.5.1.3.5 Convertible

One can indicate here whether a subcompartment at the implementation of option section 9.11.1 on page 242, Generate physical planes from the totality of convertible subcompartments must be included in the automatic conversion of the type 'with coordinates' to 'space generated between planes'. This row only appears when one has entered at the **compartiment** at the 'convertibility' that this can be set per subcompartment (see paragraph 9.5.1.2.17 on the previous page, Convertible).

9.5.1.3.6 Side

SB An asymmetrical subcompartment that is only at SB.

PS An asymmetrical subcompartment that is only at PS.

Double
A symmetrical subcompartment of which only the SB half has been defined, which is reflected to PS.

According to coordinates

Where the subcompartment is simply recorded by its coordinates, without specific symmetry assumptions. According to PIAS convention, the transverse coordinate is positive to SB, negative at PS.

9.5.1.3.7 Shape definition external subcompartments

File name and **length**, **breadth** and **height shift**, These are the parameters from the *external PIAS hull form*, a concept which is introduced in section 9.1.1 on page 195, Definitions, where the file name from the PIAS shape definition (as recorded with Hulldef or Fairway) and the shift of the origin of that definition to its position of this subcompartment are defined respectively. The filename can, by using the &-symbol, also be specified as being located in the same folder as the hull form file. Indeed, this is encouraged, reference is made to section 7.2.2 on page 172, Hullforms for some more detail.

9.5.1.3.8 Shape complexity

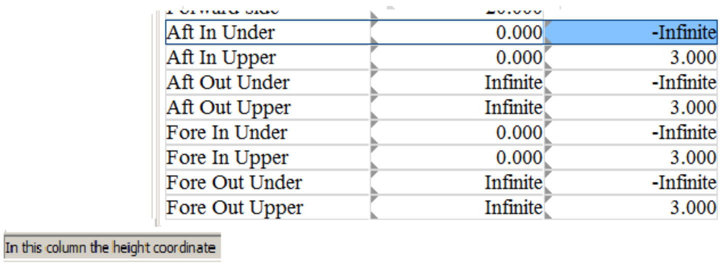
At a subcompartment of the type 'with coordinates' one can define here whether the subcompartment is a simple block, that can be recorded with six digits, or has a slightly more complex shape, for which more digits are required. One can choose here from the three types mentioned at paragraph 9.5.1.1.1 on page 214, Types of subcompartments "with coordinates".

Attention

When a subcompartment shape is of the type 'space generated between planes' then the indication "cannot be displayed in coordinates" can occur. That does not mean that the shape is wrong or useless. Such a subcompartment shape is entirely acceptable, but it only cannot be displayed with longitudinal ribs that run from aft to front. It would be possible to further split up the shape so that displayable shapes are generated, but that increases the number of subcompartments, so it was opted for not to do that.

9.5.1.3.9 Subcompartment coordinates

The final part of the subcompartment definition window consists of the coordinates, so the aft boundary, fore boundary and the other breadth and height boundaries. Where it is not evident whether it concerns a breadth of height boundary, it is indicated in text in the bottom bar, see the picture below. The coordinates are only shown at the types 'with coordinates' and 'space generated between planes'. At the last type they are only for information purposes, and they can therefore not be changed (the borders are then after all entirely determined by the physical planes). If 'With N longitudinal ribs 'has been choosen, 'N 'can be given via [cOordinates].



Indication, lower left, whether the cell contains a breadth or height boundary.

9.5.2 Calculate and print tank tables

The tank volume tables options are activated with the menu bar optie [tAnk tAbles]. It contains four suboptions that will be addressed below.

9.5.2.1 Setup: define computation scripts and output scripts

With a *script* the output of the tank tables is specified. Scripts come in two flavours, the *computation scripts* where arithmetical issues are specified, such as step size and trim range, and *output scripts* where quantities and units are specified. For the sake of flexibility for both types of scripts multiple versions can be specified.

9.5.2.1.1 Computation scripts

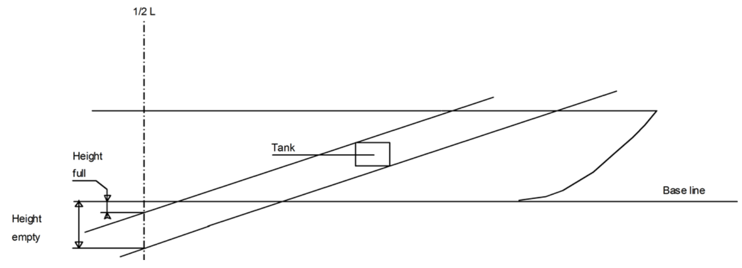
The computation script determines the way the tank tables are computed, by specifying:

- The name of the script. Multiple scripts may be defined, and per compartment the appropriate one should be selected>, and this script name serves for identification purposes.
- Which script is 'default'. If a compartment does not refer to a specific script, this 'default' is used at the calculation.
- The calculation step, which is the vertical step (in meters) on which the table is calculated.
- The output step, which is the step on which the table is printed. If in a script table in the first column a distance-quantity is used (such as height, ullage or sounding) than the table is printed with this step size in meter. If an other quantity is used in the firstcolumn (such as volume) than the table is printed with a step size that roughly corresponds with the output step in meters as specified here. In order to prevent the combination of a large computation step and a small output step, in the end the maximum of the two is applied.
- The trim, or trims. A single trim can be specified in this cell directly, for multiple trims a dedicated subscreen appears after pressing <Enter>.
- Whether tables should be produced for all angles of inclination. If filled in with 'yes' then tables will be produced for all combinations of the trims as specified here in Layout, and the angles as given in Config (as is discussed in section 5.2 on page 45, Angles of inclination for stability calculations). **If the inclinations are modified in Config then the possibly existing tank sounding tables are not declared invalid automatically.** So, they should be explicitly removed with option 'remove alle calculated tank tables', please also consult section 9.5.2.4 on page 224, Remove: remove alle calculated tank tables.

9.5.2.1.2 Output scripts

An output script specifies which parameters are to be included in a tank table and in which units. Those parameters are:

- Height, which is the height of the level of the liquid related to the PIAS system of axes, see sketch below. When a tank table is computed with trim, large positive or negative 'heights' can occur, because the 'heights' are defined from baseline at Lpp/2. The first height is at a just empty tank, the last height at the tank just completely filled.
- Ullage, this is, with a sounding pipe of at least two points, the 'dry' distance through that pipe between the last (=highest) point and the liquid level. As discussed in paragraph 9.5.1.2.8 on page 217, Sounding pipe, a "pipe" may consist of only one point. This is then the ullage reference point from which is measured downwards, and which is therefore intended to lie above or at the top of the tank. The ullage is then the perpendicular (i.e. taking into account trim and heel) distance between that point and the liquid level.
- Sounding, which is the complement of ullage, i.e. the 'wetted' distance through the pipe between the first (=lowest) point and the liquid level. A "pipe" consisting of a single point defines the sounding reference point, from which is measured upwards, and which is intended to be somewhere at the bottom of the tank. The sounding is in this case also the perpendicular distance between that point and the liquid level.
- Volume, weight, VCG, LCG and TCG will speak for themselves.
- ITransverse and ILongitudinal are the transverse and longitudinal moments of inertia.
- FSM's are the free surface moments.
- Pressure, for which the corresponding compartment must be equipped with a pressure sensor, the position of which can be specified in the menu of paragraph 9.5.1.2.9 on page 217, Special points / openings. The pressure is, obviously, calculated perpendicular to the waterline (= liquid level in the tank).



Definition of 'height' in a tank table.

Each output script has a name so for each compartment the script to be applied can be identified easily. Per script it can be specified which quantities should be printed, in which order and in which unit. Each script has an input screen the quantities and units. The first row of this screen will be the first column in the table, the second row corresponds the the second column etc., so you can specify freely the desired number of tank table columns. The last column of the script contains the label ‘category’, which is only applicable for the quantities sounding, ullage and pressure. This ‘category’ is a letter from A to J and its purpose is to identify the applicable sounding pipe or pressure gauge. For multiple pipes or sensors can be defined per compartment, and for each tank table it should be specified which of those to use. Pipes and sensors also possess such an A to J category, so serves as identification mechanism.

9.5.2.2 Calculate: compute the tank tables

With this option the tank volume tables of all selected compartmemnts are calculated, according to the specified computation scripts, and according to the setting ‘tables with everywhere the maximum free surface moment’ of Config and as discussed in section 5.3 on page 45, Settings for compartments and tank sounding tables. With the tank tables also the computation time is saved, so a re-computation can be done rather efficiently, because only those tanks that have been modified since the last computation are actually re-computed. If an unconditional re-computation of all tanks is deemed neccessary, then all existing tables should be removed, this is discussed in section 9.5.2.4 on page 224, Remove: remove alle calculated tank tables. By the way, with the switch ‘re-calculate tank tables automatically’here in Layout, see section 9.7.5 on page 235, Layout project settings and function colors, the tables are being calculated whenever required, so with this switch set it will not be neccessary to calculate here explicitly.

Attention

> Tank volume computations are, obviously, based on the shape of the tanks, but also on the location and shape of the hull frames (as defined in Hulldef). This implies that (even for tanks that do not intersect the hullform) the number of frames may have its effect on the tank computation result. For a discussion on the effect of the number of frames reference is made to section 7.2.4.1 on page 174, Number of frames.

9.5.2.3 Print: print tank tables

TANK VOLUME TABLE
Example vessel

DB 4 SB — 22 mar 2013 17:40:26
Trim = 0.000 m — Specific weight = 0.880 ton/m³

Sounding m	Ullage m	Volume m³	Volume feet³	Weight ton	VCG m	LCG m	TCG m	Mom.In.T m⁴
-	-	0.000	0.000	0.000	-0.001	42.208	0.085	0.000
0.100	1.750	0.305	10.766	0.268	0.064	41.006	0.214	0.197
0.200	1.650	1.179	41.622	1.037	0.132	40.279	0.368	1.368
0.300	1.550	2.724	96.203	2.397	0.201	39.722	0.520	4.350
0.400	1.450	4.995	176.387	4.395	0.270	39.240	0.662	9.321
0.500	1.350	8.005	282.698	7.044	0.339	38.802	0.792	16.785
0.600	1.250	11.752	415.025	10.342	0.407	38.404	0.912	26.410
0.700	1.150	16.246	573.715	14.296	0.474	38.017	1.020	37.322
0.800	1.050	21.476	758.419	18.899	0.542	37.644	1.118	50.878
0.900	0.950	27.441	969.087	24.149	0.609	37.285	1.209	65.191
1.000	0.850	34.148	1205.912	30.050	0.676	36.928	1.293	81.481
1.100	0.750	41.593	1468.849	36.602	0.743	36.576	1.370	97.975

Example of a tank table according to output script.

With this option tank tables will be printed, in a number of variants (from which three are included here as example in the figures):

- According to the output script, which allows for a good control over layout and content of the tables. If sounding or ullage is included in this table, it may contain a remark that the sounding pipe is too short. This means that the tank can be filled to a higher level than the top of the pipe, and implies that at a reading at exactly the top of the pipe, the volume may have any value between the volume which corresponds exactly with that level, and the maximum volume. In order to have an indication of the maximum volume, that value is printed. However, an additional line may be printed which is valid for a level exactly through the top of the pipe (as well as an additional line with elucidation).

- Litre tables, or sounding/litre tables, to put it more precisely. This table does not contain other data, so it is rather condensed.
- Trim tables, where for a range of trims the volumes are depicted. Trim tables come in two fashions:
  - On basis of sounding and ullage, where at each line at the sounding (in meter) for each trim the corresponding volume can be read. If a sounding pipe contains more than one point, the corresponding ullage is also included.
  - On basis of pressure, where at each line at a certain pressure (in mm water) for each trim the volume can be read.
- Correction tables, which represent the deviation in tank volume due to list and trim. These kind of tables are a bit outdated, for if the volumes are directly calculated at multiple trims (e.g. with the 'trim tables' option above) such corrections are not neccesary at all. Anyway, the *modus operandi* with the correction table is:
  - Find the tables of the intended tank (there are two tables for each tank).
  - Look up the first correction value in the trim table at the measured sounding/ullage and trim.
  - Look up the second correction value in the list table at the measured sounding/ullage and list.
  - Add those two correction values (which are in millimeters) to the sounding/ullage. This is the sounding/ullage that can be used to read out the tank volume at zero trim and list.
- Summary, which produces a table summarizing maximum volumes and COG's etc. of all (selected) compartments. Please bear in mind that maximums of volume and moments of inertia are really the maximum values that occur *anywhere* in the tank. They do not have to belong to the same filling level, e.g. with a U-tank the maximum volume is reached, obviously, with a completely filled tank, while the maximum moment of inertia will occur at partial filling, with the liquid level in the bottom part of the tank.

TANK VOLUME TABLE
Example vessel

DB 4 SB — 22 mar 2013 17:25:10
Trim = 0.000 m — Specific weight = 0.880 ton/m³

Sounding cm	Vol. litre	Sounding cm	Vol. litre	Sounding cm	Vol. litre	Sounding cm	Vol. litre	Sounding cm	Vol. litre
0	0	35	3771	70	16246	105	37779	140	67915
1	9	36	4002	71	16734	106	38526	141	68883
2	22	37	4240	72	17229	107	39282	142	69857
3	40	38	4485	73	17734	108	40045	143	70836
4	63	39	4736	74	18246	109	40816	144	71819
5	89	40	4995	75	18765	110	41593	145	72808
6	121	41	5260	76	19292	111	42377	146	73802
7	159	42	5533	77	19827	112	43167	147	74801
8	203	43	5816	78	20369	113	43964	148	75805
9	251	44	6106	79	20919	114	44769	149	76814
10	305	45	6404	80	21476	115	45580	150	77827
11	363	46	6710	81	22040	116	46398	151	78845
12	429	47	7023	82	22612	117	47223	152	79868
13	502	48	7343	83	23191	118	48055	153	80895
14	580	49	7670	84	23777	119	48893	154	81927
15	665	50	8005	85	24370	120	49738	155	82964
16	755	51	8347	86	24970	121	50590	156	84004
17	851	52	8696	87	25577	122	51447	157	85050

Example of a table of litres.

TANK VOLUME TABLE
Example vessel

Double Bottom tank 3 08 dec 2014 12:42:57

Volume and COG at maximum filling
Volume 162.416 m^3 (All volumes in cubic meters)
VCG 1.069 m
LCG 52.319 m
TCG 0.000 m

Sounding [m]	Ullage [m]	Trim in m, negative by stern, positive by bow -4.000	-3.000	-2.000	-1.000	0.000	1.000	Even keel VCG	LCG	Mom.In.T
0.000	2.200	0.577	0.719	1.188	2.234	4.378	7.962	0.124	52.539	20.058
0.100	2.100	1.628	2.105	3.311	5.535	8.801	13.321	0.189	52.499	50.783
0.200	2.000	3.518	4.614	6.901	10.251	14.496	19.832	0.253	52.462	95.592
0.300	1.900	6.446	8.475	11.884	16.207	21.304	27.356	0.316	52.432	151.453
0.400	1.800	10.578	13.702	18.077	23.244	29.086	35.775	0.379	52.408	216.102
0.500	1.700	15.980	20.106	25.328	31.235	37.737	44.995	0.441	52.389	288.002
0.600	1.600	22.545	27.554	33.513	40.075	47.166	54.928	0.503	52.373	363.354
0.700	1.500	30.137	35.915	42.522	49.663	57.276	65.489	0.564	52.361	439.746
0.800	1.400	38.629	45.078	52.258	59.916	67.996	76.614	0.625	52.351	517.380
0.900	1.300	47.909	54.951	62.644	70.764	79.263	88.244	0.686	52.343	593.506
1.000	1.200	57.889	65.458	73.609	82.143	91.016	100.323	0.746	52.337	667.240
1.100	1.100	68.490	76.530	85.090	93.994	103.204	112.805	0.805	52.332	738.669
1.200	1.000	79.645	88.106	97.034	106.271	115.785	125.654	0.865	52.328	807.508
1.300	0.900	91.296	100.134	109.393	118.932	128.721	138.834	0.923	52.325	873.609
1.400	0.800	103.393	112.571	122.130	131.942	141.982	151.578	0.982	52.322	936.685

Example of a trim table.

9.5.2.4 Remove: remove alle calculated tank tables

With this option all calculated tank tables will be removed. Actually, there is no reason to do so, because the program keeps track of the tanks whose shape has changed since the last calculation, so the tank tables is known to have become obsolete and must be re-computed. Exceptions occurs if the switch ‘tables with everywhere the maximum free surface moment’ is toggled or the angles of inclination are modified in Config (and as discussed in section 5.3 on page 45, Settings for compartments and tank sounding tables and section 5.3 on page 45, Settings for compartments and tank sounding tables respectively). In those cases the existing tables have to be removed manually with this ‘remove’ option. And, if for whatever reason you wish to remove tank tables, this is also the option to use.

9.5.2.5 Areas: print table of outer surface areas

With this option the outer surface area of the (selected) compartments will be computed and printed, with the purpose to serve as approximation of the compartment’s paint area. When applying this table one should realize that:

- Construction parts in the compartments and on the outer walls are not defined in PIAS, so their areas are not included.
- With compartments adjacent to the shell the computation is based on the envelopped frame lengths, so the accuracy is more or less proportinal to the number of frames within a compartment. Additionally, for these compartments the area computation will not be accurate if in an asymmetrical hull shape the frame locations — as far as falling within the compartment — of the PS hull are not the same as for the SB hull.

9.6 Pipe lines and piping systems

Although Layout was initially designed for defining and visualizing the geometry of internal spaces, it was expanded around the 2020s to include pipeline modelling functions. This created a system that is intended and suitable for:

- Integrated bulkhead/deck & compartment & pipe modelling and visualization.
- Including the effects of pipes on damage stability, and deriving intermediate stages of flooding on basis of the connections between compartments.
- Time-domain damage stability analysis, including the effects of fluid velocity and resistance in the piping networks.
- The exchange of PIAS’ piping data with engineering software is still on the wish list. However, due to the lack of a common interface format for this kind of data, this is not expected to become universal.

9.6.1 The piping data structure

For the sake of clarity, the piping-related manual has been split into two parts; the part concerning geometry and compartment connections fits here, in Layout, while the more fluid-flow related part better fits in the chapter on internal flooding in case of damage, see section 21.2 on page 403, Flooding through ducts and pipes: Consecutive Flooding, after 2022.

The data structure and naming conventions of PIAS' piping is inspired on the ISO-15926 standard, which has an origing in the process industry. The primary elements are:

- **Equipment**, which is a thing, not being a compartment, connected to a pipe line but not part of it. Such as an engine or a chiller. In the end, PIAS performs no actions with equipment, it is just defined in PIAS for the completeness of definition and drawings.
- A piping **system**, which is an administrative collection of pipes of the same type, for example "ballast" or "HFO". Such a system can optionally belong to a weight group — as also applied for compartments and weight items in PIAS, and as discussed in section 9.7.7 on page 236, Define weight groups. A piping system can also be assigned a color, e.g. according the ISO 14726[5] standard.
- A piping **network**, which is one closed system of connected pipes, which belongs to a piping system. A network is the core of the piping data structure.
- A piping **segment**, which is one branch of a piping network, and extends between two points without sub-branching inbetween.
- A piping **connection**, which is a part located at the extremities of a piping segment. Such a connection comes in six types:
  - Branch, where multiple piping segments meet.
  - Unprotected opening, external (so, to the outside). A nice feature of this type is that such an opening will also be included in the list of openings per compartment (see paragraph 9.5.1.2.9 on page 217, Special points / openings), as well as the overview menu of openings, section 9.7.1 on page 233, List of openings and other special points. In both lists you will be able to change the location of the openings, but for obvious reasons are not allowed to modify other properties or remove the opening.
  - Opening + WAPCD (Weathertight Air Pipe Closing Device), vent check valve.
  - Terminator, which closes a dead-end segment.
  - Compartment, or, more precisely, a connection to a compartment at a certain location. So, a compartment may have multiple connections.
  - Equipment, or, more precisely, a connection to a piece of equipment at a certain location.

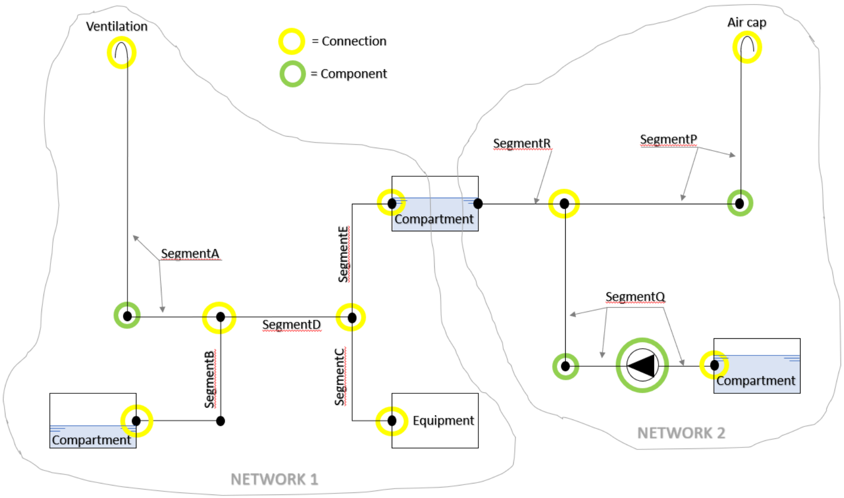
  For each connection also its position can be given, as well as the resistance coefficient, optionally including or excluding the resistance due to the energy loss at the outlet of a pipe (see paragraph 21.2.3.1.2 on page 406, Fluid outlet energy loss}.
- A piping **component**, which is a part located **in** a piping segment. These come in two types:
  - Point components, located in a (virtual) point (from which als the coordinates can be given):
    - Waypoint, which is a geometrical point without further properties. Can be applied to define the pipe geometry.
    - Elbow.
    - Valve, including open/closed indication.
    - Pressure relief valve, including the opening pressure. An important reason to include this type is for the modeling of fire control bulkheads, which are considered to be watertight until a certain water level (=pressure), and collapse at higher levels. This valve opens *inside* the pipe, and allows fluid flow in the pipe when opened.
    - Reducer, including the regular flow direction (from high to low pressure), as well as the regulared pressure at the exit. Its way of operation depends on the flow direction:
      - If the flow is in regular direction, then for the hydraulic calculations the set pressure at the exit is applied.
      - If the flow is in the opposite direction, the reducer is assumed to be closed if the pressure at the regulated side (=the original exit) is higher then the set pressure. Otherwise it is open, where the hydraulics arte being computed with calcuthe standard resistance coefficient.
    - Check valve (non-return valve), including the free flow direction indicated by an arrow.

[5] http://www.iso.org/iso/catalogue_detail.htm?csnumber=44744

    * Weathertight Air Pipe Closing Device (WAPCD, Vent check valve) including the free flow direction.
  - Pipe components, which exist between two point components. This type has just a single member:
    * A pipe section, which is simply a straight pipe between two point components.

Furthermore, for components also their resistance coeffcient can be given as well as their dimensions. Pleased observe that an opening or a WAPCD can be given both as a component as well as a connection. This seems superfluous, however there is a distinct reason:

- As a connection will be used at the end of a segment, typically as a weathertight opening to the outside, for example just above main deck.
- As a component will be used *internally* in a network, for example a weathertight opening on the vehicle deck of a Ro-Ro ship, covered by the Ro-Ro hold, which in itself is 'just' a compartiment in the ship.



Example of two networks with a number of parts.

With this structure, complex pipelines can be modelled and connected with openings and compartments, as demonstrated by the sketch above. To increase the ease of use, PIAS' pipeline system contains a number of auxiliary functions and concepts that are not essential, but can be useful, such as:

- A complete pipeline definition contains the coordinates of all its parts. Sometimes, however, it will be sufficient to define only the *connections*, without coordinates. In such cases networks or components can be specified to be **without geometry**. This can be used, for example, when specifying a standard standard vent right above a tank, in which case the exact location of the connection of the vent pipe to the tank is not important (at least, if one is satisfied that the geometry of the pipeline network is not involved in the damage stability computation).
- A ship model defined without these pipelines will still contain openings and sounding pipes. In the essence, these obviously maintain a relationship with pipelines. To facilitate the upgrading of such a model, there are auxiliary functions to convert these entities to pipelines, see the [Generate] functions in Piping systems (discussed on the following page).
- For complete flow calculations, the shape, size and resistance coefficient of all pipes and components must be provided. Since in reality there can be considerable uniformity in this respect, these data can also be provided at a higher level, e.g. for an entire pipe system, in order to save typing. This is further explained in section 21.2.3.3 on page 407, Layered definition of flow-related parameters.

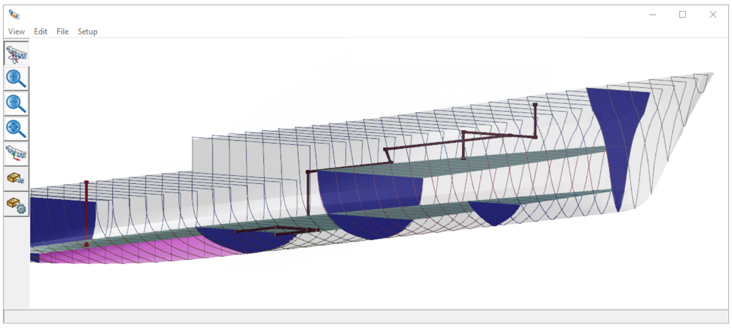
The piping's initial menu contains the following options:

Pipe lines and piping systems

1.	Piping systems
2.	Piping networks
3.	Equipment
4.	Check the input
5.	General piping settings
6.	Output of geometry, connectivity and resistance factors of piping

The following is worth noting about this list:

- The first two options are a bit duplicative, i.e. that they both provide an entry to the same individual networks. Through the first option, those are grouped by system, and through the second option, a complete list of networks comes up. Both options can come in handy.
- There is additional section with instructions on how to define particularities one may encounter in practice in the ship. However, that is discussed only after the main structure has been explained, in Modelling specific things from the real world (discussed on page 232).
- There is another place where the pipe lines can be made visible, and that is in the three-dimensional presentation — as discussed in section 9.8 on page 237, Threedimensional presentation — from which an example is included below.



Piping included in the rendered view.

9.6.2 Piping systems

This menu lets you specify a number of properties of pipeline systems. As indicated in the introduction, a **system** is a collective name for pipelines of the same type, for example “bilge/ballast”. Properties in this menu that need explanation are:

- Presentation, which can be used to specify whether and how the system will be included in the 3D presentation and subdivision plan.
- Selected, which indicates whether this system will be included at damage stability calculations.
- Sounding, which indicates whether pipes of this system are considered to be sounding pipes of compartments. If that is the case then those pipes are automatically added to the sounding pipes of the compartment they are connected to. This is convenient because then those pipes only need to be specified once. In the sounding pipe definition menu of that compartment those pipes will also be visible, but they cannot be changed there.
- For the *cross sectional properties* of the last columns reference is made to Frictional resistance from pipes lines (discussed on page 406).

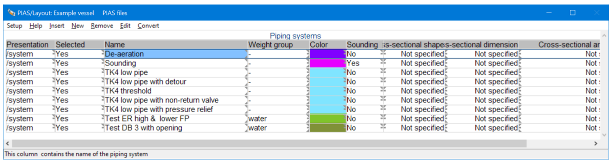
In addition, this menu has three more functions, the first two of which are primarily intended for conversion of older projects (although their use is not necessarily limited to that):

- [Convert / Openings], which, starting from already defined (de-airation) openings of compartments, generates pipelines that contain these openings. This function is clearly intended for the upgrading of elder ship files (with only single locations of openings defined) to a more complete definition which also includes the pipe between compartment and opening. Of course, the generated pipeline shape may need further detailing with elbows or other geometry (because that information was not available initially).

- [Convert / Sounding pipes] is analogous, it converts the existing (loose) sounding pipes of the compartments into pipelines that fit into the integral pipeline system as discussed here.
- <Enter>, which, if the text cursor is on a particular network, takes you one level deeper, to the details of that thereof. Into the piping network menu which is discussed below.

9.6.3 Piping networks

This option opens with the list of networks, like the example below, from which one can be entered for further editing or extension.

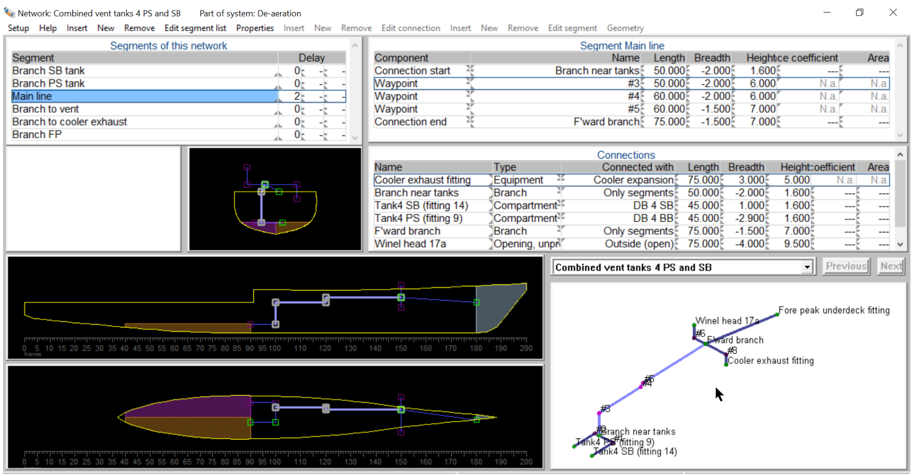


List of defined networks, with a number of properties.

The data structure around the pipe lines is sometimes a bit virtual, e.g. a ‘system’ is nothing but an administrative grouping of similar networks. The network, on the other hand, is real; that is a system of pipes into which one can pour water at one end, which can then come out at other ends.

The essence of a piping network is that it consists of segments and components, which can have their interconnections, properties and geometry. On top of that, the network itself has a number of properties, which are accessable through the [properties] menu bar button in the GUI of the figure above, or from the networks list of two figures above. However, that will be discussed later, see Properties of piping networks (besproken on page 230), first we look at the contents of this GUI:

9.6.3.1 Piping networks GUI



Piping network GUI, in Layout’s GUI.

An example of the specific pipeline network GUI in Layout is shown above. It contains the following sub-windows:

- A list of segments of this network in the top left-hand corner, see Segment list (discussed on the following page). The segment on which the text cursor is positioned is active in the rest of this GUI.
- At the top right, a list of components of the active segment, see Component list (discussed on the next page).

- Right centre a list of connections from the entire network, see Connection list (discussed on the following page).
- The black sub-windows contain the three orthogonal views of the network, including connected equipment and compartments. The locations of these cross-sections are dynamically linked to the mouse position, which works the same way as in the compartment and plane definition GUI, from the first option of the Layout main menu.
- On the bottom right, the 3D view of the network. Here one can rotate, pan and zoom in the usual way.

In the graphic sub-windows, the active segment is highlighted.

9.6.3.1.1 Segment list

This sub-window contains an overview of all segments of a network, with their names and their delay factors. The latter are used to generate *fractional* intermediate stages of filling in damage stability calculations. These are more or less the conventional intermediate stages, as discussed in section 21.2.1 on page 403, With conventional intermediate stages of flooding ("Fractional"). At the request of a PIAS user, even multiple such delays can be specified, adding to the variation in intermediate stages. However, those additional delays are optional; for regular use, a single will suffice.

With the segment sub-window selected, a [Properties] function is available in the menu bar. This relates to the entire network of the GUI, and calls the same popup box as discussed on Properties of piping networks (besproken on the following page). The reason for the presence of this redundant function is that in this GUI it may be handy to see or change network parameters that apply as default to all segments.

9.6.3.1.2 Component list

This sub-window shows all the components from the active segment. Obviously, it contains a minimum of two components, which are the references to the connections representing the start and end of the segment. Furthermore, its properties can be specified, such as position, resistance coefficient and cross-sectional area. You can do that by typing those numbers, or using [spacebar] to call a popbox that contains all these parameters. Please recall that for a reliable time-domain calculation not only the cross-sectional area and resistance coefficient of the point components (elbow, WAPCD) must be given, but also of the pipe sections, which should be explicitly specified. For a some types of components, specific properties still apply:

- For a valve, it can be specified whether it is open or closed.
- With a check valve and a WAPDC, the direction of passage is specified. This shows up as an arrow pointing up or down, indicating flow in the direction of the table's row sequence.
- With a pressure relief valve, the opening pressure can be specified. Indeed, two such pressures can be specified specified, one that applies to flow in definition order (which is flow from the component on the top row to the component on the bottom row) and another for flow in the opposite direction. Additionally, it can be specified whether opening of the valve is 'reversible' i.e. whether it closes again after the pressure is lower than the opening pressure. A spring-loaded valve will in general be reversible, the collape of a fire-resistant bulkhead not.

9.6.3.1.3 Connection list

For the entire network, this list contains all connections, which have types as dicussed on The piping data structure (discussed on page 225). In the essence, these are the nodes to which segments can be connected to, and which may themselves be connected to another thing, such as equipment, compartment or opening. Actually, there is not much to say about this sub-window, after all, many things like resistance coefficients and so on have been discussed before. Perhaps useful to mention that a connection to a compartment must be completed with a really existing compartment. If that compartment is removed then this connection too will be dead-ending. And perhaps helpful to reiterate that a WAPCD can be specified both with the components and here, with the connections. The difference between the two is explained at The piping data structure (besproken on page 225).

9.6.3.1.4 Warnings on definition errors

This involves a sub-window which will (hopefully) be mostly blank. However, should there be any incompleteness or inconsistency in the definition of a network then it will be reported here. Space here is scarce, so any defect gets only one line. By clicking on that line with the mouse, a popup box appears with, where necessary, a somewhat more detailed description. Should one need an overall list of warnings on all networks, the option from the piping main menu, described at Check the input (discussed on page 231), can be used.

Some warnings are self-explanatory, while others need some additional explanation, viz:

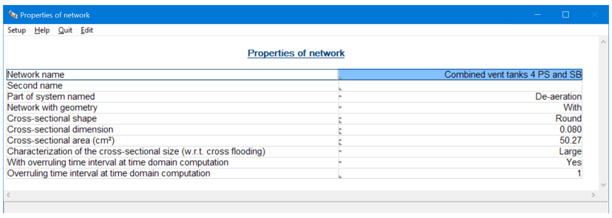
- When a piping system is declared to contain sounding pipes — which can be set in the 'Sounding' column from the piping system overview menu, as discussed at Piping systems (discussed on page 227) — conversion of the pipes defined here to sounding pipes of compartments may occasionally not be possible for a number of reasons, such as:
  - The pipe is not located inside a compartment (in other words, none of the pipe ends is connected to a compartment).
  - A network contains more than one segment (a sounding pipe may only contain one).
  - The number of components in the pipe is larger than the maximum number of sounding pipe points in Layout.
  - The number of sounding pipes in a compartment is larger than the maximum of two, in Layout.

  In such cases the related pipes can still be used related to their network, for example to play a role in damage stability calculations, however they will not be available as sounding pipes in a compartment.
- "Conflict large cross-sectional area vs. resistance coefficient > 0.". The Consecutive Flooding system features a setting of the cross-sectional area above which flow is assumed to take place immediate, which is discussed at section 5.4.9 on page 48, Minimum sectional area for instantaneous fluid passage. This provides control over the direct overflow of large amounts of ingressed sea water over e.g. a threshold at larger heeling angles of a stability calculation. The actual area of a part is then compared to this maximum, and if greater then the fluid finds no hindrance in the flow. On the other hand, one can also specify a resistance coefficient for a component. If it is greater than zero then the fluid does experience resistance = hindrance, which contradicts the effect of the large area of above. In short, the combination (area > minimum area) and (resistance coefficient > 0) may lead to conflicting results. A warning for this is already given in the definition phase, here in Layout, but it will be obvious that it applies only to the time domain calculation, and not to the fractional method.
- TODO check for other warnings to elaborate upon.

Incidentally, via the top menubar Setup, bottom option, one can choose the topics to be subjected to this check. The same is adjustable in the general pipeline settings, as discussed at General piping settings (discussed on the following page).

9.6.3.2 Properties of piping networks

A number of properties are listed in the piping networks overview menu. Via the [properties] menu bar button it opens a popup box with all the parameters of a network:



Properties of a piping network.

The non-trivial parameters are:

- Network with geometry. Geometry is the collective name for the coordinates of the components in the network. It will be obvious that a real existing network cannot exist without geometry, and to calculate the flows in the time domain, the coordinates of all components will have to be given. The fractional method, on the other hand, can in many cases quite well without coordinates. To save you typing in parameters that are not used, you can also specify that no geometry should be used. Obviously, the network cannot then be drawn, and the time domain calculation is also disabled. This also offers the choice 'not specified', then this parameter is not specified per network, but with the general piping settings, see General piping settings (discussed on the next page), all created to save you input work.
- Shape, dimensions and resistance coefficient of the cross section. This is discussed in Frictional resistance from pipes lines (discussed on page 406).
- Characterization of the cross-sectional size with respect to cross flooding. This is a network-level exception on the system wide default as discussed in General piping settings (discussed on the next page).

- For a time domain calculation, the general calculation time step to be applied can be specified in Config (see section 5.4.3 on page 46, Time domain calculation time step). However, for some networks different time step might be desired. That can then be specified here.

9.6.4 Equipment

Equipment does not play an essential role in the piping system. Such a device is simply a designation of an endpoint of a pipe segment, e.g. a generator or a chilled water maker. With the dimensions of the thing included, for the picture and recognition. On a piece of equipment, the pipe stops, so no water flows into or out of it.

9.6.5 Check the input

At Warnings on definition errors (discussed on page 229) it was explained how definition defects are reported per network (while also providing additional explanations about some of the warnings). With this option, these warnings are collected for all networks, in a unified overview.

9.6.6 General piping settings

Here, parameters are defined that are generally used for the pipelines, or for the damage stability calculations that are carried out with them lateron. These are:

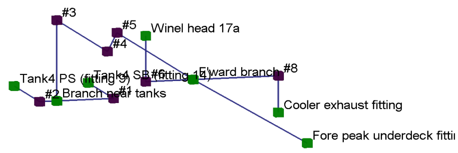
- Which network elements will be subject to input control, here in Layout. Possible choices are:
  1. Only shape and structure of the piping network. This reports e.g. unconnected pipes.
  2. The same, including specific requirements for the fractional damage stability calculation.
  3. As 1, including specific requirements for the time domain damage stability calculation. For example the requirement that somewhere in the network some flow resistance should be present (because otherwise, in theory, the fluid moves at an infinite velocity).
  4. As 1, including requirements for both fractional and time domain calculations.
- Whether networks are with or without geometry. This is a global setting of what can also be controlled at the network level —. the choice is up to the user — and what is explained there at the first option of Properties of piping networks (discussed on the preceding page).
- The method of calculating pipe outlet energy losses. There are two conventions for this, i.e. IMO res. A266 & MSC. 362(92) on the one hand, and MSC. 245(83) on the other. A further explanation of this issue can be found in paragraph 21.2.3.1.2 on page 406, Fluid outlet energy loss. These energy losses have an effect on the time-domain calculation at damage, and one may wonder why this parameter is not located at the damage stability settings. The reason is that this choice determines whether or not explicitly resistance coefficients should be specified for pipe outlets, and this is done here, in Layout. This is why it makes sense to include this general setting in Layout as well.
- Default characterization of the cross-sectional size with respect to cross flooding, a parameter explained in paragraph 21.2.3.2.2 on page 407, Choice of stability criteria with the fractional method. This is the default, for specfic networks deviating values may be given.
- Default resistance coefficients of components, connections and pipes.

The latter two parameters are components in a chain of flow parameters, as discussed in section 21.2.3.3 on page 407, Layered definition of flow-related parameters.

9.6.7 Output of geometry, connectivity and resistance factors of piping

This option prints all networks — regardless whether the system a network belongs has been selected — on paper (or what the user has set instead), see the example below. As is known, among others, resistance coefficients and cross-sectional areas can be specified on multiple levels, and what is shown with this output are the **actual** values as they are also will be used in the damage stability calculations. This makes this output suitable to be added as ‘output of the input’ appendix to the calculations, just as is customary for e.g. the input of ship hull shape and compartments.

Network: Combined vent tanks 4 PS and SB



Part of system: De-aeration

Connections in this network

Name	Type	Connected with	Cf	Area	Length	Breadth	Height
Cooler exhaust fitting	Equipment	Cooler expansion	0.350	400.00	75.000	3.000	5.000
Branch near tanks	Branch	-	0.600	400.00	50.000	-2.000	1.600
Tank4 SB (fitting 14)	Compartment	DB 4 SB	0.350	400.00	45.000	1.000	1.600
Tank4 PS (fitting 9)	Compartment	DB 4 BB	0.350	400.00	45.000	-2.900	1.600
F'ward branch	Branch	-	0.750	400.00	75.000	-1.500	7.000
Winel head 17a	Open opening	-	0.900	400.00	75.000	-4.000	9.500
Fore peak underdeck fitting	Compartment	Voorpiek	0.350	400.00	90.000	0.500	6.000

Segment: Branch SB tank

Component	Name	Cf	Area	Length	Breadth	Height
Connection start	Tank4 SB (fitting 14)	0.350	400.00	45.000	1.000	1.600
Waypoint	#1	0.350	400.00	50.000	1.000	1.600
Connection end	Branch near tanks	0.600	400.00	50.000	-2.000	1.600

Segment: Branch PS tank

Component	Name	Cf	Area	Length	Breadth	Height
Connection start	Tank4 PS (fitting 9)	0.350	400.00	45.000	-2.900	1.600
Waypoint	#2	0.350	400.00	50.000	-2.900	1.600
Connection end	Branch near tanks	0.600	400.00	50.000	-2.000	1.600

Segment: Main line

Component	Name	Cf	Area	Length	Breadth	Height
Connection start	Branch near tanks	0.600	400.00	50.000	-2.000	1.600
Waypoint	#3	0.350	400.00	50.000	-2.000	6.000
Waypoint	#4	0.350	400.00	60.000	-2.000	6.000
Waypoint	#5	0.350	400.00	60.000	-1.500	7.000
Connection end	F'ward branch	0.750	400.00	75.000	-1.500	7.000

Segment: Branch to vent

Component	Name	Cf	Area	Position	Length	Breadth	Height
Connection start	F'ward branch	0.750	400.00	-	75.000	-1.500	7.000
Waypoint	#6	0.350	400.00	-	75.000	-4.000	7.000
Vent check valve	Vent non-return ball	0.350	400.00	↓	-	-	-

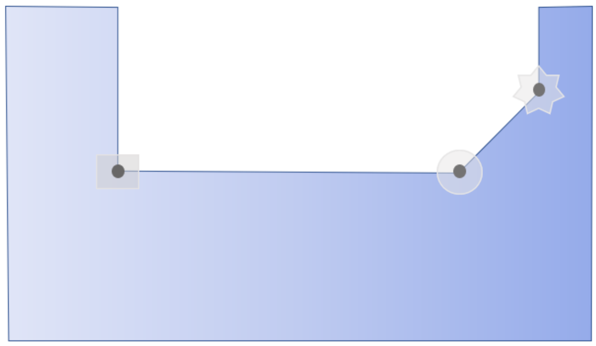
Example of the output of a network input.

9.6.8 Modelling specific things from the real world

How the system as described can be applied to systems of pipelines will be broadly clear, at this stage in the manual. In practice, however, one may encounter ship parts that cannot be immediately recognised as ‘pipeline’, even though they can be modelled as such. For example:

A hole in a bulkhead can be specified as a very short pipe that is connected on both sides to the two compartments adjacend to the bulkhead. In this respect the pipe’s *position*, i.e.nbsp;its longitudina, transverse and vertical coordinates, is determining whether or not the fluid will flow through the hole. The dimensions of the pipe do not play a role in this.

- A ship may be equipped with a fire-resistant bulkhead or door (typically an A60 class), which can withstand a limited level of water pressure. Such a bulkhead can be modelled with just a short pipe, as described above, supplemented by a pressure relief valve in the middle of the pipe. From that valve its opening pressure can be specified, as described at Component list (discussed on page 229), which for this application is equal to the collapse pressure of the bulkhead. Also, one turns off ‘reversible’ because the bulkhead will not miraculously recover if the pressure drops lateron.
- A demi bulkhead, or a threshold, can be modelled with multiple short pipes at points that are eligible to be submerged as first. See the example below, where three such ‘pipes’ are modelled. The dimensions should then be specified quite large, so that there will be no significant resistance when the fluid overflows the threshold. The exact shape and dimensions do not matter much as these are geometrically are not taken into account anyway.



Modelling a partial bulkhead by three short pipes.

9.7 Other lists, and program settings

Lists and program configurations

9.7.1 List of openings and other special points

From each compartment its openings and other special points can be given, as discussed in paragraph 9.5.1.2.9 on page 217, Special points / openings (to which we also refer for the purpose of the input cells). However, it may be convenient to collect all special points in a single list, which is performed by this option, on which the following remarks can be made:

- The order of points in this list is primarily that of the compartments. The order can be changed, for example by inserting points, or by [Sort], however, such a new order will only be temporarily. In the end the compartment order still prevails.
- [+/-], the sign of the transverse coordinate will be toggled (so the point switches from PS to SB or vv., does unfortunately not work for an entire selected column).
- With the [Sort] option points can be sorted in the order of the content of the column where the text cursor resides. Also this order is temporarily.
- Compartment openings, which are defined here, are also visible in Hulldef, please refer to section 7.2.8 on page 184, Openings for a discussion. That Hulldef list may also contain other points, such as those of the margin line, which are not relevant here. However, it may also conntain openings which are *not* connected to a compartment, and although such points play also no role here in Layout, as a service they are still included in the list of this menu option. Their compartment indication is '–Not from a specific compartment'. Because their relevance is less than that of 'real' compartment openings, their place in the list is at the bottom (while in Hulldef they are listed at the top, that is not a matter of sloppiness, the reason is that the focus is just different). This type of openings will only be stored in Hulldef format if the setting "Always create conventional PIAS compartment files at save" (please see section 9.7.5 on page 235, Layout project settings and function colors is switched on.

9.7.2 List of physical planes

Here a list appears of physical planes, with nine columns, namely:

- Name and second name: the two names of the plane. Similar as with compartments, it can be set that the second name should be generated automatically, with the setting discussed in section 9.7.5 on the following page, Layout project settings and function colors.
- Abbreviation, of eight characters at maximum.
- Abs.position: the position of the plane, in metres from App, CL or baseline. For an angled plane also a position is presented, which is — similar to orthogonal planes — the perpendicular distance between this plane and the origin, where the sign (+ or -) is not relevant. This position — not just as fixed by this column, but also by the next column — can be constrained by the presence of other planes, as explained in section 9.1.6.3 on page 201, Limited positioning of a physical plane.
- Rel.position: the position of the plane in relation to a reference plane, at least, when the plane has been relatively defined.
- Orientation: longitudinal plane, transverse plane, horizontal plane or angled plane.
- The plate thickness of the bulkhead or deck (in meters!).
- The specific weight, which is the weight/m^2. If the plane thickness is filled in, and the weight/m^2 is still zero, then the latter is adjusted according to this thickness, for shipbuilding steel. However, this is a gadget, because in reality stiffeners and girders will probably increase the specific weight (note: when exporting to Poseidon (see section 9.11.6 on page 243, Export to Poseidon (DNV•GL)) structural elements can be defined, and that information could be used to determine the specific weight here including that structure. If you are interested in such a feature you can contact SARC).
- Position is not modifiable by Constraint Management: whether this plane will maintain its location if Constraint Management is going to modify the plane positions.
- Number of constraints: here the amount of constraint that apply to this physical plane can be inputted. Then per constraint the correct constaint can be selected from the constraint table via a popup-menu.

In the menu bar of this list is, apart from the usual menu options, a number of specific options:

- When adding a new plane, one directly enters the popup menu of section 9.1.6.1 on page 199, Popup menu for geometry of points or planes.
- [Geometry], with which one can change the shape of the plane. Directions are described at section 9.4.2.6 on page 207, The shape of a plane (the green dots).
- [Sort], with which one can sort the planes by columns, i.e. in the order of the data of the column where the text cursor is. This sorting can also be undone again with [Undo].
- [Area tAbles], which will produce a table containing the area and Centre of Gravity for each plane. If also specific weights are given for all planes, then the totalized weight and CoG of the internal ship's planes are included as well. The weight of the shell plates is not included here, but can be calculated using the shell plate expansion function of Fairway, see section 6.10 on page 150, Shell plate expansions and templates.

In all but the sixth column one can enter values directly. With <Enter> one enters a popup menu with all data of the plane, and when one at the absolute or relative position presses on <Enter>, one enters the plane definition menu that is described below.

9.7.3 List of reference planes

This list contains all reference planes, and is concerning operation and content similar to the list of phisical planes, just above. However, there is an additional rightmost column which lists how often a reference plane is actually used. When entering such a cell a popup window appears with this information split out in compartments, reference planes and physical planes. On this info two remarks can be made:

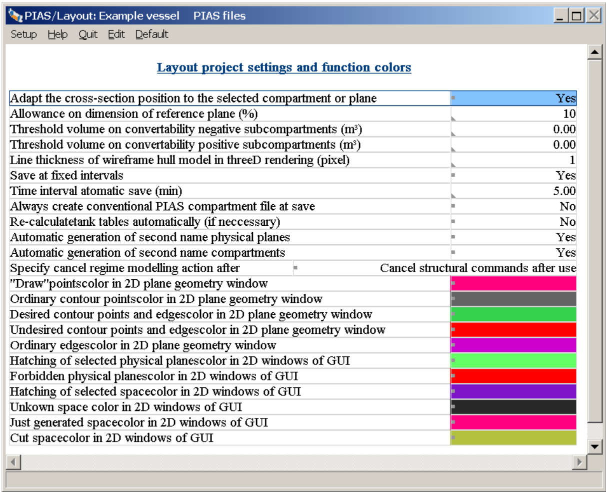
- The summation counts all parts which can be related to a reference plane, so not only the measurements of compartments and planes, but also those of sounding pipes and openings etc.
- Although the subcompartment shape might externally be of different types (such as 'simple block' and 'four longitudinal ribs', seee paragraph 9.5.1.1.1 on page 214, Types of subcompartments "with coordinates"), internally it is always stored by means of its corners. This implies that certain shape definition measurements may be counted more than once. for instance if an aft compartment boundary has four corners, and that aft boundary is defined by means of a reference plane, it will be inlcuded four times in the reference plane count. Nothing to worry about, but it is good te be informed on this phenomenon.

So, in this menu reference planes can be defined. Letting points and other planes refer to a reference plane is discussed in section 9.1.6.1 on page 199, Popup menu for geometry of points or planes. Apropos, a reference plane can be defined relative to another reference plane, etc.

9.7.4 Compartment tree

With this option there appears a popup box with the same compartment *tree view* as used in the GUI. Essentially, this tree contains no more information than the regular compartment list of section 9.5 on page 212, Compartment list, calculation and printing of tank tables, except for that it is shown here per compartments and subcompartiments in one overview.

9.7.5 Layout project settings and function colors



Project settings.

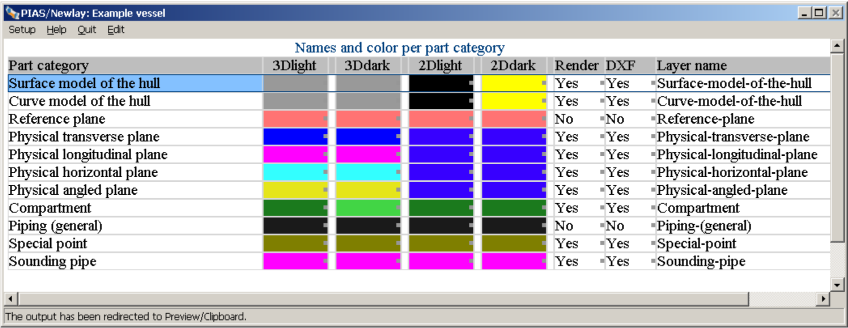
With this option appears the above popup screen, in which a number of program features and colors can be set:

- The cross-section position of the three orthogonal cross-sections is controlled anyway by clicking in such a cross-section on the right mouse button, then the position of the other cross-section is equalled to the position of the cursor at that moment. But apart from this mechanism, one may also opt for adjusting that cross-section position to a plane or compartment that has been chosen in the compartment tree or planes list. If you want this, then you have the put the first option on yes.
- Allowance: It is useful when the reference planes are drawn slightly larger than the rest of the ship. That extra size, the allowance, can be defined here.
- Threshold volume: at a conversion of a compartment configuration to a plane configuration, negative subcompartments can be included integrally. This may result, however, in a substantial increase in the number of used planes, which is not necessarily desired. That's why one can define a limit value (of volume, in cubic meters) here **under which** negative subcompartments are not included in the conversion to planes, they still exist of course, but simply as subcompartment of the type 'with coordinates'.
- Line thickness will be obvious.
- At the time interval one can define per how many minutes the model is saved automatically. That can be useful in the event of failures, then you still have at least a recent model.
- Automatic generation of the second name of physical planes or compartments. Compartments and planes have, obviously, a name. For additional identification also a second name may be given. For enhanced convenience, with these two options in wuestion it can be set that these second names are generated automatically, based on the location of the object, according to these systems:
  - Physical planes get a plane type, and their position. If frame spacings have been set (as discussed in section 7.2.1.3 on page 167, Frame spacings) then longitudinal positions are not given in meters, but in frame numbers (with possible an offset in mm) instead.
  - Compartments get a combination of aft boundary, forward boundary, side (PS, SB or 'over CL') and the weight group as assigned to this compartment.

  This mechanism permanently updates these second names, so existing manually entered names will be overwritten.

- With option [re-calculate tank tables automatically] the tank tables are (re-)calculated whenever neccessary, that is when the compartment geometry is jounger than the available table. This automatism may take some time, however, for the benefit of table consistency.
- For the cancel regime see section 9.4.2.3 on page 206, How long stays a function assigned to a mouse button?.
- The colors will be obvious. The color of 'rdesired contour points' is for that of the 'green dots'. The 'just generated space color' and the 'cut space color' are only used at a compartment color schedule 'uniform' (see for that section 9.4.3.2 on page 208, View). The 'draw'points color is as used with the draw function in the GUI, see paragraph 9.4.3.3.2 on page 209, Draw.

9.7.6 Names and color per part category



Part category	3Dlight	3Ddark	2Dlight	2Ddark	Render	DXF	Layer name
Surface model of the hull					Yes	Yes	Surface-model-of-the-hull
Curve model of the hull					Yes	Yes	Curve-model-of-the-hull
Reference plane					No	No	Reference-plane
Physical transverse plane					Yes	Yes	Physical-transverse-plane
Physical longitudinal plane					Yes	Yes	Physical-longitudinal-plane
Physical horizontal plane					Yes	Yes	Physical-horizontal-plane
Physical angled plane					Yes	Yes	Physical-angled-plane
Compartment					Yes	Yes	Compartment
Piping (general)					No	No	Piping-(general)
Special point					Yes	Yes	Special-point
Sounding pipe					Yes	Yes	Sounding-pipe

Names, colors and other part properties.

Herewith one can choose the colors of the various ship's items. In section 9.7.5 on the previous page, Layout project settings and function colors one could also configurate colors, but that was meant for program items, here we are talking about ship's items, of which seven features can be defined:

- The first column contains the identification name of a category. This is mentioned here for the purpose of recognition, and cannot be changed anymore.
- In the next two columns one can set the color as it is used in the 3D views, in the second column for a light background, and in the third column for a dark one.
- Subsequently two columns in which one can set the color as it is used in the 2D cross-sections (the orthogonal cross-sections), also for light and dark backgrounds.
- In the sixth column one can define whether this category must be included in the output of the 3D render model, the production of which is described in section 9.8 on the following page, Threedimensional presentation.
- In the seventh column one defines whether this category must be included in the DXF output of the general arrangement plan, the production of which is described in section 9.9.6 on page 241, 3D-plan to DXF file.
- In the eighth column one can define the layer name that is allocated to this category in such a DXF output. When defining the layer name one has to observe the conventions of Autocad, or another recipient CA↩ D system. Thus a layer name can have, for example, a specific maximum length, or some signs may be forbidden. For the exact nature of these restrictions, you will have to consult the documentation of the recipient system.

The color of a 'piping system' is just the default, which is assigned to a new system. If desired, for each individual system an andividual color can be assigned, see section 9.6 on page 224, Pipe lines and piping systems. The display of container positions in the layout was temporarily abolished in 2021, as PIAS was provided with a completely new container module at that time. Upon completion of the container part of Cargoquip, this will be reactivated.

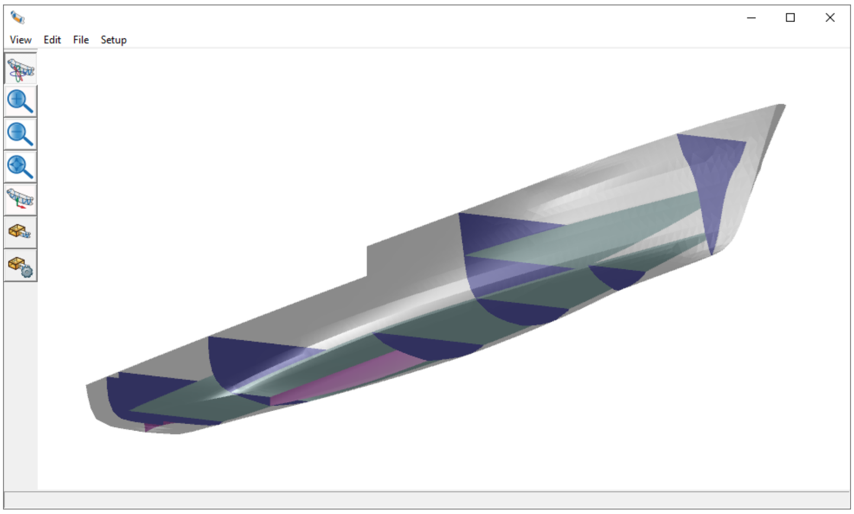
9.7.7 Define weight groups

With this option one enters a menu where features of weight groups can be defined. A weight group is a specific category of weight, of, for example, a ship or cargo. At the definition of compartments can be defined at the compartment what the weight group is of the volume for which this compartment is intended, for example *potable* *water* or *palm oil*. Color combination or output of loading conditions can take place per weight group. This menu is discussed in more detail in section 20.1 on page 395, Weight groups.

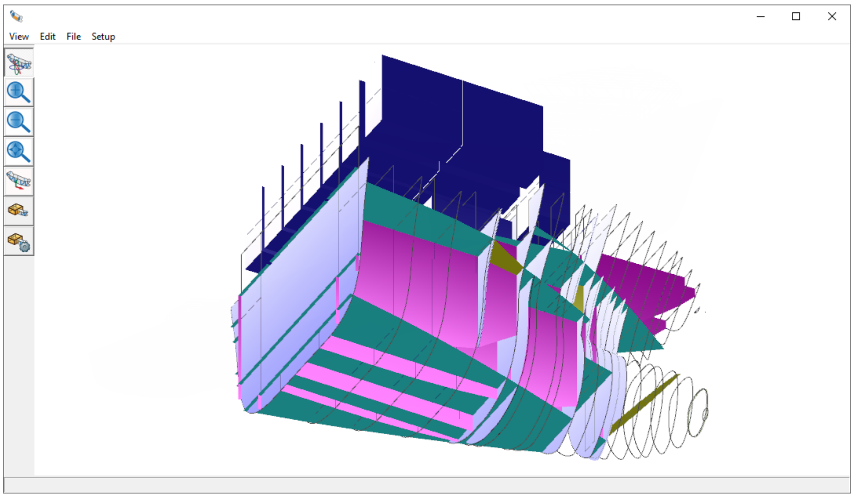
9.7.8 Notes and remarks

With this option an input screen appears with which one can make separate notes. These notes are saved at the subdivision plan, and can always be altered later on.

9.8 Threedimensional presentation



Rendered view, with surface model of hull.

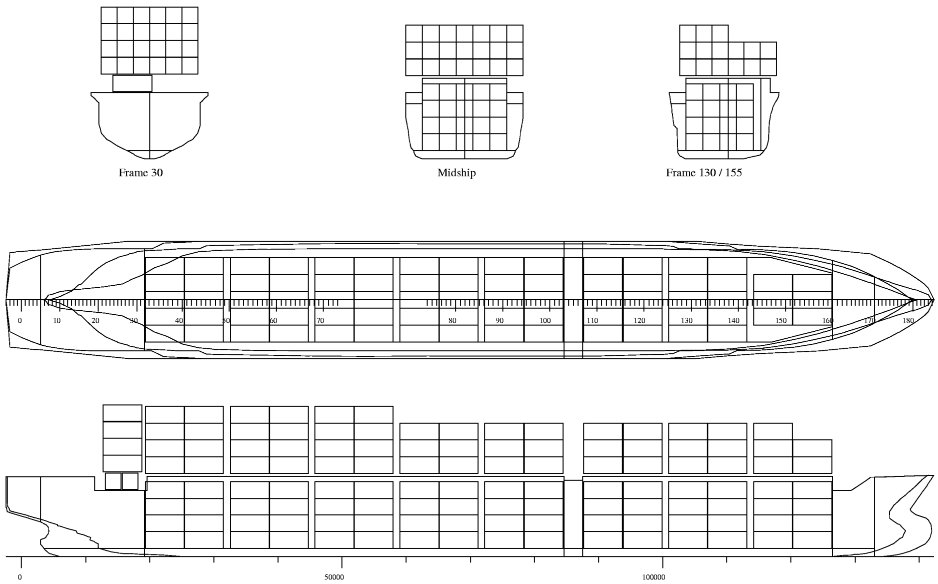


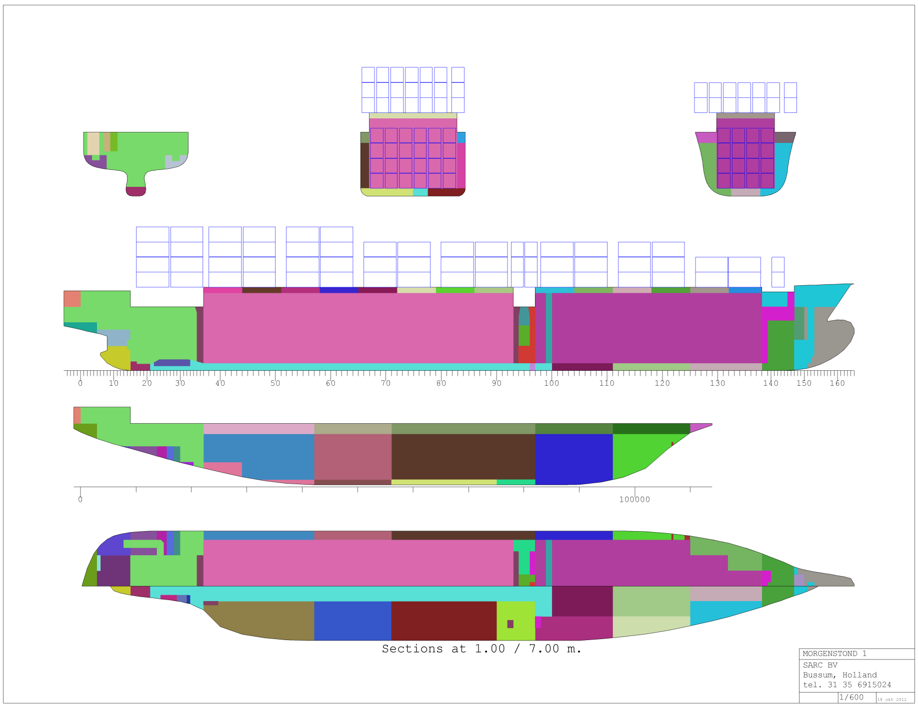
Rendered view, with sectional model of hull, including container slots.

With this option one can produce a three-dimensional rendered view, like in the above figure. In general, this is identical to the 3D view in the GUI, and naturally these functions are applcable, which are discussed in section 9.4.2.4 on page 206, Operation in the 3D subwindows. It differs insofar as that in the GUI the functions must be called upon through a popup box, which is activated with the right mouse button, while the functions are in the upper bar here, so they are more accessible. When drawing such a rendered view, one has to be aware of two issues:

- When the drawing of hull frame lines is switched off, such objects can sometimes actually be drawn. But those are the compartment frame lines, to be recognized by the color of the compartment.
- Drawing hull lines and/or planes is only possible when such a model is actually present **and** when it has been activated in module Config, see section 5.1 on page 41, Calculation methods and output preferences.

9.9 Subdivision plan





The layout of a two-dimensional subdivision plan can be recorded at this option, two examples of which are displayed above. Such a subdivision plan contains the geometric information of the internal geometry, and can be used as schematic general arrangement plan, tank plan or safety plan. The most obvious application is to have this subdivision plan serve as ‘underlayer’ for such other drawing, and to draw all additions (like deckhouses, masts, lights, valves) in other layers. When the subdivision changes later on, then only that underlayer needs to be replaced, and all other layers can be re-used. By the way, the *modus operandi* on this subdivision plan is somewhat similar to that of the lines plan in Fairway, please refer to section 6.9 on page 145, Define and generate lines plan for those details. This subdivision plan section is subdivided into six sub options:

Subdivision plan

9.9.1 Configuration subdivision plan and DXF export

- Color of the compartments, where can be defined whether any compartment gets an individual color, which is allocated per weight group, or that all compartments are drawn with the same color.
- Desired container slot size, where could be defined which of the predefined container slots should have been drawn in the subdivision plan. The slot size is defined in integer feet, for example 20 or 40. In 2021, PIAS has been refurbished with a completely new container module, and then the display of container positions in Layout was abolished. Upon strong user demand, this could be reactivated.
- Subdivision plan with color. Here is defined whether the subdivision plan is drawn with color or not.
- Draw all, or only the selected compartments. If compartments are specified to be drawn in a section or a view, at this row it can be defined whether *all* compartments should be drawn, or only the selected ones. This setting only applies to the plot of the plan, not to the 3D-DXF file.
- Coloring compartments in subdivision plan. Here is indicated whether the compartments that are drawn in the subdivision plan are also colored. The alternative is that only its circumference is drawn.
- Additional margin framework (mm). When drawing the subdivision plan on paper, the available paper space is used as much as possible. At this option one can, however, define an additional free space along the paper edge, in millimeters.
- Unit of measure of the 3D DXF file. Here one can define whether the unit of measure of the 3D DXF file is meter or millimeter.
- 3D DXF file name, here one can type either the desired file name, or (with <Enter>) call upon the Windows-*filebrowser* to indicate the file name.
- Texts drawing head. A subdivision plan that is printed on paper can contain a drawing head in the right-under corner. Here one can define how many rows this must contain, and which texts must be included.

9.9.2 Names and color per part category

This is exactly the same menu as discussed in section 9.7.6 on page 236, Names and color per part category, which for the sake of convenience also has been incorporated in this menu, in order to be able to quickly adjust a configuration.

9.9.3 Subdivision plan layout

The layout of a subdivision plan can be specified here. It is possible to define multiple layouts (maximally four), so that, for example, a subdivision plan with several pages can be defined. When one has chosen this submenu then there firstly appears a window with those various layouts. There is little to say about this, any layout has a name, one can define which one is selected to be drawn later on, and, finally, with the Copy/Paste mechanism one layout can be copied to the other. With <Enter> one enters a window where the details ot that layout in question can be defined. One can define in that screen which views are included at which position in the drawing of the subdivision plan. The desired position of a cross-section is defined in meters shift in horizontal and vertical direction. One must imagine here that a 2D scale 1:1 drawing is made, where certain views are shifted horizontally or vertically over a certain distance (in real-size measures). The scale on which is finally drawn depends on the paper sizes, which are not yet known here, and does therefore play no role yet. Per view one can define the following:

- The shift in breadth, in m, of this cross-section on the 2D scale 1:1 subdivision plan. This shift has no absolute meaning, but records the relative position of the view relative to other views. It is common practice that one view has no shift.
- Analogous to the previous row: the shift in height, in m, of this cross-section.
- The view, which may be a top view, side view or front view as desired.

- The number of cross-sections that is drawn in this view. When one wants to change this number, one enters a deeper menu.
- Whether an X-axis must be drawn with grade mark. The alternatives are: no X-axis, plain X-axis, X-axis with grade mark without legend, X-axis with grade mark and legend in millimeters, X-axis with grade mark and legend in meters, X-axis with grade mark and legend in frame number. This last option is only available at the top and side views.
- Analogous to the previous row: whether a Y-axis must be drawn with grade mark.
- The description, that is the name of this view, which is printed below the view.

As mentioned above, after choosing the fourth menu option, one enters a deeper menu, where details of the wished cross-sections of the view in question can be defined. Per view one can define here:

- The position. At front views in m from App, at top views in m from base and at side views in m from CL, positive for a cross-section at SB side and negative for a cross-section at PS.
- The side. At top views and front views one can define here whether the cross-section is located at one side (PS or SB), or extends over both sides.
- Section/Contour. Here one defines whether it is a real (inter)section, thus a cutting, or a total view (a contour). The following details are applicable here:
  - A contour can only be defined for the hull, sounding pipes, piping systems and special points. It makes no sense for other objects.
  - At a contour of the hull many points of it are projected in the view plane, and its envelope is determined. Substantial steps in the contour view can be possibly cut as a result. When this occurs, so be it.
  - Pipe lines, sounding pipes and special points can perhaps best be drawn in 'contour', because they are all shown then. Mostly there is little to see at 'cross-section', because the probability that such a pipe or point is exactly in that section is small (although the program uses a certain tolerance around the section).
- Hull. In this column one defines whether the hull must be drawn in the view in question.
- Transverse. Indicates whether transverse bulkheads must be drawn.
- Longitudinal. Indicates whether longitudinal bulkheads must be drawn.
- Deck. Indicates whether horizontal bulkheads (decks) must be drawn.
- Angled. Indicates whether angled bulkheads must be drawn.
- Compart. Indicates whether compartments must be drawn.
- Pipe line, indicates whether piping systems (zie section 9.6 on page 224, Pipe lines and piping systems) must be drawn. Because the subdivision plan would be much to crowded if all piping systems would be drawn in, in the "regular" plan only the intersections between pipe and desired section are drawn — with a 'section' anyway. Subsequently, for all piping systems which are selected with 'separate subdivision plan per piping system' a separate plot will be produced, containing the entire subdivision plan plus that single piping system. This only aplies to output on paper or file, not to output on screen. In order to emphasize tge pipe lines in those plots a bit, the compartments are drawn somewhat more pale.
- Sounding pipes. Indicates whether sounding pipes as defined within the compartments (see paragraph 9.5.1.2.8 on page 217, Sounding pipe) must be drawn.
- Sp.point. Indicates whether the special points of the compartiments (such as openings, see paragraph 9.5.1.2.9 on page 217, Special points / openings) must be drawn.
- Contain. Indicates whether the container slots must be drawn. Note: The container slots must be defined in order to be drawn. That can be done with PIAS module Cargoquip.

Attention

All physical planes (bulkheads and decks) are being drawn as far as they constitue a separation between compartments, because this gives the most realistic impression. So, this is according to the *separating planes on* switch which can be defined in the GUI, please refer to section 9.4.3.2 on page 208, View for the discussion. However, this GUI switch has no effect on the plot of the subdivision plan here.

9.9.4 Subdivision plan preview

With this option the subdivision plan is drawn on the screen. It is intended that this *preview* option is integrated in due time in the definition options of the subdivision plan, so that definition changes are directly and interactively made visible.

9.9.5 Subdivision plan to paper or file

The subdivision plan, that was drawn on the screen at the previous menu option, can be printed by the printer or plotter with this option. When one wants to make a file that contains the subdivision plan, then one can use the mechanism where output of PIAS is rerouted to a file. This can be defined at option ‘Output to’ (see section 5.1 on page 41, Calculation methods and output preferences), which offers the choice out of three formats:

- Rich Text Format, RTF, to generate a bitmap that, for example, can be read in MS Word.
- Encapsulated PostScript, EPS, to generate a file with vector data. Vectors can be displayed much more sharply than a bitmap.
- Drawing eXchange Format, DXF, to import the general arrangement plan in a CAD or drawing system. With this format one can choose whether the unit of measure is meters or millimeters. The thus generated DXF file consist of line types of the DXF type *polyline*.

9.9.6 3D-plan to DXF file

Also with this option a DXF file is made, but one that contains the complete 3D model. This file has the following features:

- Lines are of the type *DXF polyline*.
- Bulkheads and decks are displayed by means of a closed, albeit empty contour.
- Possible angled planes, for example of the hull, are first divided in triangles, and subsequently saved as wireframe model.
- Any category, such as hull, decks and 20’ containers, comes in its own layer, of which one can define the name at [Names and colors per item category], see section 9.9.2 on page 239, Names and color per part category.
- Only those elements are included of which at [Names and colors per item category] the column DXF has been placed on ‘yes’.

9.10 Print compartment input data

Print compartment input data

1.	Print input data of selected compartments
2.	Three-dimensional views of selected compartments
3.	Difference between internal and external geometry
4.	Define views/sections of compartment plan
5.	Draw compartment plan

9.10.1 Print input data of selected compartments

The input data of all selected compartments will be printed with this option. The aim of this table of inpit is to inform an outsider on the shape and core properties of the defined compartments. For this reason only raw measurements are being printed, and not auxiliary definition aids such as distances to reference planes, neither do bulkhead or deck parameters belong to the output.

9.10.2 Three-dimensional views of selected compartments

With this option all selected compartments are printed subsequently, where for each compartment the same viewing angle is applied as used in section 9.5.1 on page 213, Compartment definition window.

9.10.3 Difference between internal and external geometry

With this option a graph is drawn, where on many cross sections the difference between sectional area (derived from hullform) and the sum of the compartment areas (derived from compartment definition) is displayed. This difference, which should be zero theoretically, may indicate the presence of lacking or double compartment definition. For probabilistic damage stability calculations this check is recommended. Besides, small differences may occur due to properties of the numerical methods used. The configuration ‘difference internal/external geometry with external hullforms', as discussed in section 5.3 on page 45, Settings for compartments and tank sounding tables, is applicable to this area comparison.

9.10.4 Define views/sections of compartment plan

With this option views and sections of the compartment plan can be specified, please refer to section 20.2 on page 396, Sketches of tanks, compartments and damage cases for further discussion and an example.

9.10.5 Draw compartment plan

With this option the compartment plan, as specified with the previous option, is drawn. A compartment plan contains a number of views or sections which are printed one by one. This gives a fairly well insight in the compartment definition, and is fit to be included in stability booklets for that purpose. However, for a more versatile output method reference is made the the subdivision plan, as discussed in section 9.9 on page 238, Subdivision plan. Such a plan can also contain other items, such as bulkheads, decks and openings.

9.11 Conversion, and import and export of subdivision data

Conversion and import and export

1. Generate physical planes from the totality of convertible subcompartments
2. Apply advices on converting to physical planes
3. Import PIAS compartments from pre-2012 format
4. Clean pre-2012 PIAS compartments
6. Export decks and bulkheads to Rapid Prototyping format (STL)
7. Export to Poseidon (DNV•GL)
8. Write XML file
9. Read XML file

9.11.1 Generate physical planes from the totality of convertible subcompartments

This option does four things. First, an overlapping check is carried out, which is entirely identical to the second option in this menu (see section 9.11.2 on the following page, Apply advices on converting to physical planes), where subcompartments of the type 'generated between planes' can be converted to the type 'with coordinates'. At this test is also checked whether subcompartments overlap, and if so, the convertibility of one of the two, namely the smallest one, is switched off, so it is omitted at the conversion. Subsequently, all physical planes are removed, after which a new collection of physical planes is generated on the basis of the subcompartments of the type 'with coordinates' that mutually constitue exactly those subcompartments. Finally, all those subcompartments are converted to the type 'generated between planes'. It is not compulsory to use all available compartments at this action; through the setting 'convertible', which can be defined in the compartment list (see section 9.5 on page 212, Compartment list, calculation and printing of tank tables), at the compartment definition (see paragraph 9.5.1.2.17 on page 218, Convertible) and the subcompartment definition (see paragraph 9.5.1.3.5 on page 219, Convertible) one can exactly indicate which (sub)compartments are included in this conversion and which ones are not included.

Attention

It is advised to use this conversion option tactfully. It may be tempting to convert a complete ship, with all its compartments and tanks, to the combination of physical planes and the subcompartment type 'space generated between planes', which is possible and allowed, but one may end up with a large amount of physical planes, without any overview, as a result of which it is useless in the end. It is probably wiser to confine oneself at the conversion to compartments that belong to the main division of the ship, and is also stimulated to do so because the maximum number of convertible subcompartments is 150 (which is not a principal limit, but a practical one). Please also consult what has been written at section 9.1.2 on page 197, Use of different types of subcompartments.

9.11.2 Apply advices on converting to physical planes

This option is meant to support the generation of physical planes from subcompartments of the type ‘with coordinates’ (see section 9.11.1 on the previous page, Generate physical planes from the totality of convertible subcompartments), since that is only possible with non-overlapping subcompartments. First of all is considered here whether there already are subcompartments of the type ‘space generated between planes’, and if so, then the user is asked whether these have to be converted to the type ‘with coordinates’, because the generation of physical planes is only possible on the basis of this type of subcompartments. Subsequently is tested which subcompartments overlap. A report is given of this (when this is very long, it can be cut and pasted to word processor for further printing or further study). At all compartments of which has been defined that their convertibility is ‘automatic at conversion’, is filled in at the smallest subcompartment of the two overlapping ones that this may not be included in the conversion of option 8.1 above.

9.11.3 Import PIAS compartments from pre-2012 format

With this option compartments are imported in the format of the precursor of Layout, Compart (which has been in active use until 2012 for modeling compartments), and converted to Layout compartments of the type ‘with coordinates’. The user is supposed to choose the old file, its extension is .cmp.

9.11.4 Clean pre-2012 PIAS compartments

In the old PIAS compartment module Compart subcompartments could only be limited by eight vertices. Apart from that, there was the requirement that they should be *convex*, and in the test on convexity vertices were not allowed to converse precisely. That resulted in defining vertices for, for example, three-sides or tapered compartments with differences of millimeters. Here, in Layout, this is no longer necessary, and being more serious, those differences of millimeters are counterproductive, because when Layout is constructing physical planes, then there is actually made a plane of one millimeter, and that does not make for overview. The present option detects those differences and almost converging vertices are actually contracted. One must realise, however, that this option does not repair all anomalies, so that manual adjustments might be necessary, for example in the event of:

- Larger differences than two millimeters. The algorithm would be capable of that of course, but it remains to be seen to what extent a program must change the data imported by the user autonomously.
- A boundary of a subcompartment that is warped (so, not fully flat). Within a subcompartment of the type ‘with coordinates’ there is no objection against this property, however, this cannot be used to generate physical planes (for they have to be entirely flat).

9.11.5 Export decks and bulkheads to Rapid Prototyping format (STL)

With this option bulkheads and decks are converted to `STL format`[6], which is suitable for Rapid prototyping (or 3D printing). This option is still experimental, and only available to SARC. On `www.sarc.nl/images/publications/appendix_swz2012.pdf` is an example of how to use such STL file in order to print a ship’s subdivision three-dimensionally.

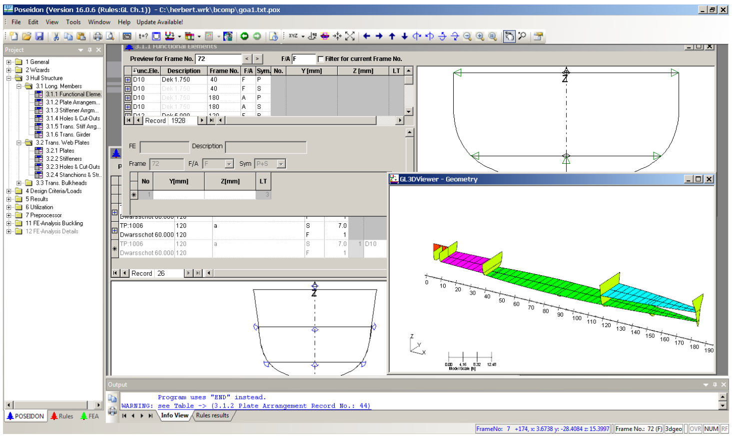
9.11.6 Export to Poseidon (DNV•GL)

With this option data from PIAS are converted to and written to a file in the format of `Poseidon`[7], the rules program of DNV•GL. The purpose of this conversion is to transfer the data as available in PIAS to some degree of completeness. The created Poseidon file contains the following data:

- Main dimensions and other general numerical data. For this purpose, please do not forget to define the ‘auxiliary particulars’ as well, see section 7.2.1.7 on page 169, Particulars of export to Poseidon. The minimum drafts aft and fore, which must be communicated to Poseidon, are derived from the draft marks. So, for a Poeidon file as complete as possible, at least two draft marks with a so-called ‘Tmax local’ must be given, see section 7.2.1.4 on page 168, Draft marks and allowable maximum and minimum drafts for further guidance. Poseidon parameters for which no data are available in PIAS (such as GL register number and ice class particulars) are left void.
- Table of frame spacings.

[6] `https://en.wikipedia.org/wiki/STL_(file_format)`
[7] `https://www.dnvgl.com/services/strength-assessment-of-hull-structures-poseidon-18518`

- The hull form, represented by all frames of the PIAS model.
- The shape of longitudinal bulkheads and decks.
- The shape of transverse bulkheads, modelled as ‘transverse web plate’ for Poseidon.
- Compartments.
- Global loads: the (envelop of) longitudinal shear forces and bending moments.
- Local loads: deck loads, compartment fillings and wheel loads.
- Girders and webs.
- Plating of shell, bulkheads and decks, with a default plate thickness.
- Longitudinal and transverse stiffeners on shell, bulkheads and decks.



Ship with bulkheads and decks exported from PIAS, imported and vizualized in Poseidon.

9.11.6.1 Enter additional Poseidon data in Layout

Many details of geometry of hull and layout are already available in Layout, so these can be converted to Poseidon without additional effort. However, Poseidon also requires data that is not at all available in PIAS, such as local loads or structural data. Layout offers some alphanumeric input menus to enter those data. For that reason with this “Poseidon” export feature, an intermediate menu will open, from which the first option invokes the actual export to Poseidon. The other options open forms in which specific Poseidon parameters, which are not available in PIAS, can be specified. Some of these parameters are all of Poseidon, so for their meaning and application reference is made to the Poseidon manual. On data of a more generic nature the following elucidation is applicable:

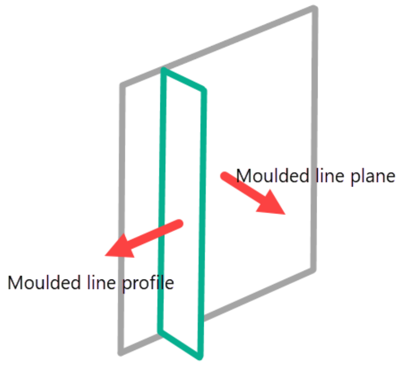
- For the purpose of Poseidons “Cargo and Decks” in one of the tables deck loads can be given. In the first instance you must specify a deck, for which one of the physical planes of Layout should be identified means of its abbreviation (one can actually give any kind of plane, including a transverse or longitudinal bulkhead, but that would be quite useless for deck cargo). The heart of the matter is that the deck cargo will be placed on that particular deck. Additionally, a compartment / subcompartment combination is given, which is used exclusively for the position and length of the cargo (the verification whether the deck is indeed adjacent to the subcompartment is up to the program user). An exception to this mechanism is a load on the upper deck. To this end one gives the Poseidon term “Tdeck” as ‘abbreviation’, as well as a compartment / subcompartment combination which is adjacent to the top of the hull. The fact that this subcompartment is located under deck instead of above does not matter, it is just used to indentify the location.
- In addition to deck loads, for “Cargo and Decks” also tank fillings can be specified . These are automatically applied to the physical plane which is located at the bottom of a compartment. A combination of tank filling and deck cargo on the same deck is obviously not feasible. Would that be specified anyway, the the deck cargo prevails above the tank filling.
- Webs (transverse girders) can be given per group, with for each group an aft boundary, forward boundary and spacing between two webs. For each web group a start and an end must be given. The orientation (from inside out, or from bottom up) is not constrained. Start and end are situated on the shell, or on a Layout plane, which can be either a reference plane or a physical plane. These planes are specified by the user by its abbreviation, or, alternatively, the Poseidon concepts “Tdeck” or “Shell” for deck or bottom/shell respectively. For start and end points are further given:

1. The Layout plane on which (in which) it lies, or, alternatively, shell/bottom/deck, is the web is attached to that. This will be called plane 1.
2. The position of this point in plane 1. Basically, a transverse or vertical coordinate, however, in order to maintain the coherence with the other components of teh ship layout, reference is made to another plane instead (or, alternatively shell/bottom/deck) — which will be baptized plane 2 — so that start/end point will be situated at the intersection of plane 1 and plane 2.
3. The inner limit of the web, which will fix the girder height at that location. For this point a third plane (plane 3) must be given, which defines the inner limit at the intersection of planes 2 and 3. Additionally, an optional offset from plane 3 can be given. Alternatively, for the web a constant girder height is specified, in which case plane 3 is omitted. If start and end points share the samen 'plane 1', and if that is a physical plane in Layout, then with a constant girder height the inside will run parallel with the outside. In this way a web along the shell can be defined.

With these parameters quite some variations in structural layout can be defined, as illustrated in section 9.11.6.2 on the following page, Examples of defining girders and webs. With a web two Poseidon-related details are relevant:

- It is not crystal clear how Poseidon determines at which side the girder height should be taken. In one instance it is at the left of 'plane 1', in another at the right. So it will require some experimentation to discover the rule behind.
- From a web it can be specified whether it is situated at PS, at SB or at both sides. In Poseidon this is interpreted loosely, which means that a web shape is simply copied to the other side, even if the plane to which the web refers does not exist at that side.

- Girders (longitudinal) are defined similar to webs, just the wording can differ now and then. A group of girders is given in transverse or in vertical direction, so angled directions are excluded. Transverse coordinates whould be given only **positively**; the switch PS/SB/Double determines the applicable side (this is, after all, the way Poseidon works).
- Specifying webs and girders in PIAS is aimed at a quick — and integrated in the Layout structure of bulkheads and decks — definition of the primary structural elements and the most commonly used web/girder configurations. However, not each and every variant is supported by this PIAS mechanism.
- Stiffeners on web plates are easily defined by giving the abbreviation of a Layout plane or Poseidon element "Tdeck" or "Shell". Now define the start and end point of the stiffeners moulded line and the centre to centre distance and number of stiffeners. Now the only things left are, on which side the stiffener should be positioned and defining the type and dimensions of the stiffener and how it's positioned in regard to it's moulded line.
- Transverse and longitudinal stiffeners on longitudinal plates are defined in a similar way as stiffeners on web plates. There two additions namely:
  - For transverse and longitudinal stiffeners on longitudinal plates one has to define a longitudinal start and end so instead of defining one begin and end point for the stiffeners moulded line there are now two, one on the longitudinal start and one on the longitudinal end. The begin and end point of the stiffeners moulded line will be interpolated in relation to the element on which the stiffener is located, decks and longitudinal bulkheads have a fixed interpolation direction but the Poseidon elements "Tdeck" and "Shell" can be set to vertical or transverse.
  - For the longitudinal stiffeners on longitudinal plates there is the extra option to define the rotation of the stiffener in relation to the moulded line of the stiffener. One can also define a rotation which is relative to the element on which the stiffener is positioned.
- Defining the plate fields is achieved by giving the abbreviation of the Layout plane or Poseidon element "Tdeck" or "Shell" and then defining a standard plate's dimensions as well as the moulded line of the Layout plane or Poseidon element. The physical area of the Layout plane or Poseidon element will be split into a bunch of plate fields with the afore mentioned standard plate dimensions. This division is handled in a specific way for each element:
  - Longitudinally, from the aft to the front; Transversally, from the centreline to the outside; Vertically, from the bottom to the highest point.
  - The specific Poseidon elements "Tdeck" and "Shell" are handled in the same way as Layout planes but the specific geometric definition of these elements is outside PIAS's control.

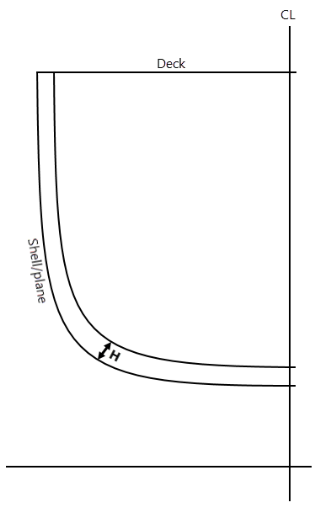


Definition moulded line plane and stiffener.

9.11.6.2 Examples of defining girders and webs

Example 1: There are always two intrinsic planes present namely 'Shell/deck' and 'maindeck'. And lets make, for convenience, 'centreline' (CL) also intrinsic. The web with fixed height H, as seen in the picture below, is defined by:

- Begin point plane 'Shell/deck' on intersection with CL.
- End point also plane 'Shell/deck' (this is neccessary because we are using a fixed girder height) on intersection with 'maindeck'.
- Fixed girder height H.



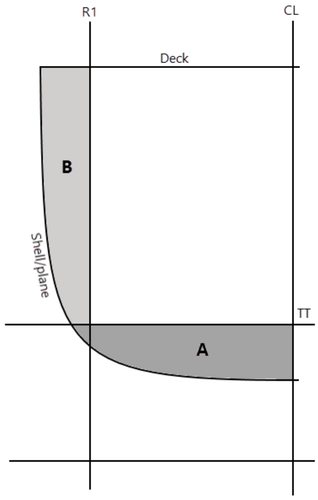
Definition girders and webs, example 1.

Example 2: Web plates A and B are defined separatly, as shown in the picture below. First web plate A:

- Begin point along plane Shell/deck on intersection with CL.
- At that begin point the inside of that web on height TT. For this the intersection of CL and TT is taken which is exactly what we want.
- End point plane Shell/deck on intersection with TT.
- At that end point the inside of that web on height TT. For this the intersection of TT and TT is used, which there are an infinite number off, but in this exceptional situation we will choose the intersection of Sheel/deck which is exactly what we want.

And web plate B:

- Begin point along plane Shell/deck on intersection with TT.
- At that begin point the inside of that web on breadth R1. For this the intersection of TT and R1 is taken and that is precisely what we want.
- End point along platen Shell/deck on intersection with main deck.
- At that end point the inside of that web on breadth R1. For this the intersection of main deck and R1 is used which is precisely what we want.



Definition girders and webs, example 2.

Example 3: Finally the construction, pictured below, with a deck and a longitudinal plate which both have been fitted with girders of different dimensions. There are four webs, web plate A is defined as follows:

- Begin point along plane TD on intersection with CL.
- End point along plane TD on intersection with LS0 or LS1.
- Fixed girder height which belongs to A.

Part B:

- Begin point along plane TD on intersection with LS0 or LS1.
- End point along plaen TD on intersection with Shell/plane.
- Fixed girder height which belongs to B.

Part C:

- Begin point along plane LS0 on intersection with Shell/plane.
- End point along plane LS0 on intersection with TD.
- Fixed girder height which belongs to C.

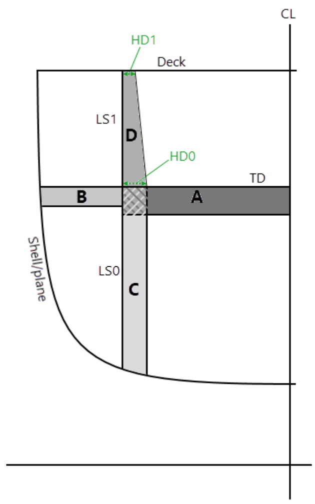
Part D:

- Begin point along plane LS1 on intersection with TD.
- At that begin point the inside of that web (as well as) along plane LS1 on intersection with TD with an offset HD0.
- End point along plane LS1 on intersection with maindeck.
- At that end point the inside of that web (as well as) along plane LS1 on intersection with maindeck with an offset HD1.

Note that the parts A and C partially overlap. This can't be helped because of the *design* model where by not every detail is been shown. For the benefit of the simplicity of the data definition (and thus the definition speed).

As mentioned before a web can be streched along multiple planes. Take for example; that C&D have a fixed girder height H (and a fixed thickness etc...) this can then be specified as follows:

- Begin point along plane LS0 on intersection with Shell/plane.
- End point along plane LS1 on intersection with maindeck.
- Girder height H.



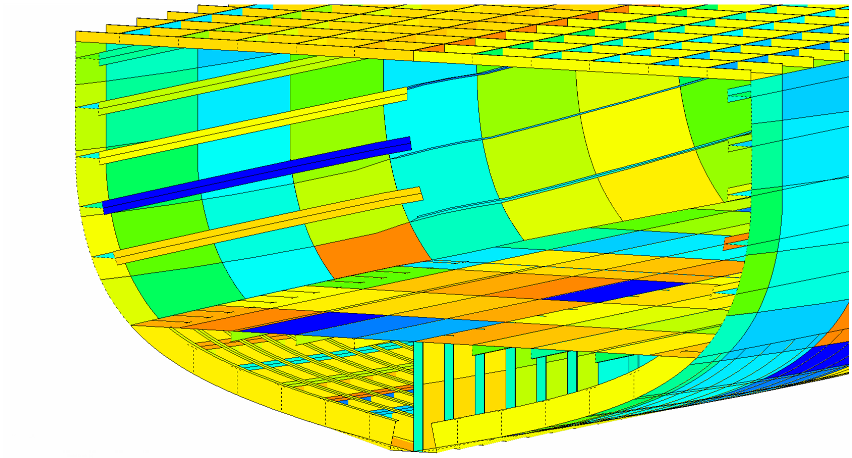
Definition girders and webs, example 3.

Lastly a description on the term 'Layout-plane'. Such a plane can be used for the definition of the positions of the webs, it's as simple as that. There are no further requirements on the presence or continuity of the planes. This gives a certain freedom which can be used as follows:

- A reference plane. This is simple, they strech along the whole vessel.
- A physical plane, which definitely can be used where it's located.
- A continuous physical plane. In the end a physical plane is nothing else that an infinite plane including a certain closed boundary. Also outside this closed boundary the infinite plane continuous and this can best be used in terms of positioning. For example, the height of a girder can be referenced on the tanktop even in places where this tanktop is not present.

9.11.6.3 Modus operandi of the PIAS to Poseidon conversion

1. Use this Layout option, which creates a text file, with .txt extension (as required for import in Poseidon).
2. Gebruik deze Layout optie, waarmee een tekstbestand wordt aangemaakt met extensie .txt (zoals vereist voor de import in Poseidon).
3. Start Poseidon, and select File/Open.
4. Choose at Files of type: Poseidon TXT File (∗.txt).
5. Indicate the file just created by Layout.



Ship, layout, plates, girders, webs and stiffeners defined in PIAS, converted to Poseidon.

9.11.6.4 Design choices, limitations and conditions

The PIAS→Poseidon conversion has, in close collaboration with the leading customer, been designed for this anticipated use:

1. Ship data and layout data which are intrinsically present in PIAS must be converted to Poseidon.
2. It is the intention of this customer to ultimately manage data of hull, layout and structure from a central design system, and to convert those to Poseidon via PIAS.
3. For practical reasons, it was decided that data types not included originally in PIAS (such as deck loads, girders and stiffeners) should, for the time being, be entered in PIAS in simple alphanumeric menus. These input menus lose their role once these data are included in the central design system. That is why these import menus are quite Spartan, and are not equipped with graphical input or feedback facilities.
4. PIAS stores these additional Poseidon data in XML format. This makes it possible for that central design system to write out the same data in that format as well, so that they are immediately available in PIAS (and hence for conversion to Poseidon).
5. The data that PIAS can supply to Poseidon is sufficient for that first customer with the current implementation. However, there are also things that are not included in PIAS and this conversion, such as pillars, man holes and welding details. For the time being, it is not the intention to extend the conversion with these components, although in specific cases extension can always be discussed with concrete users.
6. Obviously, each user is free to supplement or modify the data that is imported from PIAS with the User Interface of Poseidon.

Attention

Poseidon‘s interface format has occassionaly very specific requirements and restrictions. In this conversion option of PIAS quite some effort has been put to meet them as much as possible, and the many tests show that the geometry from PIAS transfers rather well into Poseidon. However, in general, a flawless operation in Poseidon is not guaranteed.

On this conversion the following remarks and conditions apply

- This conversion is tested with and valid for Poseidon version 17.0.8, (2018).
- Poseidon is fully dedicated to construction frames, so it is of utmost importance to define the frame spacings properly in PIAS (see section 7.2.1.3 on page 167, Frame spacings), including the position of the aftmost and foremost frames.
- Angled transverse bulkheads are excluded from the conversion.
- For Poseidon it is a prerequisite that frame 0 coincides with the APP (see chapter 2.2 of the *Fundamentals* manual of Poseidon).

In Poseidon a so-called *longitudinal element* is bounded by either another element, or the shell. However, in reality these boundaries may changed, for example the lower boundary of a longitudinal bulkhead, which may at the aft side be the tanktop and at the fore side the shell. For Poseidon this bulkhead should be split into two parts;

an aft part which is bounded by the tanktop, and a forward part bounded by the shell. PIAS will detect such a phenomenon, and split automatically. However, there is a pitfall which is related to the split location which should be positioned exactly on the intersection between two internal planes and the shell. In our example the intersection of longitudinal bulkhead, tanktop and shell. This point can easily be obtained by interpolation, however, because PIAS and Poseidon apply their own independant interpolation methods it may very well occur that on the split position as determined by PIAS, Poseidon does *just* not find an intersection between bulkhead and tanktop. In such a case the split position will have to be modified slightly, manually in Poseidon. This phenomenon is intrinsic to Poseidons *modus operandi*, in particular to the requirement that a longitudinal element must be split, and can in principle not be avoided. However, in practice its occurrence is proportional to the interpolation neccessity, so, it is advised to include in PIAS as many frames as possible, preferably all construction frames (which can be generated conveniently with Fairway).

9.11.7 Write XML file

For the purpose of communication with other software, with this option one may export the ship's subdivision data to an XML file. At present this file contains data of compartment shapes, bulkheads & dekcs and reference planes. This data is limited to that which users have really needed up to now, and includes eg. shapes, names and location of the different entities, but not all the details. E.g. the second compartment name and the sounding pipe are missing. However, if required these data can be added, please also refer to section 3.8 on page 29, Export to and import from XML. The name of the file written is `PIASfilenaam.fromLayout.XML`.

Incidentally, these XML data are not only available via a *file*, but also in direct and permanent comunication between computer programs. In this way, multiple persons can work with different CAD programs simultaneously on the same ship design. More details can be found in a paper, `www.sarc.nl/images/pdf/publications/international/2015/Koelman%20Compit%202015.pdf`, which has been presented at the Compit'15 conference.

9.11.8 Read XML file

With this function, an XML file with ship's internal layout can be imported into Layout. All comments made in the previous section, on exporting, also apply here. The expected file name is `PIASfilenaam.toLayout.XML`.

If a file is imported that has planes but not (everywhere) compartments, this will lead to spaces between those planes that are not filled with a compartment. It might be useful to have Layout then generate such compartments. Therefore, the program asks the question 'What to do with empty spaces between planes', to which one should then answer appropriately.

9.12 File and backup management

Backups of the ship's subdivision can be made and restored here. Here is also the option 'Stop without saving'. See for the details section 2.9 on page 15, Data storage and backups.

Chapter 10

Hydrotables: hydrostatics and stability tables

The Hydrotables module focuses at the computation and output of tables or diagrams of hydrostatical and stability properties which are related to hull form and/or compartments. In particular:

- *Hydrostatic tables.*
- *Tables and diagrams of cross curves (NKsin(φ) tables).*
- *Bonjean tables.*
- *Deadweight tables and deadweight scale.*
- *Tables of wind heeling moments.*
- *Tables and diagrams of maximum allowable Vertical center of Gravity (VCG'), for intact as well as damaged ship.*
- *Floodable length curves.*
- *Tables of maximum allowable grain heeling moments according the 'SOLAS Grain Code'.*
- *Trim diagram according to 'van der Ham's' method.*
- *Tonnage tables for European inland waterway vessels according the 'Convention on the measurement of inland navigation vessels, Geneva, 15-2-1966'.*

10.1 Main menu

Having started up Hydrotables, one enters the main menu, the various options of which are explained in more detail in the following sections.

Hydrostatics and stability tables

1. Parameters per table or diagram
2. Specify output sequence
3. Output according to the specified output sequence
4. Export to XML according to the specified output sequence
5. Configure the Local cloud monitors
6. Activate the Local cloud monitors
7. File management of configurations

10.2 Parameters per table or diagram

At this option, for each table or diagram the desired parameters can be specified, such as the trimming range, table increment or table type. The parameters can either be given for each individual table or diagram, or the mechanism can be used which links a parameter to the same parameter in a different table or diagram.

In this menu the selection list as printed below appears, where for each table or diagram the different parameters can be set. The *linked parameters* mechanism is only discussed at the second option, section 10.2.2 on page 253, Cross curve tables, for the application in the other tables is analogue. Also other options which are present at multiple types of tables are only discussed at their first occurrence, it will not be re-iterated.

Parameters per table or diagram

10.2.1 Hydrostatics

At this option the parameters of the (to be computed and printed) hydrostatic tables can be specified, of which the most is the combination base unit, start value, increment and end value, here numbers can be entered, or <I> or <-I>, for ∞ and -∞ respectively, which makes the table to run from the largest or smallest possible value for this particular vessel. The base unit is the unit whose tabular values are fixed in this menu, and on basis of which all hydrostatic properties are computed and printed. For the hydrostatics, the possible table values are displacement (in metric tons), (mean) draft from base line and (mean) draft from Bottom of Keel. Apart from the base unit, the table range is fixed by the start value, the end value and the increment (the step size), which are obviously expressed in the base unit (so, in meter or in ton). One could also wish to have some additional table values, apart from the regular ones, for example exactly on summer draft (which, in general, will to coincide with a regular value). Such additional entries can be specified at the option [Number of extra values]. If the option [rounded increment] is set to *yes*, then the specified increment will not be applied *exactly*, but rounded-off to the nearest 'nice' value. This rounding-off always works (with an exception for *Extra value*)(if specified so), but is justified in combination with a *linked parameter*.

Finally, the trims for the tables can be specified, as well as the output format, which come in three:

- A short table, which contains per draft (or displacement) a single line with the most important hydrostatic results.
- A long table, which contains per draft (or displacement) a column containing all primary hydrostatic results.
- An extended table, which contains per draft (or displacement) a column which also contains a number of additional or derived results, being:
  - Hull form coefficients.
  - Wetted surface, or, to be more precise, the sum of the wetted surfaces of the hull forms defined with Hulldef, without their transom areas. The transom area is omitted because that is custom for resistance predictions.
  - The *waterplane area coefficient forward of L/2*, a coefficient which is required in reg. 39 (on minimum bow height) of the Load Lines Convention.

This manual is not the place to discuss all hydrostatic parameters extensively, however, a few remarks are applicable:

- The volume of a possibly given mean shell plate thickness will be included into the moulded volume, please also refer to section 7.2.1.1 on page 167, Main dimensions and allowance for shell and appendages.
- All hull form coefficients are determined by dividing by the nominal main dimensions (such as L_{PP} and the moulded breadth), so not by the true waterline length and breadth at a certain draft.
- If waterline properties (such as area or moments of inertia) would show a bit of a jerky tendency against draft, the you may find the explanation and remedy in section 7.2.4.1 on page 174, Number of frames.

Printing the tables is initiated with option section 10.4 on page 260, Output according to the specified output sequence, where all tables specified in section 10.3 on page 260, Specify output sequence will be computed and printed in one run. For an occasional print of only the hydrostatic tables the option [Print] can be used.

10.2.2 Cross curve tables

The setup for cross curves tables is similar to that of hydrostatics: base unit, start value, increment and end value. For cross curve tables there is a choice from three formats:

- An extended table, which contains for each angle the displacements, centers of buoyancy as well as the KN sin(φ) — for which the definition is given insection 2.6 on page 9, Definitions and units. Such an extended tables also contains draft values, a heading can e.g. contain Tφ=0 of Tφ=20; which means that this is the draft at zero or twenty degrees respectively.
- A short table, which only contains the KN.sin(φ).
- An optional additional table, which lists the angle/displacement combination of downflooding (by an open opening).

Next, there is the question to which side (PS/SB), the cross curve table will be calculated and printed. This is regulated by the setting as discussed in section 5.1.7 on page 44, Calculate intact stability etc. with a heeling to :

- With setting 'portside' the table will be for heel to PS, with setting 'starboard' for heel to SB.
- With 'side of the heel' a single table will be computed for heel to SB. That is rather arbitrary, but has always been so in PIAS and there is no reason to change it.
- With 'portside and starboard' the mechanism is:
  - With only symmetric hull forms without transverse translations: a single cross curve table. The — optional — table of opening immersion is valid for the worst side of PS or SB.
  - In case of asymmetrical hull forms (or symmetric shapes with a transverse translation), two separate cross curve tables will be produced; one to PS and one to SB.

For defining the various numerical parameters the optional *link* mechanism can be applied, where a parameter can be linked to the corresponding parameter from another table. In this fashion one can conveniently link the range and steps of the cross curve table to those from the hydrostatic table. This is achieved by selecting in the top-right column 'linked to parameters of hydrostatics', after which those hydrostatic parameters show up in the right column. For each parameter one can activate the *link* in the middle column, which make the hydrostatics parameter value to be used for the cross curves. This mechanism also covers a difference in base units; assume that the hydrostatic table is defined in terms of draft, and the cross curves in terms of displacement, then the hydrostatic drafts are converted to displacements. In general this will not result in 'nice' displacement values, however, with the [rounded increment] option (as discussed in the previous paragraph) these values will be rounded-off.

Please consider that this *link* mechanism is optional; it has been included for your convenience, in order to assist you to produce and reproduce the various tables in the same range, but application is not mandatory. After al you can always use naked parameters for each table, without any link at all.

10.2.3 Cross curve diagram

The parameter menu for the cross curve diagram is similar to that of the cross curve tables. With the exception of the option [Connect points of curve of flooding through openings with straight lines]. For an explanation regarding this option see section 16.5.1 on page 320, Settings intact stability and then **Connect points of GZ-curve with straight line segments**.

In case the option [Including curve which indicates the flooding through openings] is turned on then the meaning of the drawn lines is as follows. The light blue lines show, per opening, the flooding of that open opening. If such an opening becomes downflooded before the start value or after the end value, then this line will be omitted. It is possible that the KN sin(φ) line is split into a red and a green part due to an open opening becoming downflooded. Here the red part indicates that the ship is downflooded and that green part is safe.

10.2.4 Bonjean tables

In this menu the properties are specified of the Bonjean tables, which are tables of areas and vertical centers of gravity of all frames as defined in PIAS. The parameters in this menu are similar to those in the hydrostatic table menu.

10.2.5 Deadweight tables

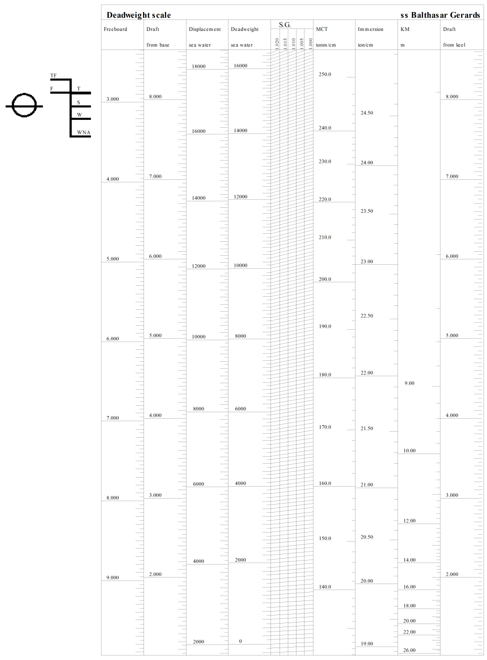
In this menu the deadweight table properties are specified. In order to produce such tables in any case the summer draft op the ship should be given correctly, this can be done in the main dimensions menu of Hulldef. If other freeboard-related drafts, such as WNA or FW, are also given (at section 7.2.1.5 on page 169, Maximum drafts and minimum freeboards) then they will be incorporated in the table. In this particular menu only the draft or displacement increment and the light ship weight must be given, which will be evident. An example of a deadweight tables is depicted below.

	IN SALT WATER S.W. = 1.025					Draft	IN FRESH WATER S.W. = 1.000			
	Frbr	Immersion	Mct	Displ	Deadwght	us keel	Deadwght	Displ	Immersion	Frbrd
	m	ton/cm	tonm	ton	ton	m	ton	ton	ton/cm	m
Trop.fresh						8.267	15236.56	17256.56	24.12	2.713
Fresh						8.102	14839.08	16859.08	24.06	2.878
Tropical	2.885	24.67	248.21	17263.28	15243.28	8.095				
Summer	3.050	24.58	245.83	16856.98	14836.98	7.930				
Winter	3.210	24.47	242.84	16464.57	14444.57	7.770				
Winter N.A	3.430	24.31	238.29	15928.14	13908.14	7.550				
	2.713	24.72	249.82	17687.97	15667.97	8.267	15236.56	17256.56	24.12	2.713
	2.992	24.61	246.79	17000.00	14980.00	7.988	14565.37	16585.37	24.01	2.992
	3.400	24.33	239.04	16000.00	13980.00	7.580	13589.76	15609.76	23.74	3.400
	3.814	23.99	229.22	15000.00	12980.00	7.166	12614.24	14634.24	23.40	3.814
	4.235	23.64	220.36	14000.00	11980.00	6.745	11638.54	13658.54	23.07	4.235
	4.661	23.28	211.48	13000.00	10980.00	6.319	10662.95	12682.95	22.72	4.661
	5.093	22.96	202.83	12000.00	9980.00	5.887	9687.35	11707.35	22.40	5.093
	5.533	22.61	195.07	11000.00	8980.00	5.447	8711.85	10731.85	22.06	5.533
	5.978	22.31	187.55	10000.00	7980.00	5.002	7736.41	9756.41	21.77	5.978
	6.429	22.00	180.39	9000.00	6980.00	4.551	6761.00	8781.00	21.46	6.429
	6.887	21.67	173.61	8000.00	5980.00	4.093	5784.88	7804.88	21.14	6.887
	7.352	21.33	166.34	7000.00	4980.00	3.628	4809.27	6829.27	20.81	7.352
	7.825	21.01	159.89	6000.00	3980.00	3.155	3833.66	5853.66	20.50	7.825
	8.305	20.65	153.13	5000.00	2980.00	2.675	2858.05	4878.05	20.15	8.305
	8.794	20.24	145.65	4000.00	1980.00	2.186	1882.44	3902.44	19.75	8.794
	9.294	19.73	136.69	3000.00	980.00	1.686	906.83	2926.83	19.25	9.294
Lightship	9.799	19.05	125.40	2020.00	0.00	1.181				

Deadweight tabel.

10.2.6 Deadweight scale

In this menu the deadweight scale parameters must be given, these come in two: In the first place the light ship weight — which is actually the same as for the deadweight tables — and secondly the question whether a Plimsoll mark should be included in the output. Obviously, for the latter the freeboard drafts should have been given at section 7.2.1.5 on page 169, Maximum drafts and minimum freeboards. In a deadweight scale with a small draft range, by the way, combined with a plot of the Plimsoll mark, the latter may be drawn rather large. The reason is that the load line marks must coincide with the drafts in the deadweight scale, which leads inevitably to such an effect.



Example deadweight scale.

10.2.7 Wind heeling moment tables

Here tables of wind heeling moments & wind levers can be configured and computed. The relationship between the various wind moment related input data is discussed in chapter 14 on page 273, Wind heeling moments, it is recommended to visit that chapter. After choosing this option a window opens where the **wind data** (as introduced in Hulldef) are listed, accompanied in the first column whether it is selected. With <Enter> (or <doubleclick left mouse button>) opens a deeper level, where the step sizes (etc.) of the wind moment tables can be entered, completely analogous to the other tables here in Hydrotables. In a very rare occasion the wind heeling levers are already available from another source, in which case those values can be entered in the input module Hulldef, see section 7.2.7 on page 183, Wind data sets.

Tables of wind arms are eventually printed for all combinations of selected **wind data** and selected **wind contours**. The printed wind moment table contains the following parameters:

- Draft.
- Displacement.
- Heeling moment.
- Heeling lever.
- Windage area.
- Wind lever, which is the distance between the center of effort of the wind, and the lateral point of the submerged hull, half draft or a fixed height, depending on which option has been chosen in Hulldef. The chosen option is printed at the end of the table. The wind lever will only be printed in case of a constant wind pressure, so not for wind pressures that vary with the height.

However, there is one exception on this output, which occurs the wind heeling levers have been entered by the user (in Hulldef, as discussed above). In that case only draft, displacement and heeling lever will be printed, as the other parameters are not available.

10.2.8 Maximum VCG' intact tables

For these tables of maximum allowable VCG' in intact condition also drafts and trims have to be specified, similar to those for the cross curves tables, as discussed in section 10.2.2 on page 253, Cross curve tables. Additional parameters for the maximum allowable VCG' are:

- *Calculate maximum VCG' on basis of*, with the choice between *user-defined angles* and *default angles*, as discussed in section 5.4.13 on page 50, Compute probabilistic damage stability on basis of.
- Number of wind contours for which the maximum VCG' tables will be computed. In the parameter window only the *number* of wind contours is given, however, on modification a selectable list of existing wind contours (as defined in section 7.2.6 on page 182, Wind contour) appears. Each selected wind contour implies an independent stability criterion and consequently results in its own maximum allowable VCG' value.
- *Show maximum VCG' results as*, with the choice between *maximum allowable VCG'* and *minimum required G'M*. This setting has no effect on the computations as such, but governs whether output value wil be VCG' or G'M.

10.2.9 Maximum VCG' intact diagrams

The parameter menu for the this diagram is equal to that of the maximum VCG' tables.

10.2.10 Maximum VCG' damaged tables and diagrams

At this option the parameters can be specified which are relevant for the computation of tables and diagrams of maximum allowable VCG' in intact condition, required to fulfill damage stability criteria in case of flooding. This is a complex set of computations, so the required input data are somewhat more extensive than for the other tables in Hydrotables. So, first the following submenu appears:

Maximum VCG' damaged tables and diagrams

1.	Specify calculation parameters
2.	Select and edit standard damage cases
3.	Generate damage case contents
4.	Define intermediate stages of flooding

10.2.10.1 Specify calculation parameters

In the first place, here the start value, end value and increment etc. can be given, which will now be familiar from the other types of tables here in Hydrotables. Specific parameters are:

- Calculate maximum VCG' on basis of 'user-defined angles' or 'default angles', a choice equal to the one at intact maximum VCG', see section 10.2.8 on the previous page, Maximum VCG' intact tables.
- Print maximum allowable VCG' in damaged condition per 'damage case' or per 'displacement'. With 'per damage case', for each damage case and each trim one table is printed with all displacements, as well as a plot of the maximum allowable VCG' as a function of the displacement. With 'per displacement' for each displacement/trim combination one table of all damage cases is printed.
- Whether diagrams of maximum allowable damaged VCG' should be added to the tabular output. An example of such a diagram is depicted below.
- Number of wind contours for which the tables of maximum allowable VCG' should be produced, as also already discussed at section 10.2.8 on the preceding page, Maximum VCG' intact tables.
- *Show maximum VCG' results as*, with the same function as with the intact VCG' tables or diagrams.

Calculation of maximum permissible KG' covers many iterations and intermediate steps (such as the determination of maximum KG' for the individual stability requirements), for which it would certainly not be practical to include all those in the output. To understand exactly *why* a maximum KG' is as it is, there used to be the possibility of intermediate results to be stored in a file, which could afterwards be browsed. That option has been repealed because there are now more convenient options. The damage cases specified here can be imported into the deterministic damage stability in Loading, see [Import] in section 20.3 on page 397, Input and edit damage cases. And once available in Loading, one can make a loading condition with exactly the critical KG' see what happens, and which criterion is most critical. Further details can be obtained by switching on in Loading the intermediate results of the stability assessment, in the column 'intermediate results' of the stability requirements menu, as described in section 15.1 on page 276, Manipulating and selecting sets of stability criteria.

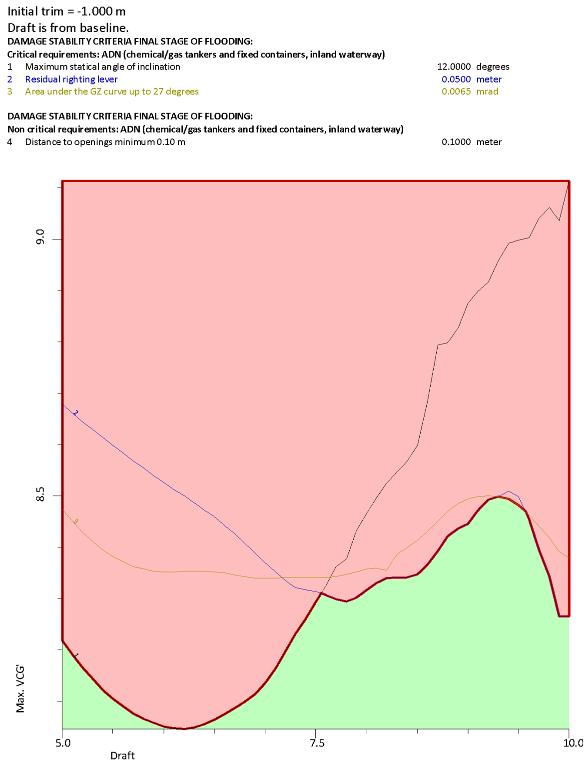


Diagram of maximum allowable VCG' for one damage case.

10.2.10.2 Select and edit standard damage cases

In this menu, which is discussed in detail in section 20.3 on page 397, Input and edit damage cases, damage cases can be defined, with a maximum of 500. Please do not forget for each damage case to verify whether a compartment contains liquid in intact condition, because that will flow out in case of damage. For those cases their weight and density should be given in the last two columns of the damage case.

10.2.10.3 Generate damage case contents

In the previous menu option it has been discussed how damage cases can be defined. For a limited number of damage cases this is not particularly laborious, however, in some occasions repeating combinations of *root damage cases* and tank fillings need to be defined (here the word ‘root damage case’ indicates only a combination of damaged compartments, without tank fillings). With N root damage cases and M tank fillings, manually Nx↩ M damage cases should be defined. At large N or M it is more convenient to define the root damage cases and the tank fillings separately, and to generate all combinations. This facility is available in this menu, through four sub-options:

- Select loading conditions. In order to prevent that for the tank fillings separate data structures should have been developped, the existing loading conditions as defined in Loading, are applied. The tank fillings of all selected loading conditions in this menu will be applied in the subsequent generation of damage cases.
- Select and edit root damage cases. Here the root damage cases can be selected. From a practical point of view those cases are the same as defined in Loading. For further discussion on this option, reference is made to section 20.3 on page 397, Input and edit damage cases.
- Generate damage cases. With this option the damage cases are actually generated, where their names are composed from the names of the root damage case and the loading condition. In this respect it should be considered that the aft and forward boundaries of the damage cases, as intended in the applicable regulation, do not necessarily have to coincide with the extreme compartment boundaries. *If* those boundaries are relevant, then they have to be defined manually. The same reasoning applies to the ‘morecomp’ column in the damage case menu, its value should be verified and possibly adapted after the generation of damage cases. This ‘generate damage cases’ option comes in two flavors, one where the existing damage cases are replaced by the newly generated, and one where the generated cases are added to the existing list.

10.2.10.4 Define intermediate stages of flooding

Here the regular intermediate stages of flooding can be given, as percentage of the final stage of flooding. The stages here are the same as defined for the deterministic damage stability calculations in Loading, see section 16.2.2.3 on page 313, Define stages of flooding. Incidentally, there are many more considerations and possibilities regarding intermediate stages, which are discussed in chapter 21 on page 402, Internal flooding in case of damage, through pipe lines and compartment connections.

10.2.11 Floodable lengths curve

With this option the calculation parameters of the floodable length curve are specified. This curve indicates the maximum length of any compartment at any longitudinal position, so that the vessel just submerges to the margin line in case this compartment is damaged. For this purpose a minimum of three points have to be defined as ‘margin line points’ in the PIAS module Hulldef. The parameters to be specified are:

- The parameters first and last ‘longitudinal position’, as well as its increment, which determine the locations where the floodable lengths are being computed. As a rule for the ‘first longitudinal location’ the APP is taken, for the ‘last’ the FPP and for the ‘increment’ a nice value, such as $L_{PP}/100$.
- The displacement, in ton, and the Longitudinal Center of Gravity, in meters from APP, of the intact ship.
- The permeability, which is constant for the entire vessel. For varying permeabilities multiple computations of the floodable lengths are required.
- The multiplication factor, as defined in the SOLAS part where the floodable lengths requirements are addressed.

10.2.12 Maximum grain heeling moment tables

With this option the parameters for a maximum allowable grain heeling moment table — from which an example is depicted below — can be specified. These moments are computed according the requirements of the International Grain Code (IMO MSC.23(59)), for a range of drafts or displacements and VCGs. Besides the usual trims, as well as the draft/displacement range these parameters comprise:

- Lowest, highest and increment of the Vertical Centres of Gravity (VCG’s) to be taken into account.
- Whether the deck edge is allowed to immerse. This criterion is applicable for vessels constructed after January 1, 1994, for which the inclination due to the shift of grain is limited to 12 degrees, or the angle of deck edge immersion. For vessel constructed before that date only the 12 degrees limitation applies. For this criterion the deck edge should be defined, this can be done in module Hulldef.

The side (PS/SB) for which this table is valid depends from the setting as discussed at section 5.1.7 on page 44, Calculate intact stability etc. with a heeling to :

- With the ‘portside’ setting, the table is valid for heel to PS, with setting ‘starboard’ or ‘side of the heel’ for heel to SB.
- With ‘portside and starboard’ the table contains the lowest allowable grain heeling moment of heel to SB or PS.

Attention

According to the *Grain Code* the angles 12° and 40° should explicitly be included into the calculations. So, please take care that these angles are part of the range of heeling angles, as specified in Config.

The trim is modified to meet constant LCB
Calculation for inclination to SB
The calculation is valid for the following heeling angles:
0.00, 2.00, 5.00, 10.00, 12.00, 15.00, 20.00, 25.00, 30.00, 35.00, 40.00, 50.00

Grain stability criteria
Maximum angle of inclination 12 degrees
Area between GZ curve and grain heeling lever 0.075 mrad
Minimum GM' 0.30 m
Deck edge not immersed

Displacement in tons

VCG	6600	6800	7000	7200	7400	7600	7800	8000	8200	8400	8600	8800	9000	9200	9400
5.50	7101	7073	7049	7029	7012	6997	6986	6977	6973	6972	6974	6980	6988	7000	7016
5.55	7028	6998	6972	6949	6930	6913	6900	6889	6882	6879	6879	6882	6888	6898	6912
5.60	6955	6923	6894	6870	6848	6829	6813	6800	6792	6786	6784	6785	6789	6796	6808
5.65	6882	6847	6817	6790	6766	6745	6727	6712	6701	6693	6689	6688	6689	6695	6704
5.70	6809	6772	6740	6710	6684	6661	6641	6623	6610	6600	6594	6590	6590	6593	6600
5.75	6736	6697	6662	6631	6602	6577	6555	6535	6519	6508	6499	6493	6490	6491	6496
5.80	6663	6622	6585	6551	6521	6493	6468	6446	6429	6415	6404	6396	6391	6389	6392
5.85	6590	6547	6507	6471	6439	6409	6382	6358	6338	6322	6309	6298	6291	6288	6288
5.90	6517	6471	6430	6392	6357	6325	6296	6269	6247	6229	6213	6201	6192	6186	6184
5.95	6444	6396	6353	6312	6275	6241	6210	6181	6157	6136	6118	6104	6092	6084	6080
6.00	6371	6321	6275	6233	6193	6157	6123	6092	6066	6043	6023	6006	5993	5982	5976
6.05	6298	6246	6198	6153	6111	6073	6037	6004	5975	5950	5928	5909	5893	5881	5872
6.10	6225	6171	6120	6073	6030	5989	5951	5915	5885	5857	5833	5812	5793	5779	5768
6.15	6152	6095	6043	5994	5948	5905	5865	5827	5794	5764	5738	5714	5694	5677	5664
6.20	6079	6020	5965	5914	5866	5821	5778	5738	5703	5671	5643	5617	5594	5575	5560
6.25	6006	5945	5888	5834	5784	5737	5692	5650	5613	5579	5548	5520	5495	5474	5457
6.30	5933	5870	5811	5755	5702	5653	5606	5562	5522	5486	5453	5422	5395	5372	5353
6.35	5860	5795	5733	5675	5620	5569	5519	5473	5431	5393	5357	5325	5296	5270	5249
6.40	5787	5719	5656	5595	5539	5485	5433	5385	5341	5300	5262	5228	5196	5168	5145
6.45	5714	5644	5578	5516	5457	5401	5347	5296	5250	5207	5167	5131	5097	5067	5041
6.50	5641	5569	5501	5436	5375	5316	5261	5208	5159	5114	5072	5033	4997	4965	4937
6.55	5568	5494	5424	5357	5293	5232	5174	5119	5068	5021	4977	4936	4898	4863	4833
6.60	5495	5419	5346	5277	5211	5148	5088	5031	4978	4928	4882	4839	4798	4761	4729
	9.93	9.76	9.61	9.48	9.35	9.23	9.12	9.02	8.92	8.83	8.74	8.66	8.59	8.52	8.46

Maximum allowable VCG according to IMO Grain Code (m)

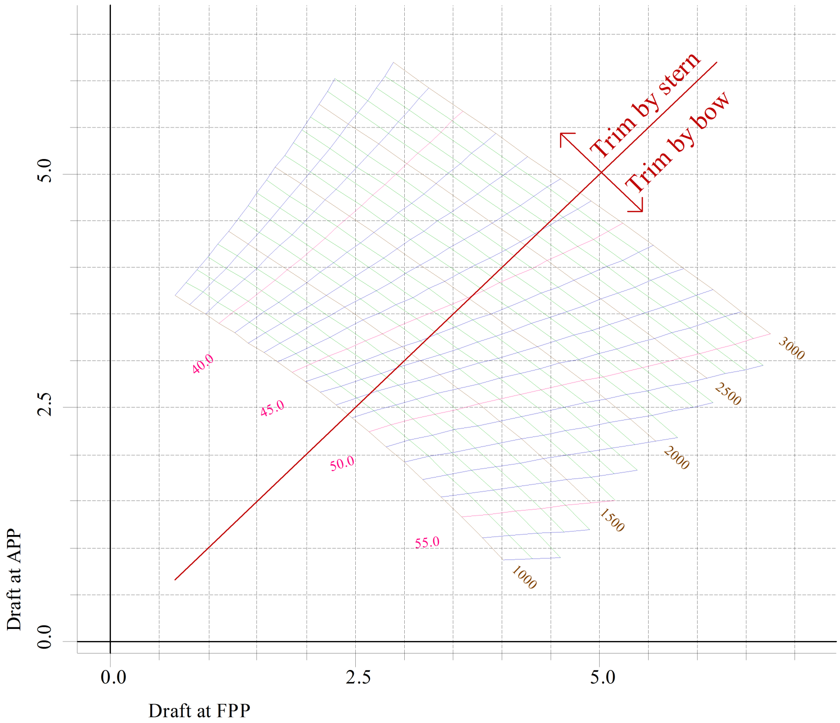
Notes: VCG is the actual vertical centre of gravity (m), the grain heeling moment is in tonm.
The actual VCG shall not only be less than the maximum according to the Grain Code, as stipulated above, but also less than any other maximum resulting from other intact or (possibly) damage stability criteria.

Example of a table of maximum allowable grain heeling moments.

10.2.13 van der Ham's trim diagram

With the option the parameters for a trim diagram according to van der Ham (as published in the Dutch *Schip en Werf* journal, 25TH edition, 1958, No. 23) can be specified. In PIAS the *Longitudinal Centre of Gravity* (in combination with displacement) is plotted, instead of the *Longitudinal Moment* as in the original definition. The reason for this is that in PIAS the origin for all longitudinal distances is the APP, instead of Lpp/2 as used by van der Ham. With a van der Ham diagram the relations between displacement/LCG and drafts fore and aft can be read instantaneously, see the example below. The parameters for this diagram are:

- Draft range or displacement range. Please note that the diagram is always drawn with drafts on both axes, and displacements along iso-displacement curves within the diagram, regardless whether in this menu drafts or displacements are specified. If drafts are specified, they will internally be converted to displacements, so for 'nice' displacement increments it is better to work on a displacement basis (or, alternatively, apply the [rounded increment] option.
- The maximum trim by bow or stern. This figure should be positive, and is used to determine the LCG range.
- The number of LCG steps, which determines by approximation the amount of LCG curves in the diagram.



Van der Ham's trim diagram.

10.2.14 IWW tonnage tables

With this option the parameters are defined for tonnage tables according the 'Convention on the measurement of inland navigation vessels, Geneva, 15-2-1966'. The only extra parameters that still need to be defined, besides the usual increment and start and end value parameters, are the trims for the *Light waterline*(start value) and the *Maximum waterline*(end value). The table will than consist of a range of drafts, running from the start to end value with their respective trim, and by those drafts the interpolated trim and the accompanying tonnage.

10.3 Specify output sequence

In this menu the tables to be printed can be specified, as well as the print sequence. For each table also a page number and chapter name can be given, which will be printed at the bottom of each page. The left column, *selected* specifies whether the output of that line is actually included in the output to be produced.

When calculating the table 'Maximum VCG' damaged and intact' the parameters of section 10.2.8 on page 255, Maximum VCG' intact tables are leading.

10.4 Output according to the specified output sequence

This option prints all tables according to their setup and print selection as specified in the previous two options. Obviously, their correctness will be dependent on the correctness of main particulars and settings, as can be given in Hulldef and Config, notably at:

- Wind heeling moment tables, where no additional parameters are required in this module because all required data are already given elsewhere, please also refer to chapter 14 on page 273, Wind heeling moments.
- Deadweight tables and deadweight scale, which depend on the various freeboard drafts as given in section 7.2.1.5 on page 169, Maximum drafts and minimum freeboards.
- Tables of maximum allowable VCG', both intact and damaged, which are governed by the stability criteria as specified in section 5.6 on page 51, Stability criteria.

10.5 Export to XML according to the specified output sequence

With this option an XML file is generated which contains all, or at least all relevant, computation results of Hydrotables. This file is intended to be consumed by other software, for it is much more robust to import an XML file than to extract figures from human-centered output files. Please also refer to section 3.8 on page 29, Export to and import from XML.

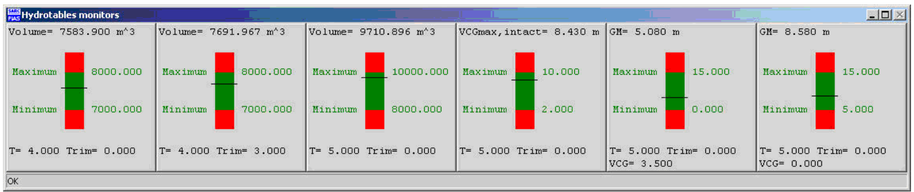
10.6 Configure the Local cloud monitors

The local cloud concept is introduced in general terms in section 2.11 on page 17, Local cloud: simultaneous multi-module operation on the same project. Hydrotables is able to 'eavesdrop' the cloud, and to compute and show a number of hydrostatics-related parameters of the hull form, we call that the 'local cloud' monitor of Hydrotables, its visual representation is shown in the next paragraph. In this menu multiple monitors can be defined, with per monitor:

- The parameter to monitor, where the choice is between volume, displacement, longitudinal center of buoyancy, KM, metacentric heigh (GM) and the maximum allowable VCG', intact or damaged.
- The draft and trim for with this parameter should be computed.
- The VCG', the vertical center of gravity, which obviously only plays a role when monitoring GM.
- Possible limits of the parameter, which are its required minimum and maximum values. If these limits are given, then the monitor shows a bar within the boundaries of these limits, so it is easily visible when the limits are exceeded. It is not required to specify these limits, in which case no bar is drawn, because that would be useless, however, the actual numerical parameter value is always shown.
- In the first column an on/off switch of the monitor.

10.7 Activate the Local cloud monitors

With the previous option the local cloud monitors can be configured, with this options they can be activated. The effect will be that the main menu of Hydrotables is replaced by one or more bars, indicating the configured parameters and limits. The figure below shows an example, where the volume is monitored at three draft/trim combinations, the maximum allowable VCG' at one draft and the GM with two different VCG's. It might be a bit exaggerated example, however, it clearly shows the possibilities. Now all hull form modifications are monitored, so if with for example Fairway the hull form is changed, in this monitor directly the corresponding volume etc. will be shown.



Local cloud monitor, monitoring and showing five parameters.

10.8 File management of configurations

Backups of the configurations for the tables and diagrams can be made and restored here. Here is also the option 'Stop without saving the configurations'. For details we refer to section 2.9 on page 15, Data storage and backups.

Chapter 11

Grainmom: calculation of grain heeling moments according to the IMO Grain Code

This module enables you to calculate the actual grain heeling moments according to the requirements of IMO Grain Code, or according to the requirements of the Dutch Shipping Inspectorate.

Calculation of grain moments

11.1 Selection and definition of grain holds

Attention

A compartment which is going to be used to compute grain heeling moments will be defined by one or more subcompartments, which have to comply to the following definition requirements:

- The subcompartments have to be defined from portside to starboard.
- No negative subcompartments are allowed.
- The vertical line through a longitudinal girder has to be a boundary of a subcompartment.

If this is a new project or no existing data is found, you will be prompted to select grain holds. By choosing "Yes", you will be redirected to a list of compartment already defined in Layout, where you are able to select any number of compartments for input through the first column [Selected]. If you choose "No", then you can navigate to the Layout compartment list by clicking on [(De)select grain holds] on the menubar. When exiting from the compartment list, the selected compartments are now visible in a table followed by their subcompartments. The compartment rows are highlighted in colour. You can double-click on a highlighted compartment row to expand/collapse its subcompartment(s) rows. Through the option [Manage] in the menubar you can choose to *Expand all* or *Collapse all* the subcompartment rows.

Selected
If "Yes" then the grainhold is included in the calculations for the output.

Name
Compartment/subcompartment name.

Void volume table
The height of the void space in [m] used for calculations and output of the volume table (acc. to Chapter VI, Section I A.a of the IMO Grain Code).

Volume step m³, Number of steps, Rounded increment
Those 3 columns are linked:

- The tables are calculated at the volume step (interval) defined here.
- Rounded increment will round (and approximate) the fraction $\frac{total\_volume}{nr\_of\_steps}$ or $\frac{total\_volume}{volume\_step}$ depending on the input.

Void grain moment table
The height of the void space in [m] used for calculations and output of the grain moment table.

Girder
Here you can define whether a girder is present. Only the girders at the topside of the subcompartment have to be defined.

Breadth CL
Breadth of the girder from centreline (SB positive, PS negative).

Underside
Height of the underside of the girder from baseline.

Position cross section
The longitudinal position of the cross-section to be drawn in the output.

Volume cross section m³
The grain level with the heeling angle of the hold in m³ to be drawn in the output.

11.2 Output of grainhold data

Output of grainhold data

Calculate volume and COGs of selected grain compartments
Calculate volume and grainmoments of selected grain compartments
Print cross section of selected grain compartments including void space
Print cross section of selected grain compartments including grainlevel

11.2.1 Calculate volume and COGs of selected grain compartments

This option calculates the table of volumes and centres of gravity, without heeling angles (see figure underneath).

TABLE OF HOLD VOLUME AND COG'S ACCORDING TO IMO Grain Code

Example calculation

Height M	Volume m^3	TCG m	VCG m	LCG m
1.223	200.000	0.000	0.693	45.177
2.204	400.000	0.000	1.205	45.112
3.162	600.000	0.000	1.698	45.082
4.108	800.000	0.000	2.182	45.064
5.048	1000.000	0.000	2.662	45.052
5.984	1200.000	0.000	3.137	45.044
6.915	1400.000	0.000	3.611	45.038
9.020	1600.000	0.000	4.149	45.033
9.564	1648.066	0.000	4.298	45.033

Table of hold volume and COG’s according to IMO Grain code

If the compartment for the grain calculation has been defined including a curved sounding pipe (see Layout), then the output table will also contain a column with the ullage. The ullage is calculated from the top of the defined curved sounding pipe. The top of the sounding pipe will normally be the top of the coaming in case of a grain compartment.

11.2.2 Calculate volume and grainmoments of selected grain compartments

This option calculates the table of grain heeling moments at the prescribed heeling angle (see figure underneath).

TABLE OF HEELING GRAIN MOMENTS ACCORDING TO IMO Grain Code

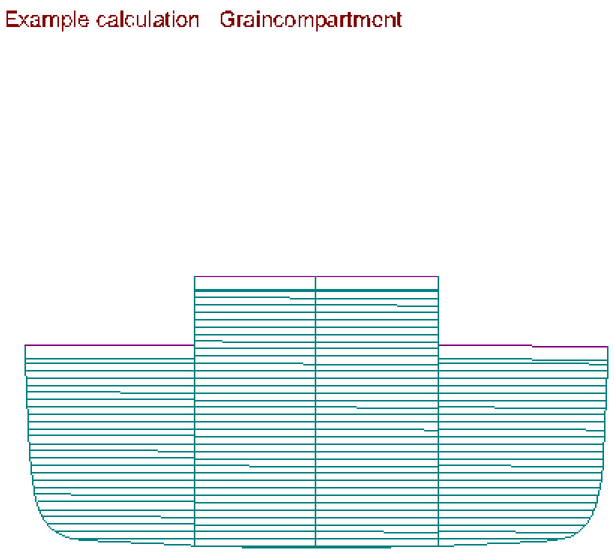
Example calculation

Volume m^3	VCG m	LCG m	Net moment m^4	Grainmoment m^4
200.000	1.739	45.129	1415	1584
400.000	2.283	45.067	2354	2637
600.000	2.728	45.046	2995	3355
800.000	3.012	45.040	3188	3571
1000.000	3.201	45.042	2852	3194
1200.000	3.475	45.040	2291	2566
1400.000	3.846	45.037	1699	1903
1600.000	4.163	45.033	527	591
1648.067	4.278	45.032	411	436

Table of heeling grain moments according to IMO Grain code

11.2.3 Print cross section of selected grain compartments including void space

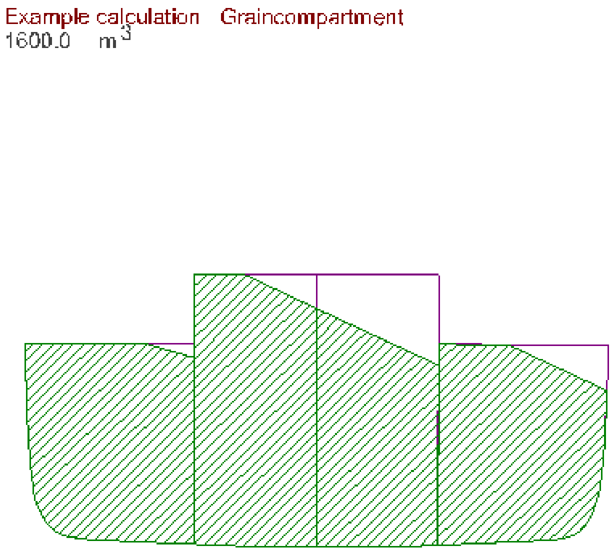
Draws the grain level including void spaces, at zero angle, see the figure below.



Graincompartment 1

11.2.4 Print cross section of selected grain compartments including grainlevel

Draws the grain level with the heeling angle as prescribed by the regulations, see the example below.

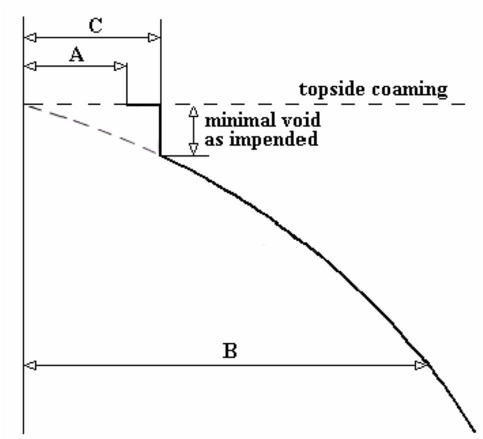


Graincompartment 2

11.3 File management

Backups of all data can be made and restored here. For details, refer to section 2.9 on page 15, Data storage and backups.

11.4 Appendix



Appendix 1

Chapter 12

Tonnage: calculation of gross and net tonnage

This module calculates the gross and net tonnage from seagoing vessels, according to "The 1969 International Convention on Tonnage Measurement of Ships".

Attention

> When using this module it is of paramount importance that compartments for cargo are selected. See description of section 12.3 on this page, Select cargo compartments for more information.

Tonnage calculations

12.1 Definition of superstructures not included in the hullform

At this stage superstructures are defined (primarily deckhouse layers) which are *not included in the hull form*. For example, a poop already included in the hullform (with a hull shape definition module Hulldef of Fairway) *does not have to be* defined in this module, while a bridge which is not already included in the hull form, *should* be defined in this module. For each superstructure, which is assumed to be rectangular, the location of the limiting bulkheads has to be specified by the user. Is the superstructure not of a rectangular shape, the mean values have to be specified. For each superstructure is specified :

- Location of aft and forward bulkhead.
- Location of low and high deck.
- *Half* the breadth.

12.2 Definition general data for net tonnage

For the calculation of the Net Tonnage several specific particulars, as defined in the Tonnage Convention, have to be specified here.

12.3 Select cargo compartments

With this option the cargo compartments selected for the calculation of the net tonnage are being defined and previewed. By entering in this option, the names of the compartments which are already selected for the calculation are listed. The user is able to adjust this list by adding or removing compartments. Adding new compartment(s) can be done by clicking the *List of compartments* button, which opens the full list of available compartments, indicating which of those are already in use. Then, the user can select the compartments to be calculated, and exit the list. The selected compartments list will now be updated. Removing a selected compartment can be done by selecting the compartment to be removed from the list and then pressing the *Remove* button. This compartment will no longer be included in the calculation of section 12.5 on the current page, Calculate and print tonnage calculation (GT and NT).

12.4 Print sideview hull plus superstructures

For checking purposes with this option the hull and the superstructure are drawn schematically in plan view.

12.5 Calculate and print tonnage calculation (GT and NT)

This option calculates and prints the tonnage calculation. For the calculation of Net Tonnage, the volume of the cargo spaces has to be determined (reg. 4 from Annex I from the Tonnage Convention). To enable this module to recognize the appropriate compartments as cargo spaces, those compartments have to be selected in the compartment list of option section 12.3 on the preceding page, Select cargo compartments. All compartments not being part of the cargo space are *not* allowed to be selected. According to the regulations, the volumes of the cargo spaces are being calculated without deduction for construction (permeability = 1).

See underneath for an example of the output of tonnage calculations.

CALCULATION GROSS TONNAGE
Container Feeder Yardo 1056

List of volumes of entire vessel 17 Oct 2013 15:01:06

Description	Aftside m	Frontside m	Underside m	Upperside m	Breadth/2 m	Area m^2	Volume m^3
Closed vessel							15244.680
First deckhouse layer	-3.000	15.000	13.000	15.000	7.000	252.000	504.000
Second layer	0.000	15.000	15.000	17.000	5.000	150.000	300.000
Third layer	0.000	15.000	17.000	19.000	5.000	150.000	300.000
Fourth layer	0.000	15.000	19.000	21.000	5.000	150.000	300.000
Wheelhouse cabin	5.000	15.000	21.000	23.800	5.000	100.000	280.000
Total volume							16928.680

Total volume	16928.680 m^3
VCG of total volume	7.455 m
LCG of total volume	42.676 m
Gross tonnage	4817

Example of the output first page

CALCULATION NET TONNAGE
Container Feeder Yardo 1056

Volumes of cargo spaces 17 Oct 2013 15:01:06

Name compartment	Length m	Breadth m	Height m	Volume m^3
Cargo 1	14.000	10.000	11.933	943.374
Cargo 2	14.000	14.000	9.109	1524.330
Cargo 3	14.000	14.000	9.089	1779.215
Cargo 4	14.000	14.000	9.066	1774.835
Cargo 5	14.000	14.000	12.000	2280.627
Cargo 6	46.000	14.000	12.000	6343.507
Total cargo volume				14645.887

(Length, breadth and height in the table are the maximum values.
All compartments have been calculated with a permeability of 1.00)

Depth	9.000 m
Moulded draught	7.200 m
Number of passengers in cabins with less than 8 berths	0
Number of other passengers	0
Net Tonnage	4149

Example of the output second page

Chapter 13

Maxchain: calculation of maximum allowable anchor handling chain forces

This module calculates the maximum allowable anchor chain forces during anchor handling, according to one of the following rules:

- *Norwegian Maritime Directorate (NMD) from 2007: Guidelines for immediate measures on supply ships and tugs that are used for anchor handling.*
- *Bureau Veritas (BV) amendments January 2014, Pt. D Ch. 14 Sec. 2 Reg. 5: Additional requirements for anchor handling vessels.*
- *Intact Stability Code (IS) 2020, part B, section 2.7.*

13.1 Input of specific ship data

The maximum allowable anchor chain forces are amongst others determined by a number of specific ship data, which are specified at the module for the definition of the hull shape and main dimensions, Hulldef. This module contains a menu option called *Main dimensions and further ship parameters*, which contains a sub menu *Anchor handler characteristics*. With this option an input window appears which contains the folowing parameters:

- The applicable stability regulation. The regulation chosen here is completely independent of any stability criterion chosen to verify the GZ-curve (as discussed in chapter 15 on page 275, Stability criteria for intact stability and damage stability). It may seem overdone to choose regulations twice, however, these choices serve different purposes:
  - In this module, the regulation is chosen that will be used for the maximum allowable forces. The regular GZ-curve of an individual loading condition does not need to be tested against this requirement (and *cannot* be tested against it because, as a rule, there is no actual anchor force included in the loading condition).
  - The stability regulation chosen in the ordinary way will only be used for testing the GZ curve. As a rule, the standard Intact Stability Code (or another standard rule) is chosen for this purpose. The anchor chain force regulations in the regular set (see section 15.2.1.1 on page 278, Seagoing) are rarely used, perhaps as a sample to verify an incidental loading condition (which then should include **all** components of an actual anchor chain force).
- Center of propeller above base, the vertical distance between the propeller center and the base line.
- Maximum chain force (wire tension), for which the winch capacity can be applied, or the holding capacity of the wire stopper if greater. If this figure cannot be determined, an ordinary 'large' value can be taken, for it is not used in the calculation as such. Its only purpose is to assist in determining the scaling factor in the polar plot.
- Wire is restricted at the extremity of the stern roller (only BV-2014 and IS-2020). This 'extremity' applies to the roller breadth. If this parameter is set to 'yes', two lines below the transverse distance between this extremity and CL should be given.
- Length from APP of aft side stern roller. Will speak for itself. Please consider that this terminology is aimed at a stern roller, however, a bow roller can be defined as well, in which case the longitudinal location of the **forward** side of the bow roller should be given here.

- 'Restricting breadth from CL of the stern roller extremity' (BV-2014 & IS-2020) or 'breadth of the stern roller extremity from CL' (NMD-2007), which means in both cases the transverse distance between a 'physical restriction of transverse wire movement' at the roller, and CL.
- Upper edge of stern roller from base line.
- Length from APP of guide pin or winch (BV-2014 & IS-2020). In this line, as well as the line below, primarily the guide pin is intended, however, if this is lacking than the particulars of the winch can be taken.
- Largest breadth from CL of guide pin or winch.
- Minimum dimensionless freeboard at the transom. Is a criterion that only applies to IS-2020, this is on the transom on CL the minimum freeboard divided by the length of the vessel. The regulatory standard is 0.005. The word "transom" has been used here because it is mentioned in the regulations. However, if the longitudinal position of the anchor roller is defined to be at the front of the vessel, this minimum freeboard is measured at the bow. Actually, the minimum freeboard is determined at the extremity of the anchor roller.
- Maximum bollard pull, in ton. Only for IS-2020, may be used for determining angle β depending from α and this bollard pull, according to reg. 2.7.2.1 from the code.

Finally, two remarks on the deck line:

- One of the stability criteria is that the deck line shall not be immerged under load of the chain force. For this it is important that the deck line is defined realistic, see section 7.2.9 on page 185, Deck line for the deck line definition menu.
- Also for the IS-2020 criterion of the minimum dimensionless freeboard at transom — or at the bow, as discussed a few lines above — the deck line must be properly defined, in particular its aftmost (or foremost) point must actually be on CL.

13.2 Main menu of this module

Main menu

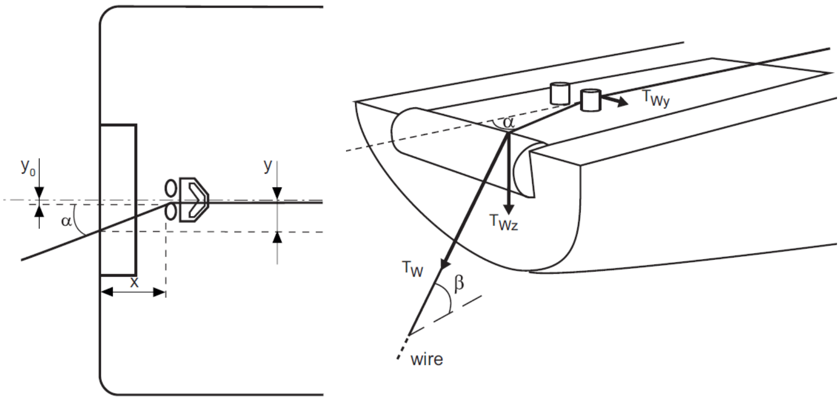
1.	Input for maximum anchor handling chain forces
2.	Define trims to calculate
3.	Calculate and print maximum anchor handling chain forces

13.2.1 Input for maximum anchor handling chain forces

In this menu the following parameters can be defined:

- Start, end and increment for the displacement, in metric tons, for the tables or diagrams to be produced.
- Similarly the start, end and increment for KG', the (virtual) center of gravity, in meter.
- Including graphs. This module will produce output in tables, which are always printed, and optional in diagrams (only NMD-2007 and BV-2014).
- Only for IS-2020: whether angle β should be fixed according to the formulas of reg. 2.7.2.1 of the code, or that a series of βs must be calculated (as can be specified in the last two rows of this menu). The latter choice is not according to the code, but can be used to explicitly investigate the relationship between maximum allowable anchor force and β.
- Whether tables for all individual stability criteria have to be included as well. All requirements contain multiple independent stability criteria, which all have to be complied with. As such, it is not required to determine the maximum anchor forces for each individual criterion, however, for the ship *designer* such a table might be handy, because it may give an indication on the most critical stability aspect, and action may be taken to improve that particular stability property. By the way, it is advised not to include such tables per individual stability criterion with ship delivery documents. Not that there is something wrong with them as such, but there is always the risk that somebody does not recognize the background of such a table, so that it may raise confusion or misunderstanding.
- The initial trim [m]. The calculation can be performed for one or more trims. Multiple trims can be defined at the second option in the main menu of this module (or, altenatively, with <Enter> in this cell). If no trims have been specified there, at this menu option a single trim can be specified. This trim is called initial trim, because with the free-to-trim effect switched on, the actual trim at larger angles of inclination may vary.

Furthermore, a number of regulation-specific parameters can be entered. For NMD-2007 these comprise start, end and increment of the chain angle (in degrees), which is the angle between chain or wire and the longitudinal plane. And for BV-2014 & IS-2020 the end value and increment — the start value is always zero — of the angles α and β, respectively the angle between the wire and the longitudinal, and between the wire and the horizontal plane, as depicted in the figure below.



Definitions of angles, BV-2014 and IS-2020.

13.2.2 Calculate and print maximum anchor handling chain forces

Several output options can be choosen, from which two examples are pasted below. The output may contain:

- For each defined chain angle a table and (possibly) a diagram with on the horizontal axis the KG', on the vertical axis the maximum allowable anchor chain force, while for each desired displacement a curve is plotted which indicates the relation between KG' and anchor chain force.
- For each chain angle a table with the maximum allowable chain force for each individual stability criterion. A short description of the applicable crition is printed in the heading of each table.
- Depending on the applied regulation: a table and (possibly) a diagram with the lowest chain force for seris of chain angles.

MAXIMUM ALLOWABLE ANCHOR CHAIN FORCES (TON)
Mv. Exempli gratia

Regulation: Intact Stability Code 2020, part B, section 2.7.
Initial trim = 0.000 m
Chain angle α (angle between chain and center plane) = 40.000°

<----------Displacement in sea water (ton)---------->

VCG	2000	2100	2200	2300	2400	2500	2600	2700	2800	2900
5.200	160.0	160.0	160.0	160.0	160.0	160.0	160.0	160.0	152.4	140.0
5.300	160.0	160.0	160.0	160.0	160.0	160.0	160.0	156.0	143.7	131.5
5.400	160.0	160.0	160.0	160.0	160.0	160.0	159.0	147.3	135.2	122.4
5.500	160.0	160.0	160.0	160.0	160.0	160.0	150.4	138.8	126.5	107.8
5.600	160.0	160.0	160.0	160.0	160.0	152.9	142.0	130.5	114.9	91.7
5.700	160.0	160.0	160.0	160.0	154.6	144.5	133.7	121.5	98.9	75.6
5.800	160.0	160.0	160.0	155.4	146.4	136.4	125.1	105.8	82.8	59.7
5.900	160.0	160.0	155.2	147.4	138.3	128.2	112.7	89.9	66.9	44.1
6.000	159.6	154.2	147.4	139.5	130.4	119.5	97.0	74.1	51.1	28.8
6.100	152.3	146.7	139.8	131.8	122.0	104.1	81.4	58.4	35.7	14.1
6.200	145.1	139.3	132.3	123.7	111.2	88.9	66.0	43.0	20.7	0.5
6.300	138.2	132.1	124.6	115.6	96.4	73.7	50.7	28.0	6.7	0.0
6.400	131.3	124.7	116.8	103.6	81.5	58.7	35.8	13.6	0.0	0.0

Table of maximum allowable anchor chain forces (IS-2020).

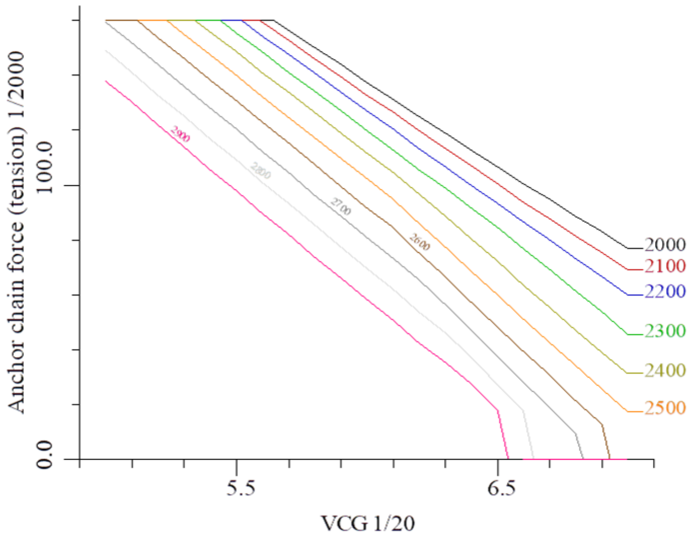


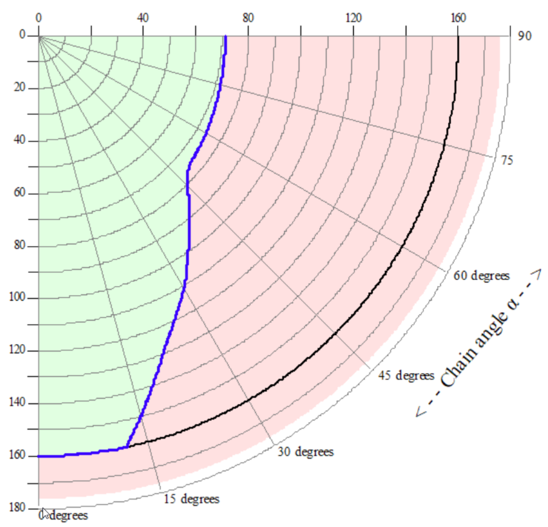
Diagram of maximum allowable anchor chain forces (BV-2014).

13.2.3 Polar diagram with maximum allowable anchor chain forces for a particular loading condition

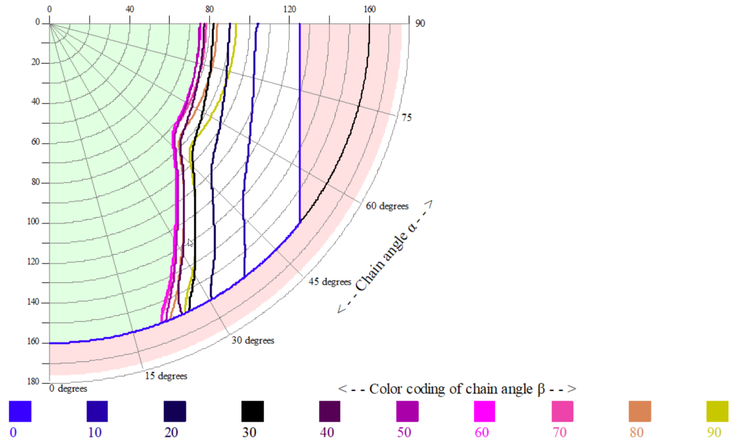
In this module, Maxchain, statical tables and diagrams of maximum anchor chain forces are computed. However, it is also possible to have each intact loading condition — as computed with Loading, section 16.5.4 on page 321, Settings polar diagram anchor handling chain forces — accompanied by a specific polar plot of maximum chain forces If desired, the following approach is adopted:

- The anhor-handling input data from Hulldef (see the first section of this chapter) are used.
- Chain angles and VCGs as specified for this Maxchain module play no role. The polar diagram is always calculated from 0°to 90°, based on the loading condition where that polar plot belongs to.
- Anchor chain forces should not be included in weight items of the loading condition, for it is the intention of this PIAS module to determine the maximum anchor chain force automatically.

For each loading condition such a diagram can be produced, where radial the maximum allowable anchor chain forces (in ton) can be read for different anchor chain angles. In a calculation according to IS-2020 the output is as shown in the figure below, here is only a single line that indicates the separation between allowed (green) and prohibited area (red) as a function of α The second polar diagram shows the output of a calculation according to BV-2014, this contains per β a line of maximum allowable anchor chain force.



Polar plot of maximum allowable anchor chain forces, according to IS-2020.



Polar plot of maximum allowable anchor chain forces, according to BV-2014.

Chapter 14

Wind heeling moments

The effect of the wind on the stability is manifest in several places, and also the input has been spread over several modules. This is no peculiarity of PIAS, but is simply the nature of wind heeling moments. To give you still a proper idea of the coherence, this is discussed in this chapter.

14.1 Input data for the wind heeling moments computations

Part of the variety at wind heeling computations is caused by the fact that there can be multiple input data, for example several windage areas — like with various loadings — and several wind pressures, for example because another pressure is used at the stability criteria for intact stability than at the damage stability criteria. You can find the exact nature of the input data, and their location, in the table below:

Wind contoru
: The wind contour belongs in principle to the ships' input data, and is therefore entered in Hulldef, see section 7.2.6 on page 182, Wind contour. It is also possible to enter several wind contours, for example for multiple types of loading on deck.

Wind contour with containers
: For standard cases, like for a stability book, several wind contours can be defined for various complete layers of containers. The trouble with this is that when a layer has not been entirely filled with containers, the windage of that entire layer is still taken into account, whereas the actual windage is obviously smaller. When a special loading module of Loading is used, like the container module (see chapter 17 on page 324, Loading tools), for any loading condition exactly the actual wind contour is constructed by laying the ship's contour (without cargo, as entered in Hulldef, see above) and the loaded container on top of each other and taking the common windage of them.

Wind data
: These are also given in Hulldef, see section 7.2.7 on page 183, Wind data sets, and regard in particular the wind pressures. Several collections of these data can be recorded, so that all kinds of combinations are possible between wind contour and wind pressures at the various stability calculations

Specific wind pressure at a stability requirement
: Although the wind contour and the wind data in principle provide sufficient data to be able to carry out further computations, some special input can take place at the stability requirements. One of them is the *specific wind pressure*, as discussed in section 15.5.4.1 on page 299, Wind lever. That seems a bit redundant, because one can give any wind pressure on the wind data from Hulldef, so why then even here? The reason is that stability requirements may, for example at a damage stability criterion, use another wind pressure than the 'standard' wind pressure which applies to intact stability. One should be very well aware of this aspect — a mistake is easily made — and even more so when creating standard criteria, because with this specific wind pressure it can directly be defined correctly. In short, this alternative, specific wind pressure is indeed redundant, but has been introduced to reduce the risk of errors and to enhance the user experience.

Multiplier on the wind heeling lever
: In addition, a multiplier for the set wind heeling lever can be specified. That also seems a bit redundant, because why not in first instance already enter, in the wind data section, a wind pressure which has already been multiplied by this factor? This would indeed be an option — it would make the program internally even less complex — but occasional standard stability criteria exist where such multiplication factors are

applied, and it is most clear to show and enter this factor in its most basic form. And by being clear the risk of mistakes is reduced.

Wind heeling levers entered by the user

It is by far the most practical to have the wind heeling levers calculated by PIAS. But only once in a while there are, however, wind heeling levers available from another source, for example from CFD calculations or model tests. In that case you can give these wind heeling levers yourself in Hulldef, at the last option of the wind data, see section 7.2.7 on page 183, Wind data sets.

14.2 Where do wind moments exert their effects?

- Tables of wind heeling levers can be printed with Hydrotables, see section 10.2.7 on page 255, Wind heeling moment tables.
- At the calculation of intact stability of a loading condition. The wind contour that is used there (as well for the calculation as possibly for the pictures) intrinsically belongs to that loading condition, and can (therefore) be entered in the upper bar of the weight items window of the loading condition, with the [Settings] option, see section 16.2.1.3 on page 310, Settings per loading condition. The wind pressure that is used there intrinsically belongs to the stability requirement, and can(therefore) be entered at the Stability criteria, see section 15.5.4.1 on page 299, Wind lever.
- Tables and diagrams of maximum allowable KG in intact condition can be calculated for several wind contours, see section 10.2.8 on page 255, Maximum VCG' intact tables. When you select N wind contours then there will be made N tables or diagrams, so you can take that table or diagram that applies to the future situation.
- Tables and diagrams of maximum allowable VCG in damage condition function as for the wind similarly as the intact condition.

14.3 Recommended working sequence

There is no obligatory working or definition sequence with regard to the wind related issues. All input or options can be modified later on. But in the event of a project where nothing has been laid down yet, the clearest definition sequence is:

1. Make one or more wind contours in Hulldef.
2. Define the wind pressure(s) in Hulldef.
3. If desired, a table of wind heeling moments can already be produced by Hydrotables, however, it is not required to do so, because this table is neither saved nor further used.
4. Define stability criteria.
5. Carry out the various computations.

Chapter 15

Stability criteria for intact stability and damage stability

In this chapter the foundations and tools for the configuration of stability criteria for intact stability and damage stability are discussed. The particular menu option for this task can be found in the Config module, which is also accessible through the [Setup]→[Project setup] function, top-left in (almost) every PIAS window.

- *The core is a **set of stability criteria**. Such a set can be valid for intact stability or damage stability.*
- *Multiples of such sets are supported, in the first place to support different criteria sets for intact stability and damage stability, but also to be able to toggle quickly between multiple types of criteria (for example in case of multiple navigation areas, where different stability criteria may apply).*
- *A set of criteria contains multiple individual criteria. Such an individual criterion has a simple structure (e.↩ g. 'minimum metacentric height' or 'area under the GZ curve within a range of 20 degrees') and can be dependant on certain **parameters** (such as 'a minimum metacentric height of **30 cm**'). Such a parameter can either be a number, or a **variable** from which the numerical value is determined dynamically by the program (such as the concept 'top of the GZ curve').*
- *From the set of stability criteria quite some standard sets of stability requirements have been preprogrammed (such as the 'Intact Stability Code'), however, this is merely a kind of service, because in the end the definition is captured within the individuals criteria, just as each user is able to do manually.*

Finally, four remarks are worth to be given before going into detail:

- *It is recommended to generate and check intermediate results in case of unclear or unexpected results. This can be set in the fourth column of the main menu of sets of stability criteria, see* *section 15.1 on the following page, Manipulating and selecting sets of stability criteria.*
- *The major part of this chapter deals with the setup of stability criteria, however, it is recommended also to visit the last two paragraphs. One contains a number of FAQs on specific stability questions, this can be found in* *section 15.6 on page 301, Answers to frequently asked questions on stability assessments.* *The other deals with the availability of criteria, and also contains a number of disclaimer remarks, please see* *section 15.7 on page 303, On the various criteria and parameters* *for those subjects.*
- *Non-watertight openings are fully taken into account while assessing the GZ curve. That does not separately has to be turned on or off.*
- *The words 'requirement' and 'criterion' are used in a mixed fashion, they have the same meaning (here).*

Attention

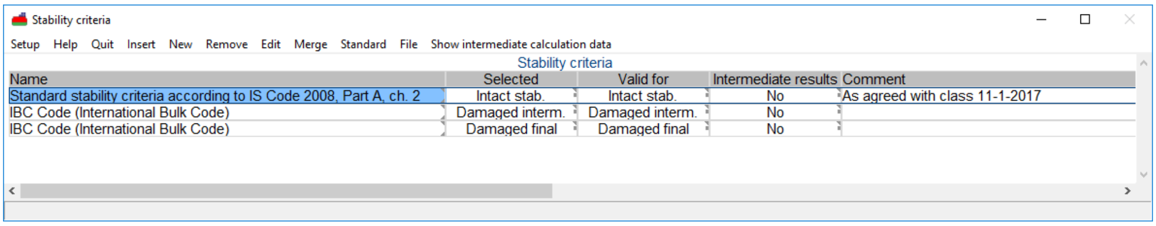
Especially read the example described in the attention of section 15.2.2 on page 282, Variants for standard sets of stability requirements about how to handle the creation of standard stability criteria.

Attention

A database is available which contains stability criteria that are not included as standard in PIAS. If one downloads this database than it is recommended to read the, in the database available, 'Readme.docx' file first. This file contains information regarding importing the '.req' files. Press `this link to download`[1] the non standard included stability criteria. There is a document available with information regarding this database but that is only transmitted on request.

15.1 Manipulating and selecting sets of stability criteria

Choosing the stability criteria definition menu opens a window with the already defined sets of criteria, which will look like this:



Set of stability requirements.

With the <Enter> key (or mouse double click) one goes one level deeper into the stability criterion, where all specific parameters can be given. That menu is discussed in section 15.3 on page 287, Manipulating individual criteria.

The columns have the following meaning:

Name
The names of the different sets of criteria can be defined and altered by the user. In the example, one name appears twice; these are sets of criteria that are applicable to different calculations, but originate from the same regulations. The user could also define different names here.

Selected
The different sets of criteria can be selected individually. This enables fast switching between different sets of criteria for the output of calculations. Upon selecting the cell in this column a popup menu will open up (from which an example is depicted below) which enables selecting a criterion set, for the types of calculations as defined in the column 'Applicable for'.

Valid for
This column shows the type of calculations to which a set of criteria applies. Changing the applicability of a set of calculations can be done through a popup menu, from which an example is depicted in the second figure below.

Intermediate results
In this column, it can be specified whether the intermediate results of the calculated stability criteria should be generated. If these intermediate results are generated then these will be of the last made calculation. With each new calculation, the intermediate results of the previous one are removed.

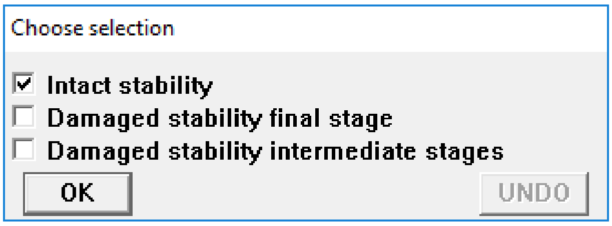
This setting is only applicable within Loading when calculating intact and damaged stability and then exclusively for the following menus, section 16.2 on page 305, Loading conditions and section 16.2.1 on page 307, Define/edit weight items. This setting is also applicable for Hydrotables if in Hulldef it has been chosen to make use of a paragraph 7.2.1.12.3 on page 171, Drafts-displacements table.

The intermediate results can be viewed via the menu bar option [Show intermediate calculation data].

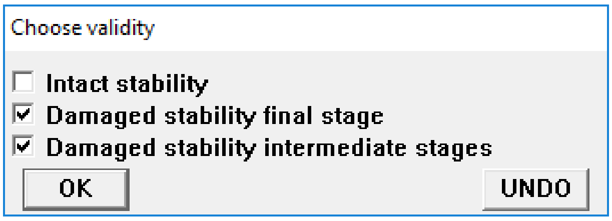
The intermediate results are written to the text file projectname.str that comes in the project folder, and can be opened with any text editor.

Comment
In this column the user can define additional comments.

[1] `https://www.sarc.nl/wp-content/uploads/2018/05/PIAS-stability-criteria.zip`



Selecting sets of criteria.



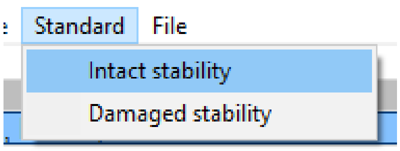
Choose applicability of criteria.

Furthermore, this menu contains a number of menu bar functions:

- With [Merge] two sets of criteria can be merged. This can sometimes prove to be handy when manipulating criteria sets, or in combination with import or export actions. The option [Merge] only works when one makes a copy of a set of criteria, through 'Copy a row entirely' see section 4.4 on page 39, Copy, paste etc., which thereafter can be merged in any other set of criteria than itself.
- With option [Show intermediate calculation data] the intermediate results file will be opened in the standard text editor.
- With [Standard], standard stability criteria can be created, this is further discussed in section 15.2 on the current page, Select standard stability criteria.
- [File] has two sub options: import and export, which can be applied to read or write a set of criteria from or to another PIAS criteria file. This option can, for example, be used to exchange a non-standard set of criteria with another project. When exporting, the selected criteria set — the one where the text cursor resides — is added to the specified file. On import a list of in the import file present criteria sets is displayed. From this list one can choose one criteria set which should actually be imported.

15.2 Select standard stability criteria

This function - activated by [Standard] in the window of criteria sets - will add a set of criteria to the already existing sets. Through popup menus a choice is made from the predefined sets of standard criteria for intact or damaged stability. First the choice for intact or damage stability criteria is made, as illustated in the figure below. Subsequenlty, depending on the choice, a popup menu with the available predefined sets of intact or damaged stability opens up, this will be discussed in the next sections.



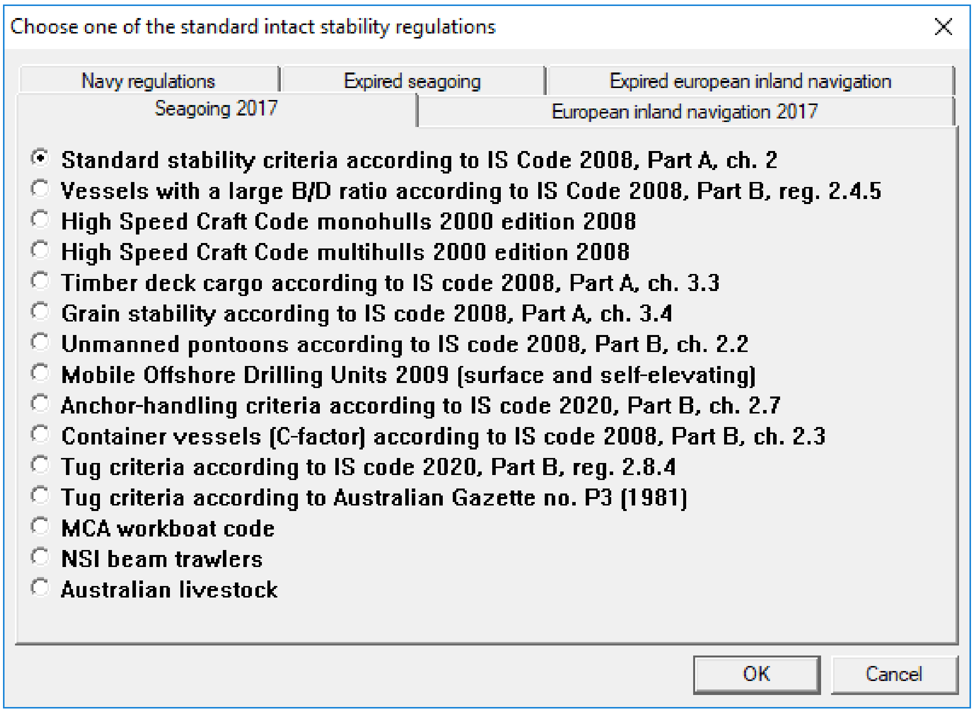
Choice for intact or damage stability criteria.

15.2.1 Standard stability criteria intact stability

A number of standard stability requirements have been preprogrammed. Below is a list with references to the chapters with short references to the sources of the requirements in that chapter.

- Seagoing
- European inland navigation
- Navy regulations
- Expired seagoing
- Expired european inland navigation

15.2.1.1 Seagoing



Standard criteria for intact stability, seagoing.

Standard stability criteria according to IS Code 2008, Part A, ch.2

Intact Stability Code 2008, almost identical to its predecessors IMO A.749 and IMO A.562.

- These regulations contain a subcriterion, stating that the statical angle due to wind may not be larger than 80% of the angle of deck immersion. Traditionally, and also today, PIAS determines this angle at Lpp/2. However, in 2006 it appeared that a regulatory body may require that the effect of trim is included. In that case the user will have to apply the variable 'Angle of deck edge immersion' instead of 'Angle of deck immersion at L/2' in this criterion.
- The Intact Stability Code also includes the so-called weather criterion, for which the roll data of the ship have to be specified, see for the definition menu section 7.2.1.2 on page 167, Roll data (for Intact Stability Code weather criterion).

Vessels with a large B/D ratio according to IS Code 2008, Part B, reg. 2.4.5

This regulation is almost identical to the earlier mentioned *Intact Stability Code standard stability criteria* regulation but then with changes for the so-called 'supply vessels', meaning that the minimum angle of GZ-max will then be decreased to 15°, which will be compensated by a larger area under the GZ curve up to that angle. In accordance with reg. 2.4.5.2 of the IS Code.

High Speed Craft Code monohulls 2000 edition 2008

International Code of Safety for High-Speed Craft (2000), 2008 Edition, annex 8.

High Speed Craft Code multihulls 2000 edition 2008

International Code of Safety for High-Speed Craft (2000), 2008 Edition, annex 7.

Timber deck cargo according to IS Code 2008, Part A, ch. 3.3

Intact Stability (IS) Code 2008, Part A, chapter 3.3.

Grain stability according to IS Code 2008, Part A, ch. 3.4
Intact Stability (IS) Code 2008, Part A, chapter 3.4. Which is practiclly similar to, International Code for the Safe Carriage of Grain in Bulk, MSC.23(59), 1 January 1994. According to this *Code* the angles 12°and 40°should explicitly be included into the calculations. So, please take care that these angles are part of the range of heeling angles, as specified in Config.

Unmanned pontoons according to IS Code 2008, Part B, ch. 2.2
IS Code 2008, part B, chapter 2.2 or IMO A749(18) chapter 4.7, 4 nov 1993: Code on intact stability for all types of ships covered by IMO instruments.

Mobile Offshore Drilling Units 2009 (surface and self-elevating)
Code for the Construction and Equipment of Mobile Offshore Drilling Units, 2009 (MODU Code). Only surface and self-elevating.

Anchor handling criteria according to IS Code 2020, Part B, ch. 2.7
These Intact Stability Code 2020 criteria are similar to the 'NMD 2007 & BV2014 anchor handling criteria', seeExpired seagoing, albeit with the following two additions:

- The residual area between the righting lever curve and the heeling lever curve should not be less than 0.070 mrad.
- The maximum residual righting lever GZ between the righting lever curve and the heeling lever curve should be at least 0.2 meter.
- The IS Code also contains a requirement on minimum required freeboard aft (of 0.005L). This requirement is not included here, because it is no stability requirement, but a draft requirement instead, which can be given in the menu of main dimensions and further ship parameters (see section 7.2.1.4 on page 168, Draft marks and allowable maximum and minimum drafts.

Just as with the NMD and BV criteria, a total load case (that is ship & anchor chain force) is tested here. With just the application of these criteria in a loading condition the *maximum permissible anchor chain force* is not determined, for that purpose module Maxchain, should be used.

Container vessels (C-factor) according to IS Code 2008, Part B, ch. 2.3
The criteria from the 'Intact Stability Code 2008' for container vessels greater than a 100 meter.

Tug criteria according to IS Code 2020, Part B, reg. 2.8.4
For 'Intact Stability Code 2020' tug criteria one or more choices might be applicable:

- Intact Stability Code 2020 (MSC 97-22-Add.1, 2.8.4.2 (self tripping))
- Intact Stability Code 2020 (MSC 97-22-Add.1, 2.8.4.3 (tow tripping))

Tug criteria according to Australian Gazette no. P3 (1981)
For 'Australian Gazette no. P3 (1981)' tug criteria one or more choices might be applicable:

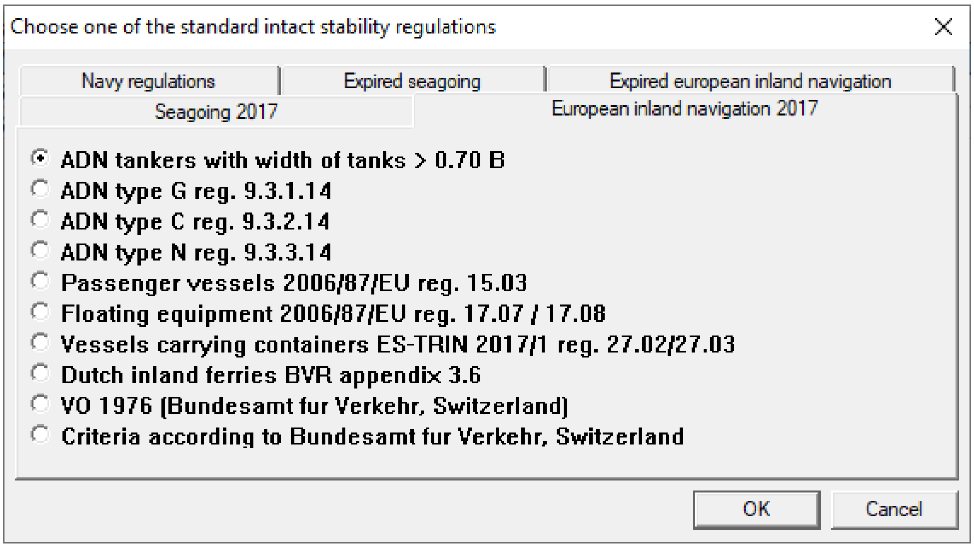
- Australian Gazette no. P3 (1981), navigation area A, B or C
- Australian Gazette no. P3 (1981), navigation area D or E

MCA workboat Code
The Workboat Code Industry Working Group Technical Standard, June 2014, MS 183/01/23.

NSI beam trawlers
According to BadS 124/1977: 20% added to the standard criteria. Netherlands Shipping Inspection (NSI).

Australian livestock
Marine Order 43 (Cargo and cargo handling — livestock) 2006.

15.2.1.2 European inland navigation



Standard criteria for intact stability, european inland navigation.

ADN tankers with width of tanks > 0.70 B
ADN, European Agreement concerning the International Carriage of Dangerous Goods by Inland Waterways, only applicable for:

- Type C tankers, reg. 9.3.2.14.2.
- Type N tankers, reg. 9.3.3.14.2.

ADN type G reg. 9.3.1.14
ADN, European Agreement concerning the International Carriage of Dangerous Goods by Inland Waterways, reg. 9.3.1.14.

ADN type C reg. 9.3.2.14
ADN, European Agreement concerning the International Carriage of Dangerous Goods by Inland Waterways, reg. 9.3.2.14.

ADN type N reg. 9.3.3.14
ADN, European Agreement concerning the International Carriage of Dangerous Goods by Inland Waterways, reg. 9.3.3.14.

Passenger vessels ES-TRIN 19.03
European Standard laying down Technical Requirements for Inland Navigation vessels, ES-TRIN 19.03.

Floating equipment ES-TRIN 22.07/22.08
European Standard laying down Technical Requirements for Inland Navigation vessels, ES-TRIN 22.07 and 22.08.

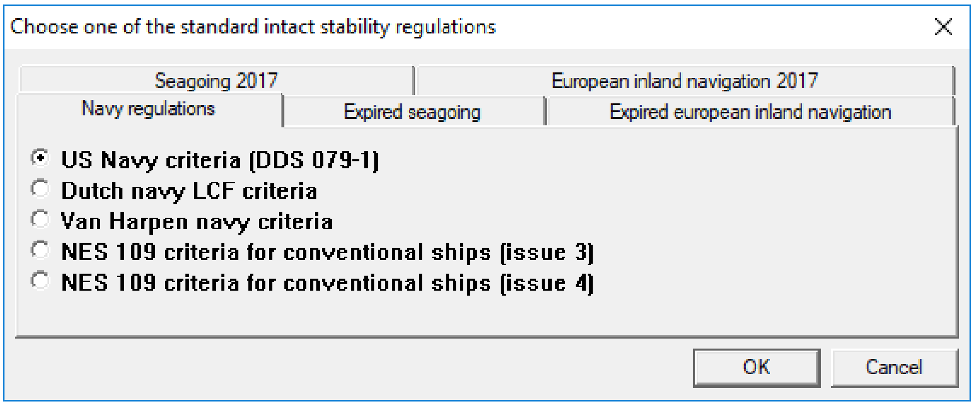
Vessels carrying containers ES-TRIN 27.02/27.03
European Standard laying down Technical Requirements for Inland Navigation vessels, ES-TRIN 27.02 and 27.03.

Dutch inland ferries BVR appendix 3.6
BVR (Binnenvaartregeling) 2009, Appendix 3.6.

VO 1976 (Bundesamt fur Verkehr, Switzerland)
Intact criteria according Bundesamt fur verkehr, Switzerland, Verordnung 1976.

Criteria according to Bundesamt fur Verkehr, Switzerland
Intact criteria according Bundesamt fur Verkehr, Switzerland.

15.2.1.3 Navy regulations



Standard criteria for intact stability navy regulations.

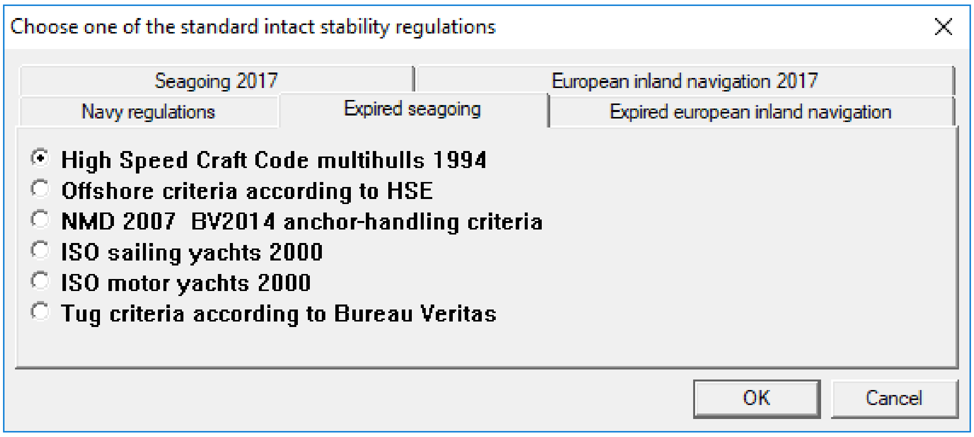
US Navy criteria (DDS 079-1)
Specific criteria for American naval vessels.

Dutch navy LCF criteria
Specific criteria for some vessels of the Royal Netherlands Navy.

Van Harpen navy criteria
Stability criteria for the Royal Netherlands Navy according to Report Nr. 21183/21021/SB of the Ministry of Defence.

NES 109 criteria for conventional ships (either 'issue 3' or 'issue 4').
NES 109 issue 3 and 4. These criteria are very similar to those of BV, 'Rules for the Classification of Naval Ships, Part B, chapter 3, section 2'. The differences are mainly in the applied criterion values.

15.2.1.4 Expired seagoing



Standard criteria for intact stability, expired seagoing.

High Speed Craft Code multihulls 1994
International Code of Safety for High-Speed Craft. MSC.36(63), 20 May 1994.

Offshore criteria according to HSE
Criteria for offshore vessels according to HSE (DoE) and NMD.

NMD 2007 & BV2014 anchor handling criteria
These criteria evaluate a total load case (which is ship & anchor chain force) for compliance with NMD's *Guidelines for immediate measures on supply ships and tugs that are used for anchor handling (2007)*. The anchor chain force should be defined acting at CL, because the NMD criteria require a certain ratio between

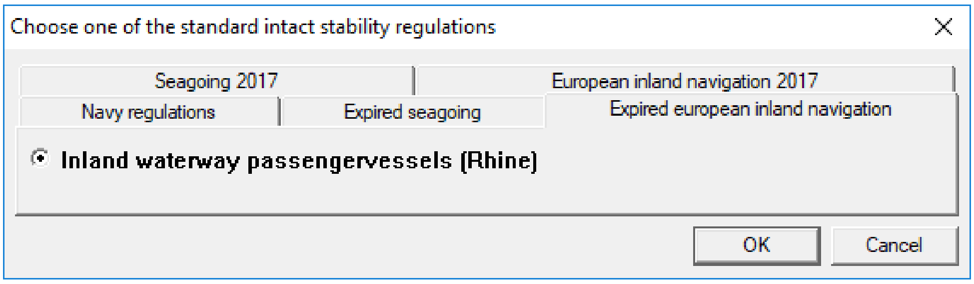
the lever at the top of the GZ curve and the lever at the intersection between righting and heeling levers. In order for this ratio to appear, one should not include the heeling moment by placing the anchor chain force off centerline. So the chain force should be assumed to exert on the real chain position longitudinally and vertically, but always at center line. By the way, with only the application of these criteria the *maximum permissible anchor chain force* is not determined, for that purpose module Maxchain, should be used.

ISO sailing yachts 2000
ISO/DIS 12217, stability and buoyancy assessment and categorization, Part 2: Sailing boats of hull length greater than or equal to 6 m, 2000-10-05.

ISO motor yachts 2000
ISO/DIS 12217, stability and buoyancy assessment and categorization, Part 1: Non-sailing boats of 6 m length of hull and over, 2000-10-05.

Tug criteria according to Bureau Veritas
The tug criteria, of Bureau Veritas (D.14.2, reg. 2.2.2), have been replaced with the 'Intact stability Code 2020' criteria for self and tow tripping. **In the Bureau Veritas regulations this can be found under, Part E, chapter 1, section 2, regulation 2.**

15.2.1.5 Expired european inland navigation



Standard criteria for intact stability, expired european inland navigation.

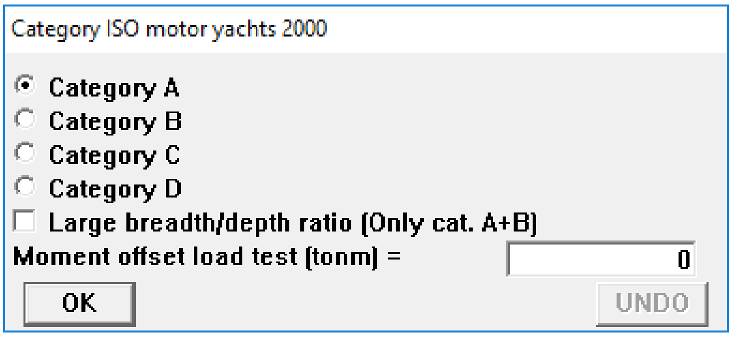
Inland waterway passengervessels (Rhine)
According to Binnenschepenbesluit, Stbl. 466, or, alternatively, according to Bundesamt für Verkehr, Switzerland

15.2.2 Variants for standard sets of stability requirements

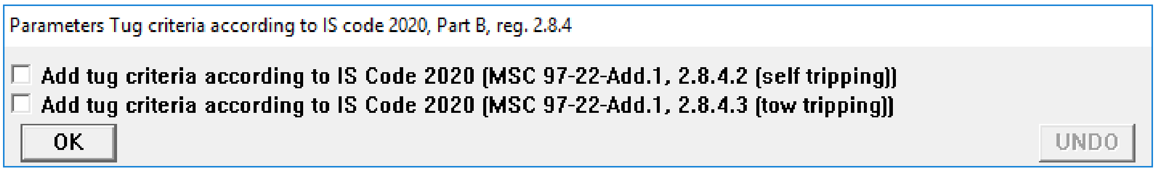
Attention

In the pre-2017 versions of PIAS, during the creation of standard intact stability criteria one would be asked the question to add other criteria, like tugs criteria, NSI interpretation and grain criteria or Australian livestock moment, to the to be created stability criteria. These choices are now integrated as standard criteria. When one for example wants to create intact criteria with grain criteria now has to create two standard sets of stability criteria, namely 'Standard stability criteria according to IS Code 2008, Part A, ch. 2' and 'Grain stability according to IS Code 2008, Part A, ch. 3.4' and then merge together them with the option [Merge], as discussed in section 15.1 on page 276, Manipulating and selecting sets of stability criteria. From now on this the way to handle standard stability criteria where this is applicable.

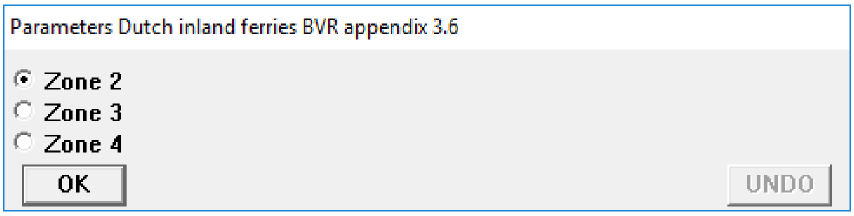
For some standard sets of stability criteria, additional choices must be made. If so, a popup menu appears after selecting the standard set of criteria. Most of the options in these popup menus determine whether or not additional criteria are applicable and sometimes a value must be entered for a variable. For a description of the variables reference is made to section 15.3 on page 287, Manipulating individual criteria and to section 15.5.4 on page 299, Defining heeling moments to be accounted for. For other sets of standard criteria, an additional choice can be made, as shown in the example below. For the determination of such choices and requested values, reference is made to the relevant regulations.



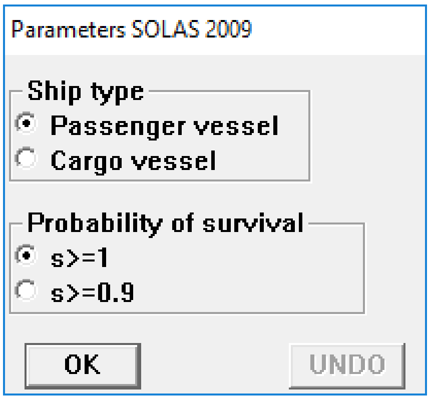
Parameters ISO motor yachts.



Supplementary parameters tugs.



Supplementary parameters Dutch inland ferries BVR appendix 3.6.



Supplementary parameters SOLAS 2009.

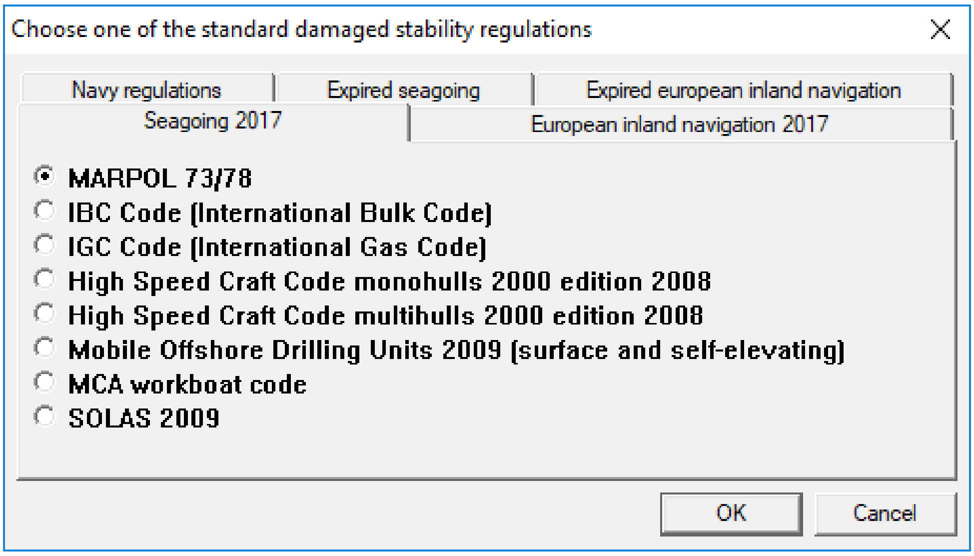
15.2.3 Standard stability criteria damaged stability

These are, similar to the intact stability criteria, also presented in a popup window. Below is a reference list to the corresponding chapter which contains a table with short reference to the source of the regulations.

- Seagoing
- European inland navigation

- Navy regulations
- Expired seagoing
- Expired european inland navigation

15.2.3.1 Seagoing



Sets of predefined criteria for damage stability, seagoing.

MARPOL 73/78
Marpol consolidated edition 2006.

IBC Code (International Bulk Code)
International Bulk Code 1998.

IGC Code (International Gas Code)
International Gas Code.

High Speed Craft Code monohulls 2000 edition 2008
International Code of Safety for High-Speed Craft (2000), 2008 Edition, annex 8. Note that the range can be reduced to 10 degrees if the area under the curve is increased by a factor of 15/range, see paragraph 2.1.1. Also note that these criteria make use of evacuation points which can be defined in section 7.2.8 on page 184, Openings.

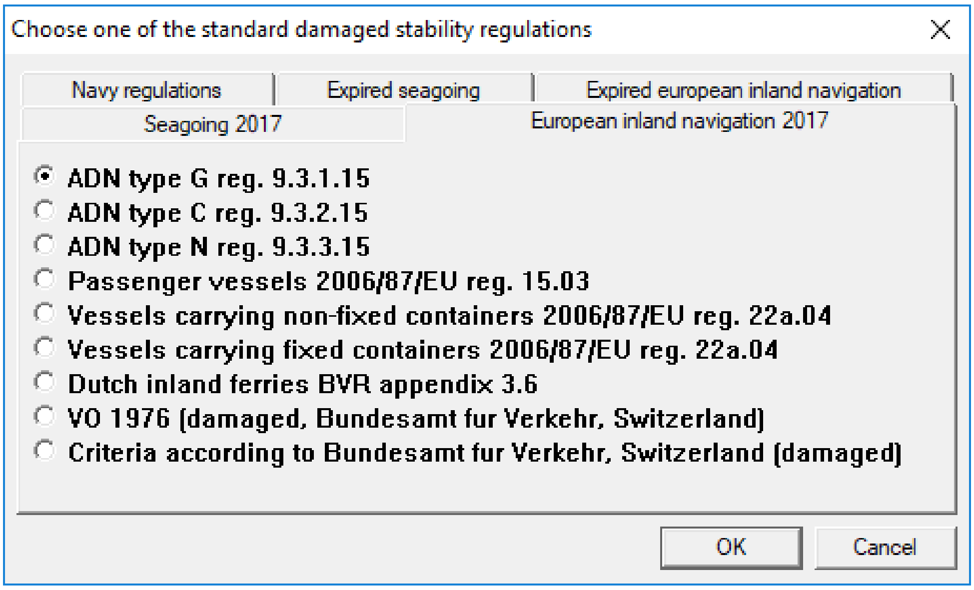
High Speed Craft Code multihulls 2000 edition 2008
International Code of Safety for High-Speed Craft (2000), 2008 Edition, annex 7.

Mobile Offshore Drilling Units 2009 (surface and self-elevating)
Criteria according MODU Code 2009, only surface and self-elevating. In article 3.4.1 it is stated that the range of stability has to be determined over the positive stability but that range is determined **without** the influence of the downflooding angle. In PIAS the downflooding angle is **always** used and also in this exceptional situation the downflooding angle has been taken into account. This is therefore in **deviation** with the stated regulations.

MCA workboat Code
The Workboat Code Industry Working Group Technical Standard, June 2014, MS 183/01/23.

SOLAS 2009
SOLAS 2009, Consolidated text 2014.

15.2.3.2 European inland navigation



Sets of predefined criteria for damage stability, european inland navigation.

ADN type G reg. 9.3.1.15
ADN, European Agreement concerning the International Carriage of Dangerous Goods by Inland Waterways, reg. 9.3.1.15.

ADN type C reg. 9.3.2.15
ADN, European Agreement concerning the International Carriage of Dangerous Goods by Inland Waterways, reg. 9.3.2.15.

ADN type N reg. 9.3.3.15
ADN, European Agreement concerning the International Carriage of Dangerous Goods by Inland Waterways, reg. 9.3.3.15.

ADN container vessels (non-fixed containers) reg. 9.1.0.95
ADN, European Agreement concerning the International Carriage of Dangerous Goods by Inland Waterways, reg. 9.1.0.95.

ADN container vessels (fixed containers) reg. 9.1.0.95
ADN, European Agreement concerning the International Carriage of Dangerous Goods by Inland Waterways, reg. 9.1.0.95.

Passenger vessels ES-TRIN 19.03
European Standard laying down Technical Requirements for Inland Navigation vessels, ES-TRIN 19.03.

Vessels longer than 110m with non-fixed containers ES-TRIN 28.03
European Standard laying down Technical Requirements for Inland Navigation vessels, ES-TRIN 28.03.

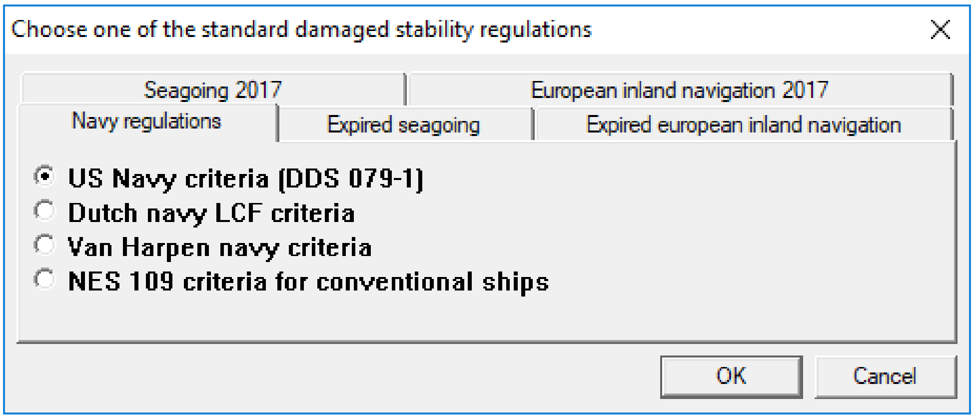
Vessels longer than 110m ES-TRIN 28.03
European Standard laying down Technical Requirements for Inland Navigation vessels, ES-TRIN 28.03.

Dutch inland ferries BVR appendix 3.6
BVR (Binnenvaartregeling) 2009, appendix 3.6.

VO 1976 (damaged, Bundesamt fur Verkehr, Switzerland)
Damaged criteria according Bundesamt fur verkehr, Switzerland, Verordnung 1976.

Criteria according to Bundesamt fur Verkehr, Switzerland (damaged)
Damaged criteria according Bundesamt fur Verkehr, Switzerland.

15.2.3.3 Navy regulations



Sets of predefined criteria for damage stability, navy regulations.

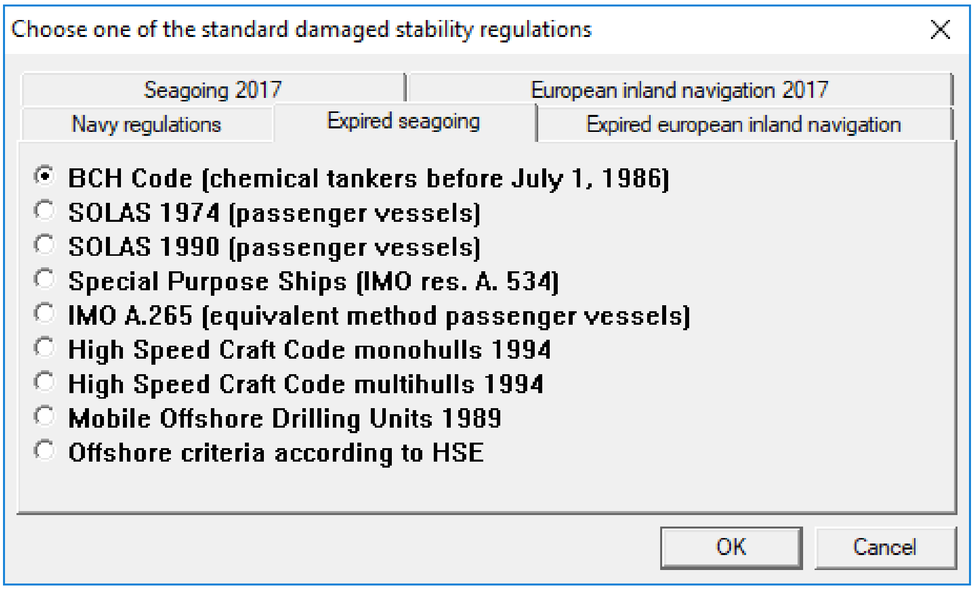
US Navy criteria (DDS 079-1)
Specific criteria for American naval vessels.

Dutch navy LCF criteria
Specific criteria for Dutch naval vessels.

Van Harpen navy criteria
Stability criteria for the Royal Netherlands Navy according to Report Nr. 21183/21021/SB of Dutch Ministry of Defence.

NES 109 criteria for conventional ships
NES 109. These criteria are very similar to those of BV, 'Rules for the Classification of Naval Ships, Part B, chapter 3, section 3'. The differences are mainly in the applied criterion values.

15.2.3.4 Expired seagoing



Sets of predefined criteria for damage stability, expired seagoing.

BCH Code (chemical tankers before July 1, 1986)
Code for the Construction and Equipment of Ships Carrying Dangerous Chemicals in Bulk, 1993.

SOLAS 1974 (passenger vessels)
SOLAS 1974.

SOLAS 1990 (passenger vessels)
SOLAS 1990.

Special Purpose Ships (IMO res. A. 534)
Special Purpose Ships Code IMO A.534.

IMO A.265 (equivalent method passenger vessels)
Regulation 5 of IMO A.265.

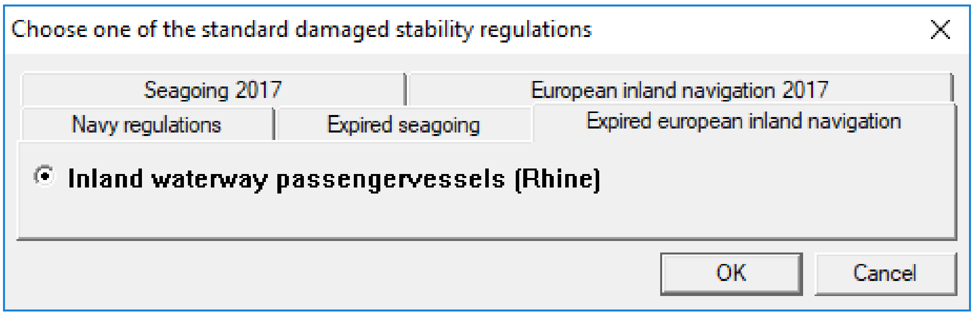
High Speed Craft Code monohulls 1994
Criteria according to HSC Code monohull passenger vessels 1994.

High Speed Craft Code multihulls 1994
Criteria according to HSC Code multihull passenger vessels 1994.

Mobile Offshore Drilling Units 1989
Criteria according to MODU Code 1989.

Offshore criteria according to HSE
Criteria for offshore vessels according to HSE (DoE) and NMD.

15.2.3.5 Expired european inland navigation

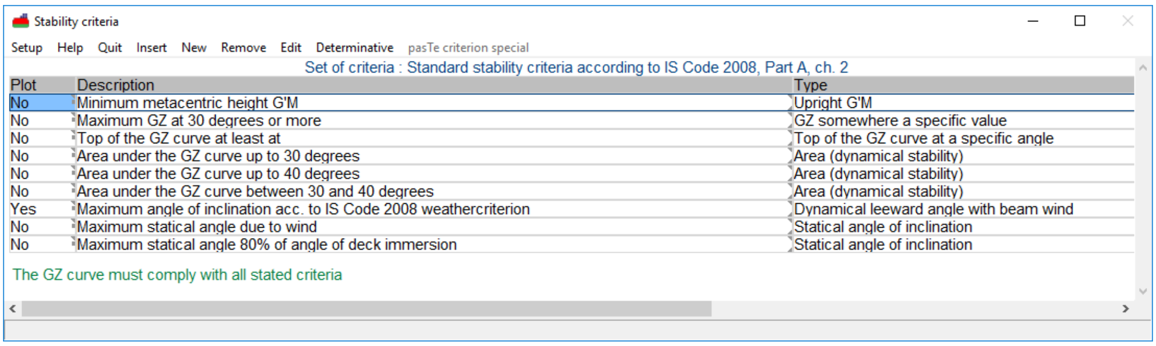


Sets of predefined criteria for damage stability, expired european inland navigation.

Inland waterway passengervessels (Rhine)
According to Dutch Binnenschepenbesluit, Stbl. 466, or, alternatively according to Bundesamt für Verkehr, Switserland.

15.3 Manipulating individual criteria

A set of criteria can be manipulated by selecting the set from the menu, section 15.1 on page 276, Manipulating and selecting sets of stability criteria. This will open a menu with the individual criteria for this set:



Defining stability criteria.

Columns of the stability criteria definition menu

1	Plot
2	Description
3	Types of basic criteria
5	Valid up to statical angle
6	Determinative (toolbar function)
7	pasTe criterion special (toolbar function)

15.3.1 Plot

For every individual criterion the user may choose to draw a GZ curve in the output. This is only useful for some types of criteria, such as a certain area under the curve or a static angle of inclination due to a (wind or other) heeling moment. In the tables of maximum allowable KG' values, no curves are drawn at all. If this option is not selected for any criterion, a GZ curve is still drawn in the output for the stability calculations for a loading condition, only without any hatching of areas etc.

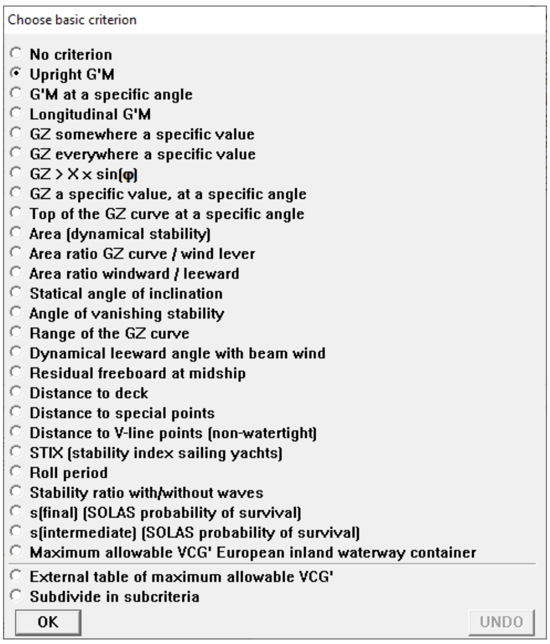
15.3.2 Description

For each criterion a name can be given. This will be used in the output for identification purposes.

15.3.3 Types of basic criteria

Attention

The criteria as listed below include some where the **other side** of the GZ curve (i.e. left of the origin) play a role, such as 'Area ratio windward / leeward' en 'Dynamical leeward angle with beam wind'. If the computation setting of section 5.1.7 on page 44, Calculate intact stability etc. with a heeling to (and its relative for damage stability) is 'Portside and starboard', then the complete GZ curve from PS to SB is computed and available, so those stability criteria can be evaluated complete and correct. However, with any other setting the stability will be computed to just a single side, and will be mirrored to the other side — after all, no other information is available in this case. In case of asymmetrical hull forms or damage cases that might be less accurate than a full computation.



Basic types of criteria.

The available types of basic stability criteria can be selected from a popup menu. This menu appears when the user selects the cell. Each requirement can be set a number of variables. Everywhere in the list below where ‘a certain value’ is used, or similar words, it is meant that the value can be set freely by the user. An overview of the criteria per basic requirement is:

No criterion
: No criterion is automatically selected for a newly added criterion, so that if a requirement is not explicitly defined, (including type), it will not influence the calculation.

Upright G’M
: The G’M value at 0° must have a certain value. Here, upright, G’M is calculated with G’M = KM - (KG+↩ GG’). In a calculation including the the shift of COGs of liquid (see section 5.1.4 on page 43, (Damage-) stability including the shift of COGs of liquid), that GG’ is determined using the free surface area of the liquid corresponding to the actual trim of the vessel. In a calculation without this setting, the basis for GG’ is formed by the free liquid surfaces at even keel (zero trim). /dd>

G’M at a specific angle
: The G’M at a certain (user=defined) angle of inclination is tested against a certain value. Here, as G’M the tangent to the GZ-curve at that angle of inclination angle is taken. There are three further explanations to be made to this criterion:

  - The tangent to the GZ-curve may vary considerably in an area where teh GZ-curve has a large curvature. To be able to determine G’M accurately, in such areas a sufficient number of angles of inclinations (see therefore section 5.2 on page 45, Angles of inclination for stability calculations) should be used (because the GZ-curve is formed by a calculation at those angles, inbetween is only interpolated).
  - If the angle at which the G’M must be determined is zero, then the G’M according to this criterion is theoretically exactly the same as according to the criterion discussed just above. And practically also (provided there are sufficient angles if the curve in that neighborhood is rather curved).
  - The comments made under section 15.6.5 on page 303, Maximum allowable VCG at criterion ‘GM at equilibrium’ also apply to this criterion.

Longitudinal G’M
: The longitudinal G’M in upright position must have a certain value.

GZ somewhere a specific value
: Within a certain range, the GZ value must be larger than a certain value *somewhere*.

GZ everywhere a specific value
: Within a certain range, the GZ value must be larger than a certain value *everywhere*.

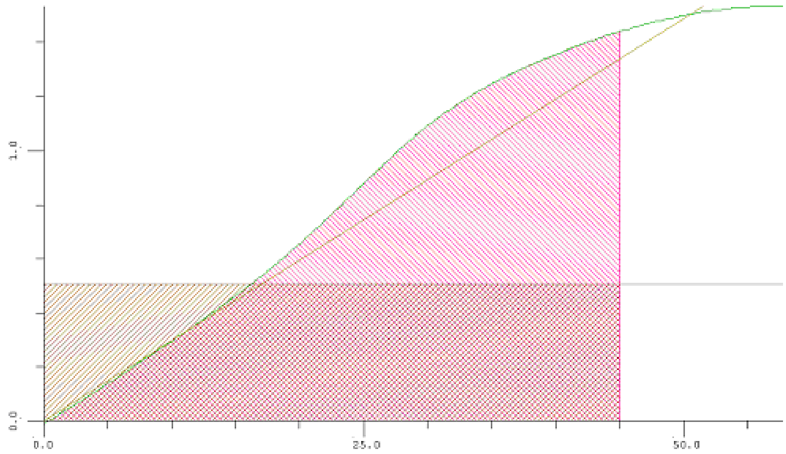
$GZ > X \times \sin(\varphi)$
: Within a certain range, the GZ value must be larger than a certain value times the sinus of the angle of inclination *somewhere*.

GZ a specific value, at a specific angle
: The GZ must be larger than a certain value at a certain angle.

Top of the GZ curve at a specific angle
: The top of the GZ curve must lie beyond a certain angle.

Area (dynamical stability)
: The area (in meter.radians) under the GZ curve must have a certain value in a certain range.

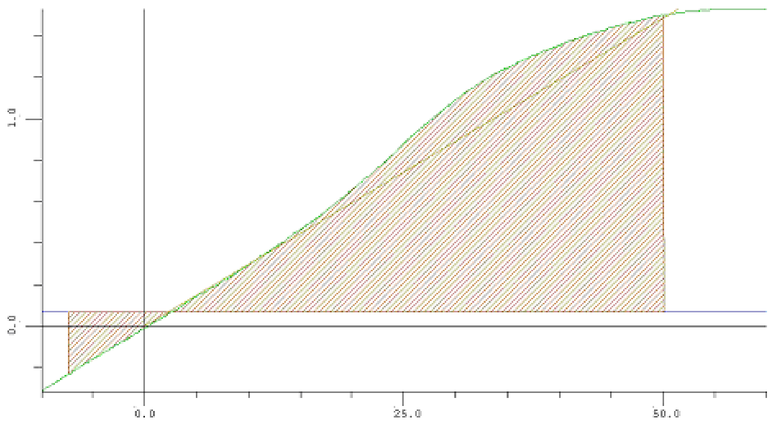
Area ratio GZ curve / wind lever
: The ratio of the areas A $\div$ B under the GZ curve and the wind lever respectively must have a certain value within in a certain range.



Area ratio GZ curve / wind lever.

Area ratio windward / leeward

The ratio of the areas A ÷ B enclosed by the GZ curve and the wind lever, calculated from a certain rollback angle windward,to a certain angle to lee must have a certain value.



Area ratio windward / leeward.

Statical angle of inclination

The static angle of inclination must be less than a certain value.

Angle of vanishing stability

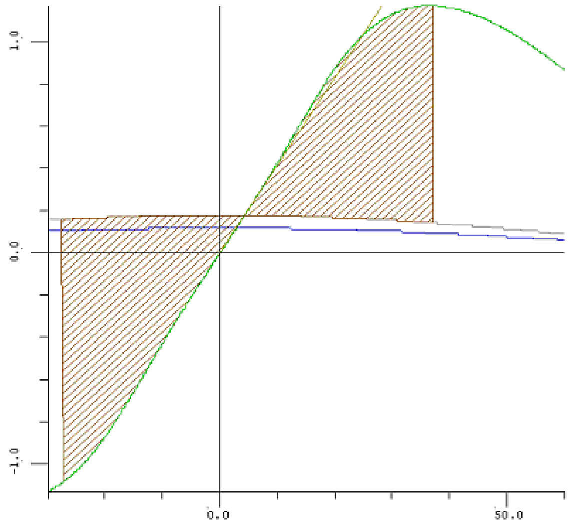
The angle of vanishing stability must have a certain value.

Range of the GZ curve

The range of the positive part of the GZ curve must have a certain value, within a certain range.

Dynamical leeward angle with beam wind

The roll angle to lee due to wind may have a certain maximum value, calculated from a certain rollback angle windward, with a certain factor for wind gust.



Dynamical leeward angle with beam wind.

The allowable angle leeward due to rolling under the influence of wind is assessed via the hatched surfaces. The area on the right side of the graph equals the area on the left side of the curve.

Note: The wind heeling lever is multiplied by a 'wind gust factor', see the differences in the wind heeling levers at section 15.5.4.1 on page 299, Wind lever. Both levers are multiplied by the cosine of the heeling angle in the example.

Residual freeboard at midship
: The remaining freeboard at half length (as determined from the depth and breadth in Hulldef) must have a certain value at the static angle of inclination.

Distance to deck
: The distance from the waterline to the deckline (as determined from the depth and breadth in Hulldef) must have a certain value at the static angle of inclination.

Distance to special points
: The distance from the waterline to openings or margin line points (as defined in Hulldef) must have a certain value at the static angle of inclination.

Distance to V-line points (non-watertight)
: The distance from the waterline to V-line points (as defined in Hulldef, see section 7.2.8 on page 184, Openings) must have a certain value at the static angle of inclination, with a certain roll margin.

STIX (stability index sailing yachts)
: The coefficient according to the STIX requirement (as defined ISO/DIS 12217, from 2000-10-05) should have a certain value.

Roll period
: The roll period as calculated from selected formulae must have a certain value. Available are estimations according to the Irish Maritime Authority ($T = 0.7 \times B \div \sqrt{G'M}$), and those according to the Intact Stability Code.

Stability ratio with/without waves
: The ratio of areas under the GZ curves for calm water and in waves may not exceed a certain value.

s(final) (SOLAS probability of survival) and s(intermediate) (SOLAS probability of survival)
: The probability of survival should be greater than the given value. Applicable to passenger vessels, according to SOLAS 2009 and 2020, part B-1, regulation 8-2. Note that the 'final' criterion is subjected to the effects of evacuation points.

External table of maximum allowable VCG'
: With this criterion the user can enter a table of maximum allowable VCG (or minimum required GM). For more details please refer to section 15.5.5 on page 301, Input of externally defined tables of maximum allowable VCG'.

Maximum allowable VCG' European inland waterway container
: The European ES-TRIN 2017/1 regulations contain rules, paragraphs 27.02 and 27.03, for the stability

of (inland waterway) container vessels navigating the European inland waterways. These have been pre-programmed, and are readily available within PIAS, see section 7.2.1.12 on page 170, Particulars for inland waterway container vessels for more information. In the end, these rules provide a maximum allowable VCG, which is used to assess stability where neccessary.

Subdivide in subcriteria
A criterion of this type can be defined as a set of criteria of all listed types. Such a set or subset of criteria can be treated, manipulated and defined as an independent set of criteria. It is even possible to create subsets within subsets.

15.3.4 Valid up to statical angle

This column is only valid for the criterion ‘External table of maximum allowable VCG’ ’, because here it can be given to which statical angle of inclination this table is valid.

15.3.5 Determinative (toolbar function)

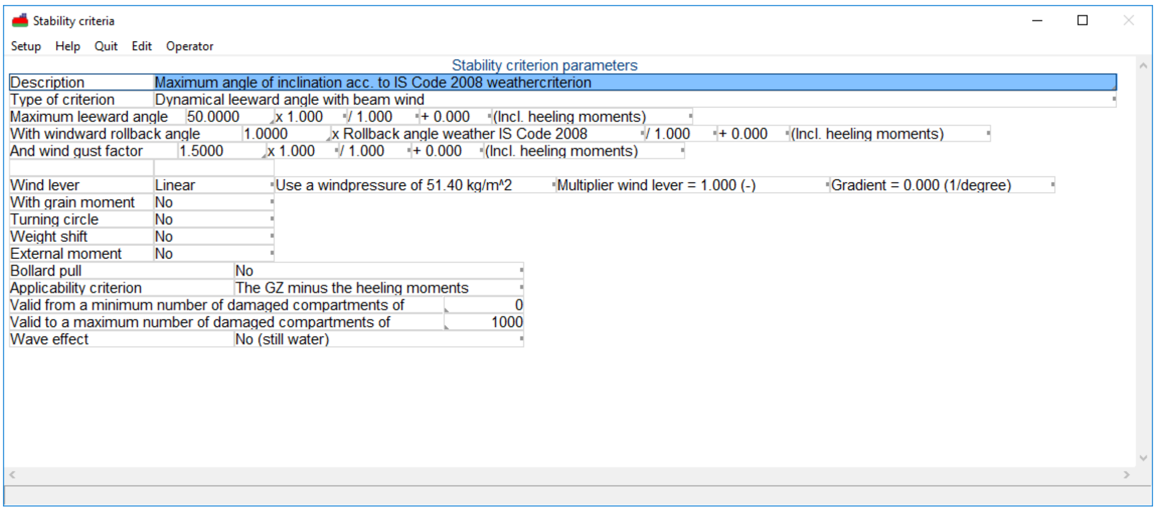
The toolbar function [Determinative] determines whether the most or least critical criterion is normative. In most cases all criteria should be complied with, in which case the bottom line in the window reads ‘The GZ curve must comply with all stated criteria’. However, it may occur that only one of the requirements needs to be complied with, which makes the bottom line to display ‘The GZ curve only needs to comply with 1 of the stated criteria’.

15.3.6 pasTe criterion special (toolbar function)

With the function [pasTe criterion special] one can copy one specific criterion from an other set of criteria to the current set of criteria. This function is only available when in a set of criteria other than itself with use of ‘Copy row’, see section 4.4 on page 39, Copy, paste etc., a copy has been made of a criterion. This also means that this function will not be available when one uses the ‘Copy row’ in the ‘current’ set of criteria.

15.4 Defining the parameters of the stability criteria

For each individual criterion, all parameters can be defined. The type of criterion determines which value is checked, how this is done, etc. The general structure of the menu for defining the parameters per criterion is explained by the example of the wind criterion of the Intact Stability Code:



Example of definition per criterion.

15.4.1 Description

The first row contains the user-defined description, see also section 15.3 on page 287, Manipulating individual criteria. The description can be changed through this menu.

15.4.2 Type

The second line contains the type of basic criterion as defined in the menu of section 15.3.3 on page 288, Types of basic criteria. The type of criterion be changed in this menu with the exception of a few basic criterion such as, 'No criterion', 'Subdivide in subcriteria' and 'External table of maximum allowable VCG' '.

15.4.3 Parameters

Depending on the type of criterion, a number of rows containing the parameters for the concerning criterion follows. In this example there are three user-defined parameters: 'Maximum leeward angle', 'With windward rollback angle' and 'And wind gust factor'. Setting these parameters is discussed extensively in section 15.5 on the following page, The nature of the stability criterion parameters. An overview of possible parameters and their meaning is given in section 15.5.1 on the next page, Types of parameters.

15.4.4 Moments

For each criterion different types of heeling moments can be accounted for (wind heeling moment, turning circle moment, etcetera). Depending on the type of moment, other settings may apply. Defining these moments and their settings is described in section 15.5.4 on page 299, Defining heeling moments to be accounted for.

15.4.5 Bollard pull

At this option the trend of the heeling moment of the bollard pull, over the angles of inclination, can be specified. Possible choices are:

- No bollard pull
- Linear moment, so constant for all angles of inclination.
- Moment decreasing with the cosine of the angle of inclination.
- Austr. 1981 ABC: moment = bollard pull × (vertical lever × cos(φ) - breadth towing hook from CL × sin(φ)). By the way, this is the same formula as proposed in IMO SDC 3 (2015).
- Austr. 1981 DE: moment decreases according to the formula from the 'Commonwealth of Australian Gazette no. P3 (May 11, 1981) sect 8, C10': moment = bollard pull × (vertical lever x cos(φ+30) - breadth towing hook from CL × sin(φ+30).
- IS Code 2020, self tripping: according to, MSC 97-22-Add.1, 2.8.4.2. Which is the same formula as used by 'Austr. 1981 ABC'.
- IS Code 2020, tow tripping: according to, MSC 97-22-Add.1, 2.8.4.3.

If applicable the bollard pull can be multiplied by a correction factor which can be entered at section 7.2.1.11 on page 170, Towing hook and bollard pull. According to regulation, the correction factor is not applied for IS code 2020 tow tripping.

Furthermore, it can also be specified which vertical lever should be used to be multiplied with the bollard force, in order to obtain the heeling moment, with options:

- The distance between towing hook and the point halfway between draft and the virtual (keel) point, which can be specified with one of the hull definition modules (see section 7.2.1.11 on page 170, Towing hook and bollard pull).
- The distance between towing hook and half the draft.
- The distance between towing hook and the COG of the lateral area, which will be computed from the underwater contour shape of the selected wind contour, for example section 16.2.1.3 on page 310, Settings per loading condition.
- The distance between towing hook and any height from base. The height is defined in section 7.2.1.11 on page 170, Towing hook and bollard pull as a virtual (keel) point.

Attention

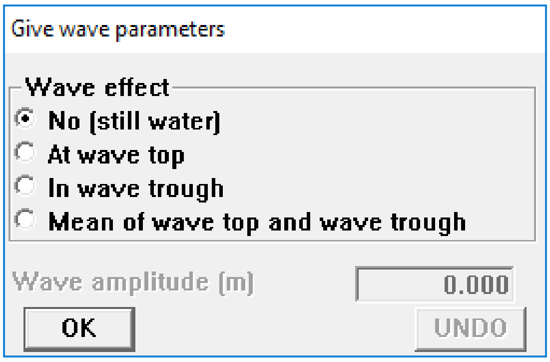
For 'IS Code 2020, tow tripping' it is not possible to specify a vertical lever because it can not be applied in the formula in a standard way.

15.4.6 Applicability of the criterion

For each criterion the GZ curve it can be specified whether the criterion should be applied on the 'The GZ minus the heeling moments', or the 'The GZ curve only'.

15.4.7 Wave effect

Some criteria may be applicable to the vessel in a wave. The way to account for a wave can be defined through a popup menu;



Wave properties.

If a wave top or wave trough is selected, the amplitude must also be specified (on the same row). The GZ curve is then calculated for this criterion with the vessel in a wave with the defined amplitude and a wave length of twice the vessel length, with the wave top, respectively the through at half the vessel's length.

If the mean of wave top and trough is selected, the GZ curve is calculated twice (for a wave top and a wave trough) and for each angle of inclination, the average GZ value is calculated. The criterion is then evaluated for the GZ curve constructed in the described way.

15.5 The nature of the stability criterion parameters

Attention

It is advised to study this section thoroughly to avoid misunderstandings. Especially due to the way parameters can be defined, as described in this section, checking stability criteria has become extremely flexible in PIAS. Unfortunately, the increased freedom involves increased complexity. Therefore, read the manual carefully in general and this section in particular

15.5.1 Types of parameters

This section describes the possible parameters that can be defined depending on the type of basic requirement. Parameter here means: the quantity appearing in the left column, in the rows directly under the 'Type of criterion', see picture below the title of section 15.4 on page 292, Defining the parameters of the stability criteria.

Between start angle
: Start angle of the range in which a parameter must reach a certain value. [degrees].

And end angle
: Final angle of the range in which a parameter must reach a certain value. [degrees].

Metacentric height
: Inclination of the tangent of the GZ curve. [m/rad].

At angle
: The angle at which a parameter must reach a certain value. [degrees].

Longitudinal metacentric height
: Metacentric height in longitudinal sense. [m].

Righting lever
: Righting moment divided by displacement. [m].

The value for X
Factor x for a basic requirement in which the key parameter varies, for example with the angle of inclination, in the type of basic criterion: $GZ > X \times \sin(\varphi)$. In this example: [m].

Angle of maximum GZ
The angle belonging to the maximum GZ value. [degrees].

Area
Area under the GZ curve. [mrad].

Area ratio
Ratio of areas under GZ curves or GZ curve and a wind heeling lever curve. [-].

To angle
To which angle from the statical angle must be rolled. [degrees].

Rollback angle windward
The rollback angle from the statical angle. [degrees].

Statical angle of inclination
Static angle of inclination at equilibrium (For damage stability at belonging stage of flooding). [degrees].

Angle of vanishing stability
The angle beyond which the GZ becomes negative or the angle at which the open openings submerse if that is smaller. [degrees].

Range of the GZ curve
Range for the positive part of the GZ curve. [degrees].

Maximum leeward angle
Allowable angle to lee for assessment of the dynamic wind heeling. [degrees].

With windward rollback angle
Angle windward for assessment of the dynamic wind heeling. [degrees].

And wind gust factor
Multiplier for wind heeling lever due to constant wind pressure, to account for a wind gust for assessment of the dynamic wind heeling. [-].

Residual freeboard at midship
Freeboard at 1/2 Lpp at the static angle of inclination. [m].

Distance to deck
Required distance from the waterline to the defined deck line points (see section 7.2.9 on page 185, Deck line). [m].

Only outside damaged region
The deck line or other points within the length of the damaged part (to be defined in the appropriate module for damage stability) will be taken into consideration, depending on the choice made here. [-].

Only applicable to deck at centerline
Defined points are considered at the defined breadth or at breadth=0, depending on the choice made here. [-].

Distance to special point
Required distance from the waterline to the defined special points (as defined at 'openings' in Hulldef). [m].

Type of special point
The type of special point for which this requirement is applicable can be selected here. [-].

Emergency exits to be included
Is specific for the special point type *emergency exit*. This specifies how emergency exits are to be taken into account. There are two ways how these are taken into account:

- All emergency exits are taken into account in the calculation.
- An emergency exit in the sense of the ADN criteria. Meaning that, when multiple emergency exits have been connected to a compartment that in the final stage one of those emergency exits should stay above the waterline with a minimum distance of 0.100 meter.

Distance to waterline
Normally a special point is compliant if its distance is **greater** than the required distance from the waterline, i.e. *Minimum*. For the special point of type *margin line*, the *Maximum* can also be taken. This means that the calculated distance is **smaller** than the required distance from the waterline.

Distance to V-line points (non-watertight)
Required distance from the waterline to the defined special points (as defined at 'openings' in Hulldef). [m].

Roll margin
Increase of static heeling angle, for which the V-line points must have the defined distance to the waterline. [degrees].

From angle of inclination
Start angle of roll, only applicable for navy vessels with Vline points. [degrees].

Stability index STIX
Required value for the stability index. [-].

Roll period
Required roll period. [s].

Estimation method
Option for choosing the estimation method for the roll period. [-].

Stability ratio
Ratio of the areas under the curves for still water and in waves. [-].

Angle for determination of area
The angle to which the area will be determined, according to NES 109, Issue 3. [degrees].

Probability of survival s(final) and Probability of survival s(intermediate)
The required probability of survival s, according SOLAS 2009 for passengervessels and in case of s(final) taking into account evacuation points. [-].

Ship type
In SOLAS 2009 s(final) contains a choice for the type of vessel. [-].

Transport of containers
If the calculation is to be made for secured or non-secured containers. Please note, when changing this parameter that it is advised to modify the description of the criterion to reflect the current state of the parameter. [-].

Calculation method
The method to be used for calculating the maximum allowable VCG according ES-TRIN 2017/1 reg. 27.↩ 02/27.03 [-].

The following methods are available:

- Approximation formulae, based on the formulaes for maximum allowable VCG, as described in the second article.
- Direct calculation, based on basic PIAS criteria as described in the first article of the afore mentioned regulations. This calculation is based on the calculated GZ-curve and curves for heeling moments. This calculation is more accurate.

15.5.2 Variables

This section describes how to define the appropriate values for each parameter. Each row contains four user-defined variables. The first one is always a user-defined number. The remaining three parameters can be either user-defined values or calculation results, see below. Some parameters can be evaluated either for the GZ curve, corrected for heeling moments (including heeling moments) or for the uncorrected GZ curve (excluding heeling moments). This setting has no effect if no heeling moments are defined for the concerning criterion. Defining the remaining three parameters is done by means of a popup menu:

Parameters for stability criteria.

If one of the parameters is selected, the corresponding value is used during evaluation of the concerning criterion. These variables are:

Statical angle
Statical angle at equilibrium.

GZ at statical angle
GZ value at angle of equilibrium (in case of defined moments).

Top of the GZ curve
Angle for which the largest value for GZ occurs.

GZ at top curve
Largest GZ value.

Angle of vanishing stability
Angle for which the GZ becomes negative.

End of the GZ curve
Largest of the calculated (defined) angles that the GZ is calculated for.

Area
Total area under the positive part of the GZ curve.

Area supply vessels
The required area under the curve for vessels with a large B/H ratio (alternative criteria according to the Intact Stability Code) = 0.055 + 0.001 $\times$ (30-angle at which the maximum GZ occurs).

Area HSC criteria
The required area under the curve according to 'High Speed Craft Code 2000 edition 2008' = 0.055 × 30 ÷ minumum(downflooding angle, angle of maximum GZ, 30 degrees).

Area MCA small multihull
The required area under the curve according to the MCA small multihull rule (MCA brown Code, §11.1.↩2.6.1) = 0.055 + 0.002 x (30-angle at which the maximum GZ occurs).

Area B semi-submersibles
The required area B for semisubs. However, this function has not yet been implemented.

Rollback angle weather Intact Stability Code 2008
The rolling angle windward as defined in the Intact Stability Code. For application of this parameter, please verify that in the Hulldef module the correct properties are defined at the applied wind contour (such as bilge shape and bilge keel area).

Rollback angle weather Russian Register 2014
According to part IV, regulation 2.1.5 of the Rules for the Classification and Construction of Sea-Going Ships from the Russian Maritime Register of Shipping, 2014. This rollback angle is virtually identical to that of the Intact Stability Code, there is a small difference in the X_1 factor, which is given for a somewhat larger range of B/d. Furthermore, there is an exception for dredgers in regulation 3.8.4.3, which is the correction factor X_3 for the windward rollback angle in case of a restricted area of navigation. If at the definition of a stability requirement a *specific wind pressure* is set (seesection 15.5.4.1 on the next page, Wind lever for that) of less than 51.4 kg/m^2, then the navigation area is assumed to be restricted, and hence the X_3 applies. It is possible to verify whether X_3 has been applied by inspection of the intermediate results, because there it will be printed if applicable.

Min. G'M IMO A.265 reg. 5
The minimum G'M based upon, IMO A.265 regulation 5.

C-factor container vessels
C factor for container vessels according to the Intact Stability Code. For this factor the vessel should be defined including hatch coaming (if present).

Displacement
Displacement (in intact condition).

Breadth^2 / 100Freeboard^2
Breadth ^{2} ÷ (100 × freeboard ^{2}). Obviously, the depth should have been correctly defined in Hulldef.

Angle of deck immersion at L/2
The angle where the deck at Lpp/2 is immersed. This function does neither apply to the geometry of the vessel, nor to the defined deck line. It is simply based on the depth, as specified with the main dimensions.

Freeboard
Depth - draft. This value is calculated with the depth as defined in Hulldef.

Angle of deck edge immersion
The angle at which one of the deck line points, as defined in Hulldef, is immersed.

VCG over VCB
VCG - VCB for zero inclination.

Absolute trim angle
The absolute trim angle in degrees, see section 15.7 on page 303, On the various criteria and parameters an application.

Heeling angle during lifting
By 'loss of load' calculations this variable is filled with the heeling angle during lifting in degrees. This variable is 1 in every other situation.

Optimal start angle range IBC/IGC
The optimal start angle of the range, according regulation 2.7.2.1 of the International Gas Code, 2016, M↩SC.370(93)

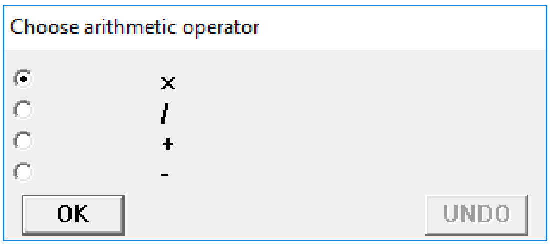
Smallest angle to special point
The first special point of the selected special points, as defined in Hulldef, to be immersed.

Res. lever v. Harpen
The remaining GZ according to formulae as defined by Van Harpen.

Angle of res. lever v. Harpen
Angle for which the residual lever according to Van Harpen occurs.

15.5.3 Operators

For each parameter the operator (the calculation to be performed) can be chosen. Choosing the operator is done via the toolbar option [Operator], if the concerning parameter is selected. A popup menu will then appear:



Selecting arithmetic operators.

Attention

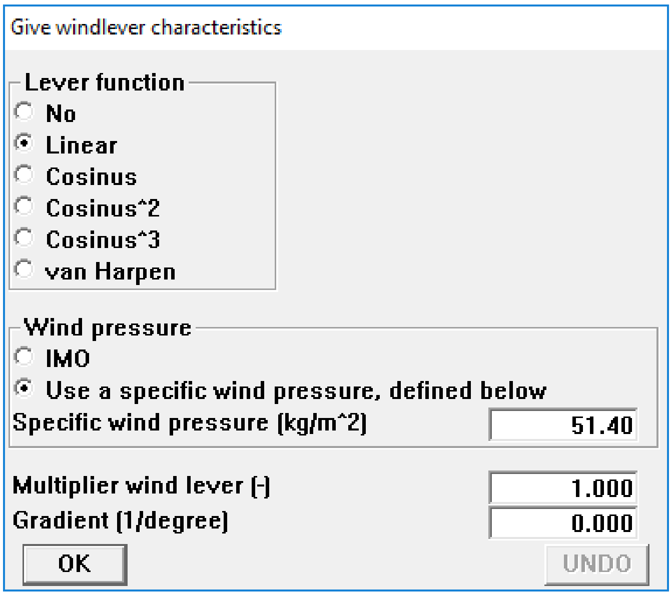
The conventional order of arithmetic does not apply to the use of these operators. The order in which the operators are evaluated is simply from left to right.

15.5.4 Defining heeling moments to be accounted for

The heeling moments to be included in the calculations can be evaluated separately or combined. If multiple heeling moments are defined for a criterion, the GZ curve can be corrected (if necessary) for the total of heeling moments.

15.5.4.1 Wind lever

Selecting the way in which the wind levers are evaluated is done in a popup menu:



Selecting type of wind lever.

As depicted in the figure above, six types of wind lever functions are predefined:

- None.
- Linear, a straight line. Here also the gradient can be given, which is the inclination of the straight line. With a gradient of zero there is no inclination, which makes the wind lever to be constant for all angles of inclination. If a gradient is given, the wind lever is multiplied by the factor (1 + angle × gradient). Assume

that for an angle of 40° the wind lever should amount 80% of the lever at 0° , then a gradient of -0.005 should be given.

- Cosine shaped, where the wind lever decreases with the cosine of the angle of inclination.
- Cosine squared, where the wind lever decreases with the square of the cosine of the angle of inclination.
- Cosine cube, where the wind lever decreases with the cube of the cosine of the angle of inclination.
- Van Harpen, where the wind lever follows van Harpens formulae (0.25 + 0.75 × cos(angle of inclination)^3).

If a wind lever is present the user also should also either select one of the predefined wind pressures (as defined in the Hulldef module, at the wind data section), or give a specific wind pressure, in in kg/m^2. Alternatively, a specific wind pressure in kg/m^2 can be directly specified. The background of this parameters is discussed under lemma 'specific wind pressure' in section 14.1 on page 273, Input data for the wind heeling moments computations.

In addition, a multiplier for the set wind lever can be specified. This parameter is also discussed in the just mentioned section, under the lemma 'multiplier on the wind lever'.

15.5.4.2 Grain heeling lever

When for the concerning criterion a grain-heeling lever is defined, this option includes or excludes it in the calculation results for this criterion.

15.5.4.3 Turning circle

If this option is selected, a heeling lever due to the centrifugal force in a turning circle is applied. The lever is given by the user as a dimensionless constant, which is multiplied by the program by (KG - T/2) in order to obtain the turning circle lever for the loading condition concerned (and by displacement × (KG - T/2) to obtain the turning circle moment in tonmeter). In general, the heeling lever of the turning circle moment decreases with the cosine of the heeling angle, however, that still has to be set by the user (leaving the freedom to choose another function, such as linear).

15.5.4.4 Shift of weight

A shift of a weight can be included for evaluation of a criterion. Both weight and dislocation can be defined. The GZ curve will be calculated including the effect of the dislocated weight.

15.5.4.5 External heeling moment

An external heeling moment can be included. The magnitude of this moment can be defined, and the behavior of this moment can be selected, similar to the wind lever, see section 15.5.4.1 on the preceding page, Wind lever.

15.5.4.6 Whether or not to apply heeling moments

It has already been discussed that in two occasions it can be specified whether or not to apply the heeling moments:

- For each individual variable, as used to compute the value of a parameter. This is specified with the cell 'Incl. heeling moments' or 'Excl. heeling moments'.
- For the entire criterion, at the field 'Applicability criterion', where 'The GZ curve only' or 'The GZ minus the heeling moments' can be entered.

These two configurations have a distinct application. The first is valid for the line on which it occurs only, and determines whether the actual value of the variable should be determined including or excluding the heeling moment effect. If, for example, in a line the variable 'Statical angle' is used, this configuration determines whether this is the statical angle determined with or without heeling moments.

The second configuration applies to the entire criterion, and determines whether it should be applied on the GZ curve only, or on the GZ curve corrected for heeling moments. Suppose a criterion requires a minimum value for the maximum GZ, than this criterion s such is not related to any heeling moment (regardless whether the criterion value should be derived taking heeling moments into account), so it is applied on the *GZ curve only*. However, should the *criterion* require a minimum lever between GZ curve and heeling moment instead, then it is applied on the *GZ curve minus the heeling moment*.

15.5.5 Input of externally defined tables of maximum allowable VCG'

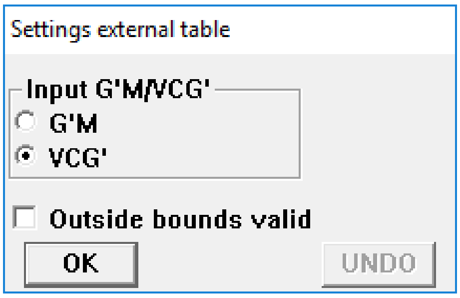
One particular type of basic criterion, as discussed in section 15.3.3 on page 288, Types of basic criteria, is the external table of maximum allowable VCG'. This criterion can, for example, be used to process maximum allowable VCG's as determined for the probabilistic damage stability, with the Probdam module, in tables or diagrams — or in the LOCOPIAS on-board loading software. These tables can be defined for multiple trims, while multiple sets of tables can be defined. Only the selected set is used for the determination of maximum allowable VCG'. Maximum VCG's for intermediate values are determined by linear interpolation. Please keep in mind that for probabilistic damage stability according to SOLAS-2009 the VCG' between the three standard drafts - light, partial and deepest - must be determined by linear interpolation on G'M. So, in this case these values must be defined at this option as G'M values.

15.5.5.1 Settings of external table of maximum allowable VCG'

In the toolbar of the external table sits an option [Settings] where several settings with regard to the whole table can be modified.

G'M or VCG'
That at the drafts the defined values are interpreted as G'M or VCG'.

Outside bounds valid
If this is **not** turned on, then in case of drafts or trims outside the defined boundaries, the stability will be assessed as insufficient. In the situation that it is **turned on**, then for the drafts or trims which are outside the defined boundaries, the nearest values are used. So, there will be no extrapolation outside the defined range.



Settings external table.

15.6 Answers to frequently asked questions on stability assessments

15.6.1 The effect of openings

All basic requirements take implicitly and irrevocably into account the non-watertight openings: weathertight openings may not be submersed at the static angle of inclination. Beyond the angle of submersion of open openings the curve is terminated, so the GZ in that range is not taken into account when evaluating a requirement.

15.6.2 Seemingly inconsistency in weather criterion from the Intact Stability Code

If the criterion 'Maximum angle in weather criterion Intact Stability Code' is the determining one, it is possible that although in the summary of a loading condition the maximum allowable VCG is less than the actual VCG', the program reports that the loading condition complies. This effect is caused by the formula for determining the 'Roll angle windward' in which the uncorrected VCG (that means, uncorrected for free surface effects) must be applied, and this value might in a real loading condition differ from the assumption for determining the maximum allowable VCG (where there is obviously no separation between real and virtually risen VCG).

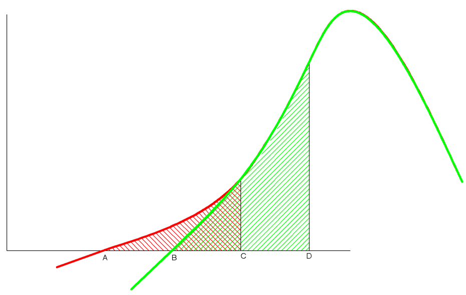
15.6.3 Determining parameter in the weather criterion from the Intact Stability Code

According to the text of the *Intact Stability Code* (IS) the ultimate parameter that determines whether the wind criterion is met, is the surface on the right side of the GZ curve b, which should be larger than that on the left,

a. PIAS uses a slightly different parameter: b is set equal to a, and the roll angle is determined at which this is exactly the case. This angle may not exceed the maximum — usually 50°, or when an opening is submerged. This presentation is a bit more appealing for a human than the IS view, because inclination angles are a more intuitive notion than some area under a GZ curve. Obviously, eventually the conclusion is exactly the same.

15.6.4 Extent for determining the minimum lever of the area under the stability curve.

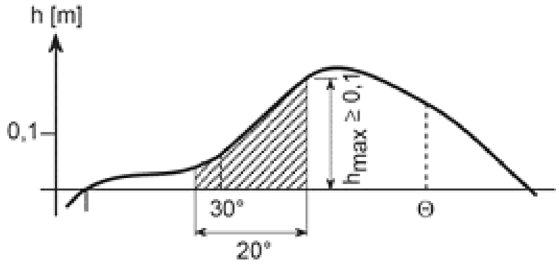
Some damage stability requirements have been nicely conceptualized, but may turn out a bit unexpected in practice. For example the requirement that 'within a range of 20° from the statical angle of inclination' a minimum area or lever should be present. Take for example the case depicted in the figure below, which shows two GZ curves. The red one is evidently better than the green one, because of its larger area and smaller statical angle of inclination. However, the 'within a range of 20°' plays foul, because the statical angle is A, which makes 'plus 20°' to be situated at C. And because of the flat character of the GZ in that region the area is small, too small to fulfill the minimum requirement. With the green curve the statical angle is B, which 'plus 20°'r leads to D, resulting in a much larger area. The result is that the better GZ curve does not comply, while the worse does.



Two curves of righting levers in damaged condition.

As such, this finding is not new, at SARC this phenomenon appeared in 1989 for the first time. And the solution is simple, by not taking this 20° from the statical angle of inclination, but from any angle between the statical inclination and the maximum allowable inclination instead. In PIAS this is achieved by subdividing this kind of criteria in many (more than ten) sub-criteria which each cover a piece of the search area, and taking the best. For the legislator this is also charted waters, as illustrated by the text of *MSC/Circular.406/Rev.1 — Guidelines on Interpretation of the IBC Code and the IGC Code — (adopted on 29 June 1990)*, which reads: '....The 20° range may be measured from any angle commencing between the position of equilibrium and the angle of 25°....'. Also classification societies might be aware of this effect, given the figure below with an interpretation by Germanischer Lloyd. However, it is never guaranteed that this position is recognized in all cases, so it is advised to inquire in advance.

2.9-0.1 The 20° range may be measured from any angle commencing between the position of equilibrium and the angle of 30°, see Fig. 2.3.



h = righting lever

Θ = progressive flooding angle

Fig. 2.3 Range of positive stability where the residual stability should be evaluated

Interpretation of GL for the calculation of the area under the GZ curve with the IGC Code.

15.6.5 Maximum allowable VCG at criterion 'GM at equilibrium'

An occasional stability regulation contains a criterion of 'G'M at equilibrium', so, at the statical angle of inclination. However, for this criterion multiple solutions may be possible, in other words, multiple KG's where G'M exactly matches the criterion value. Moreover, in general an increasing KG' will be detrimental for stability, but for this criterion this need to be not so: with the rise of K'G, in an asymmetrical damage case, the equilibrium angle will increase, so it may shift to a steeper part of the GZ curve. Where the G'M is larger, and this criterion is **better** matched! The general assumption is that a diagram of maximum allowable KG' contains a single transition line between areas of *sufficient* and *insufficient* stability, but with this particular criterion this does not hold because the diagram can contain islands of *sufficiency* or *insufficiency*. The search procedure of PIAS will find a single transition KG', but there may be more.

15.7 On the various criteria and parameters

This manual describes all functionality regarding the (damage) stability criteria. However, in the PIAS implementation a segregation into three 'packages' is applied:

- Standard (item 50.200.10 of the price list).
- Naval criteria (50.200.30), everything related to 'naval', such as DDS-079, van Harpen and NES-109.
- Extensions 2009 (50.200.40):
  - Incorporation of the bollard pull moment into the stability analysis.
  - Stability criteria for tugs, according to Bureau Veritas (2006). **Will be superseded by Intact stability Code 2020 self and tow tripping.**
  - Stability criteria for tugs, according to the Commonwealth of Australian Gazette no. P3 (11 mei 1981) sect 8, C10.
  - The parameter *Absolute trim angle*, to accomodate the combined heel and trim, as required by chapter 17.07 of the ROSR (vessels on the river Rhine).
  - The parameter *Area MCA small multihull*. This is similar to the required area for supply vessels, but with a small difference: 0.055 + 0.002 × (30-angle at which maximum GZ occurs) instead of 0.055 + 0.001 x (angle at which maximum GZ occurs). Is applied in the MCA small boat Code (brown Code) for multihull vessels (§ 11.1.2.6.1).
- Extensions 2017 (50.200.50):
  - Bollard pull function 'Self tripping' according to, Intact stability Code 2020 (MSC 97-22-Add.1, 2.↩ 8.4.2).

  - Bollard pull function 'Tow tripping' according to, Intact stability Code 2020 (MSC 97-22-Add.1, 2.↩ 8.4.3).
  - The *basic criterium* 's(final) (probability of survival) and s(intermediate) (probability of survival)' and *parameter* 'Probability of survival s(final) and Probability of survival s(intermediate)' for SOLAS 2009, with regard to passenger vessels, according to Consolidated text 2014, part B-1, regulation 8-2.

Finally, on the completeness of the stability criteria the following *disclaimer* statements are made:

- The sets of predefined standard criteria do not offer a complete and up-to-date survey of all stability criteria. The user should always check the applicable stability criteria with the authorities concerned.
- Because criteria are often prone to interpretation, users must check for themselves that the requirements are defined according to the prevailing interpretation. Users are able to change selected predefined sets of criteria to meet their own interpretation.
- A consequence of the flexibility of the system is that it is not possible to check whether **input** data are realistic or even possible. It is, for instance, possible to define contradictory criteria. PIAS users are expected to be capable to investigate whether a certain criterion yields the correct results.
- It is recommended to generate and check *intermediate results* in case of unclear or unexpected results.

Chapter 16

Loading: loading conditions, intact stability, damage stability and longitudinal strength

With this module loading conditions can be defined, and used to compute intact and deterministic damage stability, longitudinal strength and torsional moments. For a quick definition of loading conditions auxiliary tools are available, such as automatic tank reading, a visual interface for tank filling and a database of weight items.

At the start of Loading, all weight items are checked for references to tanks that have been removed. If this occurs, a choice must be made of what to do with those specific weight items. There are a few possibilities:

- Removal of the weight items with a reference to a non-existent tank.
- Convert the weight items referring to a non-existent tank to fixed weight items.
- Don't do anything automatically, these references will have to be resolved manually.

If "don't do anything automatically" is selected, and these references have not been resolved manually, then this popup will return the next time Loading is started.

Intact stability, damage stability & longitudinal strength

16.1 Graphical User Interface

Here appears the Graphical User Interface (GUI), which can be considered as the central command window of Loading, see section 17.1.1 on page 324, Main window layout. This GUI is not strictly indispensible, without it is very well possible to navigate through Loading with all functions and menu options, however, it has proven to be rather well-arranged and user friendly. The GUI is basically identical to the LOCOPIAS on-board loading software, which is derived from PIAS.

16.2 Loading conditions

With this option you enter the list of loading conditions, which can be added, removed, copied and modified from here. This list contains the following columns:

Select intact
Indicated whether the loading condition is selected for, and thus should be included in, the calculations of intact stability, longitudinal strength and torsion.

Select damage
Indicated whether the loading condition is selected for damage stability calculations.

Name of the condition
The name can freely be chosen, however, it should be unique in order to enforce some structure.

Auto-remove
This column is only included in a PIAS version with a licence for calculating the stability of hopper dredgers, and indicates that this loading condition has been generated for hopper stability. The details of these calculations are discussed in chapter 18 on page 384, Stability of open hopper vessels.

Locked
With this cell a loading condition can be locked; it will become grey then, cannot be changed anymore, and is only available for copy purposes. Locking a condition can be useful to protect special conditions from unwanted changes.
Caution: if in a loading condition weight items from the common list (which will be introduced later on) are used, then changes to these items will nevertheless have their effects in the locked loading condition.

The menu bar of this list contains these functions:

- [Manage], with sub-options:
  - [Move] and [Quitmove], with which a loading condition can be moved in the list of loading conditions.
  - [Design data from Layout], with which the weight group and density of all tanks in all loading conditions can be connected to the design weight group and design density of Layout.
- [Common list], which gives access to the common list of weight items, see section 16.3 on page 314, The common list of weight items.
- [hopPer], where parameters can be specified for (damage-)stability calculations of a hopper dredger. Background and method of working with this type of calculations is described in section 18.4 on page 385, Specify loading parameters.
- [Damstab], where damage cases can be defined and other damage stability particulars can be set, see section 16.2.2 on page 313, Damage cases and settings for damage stability.
- [File], with the following sub-options:
  - [Import], which imports loading conditions — which should be available in internal PIAS format, with the .exp extension.
  - [Export], which exports all selected loading conditions to this internal PIAS format.
  - [Maestro], which exports the single loading condition where the text cursor resides to a format suitable to be imported into the finite element software *Maestro* (please refer to `http://www.maestromarine.com`). The file that is generated contains *Comma-Separated Values* — with file extension .csv — with the following particularities:
    - If a tank name contains a comma, it will be removed, because those are simply not allowed in the content of a .csv.
    - The permeability which is passed to Maestro is that of the first subcompartment for tank capacities.
- [GUI], which invokes the GUI for the loading condition where the text cursor resides, see section 17.1.1 on page 324, Main window layout.
- [Output], to execute computations for all selected loading conditions. Details are being discussed in:
  - Settings output, section 16.4.1 on page 315, Output settings.
  - For intact stability calculations section 16.4.2 on page 315, Intact stability.
  - For longitudinal strength section 16.4.3 on page 316, Longitudinal strength.
  - For torsion calculations section 16.4.5 on page 318, Torsional moments.
  - For deterministic damage stability section 16.4.4 on page 317, Deterministic damage stability.
  - For calculations of overflow time in damaged conditions section 16.4.4.2 on page 317, Calculate cross-flooding times.
  - For a ‘Sounding table’ see section 16.4.6 on page 318, Sounding table and for a ‘Cargo/ullage report’ see section 16.4.7 on page 318, Cargo/ullage report.
  - [to XML], with this option the output of intact stability, longitudinal strength, torsional and damage stability calculations will be written in an XML-file.
- With [re-read All tanks], existing weight items which have already been read from tank tables will be read again. This option is required if compartment shapes have been changed, and the modifications should be processed in all loading conditions integrally — in which case one should not forget to re-compute the tank

tables in Layout. The tanks are read on basis of the existing percentage of filling. Also tank names are read again.

- And, finally, <Enter> brings you *into* a loading condition. This key opens the list of weight items of the loading conditions, which is further discussed below.

16.2.1 Define/edit weight items

An input window appears with all weight items for this loading condition. If weight items in the common list are defined as being a part of light ship, the first row will contain the sum of these items. This is the total light ship weight, which cannot be modified here — after all, how would such a modification be distributed over the many constituting components? If for the project weight groups are in use (please refer to section 20.1 on page 395, Weight groups for that) that the list also contains rows with the sub totals per group. If a color is assigned to a group then this sub total is printed in that color.

The volumetric properties of the tanks always matches those from Layout where those tanks are defined. Weight group and density are connected by default to the design weight group and design density as defined in Layout. Any changes in Layout will then update these data in Loading. For each individual tank in Loading, specific weight group and/or density may be given. In this case the connection to Layout will be cut, for that specific tank. Such connected parameters are indicated by a yellow background cell color.

This list of weight items contains quite many columns — scrolling will bring you to the rightmost columns — which are:

Name
: The name of the weight item.
If the temperature corrections functionality has been purchased then one can double-click on the name of a tank to enter the temperature corrections menu. See section 16.2.1.4 on page 312, Product, temperature and density for more information.

ComList
: If this is weight item from the common list of weight items (please refer to section 16.3 on page 314, The common list of weight items for a discussion) then this column contains the consecutive number of this item in that list. In this cell a number can be entered, then that weight item from the common list is included here. Because it is not convenient to manage number from a list, with <Spatie> a box pops up with the full descriptions of the items of the common list, from which you can select. Please note that in both cases the common listitem overwrites the original row content.

Type
: If a weight item is created by a loading tool, then it is assigned a ‘type’, e.g. ‘tank’, ‘crane’ or ‘containerbay’. In most cases the cells of a weight item of such a ‘type’ cannot be modified manually, after all those are being managed by the loading tool. An exception is a weight item of the ‘tank’ type, from which weight, volume, filling percentage or density can be entered, after which the other parameters (including COG) will be adapted automatically.
A weight item can be switched from such a ‘type’ to a regular (loose) weigh item with the <Space bar>. A ‘tank’ type of item has — if this functionality has been purchased — yet another option, which is the ‘flooded tank’, which is a tank in open connection with the sea.

Weight
: In ton.

VCG, LCG and TCG
: The Center Of Gravity, in meters from respectively baseline, APP and Center Line.

FSM
: The Free Surface Moment, of a liquid weight item, in tonm.

FSM type
: Different people have different visions on how to take Free Surface Moments (FSM) into account. The preferred method can be entered here, with a choice between:

1. FSM at the true volume (and true liquid level), derived from the tank shape.
2. The maximum FSM (of all liquid levels), derived from the tank shape.
3. Zero.

4. If tank filling $< 98\%$ then method 1, else zero.

If a deviating FSM has been chosen, a !-symbol is printed in the output at the FSM with the option chosen. At the bottom of the page or weight list is a statement of the chosen options. Caution: this setting is not applicable for calculations including the shift of liquid effect (as discussed in section 5.1.4 on page 43, (Damage-) stability including the shift of COGs of liquid), in which case always the real behaviour of the liquid is taken into account.

Weight group
The weight group of this weight item. For a weight item the group can be chosen in two different ways, either by entering the weight group number in this cell, or with <Spatie>, which pops up a selection window of groups. The weight group can also be connected to the design weight group as given in Layout. In that case the cell is colored with a yellow background, and changes to the Layout weight group will change it in Loading accordingly. Weight groups are discussed in section 20.1 on page 395, Weight groups, and are a convenient tool to order the weight items. If in the weight group definition menu a color is assigned to a weight group, and the column 'in table' is set to 'yes', then in the weight item list here each weight item of that group is drawn in that color.

%, Density and Volume
These three parameters are only applicable for weight item of the 'tank' type, and denote the percentage of filling, the density (=specific weight) in ton/m^3, and the volume in m^3. The density can be connected to the design density specified in Layout, similar to the weight group as discussed before.

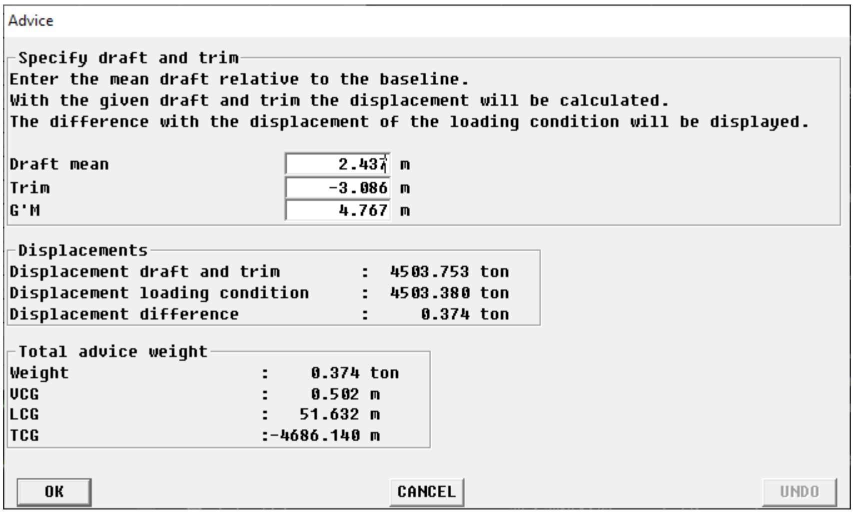
Measured, Trim sounding and Angle sounding
In the 'Measured' column a *Sounding*, *Ullage* or *Pressure* can be specified, as long as a sounding pipe and/or pressure sensor is available. With the columns 'Trim sounding' and 'Angle sounding' the trim and angle at the time of "sounding" can be specified. Note: The 'Measured' column contains the measured value associated with the specified trim and angle. Other data, i.e. columns, such as weight, volume and centre of gravity are determined at trim zero and angle zero.
If this functionality is not purchased then the 'Measured' column is only applicable to 'grain hold' weight items only, and depicts the *Ullage*, which is the distance between the top of the coaming and the grain surface. This column is only available if configured so in the Loading settings, see section 16.5.1 on page 320, Settings intact stability.

Aft and fore
Here the afmost and foremost boundaries of a weight item should be given. in meters from A_{PP}. For a 'tank' type of weight item these parameters are being derived from the tank shape. These parameters are only relevant for the longitudinal strength calculations, as discussed in section 16.4.3 on page 316, Longitudinal strength.

Furthermore, this menu contains quite some upper bar functions:

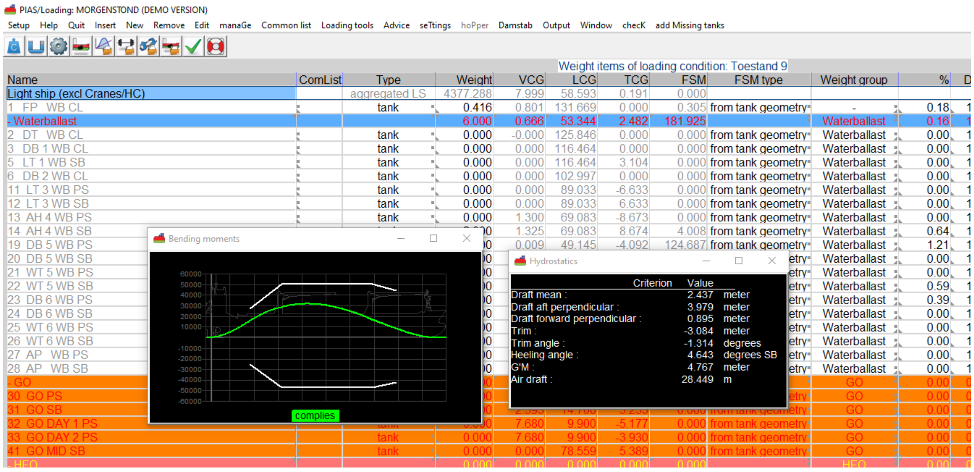
- [Manage], which can be used to adapt the menu layout to some extend:
  - As discussed the weight item list also contains sub totals per weight group. For overview purposes it might be handy to see only these totals, which can be achived by the [Collapse weight groups] function. [Expand weight groups] restored the usual layout.
  - With [Move] a weight item can be moved in the list, in a similar fashion as in the list of loading conditions, and the list of compartments in Layout.
  - With [Sort] the weight items can be sorted, in four different ways:
    - ∗ In the order in the column where the text cursor resides, to be precise in increasing order for the columns 'name' and 'weight group', and in decreasing order in the other columns.
    - ∗ On group and column (i.e. on group number, and subsequently in the same way as just sorting on column).
    - ∗ On group and location (i.e. on group number, and subsequently in decreasing order of location based on aft and forward boundary).
    - ∗ [Undo last sort] restores the previous sequence. Please note that with repeated sorting, or when Loading has been closed, the original sequence cannot be restored. There is no intrinsic order which can be restored.
- With [Common list] the common weight items for the loading conditions are processed:
  - [Edit common list] calls the edit window of the common list, see section 16.3 on page 314, The common list of weight items for a discussion.
  - [Connect] connects a weight item to the common list.

  - [Disconnect] removes the connection with common list.
- [Loading tools] contains a number of loading tools, which are very convenient, although not indispensable:
  - [Tanks], the GUI for tank filling, see section 17.2 on page 352, Graphical User Interface for tank filling.
  - [Containers], the GUI for container loading, see section 17.4 on page 362, Graphical User Interface for container loading.
  - [crAnes], the GUI for crane loading, see section 17.5 on page 370, Crane loading tool.
  - [Grain/Bulk], a tool for loading of grain or bulk, see section 17.6 on page 376, Grain/bulk.
- With [Advice] the weight addition to achieve a certain desired position (draft and triim) of the ship. Here a popup box appears, see the example below. The 'Given draft and trim' block initially displays the actual values for this loading condition but these should be modified to the desired values. In the 'Displacement' block the differences between the desired and actual displacement is displayed. Finally the 'Total advice weight' block indicates where the advice weight should be located in order to achieve the desired draft and trim. A heeling angle of '>6' of '<-6' indicates that that angle is larger than can be computed sufficiently accurate on basis of the GM'. Pressing <OK> adds the advice weight to the loading condition, <Cancel> does not.



Advice weight.

- [Settings] is used to open a popup window where all kinds of settings for this particular loading condition can be made. This is discussed in section 16.2.1.3 on the following page, Settings per loading condition.
- [hopPer], is described in section 18.5 on page 386, Parameters from an individual loading condition.
- [Damstab] is used for damage cases and settings for damage stability. Please refer to section 16.2.2 on page 313, Damage cases and settings for damage stability for more information.
- With [Output] for this single loading condition a computation can be made, as discussed in section 16.4 on page 314, The computations of stability and strength.
  - That with [Settings output] the output can be modified, see section 16.4.1 on page 315, Output settings.
  - Besides the real computation, with [Plots tanks] sketches can be generated of the tank filling for this loading condition. The format and content of these sketches are according to the settings as discussed in section 16.5.8 on page 322, Define sections for sketches of tank arrangement and damage cases. The output is send to printer, unless it is captured to a *Preview* on screen.
- [Window] contains the following options:
  - [Result windows] with which the prime results of total weights, hydrostatics, stability or strength are presented via a floating window (windowlets). The contents of these windowlets are continuously updated, so the effect of changes in weight items of the loading conditions is immediately visible. An example is shown below.



List of weight items, including two windowlets with computation results.

- [Check] verifies compliance with the criteria for all calculations that are managed by Loading (and for which sufficient data are available). This option opens a popup window with a collection of sheets. The first sheet contains the general conclusion, and the next sheets the conclusion and prime parameters of the sub-computations.
- [Add missing tanks] adds all newly created compartments to the current loading condition. Only those tanks are added for which the property ‘Automatic inclusion in loading condition’, paragraph 9.5.1.2.14 on page 218, Automatic inclusion in loading condition, is set to ‘Yes’.
- [-prev] jumps to the previous loading condition in the list of loading conditions. This option skips **Locked** loading conditions.
- [+next] jumps to the next loading condition in the list of loading conditions. This option skips **Locked** loading conditions.

16.2.1.1 Fill tanks per weight group

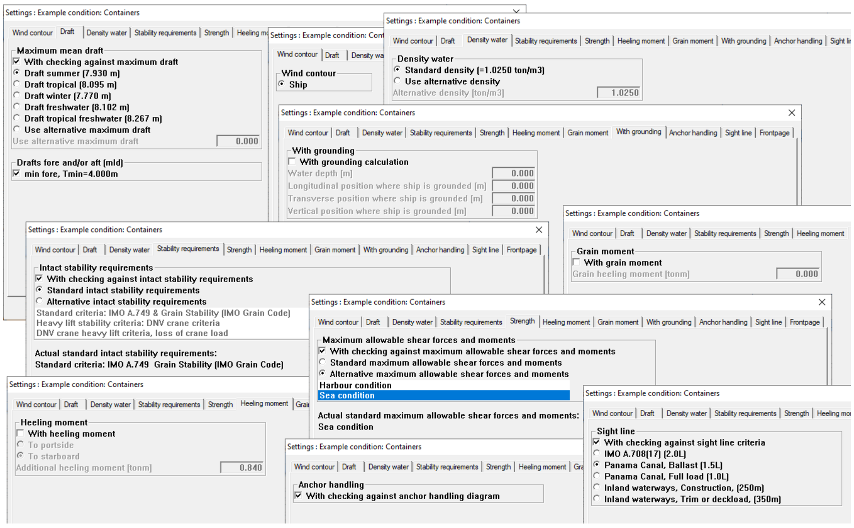
Loading conditions for design situations are often made with a uniform filling for each type of tank content, e.g. in departure conditions the fuel oil and fresh water tanks filled for 98%, and waste water for 10%. Modifications for the whole weight group can be easily applied via modifing the appropriate value on the sub total line. Possible modifications are: ‘FSM type’, ‘Weight group’, ‘Tank filling’ and ‘Density’. Do note that with ‘undo’ it is possible to restore the modification.

16.2.1.2 Read tanks as weight item

It has already been discussed that weight items can be of the ‘tank’ type, then they are linked to the shape of a compartment as defined by Layout. To use a tank as weight item it should be ‘read’ into the list of weight items. With the menu bar option [Add missing tanks], as described in section 16.2.1 on page 307, Define/edit weight items, all missing tanks will be added. A tank can not exist twice in a loading condition, not even by means of the common list.

16.2.1.3 Settings per loading condition

With Config general PIAS settings are specified, while here in Loading, with the option ‘Loading project settings’ Loading-specific settings can be made. But calculations of stability and strength be carried out in practice for very many different situations, and therefore it may be necessary for specific load conditions to deviate from the standard. You can with this option. If it is selected, it displays a popup window with several sheets for different topics that are discussed in the following paragraphs.



Several sheets for settings per condition.

16.2.1.3.1 Wind contour

If there are multiple windcontours available, then for each loading condition the appropriate wind contour can be selected. That can be done here; a list appears from which a selection can be made of the contours as defined in Hulldef.

16.2.1.3.2 Draft

In Hulldef multiple loadline drafts might have been defined, such as for 'summer' or 'fresh water' (see section 7.2.1.5 on page 169, Maximum drafts and minimum freeboards). Here, for this particular loading condition, can be chosen which of those drafts should be included in the assessment of the condition. Furthermore, in Hulldef also minimum or maximum drafts specified (see section 7.2.1.4 on page 168, Draft marks and allowable maximum and minimum drafts). If these are defined, here it can be selected which of those is applicable to this loading condition.

16.2.1.3.3 SW water

This option indicates the density (specific weight) of the outside water. Either a specific value can be used, or the standard value as defined in in Config (section 5.1.6 on page 43, Density outside water).

16.2.1.3.4 Stability requirements

At the definition of intact stability requirements (see section 16.5.6 on page 321, Definition and selection of stability requirements for that) can be specified which is the selected one. That one will be used for the stability assessment of all loading conditions, but occasionally a loading condition must be tested against other requirements. That can be set here. To do so, choose 'alternative intact stability requirements', and select the stability requirement which applies to this loading condition. Obviously, only requirements which have been defined, and are valid for intact stability can be selected.

16.2.1.3.5 Strength

This option indicates which set of maximum allowable bending moments and shear forces are used for the longitudinal strength calculations. There are two possible options:

- Use the standaard criteria.
- Specificly selected criteria for this loading condition.

See section 16.5.7 on page 321, Definition maximum allowable shearforces and moments on how to define these criteria.

16.2.1.3.6 Heeling moment

Here an additional heeling moment can be given, which is accounted for as a correction on the TCG of the displacement. This implies that this moment will be multiplied with the cosine of the angle of inclination.

16.2.1.3.7 Grainmoment or Livestock and fodder moment

Here the heeling moment (in tonm) due to the shift of grain can be specified. In case of grain, if stability critaria are in use 'with grain moment', then this moment is used in the stability calculations.

16.2.1.3.8 Grounding

Here can be specified whether the vessel is grounded on a single point, the location of that point (in ship coordinates) and the water depth at that location. To be somewhat more precise, the mechanism is that the ship is *potentially* grounded, so the local draft at the grounding location can be less than the water depth (in which case the ship is not grounded at all), but cannot be more. This grounding effect is included in all computations of Loading, while in the intact or damage stability calculation an extra page will be added with the reaction force as a function of the heeling angle (if that option has been switched on in there respective section 16.4.1 on page 315, Output settings tab page). The grounding effect will only be taken into account with intact and damage stability calculations with the free to trim effect (see section 5.1.3 on page 42, Stability calculation method)

16.2.1.3.9 Anchor handling

Additional to the regular stability output a polar diagram is plotted, which shows for each anchor chain angle the corresponding maximum anchor chain force which is still allowed by the anchor-handling stability criteria. See the example in Maxchain, section 13.2.3 on page 271, Polar diagram with maximum allowable anchor chain forces for a particular loading condition. By the way, for this option it is not required to select other than standard stability criteria.

16.2.1.3.10 Sight line

Here the criterion can be selected to which the line of sight must be tested.

16.2.1.3.11 Trim optimization

Here the speed and delta displacement for the resistance-trim graph can be set.

16.2.1.3.12 Front page

If the front page is selected for output (at the section 16.4.1.1 on page 315, Intact tab page), then here eight lines of free text can be given. In order to really include a line, the tick box just before it should be selected.

16.2.1.4 Product, temperature and density

If the temperature corrections functionality has been purchased then by double-clicking the name of a weight item, of type tank, in a loading condition, the following menu can be opened. This menu contains all the necessary parameters for processing temperature corrections.

Tank name
Same as the weight item, just for reference.

Include this tank in ullage report
If this compartment should be included in the cargo/ullage report then this field should be set to 'yes'.

Product (substance)
The name of the product, which will be used in the cargo/ullage report. If no substances have been defined yet then these can be created using the menu bar function [Substances].

Conversion table
For the calculation of the cargo weight of heated hydrocarbons, the following conversion tables are available:

- No temperature correction.
- Correction factor per degree. The 'Volume Correction Factor' is calculated according to the defined temperature and the correction factor per degree (coefficient of expansion).
- Volume Correction Factor. The 'Volume Correction Factor' can be defined directly.
- ASTM tables 54(A, B and C), 55, 53(A and B), 23(A and B), 5(A and B). The 'Volume Correction Factor' is determined according to the respective ASTM table.
- Nynas.

In case a conversion table other than *No temperature correction* is selected, this is recognisable in the weight item list by means of the yellow background colour of the name and weight of the weight item.

Temperature
The standard temperature is 15°Celsius. The volume is determined at this temperature. The actual temperature of the substance can be defined here.

Volume (not corrected for expansion)
This is the volume that is calculated according to the sounding, ullage or pressure for this weight item.

Density at 15°Celsius (in air)/(in vacuum)
The density of the substance at 15°Celsius can be defined here. If the density in air is defined, the density in vacuum is calculated automatically. These two densities are connected to each other and cannot be defined separately.

Correction factor per degree Celsius
This factor is used if the conversion table 'Correction factor per degree' has been selected, and calculates the volume correction factor.

Volume Correction Factor
This factor corrects the density at 15°Celsius of the substance for the actual temperature. This factor can be determined in a few different ways:

- This factor is defined manually, using conversion table 'Volume Correction Factor'.
- This factor is calculated with the correction factor per degree and the difference between the standard and actual temperature. The conversion table 'Correction factor per degree' must be selected.
- This factor is taken from one of the other conversion tables.

Temperature Expansion Factor
This factor corrects for the expansion of the tank at a higher temperature than 15°Celsius. This factor is calculated automatically and cannot be defined manually.

Density at {defined temperature} degrees
Density at 15°Celsius × Volume Correction Factor.

Residue On Bottom (ROB)
Volume of the residue which will be subtracted from the volume of the tank contents.

Density × Temperature Expansion Factor
Density at 15°Celsius × Volume Correction Factor × Temperature Expansion Factor.

Weight
The weight is calculated according to: Volume (not corrected for expansion) × Density at 15°Celsius × Volume Correction Factor × Temperature Expansion Factor.

16.2.2 Damage cases and settings for damage stability

Under the [Damstab] option in the loading condition menu, or the weight item list three sub options can be found:

1. Edit damage cases
2. Generate damage cases on basis of the extent of damage
3. Define stages of flooding

16.2.2.1 Edit damage cases

A maximum of 3000 damage cases can be defined. A damage case is a collection of compartments, as defined with Layout, which will be damaged simultaneously. After choosing this option a window appears where damage cases can be defined, and which is fully discussed in section 20.3 on page 397, Input and edit damage cases.

16.2.2.2 Generate damage cases on basis of the extent of damage

In general, damage cases are not chosen at will, they are derived from the extent of damage as laid down in rules and regulations instead. For that purpose PIAS contains a specialized functionality, which is discussed at section 20.4 on page 400, Generate damage cases on basis of the extent of damage.

16.2.2.3 Define stages of flooding

A maximum of ten stages of flooding can be defined. A stage of flooding is a percentage of the weight of the contents of the damaged compartments in the final, this is 100%, stage of flooding. 0% stage of flooding means the compartments are not flooded at all, so this is exactly as the intact loading condition. The 100% stage of flooding is included automatically and need not be defined explicitly. The stages here are the same as defined in Hydrotables, see section 10.2.10.4 on page 258, Define intermediate stages of flooding. Incidentally, there are many more considerations and possibilities regarding intermediate stages, which are discussed in chapter 21 on page 402, Internal flooding in case of damage, through pipe lines and compartment connections.

16.3 The common list of weight items

A loading condition is essentially nothing more than a set of weight items (including their properties, such as name and Center Of Gravity), Those weight items can be entered in a loading condition, however, in practice there ar emany loading conditions, which have quite some weight items in common. It is beneficial if these items can be managed commonly, which is offered by the 'common list of weight items'. This is a small database of weight items which can be used in multiple loading conditions. Everywhere these weight items are applied they are *the same*, which implies that a modification of such an item in one loading condition is directly transferred to the same item in another loading condition. So, the are not copied, the are being *referred to*.

The use of the common list is quite handy for weight items with a certain amount of communality, such as (obviously) the light ship (or its components) and deck loads or tank filling which occur in multiple loading conditions. However, its use is not obligatory, it is a tool, not more. In principle all weight items can directly be entered in the loading conditions, however, with the tiresome effect that later changes will have to be processed manually in all loading conditions. With the common list you save yourself these troubles, and reduce the change on errors.

16.3.1 The menu of the common list

The common list is simply a list of weight items, just as a loading condition has one. So, the input window of the common list is similar to that of a loading condition, as discussed in section 16.2.1 on page 307, Define/edit weight items. Differences are:

- A weight item can be of a certain type, for example 'tank'. Here an additional type is available: 'empty ship'. The reason of existence of this type is that the light ship is often available by numerous components, while a stability calculation should be printed with simply one single aggregated light ship (for which, by the way, also a name can be specified, as discussed in section 16.5.1 on page 320, Settings intact stability). With the aid of this type the program knows its components.
- The total light ship weight (and COG) is visible (in blue) on the row just above the first light ship component. With the aid of this row the light ship components are 'foldable', which means that by typing something there, all individual components are toggled between invisible and visible.
- Tanks and other components other than 'empty ship' and 'free weight item' can only be added to this list via the [Connect] option, as discussed in section 16.2.1 on page 307, Define/edit weight items, in a loading condition.
- The sub total line does not support any weight group modifications as described in section 16.2.1.1 on page 310, Fill tanks per weight group.

Also the available upper bar functions have all been discussed with the loading condition list, with the exception of:

- [Output]→[Print common list], which prints the common list.
- The [Import] function, which reads a text file of light ship components — which should be available in (AS↩ CII) format, with the *project file name*.klm extension. If the file contains N components then the file should contain N+1 lines, see the spec below. Please take note of the last line with zero's and the text 'stop'.

Weight(1)	VCG(1)	LCG(1)	TCG(1)	Aft boundary(1)	Fwd boundary(1)	Component name(1)
Weight(2)	VCG(2)	LCG(2)	TCG(2)	Aft boundary(2)	Fwd boundary(2)	Component name(2)
10.125	8.754	24.20	-0.52	20.1(2)	31.2	Tandwielkast
Weight(N)	VCG(N)	LCG(N)	TCG(N)	Aft boundary(N)	Fwd boundary(N)	Component name(N)
...	...	...	...	...	...	...
0	0	0	0	0	0	stop

16.4 The computations of stability and strength

In this section the merits will be discussed of the several Loading computations: intact stability, damage stability, longitudinal strength and torsional bending moments.

16.4.1 Output settings

16.4.1.1 Intact

At this tab several output options can be selected. From the listed components it can be specified whether they should be included in the output. If output for a selected option is still not produced, the reason might be that the option is not purchased or otherwise not available.

The option 'Front page' adds a front page to each stability calculation, which contains:

- A maximum of eight lines of user defined text, which can be given at the next tab page.
- A summary of the total weights for defined weight groups.
- Hydrostatics in upright position.
- Initial transverse stability.
- Drafts and trim.
- Statical angle of inclination for this loading condition.

With the option 'Moments of inertia of tanks' an additional page with the volumetric moments of inertia of the filled tanks is printed.

16.4.1.2 Strength

This tab page facilitates in the output options for the longitudinal strength calculation.

16.4.1.3 Damaged

At this tab several output options of the damage stability calculation can be selected. From the listed components it can be specified whether they should be included in the output. If output for a selected option is still not produced, the reason might be that the option is not purchased or otherwise not available.

16.4.1.4 Cargo report

Here you can specify whether a full or short 'Cargo/ullage report' is desired. See section 16.4.7 on page 318, Cargo/ullage report for more information.

16.4.2 Intact stability

To perform the stability calculations, it will be obvious that all the ship properties should be fully specified, not only hull shape and compartments, but e.g. also the light ship weight and openings. In addition, the following properties must also be set correctly in order to be able to make a complete stability calculation:

- Wind contours and wind pressures, as can be defined with Hulldef. See chapter 14 on page 273, Wind heeling moments for a discussion of the coherence of wind moment issues.
- The range of heeling angles should be chosen and entered in Config, see section 5.2 on page 45, Angles of inclination for stability calculations.
- The applicable stability criteria should be set, at the Loading settings, see section 16.5.6 on page 321, Definition and selection of stability requirements.

The intact stability output contains the following parts:

1. The weight list.
2. A table of upright hydrostatics and drafts.
3. The table of GZ and area under the GZ curve for the different angels of heel.
4. A plot of the GZ curve.
5. The conclusion where this condition is assessed against the minimum criteria.

However, this is the standard output, with the different settings of Loading it can be extended or limited significantly.

Attention

If the weather criterion from the Intact Stability Code (or a criterion with a similar nature) is the most critical one, it might occur that although in the assessment of a loading condition the maximum allowable VC← G is less than the actual VCG', the program reports that the loading condition complies. This might seem contradictory, but is not. The reason of this phenomenon is that according to the IS Code the rollback angle should be determined by a specific equation, in which the **uncorrected** VCG (that means, uncorrected for free surface effects) should be used.

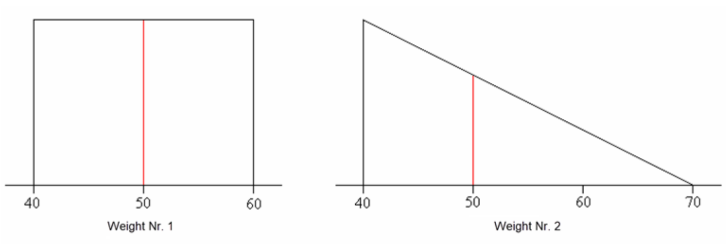
16.4.3 Longitudinal strength

Longitudinal strength comprises the longitudinal distribution of shear forces and bending moments, and, if set, deflection and elevation. A hogging moment is positive, a sagging moment negative. A positive shear force at a position indicates that behind that position the aggregated buoyancy is less than the aggregated weight. These computations are based on the longitudinal distribution of buoyancy and weight.

Each weight item is assumed to have a linear distribution between aft and forward boundary, in such fashion that the longitudinal center of gravity (LCG) equals the user-specified value. For example:

- A weight item of 100 ton, with and LCG at 50, and boundaries at 40 and 60 m.
- A weight item of 75 ton, with and LCG at 50, and boundaries at 40 and 70 m.

Have distributions according to the figure below.



Weight distributions of the two examples.

If the LCG is not within the middle 1/3 between the forward and aft boundary, then the distribution becomes partially negative. In some cases this is realistic, for example with a crane where the center of gravity of the load is even completely outside the boundaries, and sometimes it is unrealistic. Therefore it is verified at the longitudinal strength calculation whether the LCG is outside the 1/3 boundaries, and if so a warning is displayed. Also with tanks that are strongly curved in the longitudinal direction it might occur that the LCG exceeds these 1/3 boundaries. This is always considered unrealistic — the distribution of liquid in a tank can never give negative values in the weight distribution — so in these cases the aft or forward boundary is adapted in order to bring the LCG exactly to the 1/3 or 2/3 location.

If maximum allowable bending moments and shear forces have been defined (to do so please refer to section 16.5.7 on page 321, Definition maximum allowable shearforces and moments), the actual bending moments and shear forces will also be presented as percentages of the maximum values.

Then the conclusion is drawn, which represents whether all moments remain below the maximum allowable values. Please note that evaluation of actual moments against maximum allowable values is also performed at intermediate values in-between the read-out points. The maximum allowable value at these intermediate positions is found by lineair interpolation. This evaluation at intermediate points might affect the overall conclusion. So, it might occur that all read-out point values are less then the maximum allowable values, while the conclusion still is drawn that this loading condition does not comply. Somewhere between the read-out points the interpolated maximum will be exceeded in such a case.

There is also the option to calculate and output the so-called 'Envelope curve'. With this option, the envelope upper and lower limits are calculated of the occurring shear forces and moments based on the curves of the selected loading conditions. This envelope curve can be used to determine how strong the ship should be at certain longitudinal positions. Thus, it is really a design tool.

16.4.4 Deterministic damage stability

16.4.4.1 Plain deterministic damage stability

With this option for all combinations of selected damage cases, selected loading conditions and intermediate stages of flooding damage stability calculations will be produced. Suppose there are three loading conditions, five damage cases and two intermediate stages, then 3 x 5 x (2+1) = 45 damage stability calculations will be made. A regular output contains the following data:

Identification
Name of the damage case, the stage of flooding and the intact weight data.

Special points
If any margin line points have been defined (with Hulldef), then here the distance from the waterline to the margin line will be printed. If a non-watertight openings have been defined, the angle where this opening will be flooded is printed, as well as the distance from the waterline to this opening. Distances are not given in PIAS' standard system of axes; instead the distance is measured between point and waterline, measured *perpendicular* to the waterline.

Ingressed flood water, outflown intact content
For every damaged compartment, at angle of equilibrium, the following particulars are printed:

- The weight of the contents of the intact compartment (Wintact).
- The density of the contents of the intact compartment (SWintact).
- The weight of the contents of the damaged compartment (Wdamag).
- The density of the contents of the damaged compartment for this stage of flooding (SWdamag).

Table of stability parameters at larger angles
The first line indicates to which side the vessel is heeling, SB or PS. The module automatically determines the side with the worst stability, however, this can also be configured otherwise, please refer to section 5.1.8 on page 44, Calculate damage stability with a heeling to for this setting. Subsequently, for every defined heeling angle the following particulars are printed:

- Heeling angle (in degrees).
- Displacement excluding the possibly spilled cargo and including the weight of the flooded compartments.
- Draught at $L_{PP}/2$.
- Total trim on perpendiculars.
- Righting lever (meter), defined as the ratio of the righting moment and the intact displacement. So the righting lever is corrected to constant displacement (the intact displacement).
- The dynamic stability up to this heeling angle (area under the righting lever curve) in meterradian.

When instead of, or next to, these columns the text 'The vessel sinks' is printed, this indicates that a non-watertight or non-weathertight opening is submerged at an angle smaller than the statical angle. Or that the total weight exceeds the total buoyancy.

Prime parameters of the damaged stability

- Statical angle of inclination (degrees).
- Maximum righting lever (meter).

Verification against the criteria
Assessment of this damage case against the damage stability criteria, as they have been defined in section 16.5.6 on page 321, Definition and selection of stability requirements.

GZ-curve
A graph of the righting lever curve.

16.4.4.2 Calculate cross-flooding times

Attention

This is a calculation using the old method, see section 21.1 on page 402, Background from tools for ship-internal connections in PIAS for the difference between old and new. The option discussed here can be used to calculate the time required to flood a single compartment which is connected to seawater through a single connection. In the new method, for complex systems of multiple compartments, connections and components (such as vent caps), the overflow time can be calculated, integrated with a calculation of damage stability during the flooding process.

With this option the time can be calculated which is needed to let a compartment be flooded through a cross-flooding arrangement. The purpose of this option is similar to the method of IMO MSC.362(92) (formerly I↩
MO res. A.266), with the difference that the IMO resolution gives an approximation method, while this PIAS option applies a stepwise calculation (a time-domain simulation) — based on Bernoulli's law — which yields in general more accurate results. This application is covered by section 4 van MSC.362: "As an alternative to the provisions in sections 2 and 3, and for arrangements other than those shown in appendix 2, direct calculation using computational fluid dynamics, **time-domain simulations** or model testing may also be used". The parameters which are necessary for this option must be specified at 'complex intermediate stages of flooding', please refer to section 21.3 on page 408, Complex stages of flooding (before 2023).

16.4.5 Torsional moments

The torsional moments calculation is always performed for the upright vessel. Because of round-offs in the calculation and because the vessel might have an initial heeling in the loading condition, a residual moment at the end of the vessel might occur. To correct this the residual moment is linearly distributed over the total length of the vessel. The option to make torsional moment calculations for a grounded vessel has not been implemented.

The generated output of the calculation shows successively:

- One or more pages with an overview of all weight items, including name, transverse center of gravity (TCG) and fore- and aft boundary. Dependent on the Loading setting setting 'print parts of light ship summarized' (see section 16.5.2 on page 320, Settings longitudinal strength) the light ship items are printed summarized or separately.
- One or more pages with intermediate results of the torsional moment calculations. With an increment as set in the setting 'output increment', for each longitudinal position the following is calculated; weight moment (tonm/m), buoyancy (tonm/m), torsional moment (tonm), the torsional moment area (tonm^2) and the absolute torsional moment area (tonm^2).
- A page starting with a few lines displaying position and value of the maximum occuring torsional moment, torsional area and absolute torsional area. If maximum allowable torsional moments have been defined (read-out points, see section 16.5.7 on page 321, Definition maximum allowable shearforces and moments for that), at different positions of the hullform a table is printed. In this table the actual torsion moments are printed as a percentage of the maximum allowable value. So for every location the actual torsion moment and the actual moment as percentage of the maximum allowable is printed. If this percentage is lower than 100% the value is printed in green, otherwise in red. If a maximum allowable torsional area is set, see the last setting of section 16.5.7 on page 321, Definition maximum allowable shearforces and moments) then the actual absolute torsional moment area is tested against the maximum allowable value. Finally a conclusion is printed which indicates whether all torsional moments are lower than the maximum allowable. Here the same remarks applies as in the last paragraph of the discussion of the calculation of longitudinal strength, just prior.
- A graphical presentation of actual and maximum allowable torsion moments. In a sideview of the vessel the actual torsional moment is shown presented by green curve. The red curve shows the maximum allowable values.

16.4.6 Sounding table

Delivers a report where for all tanks — and then for all measuring devices, i.e. sounding pipes and pressure sensors, in the tank — at trim zero and angle zero, the current sounding, ullage and/or pressure is calculated based on the current volume.

16.4.7 Cargo/ullage report

This option allows you to print an overview of all onboard cargoes, including their weight, temperature effect, sounding and ullage etc., see example below. This list includes only those tanks of which the detail particulars (as discussed in section 16.2.1.4 on page 312, Product, temperature and density), at the second row 'Include this tank in ullage report' is switched on. Before this report is created some more questions might be asked, such as the *Bill of Lading* weight. Note that the weights of the *Bill of Lading* are only useful if the 'Product (substance)' is defined.

CARGO, SOUNDING AND ULLAGE REPORT
M.v. Exempli Gratia

28 Sep 2017 15:50:47

Trim = 1.000 m (trim by bow)
Draft from baseline on FPP = 4.100 m
Draft from baseline on APP = 3.100 m
Angle of inclination = 1.000 degrees (to SB)

Port of loading / discharge: Rotterdam
Berth: Alexander
Voyage number: 354

Tank	Product	Ullage	Sounding	Press.	Temp.	Volume	TEF	ROB	Obs.Volume	Method
30 GO PS	Gas Oil	3.826	3.386	3214	55.0	33.393	1.15522	0.100	38.476	MANUAL
31 GO SB	Gas Oil	3.848	4.348	3819	50.0	52.829	1.00423	0.200	52.852	MANUAL
43 DB 4 HFO PS	Heavy Fuel Oil	10.086	1.082	1101	50.0	200.000	1.05406	0.000	210.813	MANUAL
44 DB 4 HFO SB	Heavy Fuel Oil	9.907	1.262	1029	60.0	150.000	1.06954	0.000	160.431	MANUAL
46 HFO DAY PS	Heavy Fuel Oil		9.842		50.0	20.000	1.00082	0.500	19.516	MANUAL
50 LO CIRC CL	Lub Oil	1.066	0.926	790	80.0	10.000	1.00687	0.250	9.819	MANUAL

Tank	Table	Corr./degr.	VCF	Volume 15	Density 15 Vacuum	Density 15 Air	Weight Vacuum	Weight Air
30 GO PS	Nynas		0.9702	37.332	0.9211	0.9200	34.383	34.345
31 GO SB	Nynas		0.9739	51.474	0.9211	0.9200	47.408	47.356
43 DB 4 HFO PS		0.001000	0.9641	203.253	0.9771	0.9760	198.589	198.370
44 DB 4 HFO SB		0.001000	0.9537	153.008	0.9737	0.9726	148.985	148.822
46 HFO DAY PS	ASTM55		0.9782	19.091	0.9921	0.9910	18.940	18.919
50 LO CIRC CL	ASTM54B		0.9493	9.321	0.9011	0.9000	8.398	8.389

Volume : Volume corrected for list and trim
Obs.Volume : "Observed" volume: corrected for tank expansion (TEF)
Volume 15 : Volume at 15 degrees (corrected for cargo expansion)
Density 15 : Density at 15 degrees Celsius
TEF : Temperature Expansion Factor
ROB : Residu On Bottom
Table : Table used for temperature correction
Corr./degr. : Volume correction per degree Celsius
VCF : Volume Correctie Factor

Product	Density Air	Mean Temp.	Observed Volume	Volume 15	Barrels	Weight Vacuum	Weight Air	B/L Weight	Diff. %
Gas Oil	0.92000	52.1	91.328	88.805	558.5	81.791	81.701	81.000	0.86
Heavy Fuel Oil	0.97538	54.1	390.760	375.352	2360.7	366.514	366.111	370.000	1.06
Lub Oil	0.90000	80.0	9.819	9.321	58.6	8.398	8.389	8.250	1.65
Totals :			491.907	473.478	2977.8	456.703	456.201	459.250	0.67

For stabilised crude oil K0 = 613.9723 and K1 = 0 (for metric units)

Shipper / Receiver (On behalf of) the master

....

Example of a cargo/ullage report.

16.5 Loading project settings and tools

Loading project settings

1.	Settings intact stability
2.	Settings longitudinal strength
3.	Settings deterministic damage stability
4.	Settings polar diagram anchor handling chain forces
5.	Definition of weight groups
6.	Definition and selection of stability requirements
7.	Definition maximum allowable shearforces and moments
8.	Define sections for sketches of tank arrangement and damage cases
9.	Definition of mooring forces
10.	Define cross sectional moments of inertia (for calculating sagging)
11.	Compute tables of shear forces and moments of buoyancy
12.	Settings for ballast advice

16.5.1 Settings intact stability

Name of 'light ship' in loading conditions
It occurs quite often that the common list of weight items contains quite many components of light ship. Those need not all to be included in the loading conditions, after all that would require many pages. So, only the total weight (and COG) is included in a loading conditions, however, that items still requires a name. That can be given here.

User-specified scale of GZ-plot
The output of a stability calculations contains a GZ-curve. Its scale is selected by the program, in order to fit properly on a page. With 'yes' at this option a user-defined scale will be used (which can be entered one line lower: if you desire e.g. a scale of 1/5, then 5 should be entered there). All GZ-curves of all loading conditions will then be printed on the same scale.

Print moments in the list of weight items
With 'yes', in the intact stability output besides the weight and COG's also the moments will be included.

Print % of filling and density in weight list
If this option is set to 'yes' in the intact stability output each weight item will be printed including the percentage of filling and the density of the tank contents.

Print the deadweight in the list of weight items
With 'yes' the deadweight (and the COG of the deadweight) will be included in the output of intact stability calculations. The deadweight is total displacement minus the light ship weight.

Connect points of GZ-curve with straight line segments
A GZ-curve is usually drawn as nicely curved. And that will usually be her natural shape: a smooth curve drawn through the points formed by the calculated GZ values at the chosen angles of inclination. Sometimes, however, the trend of these points is not so smooth, e.g. when it contains a discontinuity as result of submerging an (internal) opening. The "smooth curve" method, however, does not know about these discontinuities and does its best to draw a curved line through the points, which will indeed be nicely curved, but in exceptional cases may also show a considerable loop. With this option set to 'yes' no curved curve is drawn, but the points are connected by straight line segments instead. In general slightly less accurately inbetween the points, but without curls at discontinuities. The reduced accuracy can be compensated for by having more angles calculated.

16.5.2 Settings longitudinal strength

Output increment
Enter the longitudinal interval in meter, which will be applied when printing tables of shear forces, moments etc.

Including calculation of sagging
Indicates whether sagging (as well as deflection) should also be computed and printed. If yes, then an additional row appears where Young's modulus should be given — as a rule 21000000 ton/m^2 for ordinary mild steel. The moment of inertia of the midship section structure must also be specified, which can be done at the Loading project settings, see section 16.5.10 on page 322, Define cross sectional moments of inertia (for calculating sagging).

Print parts of light ship summarized
By default all light ship items, which constitute the total light ship, are printed on the output. Here it can be set to print only one single, totalized light ship weight. just as with the intact stability calculations.

Weight item boundaries change with LCG
By default, all dimensions of each weight item have to be given, including LCG and aft and forward boundaries. If this particular option is switched on, then a little trick is applied, then the boundaries are shifted the same amount as the LCG, when the latter is modified.

Maximum allowable torsion area (Tonm.m)
Classification may impose a maximum on the maximum area under the torsion moment curve. Such a maximum, which e.g. is called AST by Germanischer Lloyd, can be defined at this option.

16.5.3 Settings deterministic damage stability

In Config the *general* settings for damage stability can be selected (see section 5.4 on page 46, General settings damage stability), while here settings specifically for deterministic damage stability (which, after all, is computed by Loading) can be given.

User-defined scale of GZ-curve in damaged condition
By default, the GZ-plot of all damage conditions will be determined by the program so that it fits nicely on the paper, but if this option is answered with 'yes' a fixed scale can be specified by the user at the next option. In this case each GZ curve will be plotted on the same scale.

Which scale to use for GZ-plot (give X in scale 1:X)
The denominator of the user-defined scale should be given here, e.g. 5 for a scale 1:5.

Sort openings and margin line points in output
In the output of damage stability calculations all (in Hulldef) defined openings or margin line points are included in the output, in combination with angles of downflooding or distances to the waterline. With a 'no' for this option the output sequence equals the definition sequence. With a 'yes' they are sorted in increasing distance to the waterline (so, the most severe point comes first).

Maximum number of openings and margin line points to print
A maximum number of to be printed points can be given if *Openings and margin line points* are sorted.

16.5.4 Settings polar diagram anchor handling chain forces

With the general ship input data in Hulldef, paramaters can be given for maximum allowable chain forces for anchor-handling vessels, zie section 7.2.1.10 on page 170, Anchor handler particulars. In that Hulldef menu a few sets of criteria can be selected, either NMD2007, BV2014 of IS code 2020. If the BV type is chosen, the polar plot can be produced for a user-specified angle of β (please refer to section 13.1 on page 268, Input of specific ship data for its definition) which can be set in this menu.

16.5.5 Definition of weight groups

Purpose and use of 'weight groups' is discussed in section 20.1 on page 395, Weight groups.

16.5.6 Definition and selection of stability requirements

The *modus operandi* of the stability criteria system is discussed in chapter 15 on page 275, Stability criteria for intact stability and damage stability, and the specific menu to enter stability criteria in section 15.1 on page 276, Manipulating and selecting sets of stability criteria. In this menu it can be indicated which of the stability requirement sets is valid for the assessment of the intact stability. This is then used for all loading conditions, but it is also possible to assess some conditions against other requirements, as described in paragraph 16.2.1.3.4 on page 311, Stability requirements.

16.5.7 Definition maximum allowable shearforces and moments

In this overview screen all created longitudinal strength criteria are displayed and how many points for the respective boundary lines have been specified. By double-clicking on a boundary line column one can modify the corresponding boundary line.

The option standard criteria allows for easy switching between different strength criteria for multiple loading conditions, if paragraph 16.2.1.3.5 on page 311, Strength have been selected for standard strength criteria.

The defined values are used in the assessment of the longitudinal strength and torsional moments calculations.

With option [Output] the input values can be printed to paper.

The percentages, which are shown in the output, are normally calculated from the zero line, but it can happen that the upper and/or lower limit are entirely or partially below or above the zero line. In such a situation the zero line is no more insight and a fictitious zero line is determined halfway between the given upper and lower limit. This allows the percentages to be determined as before, but in relation to the fictitious zero line instead of the actual zero line. It is possible that the lower limit is above the zero line and that the upper limit, which is also above the zero line, starts at a length position greater than the lower limit. In this situation, no percentage can be determined, because determining the fictitious zero line is dependent on upper and lower limit.

16.5.7.1 Definition maximum allowable shearforces and moments

In this menu the maximum allowable boundary line can be defined as a function of the length.

The following points must be taken into account when defining the maximum allowable boundary line:

- The length must be ascending.
- You can define a jump, but it can only consist of two longitudinal positions.

- That, depending on the type of boundary line, negative and positive values must be taken into account.

Using the menu bar function [Lininterpol], intermediate longitudinal positions can be interpolated linearly. The used abbreviations used in this menu are:

- Long.pos: longitudinal distance from App [m].
- Frame no.: frame number.
- Value ton: maximum allowable boundary value, shear forces [ton], bending- and torsion- moments [tonm].
- Value kN: maximum allowable boundary value, shear forces [kN], bending- and torsion- moments [kNm].

16.5.8 Define sections for sketches of tank arrangement and damage cases

Purpose and use of 'sections tank arrangement' is discussed in section 20.2 on page 396, Sketches of tanks, compartments and damage cases.

16.5.9 Definition of mooring forces

PIAS contains specialistic functionality for objects which are moored with multiple anchors. It finds equilibrium in the horizontal plane, while the vertical chain forces are included in the stability computations. In the menu the relevant chain parameters can be given. However, this menu is only available for those who have actually purchased this functionality.

16.5.10 Define cross sectional moments of inertia (for calculating sagging)

For the purpose of the calculation of the longitudinal sagging — such as is included in the longitudinal strength calculation if the *including calculation of sagging* option is switched on, see section 16.5.2 on page 320, Settings longitudinal strength — the moment of inertia of the midship structure must be specified. That can be done here, in fact, for multiple cross sections that can be specified, so that the effect of the longitudinally varying moment of inertia on the deflection can be accurately taken into account. If the variation in moment of inertia can be excluded than it will suffice to enter the midship moment of inertia only. To be exact, in the two colums you specify:

- The longitudinal distance from APP in meter.
- The moment of inertia at that location, in meters^4.

16.5.11 Compute tables of shear forces and moments of buoyancy

This is a bit of an outdated option, however, it still might occur that a longitudinal strength manual is required which also contains a scheme for the manual calculation. In that case tables of shear force and moments of only the buoyancy might be needed. With this option (if purchased) those tables can be produced. Calculation will be performed for user-specified drafts, for all longitudinal positions for which maximum allowable bending moments and shear forces are defined. After this option has been selected the following data can be specified in a menu:

- First and last draft, and draft increment.
- Trim for which the tables will be calculated.
- Draft step for determination of draft correction (correction = ($\text{value}_{\text{draft+draft step}}$ - $\text{value}_{\text{draft}}$) / draft step).
- Trim step for determination of trim correction (correction = ($\text{value}_{\text{trim+trim step}}$ - $\text{value}_{\text{trim}}$) / trim step).
- Name of the outputfile (default extension is .LST from Longitudinal Strength Tables).

The output is not designed for printing directly to paper, after all, some post-processing will be done anyway.

16.5.12 Settings for ballast advice

See section 17.7.2 on page 377, Ship-specific settings for ballast advice.

16.6 Generation of loading conditions for simulation RoRo operations

This facility is intended for simulation of RoRo operations, and is discussed in section 17.3 on page 360, Generation of loading conditions for simulation of Ro-Ro operations.

16.7 Combined output

This option can be used to perform multiple kind of calculations in one single run.

16.7.1 Definition of output sequence

In this menu can be specified which computations should be made and printed. For each type of computation also a page numer and chapter name can be given, which will be printed at the bottom of each page. The left column, *selected* specifies whether the output of that line is actually included in the output to be produced. The menu bar contains the function [layout], which can be used to choose from two kinds of output sequences. The first option is [Type of calculation], where first for the first type of calculations all loading conditions are printed, and then for the second type of calculations all conditions etc. The second option is [Loading condition], where first for the first loading condition all types of calculations are printed, then for the second condition all types etc. The chapter name as given in this menu is only used with the 'Type of calculation'.

16.8 File and backup management

Backups of the Loading data can be made and restored here. Here is also the option 'Quit module without saving the data'. See for the details section 2.9 on page 15, Data storage and backups.

Chapter 17

Loading tools

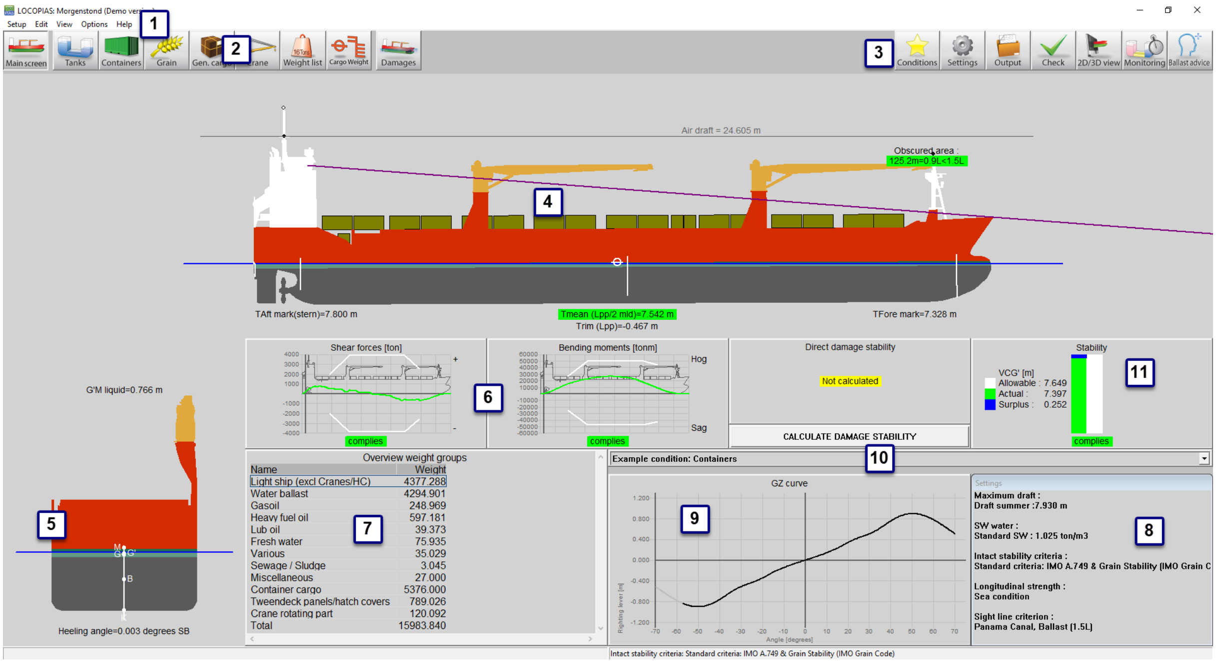
Loading contains a number of tools for particular types of loading. Because the explanation on these tools would get lost within the already extensive Loading chapter, in this chapter their modus operandi will be discussed separately. The addressed topics are:

17.1 Graphical User Interface for Loading

Here appears the Graphical User Interface (GUI), which can be considered as the central command window of Loading, and an example of which is shown in the figure below. This GUI is not strictly indispensable. It is very well possible to navigate through Loading with all functions and menu options. It has been proven to be rather well-arranged and user-friendly though. Because this GUI is similar to the main window of the PIAS-related LO↩ COPIAS loading software, the corresponding parts of the LOCOPIAS manual have been pasted into the remainder of this Loading GUI section. So it might be possible that an occasional "LOCOPIAS" pops up in the text. But since the function under consideration is identical to that of PIAS that difference in brand name is not relevant. The same argument applies to the loading tools as discussed in chapter 17 on this page, Loading tools.

17.1.1 Main window layout

A typical example layout of the main window is shown below, with an explanation of the labeled elements right below that.



Layout of the main window.

1 Menu bar

Basic functionalities are accessible through the menu bar, see Menu bar.

2 Module buttons

These tool bar buttons provide quick access to the main window and available cargo modules to load specific types of cargo.

Attention

The modules can be opened after or next to each other, see the explanation for the 'Multi-module' option.

3 Main window buttons

These buttons allow manipulation of Conditions, Settings, Output, Check and 2D/3D View.

4 Side-view

Shows the actual wind contour, drafts, actual waterline, line of sight and air draft.

5 Cross section

Shows heeling angle and initial stability (G'M).

6 Compliance windows

These windows indicate compliance with the criteria for the current loading condition. Click on a window for detailed information.

7 Overview weight groups

A summary of total weight per weight group.

8 Settings window

Shows current settings. Double-click on a setting to change it, or go to the [Settings] (discussed on page 328) dialog window by clicking the button [Settings].

9 GZ curve

Shows the GZ curve of the specific condition.

10 Drop-down list box

Shows the selected loading condition and you can select another condition.

11 Intact stability diagram

Indicates whether the vessel complies with the intact stability criteria and to what extent. Though the values for the Actual VCG' and allowable VCG' are calculated by LOCOPIAS in a manner which is correct and normally accepted by class societies, these values have not been checked by Lloyds Register and should therefore only be used as guidance!

Note

Depending on your installation, some of these elements may not be available.

17.1.1.1 Menu bar

The menu bar at the top of the main window (item 1) gives access to the following functions:

[Setup]→[Print Options]
: Select output device. Besides preview/clipboard, the default system printers are listed and can be selected here.

[Edit]→[Edit Weight Groups]
: Weight items can be grouped in so-called weight groups, were a weight group is a category of a particular content, such as 'diesel oil' or 'fresh water'. The weight groups are managed from this location in the program. The user can add, modify and delete weight groups themselves. When deleting a group, a check is made to see if there are still weight items of that group, and if there are, a notification is given and it is better not to delete the group. There are some default weight groups that are fixed in the program and cannot be changed or deleted. Editable properties are:

- The *name* of the weight group.
- The *hatching* type which is used when hatching or filling in the compartments in tank sketch plots.
- The *group color*, which is the color representing this weight group, and which is used in plots, and also as background color in text windows if the last column of this weight group is set to 'yes'.
- The *text color*, which, if the last column is set to 'yes', specifies the foreground color in textual overview windows of the texts which belong to this weight group.
- *In table*, which indicates whether the weight group color should also be used in overview tables of compartments and weight items.
- *Print summ.*, which indicates whether in the output only the subtotal should be printed. The calculation is based on all weight items though.

[Edit]→[Edit cross sections tank graphics]
: Go to this menu to add or edit cross sections and views of the tanks. These sections and views are automatically added to the output of intact stability calculations.

[Edit]→[Edit cross sections stowage plan]
: Go to this menu to add or edit cross sections and views of the stowage plan.

[View]→[3D View]
: See section 17.1.5 on page 329, 2D/3D View

[Options]→[Select stability criteria]
: See section 17.1.8 on page 330, Check

[Options]→[Export data via XML]
: Exports the current loading condition to an XML file which can be used to exchange data with third party software.

[Options]→[Environmental conditions]
: Gives the ability to simulate running aground, or check the stability in wind and/or waves.

[Options]→[Multi-module]
: This option allows you to set whether only 1 loading module is active at a time, or several side by side. The latter is especially useful if multiple screens are connected to the computer. If the multi-module option is on, the loading condition can be adjusted in different screens. The modules can then be opened only from the main screen.

[Help]→[Help reader (F1)]
: Opens this help reader.

[Help]→[Manual]→[Ship-specific data and test conditions]
: Opens the booklet containing the Ship-specific data and test conditions.

[Help]→[About LOCOPIAS]
: Opens a window with relevant data with regard to the LOCOPIAS program as well as the license conditions.

[Help]→[Not purchased]
: Shows a preview of modules which have not been purchased.

[Help]→[Enter activation code]
: Give an activation code here for modules purchased afterwards. At the moment this is only possible for the tank measurement system module for specific sytems. Please contact SARC for further details.

17.1.2 General approach

In general, you can use the following steps to define a loading condition and perform the required calculations. Please note that this workflow is *just one* way to get you started, it is not the only possible way to use LOCOPIAS. All actions can be performed in random order and frequency but it is important to check the compliance with all the appropriate criteria after a change in the loading condition has been made The functionalities will be elaborated further in the remainder of this chapter. This example starts at the main window.

Select the [Conditions] button and create a new condition. When LOCOPIAS is opened for the first time, the main window shows a preprogrammed example condition. By creating a new condition, you start with a preprogrammed default condition.

Click the [Settings] button and adjust the settings according to your situation. By adjusting the settings to the current situation before loading your cargo, useful feedback can be received during configuration of the loading condition. Settings are applicable for the current loading condition.

Go to the [Tanks] module to modify the contents of consumables e.g. fresh water, fuel oil, lubricating oil.

In the [Weight list], miscellaneous supplies, e.g. crew, provisions and stores can be entered.

Select the appropriate modules for your cargo type and define your cargo.

Open the [Tanks] module again. When all cargo is loaded, the floating position can be optimized by adding water ballast.

The [Check] button provides a quick check of stability and strength at any moment during this process.

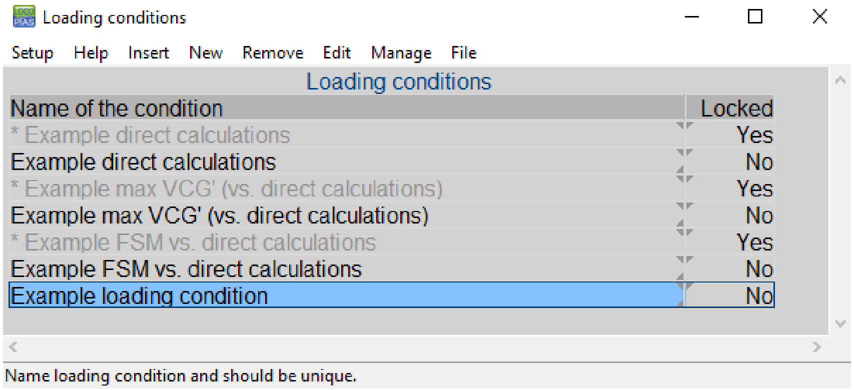
Press [Output] to perform calculations and generate output on screen or on paper.

Press [2D/3D view] to view a three-dimensional representation of your vessel, if available.

Press [Monitoring] or [Update Monitoring] to switch on the monitoring functions in LOCO↩
PIAS, if available.

17.1.3 Conditions

By pressing the [Conditions]-button, the loading conditions menu, as shown in the figure below, will appear. In this window the defined loading conditions are displayed and can be managed. You can create a new loading condition and you can delete, rename, copy/paste or export existing conditions. To modify a loading condition, select a condition and double click on it or press the <enter> key. The main window will now reflect this loading condition.



Select or create a loading condition.

New loading condition

1. Click [New].
2. Enter a new (unique) name for your loading condition.

The new condition is a preprogrammed default condition.

Delete loading condition

1. Select a loading condition.
2. Click [Remove].

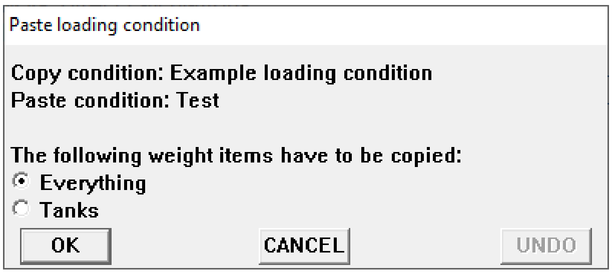
Rename loading condition

1. Click on a loading condition and press the functionkey <F2>.
2. Enter a new (unique) name.

Copy/paste a loading condition

1. Click on a loading condition, and press the [Edit]→[Copy row].
2. Now select the condition to copy to and click [Edit]→[Paste row].

Copy a loading condition and paste it over another loading condition to create a loading condition that has the same properties. If a specific module has been purchased you can choose to paste the complete condition or just the cargo defined with the specific module. The newly pasted condition will appear on the main window, as shown in the figure below.



Choose the data that have to be copied.

Import/export of selected loading condition

Import/export allows transport of data from one LOCOPIAS to another for the same vessel and same version.

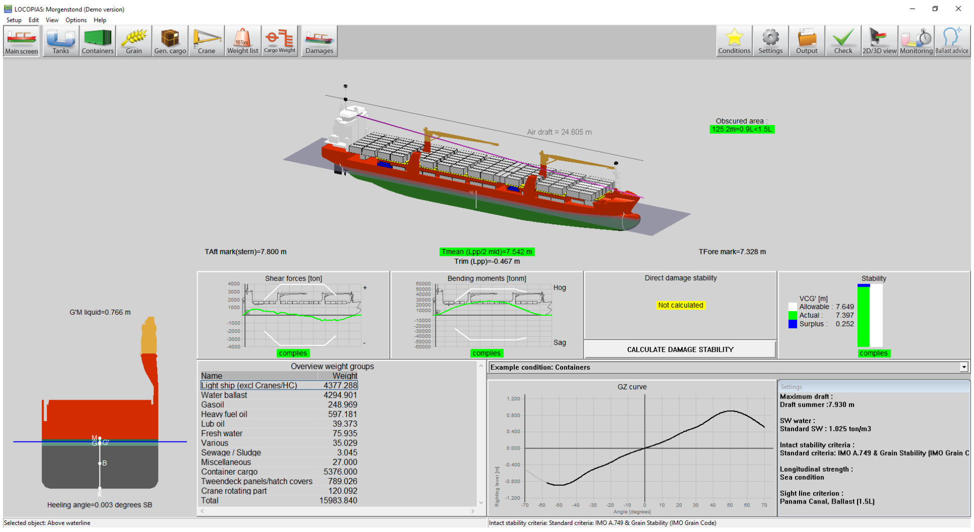
1. Press the [File]→[Export] to write the selected loading conditions to file.
2. Press the [File]→[Import] to select a file of exported loading conditions, to import them into the active version of LOCOPIAS.

17.1.4 Settings

See section 16.2.1.3 on page 310, Settings per loading condition.

17.1.5 2D/3D View

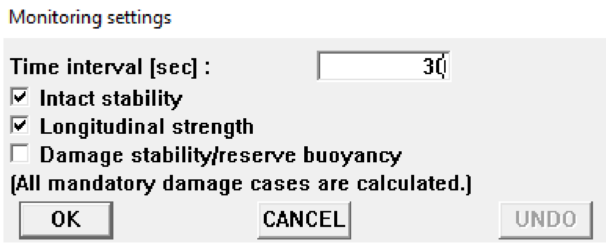
This button is only available if a 3D model of the vessel is available. It toggles between side view and three dimensional view of the hull and cargo. By choosing the menu [View]→[3D View] it is possible to edit materials, colors, and light effects of the 3D representation. The 3D image can be saved to file or sent to a printer.



Main window with 3D view switched on.

17.1.6 Monitoring

This option is only available when purchased and a connection with a tank gauge system is available. After selecting the icon for [Monitoring] a settings popup-window, as seen below, will appear. Here you can enter the time interval which will be used for reading the tank data, calculating the intact stability, longitudinal strength and damage stability (which is available and selected) and updating all data in the main screen. As long as the monitoring mode is active, it is not possible to edit loading conditions. This mode can be stopped by selecting the icon for monitoring again.



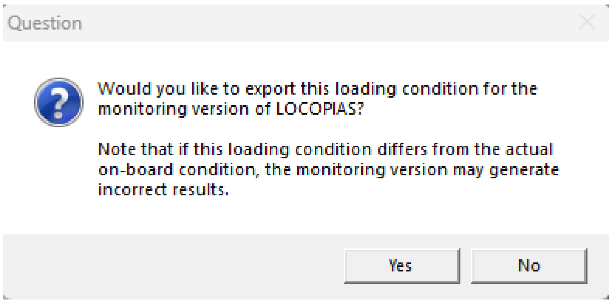
Settings for monitoring.

17.1.7 Update Monitoring

This function is only available if 'direct monitoring' is delivered with LOCOPIAS. 'Direct monitoring' is an additional feature of LOCOPIAS that can be configured to continuously send calculation results to other software, via a suitable interface. These results may include including tank fillings, weight items, results of (damage) stability and longitudinal strength, etc.

With this function the actual loading condition can be exported to update the loading condition as used in a second instance of LOCOPIAS, running in 'direct monitoring' mode. That instance LOCOPIAS will read the updated loading condition and recalculate results. Thus, monitoring need not be interrupted to define changes cargo, bunkers, or other weights on board or calculation settings.

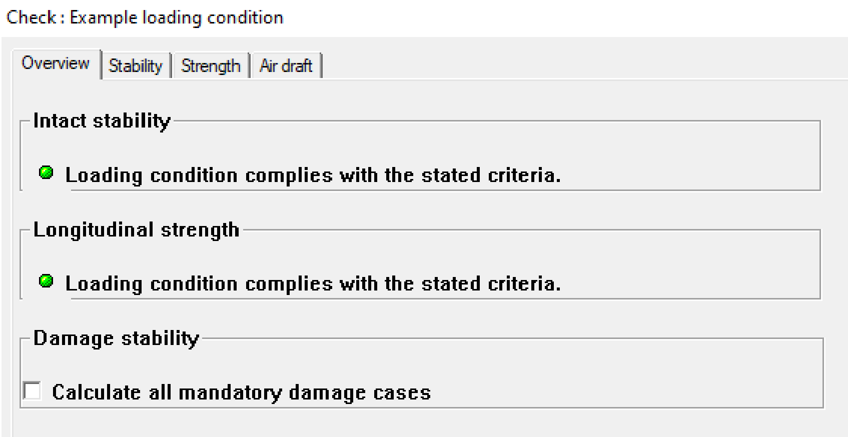
Details of the configuration of 'direct monitoring' and the interface used are described in the ship-specific documentation.



Update monitoring message.

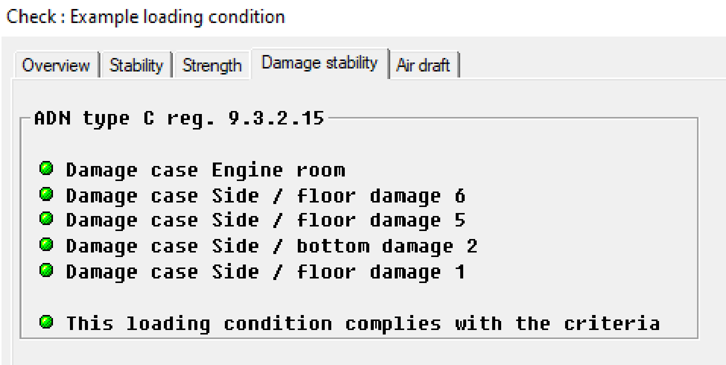
17.1.8 Check

Click the [Check]-button to check that the loading condition complies with the (damage) stability and strength requirements. After clicking the [Check]-button, a window opens with several tabs: overview, stability, strength and damage stability, if applicable. Compliance with the requirement is indicated by the color of the bullet (complies = green, does not comply = red). If, for instance, the overview shows a red bullet under intact stability, the corresponding tab provides more information as to the reason for non-compliance. Note that when the vessel operates under more than one classification society, the set of damage stability criteria applicable to the loading condition can be set via the menu bar item [Options]→[Select stability criteria]. The intact stability criteria can be set per loading condition via section 17.1.4 on page 328, Settings.



Check window.

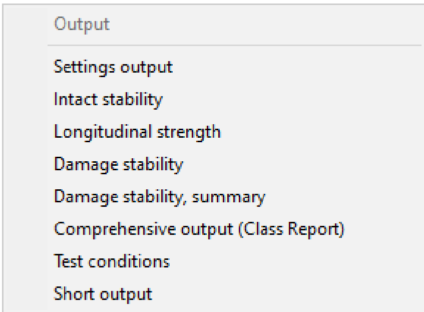
When you want to check all mandatory damage cases (type-3) select: ‘Calculate all mandatory damage cases’ and press OK. Now the Check window has generated a new tab called *Damage stability*. Here you can check whether the damage cases complies with the criteria (complies = green, does not comply = red).



Check window damage stability tab.

17.1.9 Output

You can use ‘Output’ to perform full calculations and to make a printout. If the selected printer is ‘preview/clipboard’ the output will appear on screen.



Output menu.

The following output options can be available in your version of LOCOPIAS:

Output settings
Select which data is to be printed in the full output. See Output Settings.

Intact stability
Standard format output of intact stability calculations with an overall conclusion for compliance with applicable stability requirements.

Longitudinal strength
Output of longitudinal strength calculations with an overall conclusion for compliance with selected allowable bending moment and shear force requirements.

Torsion moments
Output of torsional moments calculations with an overall conclusion for compliance with defined maximum allowable torsion moments.

Damage stability mandatory damage cases (type 3)
Full output of damage stability calculations of the mandatory (type 3) damage cases with an overall conclusion for compliance with applicable stability requirements.

Damage stability mandatory damage cases (type 3), summary
Output of damage stability conclusions of the mandatory (type 3) damage cases.

Damage stability selected damage cases
Full output of damage stability calculations of the selected damage cases with an overall conclusion for compliance with applicable stability requirements.

Damage stability selected damage cases, summary
Output of damage stability conclusions of the selected damage cases.

Comprehensive output (Class Report)
Output of the standard format of all available calculations (including mandatory damage cases, if applicable) with a common conclusion in accordance with the requirements. This 'Class report' should be printed to see if full compliance with all required criteria of the vessel is met. A hardcopy or digital copy should be saved for future reference. This 'Class report' should be printed with all the available and relevant output settings."

Short output
A summary of the loading condition and a conclusion.

Sounding table
Output for all measuring devices, for every tank, in the loading condition.

Cargo/ullage report
An overview of all onboard cargoes, including their weight, temperature effect, sounding and etc.. In this list only those tanks are included of which 'Include this tank in ullage report' is switched on.

Stowage plan
The stowage plan can be shown on screen or printed to paper. All cargoes from all available modules, except grain and bulk, are printed.

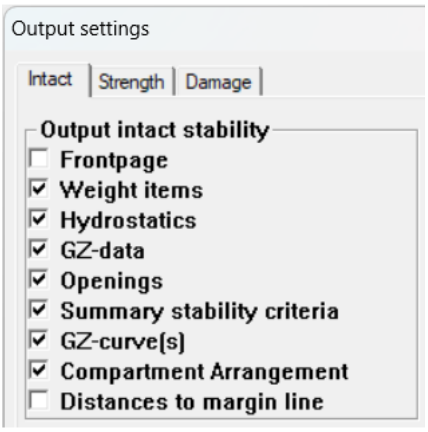
17.1.9.1 Output Settings

In 'Output Settings, one can select which data is printed in the full output. The output settings can be made for 'intact stability', 'strength' and/or 'damage stability' whatever is applicable for the type of vessel. For a full rapport, validating the compliance with the applicable criteria one should include all output, except for the 'frontpage' in intact stability. 'Frontpage' allows the user to print additional data such as information about the cargo etc. This additional information can be given at section 17.1.4 on page 328, Settings.. Not all ships will have a margin line defined and in such cases this could be left out.

Examples of the output can be found in Examples of output.

Available settings:

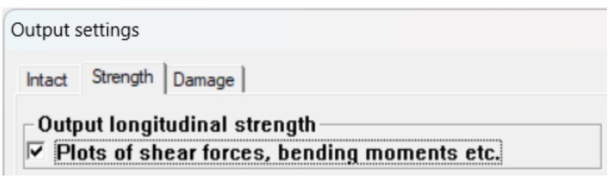
Intact stability



Output settings intact stability.

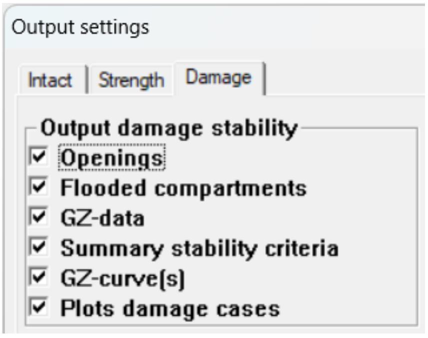
If one wants to include a frontpage, the content of this frontpage can be given at section 17.1.4 on page 328, Settings.. Distances to margin line will not be applicable to all vessels.

Strength



Output settings strength.

Damage stability



Output settings damage stability.

17.1.9.2 Examples of output

Below, examples of output for: Intact Stability, Longitudinal strength, Damage stability, Damage stability (summary) and Damage stability (summary DNV)

17.1.9.2.1 Intact Stability

Example of print-out of intact stability. Actual output and/ or results may differ according the choosen calculation, and/ or output settings.

Preview (1/6)

Quit pRint&quit Prev Next Go to page Copypage CopyAll

Lloyd's Program ID / Version Number: 08/01/2021.
Version: r28789 - 16 May 2024 16 May 2024 10:50:52

SARC Coral Nordic IMO 9919890 LOCO PIAS

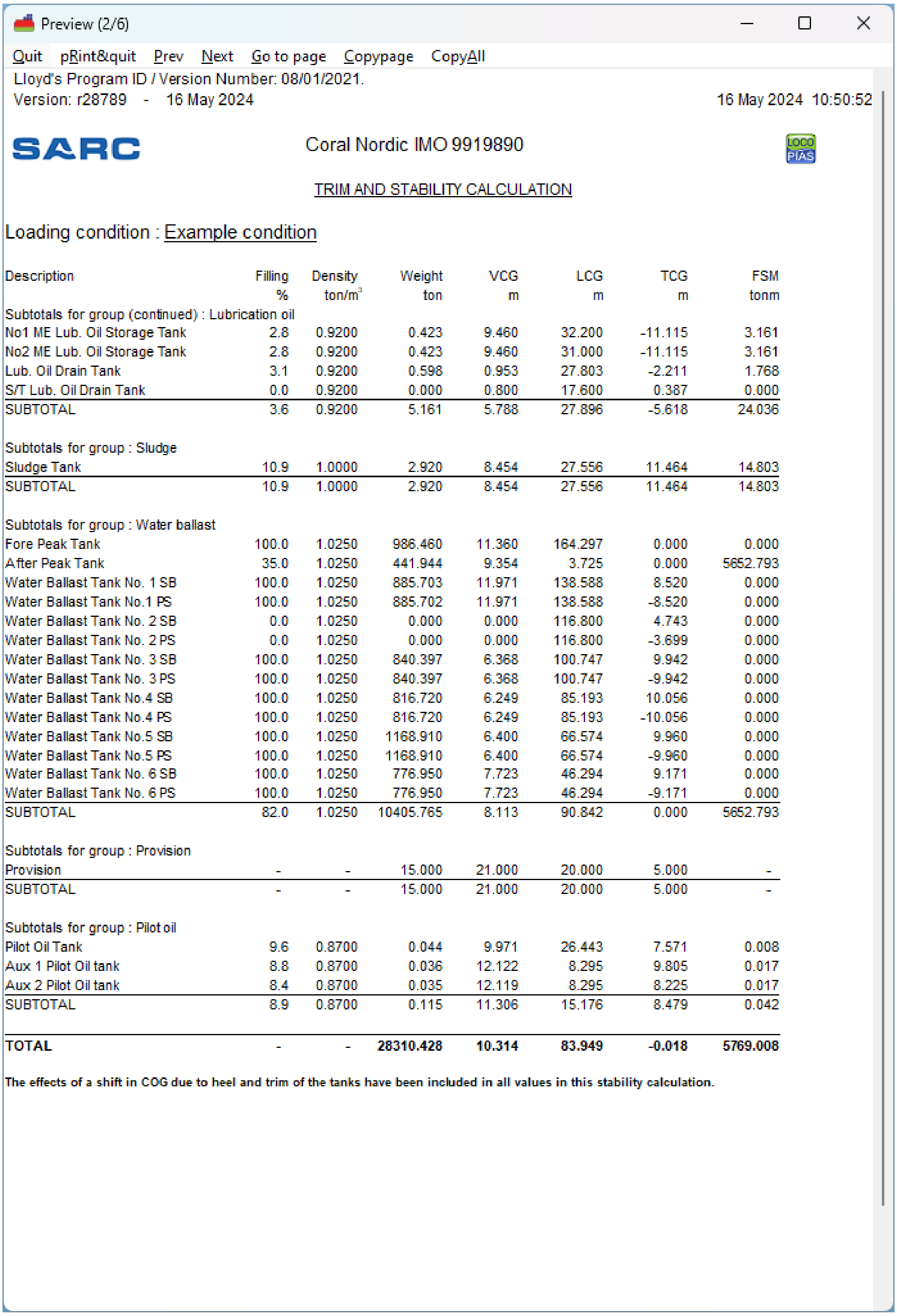
TRIM AND STABILITY CALCULATION

Loading condition : Example condition

Description	Filling %	Density ton/m³	Weight ton	VCG m	LCG m	TCG m	FSM tonm
Empty ship	-	-	11813.900	12.220	75.470	-0.040	-
Subtotals for group : Liquid cargo							
Gas Tank 1 PS	100.0	0.2000	1403.680	10.289	121.881	-5.326	0.000
Gas Tank 1 SB	100.0	0.2000	1402.400	10.289	121.882	5.326	0.000
Gas Tank 2 PS	100.0	0.2000	1608.780	10.285	61.002	-5.867	0.000
Gas Tank 2 SB	100.0	0.2000	1608.780	10.285	61.002	5.867	0.000
SUBTOTAL	100.0	0.2000	6023.640	10.287	89.362	-0.001	0.000
Subtotals for group : Crew							
Crew	-	-	6.000	27.000	25.200	0.000	-
SUBTOTAL	-	-	6.000	27.000	25.200	0.000	-
Subtotals for group : Diesel oil							
Waste Oil Tank	3.7	0.8700	0.600	0.955	31.175	-2.878	2.864
No.2 M.G.O. Service Tank	3.1	0.8700	0.713	14.455	30.402	7.110	1.350
No.2 M.G.O Settling Tank	3.0	0.8700	0.705	14.454	30.402	5.530	1.348
M.G.O. Storage Tank	2.3	0.8700	1.218	14.452	29.308	12.053	3.602
M.G.O. Overflow Tank Aft	3.2	0.8700	0.513	0.948	31.174	2.866	2.699
M.G.O. Overflow Tank Fore	2.7	0.8700	0.470	11.359	155.057	-5.167	2.011
Fore M.G.O. Tank PS	0.4	0.8700	0.522	2.353	152.993	-1.398	3.431
Fore M.G.O. Tank SB	0.5	0.8700	0.626	2.363	152.994	1.400	3.460
No.1 M.D.O. Service Tank	3.1	0.8700	0.722	14.456	30.402	8.690	1.349
No.1 M.D.O. Settling Tank	3.1	0.8700	0.748	14.456	30.402	10.290	1.444
SUBTOTAL	1.6	0.8700	6.838	10.012	59.485	5.131	23.559
Subtotals for group : Fresh water							
Swimming pool	68.5	1.0000	25.000	23.610	12.400	-6.320	10.302
Fresh Water Tank PS	2.2	1.0000	2.290	14.462	5.394	-9.594	18.823
Fresh Water Tank SB	1.8	1.0000	1.930	14.453	5.394	9.593	18.736
Cooling Water Tank	0.0	1.0000	0.000	0.769	7.200	0.000	0.000
SUBTOTAL	11.2	1.0000	29.220	22.288	11.388	-5.526	47.861
Subtotals for group : Grey water							
Bilge Well PS	0.0	1.0000	0.000	1.693	32.000	-6.362	0.000
Bilge Well SB	0.0	1.0000	0.000	1.693	32.000	6.362	0.000
Bilge Well Aft	0.0	1.0000	0.000	0.521	9.200	0.000	0.000
Bilge Water Holding Tank	2.3	1.0000	0.800	0.054	15.948	0.000	2.458
Sewage Holding Tank	1.4	1.0000	0.980	0.050	26.388	2.105	3.401
Fore Bilge Holding Tank	1.1	1.0000	0.090	0.054	152.653	0.334	0.055
SUBTOTAL	1.6	1.0000	1.870	0.052	27.999	1.119	5.914
Subtotals for group : Lubrication oil							
G/E Lub. Oil Settling Tank	2.7	0.9200	0.267	9.457	27.200	-10.300	0.494
G/E Lub. Oil Storage Tank	2.7	0.9200	0.267	9.456	27.200	-11.925	0.545
ME Lub. Oil Settling Tank	2.4	0.9200	0.478	9.451	29.600	-11.115	4.200
S/T Lub. Oil Storage Tank	2.4	0.9200	0.239	9.451	28.400	-11.115	2.108
Lub. Oil Sump Tank	12.2	0.9200	1.776	1.661	24.407	0.000	2.910
Thermal Oil Drain Tank	2.4	0.9200	0.350	9.451	31.000	-6.320	2.845
Thermal Oil Storage Tank	2.4	0.9200	0.340	9.450	32.200	-6.320	2.844

The effects of a shift in COG due to heel and trim of the tanks have been included in all values in this stability calculation.

An example of output of intact stability, page 1/6.



Preview (2/6)

Quit pRint&quit Prev Next Go to page Copypage CopyAll

Lloyd's Program ID / Version Number: 08/01/2021.
Version: r28789 - 16 May 2024 16 May 2024 10:50:52

SARC Coral Nordic IMO 9919890 LOCO PIAS

TRIM AND STABILITY CALCULATION

Loading condition : Example condition

Description	Filling %	Density ton/m³	Weight ton	VCG m	LCG m	TCG m	FSM tonm
Subtotals for group (continued) : Lubrication oil							
No1 ME Lub. Oil Storage Tank	2.8	0.9200	0.423	9.460	32.200	-11.115	3.161
No2 ME Lub. Oil Storage Tank	2.8	0.9200	0.423	9.460	31.000	-11.115	3.161
Lub. Oil Drain Tank	3.1	0.9200	0.598	0.953	27.803	-2.211	1.768
S/T Lub. Oil Drain Tank	0.0	0.9200	0.000	0.800	17.600	0.387	0.000
SUBTOTAL	3.6	0.9200	5.161	5.788	27.896	-5.618	24.036
Subtotals for group : Sludge							
Sludge Tank	10.9	1.0000	2.920	8.454	27.556	11.464	14.803
SUBTOTAL	10.9	1.0000	2.920	8.454	27.556	11.464	14.803
Subtotals for group : Water ballast							
Fore Peak Tank	100.0	1.0250	986.460	11.360	164.297	0.000	0.000
After Peak Tank	35.0	1.0250	441.944	9.354	3.725	0.000	5652.793
Water Ballast Tank No. 1 SB	100.0	1.0250	885.703	11.971	138.588	8.520	0.000
Water Ballast Tank No.1 PS	100.0	1.0250	885.702	11.971	138.588	-8.520	0.000
Water Ballast Tank No. 2 SB	0.0	1.0250	0.000	0.000	116.800	4.743	0.000
Water Ballast Tank No. 2 PS	0.0	1.0250	0.000	0.000	116.800	-3.699	0.000
Water Ballast Tank No. 3 SB	100.0	1.0250	840.397	6.368	100.747	9.942	0.000
Water Ballast Tank No. 3 PS	100.0	1.0250	840.397	6.368	100.747	-9.942	0.000
Water Ballast Tank No.4 SB	100.0	1.0250	816.720	6.249	85.193	10.056	0.000
Water Ballast Tank No.4 PS	100.0	1.0250	816.720	6.249	85.193	-10.056	0.000
Water Ballast Tank No.5 SB	100.0	1.0250	1168.910	6.400	66.574	9.960	0.000
Water Ballast Tank No.5 PS	100.0	1.0250	1168.910	6.400	66.574	-9.960	0.000
Water Ballast Tank No. 6 SB	100.0	1.0250	776.950	7.723	46.294	9.171	0.000
Water Ballast Tank No. 6 PS	100.0	1.0250	776.950	7.723	46.294	-9.171	0.000
SUBTOTAL	82.0	1.0250	10405.765	8.113	90.842	0.000	5652.793
Subtotals for group : Provision							
Provision	-	-	15.000	21.000	20.000	5.000	-
SUBTOTAL	-	-	15.000	21.000	20.000	5.000	-
Subtotals for group : Pilot oil							
Pilot Oil Tank	9.6	0.8700	0.044	9.971	26.443	7.571	0.008
Aux 1 Pilot Oil tank	8.8	0.8700	0.036	12.122	8.295	9.805	0.017
Aux 2 Pilot Oil tank	8.4	0.8700	0.035	12.119	8.295	8.225	0.017
SUBTOTAL	8.9	0.8700	0.115	11.306	15.176	8.479	0.042
TOTAL	-	-	**28310.428**	**10.314**	**83.949**	**-0.018**	**5769.008**

The effects of a shift in COG due to heel and trim of the tanks have been included in all values in this stability calculation.

An example of output of intact stability, page 2/6.

Preview (3/6)

Quit pRint&quit Prev Next Go to page Copypage CopyAll

Lloyd's Program ID / Version Number: 08/01/2021.
Version: r28789 - 16 May 2024 16 May 2024 10:50:52

SARC Coral Nordic IMO 9919890 LOCO PIAS

TRIM AND STABILITY CALCULATION

Loading condition : Example condition

Hydrostatics			Drafts and trim		
Volume	27532.984	m^3	Drafts above base :		
LCF	79.935	m	Draft mean (Lpp/2)	8.024	m
Mom. change trim	417.570	tonm/cm	Draft aft (App)	8.016	m
Ton/cm immersion	41.705	ton/cm	Draft fore (Fpp)	8.032	m
Density	1.0250	ton/m^3	Trim	0.017	m

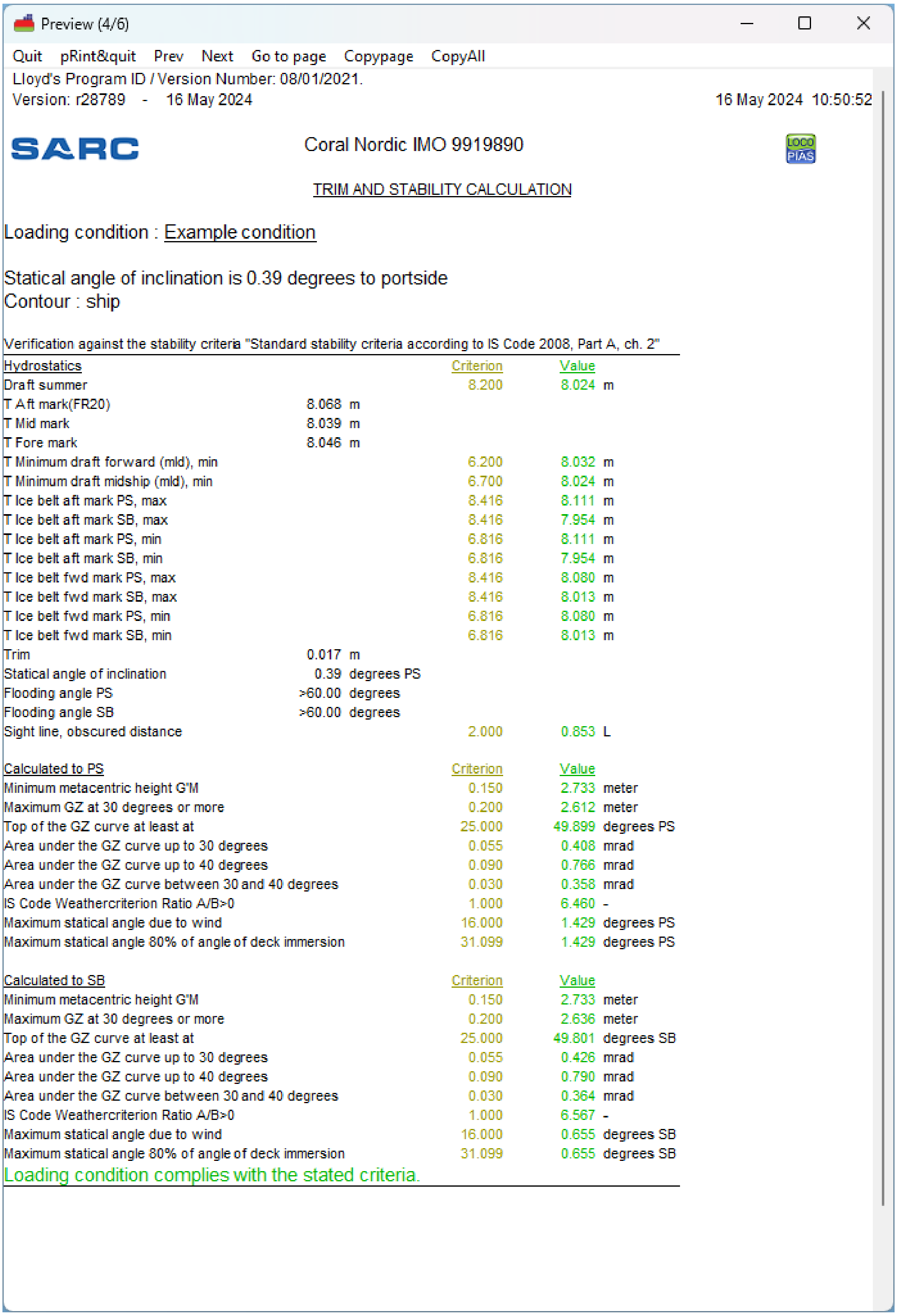
Transverse stability			Drafts on the draftmarks :		
KM transverse	13.251	m	T Aft mark(FR20)	8.068	m
VCG	10.314	m	T Mid mark	8.039	m
GM solid	2.936	m	T Fore mark	8.046	m
GG' correction	0.204	m			
G'M liquid	2.733	m	VCG'	10.518	m

The stability values are calculated for the actual trim.

Statical stability, calculated with the effect of VCG on trim:

Angle degrees		Draft mld. m	Trim m	KNsinφ m	VCGsinφ m	TCGcosφ m	GNsinφ m	Area mrad
60.00	PS	3.832	3.650	-11.387	-8.958	-0.056	-2.373	1.650
50.00	PS	5.716	2.319	-10.605	-7.922	-0.071	-2.612	1.208
40.00	PS	6.944	1.425	-9.089	-6.645	-0.082	-2.363	0.766
35.00	PS	7.363	1.124	-8.071	-5.928	-0.085	-2.058	0.573
30.00	PS	7.629	0.845	-6.972	-5.166	-0.086	-1.719	0.408
25.00	PS	7.790	0.595	-5.814	-4.365	-0.083	-1.366	0.273
20.00	PS	7.891	0.385	-4.641	-3.531	-0.076	-1.034	0.169
15.00	PS	7.956	0.221	-3.475	-2.671	-0.065	-0.738	0.092
10.00	PS	7.996	0.106	-2.315	-1.792	-0.052	-0.471	0.039
5.00	PS	8.017	0.039	-1.157	-0.899	-0.036	-0.222	0.009
2.00	PS	8.023	0.020	-0.463	-0.360	-0.026	-0.077	0.001
0.00		8.024	0.017	0.000	0.000	-0.018	0.018	0.000
2.00	SB	8.023	0.020	0.463	0.360	-0.011	0.114	0.002
5.00	SB	8.017	0.039	1.157	0.899	-0.001	0.258	0.012
10.00	SB	7.996	0.106	2.315	1.792	0.015	0.507	0.045
15.00	SB	7.956	0.221	3.475	2.671	0.030	0.774	0.101
20.00	SB	7.891	0.385	4.641	3.531	0.041	1.069	0.181
25.00	SB	7.790	0.595	5.814	4.365	0.050	1.399	0.289
30.00	SB	7.629	0.846	6.972	5.166	0.054	1.751	0.426
35.00	SB	7.363	1.125	8.071	5.928	0.055	2.089	0.594
40.00	SB	6.944	1.426	9.089	6.645	0.053	2.391	0.790
50.00	SB	5.716	2.319	10.605	7.922	0.047	2.636	1.236
60.00	SB	3.832	3.650	11.387	8.958	0.038	2.391	1.682

An example of output of intact stability, page 3/6.



Preview (4/6)

Quit pRint&quit Prev Next Go to page Copypage CopyAll

Lloyd's Program ID / Version Number: 08/01/2021.
Version: r28789 - 16 May 2024 16 May 2024 10:50:52

SARC Coral Nordic IMO 9919890 LOCO PIAS

TRIM AND STABILITY CALCULATION

Loading condition : Example condition

Statical angle of inclination is 0.39 degrees to portside
Contour : ship

Verification against the stability criteria "Standard stability criteria according to IS Code 2008, Part A, ch. 2"

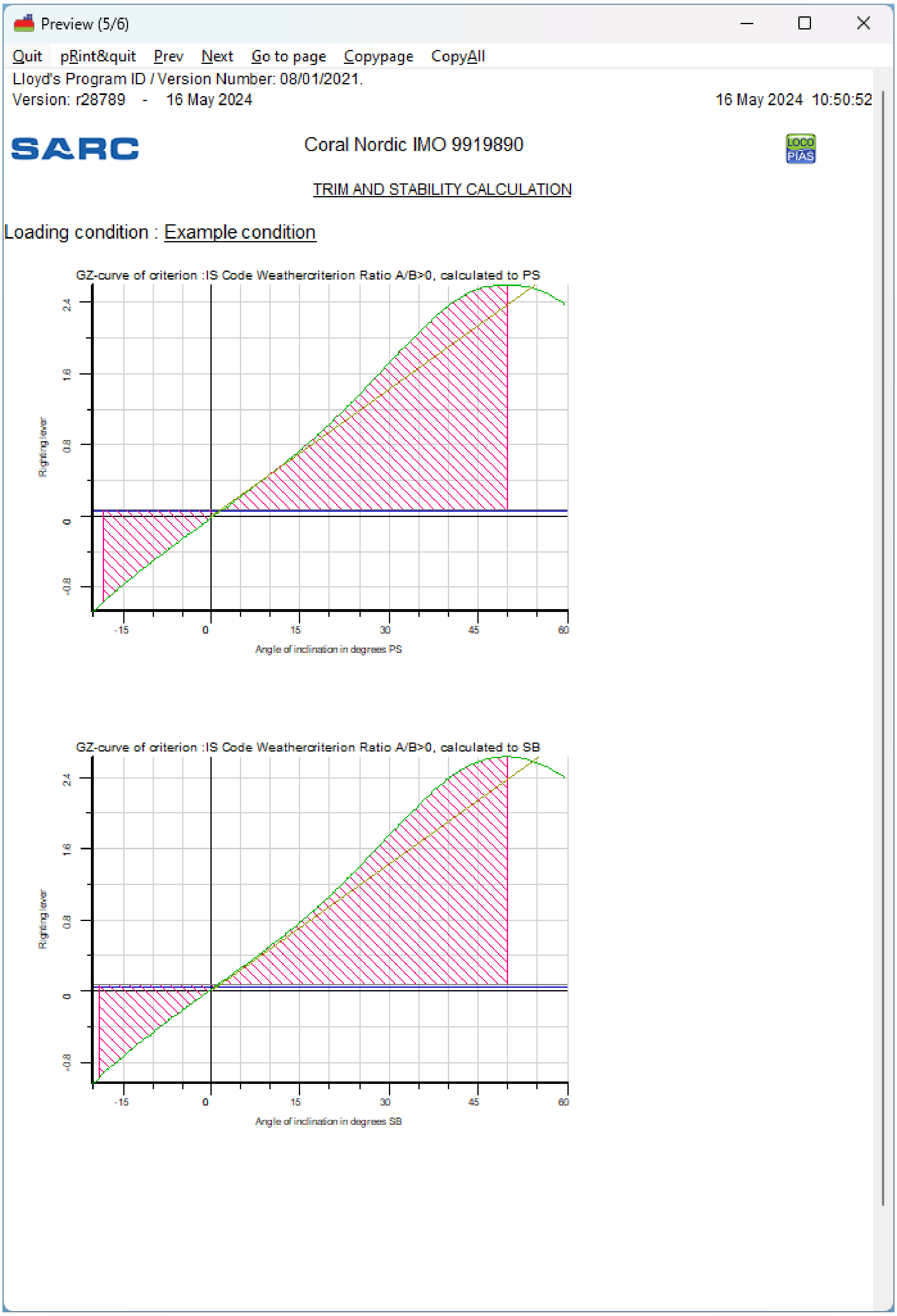
Hydrostatics		Criterion	Value	
Draft summer		8.200	8.024	m
T Aft mark(FR20)	8.068 m			
T Mid mark	8.039 m			
T Fore mark	8.046 m			
T Minimum draft forward (mld), min		6.200	8.032	m
T Minimum draft midship (mld), min		6.700	8.024	m
T Ice belt aft mark PS, max		8.416	8.111	m
T Ice belt aft mark SB, max		8.416	7.954	m
T Ice belt aft mark PS, min		6.816	8.111	m
T Ice belt aft mark SB, min		6.816	7.954	m
T Ice belt fwd mark PS, max		8.416	8.080	m
T Ice belt fwd mark SB, max		8.416	8.013	m
T Ice belt fwd mark PS, min		6.816	8.080	m
T Ice belt fwd mark SB, min		6.816	8.013	m
Trim	0.017 m			
Statical angle of inclination	0.39 degrees PS			
Flooding angle PS	>60.00 degrees			
Flooding angle SB	>60.00 degrees			
Sight line, obscured distance		2.000	0.853	L

Calculated to PS	Criterion	Value	
Minimum metacentric height G'M	0.150	2.733	meter
Maximum GZ at 30 degrees or more	0.200	2.612	meter
Top of the GZ curve at least at	25.000	49.899	degrees PS
Area under the GZ curve up to 30 degrees	0.055	0.408	mrad
Area under the GZ curve up to 40 degrees	0.090	0.766	mrad
Area under the GZ curve between 30 and 40 degrees	0.030	0.358	mrad
IS Code Weathercriterion Ratio A/B>0	1.000	6.460	-
Maximum statical angle due to wind	16.000	1.429	degrees PS
Maximum statical angle 80% of angle of deck immersion	31.099	1.429	degrees PS

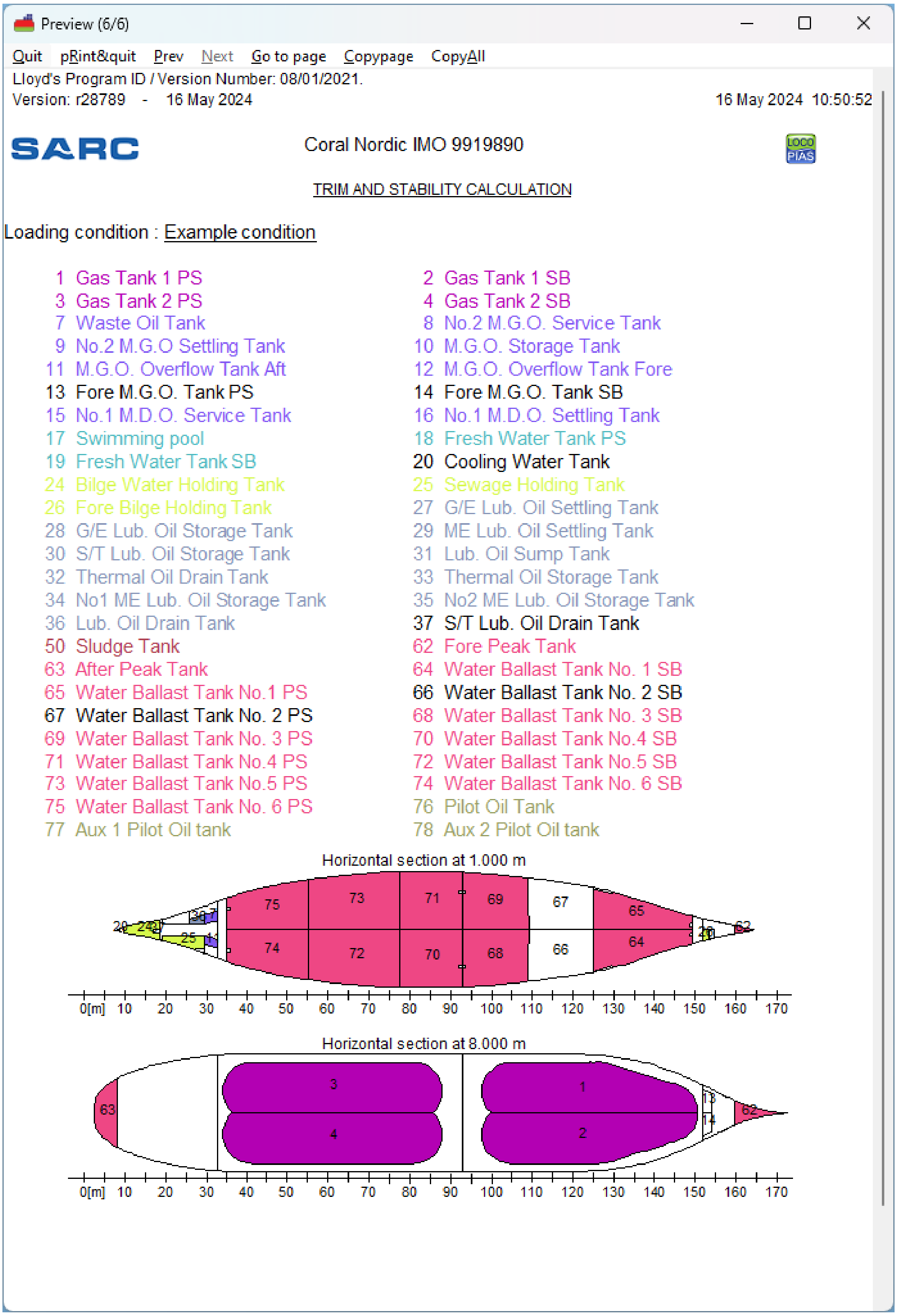
Calculated to SB	Criterion	Value	
Minimum metacentric height G'M	0.150	2.733	meter
Maximum GZ at 30 degrees or more	0.200	2.636	meter
Top of the GZ curve at least at	25.000	49.801	degrees SB
Area under the GZ curve up to 30 degrees	0.055	0.426	mrad
Area under the GZ curve up to 40 degrees	0.090	0.790	mrad
Area under the GZ curve between 30 and 40 degrees	0.030	0.364	mrad
IS Code Weathercriterion Ratio A/B>0	1.000	6.567	-
Maximum statical angle due to wind	16.000	0.655	degrees SB
Maximum statical angle 80% of angle of deck immersion	31.099	0.655	degrees SB

Loading condition complies with the stated criteria.

An example of output of intact stability, page 4/6.



An example of output of intact stability, page 5/6.



An example of output of intact stability, page 6/6.

17.1.9.2.2 Longitudinal strength

Example of print-out of longitudinal strength. Actual output and/ or results may differ according the choosen calculation, and/ or output settings.

Preview (1/6)

Quit pRint&quit Prev Next Go to page Copypage CopyAll

Lloyd's Program ID / Version Number: 08/01/2021.
Version: r29117M - 28 August 2024 28 Aug 2024 16:24:59

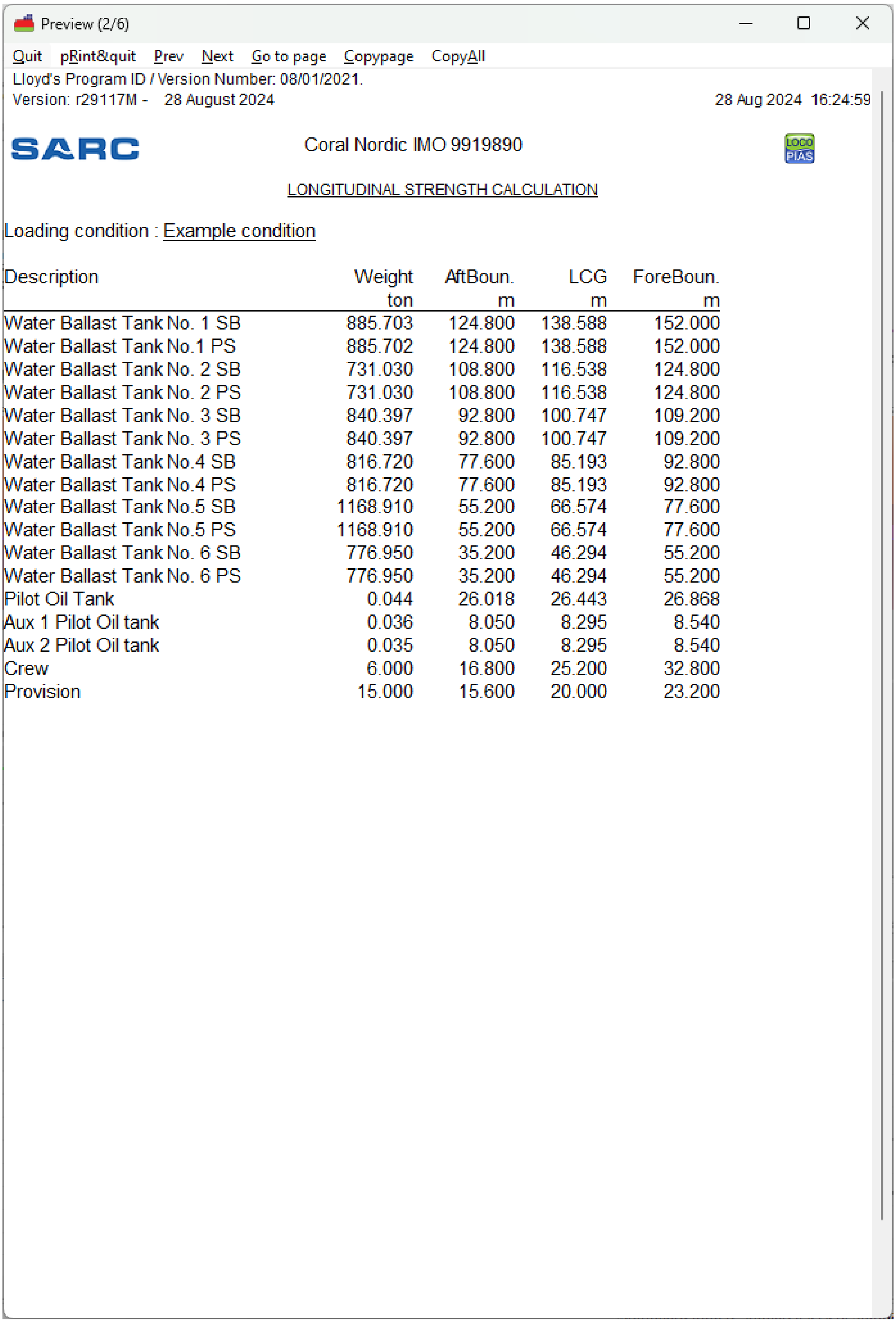
SARC Coral Nordic IMO 9919890

LONGITUDINAL STRENGTH CALCULATION

Loading condition : Example condition

Description	Weight ton	AftBoun. m	LCG m	ForeBoun. m
Empty ship	11813.900		75.470	
Gas Tank 1 PS,1	0.000	104.000	108.000	112.000
Gas Tank 1 PS,2	0.000	130.400	134.400	138.400
Gas Tank 1 SB,1	0.000	104.000	108.000	112.000
Gas Tank 1 SB,2	0.000	130.400	134.400	138.400
Gas Tank 2 PS,1	0.000	40.800	44.800	48.800
Gas Tank 2 PS,2	0.000	73.600	77.600	81.600
Gas Tank 2 SB,1	0.000	40.800	44.800	48.800
Gas Tank 2 SB,2	0.000	73.600	77.600	81.600
Waste Oil Tank	0.600	29.600	31.174	32.800
No.2 M.G.O. Service Tank	0.713	28.000	30.400	32.800
No.2 M.G.O Settling Tank	0.705	28.000	30.400	32.800
M.G.O. Storage Tank	1.218	25.600	29.304	32.800
M.G.O. Overflow Tank Aft	0.513	29.600	31.173	32.800
M.G.O. Overflow Tank Fore	0.470	154.100	155.056	156.200
Fore M.G.O. Tank PS	0.522	152.000	152.993	154.100
Fore M.G.O. Tank SB	0.626	152.000	152.993	154.100
No.1 M.D.O. Service Tank	0.722	28.000	30.400	32.800
No.1 M.D.O. Settling Tank	0.748	28.000	30.400	32.800
Swimming pool	25.000	10.400	12.400	14.400
Fresh Water Tank PS	2.290	2.400	5.392	8.000
Fresh Water Tank SB	1.930	2.400	5.392	8.000
Cooling Water Tank	0.000	5.600	7.200	8.800
Bilge Well PS	0.000	31.200	32.000	32.800
Bilge Well SB	0.000	31.200	32.000	32.800
Bilge Well Aft	0.000	8.800	9.200	9.600
Bilge Water Holding Tank	0.800	11.034	15.944	18.400
Sewage Holding Tank	0.980	19.946	26.382	29.600
Fore Bilge Holding Tank	0.090	152.000	152.653	153.400
G/E Lub. Oil Settling Tank	0.267	26.400	27.200	28.000
G/E Lub. Oil Storage Tank	0.267	26.400	27.200	28.000
ME Lub. Oil Settling Tank	0.478	28.800	29.600	30.400
S/T Lub. Oil Storage Tank	0.239	28.000	28.400	28.800
Lub. Oil Sump Tank	1.776	19.200	24.400	29.600
Thermal Oil Drain Tank	0.350	30.400	31.000	31.600
Thermal Oil Storage Tank	0.340	31.600	32.200	32.800
No1 ME Lub. Oil Storage Tank	0.423	31.600	32.200	32.800
No2 ME Lub. Oil Storage Tank	0.423	30.400	31.000	31.600
Lub. Oil Drain Tank	0.598	25.600	27.802	29.600
S/T Lub. Oil Drain Tank	0.000	16.800	17.600	18.400
Sludge Tank	2.920	20.800	27.546	32.800
Fore Peak Tank	986.460	159.700	164.297	172.300
After Peak Tank	441.944	-3.200	3.725	8.000

An example of output of longitudinal strength page 1/6.



Preview (2/6)

Quit pRint&quit Prev Next Go to page Copypage CopyAll

Lloyd's Program ID / Version Number: 08/01/2021.
Version: r29117M - 28 August 2024 28 Aug 2024 16:24:59

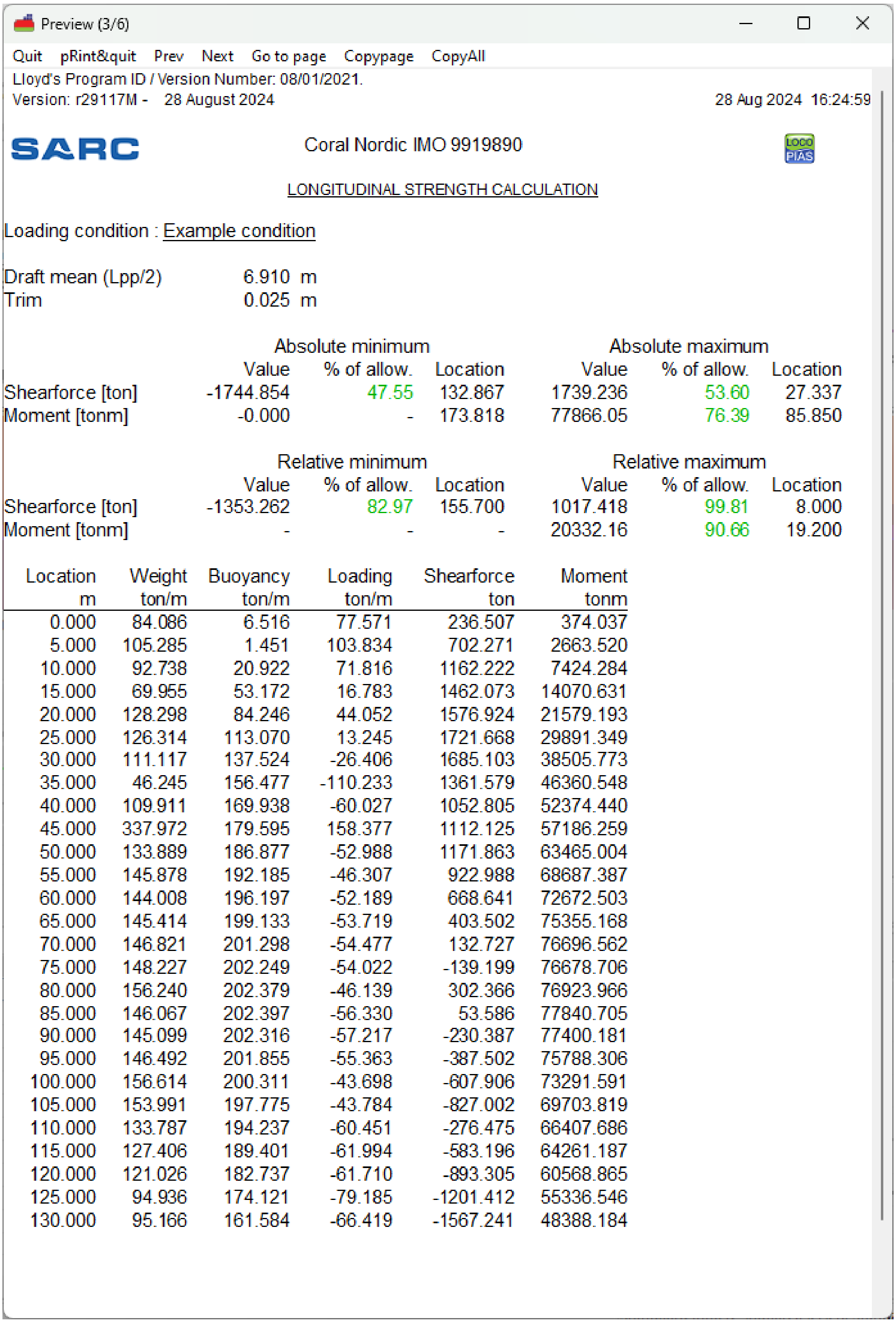
Coral Nordic IMO 9919890

LONGITUDINAL STRENGTH CALCULATION

Loading condition : Example condition

Description	Weight ton	AftBoun. m	LCG m	ForeBoun. m
Water Ballast Tank No. 1 SB	885.703	124.800	138.588	152.000
Water Ballast Tank No.1 PS	885.702	124.800	138.588	152.000
Water Ballast Tank No. 2 SB	731.030	108.800	116.538	124.800
Water Ballast Tank No. 2 PS	731.030	108.800	116.538	124.800
Water Ballast Tank No. 3 SB	840.397	92.800	100.747	109.200
Water Ballast Tank No. 3 PS	840.397	92.800	100.747	109.200
Water Ballast Tank No.4 SB	816.720	77.600	85.193	92.800
Water Ballast Tank No.4 PS	816.720	77.600	85.193	92.800
Water Ballast Tank No.5 SB	1168.910	55.200	66.574	77.600
Water Ballast Tank No.5 PS	1168.910	55.200	66.574	77.600
Water Ballast Tank No. 6 SB	776.950	35.200	46.294	55.200
Water Ballast Tank No. 6 PS	776.950	35.200	46.294	55.200
Pilot Oil Tank	0.044	26.018	26.443	26.868
Aux 1 Pilot Oil tank	0.036	8.050	8.295	8.540
Aux 2 Pilot Oil tank	0.035	8.050	8.295	8.540
Crew	6.000	16.800	25.200	32.800
Provision	15.000	15.600	20.000	23.200

An example of output of longitudinal strength page 2/6.



Preview (3/6)

Quit pRint&quit Prev Next Go to page Copypage CopyAll

Lloyd's Program ID / Version Number: 08/01/2021.
Version: r29117M - 28 August 2024 28 Aug 2024 16:24:59

SARC Coral Nordic IMO 9919890 LOCO PIAS

LONGITUDINAL STRENGTH CALCULATION

Loading condition : Example condition

Draft mean (Lpp/2) 6.910 m
Trim 0.025 m

	Absolute minimum			Absolute maximum		
	Value	% of allow.	Location	Value	% of allow.	Location
Shearforce [ton]	-1744.854	47.55	132.867	1739.236	53.60	27.337
Moment [tonm]	-0.000	-	173.818	77866.05	76.39	85.850

	Relative minimum			Relative maximum		
	Value	% of allow.	Location	Value	% of allow.	Location
Shearforce [ton]	-1353.262	82.97	155.700	1017.418	99.81	8.000
Moment [tonm]	-	-	-	20332.16	90.66	19.200

Location m	Weight ton/m	Buoyancy ton/m	Loading ton/m	Shearforce ton	Moment tonm
0.000	84.086	6.516	77.571	236.507	374.037
5.000	105.285	1.451	103.834	702.271	2663.520
10.000	92.738	20.922	71.816	1162.222	7424.284
15.000	69.955	53.172	16.783	1462.073	14070.631
20.000	128.298	84.246	44.052	1576.924	21579.193
25.000	126.314	113.070	13.245	1721.668	29891.349
30.000	111.117	137.524	-26.406	1685.103	38505.773
35.000	46.245	156.477	-110.233	1361.579	46360.548
40.000	109.911	169.938	-60.027	1052.805	52374.440
45.000	337.972	179.595	158.377	1112.125	57186.259
50.000	133.889	186.877	-52.988	1171.863	63465.004
55.000	145.878	192.185	-46.307	922.988	68687.387
60.000	144.008	196.197	-52.189	668.641	72672.503
65.000	145.414	199.133	-53.719	403.502	75355.168
70.000	146.821	201.298	-54.477	132.727	76696.562
75.000	148.227	202.249	-54.022	-139.199	76678.706
80.000	156.240	202.379	-46.139	302.366	76923.966
85.000	146.067	202.397	-56.330	53.586	77840.705
90.000	145.099	202.316	-57.217	-230.387	77400.181
95.000	146.492	201.855	-55.363	-387.502	75788.306
100.000	156.614	200.311	-43.698	-607.906	73291.591
105.000	153.991	197.775	-43.784	-827.002	69703.819
110.000	133.787	194.237	-60.451	-276.475	66407.686
115.000	127.406	189.401	-61.994	-583.196	64261.187
120.000	121.026	182.737	-61.710	-893.305	60568.865
125.000	94.936	174.121	-79.185	-1201.412	55336.546
130.000	95.166	161.584	-66.419	-1567.241	48388.184

An example of output of longitudinal strength page 3/6.

Preview (4/6)

Quit pRint&quit Prev Next Go to page Copypage CopyAll

Lloyd's Program ID / Version Number: 08/01/2021.
Version: r29117M - 28 August 2024 28 Aug 2024 16:24:59

SARC Coral Nordic IMO 9919890 LOCO PIAS

LONGITUDINAL STRENGTH CALCULATION

Loading condition : Example condition

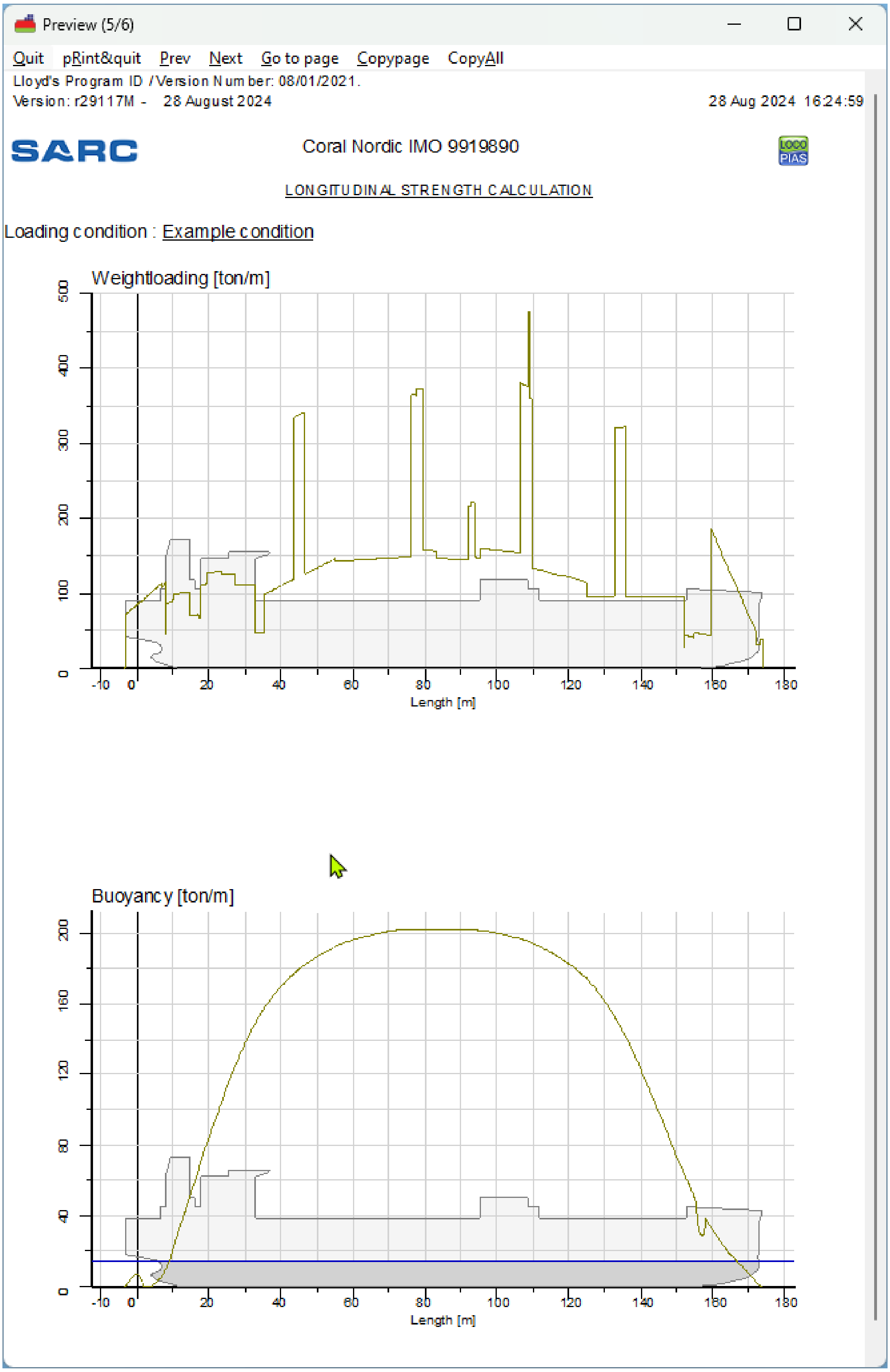
Location m	Weight ton/m	Buoyancy ton/m	Loading ton/m	Shearforce ton	Moment tonm
135.000	321.215	144.866	176.350	-1377.228	40301.516
140.000	95.625	123.441	-27.815	-1330.398	33961.630
145.000	95.855	98.913	-3.058	-1408.188	27063.112
150.000	96.085	73.602	22.483	-1359.856	20089.600
155.000	40.738	50.567	-9.829	-1351.096	13426.712
160.000	181.556	31.770	149.786	-1262.307	6780.996
165.000	126.957	17.844	109.112	-612.903	2176.894
170.000	72.357	7.300	65.057	-177.024	293.197

Pos. m	Shear force ton	% of allow.	Allow. shearf. ton
8.000	1017.418	99.81	1019
19.200	1541.885	66.80	2308
32.800	1595.595	41.19	3874
33.600	1512.504	39.05	3874
44.800	1080.462	27.89	3874
64.000	457.106	11.80	3874
77.600	45.776	1.95	2345
95.200	-398.593	12.61	-3160
108.000	-626.268	19.82	-3160
124.800	-1185.547	32.31	-3670
134.400	-1482.355	40.39	-3670
152.000	-1305.111	80.02	-1631
160.400	-1202.963	73.76	-1631

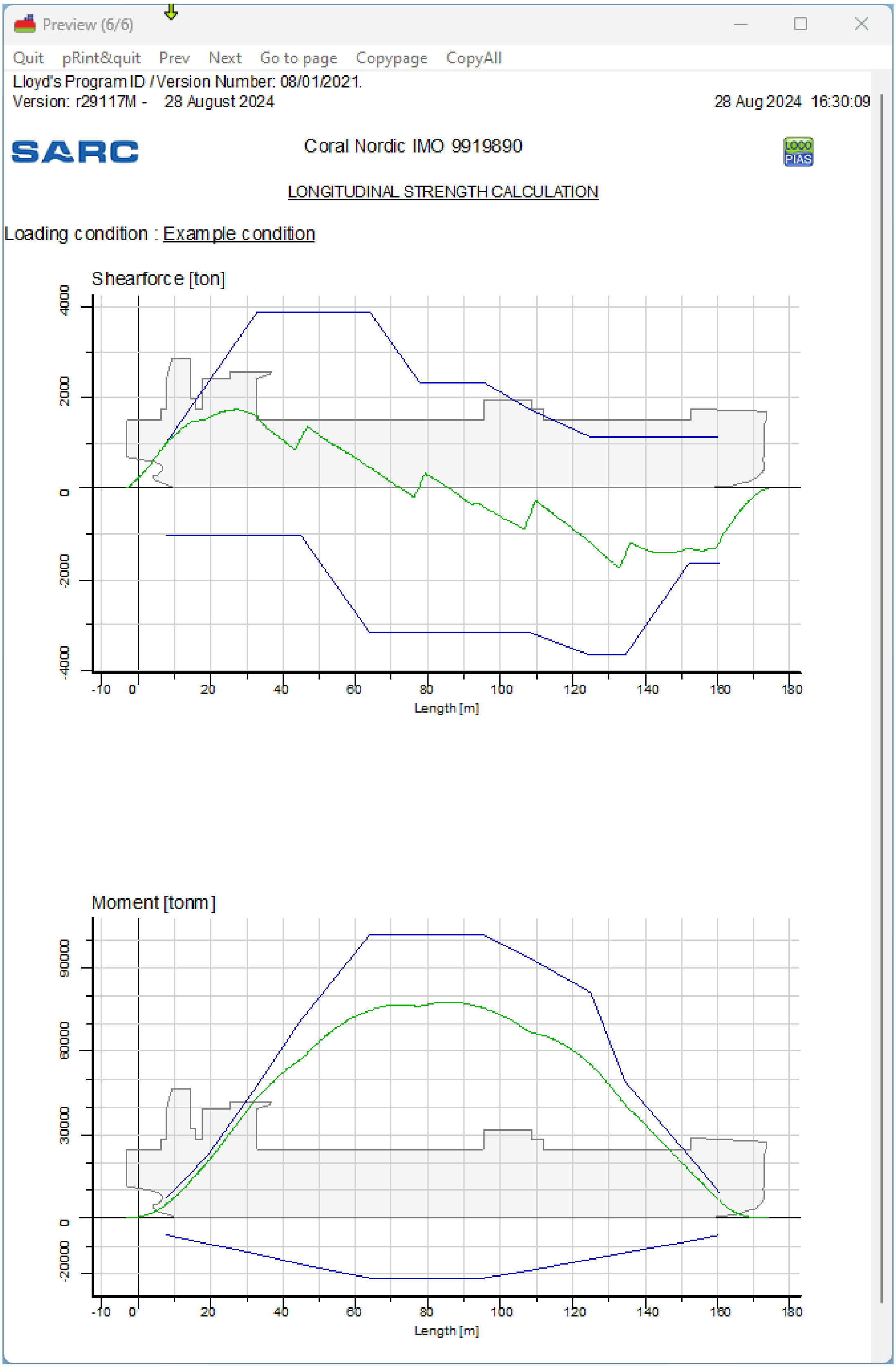
Pos. m	Bending moment tonm	% of allow.	Allow. moment tonm
8.000	5243.791	73.49	7136
19.200	20332.158	90.66	22426
32.800	43105.883	89.97	47910
44.800	56966.504	79.83	71356
64.000	74925.133	73.50	101937
77.600	76378.961	74.93	101937
95.200	75709.906	74.27	101937
108.000	67270.891	71.73	93782
124.800	55575.770	68.15	81549
134.400	41159.281	84.12	48930
152.000	17421.006	77.68	22426
160.400	6287.124	68.53	9174

Loading condition complies with the stated criteria. "Seagoing"

An example of output of longitudinal strength page 4/6.



An example of output of longitudinal strength page 5/6.



An example of output of longitudinal strength page 6/6.

17.1.9.2.3 Damage stability

Example of print-out of damage stability. Actual output and/ or results may differ according the choosen calculation, and/ or output settings.

Preview (1/5)

Quit pRint&quit Prev Next Go to page Copypage CopyAll

Lloyd's Program ID / Version Number: 08/01/2021.
Version: r28789 - 16 May 2024 16 May 2024 12:14:53

SARC Coral Nordic IMO 9919890

FLOODABILITY AND DAMAGE STABILITY

Loading condition : Example condition

Damage case	DS01
Stage of flooding	100%
Intact displacement	28310.432 ton
Fixed weight	11834.902 ton

The effects of a shift in COG due to heel and trim of the tanks have been included in all values in this stability calculation.
All calculations include the effect of VCG on trim.

Openings calculated to PS

Type of opening/point	Name	submerged at	Distance WL
Weathertight opening	VE12	45.48°	11.638 m
Weathertight opening	VE10	47.42°	11.618 m
Weathertight opening	DR01	54.08°	11.261 m
Weathertight opening	DR02	54.12°	11.254 m
Weathertight opening	OP46	54.16°	11.718 m

Openings calculated to SB

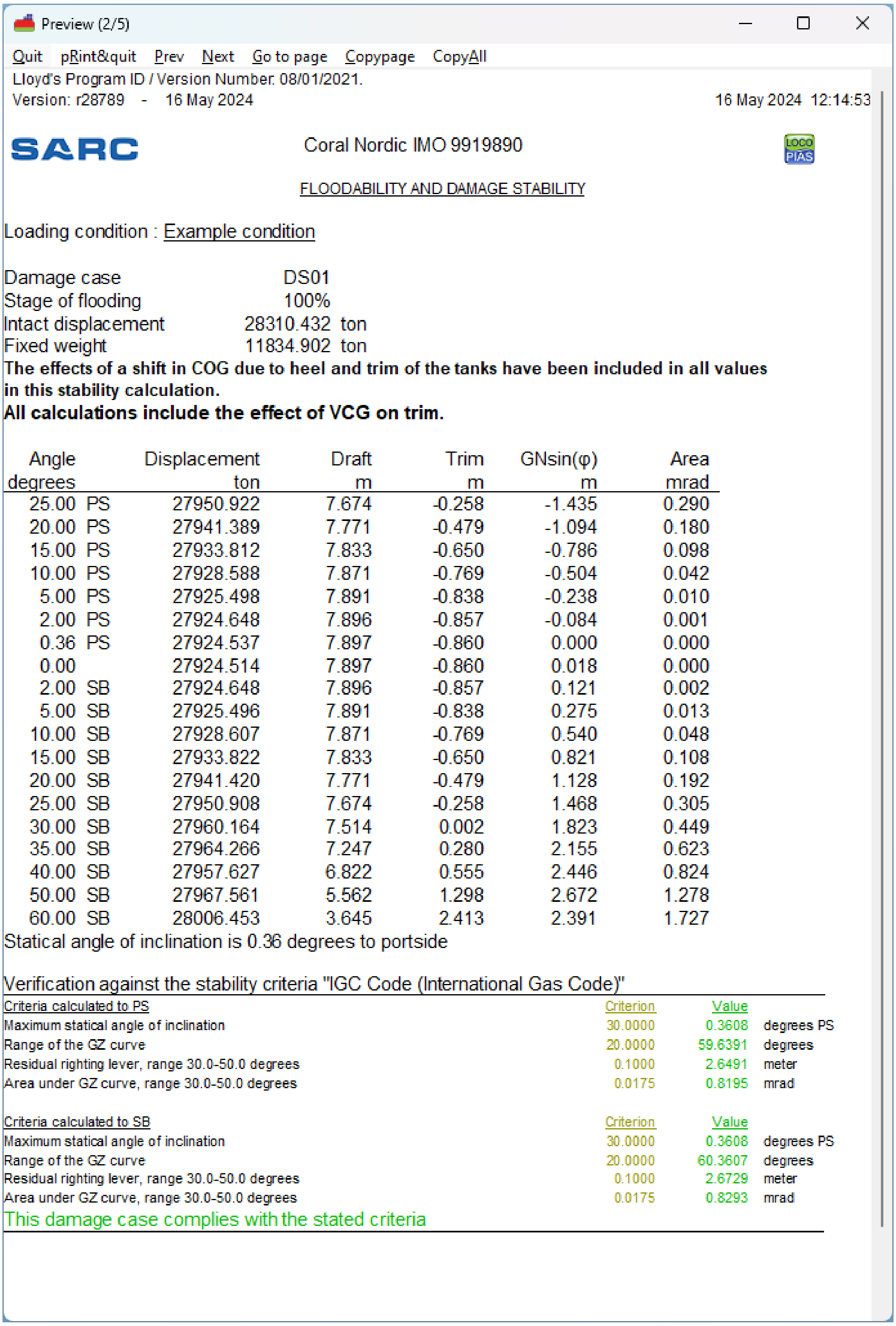
Type of opening/point	Name	submerged at	Distance WL
Weathertight opening	OP45	44.55°	11.699 m
Weathertight opening	OP50	44.55°	11.698 m
Weathertight opening	VE09	45.53°	11.777 m
Weathertight opening	OP20	48.83°	12.208 m
Weathertight opening	OP22	49.54°	12.215 m

Damaged compartments and intact compartment weights (at 0.36° PS) :

Name	Wintact ton	SWintact ton/m³	Wdamag. ton	SWdam. ton/m³
Chain Locker SB	0.000	1.0000	0.000	1.0250
M.G.O. Overflow Tank Fore	0.470	0.8700	0.000	1.0250
Fore M.G.O. Tank PS	0.522	0.8700	41.236	1.0250
Fore M.G.O. Tank SB	0.626	0.8700	40.911	1.0250
Fore Bilge Holding Tank	0.090	1.0000	7.887	1.0250
Bosun Store	0.000	1.0000	0.000	1.0250
Emergency Sea Chest	0.000	1.0000	7.108	1.0250
Echo Sounder & Speed Logger	0.000	1.0000	27.265	1.0250
Fore Hydraulic Unit Room	0.000	1.0000	0.000	1.0250
Bow Thruster Room	0.000	1.0000	228.525	1.0250
N2 Generator Room	0.000	1.0000	0.000	1.0250
Void Fore	0.000	1.0000	38.756	1.0250
Fore Peak Tank	986.460	1.0250	210.556	1.0250

Angle degrees	Displacement ton	Draft m	Trim m	GNsin(φ) m	Area mrad
60.00 PS	28004.352	3.644	2.406	-2.372	1.695
50.00 PS	27967.109	5.562	1.296	-2.648	1.250
40.00 PS	27957.508	6.822	0.555	-2.418	0.800
35.00 PS	27964.242	7.247	0.280	-2.125	0.602
30.00 PS	27960.131	7.514	0.002	-1.791	0.431

An example of output of damage stability, page 1/5.



Preview (2/5)

Quit pRint&quit Prev Next Go to page Copypage CopyAll

Lloyd's Program ID / Version Number: 08/01/2021.
Version: r28789 - 16 May 2024 16 May 2024 12:14:53

SARC Coral Nordic IMO 9919890

FLOODABILITY AND DAMAGE STABILITY

Loading condition : Example condition

Damage case DS01
Stage of flooding 100%
Intact displacement 28310.432 ton
Fixed weight 11834.902 ton

The effects of a shift in COG due to heel and trim of the tanks have been included in all values in this stability calculation.

All calculations include the effect of VCG on trim.

Angle degrees		Displacement ton	Draft m	Trim m	GNsin(φ) m	Area mrad
25.00	PS	27950.922	7.674	-0.258	-1.435	0.290
20.00	PS	27941.389	7.771	-0.479	-1.094	0.180
15.00	PS	27933.812	7.833	-0.650	-0.786	0.098
10.00	PS	27928.588	7.871	-0.769	-0.504	0.042
5.00	PS	27925.498	7.891	-0.838	-0.238	0.010
2.00	PS	27924.648	7.896	-0.857	-0.084	0.001
0.36	PS	27924.537	7.897	-0.860	0.000	0.000
0.00		27924.514	7.897	-0.860	0.018	0.000
2.00	SB	27924.648	7.896	-0.857	0.121	0.002
5.00	SB	27925.496	7.891	-0.838	0.275	0.013
10.00	SB	27928.607	7.871	-0.769	0.540	0.048
15.00	SB	27933.822	7.833	-0.650	0.821	0.108
20.00	SB	27941.420	7.771	-0.479	1.128	0.192
25.00	SB	27950.908	7.674	-0.258	1.468	0.305
30.00	SB	27960.164	7.514	0.002	1.823	0.449
35.00	SB	27964.266	7.247	0.280	2.155	0.623
40.00	SB	27957.627	6.822	0.555	2.446	0.824
50.00	SB	27967.561	5.562	1.298	2.672	1.278
60.00	SB	28006.453	3.645	2.413	2.391	1.727

Statical angle of inclination is 0.36 degrees to portside

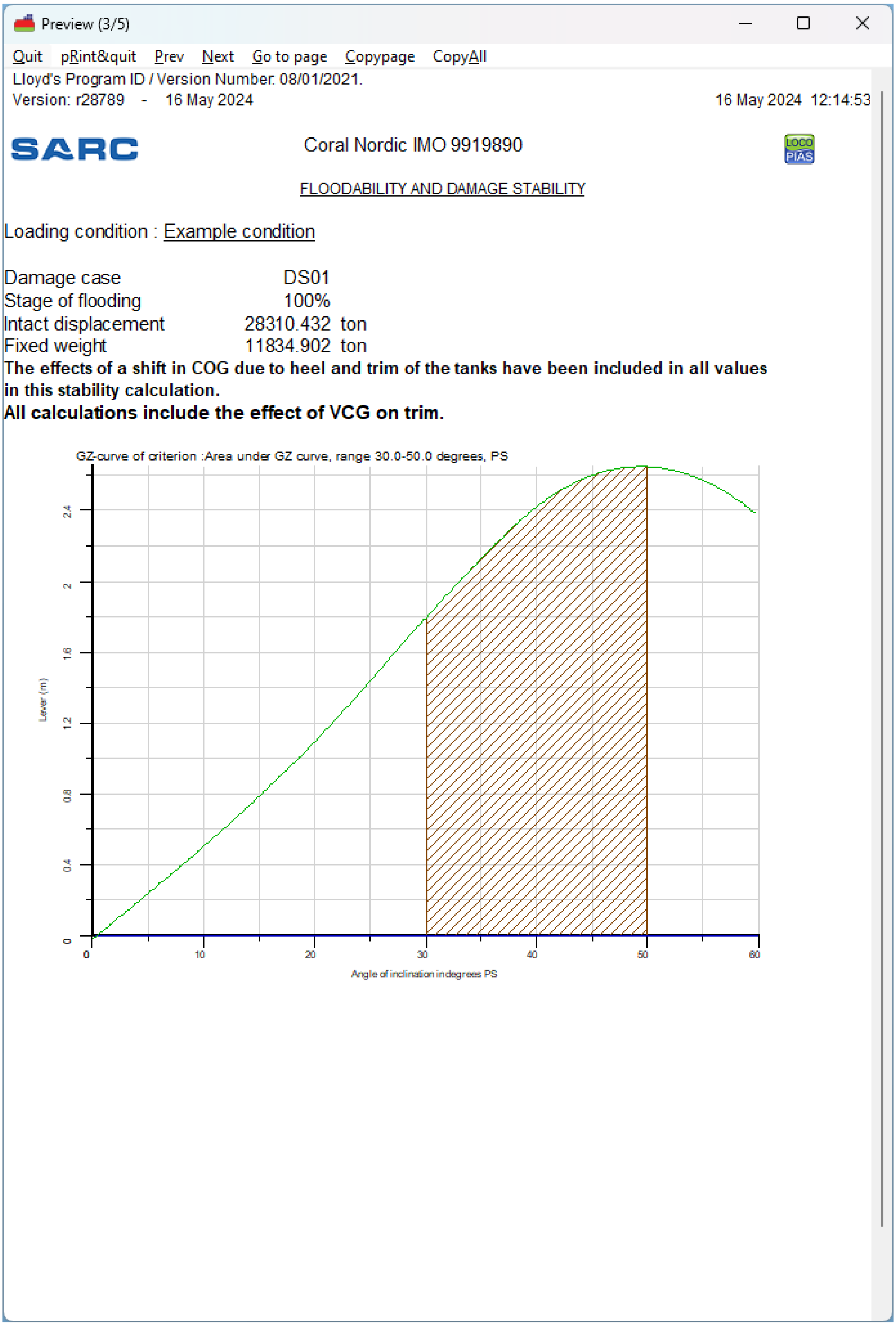
Verification against the stability criteria "IGC Code (International Gas Code)"

Criteria calculated to PS	Criterion	Value	
Maximum statical angle of inclination	30.0000	0.3608	degrees PS
Range of the GZ curve	20.0000	59.6391	degrees
Residual righting lever, range 30.0-50.0 degrees	0.1000	2.6491	meter
Area under GZ curve, range 30.0-50.0 degrees	0.0175	0.8195	mrad

Criteria calculated to SB	Criterion	Value	
Maximum statical angle of inclination	30.0000	0.3608	degrees PS
Range of the GZ curve	20.0000	60.3607	degrees
Residual righting lever, range 30.0-50.0 degrees	0.1000	2.6729	meter
Area under GZ curve, range 30.0-50.0 degrees	0.0175	0.8293	mrad

This damage case complies with the stated criteria

An example of output of damage stability, page 2/5.



An example of output of damage stability, page 3/5.

Preview (4/5)

Quit pRint&quit Prev Next Go to page Copypage CopyAll

Lloyd's Program ID / Version Number. 08/01/2021.
Version: r28789 - 16 May 2024 16 May 2024 12:14:53

SARC Coral Nordic IMO 9919890

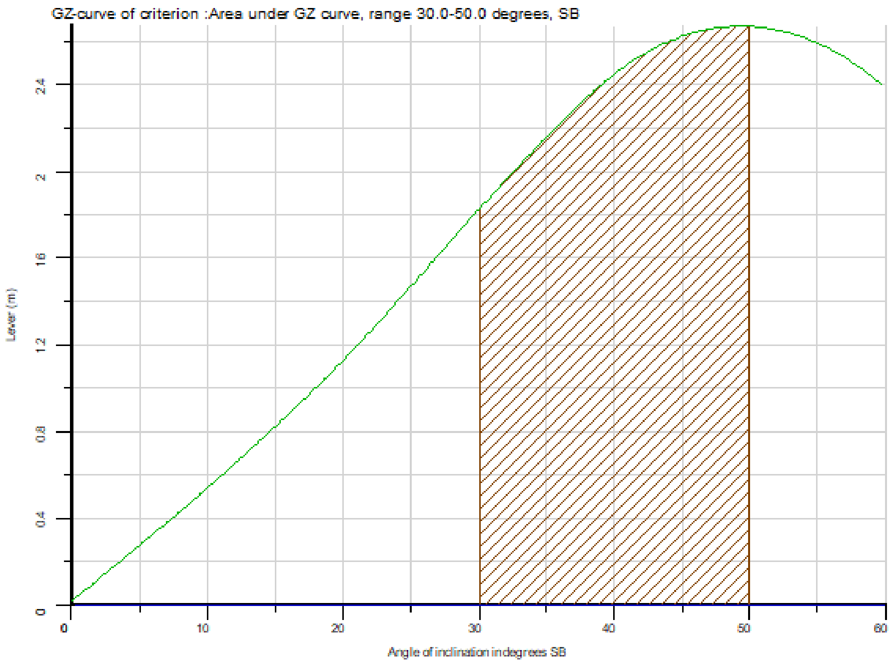
FLOODABILITY AND DAMAGE STABILITY

Loading condition : Example condition

Damage case	DS01
Stage of flooding	100%
Intact displacement	28310.432 ton
Fixed weight	11834.902 ton

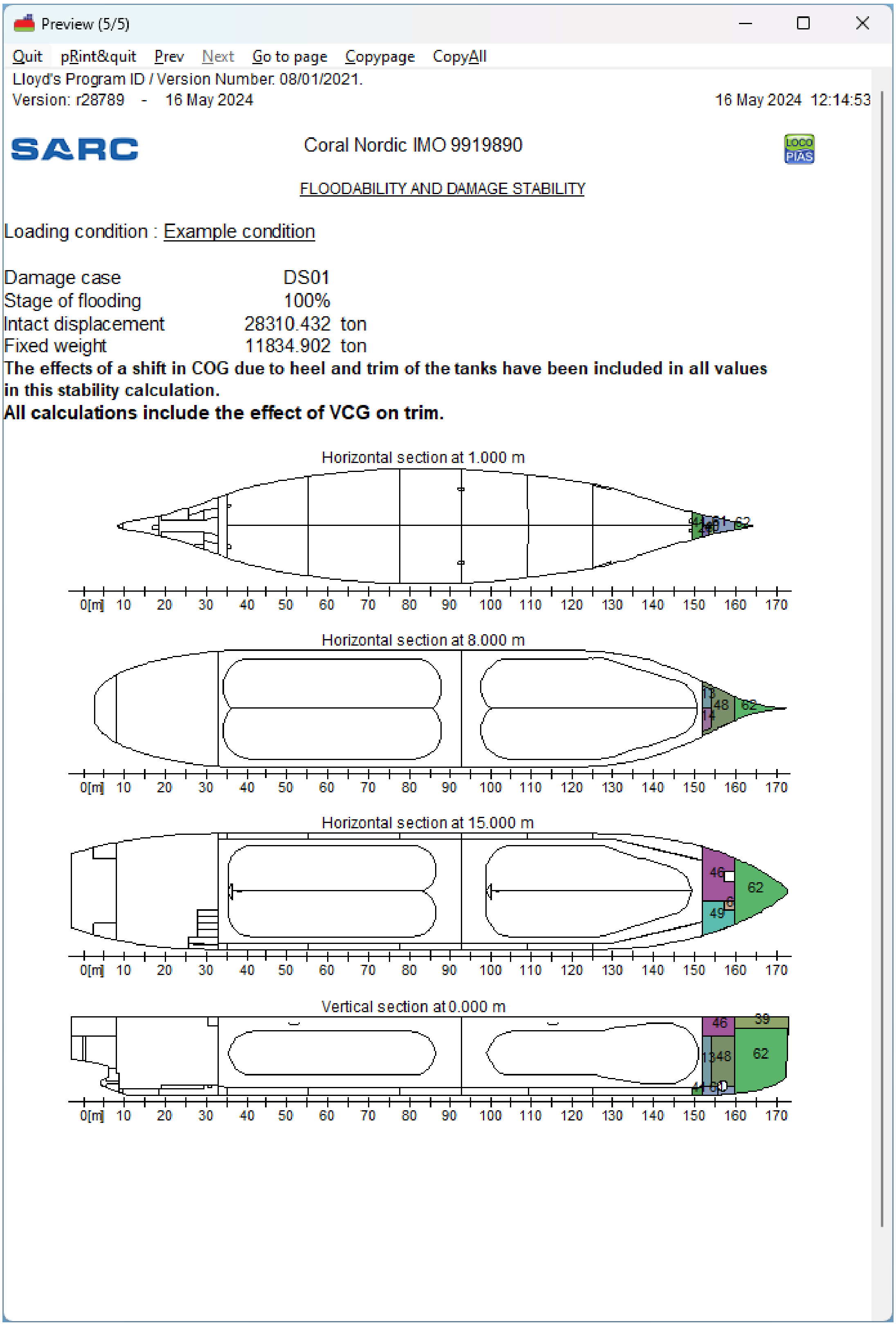
The effects of a shift in COG due to heel and trim of the tanks have been included in all values in this stability calculation.

All calculations include the effect of VCG on trim.



6 Chain Locker SB
12 M.G.O. Overflow Tank Fore
13 Fore M.G.O. Tank PS
14 Fore M.G.O. Tank SB
26 Fore Bilge Holding Tank
39 Bosun Store
40 Emergency Sea Chest
41 Echo Sounder & Speed Logger
46 Fore Hydraulic Unit Room
48 Bow Thruster Room
49 N2 Generator Room
61 Void Fore
62 Fore Peak Tank

An example of output of damage stability, page 4/5.



An example of output of damage stability, page 5/5.

17.1.9.2.4 Damage stability (summary)

Example of print-out of summarized damage stability. Actual output and/ or results may differ according the choosen calculation, and/ or output settings.

Preview (1/2)

Quit pRint&quit Prev Next Go to page Copypage CopyAll

Lloyd's Program ID / Version Number: 08/01/2021.
Version: r28245 - 03 November 2023 16 May 2024 11:42:05

SARC DEMO INLAND TANKER

FLOODABILITY AND DAMAGE STABILITY

Configuration : Direct damage & actual shift of liquids
The damage stability and the actual shift of the CoGs of liquids,
will be determined per direct calculation.

Loading condition : Example condition

Loading condition 'Example condition' complies with all calculated damage cases

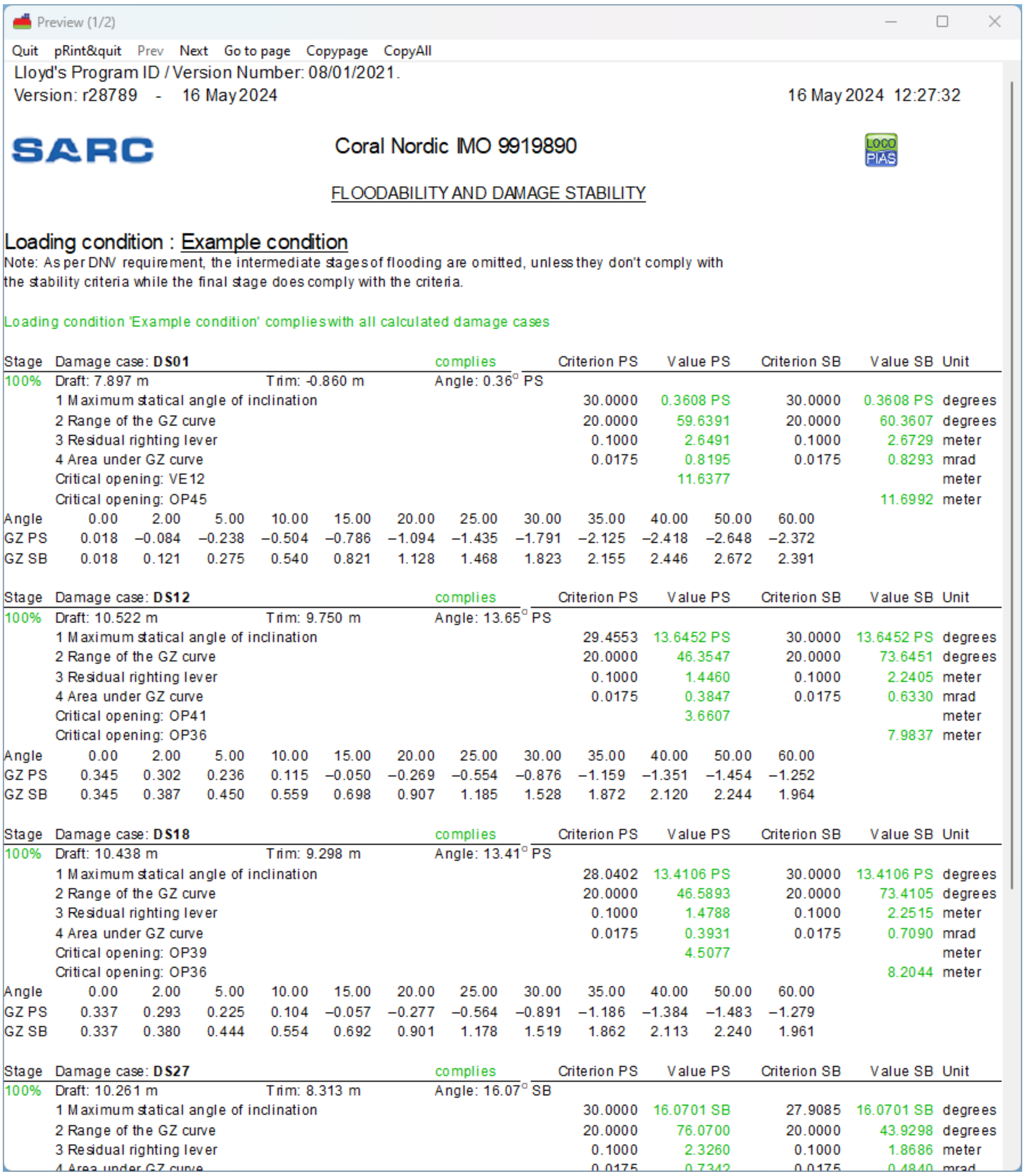
Stage	Damage case: **Engine room SB**	complies	Criterion PS	Value PS	Criterion SB	Value SB	Unit
100%	Draft: 3.572 m Trim: -1.165 m	Angle: 2.74° SB					
	1 Maximum statical angle of inclination		12.0000	2.7399 SB	12.0000	2.7399 SB	degrees
	2 Residual righting lever		0.0500	0.3727	0.0500	0.2836	meter
	3 Area under the GZ curve up to 27 degrees		0.0065	0.0920	0.0065	0.0395	mrad
	4 Distance between waterline and open openings		0.1000	1.7990	0.1000	1.7990	meter
	5 Distance between waterline and weathertight openings		0.1000	1.6519	0.1000	1.6519	meter
	6 Distance between waterline and emergency exits		0.1000	2.1296	0.1000	2.1296	meter
75%	Draft: 3.505 m Trim: -0.800 m	Angle: 2.58° SB					
	7 Residual righting lever		0.0300	0.4111	0.0300	0.3306	meter
	8 Range of the GZ curve		5.0000	32.5824	5.0000	19.3934	degrees
50%	Draft: 3.439 m Trim: -0.448 m	Angle: 1.61° SB					
	7 Residual righting lever		0.0300	0.4516	0.0300	0.3867	meter
	8 Range of the GZ curve		5.0000	31.6138	5.0000	22.2005	degrees
25%	Draft: 3.373 m Trim: -0.097 m	Angle: 0.47° SB					
	7 Residual righting lever		0.0300	0.4912	0.0300	0.4471	meter
	8 Range of the GZ curve		5.0000	30.4671	5.0000	25.2717	degrees
#1	Draft: 3.497 m Trim: -0.765 m	Angle: 3.05° SB					
	1 Maximum statical angle of inclination		12.0000	3.0460 SB	12.0000	3.0460 SB	degrees
	2 Residual righting lever		0.0500	0.5022	0.0500	0.4176	meter
	3 Area under the GZ curve up to 27 degrees		0.0065	0.1242	0.0065	0.0677	mrad
	4 Distance between waterline and open openings		0.1000	1.8240	0.1000	1.8240	meter
	5 Distance between waterline and weathertight openings		0.1000	1.8733	0.1000	1.8733	meter
	6 Distance between waterline and emergency exits		0.1000	2.3865	0.1000	2.3865	meter
#2	Draft: 3.557 m Trim: -1.086 m	Angle: 4.14° SB					
	1 Maximum statical angle of inclination		12.0000	4.1386 SB	12.0000	4.1386 SB	degrees
	2 Residual righting lever		0.0500	0.3939	0.0500	0.2827	meter
	3 Area under the GZ curve up to 27 degrees		0.0065	0.1020	0.0065	0.0370	mrad
	4 Distance between waterline and open openings		0.1000	1.7081	0.1000	1.7081	meter
	5 Distance between waterline and weathertight openings		0.1000	1.6809	0.1000	1.6809	meter
	6 Distance between waterline and emergency exits		0.1000	2.1621	0.1000	2.1621	meter

Stage	Damage case: **Engine room PS**	complies	Criterion PS	Value PS	Criterion SB	Value SB	Unit
100%	Draft: 3.567 m Trim: -1.139 m	Angle: 2.84° PS					
	1 Maximum statical angle of inclination		12.0000	2.8395 PS	12.0000	2.8395 PS	degrees
	2 Residual righting lever		0.0500	0.3490	0.0500	0.3181	meter
	3 Area under the GZ curve up to 27 degrees		0.0065	0.0777	0.0065	0.0531	mrad
	4 Distance between waterline and open openings		0.1000	1.9331	0.1000	1.9331	meter

An example of output of sumarized damage stability.

17.1.9.2.5 Damage stability (summary DNV)

Example of print-out of summarized damage stability according the format of Det Norske Veritas (DNV). As per DNV requirement, the intermediate stages of flooding are omitted, unless they don't comply with the stability criteria while the final stage does comply with the criteria. Actual output and/ or results may differ according the choosen calculation, and/ or output settings.



Preview (1/2)

Quit pRint&quit Prev Next Go to page Copypage CopyAll

Lloyd's Program ID / Version Number: 08/01/2021.
Version: r28789 - 16 May 2024 16 May 2024 12:27:32

SARC Coral Nordic IMO 9919890

FLOODABILITY AND DAMAGE STABILITY

Loading condition : Example condition
Note: As per DNV requirement, the intermediate stages of flooding are omitted, unless they don't comply with the stability criteria while the final stage does comply with the criteria.

Loading condition 'Example condition' complies with all calculated damage cases

Stage	Damage case: DS01	complies	Criterion PS	Value PS	Criterion SB	Value SB	Unit
100%	Draft: 7.897 m Trim: -0.860 m	Angle: 0.36° PS					
	1 Maximum statical angle of inclination		30.0000	0.3608 PS	30.0000	0.3608 PS	degrees
	2 Range of the GZ curve		20.0000	59.6391	20.0000	60.3607	degrees
	3 Residual righting lever		0.1000	2.6491	0.1000	2.6729	meter
	4 Area under GZ curve		0.0175	0.8195	0.0175	0.8293	mrad
	Critical opening: VE12			11.6377			meter
	Critical opening: OP45					11.6992	meter

Angle	0.00	2.00	5.00	10.00	15.00	20.00	25.00	30.00	35.00	40.00	50.00	60.00
GZ PS	0.018	−0.084	−0.238	−0.504	−0.786	−1.094	−1.435	−1.791	−2.125	−2.418	−2.648	−2.372
GZ SB	0.018	0.121	0.275	0.540	0.821	1.128	1.468	1.823	2.155	2.446	2.672	2.391

Stage	Damage case: DS12	complies	Criterion PS	Value PS	Criterion SB	Value SB	Unit
100%	Draft: 10.522 m Trim: 9.750 m	Angle: 13.65° PS					
	1 Maximum statical angle of inclination		29.4553	13.6452 PS	30.0000	13.6452 PS	degrees
	2 Range of the GZ curve		20.0000	46.3547	20.0000	73.6451	degrees
	3 Residual righting lever		0.1000	1.4460	0.1000	2.2405	meter
	4 Area under GZ curve		0.0175	0.3847	0.0175	0.6330	mrad
	Critical opening: OP41			3.6607			meter
	Critical opening: OP36					7.9837	meter

Angle	0.00	2.00	5.00	10.00	15.00	20.00	25.00	30.00	35.00	40.00	50.00	60.00
GZ PS	0.345	0.302	0.236	0.115	−0.050	−0.269	−0.554	−0.876	−1.159	−1.351	−1.454	−1.252
GZ SB	0.345	0.387	0.450	0.559	0.698	0.907	1.185	1.528	1.872	2.120	2.244	1.964

Stage	Damage case: DS18	complies	Criterion PS	Value PS	Criterion SB	Value SB	Unit
100%	Draft: 10.438 m Trim: 9.298 m	Angle: 13.41° PS					
	1 Maximum statical angle of inclination		28.0402	13.4106 PS	30.0000	13.4106 PS	degrees
	2 Range of the GZ curve		20.0000	46.5893	20.0000	73.4105	degrees
	3 Residual righting lever		0.1000	1.4788	0.1000	2.2515	meter
	4 Area under GZ curve		0.0175	0.3931	0.0175	0.7090	mrad
	Critical opening: OP39			4.5077			meter
	Critical opening: OP36					8.2044	meter

Angle	0.00	2.00	5.00	10.00	15.00	20.00	25.00	30.00	35.00	40.00	50.00	60.00
GZ PS	0.337	0.293	0.225	0.104	−0.057	−0.277	−0.564	−0.891	−1.186	−1.384	−1.483	−1.279
GZ SB	0.337	0.380	0.444	0.554	0.692	0.901	1.178	1.519	1.862	2.113	2.240	1.961

Stage	Damage case: DS27	complies	Criterion PS	Value PS	Criterion SB	Value SB	Unit
100%	Draft: 10.261 m Trim: 8.313 m	Angle: 16.07° SB					
	1 Maximum statical angle of inclination		30.0000	16.0701 SB	27.9085	16.0701 SB	degrees
	2 Range of the GZ curve		20.0000	76.0700	20.0000	43.9298	degrees
	3 Residual righting lever		0.1000	2.3260	0.1000	1.8686	meter
	4 Area under GZ curve		0.0175	0.7342	0.0175	0.4840	mrad

An example of output of sumarized damage stability, according DNV.

The specific modules to handle various types of cargo are discussed in detail in chapter 17 on page 324, Loading tools.

17.2 Graphical User Interface for tank filling

In the Tanks module you can manipulate the filling of tanks of the vessel for the loading condition under consideration.

Note

A video[1] exists in which the operation of this module is demonstrated.

[1] https://youtu.be/qSkZHbM2lp4

17.2.1 Layout

Graphical tank filling.

1 Menu bar
Basic functionalities are accessible through the menu bar.

2 Module-buttons
These buttons navigate to other modules, or back to the [Main screen].

3 Function-buttons
Special functions of the tank module.

4 Tank group buttons
Click to display a group of tanks of the same type.

5 List of tanks
Displays the list of tanks of the selected tank group.

6 Tank information
This window gives information of the selected tank. If multiple tanks are selected it gives the following message: Multiple tanks selected. The window lists the name, weight, volume, center of gravity, etc. of the selected tank. The center of gravity is calculated from the other input, which can be changed by clicking the appropriate line. An input box will appear to define the desired value.

7 Track bar
The track bar can be used to change the filling percentage of the selected tank(s).

8 Section windows
Displays top view, vertical section, and cross section. Active sections show a section of the vessel at the center of gravity of the selected tank. Fixed sections show sections at predefined locations.

9 Status bar
Gives information about the total weight of the selected tank group and which information is visible in the graphical tanks.

17.2.2 General approach

1. Select. A tank can be selected by left-clicking a tank in a *section* window 8 . Tanks can be selected by clicking near their center of gravity. A selected tank will be hatched black and white in the views. In the cross section, the actual fluid level in a tank is indicated.
2. Edit. The contents of a tank can be edited by right-clicking a tank after selecting it.
3. Output/Totals. Go to the menu [Output]→[Totals] for an overview of total weight of the selected tank group on screen.

17.2.2.1 Select

You can select tanks in one of the ways below. A selected tank is marked by black on white cross-hatching in the section windows.

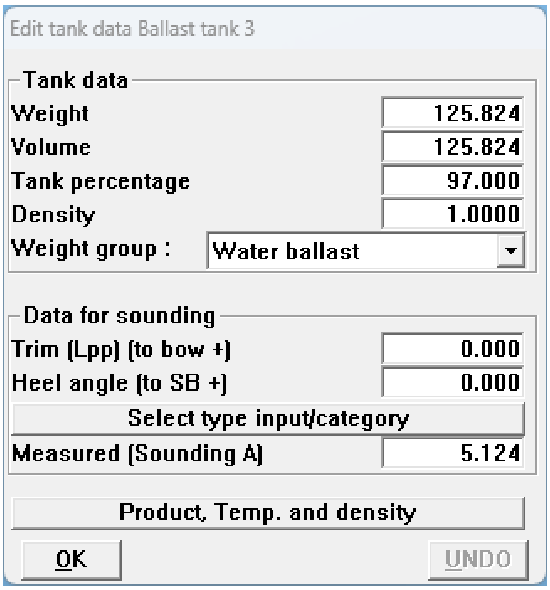
- Left-click a tank in the List of tanks-window 5 .
- Left-click a tank in one of the Section windows 8 .
- Select multiple tanks by holding the left mouse button to drag a selection box in one of the Section windows 8 .
- Select all visible tanks by pressing <Ctrl+a>.
- Add or remove a tank to/from a selection by holding Ctrl and clicking the tank in a Section window 8 , or in the List of tanks-window 5 .

The information of this tank is now shown in de Tank information-window 6 .

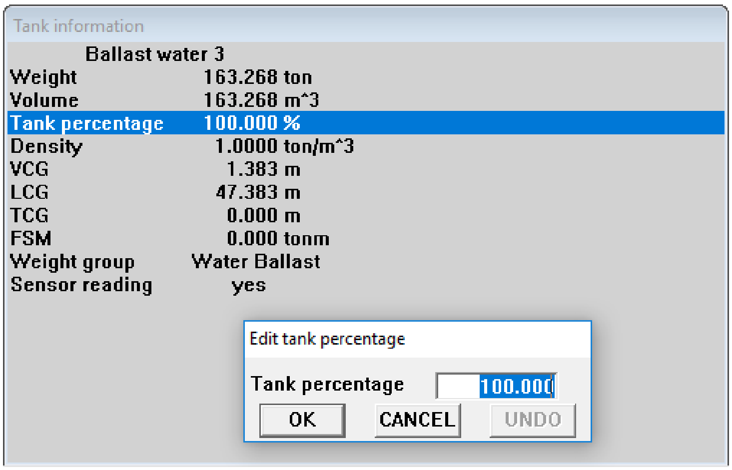
17.2.2.2 Edit

Once a tank has been selected, there are several ways to edit tank data:

- Double-click a tank in the [List of tanks]-window 5 to open the input form 'Edit tank data'.
- Right-click a tank in one of the Section windows 8 to open the input form 'Edit tank data' of the selected tank(s). When only one tank is selected, all tank data can be edited. When more than one tank is selected, only filling percentage and density of the content can be changed.
- Double-click a value in the [Tank information]-window 6 to edit that specific value, see figure below.
- Drag the track bar 7 to change the amount of fluid of the selected tank.
- Drag the surface of the content of a selected tank.
- Double click a tank to empty it or fill it to the maximum filling percentage. Use [Settings]→[Filling percentages] from the menu bar to edit the default filling percentage.
- Enter a sounding, ullage or pressure and apply temperature corrections. By right-clicking in the Section window 8 , additional fields become available in the 'Edit tank data'-window when a sounding pipe or pressure sensor has been defined. By entering trim and heeling angle together with the measured value, the tank volume is calculated according to the sounding data and input for floating position. For temperature corrections see section 16.2.1.4 on page 312, Product, temperature and density.
- Pump with track bar. Select two tanks from the same weight group, with the same density and go to [Pump] in the upright corner. Now the track bar enables you to pump fluid from one tank to another tank.



Edit tank data from List of tanks/Section-windows.

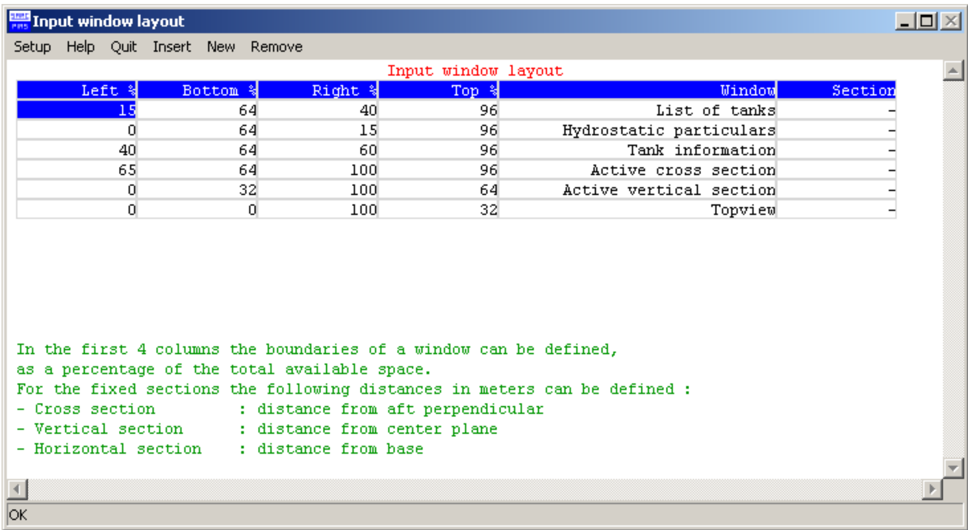


Edit tank data from Tank information.

17.2.3 Menu bar

17.2.3.1 Window layout

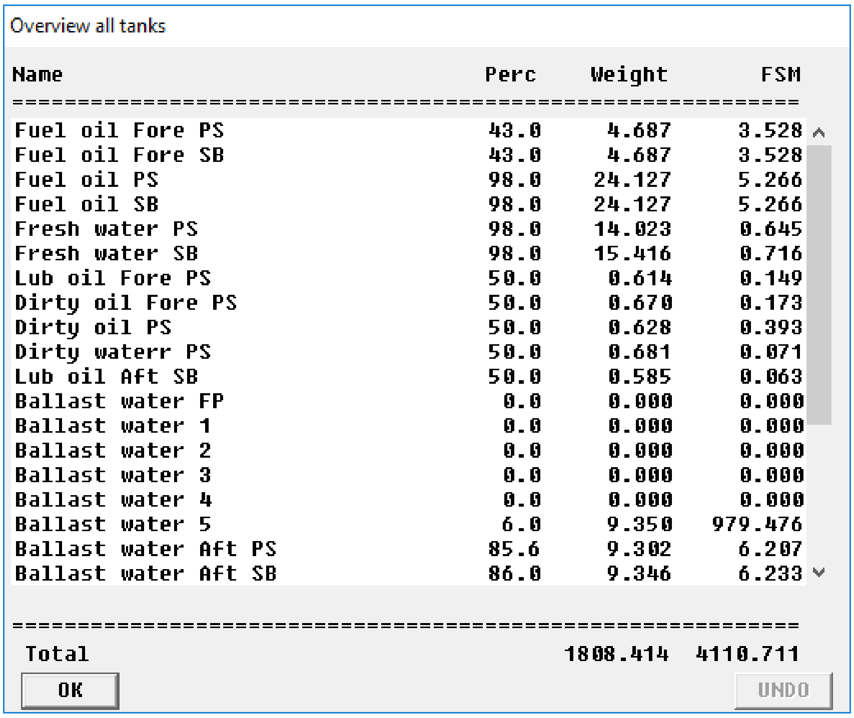
With [Window]→[Define window layout] the initial window layout can be defined. The main window contains a number (with a maximum of nine) of subwindows from which the size and position can be set here. In the leftmost four columns the position of the four edges of each window are defined as a percentage of the height and breadth of the main window. In the column 'window' the type of window can be given. If the window type is a 'fixed cross section', the cross section location can be given in the last column. In the figure below the settings for the layout of the figure above are listed. By the way, [Window]→[Default window layout] brings back the original layout.



Define window layout.

17.2.3.2 Output/Totals

With the [Output]→[Totals] option an overview of the weights of the tanks of the selected tank group is presented, as well as the total weight (at the bottom of the popup box that appears). By the way, the total weight of the selected weight group is always printed in the status bar 9 of this module.



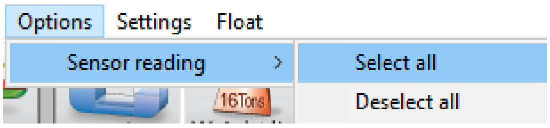
Overview all tanks

Name	Perc	Weight	FSM
Fuel oil Fore PS	43.0	4.687	3.528
Fuel oil Fore SB	43.0	4.687	3.528
Fuel oil PS	98.0	24.127	5.266
Fuel oil SB	98.0	24.127	5.266
Fresh water PS	98.0	14.023	0.645
Fresh water SB	98.0	15.416	0.716
Lub oil Fore PS	50.0	0.614	0.149
Dirty oil Fore PS	50.0	0.670	0.173
Dirty oil PS	50.0	0.628	0.393
Dirty waterr PS	50.0	0.681	0.071
Lub oil Aft SB	50.0	0.585	0.063
Ballast water FP	0.0	0.000	0.000
Ballast water 1	0.0	0.000	0.000
Ballast water 2	0.0	0.000	0.000
Ballast water 3	0.0	0.000	0.000
Ballast water 4	0.0	0.000	0.000
Ballast water 5	6.0	9.350	979.476
Ballast water Aft PS	85.6	9.302	6.207
Ballast water Aft SB	86.0	9.346	6.233
Total		1808.414	4110.711

OK UNDO

17.2.3.3 Options

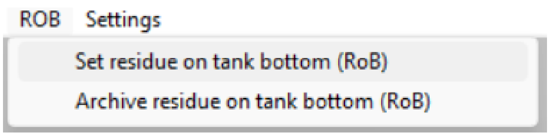
17.2.3.3.1 Sensor Reading



With this option tanks can be selected of which the data must be read from the tank measuring system.

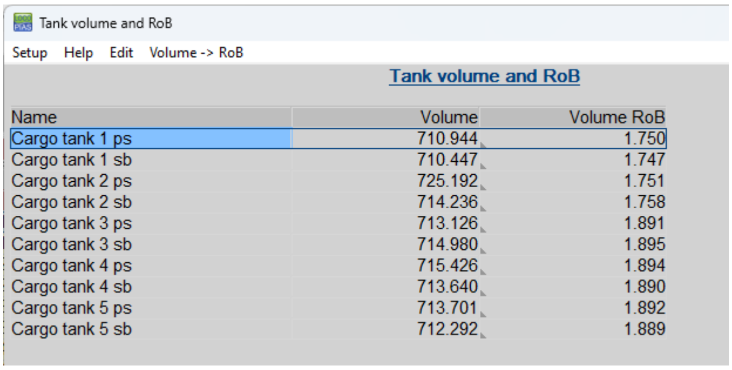
17.2.3.4 RoB (Residue on tank bottom)

17.2.3.4.1 Set residue on tank bottom (RoB)



This option allows users to set the residue on bottom (RoB) values for each cargo tank in a single menu. The purpose of this functionality is to assist users to manually adjust the RoB value of each tank, or initialize it by copying the tank volume value in it.

By selecting [RoB]→[Set residue on tank bottom (RoB)] from the toolbar, the user is transferred in a menu where all the tanks are listed, with their volume and RoB values in the corresponding columns. The user is allowed to adjust only the RoB values of the menu. However, giving a new RoB value to a tank, it may affect the tank volume field as well.

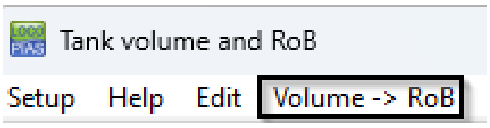


Name	Volume	Volume RoB
Cargo tank 1 ps	710.944	1.750
Cargo tank 1 sb	710.447	1.747
Cargo tank 2 ps	725.192	1.751
Cargo tank 2 sb	714.236	1.758
Cargo tank 3 ps	713.126	1.891
Cargo tank 3 sb	714.980	1.895
Cargo tank 4 ps	715.426	1.894
Cargo tank 4 sb	713.640	1.890
Cargo tank 5 ps	713.701	1.892
Cargo tank 5 sb	712.292	1.889

Residue on tank bottom (RoB) menu.

The user is able to adjust the RoB values in two ways:

1. By copying the volume value of the tank in the RoB value. This can be done by using the [Volume -> RoB] option from the toolbar.

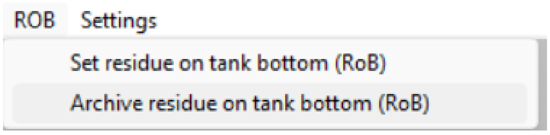


2. By manually entering the RoB values in the corresponding cell.

Both ways can be applied on a single tank or on a multiple tank selection. In both cases, the new input value is being checked and if it exceeds the 10% of the total tank volume, a confirmation message pops up in order to inform the user about the current adjustment. By confirming the popup message, the new RoB value is being assigned to the tank. Otherwise, the adjustment is dismissed.

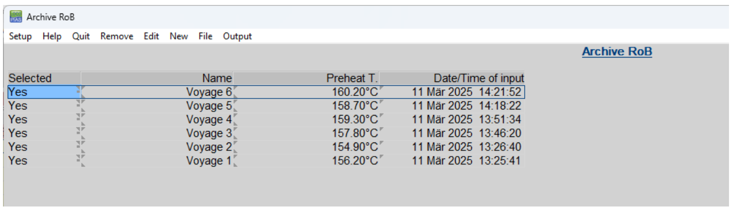
By exiting the menu, all changes are saved and the tanks have now been updated with the new volume and RoB values.

17.2.3.4.2 Archive residue on tank bottom (RoB)



With this option the user is able to see an overview of all RoB archive entries. The purpose of this function is to collect and preview all archive entries in one menu, where each entry contains a set of the main data per tank, for all tanks, which are directly taken for the current loading condition.

By selecting [RoB]→[Archive residue on tank bottom (RoB)] the user is entering into the menu, where he can see all the available archives saved in the system, sorted from the most recent one (on top of the list) to the oldest one (bottom of the list).



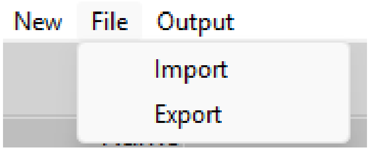
Selected	Name	Preheat T.	Date/Time of input
Yes	Voyage 6	160.20°C	11 Mar 2025 14:21:52
Yes	Voyage 5	158.70°C	11 Mar 2025 14:18:22
Yes	Voyage 4	159.30°C	11 Mar 2025 13:51:34
Yes	Voyage 3	157.80°C	11 Mar 2025 13:46:20
Yes	Voyage 2	154.90°C	11 Mar 2025 13:26:40
Yes	Voyage 1	156.20°C	11 Mar 2025 13:25:41

Archive Residue on tank bottom (RoB) menu.

The user is able to create a new archive by selecting the toolbar option [New]. In this way, a new archive entry will be added on top of the list. The name of the entry will initially be filled with the name of the current loading condition. The date and time field is the creation date/time and is filled automatically when the archive entry is being created. For each archive, the user can edit the name and fill the preheating temperature field, as well as select/deselect the archive for output and/or export.

An archive can be deleted by selecting the [Remove] option from the toolbar, after confirming the action through a popup window.

With the [File]→[Export] option, the user can select a file location and a file name, and export the selected entries in a file with (.rob) extension. In the same way, an (.rob) file can be imported through the [File]→[Import] option and all the archive entries in that file will be added to the list. The archives will be still sorted by date and any double entries will be skipped.



With the [Output] option, a report of the selected entries can be printed. When more than 1 archive is selected for output, 3 graphs are added to the output showing the progression over time.

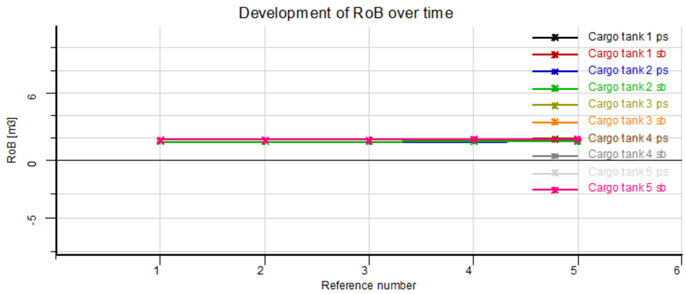
- Reference no. / RoB volume
- Reference no. / Cargo temperature
- Reference no. / Preheat temperature

ARCHIVE OF DEVELOPMENT ROB

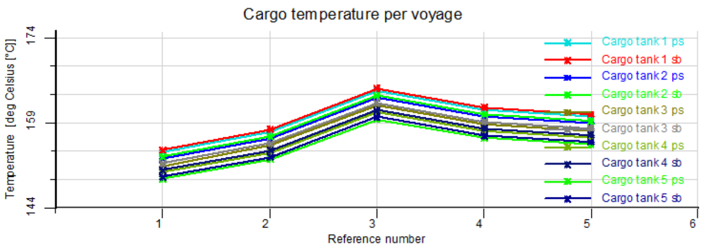
Ref.nr.	Description	Preheat T. deg Celsius [°C]	Date & time of archive
1	Voyage 1	156.20	20 Jan 2025 12:30:08
2	Voyage 2	154.90	20 Jan 2025 12:33:25
3	Voyage 3	157.80	20 Jan 2025 12:35:40
4	Voyage 4	159.30	20 Jan 2025 12:36:56
5	Voyage 5	158.70	20 Jan 2025 12:38:19

Ref.nr.	Compartments	RoB [m3]	Temperature deg Celsius [°C]	Density in air 15°C [t/m3]	Product table	Product
1	Cargo tank 1 ps	1.692	153.8	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
2	Cargo tank 1 ps	1.703	157.3	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
3	Cargo tank 1 ps	1.707	164.6	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
4	Cargo tank 1 ps	1.722	161.3	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
5	Cargo tank 1 ps	1.750	160.1	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
1	Cargo tank 1 sb	1.690	154.2	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
2	Cargo tank 1 sb	1.701	157.7	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
3	Cargo tank 1 sb	1.705	165.0	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
4	Cargo tank 1 sb	1.720	161.7	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
5	Cargo tank 1 sb	1.747	160.5	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
1	Cargo tank 2 ps	1.694	152.6	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
2	Cargo tank 2 ps	1.705	156.1	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
3	Cargo tank 2 ps	1.709	163.3	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
4	Cargo tank 2 ps	1.724	160.0	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
5	Cargo tank 2 ps	1.751	158.8	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
1	Cargo tank 2 sb	1.700	153.0	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
2	Cargo tank 2 sb	1.711	156.5	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
3	Cargo tank 2 sb	1.715	163.7	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
4	Cargo tank 2 sb	1.730	160.5	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
5	Cargo tank 2 sb	1.758	159.3	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
1	Cargo tank 3 ps	1.829	151.4	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
2	Cargo tank 3 ps	1.841	154.9	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
3	Cargo tank 3 ps	1.845	162.1	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
4	Cargo tank 3 ps	1.861	158.8	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
5	Cargo tank 3 ps	1.891	157.6	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen

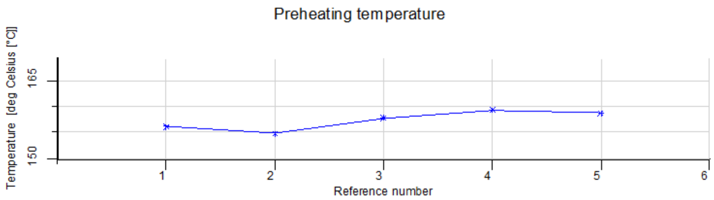
Archive Residue on tank bottom (RoB) output tables.



Archive RoB: Development of RoB over time graph.



Archive RoB: Cargo temperature per voyage graph.

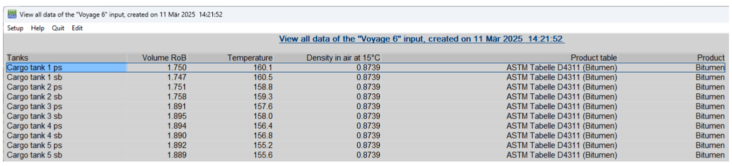


Archive RoB: Preheating temperature graph.

Furthermore, each archive entry can be entered (double-click or [Enter] on the row), and the user is presented the following data for all cargo tanks of that specific entry:

- RoB value
- Temperature
- Density on Air 15 degrees Celsius
- Product table
- Product

Note that in this overview, the information is just for preview and thus, the user cannot modify any values of the archived entries.



View all data of the "Voyage 6" input, created on 11 Mär 2025 14:21:52

Setup Help Quit Edit

View all data of the "Voyage 6" input, created on 11 Mär 2025 14:21:52

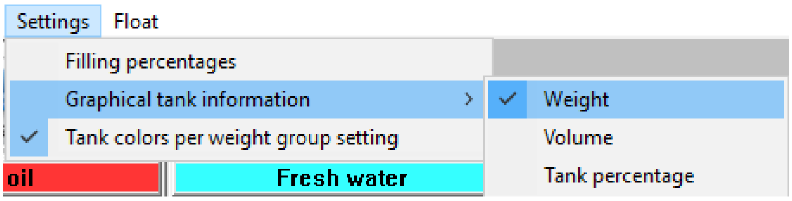
Tanks	Volume RoB	Temperature	Density in air at 15°C	Product table	Product
Cargo tank 1 ps	1.750	160.1	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
Cargo tank 1 sb	1.747	160.5	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
Cargo tank 2 ps	1.751	158.8	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
Cargo tank 2 sb	1.758	159.3	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
Cargo tank 3 ps	1.891	157.6	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
Cargo tank 3 sb	1.895	158.0	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
Cargo tank 4 ps	1.894	156.4	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
Cargo tank 4 sb	1.890	156.8	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
Cargo tank 5 ps	1.892	155.2	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen
Cargo tank 5 sb	1.889	155.6	0.8739	ASTM Tabelle D4311 (Bitumen)	Bitumen

Archive RoB entry menu: Data overview per cargo tank.

By exiting the menu, all adjustments made on the RoB archive entries are being saved.

17.2.3.5 Settings

In [Settings] you can find the option 'Filling percentages', an option for displaying graphical tank information and an option to select the color of the tanks: individually or per tank group. Under [Settings] it is also possible to show all tanks of the same weight group in the color of that tank group by enabling the setting in the menu [Settings]→[Tank colours per weight group setting].



17.2.3.6 Result windows

These options are the same as in the list of weight items, see section 16.2.1 on page 307, Define/edit weight items, option [Window]→[Result windows].

17.2.4 Function buttons

17.2.4.1 Sensor reading

With this option the tank volumes, and possibly other data, are automatically read from the tank measurement system which is used on board.

17.2.4.2 Pump

With this option the contents of a tank can be pumped from one tank to another of the same tank group. First select two tanks of the same tank group (with a selection-window or with <Ctrl>), then select the option [Pump] from the Function-buttons. Now it is possible to pump the fluid with the trackbar. In the Tank information window the data of one of these tanks will be displayed. During pumping the total volume of the contents will remain the same.

17.3 Generation of loading conditions for simulation of Ro-Ro operations

This facility is in particular intended for the simulation of Ro-Ro operations. Starting from a mother loading condition and a number of given coupled loadings which shift along a given route, you can generate many (interim) loading conditions. As for its functions, the generated loading conditions are completely identical to regular loading conditions and they can be processed and computed with Loading.

Generation of loading conditions for Ro-Ro operations

1. Entering coupled loads
2. Entering three-dimensional route followed by the loads
3. Entering the number of steps for the generation
4. Entering the name of the mother loading condition
5. Generate all loading conditions

17.3.1 Entering coupled loads

An input screen appears, where maximally ten coupled loads (such as a set of Ro-Ro vehicles) can be given. The columns have the following meanings:

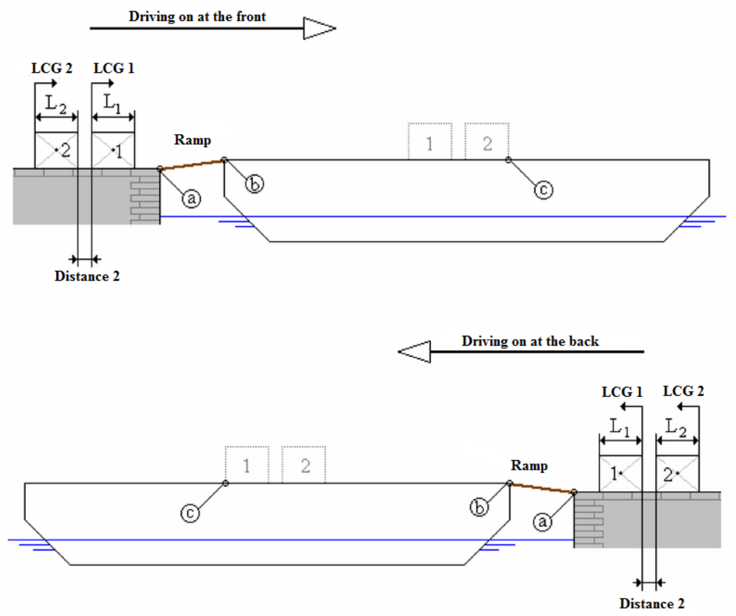
- Name: The name given to a load.
- Weight: The weight (in ton) of that load.
- Length: The length (m) of that load.
- LCG: The center of gravity in length (in m) of that load, measured from the back of the load.

- TCG: The center of gravity in breadth (in m) of that load, measured from the route the load is following. A center of gravity at the right side of the route (seen when one follows the route) is positive, a center of gravity at the left is negative.
- VCG: The center of gravity in height (in m) of that load, measured from the bottom of the load, so from the route the load is following.
- Distance: The distance (in m) from the front of this load to the back of the next load. This distance may, for instance, be equal to the pole length of a vehicle. The distance belonging to the first load is not used.

17.3.2 Entering three-dimensional route followed by the loads

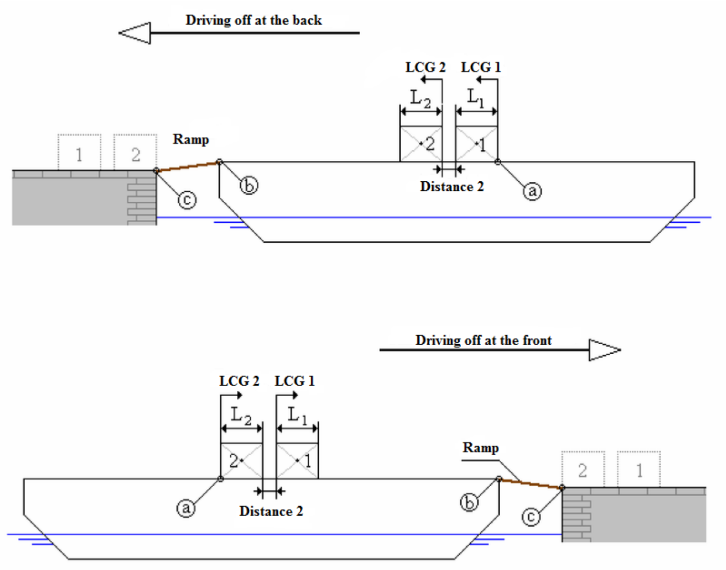
An input screen appears, where the route of the loads in the ship can be entered. The loads are assumed to follow that route, with the bottom of the loads along that route. Length, breadth and height coordinates of maximally twenty points of that three-dimensional route can be entered (in meters, in the co-ordinate system of the ship). The last column of the input screen is the column 'ramp'. At the first or the second line, the column 'ramp' can be filled with 'yes', indicating that there is a ramp at that side along which the Ro-Ro operation takes place. The point where the column 'ramp' has been filled by 'yes' is ashore, the other points are in or on the ship. The road between the two first or the two last points ((a) and (b) when driving up and (b) and (c) when driving off on the sketches) indicates the position and the dimensions of the ramp. The loads can start shifting in two manners:

- Driving up of the loads: When the column 'ramp' of the first line contains 'yes' the loads are not yet in the ship. The loads will start to shift then from the point where the front of the first load coincides with the coordinates of the first line.



Driving up the loads.

- Driving off of the loads When the column 'ramp' of the last line contains 'no'r the loads are already in the ship. The loads will start to shift then from the point where the back of the last load coincides with the coordinates of the first line.



Driving off the loads.

17.3.3 Entering the number of steps for the generation

Here is entered in how many steps the route must be divided, so how many loading conditions are generated. The minimum number of steps is 2, the maximum number is 35.

17.3.4 Entering the name of the mother loading condition

Any generated loading condition is composed of two parts:

- The weights of all loads, at the correct position along the route.
- Other occurring weight items, which are copied from the so-called ‘mother loading condition’. The name of that mother loading condition can be given at this option by filling the column for the correct loading condition with ‘yes’.

17.3.5 Generate all loading conditions

By choosing this option the loading conditions are generated. Due to an internally used numerical integration procedure it may occur that the weights of the loads in the generated loading conditions do not correspond with the given weights. However, the deviation (of maximally 0,02%) is so minor that it does not affect the computational results.

17.4 Graphical User Interface for container loading

Attention

> Before using this GUI it is essential that all ship’s container provisions, such as castings and bay / row / tier positions, are defined using the Cargoquip module.

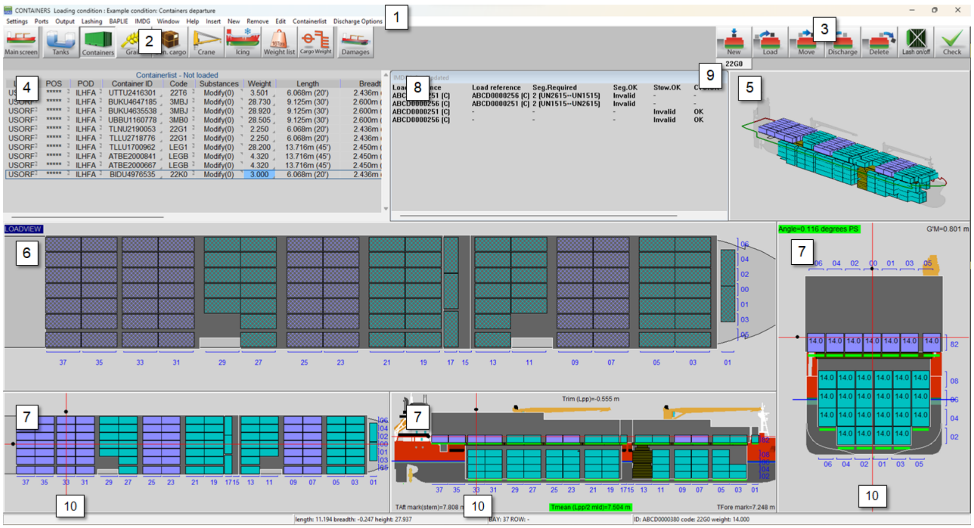
The container loading module is designed to define a particular container loading. LOCOPIAS updates the situation and informs you about the consequences for the vessel. This module is essential for vessels with a significant container capacity. It allows for the interactive positioning of containers of any size, and contains numerous loading options, amongst which electronic data exchange. Some highlights of this module are:

- *The module is founded upon a 3D representation of container distribution. It allows the user to show any desired combination of rows, bays and tiers, and to work in a sequence and orientation selected by the user.*
- *Suitable for all kinds of containers. The module has no restrictions at all with regard to the container type (20’, 30’, 40’, 45’, 48’, 52’ or every other length, with random breadth and height of each container) or loading combination. Refrigerated containers are also supported.*

- *Drawings and lists of container loading details.*
- *At any desired moment, stability or strength particulars can be evaluated and verified against the relevant criteria.*
- *Only consistent container loading is accepted.*
- *Database management functions for import and export of container data and loading conditions.*
- *Integrates seamlessly with LOCOPIAS' line of sight module.*
- *Container cargo positioned above deck is automatically included in the calculation of the wind contour of the vessel.*

17.4.1 Layout

A typical layout of the [Containers]-module is shown below. Its elements are labeled with a number and described underneath. The slots are generated automatically according to the type of container that is to be loaded.



Container module.

1 Menu bar
Basic functionalities are accessible through the menu bar.

2 Module-buttons
These buttons navigate to another module, or back to the [Main screen].

3 Function buttons
Main functions of the [Container]-module. These functions are also mapped to a keyboard key combination, see section 17.4.2.7 on page 366, Function keys.

4 Containerlist
Displays the containers that match the view options selected from the [Containerlist..]→[View] menu. You can choose between loaded, not loaded and all containers. It is also possible to copy/paste from Excel in this list.

5 3D View
3D view of the full vessel.

6 Loading view
This is the main work window of the container module. All the functions (new, load, move, discharge, delete) happen through this window.

7 Section windows
These windows show the layout of the bay, row and tier of the selected container as well as trim, draft, heeling angle, GM and actual waterline.

8 IMDG
IMDG information. If IMDG is not enabled, this window is omitted.

9 Container type/code button
With these button you can select the container type you want to load.

10 Navigation Lines

The navigation lines are present only in the Section windows 7 . Right-clicking in any of the Section windows will update the views. You can also left-click and drag the black dots.

Note

The bays and rows are always visible. The tiers are drawn when containers are loaded. For the tier numbering, see section 17.4.2.5 on page 366, Tier numbering.

17.4.2 General approach

There are three ways to load containers with the Container module. You can load a new container with the [New]-button, you can create a list of containers and load these with the [Load]-button, or you can use a BAPLIE file, see section 17.4.3.5 on page 368, BAPLIE. In general, the following approach can be used:

1. section 17.4.2.1 on this page, Select. You can select (multiple) containers.
2. section 17.4.2.2 on the current page, Load.
3. section 17.4.2.3 on the following page, Edit. After loading, you can edit the data of a container, discharge a container, switch a container from one container slot to another, discharge a container to the quay or permanently delete a container.
4. Check results and create output. Use the [Check]-button, or the [Window]→[Result windows] windows to verify your loading condition, then print the output.
5. section 17.4.3.3 on page 368, Output.

17.4.2.1 Select

You can select a container in one of these ways:

- Left-click a container in the List of containers 4 .
- Left-click a container in the Loadview 6 or any one of the Section windows 7 . All the function buttons have to be unpressed in order to select.

To select multiple loaded containers, drag cursor in one of the Section windows 7 to create a selection box. You can then right-click and choose your action from the options available. See also section 17.4.2.4 on the following page, Multiple containers. A selected container is highlighted white.

17.4.2.2 Load

Use this function to directly create a new container. You will immediately see the available slots drawn in yellow in the Loadview 6 and Section windows 7 . The type/code of the new container is determined from 9 . You can then left-click in the Loadview 6 on a green slot to position the new container. You cannot position a container in the Section views 7 . To load multiple new containers at once, see section 17.4.2.4 on the next page, Multiple containers.

Use this function to load containers from the list of containers:

1. Go to the menu [Ports]→[Input ports] to enter ports, and optionally a specific color.
2. You can add new containers of type as defined in 9 by pressing [New] in the Menu bar.
3. Edit any container data. You can also copy-paste and/or edit multiple containers at once.
4. Click the [Load]-button. You will immediately see the available slots drawn in yellow in the Loadview 6 and Section windows 7 . You can left-click in the Loadview 6 on a green slot to position the container.

Green indicators under the loaded containers in the Section views 7 turn red when the maximum loading is exceeded. You can zoom in or click on any container on the stack to check the limits. To load using a BAPLIE file, see section 17.4.3.5 on page 368, BAPLIE.

Note

For the selected type of container to be loaded, you will immediately see yellow slots drawn. If you do not see slots drawn then:

- If the vessel is equipped with initial castings and the option [Initial castings] is checked in the [Settings] menu, then there are no available slots for the *selected container type* .
- The vessel is not equipped with initial castings.

In both situations you can still place the container on the bottom, whether with [Placement Assist] (if checked) or freely ([Initial castings] and [Placement Assist] unchecked). After the lowest container has been positioned, the castings will be automatically used for the higher tiers.

17.4.2.3 Edit

To open the [Edit container data] form, you can right-click on a selected loaded container. To edit multiple containers, see section 17.4.2.4 on the current page, Multiple containers.

Use this function to discharge containers from the vessel. Activate the [Discharge] button and left-click on the Loadview 6 to discharge the containers. The discharged containers become available for loading again in the list of containers. You can also go to the menu [Discharge Options]→[Discharge All] and select ‘Discharge to containerlist’ to discharge the entire ship at once.

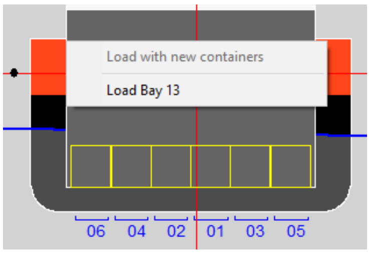
Use this function to delete containers permanently from the vessel. Click the [Delete] button and left-click on the Loadview 6 to delete the container from the vessel, the container cannot be loaded again. You can also go to the menu [Discharge Options]→[Discharge All] and select ‘Delete containers’ to empty the entire ship at once.

17.4.2.4 Multiple containers

You can load new or edit multiple containers in any one of the Section views 7 .

17.4.2.4.1 Load new containers

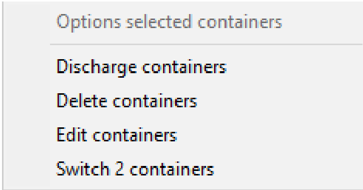
While [New] button is pressed, double-click right mouse button to load the lowest tier. The specific tier to be loaded depends on which Section view was clicked.



Load new multiple containers in bayview.

17.4.2.4.2 Edit containers

Drag cursor in any one of the Section views 7 to create a selection box. You can also hold the CTRL (Control) button and left-click on containers. The following menu will pop up after right-click:



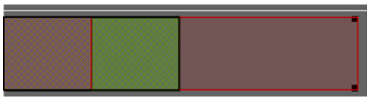
Multiple containers window.

17.4.2.5 Tier numbering

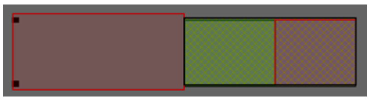
The tier numbering is done according to ISO standards. The tiers start with "02" with the height of an 8 1/2 ft standard container and rise with even numbers for each container height. Tiers on deck start with "82" and rise with even numbers above the hatch covers. Half-height containers are marked with odd numbers. Therefore containers at the same height above the keel have the same tier specification. High-cube containers are treated as standard.

17.4.2.6 Compensation pieces

In cases where containers are to be placed in a specific position but no yellow slot appears there, you can double right-click and try to manually load the container using compensation pieces. The pieces are generated accordingly and the container is positioned. The compensation pieces are automatically removed when the container below them is removed (discharged or deleted). In case this does not occur, they can be manually deleted by the function [Delete]. For proper manual placement using compensation pieces, please refer to the visual guide with 3 examples below.



Case 1. Already loaded 20ft container is highlighted in black outline. The pieces are to be placed in the fore slot to position a 40ft container on top. Green shows the correct area to double right-click.

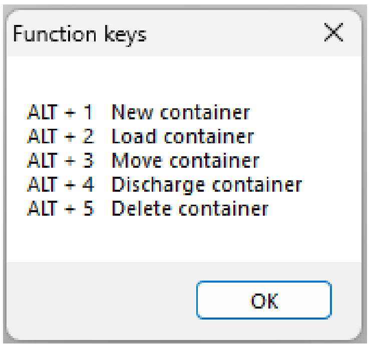


Case 2. Already loaded 20ft container is highlighted in black outline. The pieces are to be placed in the aft slot to position a 40ft container on top. Green shows the correct area to double right-click.

Case 3. No container is loaded underneath. A 45ft is to be loaded using compensation pieces on the fore and the raised tanktop on the aft. Green shows the correct area to double right-click.

17.4.2.7 Function keys

The function buttons are pressed/depressed when the corresponding key combination is used.



Functions keys.

17.4.3 Menu bar

17.4.3.1 Settings

In the [Settings] submenu the following options are available:

[Initial castings]
You can enable/disable the bottom castings, if the vessel is equipped with them.

[Placement Assist]
Placement Assist aids in container positioning. The positions add up automatically to accomodate stacking. It is useful in vessels where there are no initial castings. If [Placement Assist] is off, then you can move and place the container freely.

[Show higher tier slots]
When loading a container, the actual selected slot is always the lowest one. You can enable/disable the display of the rest here.

[Include stacking at extreme ends for types A-P]
Container types A-P have additional castings, which can generate additional available slot positions. You can enable/disable the use of these additional castings here.

[Rotated slots]
You can enable/disable the generation of rotated slots (if they exist).

[Edit container spacer]
Here you can edit the container spacer. You can choose different spacers for different sections of the vessel.

[Unit longitudinal axis]
Here you can choose your default axis; you can choose between 20ft, 40ft, single bays, frame numbers and meters.

[Bay-Row-Tier conversions]
Here you can define new names for bays, rows or tiers.

[Draw castings]
You can enable/disable the drawing of the castings.

[Edit overlap margin]
Here you can allow a margin for an overlap. It is highly recommended to avoid this, unless necessary.

[Draw cargo]
Here you can select to display all other cargo from other modules.

[Collision check]
Here you can select to check for collision with other cargo upon positioning the container.

17.4.3.2 Input

In the [Input] submenu the following options are available:

[Ports]
Here you can insert the ports where the vessel will load and discharge containers. It is also possible to add a color to a port, this can help to organize the containers.

17.4.3.3 Output

In the [Output] submenu the following options are available:

[Settings]
Here you can select container colours depending on port of load, shift, discharge or container type. You can also choose what is displayed on the containers. The selections here will also be visible in the stowage plan. Explanation of coloured circles :

- red : IMDG cargo
- yellow : empty container
- blue : refrigerated container.

[List of containers]
Standard format output of container list with detailed container information.

17.4.3.4 Lashing

[Select calculation method]
Here you can choose the calculation method.

[Reduction variants]
Depending on the selected calculation method, you can choose from all the available reduction variants. Each reduction variant carries a predefined reduction factor. The checked reduction variants will become available for the calculations.

[Configure settings]
Here you can edit some settings that will be applied on every loaded baystack.

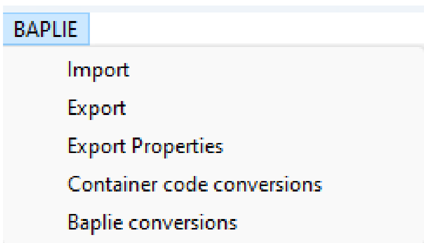
[Baystack selection]
Select baystacks for lashing forces.

[Lashing calculations]
The lashing calculations output is generated for every loaded baystack. The quick settings set in the previous entry are applied. You can work on each baystack separately by entering the [Lashing]-module.

[Lashing plan]
The lashing plan output is generated for every loaded baystack.

17.4.3.5 BAPLIE

With the BAPLIE option, you can read and write container data files with the BAPLIE-format (up till version 3.1).



Dropdown menu options BAPLIE.

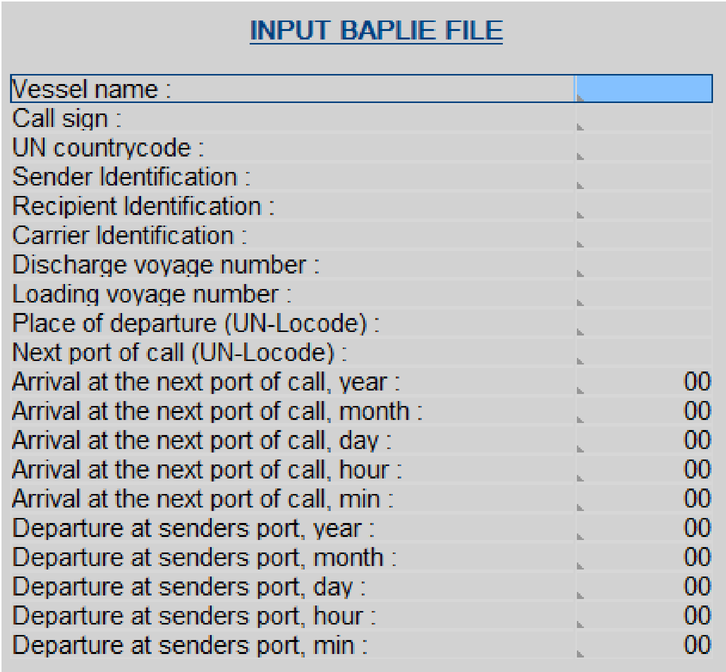
[Import]
After selecting an .edi file, the containers will be automatically loaded. It is possible that there are errors in the file and that some containers cannot be loaded. They will then show up in the Containerlist 4 .

[Export]
Create an .edi BAPLIE file.

[Export Properties]
You can enter the data for writing a BAPLIE file, see figure below. These data will be stored in a file. This information is necessary before you can [Export].

Attention

LOCOPIAS will only read those data which will be used in LOCOPIAS itself and write the data which will be available in LOCOPIAS. This means that after reading and writing a BAPLIE-file, some data will be lost.

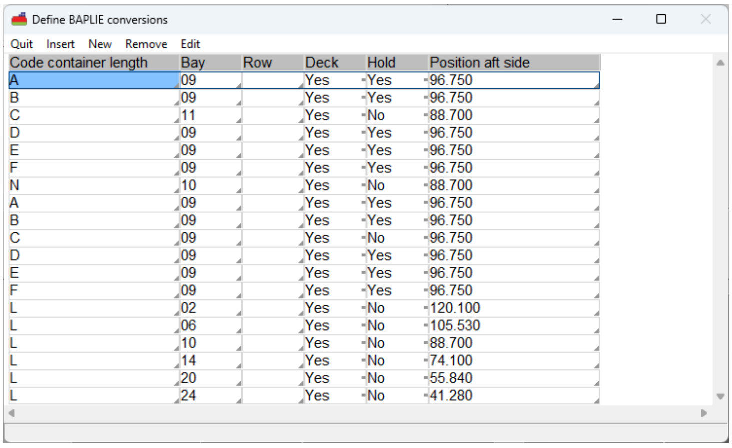


INPUT BAPLIE FILE	
Vessel name :	
Call sign :	
UN countrycode :	
Sender Identification :	
Recipient Identification :	
Carrier Identification :	
Discharge voyage number :	
Loading voyage number :	
Place of departure (UN-Locode) :	
Next port of call (UN-Locode) :	
Arrival at the next port of call, year :	00
Arrival at the next port of call, month :	00
Arrival at the next port of call, day :	00
Arrival at the next port of call, hour :	00
Arrival at the next port of call, min :	00
Departure at senders port, year :	00
Departure at senders port, month :	00
Departure at senders port, day :	00
Departure at senders port, hour :	00
Departure at senders port, min :	00

Menu input BAPLIE file.

[Container code conversions]
This tool will convert any container codes that are non-ISO to the user-specified ISO equivalents.

[Baplie conversions]
This tool will shift containers according to input when importing.



Define BAPLIE conversions

Quit Insert New Remove Edit

Code container length	Bay	Row	Deck	Hold	Position aft side
A	09		Yes	Yes	96.750
B	09		Yes	Yes	96.750
C	11		Yes	No	88.700
D	09		Yes	Yes	96.750
E	09		Yes	Yes	96.750
F	09		Yes	Yes	96.750
N	10		Yes	No	88.700
A	09		Yes	Yes	96.750
B	09		Yes	Yes	96.750
C	09		Yes	No	96.750
D	09		Yes	Yes	96.750
E	09		Yes	Yes	96.750
F	09		Yes	Yes	96.750
L	02		Yes	No	120.100
L	06		Yes	No	105.530
L	10		Yes	No	88.700
L	14		Yes	No	74.100
L	20		Yes	No	55.840
L	24		Yes	No	41.280

An example of the conversion tool.

In the [Code container length] entry, type the first letter of the container ISO-code. [Row] can either be left blank - meaning all the rows of the corresponding [Bay], or you can type a specific row for the shift to be applied.

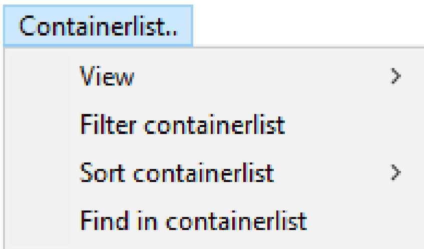
17.4.3.6 Window

Result windows
These options are the same as in the list of weight items, see section 16.2.1 on page 307, Define/edit weight items, option [Window]→[Result windows].

Reset window view
Reset to default zoom and scale.

17.4.3.7 Containerlist

In the [Containerlist] submenu the following options are available:



Dropdown menu options Containerlist.

[View]
You can choose what list you will see in the Containerlist window 4 . If you choose [Loaded] and then select a container from the Containerlist, it will be highlighted in the 3D View 5 and the Section windows 7 . However, the function buttons 3 will not respond. Option [All] opens in a new window.

[Filter containerlist]
You can choose to see only a selected type of containers, filtering by any of the 4-code digits. For example, 4∗∗∗ will show all the 40ft containers, or ∗∗R∗ will show all the refrigerated containers. The filter can be used in all views: loaded, not loaded or all. To reset the filter, type ∗∗∗∗.

[Sort containerlist]
You can sort the containerlist according to the selected column.

[Find containerlist]
You can find a specific container through its ID.

17.5 Crane loading tool

This tool is used for defining crane loads. Based on the predefined crane properties, the combined crane load and corresponding COG's are determined. This module for PIAS can be operated from a numerical input sheet, and from a Graphical User Interface (GUI) and for LOCOPIAS only from the GUI.

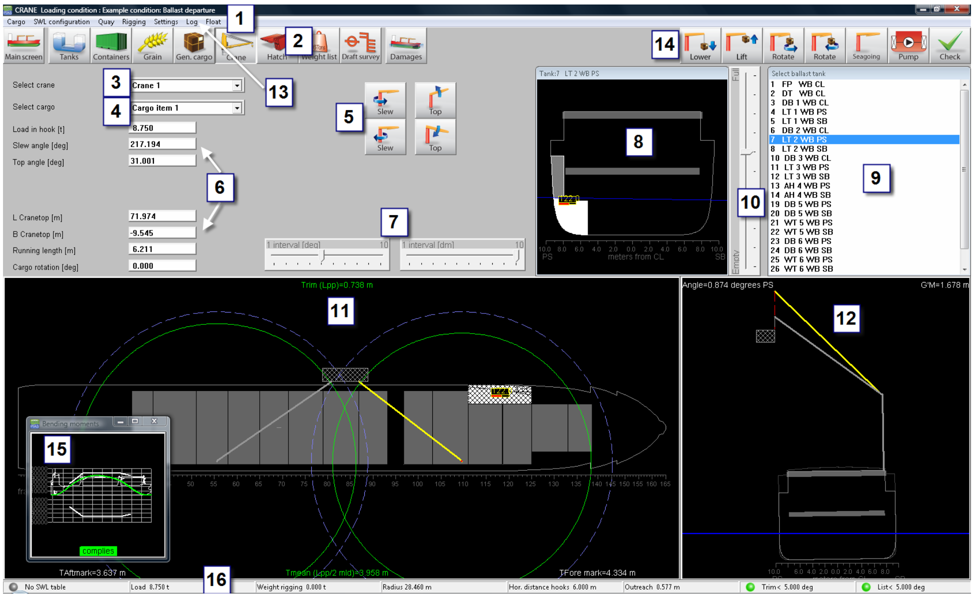
Note

A video[2] exists in which the operation of this module is demonstrated.

17.5.1 Layout

Show and manipulate defined cranes and, if applicable, ballast tanks in a Graphical User Interface, an example of which is depicted below. A description of the labeled elements follows.

[2] https://www.youtube.com/watch?v=02Sct8Lg0L4



Crane module GUI.

1 Menu bar
Basic functions are accessible through the menu bar.

2 Module buttons
These buttons navigate to another module, or back to the main screen.

3 Crane selection
Choose the crane you want to manipulate.

4 Cargo selection
From the drop-down box choose the cargo you want to load.

5 Crane operations
Use the buttons to steer the crane or manipulate cargo parameters.

6 Crane / Cargo operations
Enter values directly here.

7 Interval settings
Set interval step for meters and degrees.

8 Tank view
View on the (first) selected tank in a cross section.

9 List of tanks
Select a tank for ballasting. For transfer of tank contents between tanks change to the [Tanks] module 2 .

10 Track bar
The track bar can change the filling of the selected tank.

11 Top view
The circles in the top view indicate the maximum and current radius of the crane for the upright vessel.

12 Cross section on active crane
In the cross section view the heeling angle and position of the crane can be seen.

13 Activity log
Opens a window on top, in which all crane and ballast operations are listed. This log can be printed. Reset empties the log.

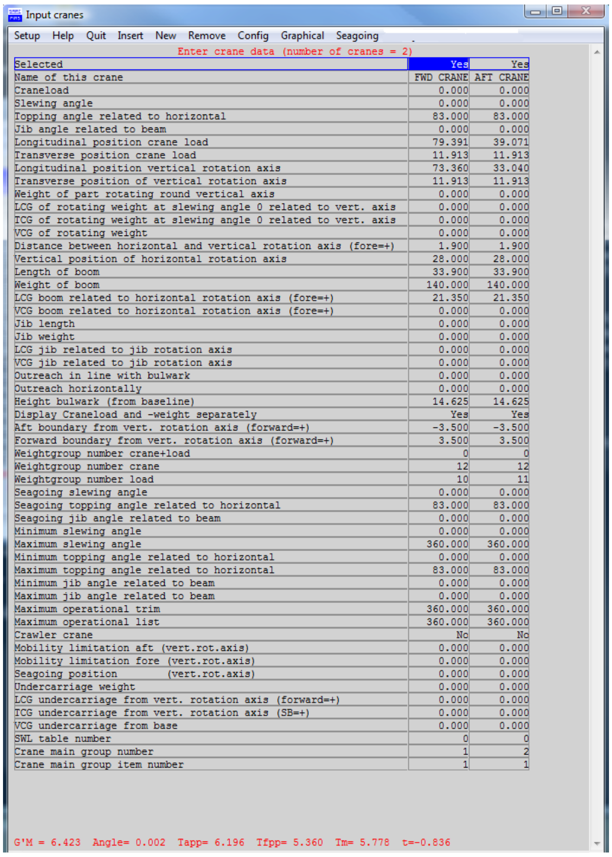
14 Function buttons
Specific functions to operate the crane, manipulate cargo, pump tank contents and verification of the current loading condition.

15 Result windows
Direct verification of stability or strength. This dynamic repositionable window can be activated by the menu bar function [Window]→[Result windows].

16 Status bar
Shows information about the selected crane.

17.5.2 General

On startup of the crane module an input menu, as shown below, is displayed in which crane data can be entered. On exit of the crane module only the selected cranes are added to the list of weight items. During input of crane data a line with hydrostatic information can be displayed at the bottom of the screen. This hydrostatic data is valid for the loading condition including all **selected** cranes. It is possible to define a maximum of 10 different cranes.



Selected	Yes	Yes
Name of this crane	FWD CRANE	AFT CRANE
Craneload	0.000	0.000
Slewing angle	0.000	0.000
Topping angle related to horizontal	83.000	83.000
Jib angle related to beam	0.000	0.000
Longitudinal position crane load	79.391	39.071
Transverse position crane load	11.913	11.913
Longitudinal position vertical rotation axis	73.360	33.040
Transverse position of vertical rotation axis	11.913	11.913
Weight of part rotating round vertical axis	0.000	0.000
LCG of rotating weight at slewing angle 0 related to vert. axis	0.000	0.000
TCG of rotating weight at slewing angle 0 related to vert. axis	0.000	0.000
VCG of rotating weight	0.000	0.000
Distance between horizontal and vertical rotation axis (fore=+)	1.900	1.900
Vertical position of horizontal rotation axis	28.000	28.000
Length of boom	33.900	33.900
Weight of boom	140.000	140.000
LCG boom related to horizontal rotation axis (fore=+)	21.350	21.350
VCG boom related to horizontal rotation axis (fore=+)	0.000	0.000
Jib length	0.000	0.000
Jib weight	0.000	0.000
LCG jib related to jib rotation axis	0.000	0.000
VCG jib related to jib rotation axis	0.000	0.000
Outreach in line with bulwark	0.000	0.000
Outreach horizontally	0.000	0.000
Height bulwark (from baseline)	14.625	14.625
Display Craneload and -weight separately	Yes	Yes
Aft boundary from vert. rotation axis (forward=+)	-3.500	-3.500
Forward boundary from vert. rotation axis (forward=+)	3.500	3.500
Weightgroup number crane+load	0	0
Weightgroup number crane	12	12
Weightgroup number load	10	11
Seagoing slewing angle	0.000	0.000
Seagoing topping angle related to horizontal	83.000	83.000
Seagoing jib angle related to beam	0.000	0.000
Minimum slewing angle	0.000	0.000
Maximum slewing angle	360.000	360.000
Minimum topping angle related to horizontal	0.000	0.000
Maximum topping angle related to horizontal	83.000	83.000
Minimum jib angle related to beam	0.000	0.000
Maximum jib angle related to beam	0.000	0.000
Maximum operational trim	360.000	360.000
Maximum operational list	360.000	360.000
Crawler crane	No	No
Mobility limitation aft (vert.rot.axis)	0.000	0.000
Mobility limitation fore (vert.rot.axis)	0.000	0.000
Seagoing position (vert.rot.axis)	0.000	0.000
Undercarriage weight	0.000	0.000
LCG undercarriage from vert. rotation axis (forward=+)	0.000	0.000
TCG undercarriage from vert. rotation axis (SB=+)	0.000	0.000
VCG undercarriage from base	0.000	0.000
SWL table number	0	0
Crane main group number	1	2
Crane main group item number	1	1

Crane load definition.

In the example above, two boom cranes are defined. '∗∗∗' means a value cannot be determined currently.

17.5.2.1 Accidental loss/drop of crane load

If this extension has been purchased, the stability criteria which are applicable after the crane load accidentally drops out of the crane can be calculated. For this calculation you should pay attention to the following items:

- The criteria as usually applicable for this vessel during crane operations should be selected.
- A set of stability criteria as applicable after the drop of crane load must be defined and NOT selected.
- The column 'loss of load' must be set to yes for this set of stability criteria.
- Then, if a crane load item defined with the crane module with a weight > 0 is found, calculations for loss of crane load will be executed.
- The loading condition will be calculated twice: first the situation with the load in the crane, secondly the situation without crane load.
- If a requirement is defined including a 'rollback angle', the static angle of heel from the first calculation is used for the value of the rollback angle in the second calculation. This statical angle can be set via the variable 'Heeling angle during lifting', see section 15.5.2 on page 296, Variables.

17.5.3 Menu bar functions

17.5.3.1 Config

A given set of crane definitions, positions and loads can be stored readily to include it in other loading conditions that use this option. Option [Config] gives an overview of all crane configurations. If a crane configuration is selected (with a 'yes' in the first column), the corresponding data will be used. Beware to give the crane configurations unique and unambiguous names, to avoid confusion later on.

17.5.3.2 Seagoing

Directly set cranes to position as defined (see section 17.5.4 on this page, Inputdata).

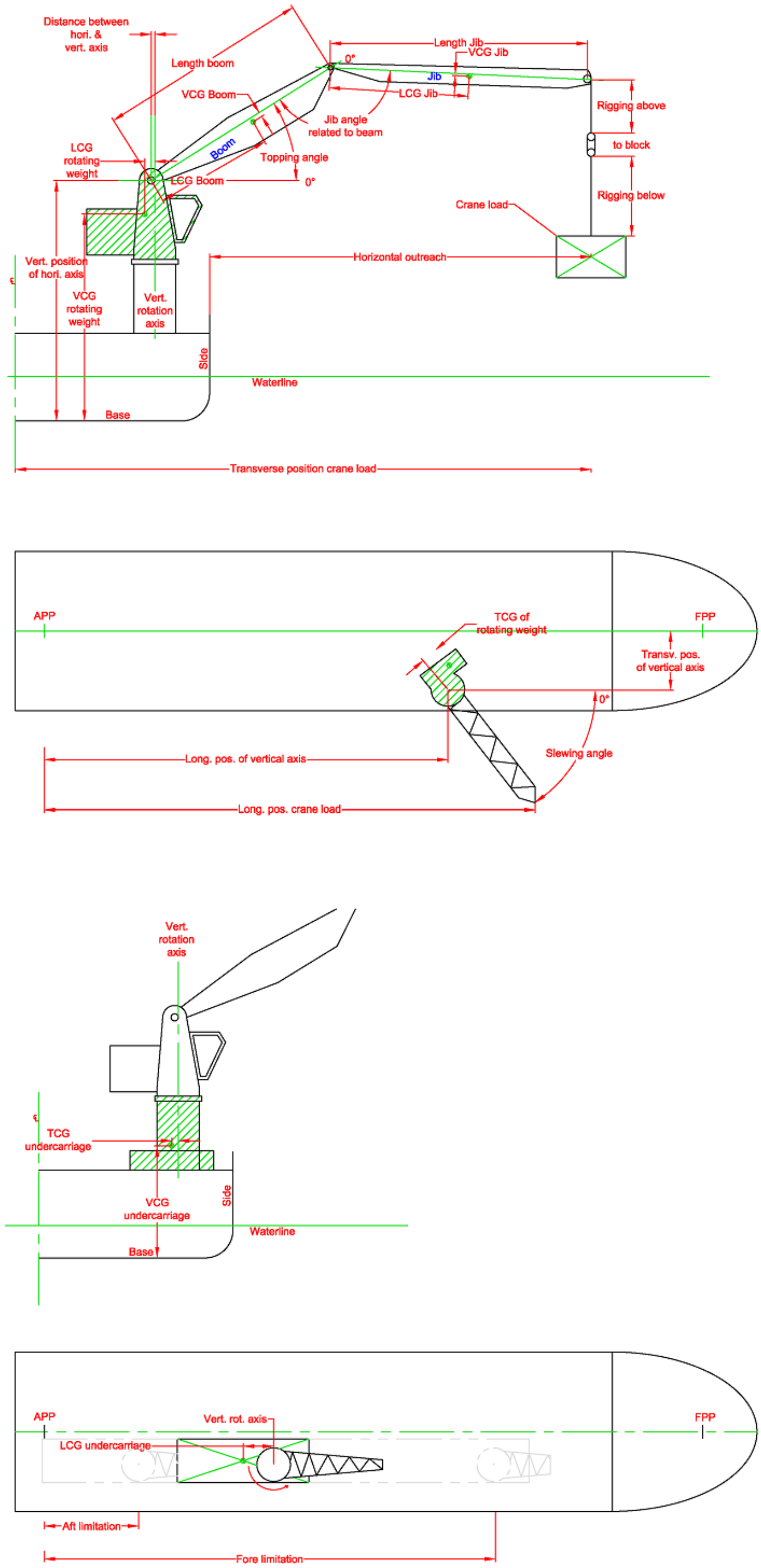
17.5.4 Inputdata

The following particulars define the crane and its position:

1. *Selected:* Selected cranes will be included in the list of weight items when you leave the crane module.
2. *Name of this crane:* The identification name for this crane.
3. *Crane load:* The weight of the crane load in metric tons can be entered here.
4. *Slewing angle:* Angle of the boom relative to the centreline of the ship. Angles to SB are positive.
5. *Topping angle:* Angle of the boom relative to the base plane of the vessel. Angles upwards are positive.
6. *Jib angle related to boom.*
7. *Longitudinal position crane load:* Length of the point of application of the crane load from APP.
8. *Transverse position crane load:* Breadth of the point of application of the crane load from CL.
9. *Longitudinal position of vertical rotation axis:* The longitudinal distance from App of the vertical rotation axis of the crane.
10. *Transverse position of vertical rotation axis:* The transverse distance from the centreline of the vertical rotation axis of the crane.
11. *Weight of part rotating round vertical axis:* The weight in metric tons of the part that exclusively rotates around the vertical axis. The boom, for example, which also rotates around the horizontal axis is defined separately.
12. *LCG of rotating weight at slewing angle 0 related to vertical axis:* The longitudinal distance from the vertical axis of rotation to the cog of weight of part rotating round vertical axis. Forward of the vertical axis is positive.
13. *TCG of rotating weight at slewing angle 0 related to vertical axis:* The transverse distance from the vertical rotation axis of rotation to the cog of weight of part rotating round vertical axis. SB of the vertical axis is positive.
14. *VCG of rotating weight:* The vertical distance from base to the cog of the weight of the part rotating round vertical axis.
15. *Distance between horizontal and vertical rotation axis (fore=+):* The longitudinal distance between the vertical and the horizontal axis of rotation. The distance is positive if the horizontal axis is forward of the vertical axis.
16. *Vertical position of horizontal rotation axis:* Vertical distance from baseline to the horizontal axis.
17. *Length of boom:* Length of the boom of the crane.
18. *Weight of boom:* The total weight of the boom.
19. *LCG boom related to horizontal rotation axis (fore=+):* Position of the lcg of the boom measured along the boom from horizontal rotation axis.
20. *VCG boom related to horizontal rotation axis (up=+):* Position of the vcg of the boom measured perpendicular to the boom from horizontal rotation axis.
21. *Jib length.*
22. *Jib weight.*
23. *LCG jib related to jib rotation axis.*
24. *VCG jib related to jib rotation axis.*
25. *Outreach in line with bulwark:* The distance perpendicular to CL, measured from 1/2 B to the transverse position of the crane load.
26. *Outreach horizontally:* The horizontal distance measured from the upper side of the bulwark to the transverse position of the load

27. *Height bulwark:* The height of the bulwark, measured from baseline to top of the bulwark.
28. *Display crane load and weight separately:* If set to ‘no’, the selected crane and its load will be included in the list of weight items as a single weight item. If set to ‘yes’, the weight is divided in two separate items: the weight of the crane itself and the load.
29. *Aft. boundary from vert. rotation axis (forward=+):* Aft boundary for the purpose of the longitudinal strength calculation.
30. *Fore. boundary from vert. rotation axis (forward=+):* Forward boundary for the purpose of the longitudinal strength calculation.
31. *Weight group number crane + load:* Crane and crane load can be given the same or separate weight group numbers, depending on the wishes of the user.
32. *Weight group number crane:* see item 31.
33. *Weight group number load:* see item 31.
34. *Seagoing slewing angle:* With the function ‘Seagoing’ from the toolbar or in the graphical interface the crane is positioned in position with a slewing angle as defined here.
35. *Seagoing topping angle:* With the function ‘Seagoing’ from the toolbar or in the graphical interface the crane is positioned in position with a slewing angle as defined here

Note: when slewing and topping angle values are changed, longitudinal and transverse positions of cargo are calculated automatically. Vice versa if longitudinal or transverse position of cargo is altered.



Crane definition aft and top view.

17.6 Grain/bulk

To be elaborated.

17.7 Ballast advice

The usual way of making or simulating a loading is that the user wants to achieve a particular objective, such as a maximum draft or maximum cargo weight, and then 'plays' with cargo and ballast in order to achieve that. With this 'ballast advice' feature that process is partly automated: the user specifies the goal and boundary conditions, and PIAS 'plays' with the ballast quantities in order to get that fixed. The word 'ballast advice' should be interpreted somewhat broader, also the fuel can be included in this process.

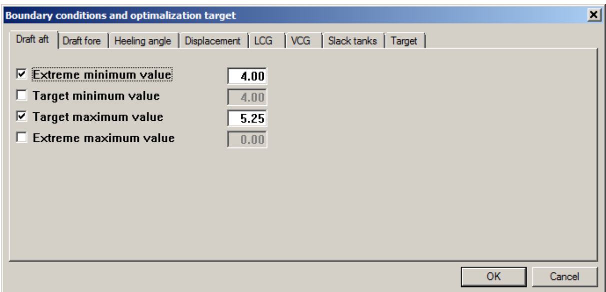
In this ballast advice process two concepts play a role:

- **Boundary conditions** which are the loading parameters to be achieved, or whose limits are indicated. Such as minimum draft, maximum heel etc.
- In general, an abundance of loadings will exist for which the conditions can be fulfilled, but only one of them is optimal. This is specified by the **optimization target**, e.g. the goal 'minimum amount of pumped ballast water' (which is actually an interpretation of the underlying goal 'to achieve the desired loading in the shortest possible time').

The advice process searches the ballast quantity that fulfils the boundary conditions, and which is optimal from the perspective of the optimization target. The operation is located in two places: the settings for a specific ship, and the actual determination of the amount of ballast water in a loading condition. We start with the latter.

17.7.1 Determining the amount of ballast water in a loading condition

The ballast advice function is invoked with the ballast advice icon in the GUI (which is discussed in section 17.1.1 on page 324, Main window layout). First, an intermediate window comes up, which shows all the tanks that can be used to achieve the desired state. Initially, all tanks are switched on, implying that from each of those tanks the content can change. If that is not desired for some tanks these can be unchecked. Next comes an input window where boundary conditions and optimization target can be set, the figure below shows an example.



The ballast advice input window.

In this window, you can specify numerical boundaries for six parameters: draft aft, draft forward, heel, displacement, LCG and VCG, actually a desired value (target value) and an extreme value. These can be a maximum and a minimum, so for each boundary condition four values can be given. The difference between target and extreme value is simply that the *extreme* is far more stringent than a *target*. If only a single boundary condition is used there will be no difference between the result with a target value and with an extreme value. However, if multiple boundary conditions are used, a global solution will be searched, in which all boundary conditions are fulfilled as much as possible. In such a case it is possible that not each and every boundary condition is rock solid, in which case that one can be set to 'target'. This may increase the efficiency of the solution. In addition, there is also a boundary condition 'slack tanks', where you can specify the maximum number of simultaneous slack tanks. Some stability rules impose a maximum on that.

The last tab of the input window contains the optimization target, one of these five:

- Minimum volume of pumped ballast water. Ballast water can, of course, be taken in from or discharged into the sea.
- Switched off, i.e. there is no target at all; the first solution that meets the boundary condition(s) is then taken.
- Minimum volume of pumped ballast water, with fixed deplacement. This solution could also be found by using the first target, combined with constraints on the displacement, but then for each loading condition the desired displacement should be specified. In this way, it is easier.
- Minimum volume of pumped ballast water with fuel volume given below. Here the boundary conditions are not met by just changing ballast, but also by the distribution of fuel over the fuel tanks. The total fuel volume can be given on the next line of this tab. This 'fuel' option is only present in this window if the setting (in the next paragraph) actually specifies that fuel tanks are involved in the ballast advice procedure.
- Minimum resistance. If the module Trim optimization is active, this optimization target can be selected to get a ballast advice which results in the trim and displacements combination that minimizes the resistance of the vessel. The dataset and internal procedures of the trim optimization module are used to achieve this.

Finally, in the last tab a check box 'Calculate ballast advice' is available. This must be switched on in order actually compute the advice. The background of this box is that the calculation of the advice might take a considerable amount of time, so that it may be desirable to leave this input window without calculation. And that can be achieved by unchecking this box. If the ballast advice has been calculated then the conclusion will be printed (or, alternatively, displayed as *preview* on screen, and the question is asked whether the loading conditions must be adjusted in accordance with the advice. If done so, the original is lost. If that is not desired you should make a copy in advance.

On the ballast advice as such some more remarks can be made:

- You can use as many boundary conditions as you like, but please consider that the solution is found iteratively. With as result that the computation time can increase significantly with the use of many boundary conditions, especially when those are nearly or completely contradictory. If there is no solution (anyway, if none is found), then that is reported, but no feedback is given on the reason thereof. As is common with iterative search methods.
- Calculating the advice may require a huge number of intermediate steps, and can sometimes take a long time. If that is deemed undesirable, it is recommended to avoid combinations with other calculation-intensive options, such as 'shift of COG of liquids', as much as possible. Another possibility to decrease the computation time is using only the major tanks and skip the small ones.
- Tanks within a 'tank combination' are filled to the same level. If this combination contains a port and starboard tank it will not be possible to neutralize a heeling angle with this combination.

17.7.2 Ship-specific settings for ballast advice

These are given from the Loading project settings menu, as these are discussed in section 16.5 on page 319, Loading project settings and tools and section 16.5.12 on page 322, Settings for ballast advice. The basic concept here is the 'tank combination', which is a group of one or more tanks which are equally filled. Usually, such a tank combination will contain a single tank, but it is known *a priori* that tank fillings must be equal, for example to prevent heel with a SB and a PS tank, then those can be taken together in one tank combination. With the first option of this settings window, the tank combination definition window is opened, where for each combination can be specified:

- The first column is a number, which is simply an automatically generated sequence number, which has no further function. With <Enter> one goes one menu deeper, and enters a compartment list where with 'Yes' or 'No' it can be specified which compartment(s) belong to this tank combination.
- The tank type, which indicates the contents of the tank combination, choice of ballast water or fuel oil.
- The relative importance, which indicates the importance of this tank combination compared to the other combinations. By default this equals one, implying that all tanks are equally important. This tank interest relates to the optimization target; suppose that this target is 'minimal amount pumped ballast water', and the tank importance tank is zero, then this tank is not included in the calculation of the total amount of pumped ballast water. Is the tank importance e.g. 100, then its influence is compelling if its water quantity changes. If a tank has, for example, a thin pipe line and a pump of limited capacity, then it will be advantageous not to transfer much water in or out of the tank, so the tank importance should be set high. Conversely, if the tank is used **by preference** then the tank importance is low. Please avoid giving all tanks an importance of zero, because then there is no effective optimization criterion anymore, and the final solution will be completely arbitrary.

- Slack, indicating whether the tanks of this combination should be included in the boundary condition 'number of slack tanks'. One could wonder why explicitly assign this property, after all "slack is slack"? The reason is that it may be defendable to exclude small tanks, with negligible free surfaces, from the count of slack tanks. This can be done by setting this 'slack' parameter to 'no'.

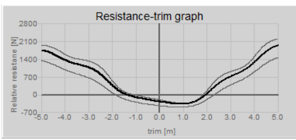
With the second option from the settings window a check box pops up, where it can be specified which boundary conditions are offered when calling the ballast advise function.

- Draft aft (on APP).
- Draft fore (on FPP).
- Heeling angle (which is the absolute value, regardless whether heel is to PS or to SB).
- Displacement.
- LCG, longitudinal center of gravity.
- VCG, vertical center of gravity.
- The maximum number of slack tanks.

17.8 Trim optimization

By interpolating between known data points it is possible to get an accurate view of the trim-resistance graph for a particular loading condition at a specified speed. This graph can be used to asses whether it is advantageous to trim the vessel. If both this module and the ballast advice module (see *section 17.7 on page 376, Ballast advice**) are active, the ballast advice module can used to calculate the optmimal trim by specifying the 'minimal resistance' optimization target.*

This module revolves around the resistance-trim graph that is available in the main window or in the loading modules using the Result Windows.



Example of a trim-resistance graph

In this graph the resistance is plotted as a function of the vessel's trim, for a defined speed and the displacement of the current loading condition. When these change the graph is updated automatically.

The speed and de delta displacement can be set for the current loading condition using the settings menu Settings, in the tab 'Trim optimization'.

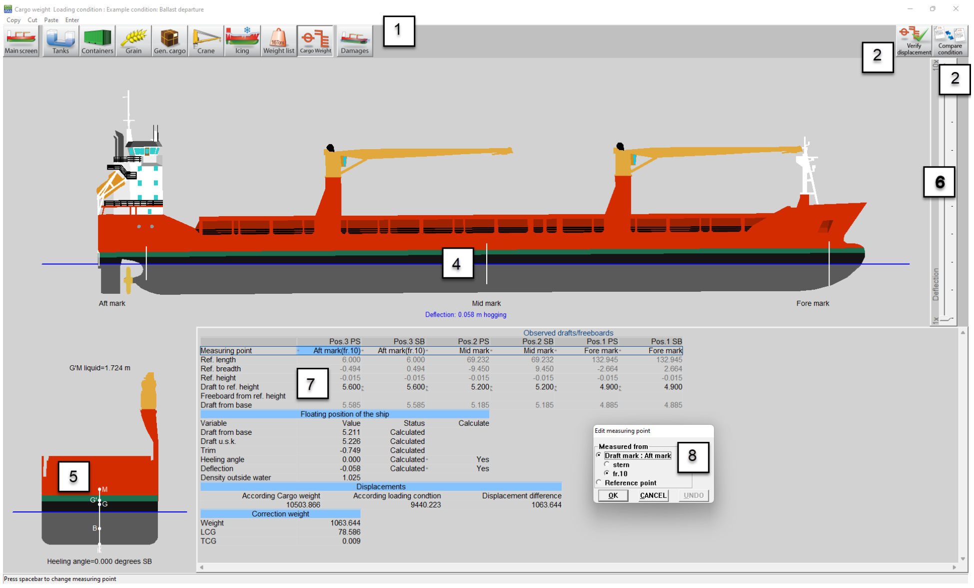
Aside from the main resistance-trim line, two other thinner lines can be plotted using the delta displace variable. The delta displacement is a deviation from the current vessel displacement in percentages, so that the effect of a change in displacement upon the resistance-trim graph can be seen at a glance. The delta displacement can be set in the settings menu, or highered or lowered using the scrollwheel of the mouse.

The unit of Y-axis of the graph is the relative resistance in Newton. The graph is scaled so that the current trim always has a value of zero on the Y-axis. The means that the effect for more or less trim is easily identified from the graph.

17.9 Cargo weight determination

The [Cargo weight] module is intended for the calculation or verification of (un)loaded cargo weight. This module can be used on any type of cargo vessel. The module can be used to verify the entered loading condition with the observed drafts / freeboards. Alternatively, the module can also be used to calculate the weight of the (un)loaded cargo, by comparing the draft or freeboard and the deductibles before and after the (un)loading operation.

17.9.1 Layout of the GUI



Layout of the cargo weight determination module.

1 **Module buttons**
These buttons navigate to another module, or back to the [Main screen].

2 **Verify displacement button**
Use this button to verify the displacement of this loading condition with the observed draft marks.

3 **Compare condition button**
Click to print the cargo weight determination report.

4 **Side view**
Shows the actual wind contour, drafts and actual waterline.

5 **Aft view**
Shows the aft view of the vessel, heeling angle and initial stability (G'M).

6 **Deflection**
With this track bar the deflection enlargment factor can be set from 1x to 10x.

7 **Observed drafts/freeboards**
Enter the observed drafts or freeboards here. Also some results are directly available.

8 **Edit measuring point window**
Type <Spacebar> at a measuring point cell 7 to open this window.

17.9.2 General approach

There are two ways to use this Cargo weight module. One can verify a loading condition with observed drafts / freeboards. Or one can compare a loading condition before and after (un)loading to calculate the (un)loaded cargo weight, perhaps better known as a draft survey. Detailed instructions for both methods can be found later on.

In general the following steps have to be done for both methods. For the compare condition method steps 1 and 2 are repeated for the initial and final loading condition.

1. Define loading condition Define the loading condition outside this module
2. Enter drafts / freeboards Enter the observed drafts / freeboards in 7
3. Calculate. For verification of one loading condition press 2 . To compare two loading conditions press 3 to determine the (un)loaded cargo weight.

17.9.3 Verify displacement method

The governing idea of this method is that the displacement of **one** loading condition can be verified by comparing it with the displacement resulting from observed drafts / freeboards. This can for example be used to check the actual cargo loaded weight against the planned loaded cargo weight as entered in the loading condition in LOCOPIAS. Or one could determine a dead weight constant, if the actual displacement always differs from the displacement according to LOCOPIAS.

Below you find the steps that should be taken to determine the difference between the displacement based on the observed drafts and the weight list of the loading condition.

17.9.3.1 Define loading condition

The first step is to define the loading condition. Perhaps this step is already finished. Otherwise go back to the [Main screen] and define the loading condition by using the other modules, see: chapter 17 on page 324, Loading tools . Enter all details like tank fillings, grain bulkheads, cargo etc., like you would normally do. When finished enter the [Cargo Weight] module again.

17.9.3.2 Enter drafts / freeboards

Now you should enter the observed drafts (default) or freeboards. In 7 you will enter the drafts on the pre-defined draft marks. Alternatively, you can also define a reference point yourself and indicate whether you want to enter drafts or freeboards. Press <Spacebar> or any other keyboard key, in accordance with the LOCOPIAS operation standard as described in section 4.3 on page 38, Content and options in the cells of selection windows and input windows at the measuring point in the [observed drafts / freeboards] window 7 to open the [edit measuring point] window 8 .

After entering the drafts / freeboards you can check in windows 4 and 5 if the vessels position is as expected. Also the expected hogging / sagging can be checked. If the hogging / sagging is not very clear, the deflection can be exaggerated by using 6 .

17.9.3.3 Calculate

Press the 'verify displacement button' 2 to make the calculation. A popup will be displayed showing the displacement according the observed drafts / freeboards, and the displacement as entered in LOCOPIAS and the weight difference between these two. If the user wants to add the weight difference as a correction weight to the loading condition, they should tick the tickbox at the bottom of the popup window. The user should give their best estimation of the vertical center of gravity of this weight difference.

If the weight difference is deemed to be a deadweight constant, the user can also tick the box 'Database'. Then this correction weight will be stored in the database for use in new loading conditions. Please refer to section 16.2.1 on page 307, Define/edit weight items for further explanation of the database.

Click on OK to add the correction weight if desired and print the 'displacement verification report' from which an example is depicted below.

DISPLACEMENT VERIFICATION REPORT

Loading conditions
Example condition: Containers

	Loading condition			Cargo weight		
Drafts [m]	Aft	Center	Fore	Aft	Center	Fore
Starboard	8.141	8.049	7.783	8.030	7.850	7.730
Portside	7.947	7.630	7.704	8.020	7.840	7.720
Mean	8.044	7.840	7.744	8.025	7.845	7.725
Hydrostatics						
Draft mean of means [m]		7.828			7.843	
Trim on Lpp [m]		-0.201			-0.322	
Angle [degrees]		1.270			0.033	
Deflection [m]		0.000			-0.034	
Density water [ton/m3]		1.025			1.025	
Actual displacement [ton]		16662.930			16716.682	
Deductibles [ton]						
Water ballast		4329.977				
Gasoil		248.966				
Heavy fuel oil		597.180				
Lub oil		39.374				
Freshwater		75.934				
Various		35.028				
Sewage / Sludge		3.045				
Miscellaneous		27.000				
Grain bulkheads		0.000				
Tweendeck panels/hatch covers		789.026				
Crane rotating part		120.092				
Zone 1		0.000				
Zone 2		0.000				
Zone 3		0.000				
Other		0.000				
Total deductibles		6265.621				
Cargo [ton]						
Cargo		0.000				
Grain / bulk cargo		0.000				
General cargo		0.000				
Container cargo		6020.000				
Crane load / rigging		0.000				
Total cargo		6020.000				
Total light ship		4377.288				
Total displacement [ton]		16662.930			16716.682	

Correction weight [ton]	**53.770**
LCG [m]	1.666
VCG* [m]	7.204
TCG [m]	-5.676

* The VCG is estimated by the crew.

Example of displacement verification report.

17.9.4 Compare load method

The method is also known as a draft survey. The governing idea of this method is that **two** loading conditions are compared. One condition is before and the other is after the (un)loading operation. The difference in displacement (resulting from observed drafts) will be the (un)loaded cargo weight. Besides a difference in cargo weight there could also be a difference in other weight items, such as ballast and consumables. To correctly calculate the (un)loaded cargo weight, these deductibles are taken into account by identifying two loading conditions in LO↩ COPIAS, which will be labelled 'initial' and 'final'. To distinguish between cargo and deductibles, every cargo weight item must be assigned to a cargo weight group. Special care should be taken when the cargo is defined in the weight list as a free weight item, since these weight items are not automatically assigned to a weight group.

Below you find the steps that should be taken to determine the (un)loaded weight.

17.9.4.1 Define the initial loading condition

Go back to the [Main screen] and define the condition before the (un)loading operation, including tank fillings, configuration of grain bulkheads, cargo etc. This is later referred to as the initial loading condition.

Note: The 'initial' and 'final' qualifiers are not fixed to a particular loading condition.

17.9.4.2 Enter the observed drafts of the initial condition

Open the [Cargo weight] module again and enter the observed drafts in this condition. Details can be found in the section 'enter drafts / freeboards'.

Now we are finished with preparing the initial loading condition.

17.9.4.3 Define the final loading condition

Now go back to the [Main screen] and create a new loading condition that will represent the situation after (un)loading. This is later referred to as the final loading condition. This new loading condition could also be a copy of the 'initial' condition. Please go to section 17.1.3 on page 327, Conditions if you need more information on how to create or copy a loading condition.

Now define this loading condition correctly, adjusting the tank fillings, grain bulkhead positions etc.

17.9.4.4 Enter the observed drafts of the final condition

Open the [Cargo weight] module again and enter the observed drafts (or freeboards) of the (un)loaded vessel.

17.9.4.5 Calculate

Click the [Compare load] button to produce a cargo weight determination report. You will be asked to select the initial condition. Only loading conditions where the observed drafts are entered are selectable as 'initial' for a weight determination computation. So if your initial loading condition is not visible, please go back to the [Main screen], switch to the initial loading condition (or create one) and follow steps 1 and 2 to set up the initial loading condition correctly. After that switch to the final loading condition again the print the cargo weight determination report.

Click on Ok to print the report from which an example is depicted below.

CARGO WEIGHT REPORT

Loading conditions
Initial : Arrival Rotterdam
Final : Departure from Rotterdam after discharging

	Initial			**Final**		
Observed drafts [m]	Aft	Center	Fore	Aft	Center	Fore
Starboard	8.195	7.930	7.740	4.560	4.470	4.385
Portside	8.200	7.900	7.720	4.550	4.460	4.370
Mean	8.198	7.915	7.730	4.555	4.465	4.378
Hydrostatics						
Draft mean of means [m]		7.915			4.453	
Trim on Lpp [m]		-0.458			-0.188	
Angle [degrees]		0.099			0.040	
Deflection [m]		-0.033			-0.002	
Density water [ton/m3]		1.025			1.025	
Actual displacement [ton]		16906.506			8816.624	
Deductables [ton]						
Waterballast		849.583			2811.201	
Gasoil		17.041			17.041	
Heavy fuel oil		76.993			530.306	
Luboil		18.572			29.487	
Freshwater		7.594			46.491	
Various		59.412			59.412	
Sewage / Sludge		15.224			3.045	
Miscellaneous		27.000			27.000	
Grain bulkheads		54.472			54.472	
Tweendeck panels/hatch covers		734.554			734.554	
Crane rotating part		120.092			120.092	
Zone 1		0.000			0.000	
Zone 2		0.000			0.000	
Zone 3		0.000			0.000	
Other		0.000			0.000	
Total deductables		1980.537			4433.101	
NET Displacement [ton]		14925.969			4383.523	
Empty ship		4377.288			4377.288	
Constant/cargo on board [ton]		10548.681			6.235	
Total discharged [ton]			**10542.446**			

Example of weight determination report.

17.9.5 Read draft sensors

To read out the sensors, click the [Sensor reading] button. The read values are copied in the measured drafts/freeboards menu. The positions for which no sensor value is available, are set to Not measured. Now the read values are used to calculate the position, deplacement and correction weight.

Chapter 18

Stability of open hopper vessels

With Loading, the standard PIAS module for intact and damage stability, also the (damage-) stability of hopper dredgers (or, in general, open-top hopper vessels) can be computed, including the effects of spillingover of cargo and pouring in of seawater. On that computation, and the corresponding modus operandi with PIAS, quite some comments can be made, which justifies this separate chapter in the PIAS manual.

18.1 Available computation methods.

The stability calculations for open hopper vessels can be executed according to the following six regulations:

- “Agreement for the construction and operation of dredgers assigned reduced freeboards”, dr-67 & dr-68.
- Bureau Veritas “Freeboard of dredgers and barges fitted with bottom dump doors”, N.I. 144, 1971.
- Russian Maritime Register of Shipping (RMRS Rules for the classification and construction of sea-going ships, 2014, Part IV, § 3.8 “Vessels of dredging fleet”).
- MCA 1999 (Merchant Shipping Regulations 1999, MSIS003/part 8, “dredgers”).
- Guideline 28, “Bijzondere voorschriften voor baggermaterieel” of the Dutch Shipping Inspectorate.
- RINA, according to the guideline below.

The RINA rules 2012 contain a section “Part E Service Notations, Ch. 13, Ships for dredging activity”, which contain the RINA rules for hopper dredgers. These can be applied with PIAS as follows:

- For the intact stability the cargo is assumed to be lagging with respect to the ship angle, according to the equation $\theta_R = (3 - \gamma).\theta_G$, for $1 < \gamma < 3$. That is exactly the same formula as applied with *Bureau Veritas*, so that one can also be used to compute intact stability according to RINA.
- For damage stability the assumption *“In the damage calculations it is to be assumed that all the cargo is lost as a result of the damage and that the bottom doors remain open leaving the spaces in communication with the sea”* applies, so this requires no real calculation with pouring in or out over the edge of the hopper. One can simply use a loading condition with empty hopper, and declare the hopper hold to be damaged. The RINA damage stability requirements can be set with the criteria set as available in PIAS, see chapter 15 on page 275, Stability criteria for intact stability and damage stability.

18.2 General procedure

Open hoppers and their contents are treated separately in a pre-programmed way, depending on the settings and the selected computation method. The first issue is to define the shape and location of the hopper(s), which can be done simply by modelling the hopper(s) in the Layout module as an ordinary compartment. Those compartments must be marked as being a hopper type, which can be done with setting as discussed in paragraph 9.5.1.2.10 on page 218, Compartment is an open hopper.

The next question is how such a hopper can be used and filled in Loading. This is also simple, because since the hopper is essentially a tank, in the weight item list it is positioned just between the other tanks and compartments (or can be added as follows, see section 16.2.1.2 on page 310, Read tanks as weight item). A hopper occupies two lines in the weight items list, once with its cargo content and the other with the (not yet poured out) water on top of the cargo. These two lines are irrevocably linked, which implies that if one line is removed, the other will also disappear.

18.3 Specify additional hopper properties

A hopper can spill over or pour in through or over special points. These are specified in Layout, where other special points of a compartment can also be specified, such as openings or a pressure sensor, see paragraph 9.5.1.2.10 on page 218, Compartment is an open hopper. For the purposes of hopper stability calculation, there are three types of such points:

- The hopper edge, which is the upper rim of the hopper, over which the cargo can spill over or the seawater can pour in freely. These points of the hoper edge must always be explicitly given, it is not the case that the top of the hopper compartment is automatically assumed to be the spill-over edge. If the stability calculation is made including the effects of trim, it is important that the forward and aft hopper edges also include modelled with these points.
- A closable overflow, through which whether or not cargo and/or seawater can flow, depending on the assumptions of the selected regularions. This opening might be adjustable in height, the range of which can be specified in column 'range' in the same menu, just right of the column where the type of overflow is given. The range is in meters, and can be positive (then the overflow can be adjusted *upwards* from the height specified at this point) or negative (in which case the opening can be adjusted *downwards*. An overflow may sometimes only be taken into account if its cross-section is sufficiently large, as required by the chosen regulations.
- A non-closable overflow. Similar to the previous type, although it cannot be closed. This setting has its effect on the overflow behaviour **during filling** (so, not on its behaviour after filling, during heel), as follows depending on the calculation method chosen:
  - Bureau Veritas and RMRS: if the overflow is potentially closable during filling, then the cargo does not flow out through the overflows, and otherwise it does.
  - All other methods: the cargo does not flow out through overflows.

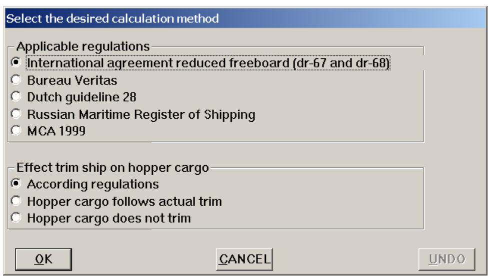
18.4 Specify loading parameters

The (damage-) stability calculations with open hopper are based on a number of loading parameters that can be specified with the [hoPper] function in the loading conditions overview in Loading, as discussed in section 16.2 on page 305, Loading conditions. These are discussed below.

18.4.1 Calculation method

In the first place one of the computation methods, which have been introduced at the beginning of this chapter, can be chosen. Additionally, it can be specified how the cargo will trim. Possible settings are:

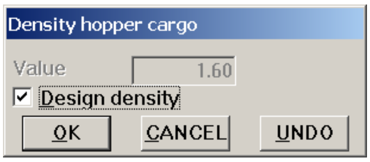
- According to regulations, i.e. that cargo and water 'trim along' with the ship, so that the cargo trim angle is dependent on the ship's trim angle as prescribed for heel in the calculation method chosen — e.g. with Bureau Veritas N.I. 144, where the cargo heeling angle depends on the ship's heeling angle and density of the cargo. With this setting, this formula will also be applied to the trim angle.
- Hopper cargo follows actual trim, which indicates that trim of cargo and water are equal to the ship's trim. In other words, surface levels of cargo and water are parallel to seawater level.
- Zero trim for hoppercargo, which sets the trim of the hopper content always to be zero, regardsless the trim of the ship.



Popup window for setting the computation method.

18.4.2 Hopper cargo density

This function opens a popup window in which the densities (=specific weights, in ton/m^3) of the cargo are defined for which the stability calculations will be made. A special form of ‘density’ is not a number, but the concept ‘design density’ which originates from the dr-67/dr-68 regulations, and which is that particular density where the ship lies exactly at her dredging draft. Such can be specified in the popup box that appears when you press fopr the density-value a key that cannot represent a number (such as <Space>), see the screendump below.



Popup window for setting the design density.

The dredging draft is specified at the ship’s main dimensions, see section 7.2.1.1 on page 167, Main dimensions and allowance for shell and appendages. The dredging draft may differ from the regular summer draft according to the Load Lines Convention, because the regulations for dredgers may allow a reduced freeboard.

18.4.3 Generate loading conditions

If everything that has been discussed so far has been entered (including the dredging draft, as discussed in the previous section), then the program has sufficient information to generate hopper loading conditions. This means that for each selected density (as discussed in the previous section) and each (in the overview list of loading conditions) loading condition which is selected for intact stability, a new condition is generated with the hopper filled. And, if applicable, with the ratio of water/cargo of the hopper determined in such a way that the vessel lies exactly on its dredging draft.

If at ’Calculation method’ for ’Effect trim ship on hopper cargo’ option ’According regulations’ has been selected, then the rule is applied as described in DR-67 / DR-68 for damage stability. This means that for lighter densities the trim of the hopper cargo is equal to that of the ship, for heavier densities the hopper cargo does not trim and for densities in between the trim of the hopper cargo is lagging behind that of the ship. For the latter densities, this means the loading condition is not optimized for liquid cargo or for solid cargo, but for something in between. This can cause the loading condition for 1 of these calculations (and sometimes for both) is just exceeding the maximum draft. There is then no filling of the hopper(s) where the ship is exactly on the dredging draft in both situations. This can sometimes be prevented by choosing ’Calculation method’ for filling ’Hopper cargo follows actual trim.’ For the calculation with liquid cargo, the ship is then on the dredging draft.

This can be used to generate a lot of loading conditions with a single command; if there are e.g. eight selected existing loading conditions and nine densities, then 72 hopper loading conditions will be generated. If a next time hopper conditions are generated again, the **question** is asked whether the previously generated conditions should be discarded. Whether to do this is up to the user. For enhanced flexibility, in the overview list of loading conditions the column ‘auto-remove’ is included at the right of the loading condition name, which will be filled with ‘yes’ if that particular condition is a generated hopper condition. If you reply *yes* to the above mentioned question, those will be discarded. If you would still like to keep some conditions, then you can mark those with ‘no’ in that column, so that they will not be automatically removed during regeneration.

18.5 Parameters from an individual loading condition

The loading parameters of each individual loading condition can also be specified, which is done with the function [hoPper] in the menu bar at the top of the list of weight items for that condition.

18.5.1 Fill hopper(s) to maximum draft

This function can perform the same operation upon a single loading condition, as section 18.4.3 on the current page, Generate loading conditions does to a whole series of conditions: filling the cargo and water in the hopper(s) so that the ship lies exactly at its dredging draft.

18.5.2 Justify overflow height(s)

Some ships have height-adjustable overflows, of which the lowest and highest position can be defined, as discussed in section 18.3 on page 385, Specify additional hopper properties. As a rule, a certain actual position of the overflow will be determined automatically, e.g. with the function from the previous paragraph. However, if you want to manually give the overflow position, you can do so with this function.

18.5.3 Fill the hopper(s) manually

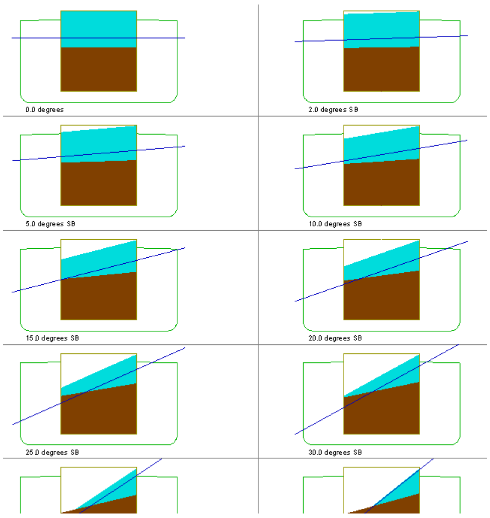
The manual filling of a hopper is so simple that it does not require a special function. In the list of weight items of a loading condition each hopper has two rows, one for the cargo, and the other for the water op top of the cargo. There you can fill in any desired weight, exactly as is the case for a normal tank. However, it should be borne in mind that in the final calculation this weight could be adjusted, e.g. if water as result of trim flows out trough an overflow.

18.6 The computations

18.6.1 Intact stablity and longitudinal strength

These calculations are simply started as conventional stability and strength calculations. All additional options available in ordinary calculations (such as tank filling plots, or verifying the GZ curve against stability crteria) can also be used in loading conditions with hopper. Specific details of hopper conditions are:

- The *Result windows* (which are discussed in section 16.2.1 on page 307, Define/edit weight items) are also functional for hopper loading. If the chosen computation method requires separate calculations for solid and liquid cargoes, the stability bar graph shows the most critical of the two.
- If the GUI shows **two** GZ curves, then one is for solid cargo, and the other for liquid cargo. It depends on the chosen computation method whether this distinction is made.
- The calculations of longitudinal strength are made on the basis of the longitudinal boundaries and the CoG from each item from the list of weight items. So without any trimming, exactly as it happens with e.g. the shift of liquid method with ordinary tanks. **If** the chosen computation method includes separate solid and liquid cargo calculations, the question is which trim is to be used for the longitudinal strength calculations. The trim for solid or for liquid cargo? No fundamental answer to this question is possible; in fact, two calculations of longitudinal strength would have to be carried out. However, such a refinement would only be of academic importance, so PIAS simply makes a single calculation, with the trim of that cargo state (solid or liquid) which happened to be calculated first.
- The output of the stability calculation can be a bit more substantial than a conventional stability calculation. First, because if the calculation method contains separate calculations for solid and liquid cargoes, **two** full calculations are printed. And secondly, because additional information is printed, such as a page with plots of cargo and water levels for each heeling angle, as well as a page with additional intermediate results, such as levels and moments of cargo and water on cargo. Some examples are given below.



Example of stability output, indicating cargo and eater levels.

Condition : Arrival, density hopper=1.5000

Calculation for : Intact stability, including possible in/outflow effects
Calculation method : International agreement reduced freeboard (dr-67 and dr-68)
Hopper : Hopper fr. 89-155
Density hoppercargo : 1.500 ton/m³

Intermediate results for LIQUID cargo calculations

Angle of ship	0.000	2.000	5.000	10.000	15.000
Angle of cargo	0.000	2.000	5.000	10.000	15.000
Volume of cargo	1241.235	1220.783	1190.109	1138.685	1086.270
Volume water on cargo	0.000	0.000	0.000	0.000	0.000
Volume water & cargo	1241.235	1220.783	1190.109	1138.685	1086.270
Level of cargo	9.176	9.051	8.861	8.541	8.209
Level of water on cargo	0.000	0.000	0.000	0.000	0.000
Draft ship	6.193	6.138	6.052	5.905	5.747
Trim ship	-0.327	-0.366	-0.419	-0.488	-0.527
Displacement	6238.742	6177.391	6085.384	5931.089	5773.871
NKsin(φ) closed ship	0.000	0.213	0.532	1.066	1.608
NKsin(φ) cargo	0.000	0.196	0.482	0.937	1.364
NKsin(φ) water on cargo	0.000	0.000	0.000	0.000	0.000
NKsin(φ) water & cargo	0.000	0.196	0.482	0.937	1.364
Righting lever (GZ)	0.002	0.014	0.036	0.088	0.166
Cargo pouring out hopper	Yes	Yes	Yes	Yes	Yes
Seawater entering hopper	No	No	No	No	No
Angle of ship	20.000	25.000	30.000	35.000	40.000
Angle of cargo	20.000	25.000	30.000	35.000	40.000
Volume of cargo	1031.763	973.898	911.762	842.804	763.798

Example of intermediate results in hopper stability calculation.

18.6.2 Deterministic damage stablity

Although none of the supported regulations include deterministic damage stability assessment, this feature is included in PIAS nevertheless, in order to be able to compute (and study in detail) a single damage case out of a set of cases of probabilistic damage stability, which are required for dr-67 & dr-68. Invoking a deterministic damage stability calculation is easy, it is completely the same as for ordinary damage case, as discussed in section 16.4.4 on page 317, Deterministic damage stability. The behaviour under heel of the cargo surface is pre-programmed to be according to dr-67 & dr-68, for those are the only regulations where damage stability apply anyhow. So the setting of computation method, as discussed in section 18.4.1 on page 385, Calculation method, has no effect here.

18.6.3 Probabilistic damage stablity

The probabilistic damage stability will be calculated according to the regulations 'Agreement for the construction and operation of dredgers assigned reduced freeboards', a.k.a. dr-68.

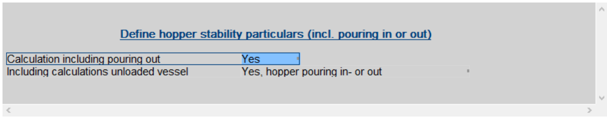
The following aspects must be considered :

- The calculation is made for loading conditions selected in Loading for damage stability.
- These loading conditions may only differ in hopper load and height of the overflow(s).

This is not necessary for the calculation itself, but dr-68 assumes a loading condition with 50% consumables. A warning will follow if there are differences in the selected loading conditions. With the present option the particulars of the probabilistic hopper damage stability calculation can be specified, in an input window as shown below:

- Whether the calculation must be done including pouring out of liquid cargo.
- Whether the light ship condition must be calculated too, and, if so, whether the hopper is assumed to be connected to the sea.

Note on the calculation: The cargo in the hopper compartment is assumed not to flow out when the hopper compartment is damaged.



Input window

18.6.4 Stability calculation with hopper open connected with outside water

To perform this calculation, the following options are available within PIAS:

- At the composed hullforms, the hopper can be subtracted from the hydrostatic model of the vessel. This possibility always exists.
- A damage stability calculation can be made, where a damage case is made with only the hopper.
- In the loading condition, the hopper can be indicated as a 'floodable tank'. This requires option 60.220.0 of the PIAS price list.

18.7 Conversion from old (pre-2018) files

Until October 2018 these hopper stability computations were available in a distinct module labelled *Hopstab*. Data from that module (such as locations of coamings or overflows) cannot be converted to the new program implementation. The amount of data is so limited that the effort of producing a conversion facility would not be worthwhile. However, as an aid in recollecting previously used data, a slimmed-down version of *Hopstab* will be available for a prolonged period of time. This version has lost all its computation functions, but kept its input menus, where the original input data can still be found. This version might even not be included in the PIAS menu, but can still be found independently, as `Hopstab.exe` between all other modules in the PIAS program directory.

Chapter 19

Sounding: calculate tank particulars including effects of heel and trim

This module computes, for compartments defined with Layout, tank capacities and corresponding COG for an arbitrary list/trim combination, if required also with temperature correction to account for expansion of cargo or tank structure. These results can be utilized in a cargo/ullage report or be exported to a loading condition.

This module has been integrated into Loading from April 2023, see section 16.2.1 on page 307, Define/edit weight items and section 17.2.2.2 on page 354, Edit for the new implementation.

Tank contents with heel and trim

1	Specify list and trim
2	Calculate tank particulars
3	Print all tank particulars on paper
4	Cargo/ullage report, and historical cargo summary
5	Export tank data to a loading condition
6	Import tank data from tank measurement systeem
7	Up-to-date overview of filling and flow rate per tank

19.1 Specify list and trim

An input window pops up, where the following data can be given:

- *Trim in meters* (Trim by bow positive), which is the difference in draft on the FPP and the APP (Tfpp - Tapp).
- *Angle of inclination in degrees* only positive angles can be filled in here.
- *The above given heeling angle is to* PS or SB.
- *Mean draft*. This draft is read out by the draft sensor, which is an option in the context of LOCOPIAS. For calculating the tank capacities only, it is not required to give a draft.

19.2 Calculate tank particulars

An input screen appears for defining all tank particulars, such as sounding or ullage, volume, density and weight. If one of these items is changed, the other items will be adjusted automatically. The ullage can only be used if a sounding pipe has been defined for the specific compartment (for which reference is made to paragraph 9.5.1.2.8 on page 217, Sounding pipe). In the alternative window the LCG, VCG, TCG and FSM can be read. By pressing <Enter>, with the cursor on the line of a compartment, a number of particulars can be defined for the specific compartment:

Tank name

As defined in the LOCOPIAS vessel model.

Include this tank in ullage report
If this compartment should be included in the cargo/ullage report (for an example see section 19.4.1 on the next page, Print Cargo/Ullage report on screen) then this field should be set to 'yes'.

Product (substance)
The name of the product, which will be used in the cargo/ullage report.

Conversion table
For the calculation of the cargo weight of heated hydrocarbons, the following conversion tables are available:

- No temperature correction.
- Correction factor per degree. The 'Volume Correction Factor' is calculated according to the defined temperature and the correction factor per degree (coefficient of expansion).
- Volume Correction Factor. The 'Volume Correction Factor' can be defined directly.
- Table 54B. The 'Volume Correction Factor' is determined according to ASTM table 54B.
- Table 55. The 'Volume Correction Factor' is determined according to ASTM table 55.

Data link
This is the value that is sent by the tank measurement system (section 19.6 on page 394, Import tank data from tank measurement systeem). The data link value is for checking purposes only.

Temperature
The standard temperature is 15°Celsius. The volume is determined at this temperature. The actual temperature of the substance can be defined here.

Volume (not corrected for expansion)
This is the volume that is calculated according to the sounding or ullage for this compartment. This volume comes from the previous window with the list of all the compartments.

Density at 15°Celsius (in air)/(in vacuum)
The density of the substance at 15°Celsius can be defined here. If the density in air is defined, the density in vacuum is calculated automatically. These two densities are connected to each other and cannot be defined separately.

Correction factor per degree Celsius
This factor is used if the conversion table 'Correction factor per degree' has been selected, and calculates the volume correction factor.

Volume Correction Factor
This factor can be determined with four different methods:

- This factor is defined manually, using conversion table 'Volume Correction Factor'.
- This factor is calculated with the correction factor per degree and the difference between the standard and actual temperature. The conversion table 'Correction factor per degree' must be selected.
- This factor is read out from the conversion table 'Table 54B'.
- This factor is read out from the conversion table 'Table 55'. This factor corrects the density at 15°Celsius of the substance for the actual temperature.

Temperature Expansion Factor
This factor corrects for the expansion of the tank at a higher temperature than 15°Celsius. This factor is calculated automatically and cannot be defined manually.

Density at {defined temperature} degrees
Density at 15°Celsius × Volume Correction Factor.

Residue On Bottom (ROB)
Volume of the residue which will be subtracted from the volume of the tank contents.

Density × Temperature Expansion Factor
Density at 15°Celsius × Volume Correction Factor × Temperature Expansion Factor.

Weight
The weight is calculated according to: Volume (not corrected for expansion) × Density at 15°Celsius × Volume Correction Factor × Temperature Expansion Factor.

19.3 Print all tank particulars on paper

With this option the tank volumes etc. (the same as in the input window of the previous option, see section 19.2 on the previous page, Calculate tank particulars) will be printed. An example is pasted just below.

TANKCONTENTS, INCLUDING EFFECTS OF HEEL AND LIST
M.v. Exempli Gratia

28 Sep 2017 15:53:21

Trim = 1.000 m (trim by bow)
Draft from baseline on FPP = 4.100 m
Draft from baseline on APP = 3.100 m
Angle of inclination = 1.000 degrees (to SB)

Compartment	Sounding m	Volume m^3	S.W. ton/m^3	Weight ton	VCG m	LCG m	TCG m	FSM tonm	Ullage m	Press. mmwater
1 FP WB CL	-0.360	0.000	1.0250	0.000	0.985	131.856	0.015	0.001	15.824	0
2 DT WB CL	-0.065	0.000	1.0250	0.000	0.079	125.002	0.092	0.000	11.201	0
3 DB 1 WB CL	-0.212	0.000	1.0250	0.000	0.023	118.711	0.934	0.001	12.572	0
4 LT 1 WB PS	-0.084	0.000	1.0250	0.000	0.081	113.411	-3.177	0.005	12.443	0
5 LT 1 WB SB	-0.061	0.000	1.0250	0.000	0.080	113.128	3.294	0.005	12.396	0
6 DB 2 WB CL	-0.264	0.000	1.0250	0.000	0.023	106.299	3.658	0.006	12.048	0
7 LT 2 WB PS	-0.179	0.000	1.0250	0.000	0.062	101.220	-5.811	0.009	11.522	0
8 LT 2 WB SB	-0.114	0.000	1.0250	0.000	0.058	100.780	5.986	0.010	11.456	0
10 DB 3 WB CL	-0.143	0.000	1.0250	0.000	0.026	92.508	3.859	0.007	11.683	0
11 LT 3 WB PS	-0.163	0.000	1.0250	0.000	0.029	91.591	-6.245	0.010	11.330	0
12 LT 3 WB SB	-0.090	0.000	1.0250	0.000	0.027	91.339	6.991	0.012	11.236	0
13 AH 4 WB PS	-0.231	0.000	1.0250	0.000	1.333	77.459	-8.606	0.013	10.054	0
14 AH 4 WB SB	-0.228	0.000	1.0250	0.000	1.333	77.461	8.740	0.013	10.051	0
19 DB 5 WB PS	-0.245	0.000	1.0250	0.000	0.024	52.968	-1.387	0.000	11.413	0
20 DB 5 WB SB	-0.105	0.000	1.0250	0.000	0.026	53.070	6.948	0.012	11.273	0
21 WT 5 WB PS	-0.114	0.000	1.0250	0.000	1.333	51.586	-8.606	0.013	9.945	0
22 WT 5 WB SB	-0.127	0.000	1.0250	0.000	1.333	51.587	8.739	0.014	9.958	0
23 DB 6 WB PS	-0.191	0.000	1.0250	0.000	0.024	38.830	-1.386	0.000	11.610	0
24 DB 6 WB SB	-0.075	0.000	1.0250	0.000	0.029	39.659	6.506	0.011	11.494	0
25 WT 6 WB PS	-0.125	0.000	1.0250	0.000	1.335	37.737	-8.575	0.013	9.956	0
26 WT 6 WB SB	-0.140	0.000	1.0250	0.000	1.337	37.929	8.674	0.013	9.971	0
27 AP WB PS	-1.998	0.000	1.0250	0.000	6.711	2.425	-1.524	0.000	10.151	0
28 AP WB SB	-1.991	0.000	1.0250	0.000	6.711	2.470	1.928	0.000	10.142	0
30 GO PS	3.386	33.393	1.0312	34.345	5.813	11.719	-6.975	0.014	3.826	3214
31 GO SB	4.348	52.829	0.8998	47.356	5.338	14.794	7.112	0.000	3.848	3819
32 GO DAY 1 PS	7.680	0.000	0.9000	0.000	7.949	9.904	-5.174	0.007		*
33 GO DAY 2 PS	7.657	0.000	0.9000	0.000	7.923	9.904	-3.925	0.005		*
40 HFO MID PS	11.133	192.327	0.9794	187.388	3.313	79.423	-5.186	0.000	0.000	977985
41 HFO MID SB	11.151	181.832	0.9710	176.564	3.721	79.597	5.642	0.000	0.000	969324
42 HFO OVERFL CL	-0.100	0.000	0.9500	0.000	4.301	80.554	0.263	0.002	6.971	0
43 DB 4 HFO PS	1.082	200.000	0.9919	198.370	0.611	65.779	-4.446	1125.585	10.086	1101
44 DB 4 HFO SB	1.262	150.000	0.9921	148.822	0.645	65.735	5.917	311.143	9.907	1029
45 HFO SETTLLING PS	6.947	0.000	0.9500	0.000	7.083	19.537	-5.273	0.005		*
46 HFO DAY PS	9.842	20.000	0.9702	18.919	8.837	18.903	-6.235	8.573		*
50 LO CIRC CL	0.926	10.000	0.8602	8.389	1.282	16.569	0.005	2.547	1.066	790
51 LO ME STORE PS	7.085	0.000	0.9000	0.000	7.163	9.615	-7.788	0.010		*
52 LO AE STORE SB	7.408	0.000	0.9000	0.000	7.950	4.801	4.567	0.005		*
53 LO GB STORE SB	7.399	0.000	0.9000	0.000	7.950	6.001	4.567	0.005		*
60 DB CW DRAIN SB	-0.119	0.000	1.0000	0.000	0.436	16.055	1.816	0.002	4.069	
61 TO DRAIN SB	-0.049	0.000	0.9000	0.000	0.476	18.348	1.956	0.002	11.249	
62 DB LEAK OIL SB	-0.040	0.000	0.9000	0.000	0.905	19.503	2.246	0.002	11.289	0
63 DB DIRTY OIL CL	-0.875	0.000	0.9000	0.000	0.087	11.027	0.072	0.001	4.194	0
64 OVERFLOW PS	-0.123	0.000	0.9000	0.000	4.379	17.408	-7.029	0.008	7.130	
65 SEWAGE SB	-0.143	0.000	1.0000	0.000	4.731	11.283	6.153	0.007	6.985	0
66 TO STORE SB	7.417	0.000	0.9000	0.000	7.950	3.601	4.567	0.005		*
68 BILGE WATER PS	-0.081	0.000	1.0000	0.000	0.257	17.664	-1.811	0.002	11.299	
69A SEPARATOR WATER PS	-0.130	0.000	1.0000	0.000	4.685	14.727	-6.857	0.009	6.782	0
69B SLUDGE FO/LO PS	-0.116	0.000	1.0000	0.000	4.374	16.210	-6.926	0.009	7.129	0
70 AP STERN CL	-0.057	0.000	1.0000	0.000	0.270	8.170	0.027	0.001	8.463	
75 AP FW PS	4.184	0.000	1.0000	0.000	4.589	7.491	-0.512	0.000		0 *
76 AP FW SB	4.164	0.000	1.0000	0.000	4.584	7.530	0.558	0.001		0 *

The tanks marked with an '*' are not corrected for list

Table with tank volumes and COGs.

19.4 Cargo/ullage report, and historical cargo summary

Cargo/ullage report, and historical cargo summary

1	Print Cargo/Ullage report on screen
2	Print Cargo/Ullage report on paper
3	Print historical cargo summary
4	View and maintain historical cargo summary

19.4.1 Print Cargo/Ullage report on screen

This option allows you to print an overview of all onboard cargoes, including their weight, temperature effect, sounding and ullage etc., see example below. This list includes only those tanks of which the detail particulars (as discussed in section 19.2 on page 390, Calculate tank particulars), at the second row 'include this tank in ullage report' is switched on. Before this report is created some more questions might be asked, such as the Bill of Lading weight, and whether this list should be stored at the historical cargo summary.

CARGO, SOUNDING AND ULLAGE REPORT
M.v. Exempli Gratia

28 Sep 2017 15:50:47

Trim = 1.000 m (trim by bow)
Draft from baseline on FPP = 4.100 m
Draft from baseline on APP = 3.100 m
Angle of inclination = 1.000 degrees (to SB)

Port of loading / discharge: Rotterdam
Berth: Alexander
Voyage number: 354

Tank	Product	Ullage	Sounding	Press.	Temp.	Volume	TEF	ROB	Obs.Volume	Method
30 GO PS	Gas Oil	3.826	3.386	3214	55.0	33.393	1.15522	0.100	38.476	MANUAL
31 GO SB	Gas Oil	3.848	4.348	3819	50.0	52.829	1.00423	0.200	52.852	MANUAL
43 DB 4 HFO PS	Heavy Fuel Oil	10.086	1.082	1101	50.0	200.000	1.05406	0.000	210.813	MANUAL
44 DB 4 HFO SB	Heavy Fuel Oil	9.907	1.262	1029	60.0	150.000	1.06954	0.000	160.431	MANUAL
46 HFO DAY PS	Heavy Fuel Oil		9.842		50.0	20.000	1.00082	0.500	19.516	MANUAL
50 LO CIRC CL	Lub Oil	1.066	0.926	790	80.0	10.000	1.00687	0.250	9.819	MANUAL

Tank	Table	Corr./degr.	VCF	Volume 15	Density 15 Vacuum	Density 15 Air	Weight Vacuum	Weight Air
30 GO PS	Nynas		0.9702	37.332	0.9211	0.9200	34.383	34.345
31 GO SB	Nynas		0.9739	51.474	0.9211	0.9200	47.408	47.356
43 DB 4 HFO PS		0.001000	0.9641	203.253	0.9771	0.9760	198.589	198.370
44 DB 4 HFO SB		0.001000	0.9537	153.008	0.9737	0.9726	148.985	148.822
46 HFO DAY PS	ASTM55		0.9782	19.091	0.9921	0.9910	18.940	18.919
50 LO CIRC CL	ASTM54B		0.9493	9.321	0.9011	0.9000	8.398	8.389

Volume : Volume corrected for list and trim
Obs.Volume : "Observed" volume: corrected for tank expansion (TEF)
Volume 15 : Volume at 15 degrees (corrected for cargo expansion)
Density 15 : Density at 15 degrees Celsius
TEF : Temperature Expansion Factor
ROB : Residu On Bottom
Table : Table used for temperature correction
Corr./degr. : Volume correction per degree Celsius
VCF : Volume Correctie Factor

Product	Density Air	Mean Temp.	Observed Volume	Volume 15	Barrels	Weight Vacuum	Weight Air	B/L Weight	Diff. %
Gas Oil	0.92000	52.1	91.328	88.805	558.5	81.791	81.701	81.000	0.86
Heavy Fuel Oil	0.97538	54.1	390.760	375.352	2360.7	366.514	366.111	370.000	1.06
Lub Oil	0.90000	80.0	9.819	9.321	58.6	8.398	8.389	8.250	1.65
Totals :			491.907	473.478	2977.8	456.703	456.201	459.250	0.67

For stabilised crude oil K0 = 613.9723 and K1 = 0 (for metric units)

Shipper / Receiver (On behalf of) the master

....

Example of a cargo/ullage report.

19.4.2 Print Cargo/Ullage report on paper

The same as previous option, albeit with output to paper.

19.4.3 Print historical cargo summary

19.4.4 View and maintain historical cargo summary

These options will speak for themselves.

19.5 Export tank data to a loading condition

A list of all defined loading conditions appears. One of these loading conditions can be selected. The selected loading condition will be copied and the tank data of the sounding module will be sent to this copy. The name of this new loading condition will be: name of selected loading condition + ‘tank reading’ + date + time.

19.6 Import tank data from tank measurement systeem

With this option the soundings or ullages of the tank measurement system can be read out and processed in the list of all tanks (section 19.2 on page 390, Calculate tank particulars).

19.7 Up-to-date overview of filling and flow rate per tank

This option opens a window in which the current filling and flow rates are displayed for each tank, as well as the remaining time until the desired filling percentage will be reached. These values are refreshed by default every five minutes, but that interval is adjustable. Obviously, this option can only work if there a connection with a tank measurement system is available.

Chapter 20

Tools for data overview, intact stability and damage stability

instead of being related to a specific PIAS module, this chapter describes a number of tools which can be applied at multiple tasks, and which are consequently included in a number of modules, which are:

- *Weight groups.*
- *Sketches of tanks, compartments and damage cases.*
- *Input and edit damage cases.*
- *Generate damage cases on basis of the extent of damage.*

20.1 Weight groups

*A weight group is a category of cargo or other loading, for example 'ballast water' or 'heavy fuel oil', and is introduced to provide some order is lists of compartments and weight items. If weight groups are being used then, where relevant, Loading will also generate subtotals (of weights and sometimes also COGs) per weight group. Please realize that the concept 'weight group' is an auxiliary tool, **its use is not obligatory**.*

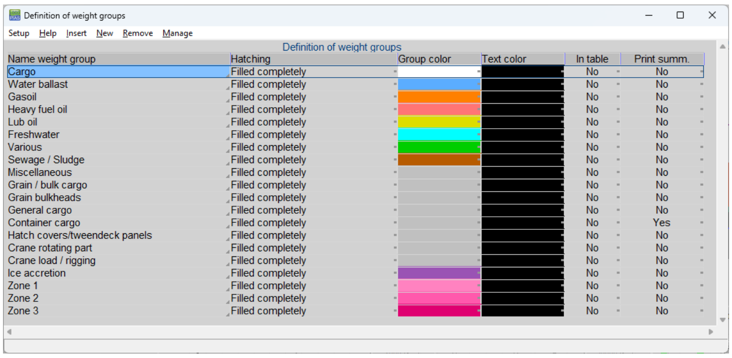
Weight groups can, up to a maximum of forty, be defined from Loading and Layout, an example of the input window is shown below. Each compartment or weight item can be assigned to a particular weight group by pressing <Spatie> there, which pops up a selection window with all full weight group names. It is also possible to create a new weight group here.

For each weight group can be given:

- The *name*, just a description.
- The *hatching* type which is used when hatching or filling in the compartments in tank sketch plots, as discussed in section 20.2 on the following page, Sketches of tanks, compartments and damage cases.
- The *group color*, which is the color representing this weight group, and which is used in plots, and also as background color in text windows if the last column of this weight group is set to 'yes'.
- The *text color*, which, if the last column is set to 'yes', specifies the foreground color in textual overview windows of the texts which belong to this weight group.
- *In table*, which indicates whether the weight group color should also be used in overview tables of compartments and weight items.
- *Print summ.*, which indicates whether in the output only the subtotal should be printed. The calculation is based on all weight items though.

Additionally, a specific function apply:

- With [Move] a weight group can be moved in the list.



Name weight group	Hatching	Group color	Text color	In table	Print summ.
Cargo	Filled completely			No	No
Water ballast	Filled completely			No	No
Gasoil	Filled completely			No	No
Heavy fuel oil	Filled completely			No	No
Lub oil	Filled completely			No	No
Freshwater	Filled completely			No	No
Various	Filled completely			No	No
Sewage / Sludge	Filled completely			No	No
Miscellaneous	Filled completely			No	No
Grain / bulk cargo	Filled completely			No	No
Grain bulkheads	Filled completely			No	No
General cargo	Filled completely			No	No
Container cargo	Filled completely			No	Yes
Hatch covers/tweendeck panels	Filled completely			No	No
Crane rotating part	Filled completely			No	No
Crane load / rigging	Filled completely			No	No
Ice accretion	Filled completely			No	No
Zone 1	Filled completely			No	No
Zone 2	Filled completely			No	No
Zone 3	Filled completely			No	No

Specify weight group properties

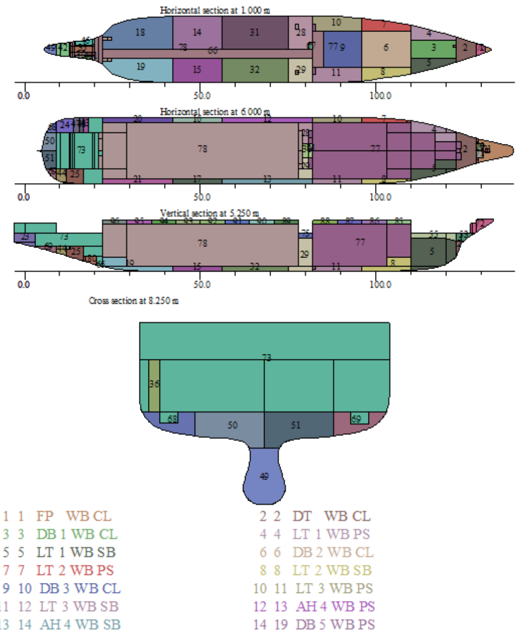
20.2 Sketches of tanks, compartments and damage cases

If this option has been purchased, it enables you to define views and section which will be used when printing compartment definition data, with damage stability calculations and with loading condition to generate large-scale (= small-size) drawings of compartments and hullform etc. The color and hatching type of these graphics can be specified per weight group, for which we refer to section 20.1 on the previous page, Weight groups. If multiple sketches are defined they will be 'pasted on paper' beneath each other, an example of the output is present at the end of this chapter.

This menu, where the sketch parameters can be given, can be called from many loactions, such as from Loading or Layout, but also from the general configurations with Config or equivalent. In each of these cases an input window appears where on each line the properties of one sketch can be given, with per column:

- The first three columns, *compart*, *intact* and *damage*, determine whether this sketch is added to the output of compartment definition output from Layout, intact stability calculations and damage stability calculations respectively. In the output of intact stability calculations each compartment which is filled to some extent will be fully hatched, so no distinction will be made between partially and fully filled tanks.
- The fourth column defined the *type of sketch*, where the choices are:
  - Vertical section.
  - Horizontal section.
  - Cross section.
  - Side view.
  - Top view.
  - List of compartments. This is an alphanumerical list of compartments which are relevant in this sketch.
  - Wind contour, which is only used in the output of intact stability.
  - Probability triangle, which is only used in the output of probabilistisc damage stability.
  - Hopper cross sections, applicable only if there is a hopper and in the output of intact stability.
  - Hopper vertical section, applicable only if there is a hopper and in the output of intact stability.
- *Location*, the location of the sketch, if applicable.
- *Damage center*, ensures that the sketch is always taken through the center of the damage case. Applies only to sketches selected for *damage*.
- A *reduction factor* on the plot size. A factor of X will reduce the size of the sketch by X.
- *Axis & scale*, which is used to specify whether a horizontal axis and a scale must be co-plotted.
- *Horizontal axis*, here can select the unity for the horizontal axis; meters or frames.
- *Identification compartment*, here you can choose how a compartment is indentified in a picture. The choice is between:
  - An automatically generated number
  - The name of the compartment
  - The second name of the compartment
  - The abbreviation
  - No identification If a legend is printed, at least a number is printed with the tank, possibly supplemented by name or abbreviation.

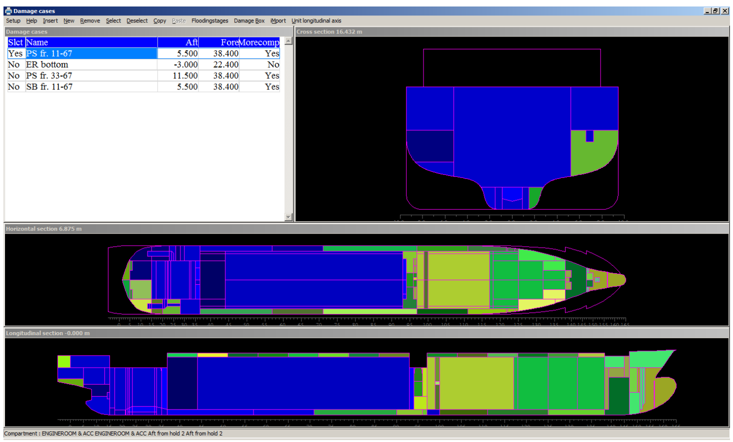
- *Waterline*, a setting that indicated whether the waterline should be drawn as well into the sketches of the intact stability and the deterministic damage stability.
- *Filling*, if in the compartments the filling level is to be hatched, only valid for intact stability.



A page with tank sketches containing 4 sections and a compartment list

20.3 Input and edit damage cases

This tool, designed to define and edit damage cases, can be called upon from all modules which perform a damage stability task, both deterministic and probabilistic. Basically, this tool is rather simple, after all compartments have already been defined (with Layout), and here they can be clicked to toggle them between 'flooded' and 'not flooded'. This tool contains a text window at the left and three graphical windows with three sections, see the example below.



Damage cases

In the graphical windows the damaged compartments are indicated with a blueish color, non-damaged compartments in green-yellow. With the mouse the cursor can be placed or moved in such a window, which will make that the other sections will be adapted to the mouse position.

Attention

Before December 2021, the Loading, Hydrotables (maximum allowable VCG in damaged condition) and Probdam modules had their own list of damage cases and there was an Import option to transfer damage cases from one list to another. From the above mentioned date all damage cases are in 1 list and each damage case can be switched on or off for each type of calculation of a module. All existing damage cases are merged into this list. By default, only those damage cases that are selected for the module in question are visible, but with the option 'View' the whole list can be made visible.

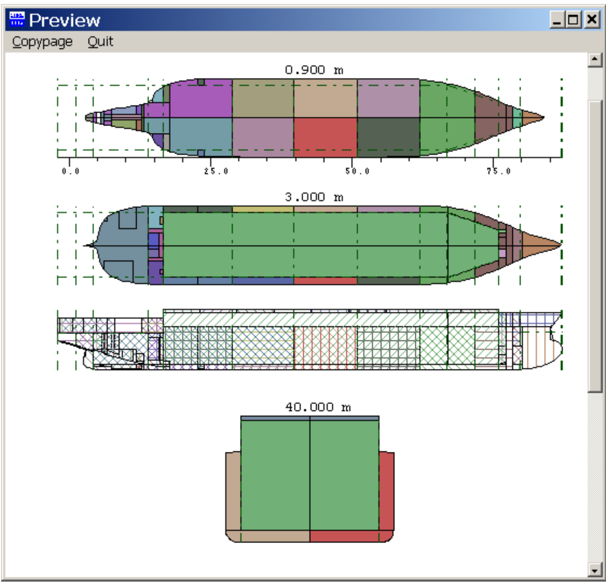
The columns in the text window have the following meaning:

- *Slct*, selected, yes or no.
- *Lock*, a damage case can be locked for changes.
- *Name*, the name of the damage case.
- *Aft* and *Fore*, being the aft and forward boundary of the damage. These parameters are relevant for the damage stabilitycriterion 'deckline not submerged outside flooded area', and for a calculation according to STAB90+50, because there the freeboard is taken into account. Recall that the freeboard is calculated from the deckline, the way of defining which is addressed at section 7.2.9 on page 185, Deck line. If these criteria do not apply, the boundaries need not be specified.
- *Upper*, *Ainside* and *Finside*, upper, aft inner and forward inner boundaries of a generated damage case in the probabilistic damage stability.
- *Crit.final* and *Crit.interm.*, one can select an alternative set of stability requirements for this damage case.
- *Level interm.*, one can set a different setting for the water level during an intermediate stage of flooding.
- *Flood.stages*, here it is indicated whether a complex intermediate stage — a concept for which reference is made to section 21.3 on page 408, Complex stages of flooding (before 2023) — has been defined for this damage case.
- *Higher subd.*, here it is indicated whether for this damage case higher subdamages should be calculated. The higher subdamage exists as interpretation of *a lesser extent of damage* as described in SOLAS part B-1 regulation 7.6: "The assumed vertical extent of damage is to extend from the baseline upwards to any watertight horizontal subdivision above the waterline or higher. However, if a lesser extent of damage will give a more severe result, such extent is to be assumed". With this option switched on, additional damage stability calculations will be made with tanks or compartments below a systematic set of horizontal boundaries non-flooded.
- *Valid Loading/Hydrot./Probdam*, here is indicated for which kind of calculation the damage case should be included, provided that the damage case is also selected.

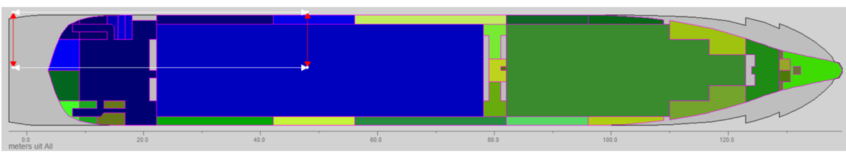
Additionally, a number of specific functions apply:

- With [Flooding stages] non-standard intermediate stages of flooding can be defined, this is futher discussed at section 21.3 on page 408, Complex stages of flooding (before 2023).
- With [damage Box], interactively a rectangle can be dragged, which makes the contained compartments flooded. This is a quick and consistent tool for declaring a large number of compartments flooded simultaneously.
  - With [Edit damage boxes] multiple damage boxes can be created that can be quickly switched between using the 'Selected' column. The other columns will speak for themselves.
  - When the option [start selected damage Box] is activated, in the three graphical windows three white rectangles pop up, which are the projections of a three-dimensional rectangular damage. The vertices of the rectangles can be dragged, which adjusts the damage size and location. If the mouse button is released the flooded compartments are colored blueish, see the example below. By pressing the right mouse button on the rectangular damage the size and location can also be modified numerically. The 'Fixed' apsects of a damage box are graphically visible by colouring the lines and arrows red if necessary. Note that selecting another damage case ensures that adjustments to the previous damage case are no longer undoable.
  - With [Create new damage case with selected compartments] the current damage case is added to the list of damage cases and a new damage case is created which can then be modified.
  - With [quit damage box]→[do Not save damaged compartments] stops the damage box and changes to the damage case are not kept.
  - With [quit damage box]→[Save damaged compartments] stops the damage box and the changes made to the damage case are saved.

- With [iMport], damage cases can be read from an XML file (if purchased).
- With [Unit longitudinl axis] the unit of the longitudinal axis can be chosen, where the choice is between meters and frames.
- With [Output] the selected damage cases can be printed and plotted, suboptions:
  - [Print input data of selected damage cases] : Produces a list, which contains for each selected damage case the names of the belonging damaged compartments.
  - [Definition of sections for sketches of damage cases] : Use and background of this option is discussed in section 20.2 on page 396, Sketches of tanks, compartments and damage cases.
  - [pLot of selected damage cases] : With this option sketches of the selected damage cases will be printed according to the specification as given in the previous option.
  - [pRint permeabilities of all compartments] : This option only appears in case of the probabilistic damage stability. With this option a list of permeabilities assigned to the distinct types of spaces is being printed.
  - [plot of Zonal boundaries] : This option only appears in case of the probabilistic damage stability with the zonal method. It will generate a single plot, which contains the compartments and the zonal boundaries, at the sectionsas specified with section 20.2 on page 396, Sketches of tanks, compartments and damage cases. See the example below.

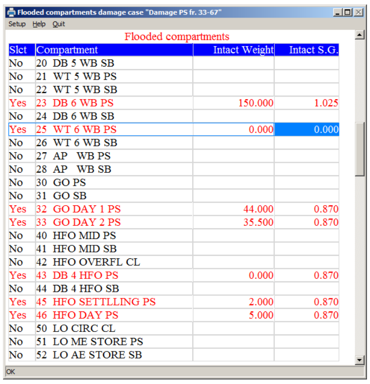


Preview zonal boundaries



Interactive damage box

Finally, with the <Enter> key a subwindow opens up, as depicted in the figure below, where form each compartment is stated whether it is flooded. The last two columns, containing the intact weight and the intact density, only appear when defining damages for *maximum allowable VCG in damaged condition*, for at those computation these intact content plays a role because it will flow out in case of damage (at the deterministic damage stability of a particular loading condition, as can be computed with Loading, the intact content obviously also flows out, but need not to be defined separately, becaause it is already known from the loading condition) .



Slct	Compartment	Intact Weight	Intact S.G.
No	20 DB 5 WB SB		
No	21 WT 5 WB PS		
No	22 WT 5 WB SB		
Yes	23 DB 6 WB PS	150.000	1.025
No	24 DB 6 WB SB		
Yes	25 WT 6 WB PS	0.000	0.000
No	26 WT 6 WB SB		
No	27 AP WB PS		
No	28 AP WB SB		
No	30 GO PS		
No	31 GO SB		
Yes	32 GO DAY 1 PS	44.000	0.870
Yes	33 GO DAY 2 PS	35.500	0.870
No	40 HFO MID PS		
No	41 HFO MID SB		
No	42 HFO OVERFL CL		
Yes	43 DB 4 HFO PS	0.000	0.870
No	44 DB 4 HFO SB		
Yes	45 HFO SETTLLING PS	2.000	0.870
Yes	46 HFO DAY PS	5.000	0.870
No	50 LO CIRC CL		
No	51 LO ME STORE PS		
No	52 LO AE STORE SB		

Flooded compartments per damage case, with their intact content

20.4 Generate damage cases on basis of the extent of damage

In general, damage cases are not chosen at will, they are derived from the extent of damage as laid down in rules and regulations instead. For example it can be stipulated that a ship has to survive a damage with a length of 10% of the ship's length, 1/5 of the vessel's breadth and unlimited height. Subsequently it is up to the designer to identify and define all resulting damage cases. this task can also be preformed with this PIAS function.

For this purpose, the currently discussed functionality is developped, where *damage dimensions* can be entered. Even multiple sets of damage dimensions, because for different regions different extents of damage can be applicable (e.g. for the forward 30% of vessel a bottom damage with a breadth of 5 meter, while for the other 70% a breadth of 3 meter will suffice). For each damage dimension can be entered:

- Description. This name is to recognize this set, and is also assigned to the damage cases generated under this set.
- Damage type. Three kinds of damage types exist: side damage SB, side damage PS and bottom damage.
- Length. The damage length, the longitudinal extent of damage.
- Penetration. With side damage this is the transverse extent of damage (measured from CWL), with bottom damage this is the vertical extent of damage (measured from the bottom).
- Dimension. With side damage this is the vertical extent of damage (which, by the way, is unlimited in most rules). With bottom damage this is the transverse extent of damage, the damage breadth.
- Aft boundary and forward boundary. These are the boundaries of applicability of this dimension set. With the mentioned example where for the forward 30% another regime applies that for the other 70%, two dimension sets have to be defined, one with boundaries aft and 70%Lpp, and the other one with boundaries 70%Lpp and forward.

Furthermore, two additional functions are available:

- [Standard]. A utility function, which can be used to calculate a standard dimension quickly. For example, a rule exists where the damage length is prescribed to be $1/3L^{2/3}$. With this function, [Standard], this equation be called and the number calculated. Because for each dimension (L, B and H) other rules apply this function works by menu cell.
- [Generate]. With this function the damage cases will be generated. Prior to that, a window pops up where the generation preferences can be specified :
  - The choice of between 'In addition to existing damage cases' and 'As replacement of existing damage cases' existing damage will be obvious.
  - With 'Prevent minor damages' the so-called *minor damages* — damages from which the compartments are a subset of aother, more extensive, damage — will not be generated. If such minor damages are indeed to be generated, then systematically damages are created with less than the maximum penetration from the side, however, the vertical and longitudinal directions are not systematically searched for that purpose. For the reason that in that fashion relatively large amounts of minor damages would be found. which, as a rule, might be expected to be less severe than the main damage. And although hundreds of minor damages can very well be managed by PIAS, it leaves humans with very little overview.

- Whether ' mutually identical damages' should be avoided. As a rule, this will be desirable, what is after all the usefulness of multiple, identical damage cases which are inflicted from multiple directions (eg. side and bottom) ?

By the way, regulations may allow that damage to the ER, or involving ER bulkheads, are not taken into account. Nevertheless these will be generated with this [Generate] function. Such superfluous damage cases will have to be removed by the user afterwards, under the slogan "throwing away is easier then adding". A possibly easier alternative is to use different regions of damage dimensions, using the aft and forward boundaries thereof. One has to use region boundaries slightly aft or before the ER bulkhead, otherwise that bulkhead will unintendently still become damaged.

Chapter 21

Internal flooding in case of damage, through pipe lines and compartment connections

When a ship becomes damaged, the flooding need not be confined to the immediately ruptured compartments, but may also extend to other compartments due to the presence of pipelines, ducts or other forms of compartment connections. To this end, PIAS is equipped with a number of tools and mechanisms that will be discussed in this chapter.

21.1 Background from tools for ship-internal connections in PIAS

If a compartment is damaged in such a way that it is open to sea water then it will obviously be flooded, which can also extend further into the vessel through all kinds of connections between compartments. In stability regulations, the word *progressive flooding* is sometimes used for this, but we rather avoid this word because it suggests that the flooding continues until it is fatal, which of course is not necessarily the case. Obviously, this process can be modelled in PIAS and computed. There are two facilities for this purpose:

- The first dates back to ±1990 and is called **Complex intermediate stages of flooding**. This works on the basis of non-uniform filling percentages per compartment, supplemented where necessary by virtual compartment connections. This allows to specify whether there was a connection between two compartments, however, it had no geometry (although it had an optional threshold height, called a 'critical point'). There was also only a single connection possible between two compartments. In section 22.3.1.6 on page 427, Define compartment connections, a table of such connections can be defined, which is used in damage case generation to co-generate such complex intermediate stages with. This Probdam function has been extended in 2018[1] and that will be the last modification to this 'complex stages' system. Although it will not be disappear, its further development has stopped, as it has been replaced by *consecutive flooding*.
- The second system was introduced in 2023 and is called **Consecutive flooding**. It was developed based on the specification *New inter-compartment flooding mechanism in PIAS*, which was drafted in 2018, in collaboration with a number of key users op PIAS. *Consecutuve Flooding* works on the basis of the actual geometry of pipes and connections, and can also calculate flooding in time domain.

The choice between these two systems can be made in Config (or through Project Setup in the upper bar of the PIAS window), Please also refer for this setting to section 5.4.1 on page 46, Calculation damage stability according to the method of, which also contains a list of PIAS features and functions which are disabled in combination with Consecutive Flooding.

This chapter discusses the following:

- The post-2021 system, Flooding through ducts and pipes: Consecutive Flooding, after 2022 (discussed on the next page).
- The pre-2023 system, Complex stages of flooding (before 2023) (discussed on page 408).
- An elucidation on the principles of calculations for both methods, Underpinning of (damage) stability computations during flooding (discussed on page 411).

[1] https://www.sarc.nl/new-generation-method-for-compartment-connections/

21.2 Flooding through ducts and pipes: Consecutive Flooding, after 2022

This system is discussed in the four sections below, viz.:

1. The basic operation of the conventional intermediate stages of flooding method in consecutive flooding; With conventional intermediate stages of flooding ("Fractional") (discussed on this page).
2. The basic operation of the time domain computation in consecutive flooding; Damage stability in time domain (discussed on page 406).
3. The background of the definition of some pipeline properties in Layout, see Hydrodynamic parameters from pipes and piping systems (discussed on page 406).
4. Finally, a short section where the different settings are summarized, in Summary of settings for Consecutive Flooding (discussed on page 408).

This order may appear to be a bit unnatural — because we should first define, before being able to use — but is nevertheless deliberately chosen this way. By the way, before doing any calculations, one will of course have to define the ducts and pipes. This is done integrated with bulkheads, decks and compartments, in Layout, as described in section 9.6 on page 224, Pipe lines and piping systems.

21.2.1 With conventional intermediate stages of flooding ("Fractional")

This method was conceptualized given two facts:

- Standard stability regulations apply the concept of “intermediates stages of floodings” of fixed percentages of flooding, i.e. 25%, 50%, 75% and 100%.
- Not all compartments are always flooded with the same percentages, i.e. with small connections between compartments the flooding of the connected compartments may lag behind the flooding of the ruptured compartment.

In the elder “compartment connection” method of PIAS the latter was facilitated by so-called “complex stages of flooding”, which support individual percentages of flooding for different compartments. This offered full freedom, however at the cost of significant manual input labour. For the numerous damage cases of probabilistic damage stability this is not practical, so module Probdam offered a specific feature to generate these complex stages, where the binary concepts of “open” and “pipe” (as discussed in paragraph 22.3.1.1.24 on page 425, Damage cases generation including "progressive flooding") offered a flexibility sufficient for the majority of cases, but not for all cases. So, in the *consecutive flooding* system a novel subsystem for unequal intermediate stages of flooding has been created which is a) flexible, b) based on *generation* so does not require much user input, and c) works for all PIAS damage stability calculation modules.

This subsystem maintains the notion of “percentual stage of flooding”, because a) this is a fundamental concept in present damage stability regulations, b) therefore this concept is familiar to authorities and classification societies and c) the concept is easy to understand. In order to have a shorthand word for this concept it was labelled “**fractional**”, because essentially it fills compartments by ‘fractions’ of the final volume. So that fraction is the unit, which enables us to introduce an integer “delay” in the flooding of connected compartments. Assume, for the time being, that the percentages of flooding are 0, 25, 50, 75 and 100%, so one fraction corresponds with 25%. If we use a delay of zero (so, no delay), then the flooding of a connected compartment will obviously be the same as for the ruptured compartment:

Fraction ruptured compartment	Fraction connected compartment
1 (=25%)	1 (=25%)
2 (=50%)	2 (=50%)
3 (=75%)	3 (=75%)
4 (=100%)	4 (=100%)

With a delay of 1, there will be a single fraction delay:

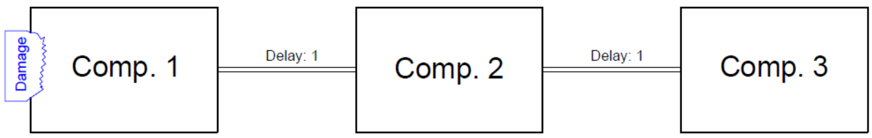
Fraction ruptured compartment	Fraction connected compartment
1 (=25%)	0 (=0%)
2 (=50%)	1 (=25%)
3 (=75%)	2 (=50%)
4 (=100%)	3 (=75%)
4 (=100%)	4 (=100%)

The last row is added because the filling should always end up with all flooded compartments filled to their final levels.

And with a delay≥4:

Fraction ruptured compartment	Fraction connected compartment
1 (=25%)	0 (=0%)
2 (=50%)	0 (=0%)
3 (=75%)	0 (=0%)
4 (=100%)	0 (=0%)
4 (=100%)	1 (=25%)
4 (=100%)	2 (=50%)
4 (=100%)	3 (=75%)
4 (=100%)	4 (=100%)

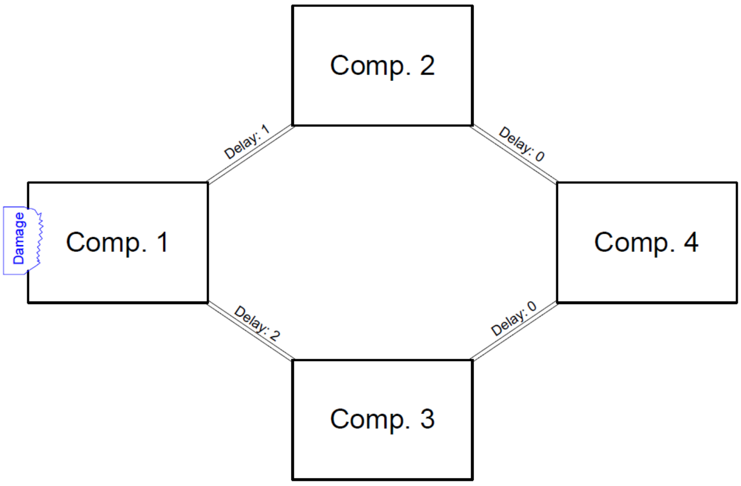
This was an example of a single connection between two compartments. But also cascaded connections are supported by nature, take for instance this configuration of three compartments and two connections (with each a delay of 1):



Three serially connected compartments

Fraction comp1	Fraction comp2	Fraction comp3
1	0	0
2	1	0
3	2	1
4	3	2
4	4	3
4	4	4

Also more complicated topologies are allowed. The delay factors for the different paths might contradict, but that poses not a real dilemma, because the smallest fraction determines the actual flooding, which is a sound logical consequence of the underlying assumptions. For example this case:

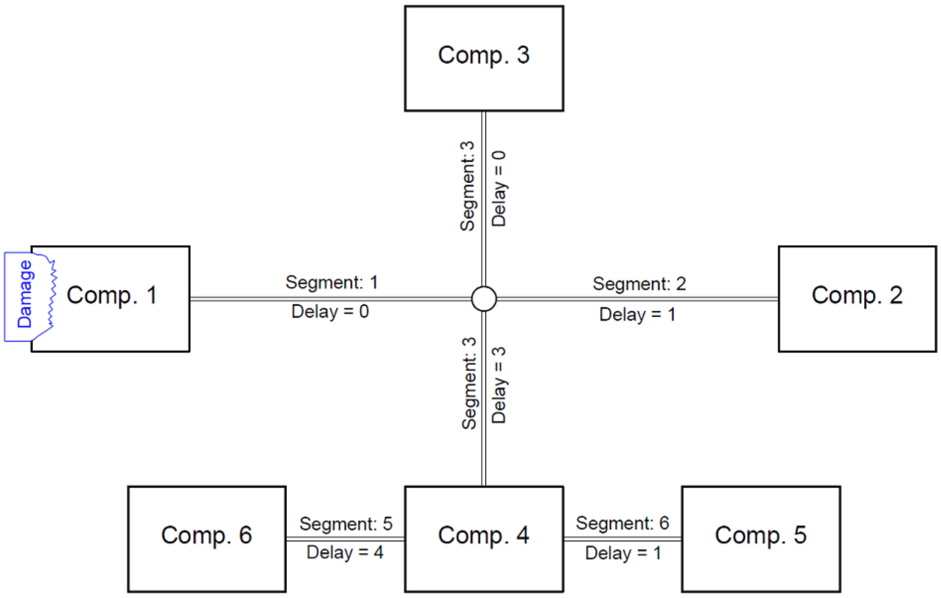


Four connected compartments

Fraction comp1	Fraction comp2	Fraction comp3	Fraction comp4
1	0	0	0
2	1	1	1
3	2	2	2
4	3	3	3
4	4	4	4

Initially the fraction sequence for Comp3 appeared us to be 0,0,1,2,3,4 but through Comp2 and Comp4 the compartment fills up faster.

Delay factors are given for each pipe segment (which is a connection between two joints or compartments, without any branch inbetween, see paragraph 9.6.3.1.1 on page 229, Segment list)). That enables a pipe topology and delay factors such as:



Many connected compartments, with varying delay factors

Which will lead to eleven different stages of flooding (including the final stage).

That's the basic idea. And how realistic will modelling method be? Just as realistic as the whole presumption of fixed percentages of flooding, as imposed by all major damage stability rules. For added flexibility PIAS goes one step further, by allowing multiple (to a maximum of three) delay factors per pipe segment. In most cases a single delay factor will suffice, but even more complex scenarios than presented in this section can be modelled with combinations of factors.

Finally, a remark on the percentages of flooding as used in the examples of this section. As demontration, multiples of 25% have been used, but you will certainly be aware that in PIAS the number of intermediate stages of flooding is user-defined, as well as the percentage of each stage. So, where we have used multiples of 25% here, in reality for this percentage the user-defined flooding percentages will be applied.

21.2.2 Damage stability in time domain

A calculation in the time domain involves analysing, for a whole series of small time steps, how fluids flow through pipes and openings and how that effects ship's position and stability. That is essentially a model which is based on physics, and therefore needs much less explanation than the fractional method, which is a bit artificial. Nevertheless, some choices and assumptions have been applied here and there, which are discussed in Basis of damage stability in time domain (discussed on page 412).

21.2.3 Hydrodynamic parameters from pipes and piping systems

The piping geometry and connectivity can be defined in combination with the internal (i.e. compartment) geometry, with module Layout. This is discussed in section 9.6 on page 224, Pipe lines and piping systems. In order to keep data of the same categories as much together as possible, the flow-related choices and parameters will be discussed in this section.

21.2.3.1 Fluid flow resistance factors

21.2.3.1.1 Frictional resistance from pipes lines

Frictional resistance through pipes is in the essence a complex issue. In practice, however, there are a number of practical methods and parameters in use, and in PIAS we have chosen to implement a selection:

- The (cross-sectional) shape. Choice of round or square, the most common shapes.
- The cross-sectional dimension. If round then diameter, if square then edge length. In meters, as commonly used in PIAS.
- The (dimensionless) resistance coefficient. Three common methods are implemented in PIAS:
  - According to IMO resolution MSC.362(92), where the frictional resistance per meter length is 0.02 ÷ hydraulic diameter.
  - With a user-specified Darcy-Weisbach[2] coefficient, where the frictional resistance per meter of length is that coefficient ÷ hydraulic diameter.
  - With a user-specified resistance coefficient per meter of length.

21.2.3.1.2 Fluid outlet energy loss

The resistance of fluid flowing through a pipe consists of two components. One is the frictional or possibly pressure resistance **in the pipe**, and the other one is due to the energy loss of the fluid outflow **at the pipe outlet**. The latter one simply follows from the fact that the fluid travels with some velocity through the pipe, and hence carries kinetic energy. After outflow (into a reservoir, or into the open air) that velocity vanishes, so the energy content also has gone. This is inevitable, also in a frictionless world. This phenomenon should be included — once, not twice. Unfortunately the different IMO regulations are not consistent in this respect. There are three IMO regulations involved, res. A266 (1973), MSC. 245(83) (2007) and MSC. 362(92) (2013). In A266 and 362 this outflow loss is included implicitly, while in 245 a energy loss factor should be included explicitly, in combination with frictional resistance coefficients of the piping configuration.

PIAS offers you this choice. It can be given in a user setting, in Layout, labelled 'Processing of outlet losses' (please see section 9.6.6 on page 231, General piping settings), with binary choice 'Explicitly user-defined by means of resistance coefficients' or 'Implicitly taken into account'. Please ascertain to set this switch in accordance with the resistance coefficients you assign to the pipe line components, and with this background in mind:

- In A266 en 362 the resistance coefficient formula is $1/\sqrt{1+\sum K}$. This factor 1 in the denominator probably represents the pipe outlet energy loss. In PIAS this is implemented so that it is only included for pipe ends that exit into a compartment or into seawater, and where actual outflow occurs (i.e., not inflow).

[2] https://en.wikipedia.org/wiki/Darcy_friction_factor_formulae

- In MSC.245 the formula reads $1/\sqrt{\sum K}$, where the result may not be taken smaller than 1. To give the user maximum freedom, that limitation to 1 is intentionally not included in PIAS. The user himself must ensure that the resistance coefficients are specified conscientiously (and, incidentally, only applies the factor of 1 for pipe ends where actual outflow occurs).
- If you find it confusing that multiple types of formulas are used in the various IMO rules, and that the background of this factor 1 is not even properly explained then you will find an ally in SARC.

21.2.3.2 Damage stability criteria to be applied

When assessing the damage stability with a cross flooding device, the question may arise which stability criteria to apply; those for final stage or those for intermediate stages of flooding? PIAS' consecutive flooding system has two mechanisms for this, one for the fractional intermediate stages and one for the time domain calculation.

21.2.3.2.1 Choice of stability criteria with the time domain method

In the essence, the application of intermediate or final criteria depends on the time span of the filling process. Some regulations (or accompanying explanatory notes) therefore contain a maximum time period until equalization. This implies that if the ship does not equalize within that time, the cross-flooding arrangement is deemed ineffective, so stability in the non-equalized condition is considerd to be a final stage, which should meet the final stage stability criterla. Well, the whole concept of *equalization* might apply to two simple interconnected tanks on either side of centerline, but for a slightly more complex tank arrangement, it is an abstraction that is not so easy to relate to reality. Nevertheless, the underlying idea is easy to transform into a universal algorithm:

- If the ship comes to rest within the time allowed, then the last time step, which represents the final position, is tested against the final-stage criteria, and earlier time steps from which the damage stability is calculated to the intermediate-stage criteria.
- If the ship does not come to rest within the time allowed, the damage stability calculations for **all** time steps are tested against the criteria for the final stage of flooding.

To this end, that maximum time allowed must be given, which can be done in the general settings for damage stability, see section 5.4 on page 46, General settings damage stability.

21.2.3.2.2 Choice of stability criteria with the fractional method

Above, we have seen that with the time domain calculation, the criterion choice can be elegantly linked to an equalisation time, but for a calculation with conventional intermediate stages of flooding the time information is missing. To allow the user to influence this, PIAS has a facility to specify whether a pipe is large or small. These concepts 'large' and 'small' have no numerical relationship to the pipe size, but only to the choice of stability criteria to be applied, in this fashion:

- Whith 'large' the cross flooding is assumed to be quick, so intermediate stages are thus checked against the (user-specified) criteria for intermediate stages of flooding. If those are not defined, then the criteria for final stage are used, as there is nothing else.
- With 'small', the flooding process is considered to occur slowly, so that also the intermediate stages are tested against the stability criteria for the final stage.

The size, large or small, of a cross section can be specified as a general piping setting (see section 9.6.6 on page 231, General piping settings) with exceptions per network where applicable.

21.2.3.3 Layered definition of flow-related parameters

For the purpose of damage stability calculations, many properties of pipe components can be specified, such as dimensions and coefficients. Because many pipes will be similar it can happen in practice that one is typing in the same numbers over and over again, and nobody likes that. In order to increase the ease of use, PIAS is therefore equipped with the feature to specify some of those parameters in 'layers&rsquo. So, a parameter can explicitly be 'not specified', which will make PIAS to look for the corresponding parameter in a higher layer.

- The pipe resistance coefficient, as discussed in Frictional resistance from pipes lines (discussed on the preceding page), is initially taken to be as specified for that pipe section (which is in a segment). If not specified then the default given in the general pipeline settings (see section 9.6.6 on page 231, General piping settings) will be used. If it is not specified either then the pipe is considered free from resistance.
- The component resistance coefficient is similar: 1) from the component and 2) the default from the general pipeline settings.

- The resistance coefficient of a connection follows the same system as that from a component.
- For the other resistance parameters (shape and size), those as specified for that **individual** pipe, component or connection are taken in the first instance. If these are not specified, then the parameters as defined with their **network** are taken. If that is not specified either, then PIAS looks one level higher, at the default shape and dimensions of the **system** where this thing belongs to. If those are not specified either, then the thing is assumed to have no resistance. There is deliberately no global default for shape and dimensions, because in general there will be no standard pipe size for the ship as a whole. For a particular *system*, e.g. ballast or sounding pipe system, on the other hand such a standard may exist.
- A time-domain calculation is performed with a fixed time interval, in seconds. This can be specified globally at the general settings for damage stability calculations (see section 5.4.3 on page 46, Time domain calculation time step), but it is also possible to set a so-called 'overruling time interval' per piping network. If this is used, it prevails. This mechanism offers the user the possibility to apply a longer interval in networks with small pipes than with large ducts.

Clearly, this system is designed to minimize user input. So, one can specify a default at a higher level (e.g. at the level of a piping system) and leave all the items below it set to 'not specified', hence corresponding to the default. Only the *exceptions* then actually have to be given individually.

21.2.4 Summary of settings for Consecutive Flooding

As can be read in this chapter, Consecutive Flooding is controlled by quite a few settings and parameters. Because these are given in various places in PIAS, the impression may arise that there was no sharp plan behind this when developing the software, but this is by no means the case. Some parameters simply belong to a *physical thing* — a pipe or a valve — and hence to its definition in Layout, while others are *calculation parameters* which belong to the general settings for damage stability calculations, or to a specific piping network. Anyway, to provide the user with an overview, the several settings are summarised in the table below.

TODO make summary in table

21.3 Complex stages of flooding (before 2023)

Preliminary remark: as mentioned in the introduction to this chapter, PIAS now has two systems that can be used to account internal flooding between compartments. The subject of this section's system, complex intermediate stages, was in development until ±2020, after which development has focused on the more advanced Consecutive flooding system, see Flooding through ducts and pipes: Consecutive Flooding, after 2022 (discussed on page 403).

This section a number of distinct facilities will be discussed:

- When necessary PIAS takes into account intermediate stages of flooding. Normally these stages are equal within a damage case for all damaged compartments. With this option a mechanism is available to define the intermediate stages of flooding more specifically, especially according to IMO regulations for seagoing passenger vessels.
- Special kinds of openings can be defined for which it will not be assumed that the vessel will immediately sinks when flooded, but for which the procedure described at the previous bullet will be adopted. Two types of such openings are available: internal openings, which connect the compartment with another compartment, and external openings, which connect a compartment with the sea.
- Calculation of cross-flooding times.
- The use of this function for the calculation of Ro-Ro ferries with water on deck (abbreviated to STAB90+50).

In each module for damage stability of PIAS, with the exception of the computation of floodable lengths, damage cases are defined at least by defining which compartments will be flooded for that case. This is discussed at section 20.3 on page 397, Input and edit damage cases, where in the menu bar the [Flooding stages] function is included. When this fuction is used the following option menu appears:

Name damage case

1	Specify calculation type, number of intermediate stages and other parameters
2	Specify intermediate stages and critical points

21.3.1 Specify calculation type, number of intermediate stages and other parameters

The first option in this menu concerns the calculation type. There are two types of calculation:

- Non-uniform intermediate stages of flooding. This type must be used if intermediate stages of flooding (expressed as percentage of the final stage) are not equal for all compartments. For this calculation type, at the second line the number of intermediate stages can be defined, with a maximum of 12. The final stage of flooding should not be defined because it is included automatically.
- Time calculation for cross-flooding arrangements. With this option for each time step it is determined how much water enters a compartment, and what time a complete flooding of a compartment requires. For this calculation type also the time step and maximum number of time steps must be specified. The time step is used as an integration step in the calculation and this step must not be too large.

21.3.2 Specify intermediate stages and critical points

21.3.2.1 Calculation type 'Non-uniform intermediate stages of flooding'

After selecting this option the following input screen appears which contains all damaged compartments:

Damage case ABC					
Compart-ment	Connected with	Via critical point			
		Length	Breadth	Height	SB&PS
DEF	-	-	-	-	-
PQR	Seawater	10.123	8.123	6.123	Yes
STU	PQR	23.123	8.123	6.123	No
XYZ	STU	43.123	8.123	6.123	No

A critical point defines an internal opening between two damaged compartments. The compartment will only then be flooded (with the percentage of flooding in a certain intermediate stage) when the level of liquid of the compartment in 'Connected with' is higher than the critical point. When for a critical point the column 'SB&PS' is set to 'Yes', than that point exists on SB and on PS (with an equal breadth from CL). The same mechanism is applicable to critical points if 'Connected with' is set to 'Seawater'.

The mechanism contains three limitations:

- Weathertight openings cannot be taken into account, but of course weathertight openings may be specified as usual in Hulldef .
- A compartment which can be flooded through a critical point may not contain any liquid in intact condition.
- With the combination of critical points and intermediate stages of flooding the following mechanism applies:
  - If the calculation is made without 'global equal liquid level' the procedure is as might be expected, that is, that every compartment has its own percentage, and its own level of filling. The compartment which can only be flooded through a critical point will only be flooded if the liquid level of the corresponding compartment exceeds the height of the critical point.
  - A calculation with 'global equal liquid level' is ligically inconsistent with the concept of 'critical point'. Therefore, for the question whether a compartiment is flooded through a critical point is solely determined at the final stage of flooding, and this condition (flooded yes/no) is also used at intermediate stages, regardless the actual liquid level at the critical point.
- If unequal percentages of flooding have been defined here, then the switch 'Equal liquid level', as dissussed in section 5.4.12 on page 49, Intermediate stages with global equal liquid level will be ignored.

With the text cursor on a specific compartment, and pressing <Enter>, the next input screen appears in which the percentage of filling for a certain intermediate stage of flooding for that compartment can be defined, for example:

Compartment XYZ			
Stage Number	Percentage of flooding	Water on deck	Stab.crit.final
1	25	No	No
2	50	No	No
3	75	No	No
4	100	No	No
5	100	No	No
6	100	No	Yes

A calculation with all compartments filled with 100% does not have to be defined, because this is the final stage of flooding which is calculated automatically.

21.3.2.2 Water on deck

According to the rules of the 'Agreement concerning specific stability requirements for Ro-Ro passenger ships undertaking regular scheduled international voyages between or to or from designated ports in North West Europe and the Baltic Sea' (Circular letter 1891), as adopted on 27-28 February 1996. A.k.a. as *'Stockholm agreement'* and by EU directive 2003/25/EC 2003 also applicable to amongst others the Mediterranean. The core of the regulations is an additional amount of water on deck, dependant on the residual freeboard. To include the effects of water on deck:

- If necessary, specify the significant wave height, as discussed in section 5.4.14 on page 50, Significant wave height for SOLAS STAB90+50 (RoRo).
- Define all spaces above deck as compartments.
- Define the deckline (section 7.2.9 on page 185, Deck line).
- Specify the correct permeability for damage stability for the compartments above deck.
- Include the relevant deck compartments in all damage cases.
- For each damage case define a complex stage of flooding, where all compartments below deck are flooded by 100%, and all above deck compartments are marked with 'Yes' in the column 'Water on deck'.
- At each damage case two calculations are made: One with the upperdeck compartments damaged, without extra water on deck, and one with the upperdeck compartments intact, with a fixed amount of water (which moves with heel and trim). At the last calculation in the last column marked '%' the height of the extra amount of water can be read.

21.3.2.3 Calculation type 'Time calculation for cross-flooding arrangements'

This calculation type is aimed at the very simple case of a compartment connected to sea water via a pipe or hole which has fluid flow resistance. Much more complex systems of pipes and connections can be addressed with Consecutive Flooding, see Background from tools for ship-internal connections in PIAS (discussed on page 402).

After chosing this calculation, a list of compartments is presented, where for each compartment can be specified:

- Whether the compartment is flooded through a cross flooding arrangement. If that is not the case the compartment is always filled for 100% (or, in other words, the water level inside is always equal to the sea water level).
- If the compartment is flooded through a cross-flooding arrangement, also the product of cross-section area S (in m^2) and a dimensionless speed reduction factor F must be specified. These parameters, as well as the calculation method for F, are further elucidated in IMO res. MSC.362(92). This PIAS computation of cross-flooding times is discussed with the menu where it can be invoked, in section 16.4.4.2 on page 317, Calculate cross-flooding times.

21.3.2.4 Output

In the output of the deterministic damage stability (with Loading) with complex intermediate stages, the percentage of flooding is not printed in the heading but in the table with weights per compartment. For the intermediate stages of flooding of the computations of maximum allowable VCG' (as discussed in section 10.2.10 on page 256, Maximum VCG' damaged tables and diagrams) only the number of the intermediate stage is printed.

21.4 Underpinning of (damage) stability computations during flooding

The method of damage stability calculations is largely fixed by rules and conventions, and obviously this forms the basis for the implementation in PIAS. However, there are also a number of issues that are less clearly elaborated — such as the question of what exactly is the amount of fluid corresponding to a certain percentage at an intermediate stage of filling, or how to deal with a small internal opening when calculating the stability curve. The choices as made for such issues in PIAS are discussed below, for the two available systems:

- Underpinning at Consecutive Flooding (after 2022)
- Underpinning at Complex Stages (before 2023)

The elder method is based on intermediate stages of flooding, and the newer one also includes a sub-method on that basis. However, the two do differ slightly, which is discussed in Difference in principles at intermediate stages.

By the way, while searching through decades-old documentation for the program approval by classification societies, we still came across the phrase that the damage stability calculation of PIAS includes the free-to-trim effect. A bit overdone to repeat that message, but just to be sure: it still does, also with *Consecutive Flooding*.

21.4.1 Underpinning at Consecutive Flooding (after 2022)

Here we discuss the effects that the internal connections and its components have on the calculations of intact and damaged stability, when using the system of *Consecutive Flooding*.

- Basis of damage stability with fractions ("intermediate stages of filling").
- Basis of damage stability in time domain.
- Basis of larger angle stability (GZ-curve).
- Effects on intact stability.

21.4.1.1 Basis of damage stability with fractions ("intermediate stages of filling")

The whole assumption behind the idea of a *fraction* (a generalization of an *intermediate stage of flooding*, see With conventional intermediate stages of flooding ("Fractional") (discussed on page 403)) is that the immediately affected compartments will be flooded through a **small** damage. After all, if the damage were large, the ingressed water would spread rapidly, and the intermediate stage would be so short that it would have no effect on ship's position and stability. So, then the intermediate stage would actually not exist. Based on this physics-based reasoning, a distinction is made between large and small damages.

To assess stability in damaged condition, the *worst-case scenario* will have to be considered and since it is not known in advance how large the damage will be, cases with both a large and small damage are calculated.

In the event of large damage, seawater can flow freely in and out of the affected compartments, so that even during roll the water level in those compartments is equal to the sea water level. Because this all happens so quickly, intermediate stages do not actually emerge.

In the case of a small damage, on the other hand, the water flows through the hole so slowly that the intermediate stages can take a long time, and thus should be considered separately. However, if the water flows slowly, then during rolling it does not have time to flow in and out significantly. So, in this case the volume of water in a compartment can be assumed to be constant for all heeling angles.

The percentages of the stages of flooding can be set by the user, this is discussed in section 16.2.2.3 on page 313, Define stages of flooding. Suppose intermediate stages of 25, 50 and 75% are given, then the complete damage stability evaluation will consist of:

Damage	**Stage**	**Water in compartiment**	**Verified against stability criteria for**
Large	Final	Freely flowing in and out	Final stage
Small	Final	Constant, as at equilibrium angle (call that W)	Final stage
Small	Intermediate	75% from W	Intermediate stage
Small	Intermediate	50% from W	Intermediate stage
Small	Intermediate	25% from W	Intermediate stage

So, this intermediate-stage system is governed by the size of the damage. However, one could argue that the size of further internal openings (or pipelines or other connections) will also play a role. Indeed, this effect on the computation of the GZ-curve is discussed in Basis of larger angle stability (GZ-curve) (discussed on the next page). Additionally, the sizes of the internal connections also determine the choice of the damage stability criteria to be applied. Also in this area PIAS gives influence to the user, which is discussed in Choice of stability criteria with the fractional method (discussed on page 407).

21.4.1.2 Basis of damage stability in time domain

The whole idea behind the time domain method is that the vessel gradually fills up through openings and pipes and so on. With each time step subdivided in these sub-steps:

1. For time step *t*, tank fillings and densities are known, as well as which tanks are through a damage connected to the sea, so that ship's position — draft, trim and heeling angle — can be determined.
2. With this, all liquid levels are known, and thus pressure differences between tanks, and between tanks and seawater can be can be determined.
3. Since resistance coefficients are also known, Bernoulli's law can be used to determine the fluid velocities in all pipes and openings. And since all cross-sectional areas are also known, the fluid quantities (the flow rates) can also be determined.
4. These flow rates are added to or subtracted from the compartments the pipes are connected to, creating new tank fillings and a new time step *t+1*.
5. With this, the process jumps back to its first step.

This is somewhat simplified — e.g. additional analyses take place, such as checking whether the points of a pipe segment are all below the liquid level, if not the segment flow is blocked — the process, which the user can control by e.g.:

- The time step, given in seconds, amongst others in Config, see section 5.4.3 on page 46, Time domain calculation time step. Obviously, the accuracy of the calculation increases with decreasing time step duration. Using very small time steps, however, leads to longer computation time and more bulky output. An optimal time step cannot be given; at SARC we do try to keep it to a maximum of a few hundred steps.
- The maximum number of time steps, see section 5.4.4 on page 46, Time domain maximum number of time steps. This prevents a calculation taking very long; if this maximum is exceeded then it stops. On the other hand, then one doesn't have a finalized result, so one will still have to restart the calculation with a different time step. This maximum can therefore best be set quite high, it is just intended to cut off extreme cases.

In principle, at each of the time steps, the (damage) stability could be calculated, but that would lead to abundant output. Therefore, there are handles that allow the user to limit those time steps, see section 5.4.3 on page 46, Time domain calculation time step and section 5.4.6 on page 47, Minimum weight difference for a GZ calculation. At the time of such a stability calculation, the fluid content (generally consisting of a mixture of intact content and ingressed sea water) is held constant, and a stability calculation is made according to the same principles as with "fractions" (see Basis of larger angle stability (GZ-curve) (discussed on the next page)). Actually a rather simple mechanism. ALthough there are still a few more details worth mentioning:

- For a piping component, the single point of its *position* is determinative, both for calculating pressure differences in the fluid and for determining whether or not a threshold (or a demi-bulkhead) will overflow. The *dimensions* of the component will not have an effect on these two aspects. The dimensions, in combination with the resistance coefficient, only play a role in determining resistance faced by the fluid flow. As also explained in the examples of section 9.6.8 on page 232, Modelling specific things from the real world.
- The inertia of the vessel and its contents is not taken into account. That means that if e.g. a threshold is overflooded, the water flows *instantaneously* over it. Nor is the momentum of the ingressed water incorporated. As a result, the vessel does not enter a harmonic motion due to the damage and flooding.
- Different types of liquids are supposed to mix. So, oil will not not float on water.
- At the end of the filling process, the system is at rest; then no further fluids flow between sea and/or compartments. At least, it is so because the system is pressure-driven, i.e. at a pressure difference fluids start to flow and mix. However, there is also a long-term effect dispersion that causes a mixture to dissolve in seawater more and more over time. So that after a long time, the mixture is completely replaced by seawater. If the end of the filling process results in a mixture of seawater and original compartment contents, then because

of *worst case* consideration, an additional "time"-step will be added with the damaged compartments (in equilibrium) filled with seawater. Damage stability will also be calculated for that step, from which the time is indicated by an ∞ in the output.

- The relevant IMO resolutions also contain a formula to determine the cross-flooding time. However, difference could arise between these IMO formulae and PIAS. For the reason that the IMO resolutions apply an approximation method, while the time-stepwise computation of PIAS will in general yield a more accurate results. This application is covered by section 4 van MSC.362 (92): "As an alternative to the provisions in sections 2 and 3, and for arrangements other than those shown in appendix 2, direct calculation using computational fluid dynamics, **time-domain simulations** or model testing may also be used".

21.4.1.3 Basis of larger angle stability (GZ-curve)

In Basis of damage stability with fractions ("intermediate stages of filling") (discussed on page 411), it has been plausibly shown that a small damage results in the volume of the compartment located behind the damage can be assumed constant during heel, whereas with a large damage the fluid can flow in and out freely during heel. Exactly the same reasoning applies to internal connections, holes and pipes. This treats internal connectionss in the same way as the damage, albeit that with the damage, since only one of these is assumed, a *worst case scenario* with combinations of small and large damages can be drawn up. Internal connections, on the other hand, can be numerous and it would take large amounts of computing time to start calculating all kinds of combinations of large and small from them. Therefore, for the internal connections, it was chosen to have the demarcation between 'large' and 'small' set by the user, which was discussed atconfig,config_damage_stability_CFminarea}. In short, if the cross-sectional area of a connection is larger than this value, then the fluid flows freely through that connection during heel, and otherwise not.

By the way, it will be obvious that due to sudden fluid transfer between heeling angles, the GZ curve will not always be nice and smooth. Traditionally, the GZ (plus associated draft and trim) is calculated at a fixed series of angles (of, e.g. 5°, 10°, 20°, 30° etcetera) but this does not model discontinuities in the GZ curve properly. After all, it will make a difference when fluid flows over an internal threshold at 21°or at 29°. Therefore, in Consecutive Flooding the GZ calculations are done at many more angles, in the order of every degree. To limit the output in loading and damage calculations, only the angles set at section 5.2 on page 45, Angles of inclination for stability calculations are printed from these. Obviously, **this larger angle range will increase computation time correspondingly**; that is the price to pay for the increased accuracy. A small price, because PIAS has quite a few features (see section 3.11 on page 32, Speed enhancing mechanisms in PIAS: PIAS/ES) that can speed up series of (damage) stability calculations by several times.

21.4.1.4 Effects on intact stability

The application of *Consecutive Flooding* is not limited to damage stability; also in intact conditions a compartment will be flooded through a submerged opening connected by a piping system. While that could be an interesting possibility, the practical importance of this is so small that it will not be included in PIAS for now.

21.4.2 Underpinning at Complex Stages (before 2023)

The method of calculation in intermediate stages of flooding is simple in the essence, with the following steps:

- For each set inclination angle, for the final stage of flooding the ship's floating position (= draft and trim) is determined, as well as the weight of ingressed sea water in each compartment. -At each intermediate stage of flooding of X%, linear interpolation is performed per compartment between the weight in intact condition (0% intermediate stage) and in fully damaged condition (100% intermediate stage). With those weights per compartment, the floating position is calculated at that gradient. And the righting moment.
- In this way, the term "intermediate percentual stage of flooding" is understood in a literal sense; it is precisely the percentual interpolation between 0% and 100%. This way provides continuity between 0 and 1% and between 99 and 100%.
- This calculation scheme is complicated a bit by a disturbing regulatory element, which is that in general for the determination of **tank** volumes the stability rules assume a permeability (μ) of 98%, while for **damage stability** the μ never needs to be taken higher than 95%. Particularly annoying and physically untenable, but a fact of life, which PIAS hides away by also interpolating the μ linearly between 0% and 100% stages of flooding, so that also in this respect continuity is achieved at 0→1% and at 99→100%.
- If a damaged compartment and an adjacent compartment are connected by a threshold or something similar then that can prevent the connected compartment from flooding. This can be specified as a so-called critical

point — please refer for that to Specify intermediate stages and critical points (discussed on page 409). In intermediate stages of flooding, such a critical point behaves like a yes/no switch; at each angle of inclination, the final stage of flooding determines whether the critical point becomes overflowed, and if so, the connected compartment is filled in intermediate stages with the usual percentages, as if that whole critical point did not exist. And if the critical point does not overflow at 100% then the connected compartment is not filled at all in intermediate stages.

This scenario will not always represent reality, but it is the best fit for the dogma of the "fixed percentual intermediate stage of floodingt". Those who do not agree with this approach will have to choose a more adequate scenario, e.g. a calculation in the time domain.

Two more details are worth knowing about the calculation method:

- It was mentioned above that with each inclination angle, location is determined, as well as the righting moment. The righting lever — GZ or GN'sin(φ) — is obviously that moment divided by the displacement. For that displacement, PIAS has a choice, see section 5.4.11 on page 48, Righting levers denominator.
- As described, floating position GZ are determined at fixed inclination angles. The equilibrium angle is not calculated directly, it is determined by (nonlinear) interpolation on the GZ curve, viz. that angle at which GZ is zero.

21.4.3 Difference in principles at intermediate stages

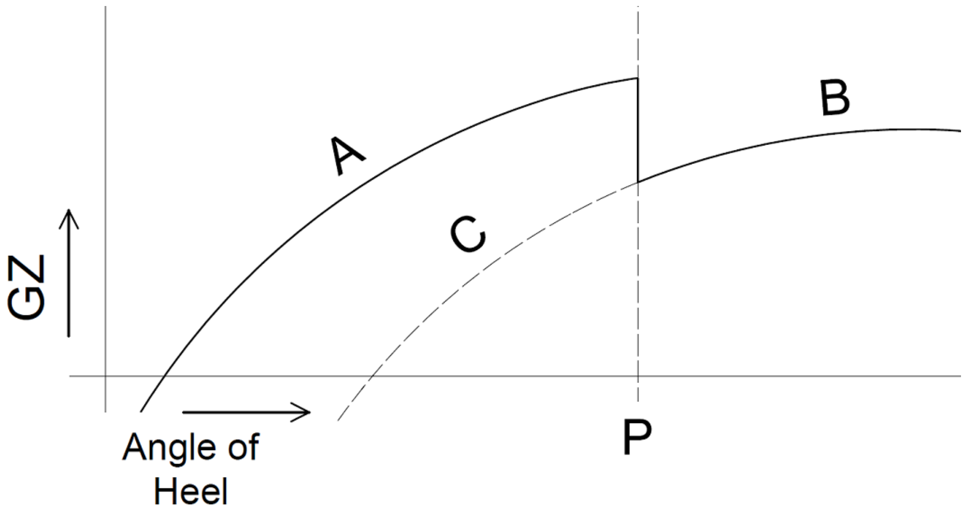
Both the elder (pre-2023) damage stability calculation system, and the *Consecutive Flooding* system contain calculations with intermediate stages of flooding. Perhaps one would expect their results to be the same, but that will by not always be the case. This is not so surprising, as both methods have their own calculation bases. The details of these have been discussed in previous sections, but in summary they boil down to this:

- The elder system is entirely based on the idea of **continuity**: at the final stage of flooding, for each compartment at each heeling angle is known how much seawater it contains, and in intact condition likewise, so that the water quantity at each intermediate stage can be interpolated between them. This has the advantage of consistency: after all, a 1% intermediate stage will be (almost) equal to the intact state, and a 99% stage to the final stage of flooding. Hence, the case 'grows' gradually and traceable from intact through the intermediate stages to the final stage of flooding.
- The *Consecutive Flooding* system is more based on **physical reasoning**, looking at the physically expected effect under the various scenarios. Here, the 99% stage need not be (nearly) the same as the final stage of flooding at all. Indeed, two 100% stages occur here, one being an intermediate stage and the other the final stage.

One might ask which method is best. In general, that question cannot be answered. After all, the first method has the advantage of elegance, adding that many thousands of such calculations have been approved by classification societies and Shipping Inspectorates over the past decades. While the second method has the advantage that diverse configurations with varying sizes of damages, openings, pipe lines, holes and connections rest on a certain physical foundation. Because *Consecutive Flooding* supports just such a large variety, it was necessary to switch to the second method for that.

21.5 Effect of internal openings on the GZ-curve

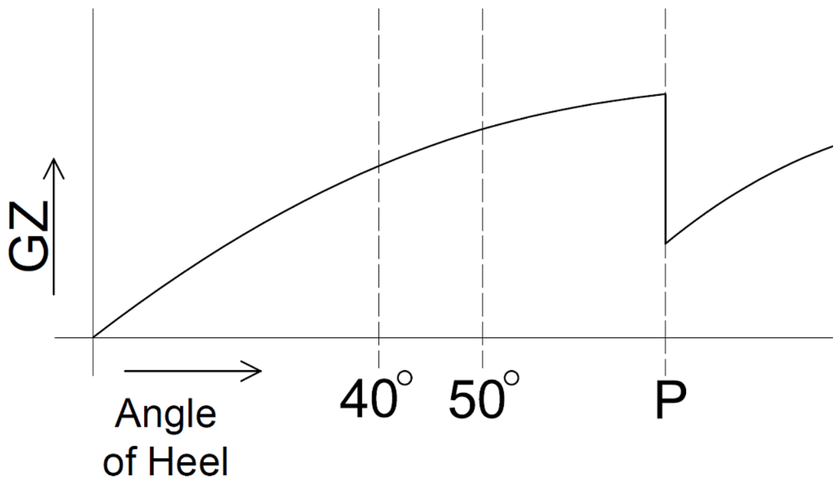
How does PIAS deal with an internal threshold or pipe which is submerged, and hence allows transfer of water, at an angle of heel which is beyond the static equilibrium? Take for example the GZ-curve as sketched below, where at angle P the upper edge of a partial bulkhead overflows, leading to the filling of an adjacent compartment and hence a deteriorated stability. It will be undebatable that the GZ will initially follow curve A, until angle P is reached, where a greater amount of ingressed water will lead to reduced curve B. However, the question is what happens on the "way back", i.e. with decreasing angle of heel? The water will not fully flow back over the bulkhead, so a curve more or less as indicated by C can be expected. And the subsequent question is which curve to use for the verification of GZ against stability criteria, A+B or C+B?



GZ-curve with internal opening submerged at angle P

In PIAS the past decades A+B has always been used — numerous calculations have been issued at classification societies and shipping inspections, and approved — based on the reasoning that the notion "way back" is never properly addressed, neither in literature nor in regulations. A few more arguments can be made in favour of this choice:

- The example above is expressive, but counter examples also exist. Take the GZ-curve as sketched below, with the partial bulkhead now immersed at an angle P which is much larger. If the vessel is subject to IMO's Intact Stability Code then the maximum heel for criteria evaluation is 50°— the IMO weather criterion — while angle P is much larger than 50°now. So, this loading condition meets all stability criteria long before P is reached, and a reduced C-branch will not be applicable.
- Will 50°then be the determining angle? In many cases not, because dynamic stability equality (area A=B from the weather criterion) may have been reached at a much smaller angle. So, the possible branching of the GZ-curve should be related to the applicable stability criteria, one way or another.
- Assume now that at the same large angle P not an internal opening spills over, but instead an external opening (e.g. a ventilation inlet), which sinks the ship. Then beyond P, the GZ curve will vanish, so also branch B. If one would argue that with an internal opening branch C should be taken, then the same reasoning should be applied to external ones. However, with branch B also C has vanished, so using this branch will render the whole GZ-curve non-existent. Nobody — user, researcher, authority nor classification society — has ever suggested such a 'solution', because it would be unrealistic.



GZ-curve with internal opening submerged at a large angle

Supported by these arguments, it was — in 2022, during the re-evaluation at the introduction of *consecutive flooding* — chosen to keep the computation method for this subject in PIAS as it always has been. Please understand that this is an implementation choice, not the irrevocable result of the modelling method in PIAS. So alternative choices could be made, if there would be a reason for that, such as a generally accepted convention. Other reasons could be clear and unambiguous guidance by rules or regulations or unified interpretations from institutions, such as IMO, IACS or national authorities.

Chapter 22

Probdam: probabilistic damage stability

With this module the damage stability can be computed on a probabilistic basis. Summarized briefly, the probabilistic method encompasses that, assuming the vessel is damaged, the probability that this damage is located in a certain area is determined, as well as the probability that damage in that area is survived. The product of these two represents the probability of survival in the event of damage in that area. By calculating these probabilities for many areas and adding them up, the total probability of survival is determined (at multiple drafts). This probability of survival must be greater than the minimum as imposed by the regulations.

22.1 The background of the probabilistic damage stability method

This PIAS manual is not the proper place for an in-depth explanation of the probabilistic damage stability and all its merits, for that purpose the space is lacking and the subject is simply too extensive. Furthermore, a program manual is not a tutorial. For the background we therefore refer to the reference list, which is included at the end of this chapter. Recommended is the nice book of Pawlowski [1], which gives a complete and thorough overview. The backgrounds of specific configurations and calculation methods are discussed in the papers [2], [3], [4] and [5], which themselves also refer to elder literature. And last but not least, the relevant regulations and their *explanatory notes* will have to be kept at hand.

You will neither find design advices here, nor recommended practices or configurations. This module must be regarded as a toolbox, from which the user can select his preferred tool. However, we do pursue that the properties of all choices and options are clearly explained, and that the calculation process itself is clear.

22.2 Introduction to the module

22.2.1 General

To calculate the probabilistic damage stability only a few characteristics of the intact vessel, some configuration parameters and the damage cases have to be defined (which can be generated automatically). A damage case consists essentially of a number of selected compartments which are damaged. These compartments have to be defined using module Layout. This module is capable of automatic determination of damage cases and damage boundaries, but the price that must be paid for this comfort is that the vessel has to be defined in compartments totally and uniquely. This means that every point in the vessel must be part of one compartment and may not be part of more than one compartment. Layout contains tools to assist in this process, for example the one discussed in section 9.10.3 on page 241, Difference between internal and external geometry.

22.2.2 External compartments

PIAS subcompartments are available in two flavors, of the type ‘bulkheads’, which are bounded by plane bulkheads or the side shell or bottom shell, and of the type ‘external’, which can have any arbitrary shape and for which we refer to the discussion in paragraph 9.5.1.3.7 on page 220, Shape definition external subcompartments on the definition method. For probabilistic damage stability subcompartments of both types can be used, however, for external subcompartments the following remarks apply:

- With subcompartments of the type ‘bulkhead’ the damage boundaries will always be determined by one of the four bulkhead locations. With ‘external subcompartments’ such single points can not be assumed a priori, so the whole shape of the external subcompartment has to be taken into account. As such, this will obviously not be a problem, but the effect is that much more points have to be considered, and that the processing time of the automatic determination of damage boundaries might increase considerably.

- A similar effect does occur with the generation of damage cases: with subcompartments based on bulkheads the program can assume that any of those bulkheads might possibly be a damage case boundary. with external subcompartments such assumptions do not apply, so internally the subcompartment is subdivided into may smaller portions, and it is investigated whether each of those portion boundaries is a damage case boundary. On the one hand this leads to increased processing times, while, on the other hand, it will never be certain that the amount of portions is sufficient to find each and every damage case boundary.

Which leads to the conclusions:

- External subcompartiments can be used, but as sparsely as possible.
- When using external subcompartments, be sure to verify the generated damage case thoroughly.

22.3 Main menu of the module

Probabilistic damage stability

1.	Calculation method, configurations and ship parameters
2.	Generation of damage cases
3.	Select and edit damage cases
4.	Remove (parts of) saved information
5.	Execute and/or print calculations
6.	File and backup management

This menu contains the main options, which will be discussed just below. Additionally, this menu contains details of the damage cases. Not only the number of cases claims is presented, but also whether there are useful results from previous calculations present, either the probabilities of damage (the *prv* values) or the probabilities of survival (the *s* values). These results may be reused later, so that any subsequent calculations could be performed significantly faster.

22.3.1 Calculation method, configurations and ship parameters

This is the central configuration part of this module, and contains the following sub options:

Calculation method, configurations and ship parameters

1.	Calculation method, configurations and ship parameters
2.	Drafts, trims and VCG's
3.	Define hopper stability particulars (incl. pouring in or out)
4.	View scheme of standard permeabilities
5.	Edit scheme of user-defined permeabilities
6.	Define compartment connections
7.	Define zonal boundaries
8.	Notes (free text)
9.	Determine the VCG' for which A=R

22.3.1.1 Calculation method, configurations and ship parameters

Attention

In this section many choices and options for probabilistic damage stability are being discussed. The number of options is large, options even exist which are not backed by one of the regulations. Nevertheless these options are kept, they might have a historic origin and, as such, required for elder projects of vessels. At some options question marks can even be put, but each one has its own background and if this field would be pruned not each variant would be available anymore. That would be user-unfriendly.

When this option is selected a setup window will appear. This screen has a variable layout, its content is dependent on the selected calculation method and applied regulations. In this window all kinds of choices and configurations can be made, with the aim to offer the user as many options as possible and (consequently) to pre-program as few as possible. Fortunately, this menu offers the function button <Default>. With this function the configuration is chosen according to the selected regulations, at least, in the opinion of SARC, in January 2022. Of course other institutions or persons may prefer or prescribe other choices, so the is advised to verify that you agree with the programs 'default' choices, which are:

Option	**SOLAS 1992 & IMO A.265**	**SOLAS 2009 & 2020**
TCG in intact condition	Upon no heel	Upon no heel
Reference point for penetration depth	Waterline	Waterline
Application penetration limitation (b1,b2)	Apply rule, except at damage to center line	Apply rule, except at damage to center line
Type of penetration limitation	b1,b2 < 2.min(b1,b2)	b_{mean} < 2.min(b1,b2)
'Mean' or 'Minimum' penetration	Mean	Mean
Penetration rule multicomparttment damages	Local	Local
Damage penetration over CL	No	Yes, except with calculation method 'numerical integration'
r outside brackets in product r x (p123-p12-p23+p2)	No	No
Probability of damage never negative	No	No
With intermediate stages of flooding	Yes	Passenger vessels: yes, cargo vessels: no
Generate including horizontal subdivision	SOLAS 1992: yes, IMO A.265: no	Yes

The exact background and intention of each options will be discussed in the paragraphs below.

22.3.1.1.1 Applied regulations

Here the user can select one of the following regulations:

SOLAS 2020
The regulations for cargo vessels over 80 m as well as passenger vessels, into force from January 1, 2020.

SOLAS 2009
The so-called harmonized regulations, for cargo vessels over 80 m as well as passenger vessels. Applicable from January 1, 2009 to January 1, 2020.

SPS 2008
The probabilistic ruleas according to the *Special Purpose Ship Code* from 2008.

SOLAS 1992
For cargo vessels with a length over 80 m. Has been replaced by SOLAS 2009

IMO res. A.265
The equivalent rules for passenger vessels, according to IMO res. A.265 of 1973. Also replaced by SOLAS 2009 or 2020

Reconstructed SOLAS 1992
This 'regulation' is not relevant in regular ship design, partly because it is only applicable to the calculation method 'numerical integration', The background of this pseudo-regulation is discussed in [3] and [5], and its existence originates from the inconsistent processing at combinations of transverse and longitudinal subdivision in SOLAS 1992. As a consequence of this anomaly the results from numerical integration may differ from those as obtained by direct application of the SOLAS rules. Because numerical compatibility has its practical side, we have derived a new probability function by means of *reverse engineering*, and this is the *reconstructed SOLAS 1992*. Summarized, 'SOLAS 1992' is theoretically in agreement with the regulations, but gives numerically deviant results, while the 'reconstructed SOLAS 1992' is theoretically nonsense, but

provides answers in line with a conventional calculation. By the way, this whole aspect does not come into play with SOLAS 2009, because the whole foundation of this method is much more solid, thanks to a common treatment of *p* and *r*.

Attention

This modules only takes the probabilistic damage stability aspect of these regulations into account. Other matters, such as possible additional deterministisc damage stability requirements, are not considered. Thus, it is necessary to verify whether there are any other than probabilistic demands, and to treat them separately.

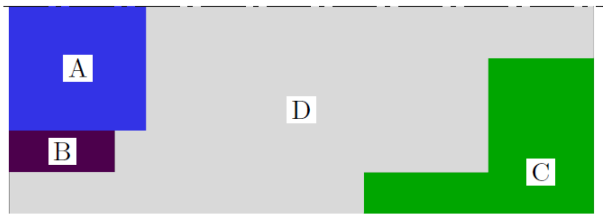
22.3.1.1.2 Ship type

Exclusively for the SOLAS 2009 and 2020 regulations here the choice can be made between 'cargo vessel' and 'passenger vessel'.

22.3.1.1.3 Calculation method probability of flooding

Here a choice can be made between four methods for the calculation of the probability of damage *p.r.v*: 'numerical integration', '1 damage per compartment', '1 damage per subcompartment' and '1 damage per zone'. The backgrounds of these methods are discussed in [3], [4] and [5], summarized briefly it touches a corner stone of the probabilistic method, which is the assignment of a probability of damage to each portion of the vessel. In principle it is indifferent which atomic (i.e. undividable) portion is taken, as long as the sum of all portions covers the whole vessel. In practice a number of choices for these portions has surfaced, enumerated from coarse to fine:

- A **zone**, whereas a zone is a portion of the vessel between two longitudinal boundaries (e.g. transverse bulkheads). he use of the zonal concept forces the subdivision model into regularity, thus avoiding certain pitfalls of a more refined subdivision. However, the zone-model is artificial; it is an abstraction of the actual subdivision, and as such will produce a less accurate result. It is funny to see that the zonal concept is rather popular, although it is not even mentioned in SOLAS 1992 (however, it is mentioned in the explanatory notes). In SOLAS 2009 the terms 'zone' and 'compartment' are entangled, however, the 'zone' is not defined at all.
- A **compartment**. This is the most obvious choice, for it corresponds to the actual subdivision.
- A **subcompartment**. A compartment as an atomic entity may not even be small enough. It might occur that there exists no single damage (with flat aft, forward and inner boundaries) which affects a compartment completely, ut that a finer subdivision is needed to cover the entire compartment. An example is shown in the figure underneath, where the assumption that each compartment is affected by a single damage does not hold for compartment 1. A further division of this compartment, for instance along the dotted line, will make it affected by two damages: B-C and D-E. More complex compartment shapes are by nature in PIAS composed of subcompartments, so they can readily be used as an atomic entity. Obviously, for the determination of the probability of survival the compartment is always flooded in its entirety.
- Tiny pieces of volume ("*voxels*"), which together fill the whole vessel, and which are independant of the subdivision as such. If the PDF s are not *a priori* integrated, the whole usage of crisp boundaries disappears. Consequently, there is no need for any atomic portion concept. If, as proposed in [3], the probability functions are integrated numerically in combination with the actual geometry of the compartment (or a group of compartments), then any compartment shape can be processed, including possible niches, irregularities and warped or even curved compartment boundaries. On this **numerical integration method** some additional remarks can be made:
  - The application of numerical integration in this context can be compared with the developments in the area of structural strength; initially analytically determined standard solutions for the deflection of beams were utilized, but for more complex structures the division into very small, *Finite Elements* proved to be more flexible.
  - The option to determine the probability of damage using numerical integration is not mentioned in S↩ OLAS, however, it is a recognized method for the calculation of outflow from oil tankers, see res. M↩ EPC.66(37) of September 14, 1995, and the "Revised interim guidelines for the approval of alternative methods of design and construction of oil tankers under Regulation 13F(5) of Annex I of MARPOL 73/78, MEPC.110(49)" from July 18, 2003.
  - [6] and [7] report on the same approach, at the University of Hamburg.
  - This method is also known as **Monte Carlo method**.
  - The combination of this method with the IMO res. A.265 regulations has not been implemented in PIAS.



A space where 'zone' and 'compartment' are too coarse

22.3.1.1.4 Damage at

Here SB or PS can be given. According to the *explanatory notes* of (at least) SOLAS 1992, if asymmetry is present in hullform or compartmentation, the calculation must be made to SB as well as to PS, where as attained subdivision index A both the lowest value and the average value of the two may be used. Apart from that, with Config it can be specified for damage stability calculations whether the inclination is to SB, to PS or is determined automatically, but this setting is **not** applicable for the probabilistic damage stability calculations.

22.3.1.1.5 Light ship draft or light service draft

Here the light ship draft (for SOLAS 1992 and IMO A.265) or the light service draft (for SOLAS 2009 and SOLAS 2020) must be given. These drafts are the same as can be given Hulldef, as discuseed in section 7.2.1.9 on page 170, Particulars for SOLAS chapter 2, part B1.

22.3.1.1.6 Subdivision draft

Here the subdivision draft must be given, with the same remark about SOLAS and Hulldef as in the previous paragraph.

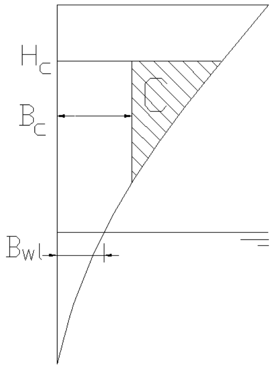
22.3.1.1.7 TCG in intact condition

Here one can choose between *Coincides with centerplane* and *Is determined upon no heel*. This switch is only relevant with vessels with an asymmetrical hull shape, With the first choice, TCG at centerplane, an initial list will occur, while with the second option the Transverse Center of Gravity is determined so that no list will occur.

22.3.1.1.8 Reference point for penetration depth

The penetration depth b is the penetration (in meters) up to the inner damage boundary. However, if this distance is measured from the waterline, and the inner boundary is located beyond the local waterline breadth, a logical inconsistency pops up, see the cross section figure below. With damage to compartment C only, the matter is that Bc $>$ Bwl, so the penetration = Bwl - Bc $<$ 0, and consequently the probability of damage p will be zero. And that is the problem, because we have a real damage case at hand, with a real penetration through real steel, which is not included arithmetically as a damage case in the context of probabilistic damage stability. For this reason this option offers the choice between two alternatives:

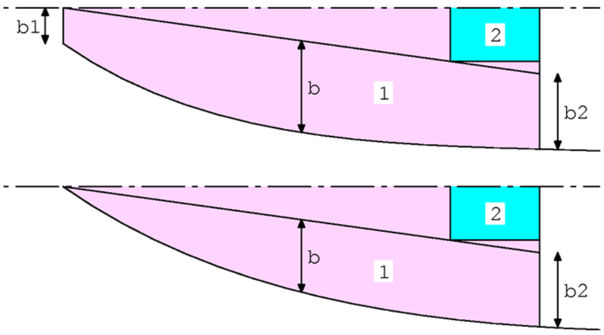
- Waterline, where the penetration is determined from the waterline at deepest subdivision draft.
- Upper boundary of damage case, where the penetration is determined at the level of the upper boundary of the damage case under consideration, in our figure height Hc in our figure. In that case a realistic probability will be assigned to the damage case of compartment C. According to the table above, the default for this options is 'waterline', however, between the end of 2009 and the first week of January 2011 it was configured as 'upper boundary of damage case'.



Penetration depth in cross section

22.3.1.1.9 Application penetration constraint (b1,b2)

According to the *explanatory notes* of SOLAS 1992 the penetration at side compartments is limited by the rule that the maximum penetration shall not exceed twice the minimum penetration. This constraint can limit the penetration depth very severely, see e.g.the figure below, where only compartment 1 is damaged. The evident penetration is according to the angled line, with the penetration depth indicated by *b*. However, in this case the minimum penetration, *b*1, is zero, so the maximum penetration *b*2 is always greater than twice the minimum penetration, violating the penetration constraint rule. The only solution to comply with this rule is to set b2 also to zero, which results in (an unrealistic small) penetration depth *b* as depicted in the second figure. With a hollow waterline *b* would even become negative, which leads to a probability of damage of zero for this rather realistic damage case.



Penetration limitation rule with b1=0

At this option 'Application penetration constraint (b1,b2)' it can be specified whether, and how, this rule is applied. The options are:

- Without this penetration constraint rule.
- With this rule, except for damages which extend to CL.
- With this rule, except for damages with an inner boundary // CL (where '//' denotes 'parallel to').
- With this rule, also for damages to CL (so the rule is always applied).

22.3.1.1.10 Type of penetration constraint

As mentioned, the penetration constraint rule of SOLAS 1992 reads that the maximum penetration may not be larger than twice the minimum. For SOLAS 2009 & 2020 the rule is slightly different, it reads that the *mean* penetration may not be larger than twice the minimum. Therefore the program offers a choice of two alternatives, where the SOLAS 1992 version is formulated as 'b1,b2 < 2.min(b1,b2)', and the 2009 & 2020 one 'bmean < 2.min(b1,b2)'.

22.3.1.1.11 At exceeding of the (b1,b2) constraint

Here it can be specified (onl;y with the zonal method) what the program should do when the (b1,b2) penetration constraint is exceeded:

- Warning only, which implies that the dimensions of the specified zonal boundaries are still used, and that the program specifies on the output in which damage cases the (b1,b2) rule is violated. It is up to the user to adapte the zonal boundaries for these cases.
- Let the program adapt the penetration automatically to the (b1,b2) constraint rule.

22.3.1.1.12 'Mean' or 'Minimum' penetration

This choice is related to the determination of penetration breadth b (see e.g. SOLAS 1992 reg. 25-5.2.2). Three choices are possible here:

- According to (at least) the Dutch Authorities it can be interpreted as 'b = the *minimum* transverse distance...'. See also the report SLF 34/WP.11, item 5.3.10, and the paper 'IMO Circular letter 1338: Interpretations by the Netherlands Administration' (section 22.7.1 on page 435, Appendix 1: mean/minimum penetration, IMO circular letter 1338, in Dutch).
- Literally 'b = the *mean* transverse distance...', an interpretation which is likely to be used by many authorities.
- In 1991 we have propounded the question how to handle in case b1=0 or b2=0 to the Dutch authorities (see section 22.7.2 on page 436, Appendix 2: mean/minimum penetration, the question and section 22.7.3 on page 437, Appendix 3: mean/minimum penetration, the reply by NSI, in Dutch). The answer was that in such a case always *minimum* penetration depth may be used. That leads to the (for the ship designer) most favorable situation, because in cases where b1 as well as b2 are greater than zero the *mean* depth can be used (which is in general the largest), while in case of b1=0 or b2=0 one can switch to minimum, so there is no penalty of a small penetration depth. This choice is incorporated in PIAS, and is called 'Largest of mean and minimum' (This 'Largest of mean and minimum' rule has not been implemented. The same effect can be obtained with a is not applied on damages with an inner boundary which is parallel to centerline.).

The choice between the three alternatives is left to the user, we can only mention to have observed that the *minimum* interpretation requires the least computation time.

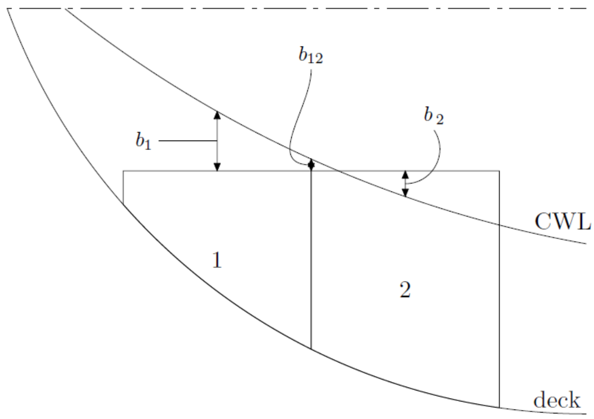
22.3.1.1.13 Determine r at multi-compartment damages

This option determines the way how with a multi-compartment damage with longitudinal subdivision the reduction factor *r* is determined. The first possibility is *local dimensionless penetration b/B*, where the *r* of each to-be-subtracted damage case is determined with the individual b/B as measured halfway the waterline of that case. The other choice has the effect that the *r* of each damage case (so the main damage as well as the damages to be subtracted) is determined on basis of a common dimensionless penetration b/B. The exact implementation depends on the selected calculation method:

- With the (sub-)compartment methods cases the *global dimensionless penetration b/B* is applied, which is determined halfway the main damage (In paragraph 3.5 of [2] it is explained why this is the only viable option for the general case).
- With the zonal method the *minimum dimensionless penetration b/B* is applied, where each reduction factor *r* is determined with the b/B which is the minimum of all involved b/B's (so the b/B from the main damage as well as from the damages to be subtracted). The explanatory notes suggest this approach in fig. a-3 (IMO res. A.684 for SOLAS 1992) respectively paragraph 1.2 (SLF 49/17 for SOLAS 2009). Because with the zonal method the subdivison is regular, this method is in this particular case feasible. SARC has no preference for one method over another. By the way, until November 2001 PIAS had no external mechanism to select this calculation option. Before that date always *local b/B* was used. From the viewpoint of logic and consistency the use of *local* penetration can be favored above *global or minimum*, because with *local* penetration the probabilities of the damages to be subtracted are always equal to the probability with which they have been added. With *global or minimum* this is not the case, which might be confusing. Another aspect is illustrated in the figure alongside. With *local* penetration is $p12 = p12.r12 - p1.r1 - p2.r2$, with $r12$ determined on the basis of $b12$, $r1$ on the basis of $b1$ and $r2$ on the basis of $b2$. In the example $b1<0$, so $r1=0$, $b12<0$, so $r12=0$ and $b2>0$, so $r2>0$. Consequently $p12$ becomes negative, regardless the nature of the formula of r as a function of b. With *global or minimum* penetration $r1$, $r2$ and $r12$ are all determined on a common basis (With global dimensionless penetration on basis of b12, and with minimum dimensionless penetration on basis of the minimum of b1, b2 and b12), so a negative probability cannot occur (at least not due to an anomaly in to the processing of to-be-subtracted damage cases).

Attention

> This whole subject with mean, minimum, penetration rule multi-compartment damages and b1/b2 penetration rule does not play a role, cannot even play a role, with the numerical integration method. With that, by the way, no negative probabilities occur.



22.3.1.1.14 Damage penetration over CL

In 1992 SOLAS and IMO A.265 the penetration of side damage was limited to centerline. In SOLAS 2009 the penetration is B/2, without further addition. That implies that with SOLAS 2009 in the regions of the narrowing waterline, in foreship and aftship, the damage will extend beyond centerline. With this option one can choose between these two variants. This option is, for now, not available for the 'numerical integration' method. It is not because this would be infeasible, on the contrary, but the combination has never been required.

22.3.1.1.15 r outside brackets in product r x (p123-p12-p23+p2)

Here is can be specified how to process a combined longitudinal and transverse subdivision. If *r* is taken outside brackets than the equation for a three-compartment damage reads *pr* = *r*123.(*p*123 - *p*12 - *p*23 + *p*2), if *r* is not taken outside brackets *pr* = *r*123.*p*123 - *r*12.*p*12 - *r*23.*p*23 + *r*2.*p*2 is used.

22.3.1.1.16 Probability of damage never negative

As illustrated by the text and the figure just above, the formulae structure as laid down in the legislation can give rise to the occurrence of negative probabilities of damage. Some people accept this as the mere outcome of the SOLAS system, while others get restless at this, because it is obviously mathematical nonsense. For the latter this setting is available; if one enters 'yes' here, then the probability of damage is maximised to zero (i.e. if the probability is less than 0 then 0 is taken). With 'no', for the collision probability the SOLAS formulas are simply followed, without special treatment of negative probabilities.

Please see that the choice 'yes' is not supported by SOLAS. This provision is, among other reasons, included in PIAS because reportedly other computer programs exist where default (and invisible) negative probabilities are raised to zero, so that they are hidden away. With this setting, PIAS can do the same.

22.3.1.1.17 Including intermediate stages of flooding

With 'no' the damage stability calculations are only performed for the final stage of flooding, with 'yes' also the intermediate stages 25%, 50% and 75% are computed.

22.3.1.1.18 Combine damage case generation with the calculation

Besides for the computation of probabilities of damage, the numerical integration method can rather well be used for the generation of damage, so a distinct generation step can be considered a bit redundant. With 'yes' at this option it is specified that the generation of damage cases and the execution of the calculation must be combined, with 'no' they are separated.

22.3.1.1.19 Maximum damage length for damage case generation

So that no damages will be generated larger than this value. At SARC, in general we see little reason to deviate from the default value of hald the ship length.

22.3.1.1.20 Maximum number of damaged (sub-)compartments per damage case

With Z given here, no damage cases will be generated with more than Z simultaneously damaged compartments (or subcompartments, in case of the subcompartment method). This parameter should be chosen with care; a too small value will result in a limited number of damage cases, and hence a low *attained subdivision index*. A high number will produce the maximum **A**, but that may require a high number of damage cases with a corresponding long computation time. A scrisp number cannot be recommended, that depends on the nature of the subdivision and on the number of compartments. At SARC we tend to start at ten, and may adapt this parameter dependant from the computation result.

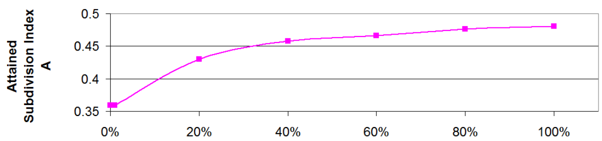
By the way, the absolute maximum number of damage cases for this module is 5000, that has nothing to do with this parameter.

22.3.1.1.21 Maximum number of damaged zones per damage case

When using the zonal method the program will generate all combinations of adjacent zones. This could lead to large amounts of damages, from which the majority of multi-compartment damages could preponderantly not contribute to A. In order to limit the number of damage cases, with this option the maximum number of zones per damage can be specified.

22.3.1.1.22 Accuracy index numerical integration (0-100)

With the numerical integration method the integration step size plays a role; it may be clear that the accuracy increases with a decreasing step size. The real step size is determined by means of an algorithm, on the basis of the number of compartments etc., and it is not considered relevant to confront the user with it. For that reason the accuracy is defined by means of an index, where 0 indicates the least accuracy and 100 the greatest. One might be inclined towards the greatest possible accuracy, but that of course also induces a longer processing time. The exactly required accuracy cannot be indicated in general. Apart from your patience it does also depend on the vessel's layout. In order to give an impression one can imagine two extremes, one extreme is a barge with completely rectangular subdivision. Because the program applies an integration step at each (sub-)compartment boundary, all spaces are already taken into account completely, so the result will (approximately) be independent from the accuracy index. On the other hand, in case of a vessel with angled bulkheads the number of integration steps does play a role, for illustration purposes we have created a barge with only a single longitudinal bulkhead, which is extremely twisted. The required subdivision index *A* as a function of the accuracy index is plotted in the figure. It will be obvious that in reality the quantitative effect of this index will be somewhere between these extremes.



Accuracy index

22.3.1.1.23 Generate including horizontal subdivision

With 'yes' the damage cases will be generated including horizontal subdivisions (decks) between the compartments. With 'no' the horizontal subdivisions will be ignored, so the damages will extend from baseline to the uppermost deck, leading to less damage cases.

22.3.1.1.24 Damage cases generation including "progressive flooding"

If compartment connections have been defined (see section 22.3.1.6 on page 427, Define compartment connections for more details), then with this option it can be specified that at the generation of damage cases the program will model the flooding of compartments through pipes by means of complex intermediate stages of flooding. This option is only relevant for the (pre 2021) method of complex stages, please refer for that method to section 21.3 on page 408, Complex stages of flooding (before 2023).

22.3.1.1.25 Intermediate results in text file incl. all integration steps

For verification purposes the contributions of all numerical integration steps are also included in the text file with intermediate results (When PIAS' multithreading function is active, multiple integration cycles are executed concurrently. A side effect is that these intermediate results are not included in the textfile in a synchronized fashion. If synchronicity is required, multithreading should be switched off, which can be done with the external variable *no_multithreading*, as discussed in Fairway). With a large accuracy index the number of integration steps can be quite large, and the text file might consequently rise to enormous proportions. To avoid that, it can be specified at this option to exclude the integration steps from the text file.

22.3.1.1.26 Store intermediate results in spreadsheet file

The text file, as discussed in the previous option, is intended for human interpretation. For an analysis of the computational results it might be handy to have the figures available in a spreadsheet. If that is specified at this option, the results will be written in a .CSV (Comma Separated Values) file, such a file can be read with most spreadsheet programs. However, such a spreadsheet file can only be generated if the configuration options listed below (Re-use unmodified results of former calculations) is set 'off'. The reason is that all results actually must have been calculated before they can be included in the spreadsheet file.

22.3.1.1.27 Re-use unmodified results of former calculations

In order to increase the computation speed this module keeps quite some results of former calculations in memory. These can be re-used in subsequent calculations. For example, for each damage case the KNsin(φ) are stored, so that in case of future modifications of VCG' the GZ in damaged condition can be rapidly determined. In general it is recommended to use this facility, but if it is not desired for whatever reason it can be switched off at this option.

- Nb 1. The results of former calculations can also be removed explicitly, see section 22.3.4.1 on page 431, Remove all results of former calculations.
- Nb 2. The re-usage of existing results is a completely module-internal matter. It has nothing to do with the intermediate calculation data (see section 22.3.5.4 on page 433, Show intermediate calculation data which is intended solely for human interpretation.
- Nb 3. When this option is used and PIAS uses the results of former calculations, this will be mentioned in the intermediate calculation data as statement instead of the full data for the corresponding damage case(s). When you require complete intermediate calculation data, this option should not be used or results should be removed explicitly as described above.

22.3.1.1.28 Orientation damage case plots

Damage cases can be plotted. With the option under consideration it can be specified if these plots are requested in portrait or in landscape format.

22.3.1.1.29 Wind pressure for calculation of heeling moment (kg/m2), passenger moment, life boat moment & selected wind contour

Here the data for the various components of the heeling moment can be given, which are required for the calculation of passenger vessels.

22.3.1.2 Drafts, trims and VCG's

In section 22.3.1.1 on page 418, Calculation method, configurations and ship parameters the subdivision draft and light draft have already been specified, with which the calculation drafts are fixed. In the present menu for each of those calculation drafts the following particulars can be entered:

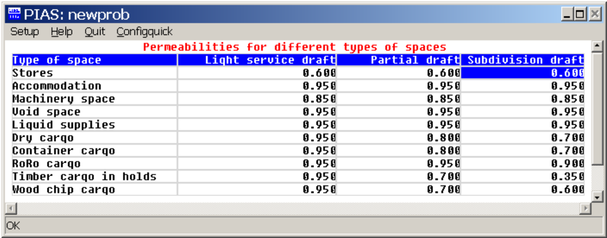
- The trim.
- The VCG' / MG' combination. Enter the VCG', and the MG' is automatically adapted, and the other way round.
- Whether this is the draft on which the VCG' must automatically be determined, in order for the attained subdivision index to become equal to the required subdivision index: A = R (see section 22.3.1.9 on page 429, Determine the VCG' for which A=R).

22.3.1.3 Define hopper stability particulars (incl. pouring in or out)

This module is also capable to calculate the probabilistic damage stability according to the regulations 'Agreement for the construction and operation of dredgers assigned reduced freeboards', a.k.a. dr-68. See for all details of hopper stability section 18.6.3 on page 388, Probabilistic damage stablity.

22.3.1.4 View scheme of standard permeabilities

According to SOLAS 2009 & 2020 the several kinds of space types must be computed with different permeabilities (μ). The μ's which belong to each type of space can be inspected with this menu option. With module Layout the correct space type will have to be assigned to each compartment, where one should verify that all subcompartments of such a compartment have their 'Autopermeability' set to 'yes', otherwise the static μ will still be used.



Permeabilities for different types of spaces

Type of space	Light service draft	Partial draft	Subdivision draft
Stores	0.600	0.600	0.600
Accommodation	0.950	0.950	0.950
Machinery space	0.850	0.850	0.850
Void space	0.950	0.950	0.950
Liquid supplies	0.950	0.950	0.950
Dry cargo	0.950	0.800	0.700
Container cargo	0.950	0.800	0.700
RoRo cargo	0.950	0.950	0.900
Timber cargo in holds	0.950	0.700	0.350
Wood chip cargo	0.950	0.700	0.600

Permeabilities for different types of spaces

22.3.1.5 Edit scheme of user-defined permeabilities

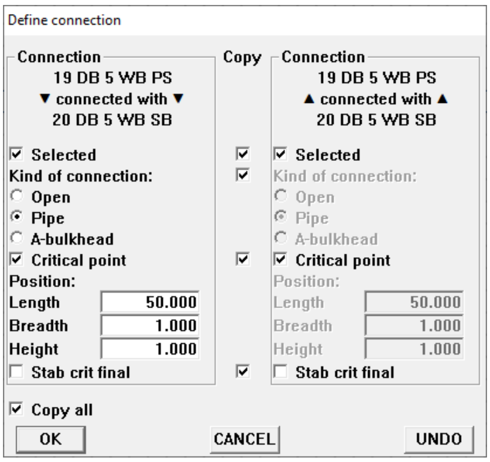
The permeability list of section 22.3.1.4 on the previous page, View scheme of standard permeabilities covers those types of spaces which are explicitly defined in SOLAS 2009 & 2020. Possible additional categories can be defined with this option.

22.3.1.6 Define compartment connections

This function co-operates with PIAS' system for complex intermediate stages, the development of which was frozen in 2021 because a more capable system called *Consecutive flooding* was released by then. When using that, this specific Probdam function is no longer needed, so it will be inactive then. For more information, see section 21.1 on page 402, Background from tools for ship-internal connections in PIAS.

In this menu for each compartment can be specified which compartments are flooded in the case of damage to the compartment under consideration. In particular one should consider the flooding via damaged pipes. This menu consists of a matrix of connections, between all compartments, where in the first column all compartments are linked to a number that corresponds to the numbers in the header line. With these numbers it is clear which two compartments are connected. The connection between the two compartments can also be found textually in the status bar.

In almost every cell, two connections can be specified for two compartments by using the 'Connections popup'. All settings in the popup are explained in more detail below, using the compartment connections menu. In addition to the popup, an 'Open', 'Pipe' or 'A-bulkhead' connection can be made between the two compartments using the <O>, <P> and <A> keys respectively.



Connections popup

Double-clicking on a compartment name — in the second column — opens a menu where all connections to and from the selected compartment can be defined. For each connection it must be specified whether it exists by means of 'Selected'. If such a connection exists, the type of connection must be defined, which can be one of the following types:

- [Open], means a connection with a large cross-section, through which the seawater can pass freely, also in intermediate stages of flooding.

- [Pipe], means a connection with a small cross-section through which the seawater cannot pass freely in intermediate stages. When a connection is of the [Pipe] type, a complex intermediate stage of flooding will be generated (if this option is purchased), in which the compartment connected by means of a pipe will not be flooded.
- [A-bulkhead], is treated exactly the same as a connection of the type [Pipe] in the damage stability calculations.

Additionally, it can be specified whether the connection always exists, or only after the water level at a certain point is exceeded. The latter can e.g. occur with a bulkhead which does not extend over the full height. In this case the column 'Cr.pnt' — an abbreviation of Critical Point — should be marked 'yes', while at Lcrit, Bcrit and Hcrit the longitudinal, transverse and vertical coordinates of that point must be specified. In addition to the 'Critical Point', one can also specify whether the connection should be checked against the criteria for the intermediate or final stage of flooding. Normally this setting is *No*, but in some situations it may be desirable or mandatory to test against the criteria for the final stage. Please note that, if the regulations have not specified criteria for intermediate stages - eg for cargo vessels under SOLAS 2009 - the criteria for the final stage will be used.

In this menu there is also a 'Copy' column with which the entire connection between the compartments to be connected can be copied. Normally the following applies to a connection; if one compartment, say compartment A, is connected with another compartment, say compartment B, it does not imply that B is automatically connected with A. In reality that does also not necessarily has to be the case, e.g. when a pipe is fitted in compartment A which will flood B if damaged. That mechanism does not apply reversely, so at damage of compartment A, B will also be flooded, but at damage of compartment B, A remains unaffected. If there exists a permanent opening between A and B, one should *explicitly* specify that A is flooded in case of damage to B. This *explicit* task can be taken care of by setting the copy to 'Yes'. It is worth noting that in the connection popup, from the matrix menu, copying can be done in a more selective way.

The values specified here are solely used at the ***generation*** *of the damage cases*, in other words, these data are not used independently, but are processed in each individual damage case. So modified data will only be processed after re-generation of damage cases.

Two more remarks can be made on this option:

- The generated data can be inspected at the damage cases (section 22.3.3 on page 430, Select and edit damage cases) after choosing the function <Flooding stages>. With the second menu option a list appears which shows which compartments are connected with a damaged compartment from that particular damage case, and what is the position of a possible critical point. This specification can also be edited, see for more details the section on complex stages of flooding, section 21.3 on page 408, Complex stages of flooding (before 2023).
- If one defines the following two pipe connections, A to B and B to C, then in the event of direct damage to A, compartment B will be flooded, but as a result of the flooding of B, compartment C will also be flooded. This situation ultimately translates into two complex intermediate stages of flooding, whereby compartment B is 100% filled after the first stage and compartment C is 100% filled after the second stage. This is shown below in a short table, in which the percentages of flooding are printed under compartments A, B and C. The **Final** stage is printed purely for completeness.

Stage	A	B	C
1	100	0	0
2	100	100	0
Final	100	100	100

22.3.1.7 Define zonal boundaries

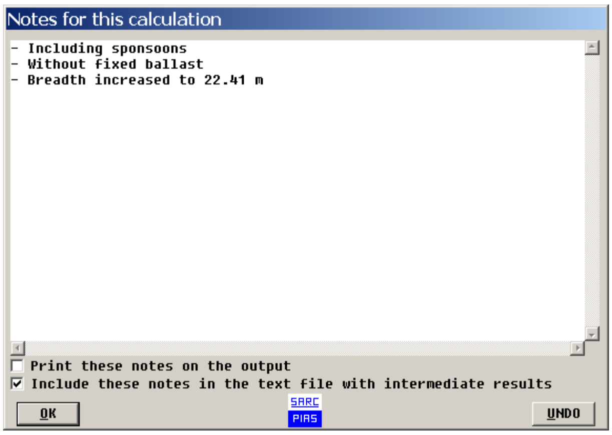
The compartments defined in PIAS are the real enclosed spaces of the ship. However, in a vessel on a less detailed level often 'primary spaces' or 'zones' can be identified, where multiple compartments can be situated in such a zone. An example of a zone is the engine room, regardless the exact layout of the many consumable compartments. So a zone boundary is an abstract notion, it does not always have to coincide with a physical bulkhead. Zonal boundaries can be specified for two purposes:

- For a calculation with the zonal method. For this purpose the user will have to define transverse zonal boundaries, and possibly also longitudinal and horizontal boundaries. At the aft and forward extremities (more precise, at *aft terminal* and *forward terminal*) obviously zonal boundaries should be placed. If these are not present the program will add them at the aftmost and foremost frame of the hull.

- With the (sub-)compartment method the damage cases can be collected into zones with section 22.3.2.4 on the next page, Collect damage cases into damage zones. Other software for the calculation of probabilistic damage stability may work on a zonal basis, and for comparison purposes it might prove handy if the output of PIAS is also sorted in zones. (see also section 22.3.5.3 on page 432, Print calculation results, subtotalized by zone).

22.3.1.8 Notes (free text)

With this option a window appears where free notes about the calculation or the configuration can be written. These notes are saved with the calculation. It can also be specified whether these notes must be printed on the output, or in the text file with intermediate results.



Notes

22.3.1.9 Determine the VCG' for which A=R

Except from the subdivision, the attained subdivision index A is also dependant on the vertical center of gravity VCG'. In general the index will rise with decreasing VCG', so there will be a single VCG' for which A is exactly R. With this option this critical VCG' is determined. One aspect is that multiple drafts with belonging VCG's are in play, and that no general rule can be postulated for the distribution of the VCG' variation over the drafts. Therefore, at section 22.3.1.2 on page 426, Drafts, trims and VCG's the user has to assign the single draft for which the critical VCG' is determined.

22.3.2 Generation of damage cases

22.3.2.1 Generate ALL possible NEW damage cases

With this option all existing damage cases will be deleted and all possible (up to a maximum of 5000) damage cases will be generated. This might, notably with a large number of damage cases, take quite some time, which significantly be reduced with the use of multithreading see section 3.11 on page 32, Speed enhancing mechanisms in PIAS: PIAS/ES

The name by which each generated damage case is identified depends on the setting of 'Identification of the compartment in the tank sketches' as discussed at section 5.3 on page 45, Settings for compartments and tank sounding tables :

- Automatic tank number : The name will be 'N.M' where N is the number of simultaneously flooded compartments and M is the sequence number in the series of damage cases with the same number of flooded compartments.
- Name/second name/abbreviation compartment : The name consists of the selected identification of all flooded compartments. Within section 22.3.1.1 on page 418, Calculation method, configurations and ship parameters the maximum damage length and maximum number of damaged compartments can be given, in order to keep the amount of damage cases within reasonable limits. It is recommended to start the calculations with a short damage length and a modest number of simultaneously flooded compartments. Only in cases when the vessel does not comply with the required subdivision index R the damage length and a number of compartments can be increased.

With the (sub-)compartment method the progress will be shown in the section 3.11.6 on page 33, Multithreading task monitor. Damage cases are generated from the forward- to the aft side of the vessel and per thread it is shown from which forward boundary damages are generated.

With the (sub-)compartment method the damage cases as generated with this option are only based on the (sub-)compartment geometry. So the initial damage cases are determined by the extreme compartment boundaries. During the computation of the damage stability calculation the true damage boundaries will be determined, taking into account all boundary conditions. For this reason it might occur that a generated damage case after all is not able to exist, within all statutory limitations. Especially the penetration constraint rule b1/b2 plays a role to this effect. If a damage case does not exist, the warning ‘Damage impossible’ is included in the .PD0 file with intermediate results, please also refer to section 22.5 on page 433, Warnings.

22.3.2.2 Generate additional damage cases

With the previous option all existing damage cases will be removed and new damage cases are generated. The present option also generates damages cases, but these will be added to the existing damage cases. Four selection criteria must be specified here:

- Aft boundary of the area within which the damage cases have to be generated.
- Fore boundary of the area within which the damage cases have to be generated.
- Minimum number of compartments per damage case.
- Maximum number of damaged compartments per damage case. The criteria ‘maximum damage length’ and ‘maximum number of flooded compartments’ are *not* applicable here.

22.3.2.3 Generate high sub damages as complex stages of flooding

With this option complex intermediate stages of flooding were generated, where compartments **below** a certain, user-specified,height get a zero percentage of flooding. This option could be used to simulate minor damages. After December 2021 the high sub damages are generated and calculated automatically, if the setting at the input of damage cases is selected.

22.3.2.4 Collect damage cases into damage zones

With this option, which sorts and collects damage cases, is only relevant for the (sub-)compartment method, it is not related to the zonal method as distinct calculation method for the probability of damage. This sorting is based on the zonal boundaries as specified in section 22.3.1.7 on page 428, Define zonal boundaries. After sorting, those damage cases which fall into the same zone are collected, while subtotals for all zones and combinations of zones are printed. One must recognize the fact that the calculation process of PIAS does not change when using zones, it only affects the presentation of the output. While sorting, the damage cases can be renamed if you have chosen to do so. In that case the name of a damage case is composed in the form of K.L.M.N, where:

- K: A number which indicates the number of zones in which the damage case extends.
- L: The zone number (at a single-zone damage), or the first-last zone number combination (at a multi-zone damage). The first zone number is always zero.
- M: A number to indicate the number of PIAS compartments which are flooded in the damage case.
- N: The sequence number of this damage case.

22.3.3 Select and edit damage cases

With the (sub-)compartment method the user has to specify all damage cases (where a damage case is a collection of one or more damaged compartments) which can occur within the vessel. That can be done with the present option, which is discussed in general at section 20.3 on page 397, Input and edit damage cases. In addition, here at the probabilistic damage stability a number of additional columns are present in the text window:

- Aft: Aft boundary of the damage.
- Fore: Forward boundary of the damage.
- Upper: Upper boundary of the damage.
- Ainside: Inside boundary of the damage, at the aft boundary.

- Finside: Inside boundary of the damage, at the forward boundary. The columns 'Aft' up to 'Finside' can only be edited by the user with a user-defined (not generated) damage case. The values presented in the columns 'Aft' up to 'Finside' can only be precise for each damage case which has been fully calculated, otherwise approximate values are displayed.

If a new calculation is performed, the former exact determined values can sometimes change a bit. The listed values are used as the initial values for the new calculation of damage boundaries, and if the initial values for two calculations differ to some extent, the final boundaries may also slightly differ, please also refer to section 22.4 on page 433, Computing the damage boundaries.

Because the probability of survival is composed of the sum of the probabilities for all damage cases, all cases normally have to be included, so all 'Slct' values should be set to 'yes'. In certain (investigation-)cases it could be handy to leave cases temporarily out of the calculation, in which case those cases can be set to 'no'.

If the text cursor is placed over a damage case and <Enter> is pressed, a window appears where can be specified which compartments (as defined in Layout) are simultaneously flooded in the damage case under consideration. this is also described at section 20.3 on page 397, Input and edit damage cases. In the context of the probabilistic damage stability an additional column is present, 'valid for pi, vi and ri', which is normally to be set at 'yes', which means the flooded compartment is taken into account for the determination of the probability of damage, p_i, v_i en r_i. If this column is set to 'no', this compartment is not included in the probability of damage, however, it will obviously still be included as a flooded compartment. This might be the case when a compartment is not located in the damaged region but yet flooded anyway, for instance if it is connected to a damaged compartment by means of a pipe.

22.3.4 Remove (parts of) saved information

22.3.4.1 Remove all results of former calculations

As mentioned, in the module paragraph 22.3.1.1.27 on page 426, Re-use unmodified results of former calculations a mechanism is incorporated with which the results of unmodified formerly calculated damage cases are maintained. With this mechanism the processing time at repeating calculations with slight variations can strongly be reduced, because only those damage cases which have actually been modified need to be re-calculated fully. This module automatically keeps books of compartments, damage cases and intact particulars. A prerequisite for a correct bookkeeping is that the computer clock functions well. With the present option the user can remove all these accumulated results.

22.3.4.2 Remove all complex intermediate stages of flooding

With this option all complex intermediate stages of flooding will be removed (see section 21.3 on page 408, Complex stages of flooding (before 2023) for the 'complex stages' mechanism).

22.3.4.3 Remove all damage cases with a non-positive probability of damage

Damage cases with a negative probability of damage will decrease the attained subdivision index A. With this option they can be removed. Please realize that as a consequence of this action the whole damage case constellation will change, and, with the (sub-)compartment method, consequently the whole damage case subtraction scheme, which might on its turn cause other damage cases to get negative probabilities.

22.3.4.4 Remove all damage cases which do not contribute to "A"

Damage cases with a non-positive a_i are removed. It is better not to use this option when in a later stage the VCG' could be decreased. Damage cases with zero probability of survival will be deleted with this option, and it might very well be that with a lower VCG' some of those damage cases would have a positive survival probability.

22.3.4.5 Remove all damage cases

With this option all existing damage cases are simply removed.

22.3.5 Execute and/or print calculations

22.3.5.1 Execute and print the calculation

With this option the computation is executed, and printed. This might, notably with a large number of damage cases, take quite some time, which significantly be reduced with the use of multithreading, see section 3.11 on page 32, Speed enhancing mechanisms in PIAS: PIAS/ES.

The progress will be shown in the section 3.11.6 on page 33, Multithreading task monitor. Each thread shows the damage case it is calculating with the start-, and elapsed time and the status of the thread.

An example of the output of a dry cargo vessel is given in section 22.7.4 on page 438, Appendix 4: output of an probabilistisc damage stability calculation for a dry cargo vessel, and that of an open hopper vessel (according to regulation dr-68) in section 22.7.5 on page 439, Appendix 5: output of an probabilistisc damage stability calculation for a hopper dredger.

In this output p_i is the probability of damage, so including the effect of reduction factor r (which accounts for transverse penetration) as well as reduction factor v (which accounts for horizontal watertight subdivision above the waterline). Probability of survival s_i, as included in the output, is what the name implies; the pure survival probability, based on properties of the residual GZ-curve. This is a slightly different grouping than applied in the regulations — where $p \times r$ is baptized "probability of flooding" and $s \times v$ "probability of survival" — however, that is a bit of an awkward formulation because it stirrs up the two types of probability. Fortunately, for the final results this grouping issue is irrelevant, because the contribution a_i to the Attaineed Subdivision Index A will be $p \times r \times v \times s$ in all cases.

As mentioned earlier, in order to enhance execution speed as much as possible, the program keeps track of results of former calculations. Under the condition that the underlying ship data are not changed, these results can be re-used, entirely or partially, at subsequent computations. This mechanism can be the reason that where an initial calculation can take hours to complete, repetitive and identical calculations can be finished within minutes.

Furthermore, please observe that at the end of the table with damage cases, on the output, the sum of all probabilities of damage p is listed. This sum, Σp, represents the 'total probability of damage when damaged' and gives an indication whether the selected damage cases are correct and complete. There are three possibilities:

- Σp is exactly 1. In this case all possible damage cases are well covered by the selected damage cases, because the 'total probability of damage when damaged' should theoretically exactly be 1.
- Σp is smaller than 1. In this case the set of selected damage cases is incomplete, it does not represent all possible damage cases. This is not incorrect, but more damage cases could be defined, which could contribute to the attained subdivision index A.
- Σp is larger than 1. In this case overlapping or identical damage cases have been defined, which is incorrect.

Unfortunately, the practice is often stronger than the doctrine, will Σp often not become exactly 1, also with a correct set of damage cases. So, in practice we are already satisfied if it is in the neighbourhood, e.g. between 0.9 and 1.13 — or 1.15, or 1.17, this limit is not strict.... This phenomenon applies in particular to the (sub-)compartment method, and is discussed in [2], [3] en [8].

Finally, it must be mentioned that besides probabilistic criteria the regulations could also contain deterministic damage stability criteria,. The computation of deterministic damage stability is not a task of this module, that can be performed with another (dedicated) PIAS module Loading. For instance reg. 25-6.2 of SOLAS 1992 requires essentially that every fore peak damage must be survived with s=1 (that is, $GZ_{MAX} \geq 0.10$ m, range $\geq 20°$and $\theta_{statical} \leq 25°$). That this rule is not verified is reported by the program by the message "Compliance with SOLAS reg. 25-6.2 has not been verified".

22.3.5.2 "Only execute" and "Print the complete calculation"

These options will speak for themselves. With the first computations are only executed, nothing is printed. With the second the available results of the last computation are printed (again).

22.3.5.3 Print calculation results, subtotalized by zone

Complete calculation results can, because of their extent, be rather indigestible. It could be convenient to structure the results within zones. If with section 22.3.1.7 on page 428, Define zonal boundaries zonal boundaries have been defined, then with the present option the output can be arranged and printed in zones, see section 22.7.6 on page 440, Appendix 6: collecting the output in zones for an example. This restructuring within zones can be applied with the zonal method and the (sub-)compartment method. In principle the numerical integration method does not use the 'damage boundary' concept, and consequently it cannot be determined in which zone a damage case is located.

22.3.5.4 Show intermediate calculation data

At the execution of the computations many intermediate results are generated. In order to provide the possibility to analyse or verify a certain computation, these intermediate results are saved. With this the intermediate results are shown. Only the intermediate data of the last calculation will be available via this option (the file will be rewritten when a new calculation is performed). If the user desires, these intermediate results can be saved for future reference. Please note that if the option paragraph 22.3.1.1.27 on page 426, Re-use unmodified results of former calculations is used, this will be mentioned in the intermediate calculation data as statement instead of the full data for the corresponding damage case(s). When you require intermediate calculation data with all information, this option should not be used or be removed explicitly, see section 22.3.4.1 on page 431, Remove all results of former calculations.

22.3.6 File and backup management

Backups of the Probdam data can be made and restored here. Here is also the option 'Quit module without saving the data'. See for the details section 2.9 on page 15, Data storage and backups.

22.4 Computing the damage boundaries

There is a direct relationship between compartment boundaries and damage boundaries, however, when applying the zone method the user will have to define the zonal boundaries manually, so no use will be made of the already available compartment geometry. When the compartment method (as weel as with the subcompartment method, see paragraph 22.3.1.1.3 on page 420, Calculation method probability of flooding for those methods), the relationship is evaluated, so that the damage boundaries can be automatically determined. For this prupose, a search algorithm is used that 'plays' with the damage boundaries so that:

- All compartments intended to be flooded are indeed struck bij that damage.
- All other (non-flooded) compartments are not struck.
- The damage is as large as possible.
- Required constraints are taken into account, such as the b1/b2 ratio as discussed in paragraph 22.3.1.1.9 on page 422, Application penetration constraint (b1,b2).

Incidentally, multiple damage bondaries may exist, which all lead to the flooding of the intended compartments, and all have more or less the same size. One of those is chosen, by "coincidence". It can even occur that at one time a slightly different combination of damage boundaries is found than at another time, which might be caused by one of these reasons:

- By a slightly different set of start values for the search algorithm, which will lead to a slightly different result.
- Because PIAS in last instance may search the boundaries by means of a `genetisch algoritme`[1], which is a process where *change* plays a role.

This is not disturbing, both solutions can (and will) both be correct. Moreover, by application of this genetic algorithm it might occasionally occur that damage boundaries are sometimes found and sometimes not. Could that be made more consistent? Yes, but only against a serious increase in calculation times, which is a (too) high price to pay for this occasional effect in damage cases with a marginal effect.

22.5 Warnings

During the calculation Windows might occassionaly report that the program is not responsive anymore. This message is incorrect and can be ignored, for its background please see section 2.12 on page 18, Frequently asked questions. Sensible warnings may be included in the text file with intermediate results (.PD0 file), and have the following meanings:

- *Warning: This damage case is redundant.* Means that this damage case is multiply defined. The multiple definition of damage cases always gives incorrect results, due to incorrect subtraction of subdamages. Please be informed that the sequence of definition of damage cases, as well as their being 'selected', is irrelevant for the mechanism of subtraction of subdamages, so it is also irrelevant for this warning.

[1] `https://en.wikipedia.org/wiki/Genetic_algorithm`

- *Warning: This case contains incomplete subtracted subdamages.* Means that subdamages are being subtracted which combined do not cover the entire damage. This might be correct, but special attention must be given to this case, because it may indicate that damage cases are missing.
- *Warning: Damage impossible, pi, ri and vi are zero.* Means that for this damage case no boundaries can be found (also see the remark at the 'Generation of damage cases' chapter). This damage case is left out in the calculation.
- *Warning: The damage boundaries enclose also negative subcompartments. In combination with the current calculation method (a single damage per subcompartment) such damage boundaries can be expected to be correct, but it is not guaranteed. Please check.* This message, which can only be given with the subcompartment method, indicates that negative subcompartments are damaged within the boundaries of the damage cases. Essentially that should not be problematic, but one should realize that negative subcompartments cannot be damaged on their own, their damage is only realistic where they overlap a positive subcompartment. So it is advised to verify the damage boundaries as found by the program.

22.6 References on probabilistic damage stability

- [1] M. Pawlowski. *Subdivision and damage stability of ships.* Politechnika Gdańska, Gdańsk, Poland.
- [2] H.J. Koelman & J. Pinkster. Rationalizing the practice of probabilistic damage stability calculations[2], International Shipbuilding Progress 50, no.3 (2003) pp.239-253.
- [3] H.J. Koelman. On the procedure for the determination of the probability of collision damage[3], International Shipbuilding Progress 52, no.2 (2005), pp.129-148.
- [4] H.J. Koelman. Re-evaluation of the method to determine the probability of damage[4], IMDC 2006, 16–19 May 2006, University of Michigan, USA.
- [5] H.J. Koelman. A new method and Program for Probabilistic Damage Stability[5], COMP↩ IT 2006, Oegstgeest, The Netherlands, 8-11 May.
- [6] H. Dankowski & S. Krüger. On the safety level of the SOLAS 2009 damage stability rules for RoPax vessels[6], Proc. PRADS 2010.
- [7] H. Dankowski & S. Krüger. Progressive Flooding Assessment of the Intermediate Damage Cases as an Extension of a Monte-Carlo based Damage Stability Method[7], Proc. PRAD↩ S 2013, 20-25 October, Korea.
- [8] H.J. Koelman. Damage Stability Rules in Relation to Ship Design[8], WEMT95 Copenhagen, Denmark, 17-19 May 1995.

22.7 Appendices

[2] http://www.sarc.nl/images/publications/probilistic_isp_v50_2003.pdf
[3] http://www.sarc.nl/images/publications/abstract_isp_v52_2005.pdf
[4] http://www.sarc.nl/wp-content/uploads/2016/07/imdc06.pdf
[5] http://www.sarc.nl/images/publications/probabilistic_compit2006.pdf
[6] http://www.ssi.tu-harburg.de/doc/webseiten_dokumente/ssi/veroeffentlichungen/Prads10_DankowskiKrueger.pdf
[7] http://www.ssi.tu-harburg.de/doc/webseiten_dokumente/ssi/veroeffentlichungen/PRADS2013_DankowskiKrueger.pdf
[8] https://www.sarc.nl/wp-content/uploads/1996/05/WEMT95-Koelman-Probdamstab.pdf

22.7.1 Appendix 1: mean/minimum penetration, IMO circular letter 1338

EdW/90/044

IMO Circular Letter No. 1338 (Annex)

Interpretations by the Netherlands Administration

1. Regulation 25-2, sub 1.3: Partial load line (light ship draught)

   The light ship draught is the draught, assuming level trim, corresponding to the ship lightweight (lightweight is defined in SOLAS 1974, Chapter II-1, Part A, Regulation 3, sub 22).

2. Regulation 25-2, sub 2.1: Subdivision length of the ship (L_s)

   The subdivision length of the ship (L_s) is the greatest projected moulded length of that part of the ship at or below the deck or decks limiting the vertical extent of flooding with the ship at the deepest subdivision load line.

3. Regulation 25-4, sub 7: Control of minor progressive flooding

   Demonstration of the control of progressive flooding is applicable to Regulation 25-6, sub 1.1 and sub 1.2.

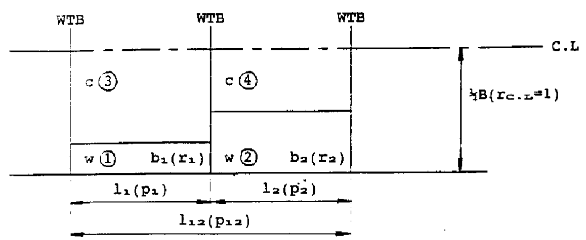
4. Regulation 25-5, sub 1.1.6

   This sub-paragraph should be deleted and a new sub-paragraph Regulation 25-5, sub 3.2 should be added with the text of the existing sub-paragraph sub 1.1.6.

5. Regulation 25-5, sub 2.2: Definition of b

   In the definition of b the expression "mean transverse distance" should be deleted and the wording "minimum transverse distance" should be inserted.

6. Regulation 25-5, sub 3.1



① + ② : $p_{12}r_1 - p_1r_1 - p_2r_1$

① + ② + ③ : $p_{12}r_2 - p_1r_2 - p_2r_2 - (p_{12}r_1 - p_1r_1 - p_2r_1)$

$= p_{12}(r_2-r_1) - p_1(r_2-r_1) - p_2(r_2-r_1)$

① + ② + ③ + ④ : $p_{12}(1-r_2) - p_1(1-r_2) - p_2(1-r_2)$

Note 1: Because of the assumption of a rectangular damage the damage case ① + ② + ④ does not exist.

Note 2: The subscripts of $p_m r_n$ have the following meaning;
m indicates that the value of p is determined on the basis of l_m and n indicates that the value of r is determined on the basis of b_n and the associated l_m.

Note 3: The p_i-value of a group of three or more adjacent compartments are based on a similar approach.

22.7.2 Appendix 2: mean/minimum penetration, the question

Scheepsbouwkundig Advies
en RekenCentrum SARC BV
Elkenlaan 3
1406 PK Bussum
Nederland
Tel : 02159 - 15024
Fax : 02159 - 18303
K.v.K. Hilversum dossier nr. 41.391
Postbank 439 66 88
Bankrekening AMRO-bank 48.56.67.282

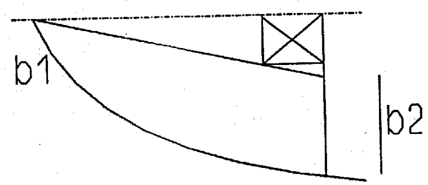
DGSM SI Zeevaart, afd. beleid
t.a.v. J. Vermeer
Postbus 5817
2280 HV Rijswijk

Betreft : Interpretatie
prob. lekberekening

Bussum, 15 juni 1991

Mijne heren,

De vraag betreft de vaststelling van de "mean" penetratiediepte in het volgende geval :



We nemen aan dat het hoofdcompartiment lek moet worden en het kleine compartiment aan HS aan de voorkant *niet* lek mag worden. De binnenbegrenzing van de schade zou in mijn ogen moeten zijn zoals getekend.
Deze oplossing wordt door ons programma echter verworpen omdat het programma werkt met de regels uit de "explanatory notes", welke de verhouding b2/b1 maximeren op 2. Omdat b1 altijd nul is, zorgt het programma ervoor dat b2 ook nul is, zodat als binnenbegrenzing van de schade een rechte lijn vlak langs de huid gevonden wordt.

De vraag is nu hoe in dergelijke gevallen met de b1/b2 verhouding om te gaan.

Overigens wil er op wijzen dat bovenstaand voorbeeld geen gedachtenexperiment is. Het computerprogramma liep bij het eerste het beste "echte" schip wat berekend werd hier tegen aan.

Met vriendelijke groeten,

Herbert Koelman

22.7.3 Appendix 3: mean/minimum penetration, the reply by NSI

Ministerie van Verkeer en Waterstaat

Directoraat-Generaal Scheepvaart en Maritieme Zaken

Aan

S.A.R.C.
t.a.v. de heer H. Koelman
Eikenlaan 3
1406 PK BUSSUM

Contactpersoon: -

Doorkiesnummer: 070-3955620

Datum: 27 juni 1991

Bijlage(n): -

Ons kenmerk: S/V-21.331/91/VTS

Uw kenmerk: -

Onderwerp: Interpretatie "Probabilistische methode lekberekening vrachtschepen"

Geachte heer Koelman,

In antwoord op uw schrijven d.d. 15 juni jl. deel ik u het volgende mede:

1. Toepassing van de definitie van "minimum" penetratiediepte is toegestaan als alternatief voor "gemiddelde" penetratiediepte.

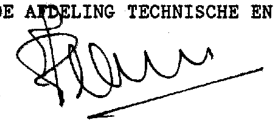
2. Voor eindcompartimenten dient, indien geen gebruik wordt gemaakt van de definitie van "minimum" penetratiediepte de "gemiddelde" penetratiediepte te worden bepaald door de aansnijding van het denkbeeldige langsschot met de huid aan te nemen ter plaatse van de C.L. zoals aangegeven in uw voorstel. (Het probleem zou overigens kunnen worden voorkomen door een hypothetisch dwarsschot te plaatsen t.p.v. achterzijde van kleine compartiment).

Ik hoop dat dit standpunt u voldoende houvast geeft voor een computermatige oplossing.

Met vriendelijke groeten,

HET HOOFD VAN DE AFDELING TECHNISCHE EN BEMANNINGSZAKEN,
(l.-Hfd.),

Ir. H. Vermeer

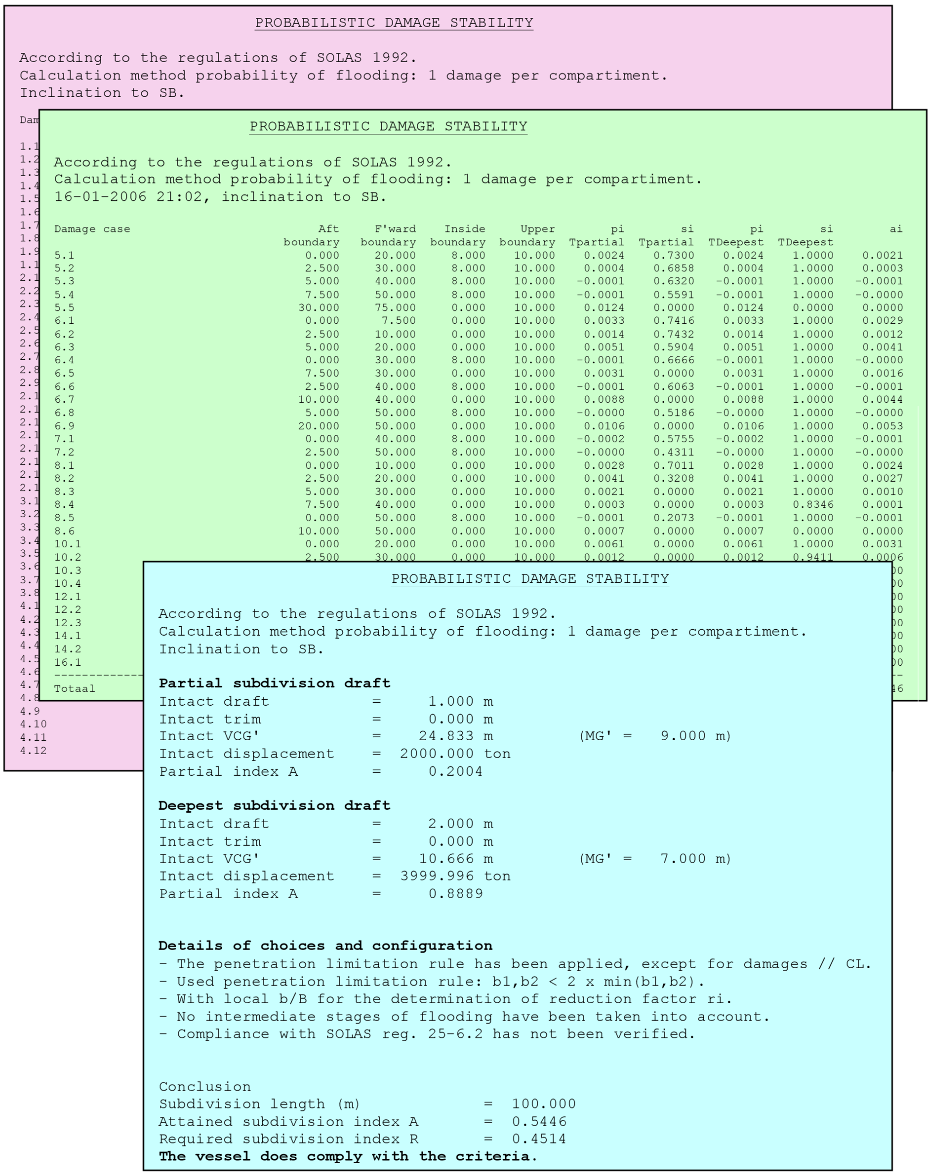


Postadres postbus 5817, 2280 HV Rijswijk
Bezoekadres Bordewijkstraat 4

Telefoon 070-3955555
Telefax 070-3996274
Telex 31040 dgsm nl

Bereikbaar met Buslijn 30 vanaf station H.S. Buslijn 23 vanaf station Voorburg (richting Kijkduin)

22.7.4 Appendix 4: output of an probabilistisc damage stability calculation for a dry cargo vessel



22.7.5 Appendix 5: output of an probabilistisc damage stability calculation for a hopper dredger

PROBABILISTIC DAMAGE STABILITY "AGREEMENT DREDGERS REDUCED FREEBOARDS" (dr-67)

According to the regulations of SOLAS 1992.
Calculation method probability of flooding: 1 damage per compartiment.
Basic loading condition :50 % ballast tk 20,21,24

Damage case	Aft boundary	F'ward boundary	Inside boundary	Upper boundary	pi SW=1.200 SW=1.800 SW=1.839	si SW=1.200 SW=1.800 SW=1.839	pi SW=1.400 SW=2.000 Unloaded	si SW=1.400 SW=2.000 Unloaded	pi SW=1.600 SW=2.200	si SW=1.600 SW=2.200
3.15	25.600	36.400	5.899	7.707	0.0138	1.0000	0.0138	1.0000	0.0138	1.0000
					0.0138	1.0000	0.0138	1.0000	0.0138	1.0000
					0.0138	1.0000	0.0138	1.0000		
3.16	36.400	37.601	0.000	9.800	0.0000	1.0000	0.0000	1.0000	0.0000	1.0000
					0.0000	1.0000	0.0000	1.0000	0.0000	1.0000
					0.0000	1.0000	0.0000	1.0000		
3.17	37.599	44.800	2.300	9.800	0.0269	1.0000	0.0269	1.0000	0.0269	1.0000
					0.0269	1.0000	0.0269	1.0000	0.0269	1.0000
					0.0269	1.0000	0.0269	1.0000		
3.18	44.199	44.800	0.000	9.800	0.0000	1.0000	0.0000	1.0000	0.0000	1.0000
					0.0000	1.0000	0.0000	1.0000	0.0000	1.0000
					0.0000	1.0000	0.0000	1.0000		
3.19	44.200	50.200	5.899	9.800	0.0008	1.0000	0.0008	1.0000	0.0008	1.0000
					0.0008	1.0000	0.0008	0.9730	0.0008	1.0000
					0.0008	1.0000	0.0008	1.0000		
3.20	44.800	50.201	2.300	8.600	0.0118	1.0000	0.0118	1.0000	0.0118	1.0000
					0.0118	1.0000	0.0118	1.0000	0.0118	1.0000
					0.0118	1.0000	0.0118	1.0000		
3.21	44.800	59.200	5.899	7.707	0.0091	1.0000	0.0091	1.0000	0.0091	1.0000
					0.0091	1.0000	0.0091	1.0000	0.0091	1.0000
					0.0091	1.0000	0.0091	0.9937		
3.22	50.199	59.200	2.300	8.600	0.0282	1.0000	0.0282	1.0000	0.0282	1.0000
					0.0282	1.0000	0.0282	1.0000	0.0282	1.0000
					0.0282	1.0000	0.0282	1.0000		
3.23	50.200	62.201	5.899	8.600	0.0056	1.0000	0.0056	0.0000	0.0056	0.2904
					0.0056	0.0000	0.0056	0.0000	0.0056	0.0000
					0.0056	0.0000	0.0056	0.0000		
3.24	59.200	62.201	0.000	8.600	0.0004	1.0000	0.0004	1.0000	0.0004	1.0000
					0.0004	1.0000	0.0004	0.0000	0.0004	0.0000
3.25										
3.26										
3.27										
3.28										
3.29										
3.30										
3.31										
4.1										
4.2										
4.3										
4.4										
4.5										
4.6										
4.7										
4.8										
4.9										
4.10										
4.11										
4.12										
4.13										
4.14										

PROBABILISTIC DAMAGE STABILITY "AGREEMENT DREDGERS REDUCED FREEBOARDS" (dr-67)

According to the regulations of SOLAS 1992.
Calculation method probability of flooding: 1 damage per compartiment.
Basic loading condition :50% consumables.

Results for a cargo S.W. of 2.000 ton/m^3
Hopper number 1
    Intact cargo volume = 837.000 m^3
    Intact volume of water on cargo = 0.000 m^3
Hopper number 2
    Intact cargo volume = 886.700 m^3
    Intact volume of water on cargo = 0.000 m^3
A loaded = 0.4487 (OK)
A unloaded = 0.6797 (OK)
A = 0.5642 (OK)

Results for a cargo S.W. of 2.200 ton/m^3
Hopper number 1
    Intact cargo volume = 806.000 m^3
    Intact volume of water on cargo = 0.000 m^3
Hopper number 2
    Intact cargo volume = 806.454 m^3
    Intact volume of water on cargo = 0.000 m^3
A loaded = 0.4427 (OK)
A unloaded = 0.6797 (OK)
A = 0.5612 (OK)

Results for a cargo S.W. of 1.839 ton/m^3
Hopper number 1
    Intact cargo volume = 914.313 m^3
    Intact volume of water on cargo = 0.000 m^3
Hopper number 2
    Intact cargo volume = 914.315 m^3
    Intact volume of water on cargo = 0.000 m^3
A loaded = 0.4685 (OK)
A unloaded = 0.6797 (OK)
A = 0.5741 (OK)

Details of choices and configuration
- Inclination and trim according to the shifting law formulae from paragraph 6.2.1. of DR-67.
- Calculation inclusing ouring in and out of cargo or sea water.
- The penetration limitation rule has been applied, except for damages // CL.
- Used penetration limitation rule: b1,b2 < 2 x min(b1,b2).
- With local b/B for the determination of reduction factor ri.

22.7.6 Appendix 6: collecting the output in zones

PROBABILISTIC DAMAGE STABILITY

According to the regulations of SOLAS 1992.
Calculation method probability of flooding: 1 damage per compartiment.
Iinclination to SB.

Damage case	Aft boundary	F'ward boundary	pi Tpartial	ai Tpartial	pi TDeepest	ai TDeepest	ai
1-zone damages							
Zone 0	*	2.500	0.0057	0.0046	0.0057	0.0057	0.0051
Zone 1	2.500	5.000	0.0012	0.0009	0.0012	0.0012	0.0010
Zone 2	5.000	7.500	0.0013	0.0010	0.0013	0.0013	0.0011
Zone 3	7.500	10.000	0.0014	0.0011	0.0014	0.0014	0.0012
Zone 4	10.000	20.000	0.0230	0.0170	0.0230	0.0230	0.0200
Zone 5	20.000	30.000	0.0287	0.0214	0.0287	0.0287	0.0250
Zone 6	30.000	40.000	0.0344	0.0257	0.0344	0.0344	0.0301
Zone 7	40.000	50.000	0.0402	0.0301	0.0402	0.0402	0.0351
Zone 8	50.000	75.000	0.2040	0.0000	0.2040	0.2040	0.1020
Zone 9	75.000	100.000	0.2520	0.0000	0.2520	0.2519	0.1260
Total contribution of all 1-zone damages			0.5917	0.1017	0.5917	0.5917	0.3467
2-zone damages							
Zone 0-1	*	5.000	0.0059	0.0046	0.0059	0.0059	0.0052
Zone 1-2	2.500	7.500	0.0022	0.0018	0.0022	0.0022	0.0020
Zone 2-3	5.000	10.000	0.0024	0.0019	0.0024	0.0024	0.0022
Zone 3-4	7.500	20.000	0.0090	0.0063	0.0090	0.0090	0.0077
Zone 4-5	10.000	30.000	0.0350	0.0082	0.0350	0.0350	0.0216
Zone 5-6	20.000	40.000	0.0428	0.0226	0.0428	0.0428	0.0327
Zone 6-7	30.000	50.000	0.0506	0.0294	0.0506	0.0506	0.0400
Zone 7-8	40.000	75.000	0.0763	0.0000	0.0763	0.0763	0.0381
Zone 8-9	50.000	100.000	0.0960	0.0000	0.0960	0.0000	0.0000
Total contribution of all 2-zone damages			0.3202	0.0749	0.3202	0.2242	0.1496
3-zone damages							
Zone 0-2	*	7.500	0.0050	0.0038	0.0050	0.0050	0.0044
Zone 1-3	2.500	10.000	0.0021	0.0016	0.0021	0.0021	0.0018
Zone 2-4	5.000	20.000	0.0075	0.0048	0.0075	0.0075	0.0062
Zone 3-5	7.500	30.000	0.0045	0.0010	0.0045	0.0045	0.0027
Zone 4-6	10.000	40.000	0.0121	0.0022	0.0121	0.0121	0.0072
Zone 5-7	20.000	50.000	0.0145	0.0027	0.0145	0.0145	0.0086
Zone 6-8	30.000	75.000	0.0174	0.0000	0.0174	0.0050	0.0025
Zone 7-9	40.000	100.000	0.0000	0.0000	0.0000	0.0000	0.0000
Total contribution of all 3-zone damages			0.0630	0.0160	0.0630	0.0507	0.0333
4-zone damages							
Zone 0-3	*	10.000	0.0041	0.0030	0.0041	0.0041	0.0035
Zone 1-4	2.500	20.000	0.0060	0.0027	0.0060	0.0060	0.0044
Zone 2-5	5.000	30.000	0.0029	0.0006	0.0029	0.0029	0.0017
Zone 3-6	7.500	40.000	0.0003	-0.0000	0.0003	0.0002	0.0001
Zone 4-7	10.000	50.000	0.0003	-0.0002	0.0003	-0.0004	-0.0003
Zone 5-8	20.000	75.000	0.0000	0.0000	0.0000	0.0000	0.0000
Zone 6-9	30.000	100.000	0.0000	0.0000	0.0000	0.0000	0.0000
Total contribution of all 4-zone damages			0.0136	0.0060	0.0136	0.0129	0.0095

```
Calculation met
Inclination to

Damage case

5-zone damages
Zone 0-4
Zone 1-5
Zone 2-6
Zone 3-7
Zone 4-8
Zone 5-9
---------------------
Total contribution o

6-zone damages
Zone 0-5
Zone 1-6
Zone 2-7
Zone 3-8
Zone 4-9
---------------------
Total contribution o

7-zone damages
Zone 0-6
Zone 1-7
Zone 2-8
Zone 3-9
---------------------
Total contribution o

8-zone damages
Zone 0-7
Zone 1-8
Zone 2-9
---------------------
Total contribution o

9-zone damages
Zone 0-8              *     75.000   0.0000   0.0000   0.0000   0.0000   0.0000
Zone 1-9          2.500    100.000   0.0000   0.0000   0.0000   0.0000   0.0000
---------------------
Total contribution o

10-zone damages
Zone 0-9
---------------------
Total contribution o

=====================
Total
```

PROBABILISTIC DAMAGE STABILITY

According to the regulations of SOLAS 1992.
Calculation method probability of flooding: 1 damage per compartiment.
Inclination to SB.

```
Partial subdivision draft
Intact draft            =     1.000 m
Intact trim             =     0.000 m
Intact VCG'             =    24.833 m        (MG' =   9.000 m)
Intact displacement     =  2000.000 ton
Partial index A         =     0.2004

Deepest subdivision draft
Intact draft            =     2.000 m
Intact trim             =     0.000 m
Intact VCG'             =    10.666 m        (MG' =   7.000 m)
Intact displacement     =  3999.996 ton
Partial index A         =     0.8889


Details of choices and configuration
- The penetration limitation rule has been applied, except for damages // CL.
- Used penetration limitation rule: b1,b2 < 2 x min(b1,b2).
- With local b/B for the determination of reduction factor ri.
- No intermediate stages of flooding have been taken into account.
- Compliance with SOLAS reg. 25-6.2 has not been verified.


Conclusion
Subdivision length (m)              =  100.000
Attained subdivision index A        =  0.5446
Required subdivision index R        =  0.4514
```

Chapter 23

Outflow: probabilistic computation of oil outflow, with the MARPOL simplified method

The MARPOL regulations set a limit on the amount of oil outflow in case of damage. With this module this can be calculated, and tested against the criteria.

23.1 Background of the probabilistic oil outflow calculations

MARPOL contains two sets of probabilistic outflow regulations:

- Probabilistic oil outflow for oil tankers >5000 ton dwt. This is applicable from construction date January 1, 2007.
- Probabilistic oil outflow of fuel oil, for vessels with a total fuel capacity >600 m^3. This applies to vessels with contract date on or after August 1, 2007, keel laying February 1, 2008 or delivery August 1, 2010. Also applies to 'major conversions'.

This PIAS module performs a *simplified calculation*, as prescribed in detail in MARPOL. The alternative would be an *exact calculation*, as e.g. available in Probdam for the computation of prbabilistic damage stability. Such a calculation is explicitly accepted according to reg. 23.10 for oil tankers. For fuel oil tanks this method is not referred to, however, because the calculations for cargo oil and fuel oil are identical, besides for some detail, it could very well be applied to fuel oil tanks too. By the way, the explanatory notes — explanatory notes on matters related to the accidental oil outflow performance under regulation 23 of the revised MARPOL Annex I, 15 October 2004, MEPC.122(52) — make mention of the fact that in non-rectangular cases the simplified method gives a higher outflow than a more exact approach. If desired, SARC will implement a calculation on the basis of the numerical integration method, if:

- Such a method is considered to be necessary. Suppose that with the simplified approach the outflow criterion can easily be matched, then there will be no *raison d'etre* for a more accurate approach.
- With a certifying organisation can be agreed that this numerical integration calculation will indeed be accepted.

23.2 Introduction to this module

The requirements contain rules for determining the probability of an *average outflow* on the basis of many tank parameters (like distances to shell and bottom, volume, tank boundaries). The ship complies with the rule when that *average outflow* is smaller than a certain maximum.

This module automatically determines all these tank parameters, but, as usual, not all rules are equally objective. Take, for example, the determination of the distance y, which is the 'minimum horizontal distance between compartment and side shell'. Such a definition raises the question what the 'side shell' exactly is. Does it run on into the bottom, or into the bilge? And what in case of a rounded gunwhale? In order to have some certainty about this definition, one has added, at the rules for fuel oil tanks anyway: *In way of the turn of the bilge, y need not be considered.....* This does not solve the problem, however; because where exactly is that *way of the turn of the bilge*, and what about the fore and aft parts, where there is not a real bilge, but where everything is just curved?

Because the determination of certain dimensions is therefore sometimes subjective, these can also be given manually as the occasion rises. In any case, it is strongly recommended to check the penetration and tank dimensions thoroughly. Finally, two more remarks:

- In the program and at the output the same symbols are used as in the regulations (although without the typografic refinement of subscripts). In general we are not in favour of including cryptic codes in the input or output, but in this case their meaning has been laid down correctly in the text of the regulations.
- This module calculates the average oil outflow. The position of the tanks in relation to the outer shell of the ship (like, for example, described for fuel oil tanks in paragraphs 6, 7, 8 and 11.8 of reg. 13A) is not determined, and the consequences of that location (for example, the question whether there have to be made outflow calculations at all) have to be verified by the user himself.

23.3 Main menu of this module

This module is activated by chosing from PIAS' main menu, via option *Other*, the module Outflow. After specifying the PIAS file name this module's main menu appears:

Probabilistic computation of oil outflow, with MARPOL's simplified method

1.	Settings for the oil outflow computations
2.	Specify damage boundaries and geometrical parameters
3.	Execute the oil outflow calculations

23.3.1 Settings for the oil outflow computations

1	Type of outflow calculation
2	Calculation method
3	Ship and compartments are symmetrical
4	Light ship draft
5	Load line draft
6	Which oil density to apply
7	Generic oil density
8	Which tank permeability to apply
9	Generic permeability of all tanks
10	With fixed minimum height for determination of y
11	Fixed minimum height for determination of y
12	There are 2 continuous longitudinal bulkheads in the cargo tanks

23.3.1.1 Type of outflow calculation

At this option one can choose between calculations for fuel oil or cargo oil. Each tank must be assigned the correct content destination type in Layout, see paragraph 9.5.1.2.13 on page 218, Oil outflow parameters.

23.3.1.2 Calculation method

In the future this option will be the place to toggle between a *simplified calculation* and a calculation *on the basis of numerical integration*.

23.3.1.3 Ship and compartments are symmetrical

If hullform and compartment are completely symmetrical, then it is sufficient to perform the calculation for one side only (we chose SB). In case of asymmetry the calculation is performed to both SB and PS, and the result is averaged. So, this choice is not governed by the setting of the side of calculation as given at the general project configurations (section 5.1.7 on page 44, Calculate intact stability etc. with a heeling to).

23.3.1.4 Light ship draft

For the determination of the calculation draft. The drafts as entered here are not integrated with the corresponding data from Probdam.

23.3.1.5 Load line draft

Please refer to the remark just above, at Light ship draft

23.3.1.6 Which oil density to apply

Here two choices can be made:

- Apply the density as specified per tank in module paragraph 9.5.1.2.6 on page 216, Design density. This method can only be applied if none of the tanks is designated a 'variable density'.
- With a generic density, as specified in the next line, for all tanks.

23.3.1.7 Generic oil density

If the second option is selected at the previous line, then at this line the uniform density for all tanks can be given.

23.3.1.8 Which tank permeability to apply

Here two choices can be made:

- Apply the permeability as specified per sub-compartment in module Layout.
- With a generic permeability, as specified in the next line, for all tanks.

23.3.1.9 Generic permeability of all tanks

If the second option is selected at the previous line, then at this line the uniform permeability for all tanks can be given.

23.3.1.10 With fixed minimum height for determination of y

As indicated in the introduction, the penetration of side damage, y, needs only to be determined from the side shell (and, with fuel oil tanks, not below h=min(B/10,3)). As an aid for the question where the side shell ends, at this option a certain minimum height for the determination of y can be specified (by the way, the h=min(B/10,3) will always be applied in case of fuel oil tanks).

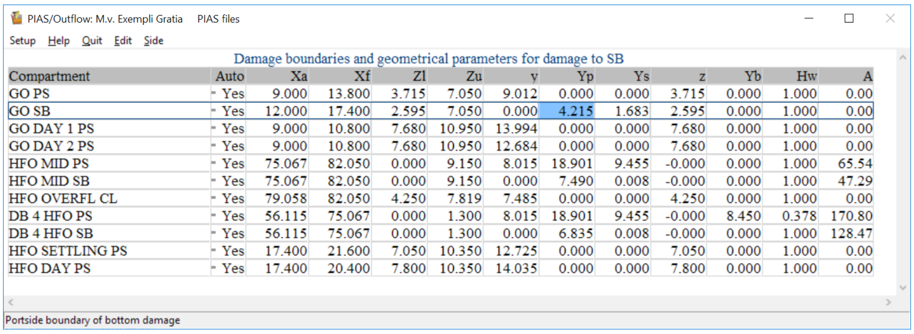
23.3.1.11 Fixed minimum height for determination of y

If the previous line is set to 'Yes', then at this line the minimum height (in meters from baseline) can be given.

23.3.1.12 There are 2 continuous longitudinal bulkheads in the cargo tanks

The answer to this question is relevant for the determination of factor C3, see reg. 23.6 of the cargo oil rules.

23.4 Specify damage boundaries and geometrical parameters



Compartment	Auto	Xa	Xf	Zl	Zu	y	Yp	Ys	z	Yb	Hw	A
GO PS	Yes	9.000	13.800	3.715	7.050	9.012	0.000	0.000	3.715	0.000	1.000	0.00
GO SB	Yes	12.000	17.400	2.595	7.050	0.000	4.215	1.683	2.595	0.000	1.000	0.00
GO DAY 1 PS	Yes	9.000	10.800	7.680	10.950	13.994	0.000	0.000	7.680	0.000	1.000	0.00
GO DAY 2 PS	Yes	9.000	10.800	7.680	10.950	12.684	0.000	0.000	7.680	0.000	1.000	0.00
HFO MID PS	Yes	75.067	82.050	0.000	9.150	8.015	18.901	9.455	-0.000	0.000	1.000	65.54
HFO MID SB	Yes	75.067	82.050	0.000	9.150	0.000	7.490	0.008	-0.000	0.000	1.000	47.29
HFO OVERFL CL	Yes	79.058	82.050	4.250	7.819	7.485	0.000	0.000	4.250	0.000	1.000	0.00
DB 4 HFO PS	Yes	56.115	75.067	0.000	1.300	8.015	18.901	9.455	-0.000	8.450	0.378	170.80
DB 4 HFO SB	Yes	56.115	75.067	0.000	1.300	0.000	6.835	0.008	-0.000	0.000	1.000	128.47
HFO SETTLING PS	Yes	17.400	21.600	7.050	10.350	12.725	0.000	0.000	7.050	0.000	1.000	0.00
HFO DAY PS	Yes	17.400	20.400	7.800	10.350	14.035	0.000	0.000	7.800	0.000	1.000	0.00

Menu with damage boundaries and other geometrical parameters

As motivated in the introduction, it can be desirable to give certain dimensions or distances manually. That can be done in this menu, where the different columns have the following meaning:

- Auto: with ‘Yes’ these compartment parameters are computed by the program, with ‘No’ entered by the
- Xa: aft boundary of damage.
- Xf: forward boundary of damage.
- Zl: lower boundary of side damage.
- Zu: upper boundary of side damage.
- y: penetration of side damage.
- Yp: portside boundary of bottom damage.
- Ys: starboard boundary of bottom damage.
- z: penetration of bottom damage.
- Yb, Hw and A: Only applicable to fuel oil tanks, see MARPOL reg. 13A.11.3

23.5 Execute the oil outflow calculations

With this option the calculation is executed and printed. An output example is included just below.

M.v. Exempli Gratia

Calculation with damage to SB.
(The vessel is symmetrical, the calculation is only made for the SB side)

Tank	PS	OS	OMS	PB	CDB	OB(0)	OB(tide)	OMB(0)	OMB(tide)
GO PS	0.0000	32.71	0.000	0.000	1.0	10.96	32.71	0.000	0.000
GO SB	0.0691	51.77	3.575	0.002	1.0	9.02	50.47	0.015	0.085
GO DAY 1 PS Not taken into account, with an individual capacity not greater than 30 m^3									
GO DAY 2 PS Not taken into account, with an individual capacity not greater than 30 m^3									
HFO MID PS	0.0000	120.63	0.000	0.181	1.0	65.54	65.54	11.868	11.868
HFO MID SB	0.1054	112.64	11.870	0.152	1.0	47.29	49.82	7.206	7.593
HFO OVERFL CL	0.0000	37.68	0.000	0.000	1.0	18.55	37.68	0.000	0.000
DB 4 HFO PS	0.0000	215.95	0.000	0.182	1.0	64.59	64.59	11.752	11.752
DB 4 HFO SB	0.0163	153.95	2.509	0.144	1.0	128.47	128.47	18.436	18.436
HFO SETTLING PS	0.0000	42.66	0.000	0.000	1.0	42.66	42.66	0.000	0.000
HFO DAY PS Not taken into account, with an individual capacity not greater than 30 m^3									
			17.954					49.278	49.734

OMB (Mean oil outflow for bottom damage) = 49.415 m^3

PS	:	Probability of side damage
OS	:	Oil outflow for side damage (m^3)
OMS	:	Mean oil outflow for side damage (m^3)
PB	:	Probability of bottom damage
CDB	:	Oil capture factor
OB(0)	:	Oil outflow for bottom damage, after stranding, without tide (m^3)
OB(tide)	:	Oil outflow for bottom damage, after stranding, with tide (m^3)
OMB(0)	:	Mean oil outflow for bottom damage, without tide (m^3)
OMB(tide)	:	Mean oil outflow for bottom damage, with tide (m^3)

Final conclusion
Probabilistic outflow calculation, using the simplified method, for fuel oil
According to MARPOL reg. 13A

Mean oil outflow for side damage	OMS =	17.954	m^3
Mean oil outflow for bottom damage	OMB =	49.415	m^3
Total volume of oil	C =	806.066	m^3
Dimensionless mean oil outflow	OM =	0.0457	
Maximum allowable OM	OMmax =	0.0148	

The vessel does **not** meet the probabilistic requirement of maximum oil outflow

Output example

Chapter 24

Resistance: resistance prediction with empirical methods

With this module resistance predictions can be made for different ship types, with nine published empirical methods, viz:

- *Hollenbach, for displacement vessels.*
- *Holtrop & Mennen, for displacement vessels.*
- *Van Oortmerssen, for smaller displacement vessels.*
- *Britisch Columbia, for smaller displacement vessels with a low L/B ratio.*
- *MARIN for barges.*
- *Mercier & Savitsky for vessels in the transition region between displacement and planing mode. A.k.a. pre-planing range.*
- *Savitsky, for planing chine vessels.*
- *Robinson, for planing chine and round bilge vessels.*
- *Keuning, Gerritsma & van Terwisga, for planing chine vessels with large deadrise angle.*

24.1 Overview and applicability of the calculation methods.

Note

In this section for each method the applicability and ranges of validity are given. It will be evident that if parameters fall outside these ranges, the results will be unreliable. But even apart from that one should realise that all applied resistance prediction methods used are based on statistics, so it is recommended to always take a critical look at the results.

24.1.1 Hollenbach

With this method a resistance approximation is made for the displacement condition. The approximation is based on *Dr.Ing. Uwe Hollenbach, SDC Ship Design & Consult GmbH, Germany, "Estimating resistance and propulsion for single-screw and twin-screw ships in the preliminary design" 10th International conference on computer applications in shipbuilding, 7–11 June 1999*. The parameters should be within the following limits:

Dependent parameter	Single screw ships				Double screw ships	
	Design draft		Ballast draft		Design draft	
	Min.	Max.	Min.	Max.	Min.	Max.
Lpp [m]	42.0	205.0	50.2	224.8	30.5	206.8
Lpp / $\nabla^{1/3}$	4.49	6.01	5.45	7.05	4.41	7.27
Cb	0.60	0.83	0.56	0.79	0.51	0.78
Lpp / B	4.71	7.11	4.95	6.62	3.96	7.13
B / T	1.99	4.00	2.97	6.12	2.31	6.11
Los / Lwl	1.00	1.05	1.00	1.05	1.00	1.05
Lwl / Lpp	1.00	1.06	0.95	1.00	1.00	1.07
Dp / Ta	0.43	0.84	0.66	1.05	0.50	0.86

Hollenbach limits of applicability.

24.1.2 Holtrop and Mennen

This method approximates the open water resistance according to the following publications:

- *J. Holtrop & G.G.J. Mennen "An approximate power prediction method" International Shipbuilding Progress, 1982.*
- *J. Holtrop "A Statistical re-analysis of resistance and propulsion data", International Shipbuilding Progress, 1984, pp.272–276.*

The parameters should be within the following limits:

- The approximation is valid for seawater (1.025 ton/m^3) of 15°Celsius, for calm water.
- Cross-sectional area of the bulb must be less than 20% of the miship sectional area.
- Midship coefficient between 0.5 and 1.0.
- Lwl/B ratio between 3.5 and 9.5.
- LCB between -5% and +5% of Lwl/2.
- Prismatic coefficient between 0.40 and 0.93.
- Half angle of waterline entrance maximum 70°.
- Resistance coefficient of bow propeller between 0.003 and 0.012.

Holtrop and Mennen.

24.1.3 Oortmerssen

This methode makes an estimation of the resistance of a vessel according to *G. van Oortmerssen, "A Power Prediction method and its application to small ships", International Shipbuilding Progress Vol.18, no 207.* The parameters should be within the following limits:

- Froudenumber (= V / $\sqrt{}$(g × Lwl) where V = speed in m/s, g = 9.81 m/s^2, Lwl = length waterline in m) between 0 and 0.5.
- Length between perpendicular between 9 and 80 m.
- Wetted area between 0 and 1500 m^2.
- Volume between 0 en 3000 m^3.
- Long. centre of buoyancy between -8% and 4% of Lpp before half Lpp.
- Prismatic coefficient between 0.5 and 0.73.
- Half entrance angle waterline between 10°and 46°.
- Breadth / draft ratio between 1.9 and 4.0.
- Midship coefficient between 0.72 and 0.97.
- Length / breadth ratio between 3.0 and 6.5.
- Water density between 1.0 and 1.03 ton/m^3.
- Appendage coefficient (= multiplicationfactor for the volume to get to volume & appendages) between 1.0 and 1.10.

24.1.4 British Columbia

This method can be used for the somewhat smaller vessels with a small length / breadth ratio. The method is based on *Dr. Sander M. Çalişal & Dan McGreer, University of British Columbia, Vancouver, BC, Canada. "Model resistance tests of a systematic series of low L/B vessels" A paper presented to the spring meeting of the Pacific Northwest section of the society of naval architects and marine engineers.* The parameters should be within the following limits:

- Froudenumber (= V / $\sqrt{}$(g × Lwl) where V = speed in m/s, g = 9.81 m/s^2, Lwl = length waterline in m) between 0 and 0.5.
- Wetted area between 0 and 1500 m^2.
- Volume between 0 en 3000 m^3.
- Blockcoefficient between 0.531 en 0.614.
- Breadth / draft ratio between 1.5 and 3.5.
- Length / breadth ratio between 2.0 and 4.5.
- Water density between 1.0 and 1.03 ton/m^3.
- Appendage coefficient between 1.0 and 1.10.

24.1.5 Barge

This method can be used to approximate the resistance of a barge. The calculations are according to the method described in *MARIN report no.49791-1-RD, "Een empirisch model voor de weerstandspredictie van bakken"*. The prediction is valid for deep and calm water, and has the following applicability:

- Froudenumber (= V / (g × breadth)$^{1/2}$, where V = speed in m/sec and g = 9.81 m/sec^2) smaller than 0.60.
- Length / breadth ratio between 2.25 and 8.0.
- Breadth / draft ratio smaller than 10.
- Prismatic coefficient between 0.7 and 1.
- Length of fore body, with a minimum of 0.01 m on model scale (note: at SARC this is considered to be a bit curious criterion, for what would be the 'model scale' of a scale 1:1 barge? But anyway, this is how it is listed in MARIN's publication).

24.1.6 Preplan

With this method you can calculate the resistance of a vessel in the area between displacement and planing condition. The calculations are according to the method as published in *J.A. Mercier & D. Savitsky, "Resistance of transom shear craft in the pre-planing range", Davidson Labatory Report 1667, Stevens Institute of Technology, June 1973*. The method has the following area of applicability:

- Froude number based on the volume (= V / $\sqrt{}$(g × (volume$^{1/3}$)) where V = velocity in m/sec, g = 9.81 m/sec^2, volume = volume in m^3) between 1 and 2.
- Half angle of entrance of the waterline between 10°and 55°.
- Ratio length / volume$^{1/3}$ between 2 and 12.

- Ratio transomarea / maximum cross-sectional area between 0 and 1.
- Ratio length / breadth between 2 and 14.
- Water density between 1.00 and 1.03 ton/m^3.
- Appendage coefficient between 1.0 and 1.10.

24.1.7 Savitsky

With this method you can calculate the resistance of a planing hull. The method is based on the following two publications:

- *D. Savitsky "Hydrodynamic design of planing hulls", Marine Technology, Vol.1, No.1, Okt. 1964, pp. 71–75.*
- *Donald L. Blount & David L. Fox "Small craft power prediction", Marine Technology Vol.13, No.1, Jan. 1976, pp. 14–45.*

For the resistance calculations all forces are assumed to go through the centre of gravity of the vessel. For calculations in the preliminary design stage this is assumed to be sufficiently accurate. The output shows two values for the resistance:

- Resistance according to Savitsky which is valid for test conditions.
- Corrected resistance for true operational conditions according to the method of Blount and Fox. This method calculates two correction factors:
  - Correction for test resistance to true operational resistance for the naked hullform.
  - Correction for the influence of appendages.

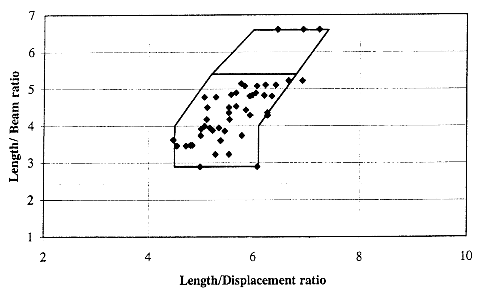
The method has the following area of applicability:

- Rise of floor (deadrise angle) between 0°and 35°.
- Density of water between 1.00 and 1.03 ton/m^3.
- Appendage coefficient between 1.0 and 1.10.
- Speed ratio Cv (= V / $\sqrt{}$(g × Bpx), with V = speed in m/sec, g = 9.81 m/sec^2 and Bpx = maximum knuckle breadth in m) between 0.6 and 13.
- Length of wetted keel / maximum knuckle breadth larger than 4, this implies that the length on the waterline / maximum knuckle breadth larger is than 4.

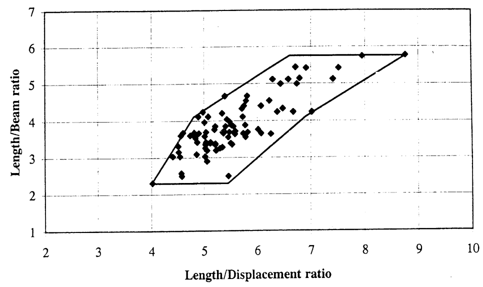
24.1.8 Robinson

With this method the resistance of a planing vessel can be calculated. The method is based on *John Robinson, Wolfson Unit MTIA, University of Southampton, "Performance prediction of chine and round bilge hull forms", Hydrodynamics of High Speed Craft, 24 and 25 November 1999, London.* The following limits for the parameters have to be taken into account:

- Volumetric Froude number between 0.5 and 2.75.
- See following figures:



Hard chine regression data boundary



Hard bilge regression data boundary

24.1.9 Delft

With this method the resistance of a planing vessel can be calculated. The method is based on *J.A. Keuning, J. Gerritsma and P.F. van Terwisga, "Resistance tests of a series planing hull forms with 30 degrees deadrise angle, and a calculation model based on this and similar systematic series", International Shipbuilding Progress 40, No.424, (1993) pp. 333–385.* The following parameter limits have to be taken into account:

- Volumetric Froude number between 0.75 and 3.00.
- Volume between 2.5 and 5000 m^3.
- Rise of floor (deadrise angle) between 12.5°and 30°.

24.2 Shallow-water correction

A shallow water correction method has been implemented, based on *H.C. Raven, "A new correction procedure for shallow-water effects in ship speed trials", Proceedings of PRADS2016 (2016), Kopenhagen, Denmark.* In 2017 the ITTC has accepted this method: *International. Towing Tank Conference, "Report of specialist committee on performance of ships in service", Proceedings of 28th ITTC (2017), Wuxi, China. .*

Raven's method has been implemented for five of the resistance prediction methods: Hollenbach, Holtrop and Mennen, Oortmerssen, Barge and British Columbia.

The method knows the following limits of applicability:

- Froude depth number lower than 0.65.
- Draft / water depth ratio below 0.5.
- Displacement increase due to additional sinkage below or equal to 5%.

24.3 Main menu

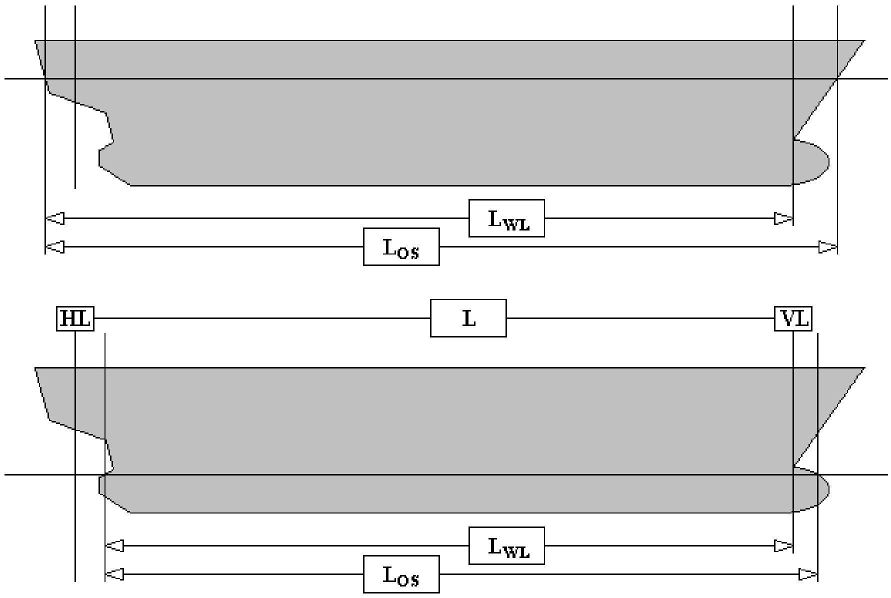
Resistance prediction

1.	Input data resistance prediction
2.	Calculate and print
3.	Diagram of resistance components
4.	Calculate and send to Propeller
6.	Local cloud monitor
7.	File and backup management

24.3.1 Input data resistance prediction

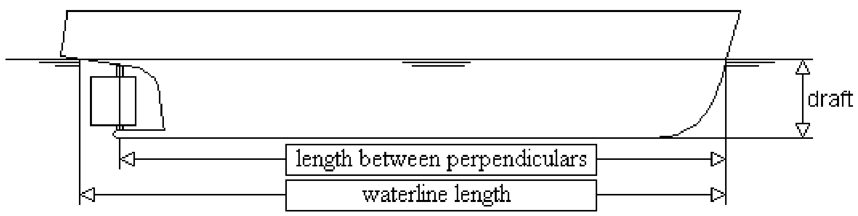
In this window all ship's parameters which are applicable for a certain method should be given. In the list below **all** existing parameters are included, however, in reality only those relevant for a particular method are shown. The definition of all parameters is strictly according to the convention of the applied resistance prediction method, which does not have to agree with the PIAS convention. Below, some guidance on these conventions is given, nevertheless it is recommended to keep the source publications at hand for the details.

- Method: the chosen resistance prediction method.
- Name and Identification name: just for identification purposes
- Density of the water, in ton/m^3.
- Ship type: choice between single / twin screw vessel, or chine / round-bilge vessel (dependant on the selected prediction method).
- Length of underwater body: see figure below.



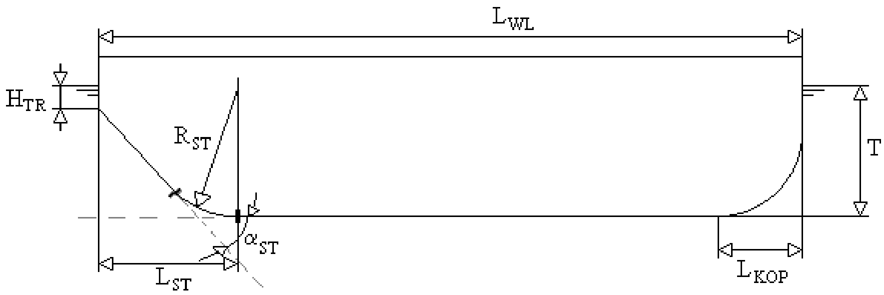
Definition of lengths for the Hollenbach method

- Length waterline and length between perpendiculars: The aft perpendicular is the center of the rudder stock, the forward perpendicular is the intersection of the waterline and the stem. Length between perpendiculars is the distance between those perpendiculars, length waterline is the distance between the forward perpendicular and the intersection between waterline and the stern, as depecited in the figure below.



Definition length for the Oortmerssen method.

- Length of forebody: length L_{KOP}, see figure below.
- Length of non-prismatic aftbody: length L_{ST}, see figure below.
- Draft for calculation: choice between design and ballast-draft.
- Mean draft: the vertical distance between the baseline and the waterline at half the waterline length.
- Wetted surface: the wetted surface area of the hull form that is under the waterline.
- Midship coefficient to be determined on the largest cross-sectional area.
- Waterline coefficient: If yet unkown, it can be roughly approximated by 1/3 + 2/3 $\times$ block coefficient.
- Prismatic coefficient of aftbody: should be given for the non-prismatic part of the aftbody.
- Angle of aftbody buttocks to the baseline: angle α_{ST}, see figure below.
- Radius flat bottom-aftbody: radius R_{ST}, see figure below.



Definitions for the Barge method.

- Half entrance angle of the waterline: the half entrance angle of the faired waterline, without the local stem correction.
- Longitudinal centre of buoyancy: defined as an percentage of the waterline length, relative to half waterline length, positive forward and negative aft.
- Cstern: the C_{STERN} shape coefficient value from the tables below.

For the coefficient C_{stern} the following tentative guidelines are given:

Afterbody form	C_{stern}
V-shaped sections	– 10
Normal section shape	0
U-shaped sections with Hogner stern	+ 10

The Cstern table from the Holtrop & Mennen paper.

- Appendage coefficient: the appendage area (in m^2) as well as the resistance coefficient (labelled '1+K2') according to the table below. For a composition of multiple appendages, the weighted average of the 1+K2 coefficients should be taken.

Approximate $1 + k_2$ values

rudder behind skeg	1.5 – 2.0
rudder behind stern	1.3 – 1.5
twin-screw balance rudders	2.8
shaft brackets	3.0
skeg	1.5 – 2.0
strut bossings	3.0
hull bossings	2.0
shafts	2.0 – 4.0
stabilizer fins	2.8
dome	2.7
bilge keels	1.4

The 1+K2 table from the Holtrop & Mennen paper.

- Submerged transom area: the submerged transom area (in m^2).
- Mean immersion depth of the transom: Mean immersion depth of the transom H_{TR}. See figure of "Definitions for the Barge method".
- Number of bow thrusters: the number of bow thrusters, in combination with the diameter (in m) and resistance coefficient (between 0.003 and 0.012) for every bow thruster opening.

- Bulb: choice between bulb or no bulb. With the cross-sectional area at FPP (in m^2) of the bulbous bow and the VCG (in m) of this cross-sectional area from baseline.
- Planing length: length of the projected planing bottom area.
- Planing breadth: breadth over the chines.
- Planing area: projected planing bottom area.
- Twisted bottom: choice between twisted bottom or no twisted bottom.
- Twist angle: difference in rise of floor between foreship and aftship.
- Centreline angle: average angle between centreline and baseline, over the aft half of the ship. Positive if draft aft is greater than draft at half length.
- Breadth: for method Robinson the breadth over all. For other methods, moulded breadth of the ship.
- Breadth at chine: the breadth of the ship at chine.
- Volume including shell & appendages: the volume of the ship including the volume of the shell and appendages.
- Moulded volume: the moulded volume of the ship.
- Model ship correlation coefficient: is a coefficient which is used to get from specific method scale model values to real life scale.
- Submerged cross section area: the submerged cross section area.
- Longitudinal centre of gravity: is the location of the centre of gravity in longitudinal position, measured from the transom along the keel line of the ship.
- Rise of floor: in a cross section, the angle between the baseline and the bottom. Deadrise angle.
- Speed interval: at which interval, between start speed and end speed, the resistance will be calculated.
- Input air resistance: choice between with or without air resistance.
- Projected area air resistance: the projected area which is subjected to air resistance.
- Stabilisation vins: choice between with or without stabilisation vins.
- Wetted surface area stabilisation vins: the wetted surface area of the stabilisation vins.
- Bilge keels: choice between with or without bilge keels.
- Wetted surface area bilge keels: the wetted surface area of the bilge keels.
- Dome: choice between with or without dome.
- Wetted surface area dome: the wetted surface area of the dome.
- LCG relative to half length of planing area: in percentage, positive forward and negative aft.
- Waterdepth: The waterdepth in meters that is used for the optional shallow water correction. Must be at least twice the draft.

Besides the columns for specifiying the parameter velues, this menu contains at the right side a column labelled ‘source’, which can be:

- User-defined: a value given by the user.
- From hullform: the parameter value of this line is extracted from the PIAS hullform (as defined with Fairway of Hulldef).
- Estimation: this parameter is to be determined with empirical estimation equations, which are available in some resistance methods. Obviously, this option (as well as ‘From hullform’) is only applicable to rows where supporting functions are indeed available for the corresponding parameters.

Attention

With the ‘From hullform’ option one should realize that a Hulldef ship hull representation contains sections only. This implies that the waterline extremities are not exactly available, because they will in general fall between two sections. PIAS uses extrapolation to estimate the waterline shape in those regions, however, it is advised to verify the waterline-oriented parameters — typically ‘Half entrance angle of the waterline’ and ‘Length waterline’ — well and correct if neccessary. It will be obvious that this warning evaporates a bit with a dense frame set, e.g. as can be generated with Fairway.

24.3.2 Calculate and print

For the parameters as given at the first menu option, here the resistance approximation is made, and the results are printed in a table.

24.3.3 Diagram of resistance components

Similar to the previous option, with the resistance and its components plotted in a graph.

24.3.4 Calculate and send to Propeller

The input data as well as the calculated resistance is sent to the propeller calculation module Propeller.

24.3.5 Local cloud monitor

See section 2.11 on page 17, Local cloud: simultaneous multi-module operation on the same project.

24.3.6 File and backup management

With this option design version can be managed, with a mechanism as described in section 2.9 on page 15, Data storage and backups.

Chapter 25

Propeller: propeller calculations with standard propeller series

With this module properties can be computed from propellers of the following, published, empirical propeller series:

- *Open water propellers of the systematic B-series of MARIN, Wageningen, The Netherlands.*
- *Ducted propellers of the systematic Ka-series, and one from the Kd-series of MARIN.*
- *Propeller series by Gawn.*
- *The Japanese Au series for three-bladed, four-bladed and six-bladed propellers.*

25.1 Overview and applicability of the calculation methods

Note

Please take the warning in the note of section 24.1 on page 446, Overview and applicability of the calculation methods. — on the attitude with respect to empirical/statistical prediction methods — heartily.

25.1.1 B-serie

The calculation is based on the methode of *M. Oosterveld & P. van Oossanen, NSP 1974*, and is valid for pitch/diameter ratios between 0.6 and 1.4. The applicable blade numbers and blade area ratios are listed in the table below:

Number of blades	2	3	4	5	6	7
Expanded area ratio	0.3	0.35-0.8	0.4-1.0	0.45-1.05	0.5-0.8	0.55-0.85

25.1.2 Ka/Kd-series

The calculation is based on the method of *M. Oosterveld, 'Ducted propeller characteristics', RINA 1973*, and is valid for pitch/diameter ratios between 0.5 and 1.6, and for the following propeller/nozzle combinations:

Propeller	Nozzel	Number of blades	Blade area ratio
Ka 3-65	19A	3	0.65
Ka 4-55	19A	4	0.65
Ka 4-70	19A	4	0.70
Ka 4-70	22	4	0.70
Ka 4-70	24	4	0.70
Ka 4-70	37	4	0.70
Ka 5-75	19A	5	0.75
Ka 5-100	33	5	1.00

The different nozzles have the following particulars, where L is the length of the nozzle and D the propeller diameter.

- nozzle 19A - L/D = 0.5, accelerating flow type
- nozzle 22 - L/D = 0.8, accelerating flow type
- nozzle 24 - L/D = 1.0, accelerating flow type
- nozzle 33 - L/D = 0.6, decelerating flow type
- nozzle 37 - L/D = 0.5, accelerating flow type

Nozzle 22 and 24 are similar to 19A, except for the L/D ratio which is higher, which is favourable for tugs. Nozzle 37 has a thick trailing edge which results in better performance with power astern. Nozzle 33 has a higher cavitation limit which is favourable to decrease the level of vibrations and noise.

25.1.3 Gawn-series

The calculation is based on the method of *R. Gawn, 'Effect of pitch and blade width on propeller performance', RINA 1952*, and has the following application area:

- Only 3 bladed propellers.
- Expanded area ratio should be between 0.2 and 1.1.
- Pitch/diameter ratio should be between 0.8 and 1.4.

25.1.4 AU-series

The calculation is based on the method of *A. Yazaki, 'Design diagrams of modern four, five, six and seven-bladed propellers developed in Japan', 4th Naval Hydronamics Symposium, National Academy of Sciences, Washington, 1962*. The parameters should be within the following limits per propeller:

Propeller name	N-AU 3-35	N-AU 3-50	AU 4-55	AU 4-70	AUw 6-55	AUw 6-70	AUw 6-85
Number of blades	3	3	4	4	6	6	6
Expanded area ratio	0.35	0.5	0.55	0.7	0.55	0.7	0.85
Pitch/diameter ratio	0.4-1.2	0.4-1.2	1.0-1.6	1.0-1.6	0.9-1.5	0.9-1.5	0.9-1.5

25.2 Main menu

Propeller calculations

1.	Input of ship hull parameters
2.	Input of propeller data
3.	Input of speed and resistance range
4.	Calculate propeller with maximum efficiency in a range of diameters
5.	Calculate propeller with revolutions variation
6.	Calculate resistance with fixed propeller dimensions
7.	Calculate speed-power curve with fixed propeller dimensions
8.	Calculate thrust force for a fixed pitch propeller
9.	Calculate thrust force for a controlable pitch propeller
10.	Local cloud monitor
11.	File and backup management

25.3 Input of ship hull parameters

Here an input window with ship hull parameter appears. **Excepts for the drafts, they will only be utilized for the estimation of wake and thrust deduction coefficients (by Holtrop & Mennens method), not for the propeller calculations as such**. If the resistance values have been transferred from Resistance to Propeller, those parameters are already filled in, because they have already been defined in Resistance and have been cotransferred. So, for their description reference is made to section 24.3.1 on page 450, Input data resistance prediction.

25.4 Input of propeller data

- Number of propellers: 1 or 2, so, single screw or twin screw propulsion. With two propellers, all over the computations the presence of two propellers are taken into account, and the final resulting shaft power applies obviously to the entire vessel, not for a single propeller.
- Number of blades per propeller will speak for itself. The minimum or maximum number depends on the selected propeller series.
- Losses in propeller axis: mechanical loss in percent.
- Diameter start-increment-end can be specified in order to make calculations for a series of propeller diameters.
- revolutions start-increment-end is specified to make calculations for a range of propeller revolutions. However, this is only used in calculation 5.
- Pitch/diameter ratio: the pitch/diameter ratio has to be filled in by calculations 6 and 7. With calculation 4 and 5 this value is determined by the program.
- Determination center shaft - base: if this option is set '0.53 × diameter' then the distance between the center of the propeller shaft and baseline is recalculated for each diameter (with this = 53% of the propeller diameter). If the option is set to 'User defined' than the user-specified value is used.
- Expanded area ratio: the expanded area ratio (A_E/A_0) of the B-series and Gawn propellers can be estimated by the program (according to the cavitation criterion of Keller) by setting the field 'Determination method blade-area ratio' to 'Calculate'.
- Wake and thrust deduction: There are three methods:
  - Estimate according Holtrop & Mennen: with this estimation method (see Resistance for the references) on basis of the ship hull parameters, like defined in the first menu option(see section 25.3 on the preceding page, Input of ship hull parameters), the wake and thrust deduction is estimated. If this method is not used, then these ship hull parameters (except for the drafts) no not neccessarily have to be given.
    Unfortunately, experience has shown that for full single-propeller vessels the Holtrop & Mennen formulae tend towards unrealistic high values. In such as case — higher than the tentatively selected wake factor of 0.45 — an alternative formula is used, namely that of Schneekluth (1988). This is a bit of a ramshackle, but that is not uncommon with empirical estimation methods.
  - Fixed, user-specified values: Here you defined one wake and thrust deduction which will be used for every speed and diameter.
  - User defined per speed-diameter: In this menu the wake and thrust deduction can be defined per speed-diameter. If changes are made to the number of speeds-diameters than you have to check if these user defined wake and thrust deduction values are still valid for there respective speed-diameter combination.
- Propeller series: The to be used propeller series. With the Ka- and AU-series the following input parameters are automatically determined in and cannot be changed:
  - The number of blades.
  - The expanded area ratio.
  - The type of propeller.

25.5 Input of speed and resistance range

In this input screen you can enter up to twenty speeds with the resistance. Enter the speed in knots and resistance in KN.

25.6 Calculate propeller with maximum efficiency in a range of diameters

This option is for calculating a propeller at a given speed and resistance when the number of revolutions and the pitch of the propeller is unknown. The determined propeller has a maximum possible open water efficiency.

25.7 Calculate propeller with revolutions variation

This option calculates a propeller at a given speed, resistance and (range of) number of revolutions. The pitch is determined so that the delivered shaft horse power equals the required shaft horse power. The previous calculation option (maximum efficiency) calculates all combinations of speed and diameter. The present revolutions variation does, however, its calculations only for the very first diameter.

25.8 Calculate resistance with fixed propeller dimensions

This option is to determine the resistance at the trial trip. All propeller characteristics have to be defined by the user. Using the defined propeller dimensions the resistance is calculated.

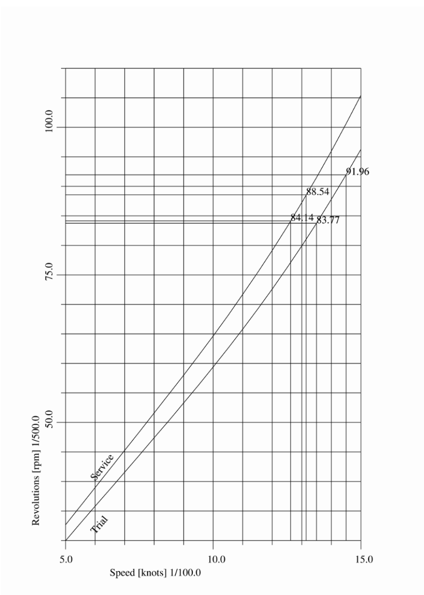
25.9 Calculate speed-power curve with fixed propeller dimensions

The following menu will be displayed on the screen, where with the first two options the speed-power curve is actually computed and plotted. The first option is for a fixed pitch propeller (and consequently varying revolutions), and the second for a controlable pitch propeller, with fixed revolutions. The other options can be used to configure the nature and content of the graph.

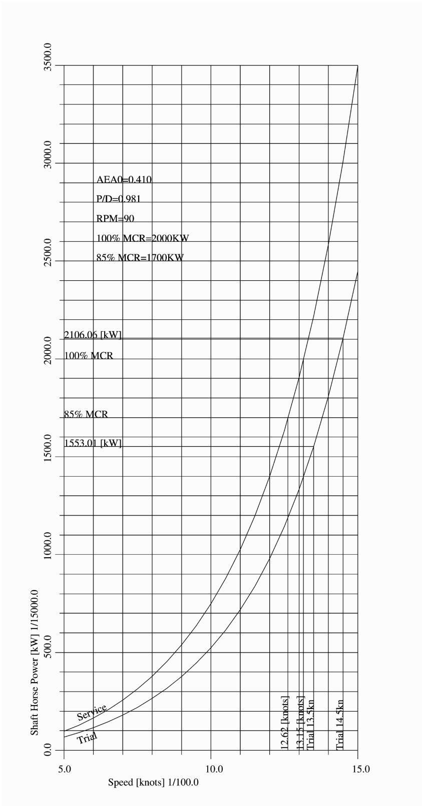
Speed-power curve for a fixed propeller dimensions

1. Speed-power curve for a fixed pitch propeller
2. Speed-power curve for a controlable pitch propeller
2. Legends at the graph
3. Allowances
4. Efficiency reduction at constant revolutions
5. Intersections in the graph
6. Layout

For a propeller with fixed dimensions a graph can be plotted, which gives the relationship between the speed and the required shaft horse power. With the standard version the calculation can be performed for a fixed pitch propeller. The graphical extensions also allow calculations with a controllable pitch propeller. If there are any losses due to pitch variation, these can be defined by an allowance on the required power. The graph shows the shaft power at the speeds you specify. It is therefore important to specify sufficient speeds, with a small interval to obtain a smooth graph. Below you will find an example of such a graph.



Power curve.



Power curve with generated text labels.

25.10 Calculate thrust force for a fixed pitch propeller

With this option the thrust force can be calculated for a range of speeds, by varying the revolutions. The available shaft power (in KW) is defined at option 3, by entering the power instead of resistance. If the speed is zero, the wake factor and the thrust deduction fraction are default set to 0.05. This value can be changed by entering the wake and thrust factors by hand. The thrust force at speed zero is the bollard pull. The calculated thrust force reduced with the resistance, at speeds larger than zero, results in the available thrust force, for example for towing fishing gear.

25.11 Calculate thrust force for a controlable pitch propeller

With this option the thrust force can be calculated for a range of speeds, by varying the pitch-diameter ratio, for a fixed number of revolutions (for which the first revolutions value is taken from the range as given in option 2 (section 25.4 on page 457, Input of propeller data. Also for this option the shaft power should be given at option 3, by entering it (in KW) in the 'Resistance' column. For wake and thrust deduction factors, as well as bollard pull — the same remarks as from the previous option apply.

25.12 Local cloud monitor

This option pops up a window with a power curve, as discussed in section 25.9 on the preceding page, Calculate speed-power curve with fixed propeller dimensions. However, here, in *cloud* context, this diagram is dynamic, which means it is recomputed and redrawn every time when data in the cloud which affects the power results changes. For more information on the cloud reference is made to section 2.11 on page 17, Local cloud: simultaneous multi-module operation on the same project.

25.13 File and backup management

With this option design version can be managed, with a mechanism as described in section 2.9 on page 15, Data storage and backups.

Chapter 26

Motions: ship motions estimation

With this module the ship motions in natural seaways can be estimated in the frequency domain, using one of the following methods:

- *Jensen, an emperical method, meant for quick estimations in the design phase. An advantage of this method is that the calculations are based on the global hull form coefficients, so no hull form is required yet.*
- *A strip theory based method which performs calculations using the hullform as defined in PIAS.*

26.1 Overview and applicability of the estimation methods

It is important to note that this manual will only discuss the PIAS implementation of the methods. The user is assumed to have knowledge of seakeeping and wave theory, as can be found in standard text books and is taught in universities.

26.1.1 Jensen

Note

> Please take the warning in the note of section 24.1 on page 446, Overview and applicability of the calculation methods. — on the attitude with respect to empirical/statistical estimation methods — at heart.

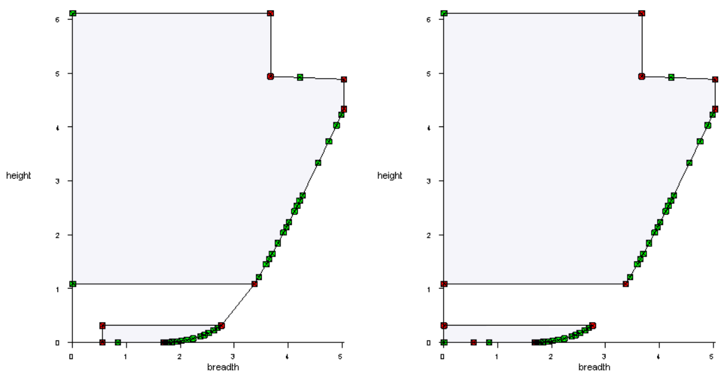
This method is based on *Jensen, J.J., Mansour, A. & Olsen, A.S. (2004). Estimation of ship motions using closed-form expressions. Ocean Engineering, 31(1), 61–85*. They developed closed-form semi-analytical expressions for the response function for heave, pitch and roll and the vertical motion, velocity and acceleration of monohull ships.

In the paper it is stated that its formulae estimate the motions fairly accurate, with the exceptions of:

- The heave motion is too small for wavelengths larger than the ship length.
- The pitch motions is too large where the wave length is around the same as the ship length for Froude numbers larger than 0.2.
- The roll motion is too large around the resonance frequency.

26.1.2 Strip theory

A strip theory method has been implemented based on *Bertram, V., Veelo, B., Söding, H., Graf, K. (2006, May). Development of a freely available strip method for seakeeping. In COMPIT (Vol. 6, pp. 356–368)*. The program PDstrip which is presented in this paper is used as the computational engine in Motions. In order to use the strip theory method a hullform is necessary. Motions uses the PIAS frame model that is described in section 2.10.2 on page 16, Hull form representations. The program is currently only suitable for monohull vessels, while added hullforms will not be taken into account. The current implementation of the program mirrors starboard and portside parts of the hullform in order to speed up the calculations, and thus is only suited for symmetrical hullforms. For common hullforms 30 to 40 frames are considered sufficient to properly capture the hullform. Although the program can handle the PIAS maximum number of frames (currently 500), and accounts for double frames. Gaps in a frame are accepted, provided that the frame does not intersect itself. An example of what is not allowed and what is allowed is shown in the image below.



Example of a gap in a frame. On the left the frame intersects itself, which Motions does not accept. On the right is an example of a gap that does work in Motions.

Only the underwater part of the hull is used in calculations, which is 'cut off' from the hullform at the user-defined draft and trim. When multiple intersections with the waterline are present in a frame, as may be the case in a propeller tunnel which is not wholly submerged, the part of the frame that is present after the first intersection of the frame shape with the waterline is neglected.

26.1.3 Wave spectra

In order to estimate the probability of exceedance of a motion, a wave spectrum is required. Six wave spectra have been pre-programmed, according to the equations in *Stansberg, C.T., Contento, G., Hong, S. W., Irani, M., Ishida, S., Mercier, R., & Kriebel, D. (2002). The specialist committee on waves final report and recommendations to the 23rd ITTC. Proceedings of the 23rd ITTC, 2, 505–551*. These spectra are discussed below.

Note

> The quasi-static wave as specified with config, see section 5.7.1 on page 52, Wave amplitude, has no influence on the ship motions computations of this module. After all, in Motions the calculation is not based on a single static wave, but on a distribution (a spectrum) of many waves.

26.1.3.1 Jonswap

The JONSWAP spectrum defines seas with finite fetch. It requires input of a *peak frequency and a peak enhancement factor*. The approximations for this spectrum are believed to be correct for a range of the *peak enhancement factor* between 1 and 7. When the wind speed U and the fetch F are known, the peak frequency of the spectrum can be calculated according to $f_\mathrm{p} = \frac{g\hat{F}^{-1/3}}{U}$ where $\hat{F} = \frac{gF}{U^2}$.

26.1.3.2 Spectra of the generalized Pierson Moskowitz form

The other spectra are of the generalized Pierson Moskowitz or Bretschneider form. These spectra define fully developed seas. The spectra and their required input are listed below:

- One-parameter Pierson-Moskowitz: requires input of either wind speed, peak frequency or significant wave height. When one of these values is inputted into the program, the corresponding values for the other two parameters are automatically updated.
- Two-parameter Pierson-Moskowitz: requires input of the peak frequency and the significant wave height.
- ISSC: requires input of the mean frequency and the significant wave height.
- ITTC: requires input of significant wave height and one of the energy, peak, mean or zero-crossing periods.
- Liu: requires input of the wind speed and the fetch.

26.2 Main menu

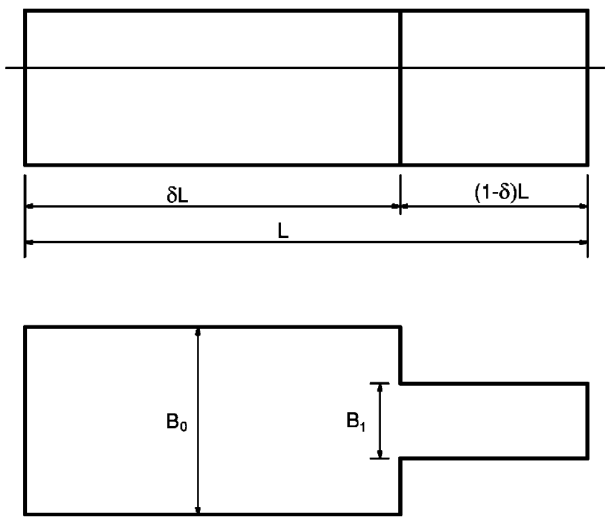
Ship motion estimation

1.	Input data for ship motion analysis
2.	Specify points of interest
3.	Calculate and print output
4.	Filemanagement

26.2.1 Input data for ship motion analysis

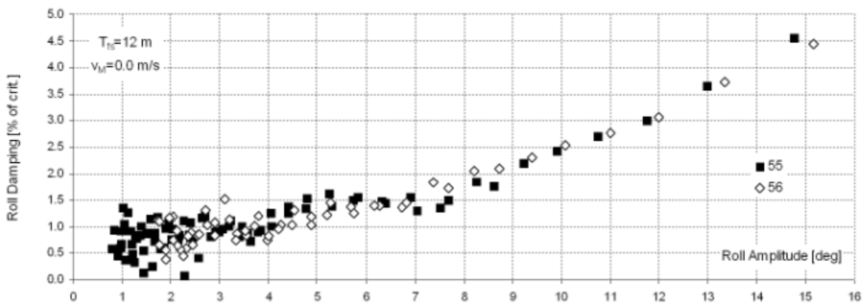
In this window the parameters which are needed for the various methods are given. The list below now only lists those for the Jensen method. For additional detail it is advised to consult the source publication.

- Method: the chosen calculation method.
- Name of the vessel: can be taken from the PIAS hullform if a project name is defined in section 7.2.1.1 on page 167, Main dimensions and allowance for shell and appendages.
- Start frequency, frequency interval and end frequency: the frequency range for which the motions are calculated is defined here. Should the frequency interval be zero or larger than (end frequency – starting frequency), the calculation shall only be performed for the starting and end frequency.
- Draft: when derived from a hullform the design draft as defined in Hulldef is used.
- Trim: according to the definition in section 2.6 on page 9, Definitions and units.
- Water depth: distance from the surface of the water to the bottom.
- Main hullform parameters: length waterline, breadth, draft, block coefficient, waterline coefficient: if a PIAS hullform is available, these parameters can be derived from it.
- Calculate roll RAO: choice to calculate roll with the Jensen method. Can be turned off if the required additional input of the waterplane coefficient, metacentric height or the natural roll period is not available. The length ratio is chosen and the critical damping is optional.
- Metacentric height (GM): is linked to the VCG.
- VCG: height of the vertical center of gravity (is linked to the metacentric height).
- Natural roll period: if the natural period is not known it can be estimated. For this the *IMO Intact Stability Code 2008 A.2.3.4* is used, which uses the length, breadth, draft and metacentric height parameters as input.
- Length ratio: the Jensen method simplifies the vessel into two prismatic beams for the roll calculation, with the same draft but different breadth and cross-sectional areas. The length ratio δ defines the length of these two prismatic beams, as seen in figure below. The value of the length ratio must be between 0 and 1, but not greater than the waterline coefficient.



Length ratio delta (Jensen, Mansour & Oslen, 2004)

- Critical damping: With the Jensen method the viscous roll damping is approximately accounted for by adding a percentage of critical damping to the estimated inviscid damping. It is inputted as a percentage.
- Wave spectrum: select one of the wave spectra mentioned in section 26.1.3 on page 462, Wave spectra.
- Number of speeds: a maximum of 50 can be given.
- Number of wave headings: a maximum of 100 can be given. A wave heading of 0 degrees means following waves, a wave heading of 90 degrees beam waves from portside, and a wave heading of 180 degrees signifies head waves.
- Radius of gyration kxx: the radius of gyration about the longtitudinal axis through the centre of gravity. An estimation method based on *IMO Intact Stability Code 2008 A.2.3.4* is available. For a more accurate estimation we recommend *Grin, R., Ruano, F.S., Bradbeer, N., Koelman, H., On the prediction of weight distribution and its effect on seakeeping, Proceedings of PRADS 2016, 4–8 September, 2016, Copenhagen, Denmark. pp. 227–235*, which is available on the `SARC website.`[1].
- Radii of gyration kyy and kzz: the radii of gyration about respectively the transverse and vertical axes through the centre of gravity. Estimations are available, based on the common estimation formula $k_{yy} = k_{zz} = 0.25 * L_{pp}$. For a more accurate estimation we refer to the paper mentioned above.
- Radii of gyration kyx, kyz and kxz: the radii belonging to the combinational products of inertia longitudinal-transverse, transverse-vertical and longitudinal-vertical respectively.
- Damping ratio: the ratio between the actual and critical damping of the free roll motion of the vessel. Also known as 'the damping coefficient'. The implementation is based on *el Moctar, B.O., Schellin, T.E. & Söding, H. (2021). Numerical Methods for Seakeeping Problems, Springer International Publishing*. The damping ratio is used to add viscous damping to the calculations of the strip theory method. After all, the strip theory is based on the potential flow theory, which does include damping, although not the so-called *viscous* damping (which is the effect of fluid friction). When applying the 'bare' strip theory, this may cause the real damping to be underestimated, notably for the roll motion. It is possible to measure the damping ratio, which is one of the reasons that this method has been implemented. If this parameter cannot be measured it can be estimated instead, for example with the formula from `ITTC procedure for numerical estimation of roll damping`[2]. As an illustration, the figure below shows the measured rolldamping of a 14000 TEU container vessel (from *Moctar, O.E., Shigunov, V. &Zorn, T. (2012). Duisburg Test Case: Post-panamax container ship for benchmarking. Ship Technology Research, 59(3), 50–64.*. Please notice that the values in the graph are in percent, while the input in Motions is in ratio (e.g. 1% is a ratio of 0.01).



Damping ratio of Duisburg Test Case.

26.2.2 Specify output

In these menus the points of interest and the specific output can be specified. The points of interest are the points for which specifc output can be defined in the program, such as the vertical accelerations and the probability that a defined threshold value is exceeded in the defined wave spectrum. The choice has been made to separate the definition of the points and the definition of the output, so that it is easier to define multiple outputs for a single point. In this way it is comparatively easy to define multiple threshold values for a single point, while it is also possible to specify the output of displacements and accelerations for that point.

One specific point of interest that is always defined is the center of gravity.

[1] `https://www.sarc.nl/publications/on-the-prediction-of-weight-distribution-and-its-effect-on-seakeeping/`
[2] `https://www.ittc.info/media/8151/75-02-07-045.pdf`

- For the strip theory method the location of the the center of gravity is derived from the hullform and the input parameters.
- For Jensen the longitudinal coordinate of the center of gravity is half the waterline length, as specified by the method.

26.2.2.1 Specify points of interest

In this menu the so-called 'point of interest' can be defined. A point is defined by a name, abbreviation and location coordinates. For the limited Jensen method, this location is only defined by a longitudinal position, the transverse position is always on the centerline and the vertical position is always on the waterline. For the strip theory based method all three coordinates are required.

26.2.2.2 Specify output at point of interest

In this menu the output for a point of interest is specified. First, after a new line has been created, a point of interest can be selected from a popup menu containing all of the points of interest defined in the previous menu. Alternatively, in the second column an abbreviation of one of the points of interest can be manually entered.

The column 'output type' opens a menu to select all the output for the point of interest. The selected (relative) displacements, velocities and accelerations are calculated on the specified location of selected point of interest. As only three of the six motion transfer function can be calculated using the Jensen method, and information about the phase angles of these motions are not calculated, only vertical motions on the centerline of the vessel can be calculated. A more complete calculation, where also the transverse and longitudinal motions, velocities and accelerations are calculated will be introduced with the strip theory method.

If a wave spectrum is defined in menu section 26.1.3 on page 462, Wave spectra, and only a single type of output has been selected, then a probability of exceedance of a defined threshold value of the output type against the wave spectrum can be calculated. When this option is selected, a threshold value must be defined. An acceptable probability of exceedance can optionally be defined, which is used to clarify the graph output. The unit of the threshold value naturally depends on the selected type of output, and is shown in the cell comment when the cell is selected. The acceptable probability is given in percentages (0%-100%).

26.2.2.3 Specify output options

In this menu the printing of the RAO tables can be enabled or disabled.

26.2.3 Calculate and print output

Using the parameters given in the first menu option, the output specified in the second menu option is calculated and printed as specified. A complete output contains:

- For each speed and wave heading an output table as specified in section 26.2.2.1 on this page, Specify points of interest. If the option to calculate probabilities of exceedance has been enabled these are also printed in a table and plotted in a polair diagram.
- For each speed and wave heading the RAO tables if these have been enabled in section 26.2.2.3 on the current page, Specify output options.
- An overview of the input that has been used to perform the calculations.

TABLE OF MOTION OUTPUT
DTC schip
16 Mar 2022 16:06:13

Point of Interest:

Name	Abbr.	Length	Breadth	Heigth
Bow	Bow	355.000	0.000	31.500

Speed: 0.000 knots
Wave heading: 150.000 degrees

Frequencies		Displacements							
Wave freq.	Enc. freq.	Long.		Trans.		Vert.		Vert. rel.	
		Ampl	Phase	Ampl	Phase	Ampl	Phase	Ampl	Phase
rad/s	rad/s	m	rad	m	rad	m	rad	m	rad
0.100	0.100	0.826	-1.566	0.487	-1.441	1.010	0.156	0.010	0.124
0.200	0.200	0.754	-1.575	0.493	-1.081	1.162	0.565	0.175	0.207
0.300	0.300	0.575	-1.595	0.502	-0.492	1.587	0.997	0.783	0.457
0.400	0.400	0.283	-1.622	0.586	-0.485	1.875	1.223	1.860	0.681
0.500	0.500	0.025	-1.510	0.315	0.368	1.158	1.282	2.089	1.048
0.600	0.600	0.050	1.878	0.077	2.647	0.449	-3.137	1.386	2.697
0.700	0.700	0.019	2.485	0.139	-1.762	0.579	-2.881	1.350	-2.144
0.800	0.800	0.014	-0.956	0.079	0.464	0.176	-0.501	1.090	0.462
0.900	0.900	0.004	1.902	0.052	2.539	0.044	-0.347	0.961	-2.990
1.000	1.000	0.001	-1.007	0.039	-1.079	0.055	2.549	0.954	-0.002
1.100	1.100	0.000	1.901	0.027	1.898	0.036	-0.639	0.974	-2.999
1.200	1.200	0.001	-2.546	0.017	-1.196	0.020	2.563	0.992	0.599
1.300	1.300	0.001	0.314	0.008	2.183	0.005	-0.100	0.999	-1.778
1.400	1.400	0.001	-3.019	0.002	1.631	0.006	-1.670	0.996	2.454
1.500	1.500	0.000	-1.925	0.008	-1.052	0.004	2.099	1.001	0.715
1.600	1.600	0.001	0.579	0.005	1.503	0.006	1.734	0.996	-0.709
1.700	1.700	0.000	1.361	0.006	2.489	0.004	-2.307	1.004	-1.825
1.800	1.800	0.000	0.304	0.004	-1.855	0.008	-1.272	1.002	-2.612
1.900	1.900	0.001	1.040	0.005	-0.623	0.009	-0.108	0.991	-3.102
2.000	2.000	0.001	1.962	0.005	0.766	0.009	1.312	0.999	3.001

Example of an output table of calculated displacements

TABLE OF RESPONSE SPECTRUM OUTPUT
DTC schip
16 Mar 2022 16:06:14

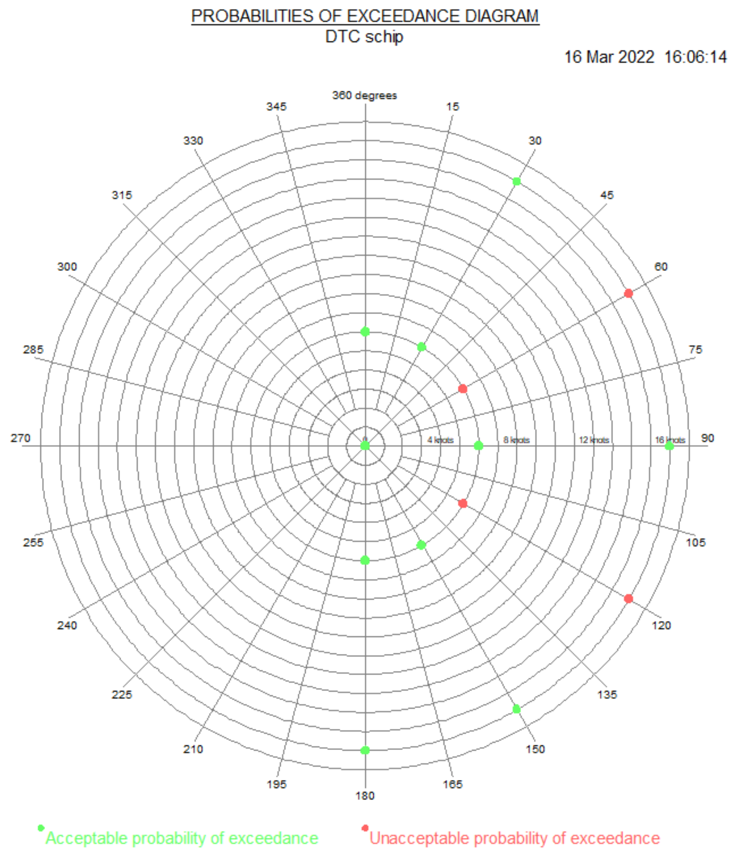
Point of Interest:

Name	Abbr.	Length	Breadth	Heigth
Bow	Bow	355.000	0.000	31.500

Response spectrum output for vertical displacement.
Probability of exceedance for a threshold value of 5.000 m.
A probability under 25.0% is deemed acceptable.

Speed	Wave heading	m0	m1	m2	RMS	Significant amplitude	Probablility of exceedance	Acceptable
kts	deg				m	m	%	Y/N
0.000	0.000	2.982	1.251	0.537	1.727	3.453	1.5	Yes
0.000	30.000	4.114	1.754	0.770	2.028	4.056	4.8	Yes
0.000	60.000	9.607	4.699	2.361	3.100	6.199	27.2	No
0.000	90.000	7.983	4.191	2.286	2.825	5.651	20.9	Yes
0.000	120.000	9.983	5.019	2.589	3.160	6.319	28.6	No
0.000	150.000	4.762	2.121	0.967	2.182	4.364	7.2	Yes
0.000	180.000	3.480	1.489	0.655	1.866	3.731	2.8	Yes
6.000	0.000	2.697	0.979	0.361	1.642	3.284	1.0	Yes
6.000	30.000	3.748	1.412	0.544	1.936	3.872	3.6	Yes
6.000	60.000	9.335	4.163	1.900	3.055	6.110	26.2	No
6.000	90.000	7.012	3.674	2.000	2.648	5.296	16.8	Yes
6.000	120.000	11.477	6.172	3.413	3.388	6.776	33.7	No
6.000	150.000	5.497	2.707	1.362	2.345	4.689	10.3	Yes
6.000	180.000	3.987	1.909	0.937	1.997	3.994	4.4	Yes
16.000	0.000	-	-	-	-	-	-	-
16.000	30.000	3.329	0.967	0.282	1.824	3.649	2.3	Yes
16.000	60.000	9.268	3.456	1.307	3.044	6.089	26.0	No
16.000	90.000	5.611	2.919	1.582	2.369	4.737	10.8	Yes
16.000	120.000	13.273	7.717	4.602	3.643	7.287	39.0	No
16.000	150.000	6.541	3.618	2.081	2.557	5.115	14.8	Yes
16.000	180.000	4.746	2.623	1.482	2.179	4.357	7.2	Yes

Example of a response spectrum output table



Example of an polar diagram of the acceptability of probabilities of exceedance

TABLES OF RESPONSE AMPLITUDE OPERATORS
DTC schip
16 Mar 2022 16:06:14

Speed: 0.000 knots
Wave heading: 150.000 degrees

Frequencies		RAOs											
Wave freq.	Enc. freq.	Surge		Sway		Heave		Roll		Pitch		Yaw	
		Ampl	Phase	Ampl	Phase	Ampl	Phase	Ampl	Phase	Ampl	Phase	Ampl	Phase
rad/s	rad/s	-	rad	-	rad	-	rad	rad/m	rad	rad/m	rad	rad/m	rad
0.100	0.100	0.854	-1.566	0.499	-1.577	0.998	-0.000	0.001	1.548	0.001	-1.569	0.000	0.005
0.200	0.200	0.864	-1.579	0.510	-1.602	0.959	-0.000	0.003	1.525	0.004	-1.607	0.001	-0.041
0.300	0.300	0.807	-1.602	0.701	-1.596	0.796	0.032	0.017	1.603	0.007	-1.621	0.003	-0.132
0.400	0.400	0.594	-1.670	0.225	-0.977	0.390	0.072	0.006	-3.070	0.010	-1.714	0.003	-0.123
0.500	0.500	0.213	-1.946	0.033	-1.624	0.178	2.303	0.002	2.188	0.006	-2.001	0.002	0.315
0.600	0.600	0.093	1.272	0.126	-2.949	0.313	2.303	0.005	0.593	0.002	0.773	0.000	-2.825
0.700	0.700	0.084	0.394	0.054	-1.887	0.050	-2.470	0.002	0.896	0.003	0.223	0.001	-1.912
0.800	0.800	0.025	-2.813	0.054	0.449	0.072	-1.944	0.002	-2.488	0.001	3.044	0.000	0.605
0.900	0.900	0.012	2.047	0.020	2.785	0.028	0.791	0.000	-0.032	0.000	2.134	0.000	2.508
1.000	1.000	0.010	-0.788	0.011	-0.932	0.009	-2.564	0.000	1.118	0.000	-0.759	0.000	-1.246
1.100	1.100	0.007	2.364	0.007	2.054	0.004	0.397	0.000	-1.881	0.000	2.400	0.000	1.742
1.200	1.200	0.003	-0.961	0.004	-0.645	0.003	-2.681	0.000	1.481	0.000	-0.726	0.000	-1.396
1.300	1.300	0.001	0.877	0.004	-3.061	0.003	0.684	0.000	0.646	0.000	2.536	0.000	1.738
1.400	1.400	0.001	2.545	0.004	0.317	0.002	-1.864	0.000	-2.333	0.000	1.532	0.000	2.468
1.500	1.500	0.001	-1.531	0.002	3.104	0.001	2.640	0.000	0.756	0.000	-1.214	0.000	-1.019
1.600	1.600	0.001	-0.858	0.003	-2.389	0.001	-0.449	0.000	0.235	0.000	-1.337	0.000	1.355
1.700	1.700	0.001	0.785	0.002	-0.409	0.001	0.117	0.000	1.368	0.000	0.654	0.000	2.663
1.800	1.800	0.002	1.765	0.002	0.989	0.001	1.575	0.000	2.096	0.000	1.838	0.000	-1.714
1.900	1.900	0.002	2.763	0.002	2.323	0.002	-2.996	0.000	-2.822	0.000	3.073	0.000	-0.525
2.000	2.000	0.001	-2.402	0.002	-2.973	0.002	-2.004	0.000	-1.618	0.000	-1.866	0.000	0.779

Example of a RAO output table

26.2.4 Filemanagement

With this option design versions can be managed, with a mechanism as described in section 2.9 on page 15, Data storage and backups.

Chapter 27

Loadline: freeboard calculation according to the Load Lines Convention

This module calculates the minimum freeboard according to the International Convention on Load Lines for type A or type B vessels.

27.1 Introduction

The following regulations from the ICLL are included:

- Chapter I (general):
  - regulation 3, § 1, 4, 5, 6, 7, 8 and 10.
  - regulation 4.
  - regulation 5.
  - regulation 6, § 1, 2a up to 2f.
- Chapter III (freeboards):
  - regulations 27 up to 31.
  - regulation 33.
  - regulation 34, § 1.
  - regulation 35, § 1 up to 3.
  - regulation 36, § 1g, 1h, 2 and 3.
  - regulation 37.
  - regulation 38, § 8 up to 12 and 14 up to 16.
  - regulation 39, § 1 and 5.
  - regulation 40, § 1, 3, 5 up to 7.

Prior remarks, and disclaimer:

- For the correct application of this module it is strongly advised to consult the regulations from the International Conference on Load Lines. This module does not pretend to make the use of the Load Lines Convention text redundant.
- The regulations which are not included in this module should be manually processed and accounted for.
- Parameter definitions in this module in general deviate from the PIAS standard, because the Load Lines Convention definitions prevail.
- **All units are in meters, except for the bow height and sheer, which should be given in millimeters.**

27.2 Main menu

Having started up loadline, one enters the main menu, the various options of which are explained in more detail in the following sections.

Freeboard calculation

1.	Main dimensions and other input parameters
2.	Superstructures
3.	Points of the sheer line
4.	Calculate freeboard with output to paper
5.	File and backup management

27.2.1 Main dimensions and other input parameters

The primary parameters are specified in this menu. A number of parameter can be derived from data that is already available in PIAS, that is to say, if a hull shape has been defined with Hulldef or designed with Fairway. If that is desired, in the last column 'from hullform' can be given (in contrast to 'own value', which allows the parameter to be entered manually). This is the same mechanism as used in Resistance to distill shape parameters from the hull form. Many parameters will speak for themselves or are defined in the Loadline Convention. Others are explained below.

27.2.1.1 Pre/post 2005 legislation

In 2005 the legislation has been revised, and this switch you can choose to apply either the old (pre 2005) or the present (post 2005) rules.

27.2.1.2 Depth

At the definition of the depth there is a difference between the setting before and after 2005. **Pre 2005** the depth can be given in two fashions:

- With the switch 'determination depth' set to 'manual': enter the depth according to regulation 3(6) as well as the depth at 85% of the minimum moulded depth.
- With the switch 'determination depth' set to 'constructed': enter the moulded depth, the thickness of the deck stringer plate and the thickness of the exposed deck sheating (T) according to regulation 3(6a). The depth at 85% of the moulded depth is then computed with the value of the moulded depth.

If the first method is used, the cells for thickness of the deck stringer plate and the thickness of the deck sheating and the moulded depth are grayed. If the second method is used and the thickness of the deck sheating does not equal zero then a questionmark is displayed at the depth if the total length of the superstructures is not yet known.

27.2.1.3 Bow height

The bow height is defined on the forward perpendicular, **in millimeters above the waterline corresponding to the assigned summer freeboard**. With setting 'pre 2005' if the bow height is given by the user it will be verified aginst the minimum required bow height. When a height of zero is entered, the program will compute the minimum bow height, which will be included on the output. With the 'post 2005' setting the bow height is no user input; it will simply be computed and printed by the program.

27.2.1.4 Miscellaneous freeboard parameters

- If the facility to derive the hull parameters from the hull form is not used — but, why should one not want to use that? — the waterline coefficient of the foreship can be found in extended hydrostatic table of Hydrotables.
- A reduction percentage on the base freeboard according to reg. 27 (the so-called B-60 freeboard) can be given, as well as the allowance according to the same regulation, however, both are mutually exclusive.
- At 'classification society' the two characters of the classification society or national authority can be given, those will be plotted into the Plimsoll mark. With this purpose the number of letters is limited to two: one at each side of the Plimsollmerkcircle.
- With 'plot Plimsoll mark' to 'yes' a plot of the Plimsoll mark is added to the tabular output.

27.2.2 Superstructures

In this menu the parameters of the superstructures can be given, where each superstructure should comply with the requirements of reg. 3(10). To determine the effective length (E), the breadth of each superstructure is tested to be at least 92% of the local breadth of the ship or 60% in the case of a trunk. The height of a forecastle or poop is measured at the perpendicular when the sheer correction should be applied. For determination of the effective length of a superstructure the height must be the minimum height of the superstructure, according to chapter 1, reg. 3(10). Superstructure parameters to enter are:

Name
: A textual description, just for identification.

Type superstructure
: This column defines whether the superstructure is a forecastle, poop, trunk, raised quarterdeck or other type of deckhouse ('general superstructure'). This distinction is necessary in relation to the regulations 31 and 38 (depth correction and sheer correction) and regulations 35 and 36 (effective length of superstructure).

Breadth ship
: For each superstructure, here the ship breadth at mid-length of the superstructure should be given. If the breadth of a superstructure equals the local breadth of the ship, you may choose to use [Breadth] in order to make the superstructure breadth equal to the local ship breadth.

Sheer correction
: If a poop or forecastle should be taken into account for determination of the sheer correction, thie column must be set to 'yes'

In eff.length
: If a superstructure should be taken into account for determination of the effective length of superstructure, which generally is the case, this column 'yes'.

Length>0.6L
: Should be set to 'yes' if the superstructure length is more than 0.6L. Only applicable to a 'general superstructure'.

Amidship
: Should be set to 'yes' if the superstructure extends over amidship. Also only applicable to a 'general superstructure'.

If only one superstructure is present, of the type *trunk*, the length of the trunk should be at least 0.60L to be taken into account. The length of a superstructure is the length within the length between perpendiculars according to reg. 3(2).

27.2.3 Points of the sheer line

Here a menu appears where the height of the sheer line can be given on the six standard ordinates — APP, 1/6↩ L_{PP}, 1/3L_{PP}, 2/3L_{PP}, 5/6L_{PP} en FPP. The standard sheer profile according to regulation 38(8) can be generated with [Standard sheer]. The sheer is measured **in millimeters**, as defined in reg. 38 (i.e. relative to a straight line, parallel to the construction waterline, passing through the sheer line at midship).

27.2.4 Calculate freeboard with output to paper

Will speak for itself, an example of the output is presented below.

CALCULATION OF FREEBOARD

Mv. Exempli Gratia

Calculation according the post 2005 criteria.

Type of ship	B	
Length waterline on 85% depth	95.000	m
Length stem - rudder stock	96.800	m
Length for freeboard calculation	96.800	m
Breadth	20.500	m
Depth moulded	8.350	m
Thickness deck stringer plate	0.015	m
Depth acc. freeboard	8.365	m
Volume at 85% depth	12625.000	m^3
85% depth moulded	7.098	m
Block coefficient 85% depth	0.896	
The waterline coefficient fore	0.891	
Displacement seawater at Tsummer	14100.000	ton
Tons/cm immersion salt water on Tsummer	15.100	ton
Reduction on base freeboard (reg. 27.8)	0	%
No allowance according (reg. 27.6)		

Superstructures

Name	Type superstructure	Length	Breadth	Height	Breadth ship	Sheercorrection	Superstructure length	Length>0.6L
Amidship								
Poopdeck	Poop	20.000	20.500	3.000	20.500	Yes	Yes	N.a.
F'castle	Forecastle	10.500	20.500	2.800	20.500	Yes	Yes	N.a.
RQD	Raised quarterdek	10.500	20.500	2.500	20.500	Yes	Yes	N.a.
Deck house	Superstructure	20.000	19.880	2.950	20.500	N.a.	Yes	No

Sheercorrection : Take into account for sheer credit acc. reg. 38 paragraph 12.
Superstructure length : Take into account for the effective length of superstructure acc. reg. 35.

Sheer (mm), Standard sheer profile according to regulation

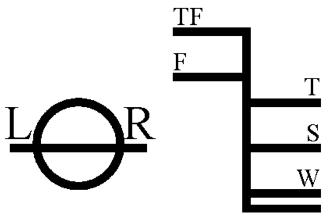
APP	1/6fAPP	1/3fAPP	1/3aFPP	1/6aFPP	FPP
1056.667	469.160	118.347	236.693	938.320	2113.333

The tabular freeboard is	1205.200	mm
The allowance (reg. 27.6) is	0.000	mm
The deduction (reg. 27.8) is	0.000	mm
The effective superstructure length (reg. 35) is	60.395	m
The allowance for L<100 m. (reg. 29) is	0.000	mm
The block coefficient is	0.896	
Freeboard	1205.200	mm
Correction for the block coefficient	1.159	
Depth-correction	385.520	mm
Freeboard after correction for depth (reg. 31)	1782.479	mm
Superstructure-correction	-506.413	mm
Sheer-correction	-58.040	mm
Freeboard according to the regulation is	***1218.026***	mm
The required minimum bowheight (reg. 39.1) is	4398.466	mm
The required minimum area (reg. 39.5) is	42.247	m^2

	Freeboard m	Draft m
Trop. fresh	0.836	7.529
Summer	1.218	7.147
Tropical	1.069	7.296
Winter	1.367	6.998
Winter NA	1.417	6.948
Freshwater	0.985	7.380

Freeboard computation details in the output.

	Freeboard	Draft
	m	m
Trop. fresh	0.836	7.529
Summer	1.218	7.147
Tropical	1.069	7.296
Winter	1.367	6.998
Winter NA	1.417	6.948
Freshwater	0.985	7.380
Scale 1/10		



Plot of Plimsollmerk in the output.

27.3 File and backup management

Backups of the input data can be made and restored here. Here is also the option ‘Stop without saving’. See for the details section 2.9 on page 15, Data storage and backups.

Chapter 28

Incltest: inclining test or light weight check report

This module generates an inclining test report or a light weight check report. After entering the measured data of the test, the lightship weight is calculated including the position of the CoG (Center of Gravity). In this manual, mainly the operation of the program will be discussed, altough in particular the last section is brought under your attention, for that is the place where some computational backgrounds are discussed.

Inclining test or light weight check

28.1 General data and settings

Here the general data of the inclination test or light weight check can be given, from which quite some will speak for themselves. That is just textual input — such as for 'Condition outside water' or 'Number of persons present' — which can be given and which will be printed on the measurement report. However, a number of calculation settings require some elucidation:

Density of outside water

This is the density (the specific weight) of the outside water (in ton/m^3) during the inclination test or light weight check, so it is not related to the design density as given in section 5.1.6 on page 43, Density outside water.

Inclining test or light weight check

Here is defined what type of calculation is made and thus what type of report is generated. Furthermore it should be noted that, when an inclining test is selected but only the zero measurement is defined or there are no test weight defined at the zero measurement, that automatically a light weight check is calculated.

Calculation with correction for sagging

With this switch the method of calculation of the volume (and hence the weight during the inclining test or light weight check) and Longitudinal Center of Buoyancy can be chosen. Obviously, sagging can only be taken into account when drafts/freeboards have been measured at at least three locations. At a calculation *without* correction for sagging a *straight* waterline is determined as close as possible to the measured drafts. At a calculation *with* this correction a *parabolic* waterline is constructed. In both cases the waterplane construction is done with the least squares method.

Correction of trim & VCG on CoG

Please read for the background of this issue first the discussion at 'With free-to-trim effect, including the effect of VCG on trim', see section 5.1.3 on page 42, Stability calculation method. The point is that trim & VCG on the one hand, and LCG & TCG on the other hand have a mutual relationship. In conventional calculations this effect was taken into consideration, but here in PIAS it can be incorporated. If one should wish the results of the elaboration of the inclining test to be fully compatible with those of the loading conditions, then this setting here in Incltest should be set the same as the referred setting in Config. **Please note** that if the ship has a heel angle during the light weight check then it is advised to use this option.

VCG for correction of trim & VCG on CoG
If this 'Correction of trim & VCG on CoG' is switched on, the VCG should obviously be known. When elaborating an inclining test that is always the case, because then the VCG is just computed! However, when **only a light weight check** is done, then the VCG is not known, so it must be user-specified, which can be done here. If the VCG is not known exactly, then one should be satisfied with an estimation.

Processing the measurements in an inclining test.
If the vessel's hydrostatics remain constant throughout the inclining test, the 'Conventional' option can be selected, whereby the overall G'M is determined in 1 time from all measurements using the least squares method, and the VCG is calculated using the KM of the zero measurement. If the hydrostatics are not constant, the 'Non-conventional' option can be chosen where the VCG is calculated per measurement with the actual KM and FSM of that measurement. The KM may differ if there is a difference in waterline surface even at small angles, and the FSM may start to differ if tanks are used as inclining test weights. Also, when tanks are used, the displacement may not be constant, and the VCG of the inclining test weights itself may change. The latter may also be applicable for fixed inclining test weights.

Note

For a calculation to be submitted at Bureau Veritas, the "Conventional" option should be used, as this has been shown to be the only method suitable for a comparison with their software (status 2024).

Determination of G'M
At an inclining test, multiple measurements of G'M are performed. From these measurements the single resulting G'M must be determined. The probably most evident method is to take the mean of all measurements. An alternative comes up if we plot the measured angles of inclination against the heeling moments — such a graph is included in the inclining test report as produced by Incltest. If, subsequently, a straight line is drawn through these points, the inclination of this line represents G'M. That straight line is being determined by the least squares method, so this is also the name of this method. If the option 'Conventional' is selected, this method will always be used.

Selected wind contour for pictures in report
The output of Incltest may contain a side view of the ship, where draft, trim and sagging are depicted. For such a sketch a wind contour can be used — as discussed in section 7.2.6 on page 182, Wind contour — and at this setting one may specify *which* contour to use for this purpose.

Trim and heeling angle for sounding/ullage of tanks
Here can be given which trim and heel are to be used for the determination of tank volumes based on measured sounding or ullage. As a rule, these will be according the drafts, which have only been recorded once, at the zero measurement, in which case the setting will be 'Taken from zero measurement'. In practice, it might occur that preparations are not yet ready for the zero measurement, while there is time available to gauge the tanks. For such a case the alternative setting 'As defined below' is available. This choice will enable the last two rows of this menu, where trim (in meters) and heeling angle (in degrees, positive to SB) can be given as they have been recorded during gauging of the tanks. In the case that 'Taken from zero measurement' is used then the last two rows will show there respective values from the zero measurement.

28.2 Inclination measuring equipment particulars

This is a simple form where inclination gauging particulars can be given. PIAS support ten such devices, of three types:

- A conventional pendulum, from which name and length should be specified.
- An (electronic) inclinometer — an inclination gauge — e.g. the one as integrated in PIAS through module Inclmeas. From this equipment only the name has to be given.
- A tube, from which name and distance between ends should be specified.

28.3 Measurements

This table contains the heart of the matter, here the measurements are being entered in PIAS. The first row is always present in this table, which is the so-called "zero measurement", which fixes the initial condition. For each measurement there is one line, which contains:

- The name of the measurement.
- Whether the drafts of this measurement deviate from the zero measurement. For the zero measurement the drafts or freeboards obviously have to be given, because, as a rule, they are valid for the entire test. Deviant drafts are a bit of an exception (but will be processed in PIAS properly, if being given). This option is available only if 'Non-conventional' is selected for measurement processing.
- Whether or not one whishes to use 1 total moment. In the majority of cases it will probably most convenient to give for each individual test weight its position, and to let PIAS add their heeling moments. In such a case a user does **not** give 1 total moment.
- If one does wish to give 1 total moment, then in the 4^{TH} column that moment (in tonmeter) can be specified relative to centerline.
- In the last columns the measured inclination angles are being given. Either as angle in degrees, in case of an inclinometer, or as the stroke **in meters** of a pendulum or tube. In the case of a tube, the measured values for the PS and SB side are given separately. These angles and strokes should be given as the actual measured values. It is also possible to 'Reject' a measurement; the measurement is then not used in the determination of the G'M.
- In due time, this input form will be equipped with a function to import and utilize inclination angles as measured by Inclmeas. The graph of the measured angle vs. time will then also be included in the inclination test report.

With the text cursor on the row of a measurement, one can with <Enter> continue to input forms of freeboards/drafts and inclining test weights, as dicussed just below.

28.3.1 Measured freeboards/drafts

This is the input form of freeboards and/or drafts, which contains seven columns:

- The first column is called 'Measuring point', where the name of this measuring point can be given.
- In the second and third column the longitudinal and transverse positions of the measuring point are given.
- In the fourth column, 'Ref.height from base', the reference height of the measuring point is given.
- In column five 'Measured type' is given, in case of a 'Reference point' it can be specified, if the measured value is defined as a 'Freeboard' or 'Draft'. In case of a draftmark only a 'Draft' can be used.
- In column six the 'Measured value' can be given. If a draft is entered, it will be added to the reference height (at the longitudinal and transverse position of this measuring point), and if a freeboard is given it will be subtracted from the reference height. In this fashion the measured position of the waterline at this measuring point is specified.
- In seventh column the draft form base is shown. This only for your information and cannot be modified.

With [New] one always creates a 'Reference point'. For reading on a draft mark the function [Draft marks] is available. With this function one can read draft marks, as defined in PIAS as discussed in section 7.2.1.4 on page 168, Draft marks and allowable maximum and minimum drafts. If the specific measuring point is a draft mark then the columns 1 to 5 have been filled with data from the defined draft mark and for that reason cannot be modified.

28.3.2 Locations of inclining test weights

In this form the inclining test weights are defined. At the zero measurement inclining test weights can be added or removed, in all the other measurements only specific data can be changed. Inclining test weights which have been create in the zero measurement are available in all the other measurements.

In PIAS two types of inclining test weights are supported, a conventional solid weight (such as a block of concrete or a barrel filled with water), or a tank. From each inclining test weight should be given:

- The name, preferably unique.
- The type (solid weight or tank. In the latter case it must have been defined in PIAS module Layout).
- The weight in ton, in case of a solid weight.
- If this test weight was on board during draft reading.
- A further textual description.

With [New] one always creates a test weight of type 'solid weight'. For adding weights of type 'tank' the function [Tank list] is available. Here is a list of all the tanks that can be used as a test weight. This function does the same as the function with the same name in Loading — which is discussed in section 16.2.1 on page 307, Define/edit weight items under function [Misc].

The particulars to be entered for a specific type of test weight are:

- From a (conventional) solid weight the longitudinal, transverse and vertical position of its CoG is given, relative to APP, centerline and baseline. The 'weight', in metric tons, of the solid weight has to be defined only once at the zero measurement.
- From a tank used as inclining test weight the (weight-) content can be given. Here the user has the choice to use the most convenient parameter from weight, percentage of filling, volume, sounding or ullage. Sounding and ullage are only available if the tank in question has a properly defined sounding pipe, see paragraph 9.5.1.2.8 on page 217, Sounding pipe.

28.4 Weights to be added, subtracted or relocated

When conduction an inclining test or light weight check, not each and every weight will be on its final position. In practice always some weight adjustments will have to be made, commonly called *weights to add or to deduct*. That can be done by this menu option, which is subdivided in three catagories.

28.4.1 Weights to be added to lightship

Weight to be added to light ship, where for each weight item the name, weight and CoG should be given.

28.4.2 Weights, present on-board during the test, not belonging to light ship

Weight, present on board during the test, not belonging to light ship. Just as with the previous category this can be simply unconnected items, with name, weight and CoG. However, one can also use the function [Tank list], as described in section 28.3.2 on the preceding page, Locations of inclining test weights, to add weights of type *tank*, as defined in PIAS (with Layout). If feasible, this feature is certainly convenient to use, because then giving just a percentage of filling, or sounding or ullage will suffice, and be used to calculate volume and CoG of the tank content, taking into account heel and trim.

28.4.3 Weights, present on-board during the test, but not on their final position

Weights, present on board during the test, but not yet on their final position. In this menu for each item the weight should be given, as well as its position during the test as well as the final position in light ship.

28.5 Print measurement report

Prints the inclining test or light weight check report, below an example is included of the output with the option 'Non-conventional'.

INCLINING TEST REPORT
M.v. Exempli Gratia

26 Jun 2017 16:46:10

General data

Description	Final inclining test, ship ready for delivery
Ship's name	Exempli Gratia
Type of ship	Multi Purpose
Date	December 8, 2016
Location	Bussum harbour, The Netherlands
Customer	SARC BV
Number of persons present	2
Person 1	Mr. Charles Magne
Person 2	Ir. Jan Salie

Environmental conditions during test

Condition outside water	Waves 0.1 - 0.3 m
Current	None
Windforce	Approx. 3 Beaufort
Wind direction relative to the ship	Wind from the stern
Specific weight of outside water	1.0000 ton/m^3
Water depth	Sufficient

Situation of the ship
Moored along quay, mooring lines slack

Settings for calculations

Inclining test or light weight check	Inclining test
Calculation with correction for sagging	Yes
Correction of trim & VCG on COG	Yes
Determination of G'M	Least square

Measuring instruments

Name	Type	Length pendulum [m]
Pendulum aft	Pendulum	5.432
Inclining gauge	Inclinometer	n.a.

Used inclining test weights

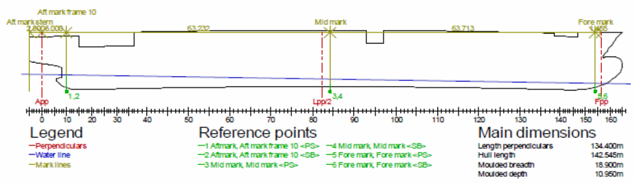
Name	Type of weight	Weight	Aboard during light weight check	Description
Inclining test weight 1	Solid weight	15.100	No	Steel block
Inclining test weight 2	Solid weight	15.121	No	Steel block
Inclining test weight 3	Solid weight	14.980	No	Concrete 160 x 160 x 140 cm
Inclining test weight 4	Solid weight	15.212	No	Steel block
Total		60.413		

Zero measurement

NO inclining test weights were aboard during draft or freeboard measurement.

Measured drafts and/or freeboards

Name	Length	Breadth	Ref. point from base	Measured type	Measured value	Draft from base
	m	m	m		m	m
Aftmark, Aft mark frame 10 <PS>	6.000	-0.269	0.000	Draft	3.910	3.910
Aftmark, Aft mark frame 10 <SB>	6.000	0.275	0.000	Draft	3.890	3.890
Mid mark, Mid mark <PS>	69.232	-9.450	0.000	Draft	1.910	1.910
Mid mark, Mid mark <SB>	69.232	9.450	0.000	Draft	1.890	1.890
Fore mark, Fore mark <PS>	132.945	-1.433	0.000	Draft	2.010	2.010
Fore mark, Fore mark <SB>	132.945	1.418	0.000	Draft	1.990	1.990



Calculated drafts, with the "least square" method (parabolic)

Lpp	134.400	m
Draft App	4.183	m
Draft at 1/2 Lpp	1.917	m
Draft Fpp	2.012	m
Trim	-2.171	m
Inclination	-0.070	degrees
Sagging	-1.181	m

Which result in (NO inclining test weights aboard)

Volume & appendages	3867.286	m^3
Displacement ==> S.W.=1.0000 ton/m^3	3867.286	ton
KM transverse	14.686	m
Longitudinal center of buoyancy	61.699	m
Transverse center of buoyancy	-0.016	m
Vertical center of buoyancy	1.200	m

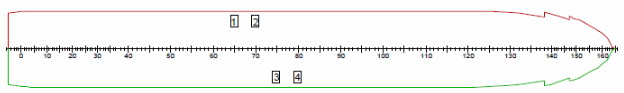
Which result in (ALL inclining test weights aboard)

Volume & appendages	3927.742	m^3
Displacement ==> S.W.=1.0000 ton/m^3	3927.742	ton
KM transverse	14.531	m
Longitudinal center of buoyancy	61.634	m
Transverse center of buoyancy	-0.016	m
Vertical center of buoyancy	1.218	m

Inclining test weights

No.	Name	Weight	VCG	LCG	TCG	FSM	Type of weight
		ton	m	m	m	tonm	
1	Inclining test weight 1	15.100	15.320	50.000	-6.900	0.000	Solid weight
2	Inclining test weight 2	15.121	15.320	55.000	-6.900	0.000	Solid weight
3	Inclining test weight 3	14.980	15.320	60.000	6.900	0.000	Solid weight
4	Inclining test weight 4	15.212	15.320	65.000	6.900	0.000	Solid weight
	Total	60.413	15.320	57.508	-0.003	0.000	

Positions of inclining test weights



Measurement 1

Inclining test weights

No.	Name	Weight ton	VCG m	LCG m	TCG m	FSM tonm	Type of weight
1	Inclining test weight 1	15.100	15.320	50.000	6.900	0.000	Solid weight
2	Inclining test weight 2	15.121	15.320	55.000	6.900	0.000	Solid weight
3	Inclining test weight 3	14.980	15.320	60.000	6.900	0.000	Solid weight
4	Inclining test weight 4	15.212	15.320	65.000	6.900	0.000	Solid weight
	Total	60.413	15.320	57.508	6.900	0.000	

Measured drafts and/or freeboards

Name	Measurement	Unit	G'M [m]	VCG [m]
Pendulum aft	0.1890	m	3.052	11.484

$$G'M = \frac{(60.413 \times 6.900 - 60.413 \times -0.003) \times 5.432}{3927.742 \times (0.1890 - 0.0000)} = 3.052 \text{ m}$$

VCG = 14.536 - 3.052 - (4.313 / 3927.742) = 11.484 m

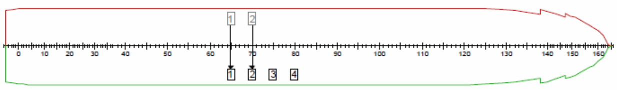
Name	Measurement	Unit	G'M [m]	VCG [m]
Inclining gauge	1.9980	degrees	3.045	11.490

$$G'M = \frac{(60.413 \times 6.900 - 60.413 \times -0.003) \times 57.296}{3927.742 \times (1.9980 - 0.0000)} = 3.045 \text{ m}$$

VCG = 14.536 - 3.045 - (4.313 / 3927.742) = 11.490 m

Shifts/moment

Grey weights are weights from measurement: Zero measurement



Measurement 2

Inclining test weights

No.	Name	Weight ton	VCG m	LCG m	TCG m	FSM tonm	Type of weight
1	Inclining test weight 1	15.100	15.320	50.000	-6.900	0.000	Solid weight
2	Inclining test weight 2	15.121	15.320	55.000	-6.900	0.000	Solid weight
3	Inclining test weight 3	14.980	15.320	60.000	6.900	0.000	Solid weight
4	Inclining test weight 4	15.212	15.320	65.000	6.900	0.000	Solid weight
	Total	60.413	15.320	57.508	-0.003	0.000	

Measured drafts and/or freeboards

Name	Measurement	Unit	G'M [m]	VCG [m]
Pendulum aft	0.0045	m	3.126	11.403

$$G'M = \frac{(60.413 \times -0.003 - 60.413 \times 6.900) \times 5.432}{3927.742 \times (0.0045 - 0.1890)} = 3.126 \text{ m}$$

VCG = 14.531 - 3.126 - (4.313 / 3927.742) = 11.403 m

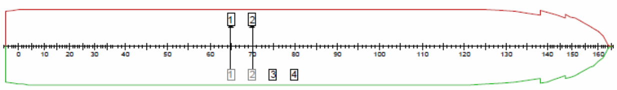
Name	Measurement	Unit	G'M [m]	VCG [m]
Inclining gauge	0.0550	degrees	3.131	11.398

$$G'M = \frac{(60.413 \times -0.003 - 60.413 \times 6.900) \times 57.296}{3927.742 \times (0.0550 - 1.9980)} = 3.131 \text{ m}$$

VCG = 14.531 - 3.131 - (4.313 / 3927.742) = 11.398 m

Shifts/moment

Grey weights are weights from measurement: Measurement 1



Measurement 3

Inclining test weights

No.	Name	Weight ton	VCG m	LCG m	TCG m	FSM tonm	Type of weight
1	Inclining test weight 1	15.100	15.320	50.000	-6.900	0.000	Solid weight
2	Inclining test weight 2	15.121	15.320	55.000	-6.900	0.000	Solid weight
3	Inclining test weight 3	14.980	15.320	60.000	-6.900	0.000	Solid weight
4	Inclining test weight 4	15.212	15.320	65.000	-6.900	0.000	Solid weight
	Total	60.413	15.320	57.508	-6.900	0.000	

Measured drafts and/or freeboards

Name	Measurement	Unit	G'M [m]	VCG [m]
Pendulum aft	-0.1920	m	2.932	11.603

$$G'M = \frac{(60.413 \times -6.900 - 60.413 \times -0.003) \times 5.432}{3927.742 \times (-0.1920 - 0.0045)} = 2.932 \text{ m}$$

VCG = 14.537 - 2.932 - (4.313 / 3927.742) = 11.603 m

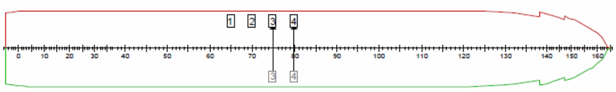
Name	Measurement	Unit	G'M [m]	VCG [m]
Inclining gauge	-2.0200	degrees	2.929	11.607

$$G'M = \frac{(60.413 \times -6.900 - 60.413 \times -0.003) \times 57.296}{3927.742 \times (-2.0200 - 0.0550)} = 2.929 \text{ m}$$

VCG = 14.537 - 2.929 - (4.313 / 3927.742) = 11.607 m

Shifts/moment

Grey weights are weights from measurement: Measurement 2



Measurement 4

Inclining test weights

No.	Name	Weight ton	VCG m	LCG m	TCG m	FSM tonm	Type of weight
1	Inclining test weight 1	15.100	15.320	50.000	-6.900	0.000	Solid weight
2	Inclining test weight 2	15.121	15.320	55.000	-6.900	0.000	Solid weight
3	Inclining test weight 3	14.980	15.320	60.000	6.900	0.000	Solid weight
4	Inclining test weight 4	15.212	15.320	65.000	6.900	0.000	Solid weight
	Total	60.413	15.320	57.508	-0.003	0.000	

Measured drafts and/or freeboards

Name	Measurement	Unit	G'M [m]	VCG [m]
Pendulum aft	-0.0050	m	3.081	11.448

$$G'M = \frac{(60.413 \times -0.003 - 60.413 \times -6.900) \times 5.432}{3927.742 \times (-0.0050 - -0.1920)} = 3.081 \quad m$$

VCG = 14.531 - 3.081 - (4.313 / 3927.742) = 11.448 m

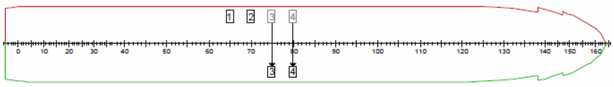
Name	Measurement	Unit	G'M [m]	VCG [m]
Inclining gauge	-0.0200	degrees	3.039	11.491

$$G'M = \frac{(60.413 \times -0.003 - 60.413 \times -6.900) \times 57.296}{3927.742 \times (-0.0200 - -2.0200)} = 3.039 \quad m$$

VCG = 14.531 - 3.039 - (4.313 / 3927.742) = 11.491 m

Shifts/moment

Grey weights are weights from measurement: Measurement 3



Overview measurements

No.	Name	Moment tonm	G'M1 m	VCG1 m	G'M2 m	VCG2 m
1	Measurement 1	417.050	3.052	11.484	3.045	11.490
2	Measurement 2	-417.050	3.126	11.403	3.131	11.398
3	Measurement 3	-416.650	2.932	11.603	2.929	11.607
4	Measurement 4	416.650	3.081	11.448	3.039	11.491

Measuring instruments
1 Pendulum aft
2 Inclining gauge

G'M determination with a pendulum as measuring instrument:

$$G'M = \frac{(\text{inclining test weight}(N) \times TCG(N) - \text{inclining test weight}(N-1) \times TCG(N-1)) \times \text{pendulum length}}{\text{displacement} \times (\text{pendulum stroke}(N) - \text{pendulum stroke}(N-1))}$$

G'M determination with an inclinometer as measuring instrument:

$$G'M = \frac{(\text{inclining test weight}(N) \times TCG(N) - \text{inclining test weight}(N-1) \times TCG(N-1)) \times \text{radian}}{\text{displacement} \times (\text{inclination}(N) - \text{inclination}(N-1))}$$

- Applicable to measurement (N), with respect to the previous measurement (N-1).

VCG determination:
VCG = KM - G'M - FSM correction = KM - G'M - (FSM / displacement)

- See manual for further explanation.

Weights to be added

Name	Weight ton	VCG m	LCG m	TCG m
COUPLING ME-GB	2.092	3.720	13.100	0.000
COUPLING GB-PTO	0.150	4.250	10.550	0.000
LIFE RAFT FWD	0.050	14.150	127.000	-2.160
LIFE RAFTS AFT	0.300	17.000	5.920	0.000
EMBARKATION LADDERS	0.200	17.200	8.440	0.000
GALLEY EQUIPMENT	0.750	14.750	6.600	-0.750
PILOT CHAIRS	0.150	25.750	6.100	0.000
SHELVES FORECASTLE	0.500	12.300	135.000	0.000
FILLING CW SYSTEM	2.400	5.000	16.400	0.000
WORKSHOP TOOLS	0.500	7.150	12.000	7.500
spare parts in f'castle	0.500	12.000	135.000	0.000
metal sheeting exhaust pipe	0.200	9.750	6.000	-0.600
metal sheeting E.R.	0.300	6.220	22.200	0.000
floorplates pumproom	0.300	1.600	80.000	0.000
floorplates BTRoom	0.300	1.600	126.000	0.000
linoleum	0.500	14.200	6.000	0.000
furniture	1.000	14.800	6.000	0.000
paint superstructure outside	0.618	20.300	6.300	0.000
paint superstructure inside	0.216	20.300	6.300	0.000
paint double bottom hold	0.156	0.650	53.600	0.000
paint maind, poopd, fcd, cc	0.485	12.000	68.000	0.000
paint E.R.	0.232	6.850	13.200	0.000
paint hatch covers	0.322	13.950	70.200	0.000
medical oxigen bottle	0.050	17.250	-0.300	-4.890

safety equipment	1.000	12.000	5.500	0.000
outboard motor mob boat	0.050	17.060	0.630	-8.190
tow line	0.250	11.400	134.500	0.000
fastenings hatches	0.500	13.620	70.000	0.000
hoisting boxes (2)	0.200	11.700	65.000	0.000
crates (6)	0.300	11.500	129.500	0.000
bolts propellor flange (12)	0.120	2.600	10.260	0.000
luboil gearbox	0.990	1.850	11.650	0.000
filling contr. pitch prop.	0.300	3.000	13.200	0.000
seachest PS	8.026	3.815	19.232	-7.069
seachest SB	8.901	3.858	18.960	7.031
Total	32.908	6.475	25.580	0.248

Weights to be subtracted

Name	Weight ton	VCG m	LCG m	TCG m	FSM tonm	Type of weight	Filling %	S.W. ton/m³	Volume m³	Sounding m	Ullage m	Measured
7 LT 2 WB PS	203.443	4.009	102.805	-7.677	0.000	tank	100.000	1.0250	198.481	9.537	1.806	Filling%
21 WT 5 WB PS	23.652	1.851	48.894	-8.675	4.313	tank	14.000	1.0250	23.075	1.119	8.712	Filling%
temporary scaffolding	11.000	13.050	66.200	0.000	0.000	Solid weight	-	-	-	-	-	-
accumulator	0.450	16.850	-1.200	-8.200	0.000	Solid weight	-	-	-	-	-	-
cables	0.510	13.000	9.000	0.000	0.000	Solid weight	-	-	-	-	-	-
connection box (10)	0.100	12.000	68.000	0.000	0.000	Solid weight	-	-	-	-	-	-
lamps (100)	0.100	12.000	68.000	0.000	0.000	Solid weight	-	-	-	-	-	-
floodlight (3)	0.100	9.000	80.000	0.000	0.000	Solid weight	-	-	-	-	-	-
rubber air hoses	0.600	11.000	80.000	0.000	0.000	Solid weight	-	-	-	-	-	-
people on fc deck	0.320	15.700	130.300	0.000	0.000	Solid weight	-	-	-	-	-	-
people on hatches	0.880	15.320	64.000	0.000	0.000	Solid weight	-	-	-	-	-	-
people on aft ship	0.640	14.750	3.000	0.000	0.000	Solid weight	-	-	-	-	-	-
Total	241.794	4.364	95.011	-7.323	4.313							

Tanks are calculated with:
Trim : -2.171 m
Inclination : -0.070 degrees

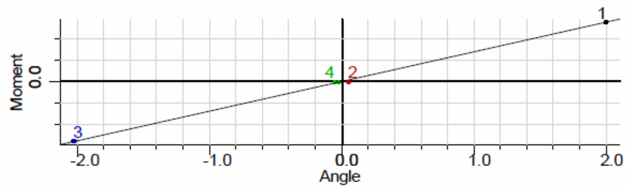
Already onboard but not yet at the correct position

Name	Weight ton	Old VCG m	Old LCG m	Old TCG m	New VCG m	New LCG m	New TCG m
SPARE PARTS MAINDECK	2.590	11.500	14.500	-0.300	7.600	13.200	4.820
Total	2.590	11.500	14.500	-0.300	7.600	13.200	4.820

Light ship weight

	Weight ton	VCG m	LCG m	TCG m	FSM tonm
Ship during inclining test	3927.742	11.491	61.800	-0.004	
Total inclining test weights	-60.413	15.320	57.508	-0.003	0.000
Total weights to be subtracted	-241.794	4.364	95.011	-7.323	4.313
Total to be added weights	32.908	6.475	25.580	0.248	
Total to be moved weights (old position)	-2.590	11.500	14.500	-0.300	
Total to be moved weights (new position)	2.590	7.600	13.200	4.820	
Light ship weight	3658.443	11.851	59.349	0.486	

Plot of heeling moment vs. angle of inclination



28.6 File management

Backups of inclining test data can be be made and restored here, please refer to section 2.9 on page 15, Data storage and backups for more details.

28.7 Print pre-2017 measurement report

Prints an inclining test report or light weight check report, according to the method as being applied by PIAS until december

1. For an elucidation on the differences between the methods of then and now, reference is being made to section 28.8.2 on the following page, Compatibility with the pre-2016 program version. This option is included for backward compatibility reasons only; now lacking the pre-2017 program version, there would otherwise be no possibility to produce a test report with an elder data file, while with this option there is. Obviously, this option does not work when the input data haven been defined with the contemporary program version.

28.8 Background and computation method

28.8.1 A brief elucidation on the computation of GM and KG

The following applies to the "Non-conventional" option, for the "Conventional" option the calculation of the G'M and VCG is in the output.

For a pendulum measurement applies:

$$GM = \frac{(Testweight_N.TCG_N - Testweight_{N\text{-}1}.TCG_{N\text{-}1}).Pendulumlength}{Displacement.(Pendulumstroke_N - Pendulumstroke_{N\text{-}1})}$$

and for an inclinometer:
$GM = \frac{(Testweight_{N}.TCG_{N} - Testweight_{N-1}.TCGN-1).1Radian}{Displacement.(Inclinationangle_{N} - Inclinationangle_{N-1})}$

- Applicable to measurement N, with respect to the previous measurement (N-1).
- For measurement N=1 the previous measurement is the zero measurement (after all, N-1 is zero in that case).
- TCG is the transverse center of gravity of the inclining test weights. Test weight (ton) and TCG (m) are taken from the **total** of inclining test weights.

The vertical center of gravity (VCG) is computed by:
$VCG = KM - G'M - GG' = KM - G'M - \frac{FSM}{displacement}$

where:

- The KM is determined at the draft and trim of the zero mesurement and the angle of inclination of the current measurement (unless one or more measurements independent drafts have been given, in which case the KMs for those measurements are determined for those drafts).
- G'M is that from the measurement itself.
- The free surface moment FSM is the total of all FSMs (so, from the weights to be decuted, as well as potentially the inclining test weights).
- The VCG of 'ship during inclining test' (in the final ouput sheet) is computed with the average of all VCGs (of all measurements and all measuring devices), or with the least squares method, dependant on the applied setting.
- Furthermore, in a inclining test with solid test weights it is assumed that the light ship VCG (including the test weights) is constant during the test. However, if (also) tanks are in use as test weights then this assumption would not be realistic, and will the VCG of the ship (excluding test weights) be computed per individual measurement.
- Because small differences in KM, displacement, trim, etc. between the measurements are included, this can produce a small difference to the classical method, which assumes that these data are constant throughout the test.

28.8.2 Compatibility with the pre-2016 program version

The design of the original version of Incltest was drawn up around 1988. In the course of the years, quite some changes and extensions have been made, but the basic structure has roughly remained the same. Comments and desires of many users have been collected over the years, and haven been used in the implementation of a completely new module which replaced the elder one in December 2016. Many of these changes do not affect the basic methodology of input and calculation, however, there is one essential difference that needs to be explained in more detail:

In the elder Incltest module the movements of the inclining test weights as well as the pendulum stroke or measured inclination were given **with respect to the previous measurement**. In this new module the positions of the inclining test weights are given in the ship's coordinate system, and the stroke / inclination relative to the **zero measurement**. With old inclining test data, movements and pendulum strokes are in first instance interpreted in the old way, so that the calculation yields the same results as previously. All the old inclining test data can be found in section 28.6 on the previous page, File management. Old measurement reports, if available, can be printed with the menu option section 28.7 on the preceding page, Print pre-2017 measurement report. If one would like to use the old inclining test data then one can *restore* the data with use of section 2.9.3 on page 15, Restore data from backup. The restored data is then immediately interpreted in the new fashion, and the calculations will (consequently) no longer be correct. One should now re-enter movements and strokes in the new convention.

It is not possible to, proceed according the old method, and newly defined inclining test data cannot be interpreted upon the old method.

Chapter 29

Inclmeas: registration and processing of digital inclination measurement

This module measures the inclination during the inclining test. For every measurement you can specify the translation and size of the test weight. The measurements can be exported to the inclining test report module. Furthermore it is possible to present a graphical representation on the screen or send it to a printer.

29.1 Guidelines for installation

The inclination sensor is equipped with a USB cable, which is used to connect the sensor to the computer. It is required to install the driver which belongs to the sensor. The sensor can be placed in an arbitrary place in the ship, however, the underground should be firm. During measurement the sensor should not wiggle or be relocated.

Digital inclining measurement

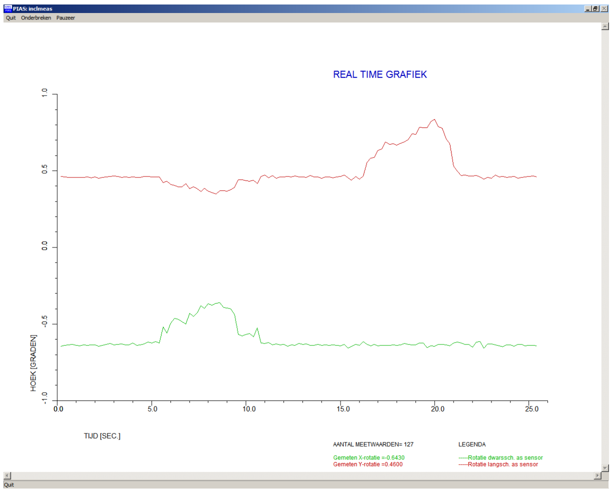
29.2 Main menu

Digital inclining measurement

1.	Test measurements
2.	Inclination measurement
3.	Last measurement again
4.	Remove all saved measurements
5.	Output of measurements to screen
6.	Output of measurements to printer
7.	Output of measurements to ASCII file
8.	Settings of digital inclinometer

29.2.1 Test measurements

The module reads out the serial port and the results are printed on screen. The presented angles are the angles of both axes of the measurement gauge box. The range of the gauge is -5 to + 5 degrees. Thus the gauge must be positioned such that the measurement values lie within the limits. Besides checking whether the gauge is positioned correctly, the gauge can be used to check the weight displacement that is necessary to realize a certain inclination (for example 1 degree).



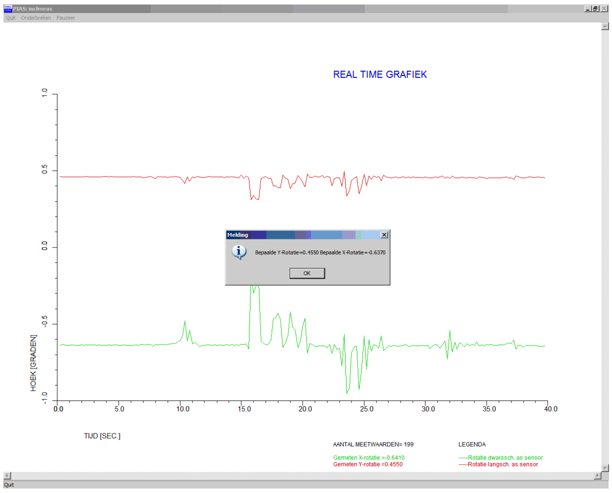
Real time graphic

29.2.2 Inclination measurement

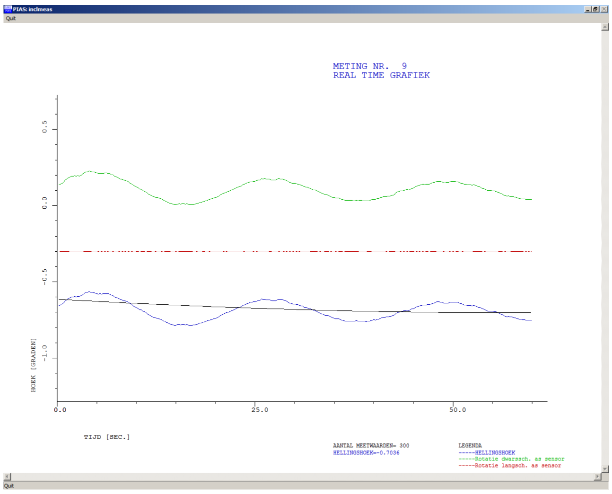
After you have selected this option, the measurement starts immediately. The following is displayed on the screen:

- Number of measured values. The number of readings until that moment. The measurement frequency is 5 Hz.
- Measured X-axis rotation. These angles have been determined at the zero measurement relative to a horizontal plane. At subsequent measurements the measured rotations are measured relative to the angles of these axes that have been determined during the zero position or previous inclination measurement.
- Measured Y-axis rotation. These angles have been determined at the zero measurement relative to a horizontal plane. At subsequent measurements the measured rotations are measured relative to the angles of these axes that have been determined during the zero position or previous inclination measurement.
- Measured inclination. This inclination is the change in angle relative to the zero position or previous inclination measurements. During the zero measurement (first measurement) this value is not displayed, only the position of the separate X- and Y-axis of the inclination sensor relative to a horizontal plane is determined.
- Graphic presentation measured X-Y rotation and inclination.

The first image below is a zero measurement, the second one a later measurement (after displacement of inclination weight), where also an inclination is measured and displayed.



Zero position



Measurement after relocation

During the measurement one has the following possibilities:

- One has the measurement finished automatically when the module has determined the correct inclination or the maximum number of measurements has been reached.
- [Quit]. The measurement is stopped, measured values are not saved.
- [Interrupt]. After you have selected this option, the following question is asked. The inclination or zero position determined so far is saved. This may result in an incorrectly determined inclination or zero position, because the measurement had not yet been stopped automatically.
- [Pause].

29.2.3 Last measurement again

After choosing this option the last measurement will be repeated. Earlier determined angles and measurements will be erased.

29.2.4 Remove all saved measurements

All measurement data will be ereased. If a new measurement is started a zero measurement will be done fisrt.

29.2.5 Output of measurements to screen

When you choose this option, a menu appears to choose which set of data should be displaced. The data sets are called by the descriptions given in Inclination measurement. After selecting the desired set, the data as given in option Inclination measurement, the determined inclinations and the number of measured inclinations will be presented on screen. With <Enter> you continue to the graphical representation. This is a graph of the data as measured and the function used to determine the correct inclination. The correct inclination is the value of the function completely at the right in the graph. Also the description of the set as given in Inclination measurement is presented.

29.2.6 Output of measurements to printer

All graphical information as presented in the previous option is sent to the printer.

29.2.7 Output of measurements to ASCII file

With this option the measurement data (of all measurements) can be exported to a ASCII file.

29.2.8 Settings of digital inclinometer

There are four settings:

- Type of sensor
- Serialport
- Minimum/maximum number of readings

Chapter 30

Launch: launching calculation

This module computes pressures, forces, speeds, anti-tipping moments etc. during longitudinal launching.

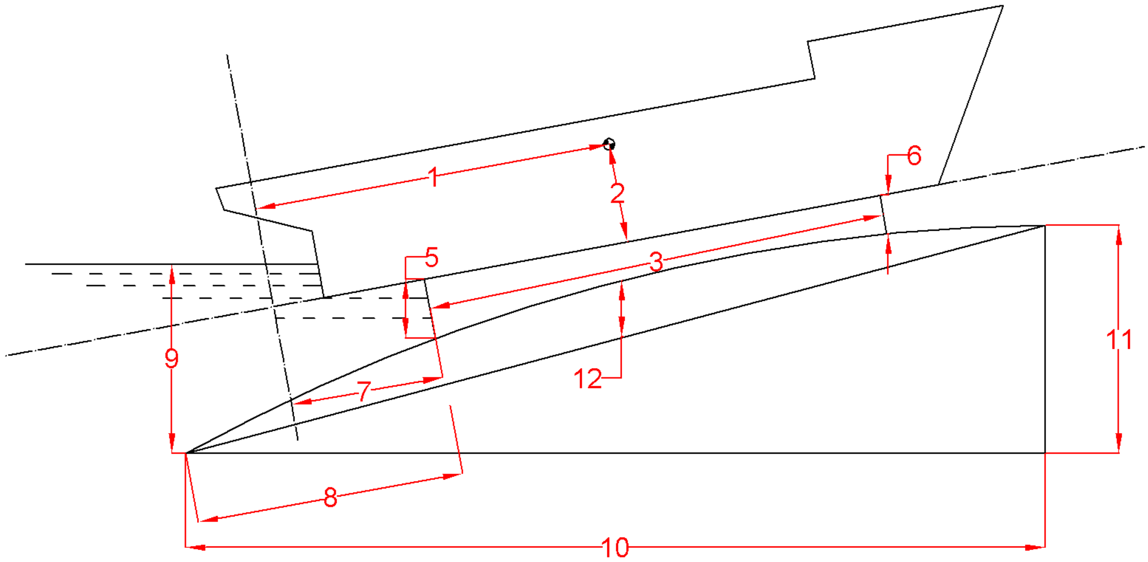
Longitudinal launching

30.1 Input data of ship and slipway

For ship and slipway, the following parameters — which are also illustrated in the sketch at the end of this section — must be provided.

- Identificatie, which is simply a name to be included in the output.
- The weight of the ship, during launching [ton].
- Longitudinal centre of gravity of the vessel is measured in the plane of the vessel in relation to APP.
- Vertical centre of gravity of the vessel is measured in the plane of the vessel in relation to the baseline.
- Length and breadth of the cradle are measured in the plane of the cradle.
- Length of fore poppet in relation with length of cradle [%], is that part of the cradle that is supposed to give upward force after floating of the aftship.
- Cradle height aft and for is measured perpendicularly to the cradle.
- Distance of aftside of the cradle to APP, and to the end of the slipway are measured alongside the slipway.
- Water level above the end of the way is measured perpendicularly to the water level.
- Density of outside water, which will (only for Launch) overwrite the density setting of Config — in 'Calculation methods and output preference', section 5.1.6 on page 43, Density outside water.
- Camber of the slipway is measured at half-length of the way, perpendicularly to the slipway.
- Time interval for calculation, should be set to approximately 0.5 to 2 seconds in order to get an accurate calculation.
- Margin speed at which the calculation ends, has to be set because the calculation will otherwise continue endlessly when no dragging forces have been specified. The reason for this is that the water resistance will approximate to zero but will never become zero.

This parameter menu has an addition function [PAper size] which can be used to toggle between A3 and A4 paper size for formatting the output.



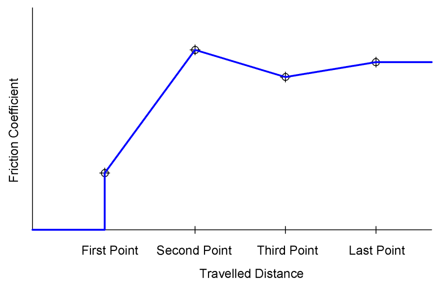
Geometric parameters, with the numbers referring to the table below.

1. Longitudinal center of gravity
2. Vertical center of gravity
3. Length of the cradle
5. Crade height aft
6. Crade height forward
7. Distance from aftside of the cradle to APP
8. Distance from the aftside of the cradle to aftside of the slipway
9. Waterlevel above the aftside of the slipway
10. Length of the slipway
11. Height at the forward side of the slipway
12. Camber at half length of the slipway

30.2 Input data of the friction coefficient of the cradle

One or more dimensionless friction coefficients as a function of the travelled distance are defined in this input screen, as illustrated in the figure below. The friction coefficient is defined as *ship weight / friction force*, both in the same dimension. The friction coefficient is assumed to be independant of speed. In this input menu a number (maximum 40) of travelled distance & friction coefficient combinations can be defined. Intermediate coefficients are calculated by linear interpolation, with at the extremities the provisions as illustrated in the figure below:

- At a travelled distance less that the value of the first row, the friction coefficient will be taken as zero.
- At travelled distances larger than the last row, the friction coefficient from the last row will be taken.



travelled distance & friction coefficient.

30.3 Input data of the resistance coefficient of the wetted hull

One or more resistance coefficients of the wetted hull can be defined here as a function of the speed. This coefficient is defined in its simplest form as follows: coefficient = resistance / displacement / speed^2 [sec^2/m^2]. For interpolation of intermediate values the same procedure is adopted as described above.

30.4 Input data of the dragging forces

One or more dragging forces can be defined here as a function of the speed. Intermediate values are calculated as described above.

30.5 Execute launching calculation

For every step in time the speed, pressure, forces, and travelled distance is printed, as in the example below. By the way, the assumption in the cradle pressure is that it is linearly distributed over the cradle length, and that it is only positive (after all the cradle cannot exert a tension force).

DEMO LAUNCHING CALCULATION
Mv. exempli gratia

Time	Distance	Speed	Acceler	Position	Anti-tip	Max.press	Loc.max	G'M	Fstem
0.00	0.00	0.00	0.269	Following way	60516.9	17.97	94.15	-	-
0.50	0.03	0.13	0.269	Following way	60483.1	17.97	94.15	-	-
1.00	0.13	0.27	0.269	Following way	60381.5	17.97	94.15	-	-
1.50	0.30	0.40	0.269	Following way	60212.3	17.97	94.15	-	-
2.00	0.54	0.54	0.269	Following way	59975.3	17.97	94.15	-	-
....									
12.50	20.97	3.35	0.260	Following way	39352.7	19.82	11.97	-	-
13.00	22.68	3.48	0.256	Following way	37651.2	20.97	13.68	-	-
13.50	24.45	3.61	0.252	Following way	35902.1	22.14	15.45	-	-
14.00	26.28	3.73	0.246	Following way	34113.7	23.31	17.28	-	-
14.50	28.18	3.86	0.238	Following way	32296.2	24.45	19.18	-	-
15.00	30.14	3.98	0.228	Following way	30462.1	25.53	21.14	-	-
15.50	32.16	4.09	0.217	Following way	28626.4	26.51	23.16	-	-
16.00	34.23	4.20	0.202	Following way	26806.7	27.31	25.23	-	-
16.50	36.35	4.30	0.185	Following way	25023.7	27.86	27.35	-	-
17.00	38.52	4.39	0.164	Following way	23300.6	28.04	29.52	-	-
17.50	40.74	4.47	0.140	Following way	21664.4	27.73	31.74	-	-
18.00	42.99	4.54	0.112	Following way	20144.2	26.73	33.99	-	-
18.50	45.28	4.60	0.080	Following way	18772.2	24.81	36.28	-	-
19.00	47.59	4.64	0.044	Following way	17582.1	21.66	38.59	-	-
19.50	49.91	4.66	0.004	Following way	16608.5	17.81	94.15	-	-
20.00	52.25	4.66	-0.040	Following way	15893.2	22.47	94.15	-	-
20.50	54.57	4.64	-0.087	Following way	15473.7	29.10	94.15	-	-
21.00	56.88	4.60	-0.134	Following way	15381.5	48.79	94.15	-	-
21.50	59.17	4.53	-0.165	Lifting aft	-	131.07	94.15	5.568	346.384
22.00	61.41	4.45	-0.158	Lifting aft	-	129.58	94.15	5.949	342.422
22.50	63.62	4.37	-0.151	Lifting aft	-	128.00	94.15	6.327	338.265
23.00	65.79	4.30	-0.145	Lifting aft	-	126.35	94.15	6.708	333.893
23.50	67.92	4.22	-0.140	Lifting aft	-	124.60	94.15	7.083	329.285
24.00	70.01	4.15	-0.135	Lifting aft	-	122.76	94.15	7.458	324.416
24.50	72.07	4.09	-0.131	Lifting aft	-	120.82	94.15	7.826	319.292
25.00	74.10	4.02	-0.127	Lifting aft	-	118.76	94.15	8.193	313.831
25.50	76.09	3.96	-0.123	Lifting aft	-	116.56	94.15	8.553	308.017
26.00	78.05	3.90	-0.120	Lifting aft	-	114.21	94.15	8.908	301.807
26.50	79.99	3.84	-0.117	Lifting aft	-	111.69	94.15	9.256	295.151
27.00	81.89	3.78	-0.115	Lifting aft	-	108.99	94.15	9.594	288.012
27.50	83.76	3.72	-0.113	Lifting aft	-	106.07	94.15	9.920	280.311
28.00	85.61	3.66	-0.112	Lifting aft	-	102.92	94.15	10.234	271.986
28.50	87.43	3.61	-0.110	Lifting aft	-	99.51	94.15	10.535	262.970
29.00	89.22	3.55	-0.110	Lifting aft	-	95.81	94.15	10.820	253.190
29.50	90.98	3.50	-0.109	Lifting aft	-	91.80	94.15	11.089	242.588
30.00	92.71	3.44	-0.109	Lifting aft	-	87.46	94.15	11.340	231.129
30.50	94.42	3.39	-0.110	Lifting aft	-	82.80	94.15	11.575	218.813
31.00	96.10	3.33	-0.110	Lifting aft	-	77.85	94.15	11.793	205.722
31.50	97.75	3.28	-0.111	Lifting aft	-	72.65	94.15	11.997	191.987
32.00	99.38	3.22	-0.112	Lifting aft	-	70.35	94.15	12.189	177.761
32.50	100.98	3.17	-0.259	Lifting aft	-	112.01	94.15	12.371	163.179
33.00	102.53	3.04	-0.254	Lifting aft	-	355.21	94.15	12.540	148.491
The stem of the vessel has fallen from the slipway									
33.50	104.01	2.91	-0.306	Floating	-	0.00	0.00	13.806	-
34.00	105.43	2.76	-0.290	Floating	-	0.00	0.00	13.806	-
34.50	106.77	2.61	-0.276	Floating	-	0.00	0.00	13.806	-
35.00	108.04	2.47	-0.264	Floating	-	0.00	0.00	13.806	-
35.50	109.25	2.34	-0.253	Floating	-	0.00	0.00	13.806	-
36.00	110.38	2.21	-0.243	Floating	-	0.00	0.00	13.806	-
36.50	111.46	2.09	-0.234	Floating	-	0.00	0.00	13.806	-
37.00	112.48	1.98	-0.226	Floating	-	0.00	0.00	13.806	-
37.50	113.44	1.86	-0.218	Floating	-	0.00	0.00	13.806	-
38.00	114.34	1.75	-0.211	Floating	-	0.00	0.00	13.806	-
38.50	115.19	1.65	-0.205	Floating	-	0.00	0.00	13.806	-
39.00	115.99	1.55	-0.200	Floating	-	0.00	0.00	13.806	-
39.50	116.74	1.45	-0.194	Floating	-	0.00	0.00	13.806	-
40.00	117.44	1.35	-0.190	Floating	-	0.00	0.00	13.806	-
40.50	118.09	1.25	-0.186	Floating	-	0.00	0.00	13.806	-
41.00	118.69	1.16	-0.182	Floating	-	0.00	0.00	13.806	-
41.50	119.25	1.07	-0.179	Floating	-	0.00	0.00	13.806	-

Example of a single page of output.

Abbreviations and units :

Time	Elapsed time (sec)
Distance	Travelled distance (m)
Speed	Speed (m/sec)
Acceler	Acceleration (m/sec^2)
Position	Position of the vessel
Anti-tip	Anti tipping moment (tonm)
Max.press	Maximum pressure on the cradle (ton/m^2)
Loc.max	Loaction of the maximum pressure on the cradle (m)
G'M	Virtual metacentric height (m)
Fstem	Force on the stem while lifting aft (ton)
Taft	Draft aft (m)
Tfore	Draft fore (m)
Displa	Displacement (ton)
LCB	Longitudinal center of buoyancy (m)
Lcrad	Actual length of cradle (m)
Pr.aft	Pressure on cradle aft (ton/m^2)
Pr.fore	Pressure on cradle fore (ton/m^2)
Fdriv	Driving mass force (ton)
Ffrict	Frictional force of the cradle (ton)
Fresist	Resistance force of the displacement (ton)
Fdrag	Dragging force (ton)

Explanation of the output parameters.

30.6 File and backup management

Backups can be made and restored here. Here is also the option ‘Quit module without saving the data’. For details we refer to section 2.9 on page 15, Data storage and backups.

Chapter 31

Cargoquip: definition of cargo equipment

Module Loading contains many useful tools for exploiting specific characteristics of cargo types or loading tools, such as for container loading, RoRo loading, grain loading and shipboard cranes. All these categories have rather ship-specific features, such as container castings, power connections for reefer containers, decks and lanes for RoRo cargo, attachment points for the lashings of containers and trucks, crane dimensions and capacity, grain bulkheads etc. This is ship-specific input, and it has been considered to give it a place in other input modules, such as Hulldef or Layout. However, in order not to overload these modules with options, it was decided to segregate this input into a separate module, called Cargoquip.

The level of discussion in this module is somewhat more general than in the rest of this manual. After all, by the time the user has arrived at these static ship details, he or she will have gone through the entire cycle of hull and compartment input, as well as performing many calculations. For such a seasoned user, not every menu needs to be covered in great detail. Therefore, the concepts of and relations between the various data will be more important here than the precise menu structure.

Attention

Unless stated otherwise, all point coordinates must be given in the PIAS ship axis system.

Input of cargo equipment

1.	Input of spaces
2.	Input of panels
3.	Input of specific data per cargo type
4.	Input of IMDG data
5.	Create data files
6.	File Management

31.1 Input of spaces

In this menu you can define all spaces and their subcategories. You can add any number of spaces, the program will automatically select the necessary ones for every required process.

Name
Type a name for the space.

Compartment
Press space to choose from the available compartments defined in Layout. It is not necessary for a space to come from Layout, its solid shape can be manually defined in section 31.1.3 on page 495, Solid shape definition.

Decks
Double-click or Enter in order to add deck data. Refer to section 31.1.1 on the next page, Input of decks.

Panel positions
Double-click the cell in order to add panel positions, see section 31.1.2 on page 495, Input of panel positions. The panels themselves are defined in section 31.2 on page 496, Input of panels.

Type
Choose between [General] and [IMDG].

Location
Choose between [Above deck] and [Below deck].

Defined
If the selected compartment has been correctly defined in Layout , then green text *With compartment* will appear and the solid shape definition menu is not accessible. Otherwise, the solid shape needs to be manually defined. Double-clicking will enter a new menu to define the solid, see section 31.1.3 on the following page, Solid shape definition.

31.1.1 Input of decks

In this menu you can define any number of decks that will belong to the selected space. Depending on the type of deck (RoRo or Container), some options might become unavailable. [Duplicate] will duplicate the selected deck, including all its points and settings.

For both types, you can define:

- Name
- Deck type
- Contour points
- Lashing points

For RoRo you can additionally define:

- Floor points
- Ceiling points
- Maximum deck load
- Load allowed : whether RoRo cargo can be loaded on the specific deck
- Line pattern (for drawing)
- Line type (for drawing)
- Line colour (for drawing)
- Line thickness (for drawing)

For Container cargo you can additionally define:

- Castings
- Grid points
- Reefer points
- Maximum stackload for 20ft containers (including types 1,A,B,C,D,E,F)
- Maximum stackload for 30ft containers
- Maximum stackload for 40ft containers (including types G,H,K,LM,N,P)
- Tier offset : The standard offset for containers in the hold is 0. For containers on deck this is 80. Increase/decrease with steps of 2 for a raised/lowered deck.

31.1.1.1 Deck 3D points

By pressing Space you can connect any point to a reference plane. Not all of these points need to be defined. It depends on the type of vessel, type of space and type of deck.

Contour points
Contour points have to be manually defined. The first and last point are always identical, indicating a closed contour line.

Castings
Casting points can automatically derive a height from the solid shape if the latter has been defined properly. When filling up the castings list, please be aware to include the length and breadth offsets because they will not be automatically added later. You can also paragraph 31.1.1.1.1 on the next page, Use table to create castings from container slots. The user-defined points in the list will not be affected by the table.

Grid points
Grid points can automatically derive a height from the solid shape if the latter has been defined properly. A [Bay] and a [Row] also need to be entered.

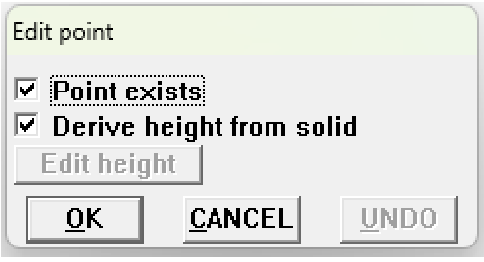
Lashing points
Lashing points can automatically derive a height from the solid shape if the latter has been defined properly.

Reefer points

Reefer points can automatically derive a height from the solid shape if the latter has been defined properly.

31.1.1.1.1 Use table

In this table you can input the coordinates of 20ft container slots. New rows can be added by [New] or [Insert] and deleted by [Remove]. New columns can be added by [new Column] and deleted by [remove cOlumn]. The first row is the longitudinal coordinate (LCG), the first column the transverse coordinate (TCG). These coordinates can also be connected to a reference plane. For the height, you can either type the slot underside in the selected cell, or press Enter or double-click for a popup.



Point manipulation popup.

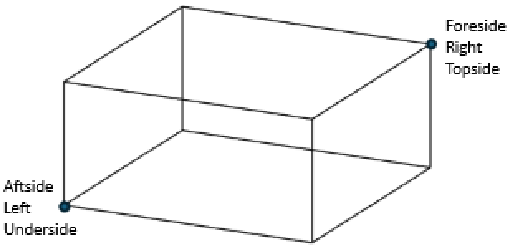
The slot point can be present or not. If it is, then you can select how the height coordinate is defined↩
: automatically from the solid shape or manually. In [Edit height] you can connect the point to a reference plane. If the setting [Derive height from solid] is enabled, editing either the first row or the first column will automatically update the full column or row respectively. When exiting this table, 4 casting points are automatically generated for each existing slot. Please be aware that these 4 points are persistent to their slot and will be re-generated upon every exit of the table.

31.1.2 Input of panel positions

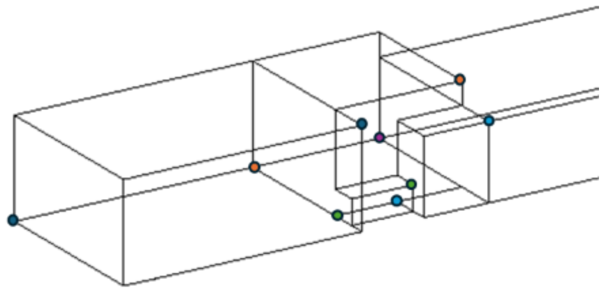
Here you can input panel positions per space. These panel positions will become available to choose from in the section 31.2 on the following page, Input of panels.

31.1.3 Solid shape definition

If the space is not connected to a compartment from Layout , then for full functionality the solid shape has to be manually defined. The solid shape is either a rectangle or a set of merged/connected rectangles. These rectangles are defined by 6 points: [Aftside], [Left], [Underside], [Foreside], [Right], [Topside].



Solid shapes are defined by 2 points.



A more complicated shape created by merging. Each pair of points is coloured differently.

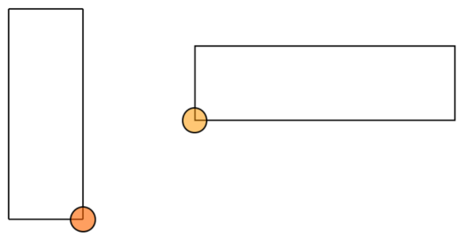
If the space is defined by more than 1 rectangle, exiting the solid shape definition menu will automatically create the solid shape by merging all the solids. If the merging operation is successful, the text in the section 31.1 on page 493, Input of spaces menu will be green, otherwise red. Solid shapes that cannot be created by one or more rectangles have to be defined in Layout .

31.2 Input of panels

You can define all panel data in this menu. The length, breadth and height positions of the panel are given in the PIAS ship axis system.

Attention

The axial system of a panel is Aftside, CenterLine, Underside. LCG, TCG, VCG and Castings' coordinates must be given in this local axis system.



Reference point vertical position (left) and horizontal position (right).

In [Positions], you can double-click to select from a list of available positions defined in section 31.1.2 on the preceding page, Input of panel positions.

31.3 Input of specific data per cargo type

31.3.1 Main screen

Here you can define the window layout for the Main screen of Locopias by Double-clicking in [Number of defined windows]. [Use default] in the menubar for a default window configuration.

31.3.2 Weight list

Here you can set minimum and maximum values for (free) weight items in the weight list.

31.3.3 Tanks

Here you can define the window layout for the Tanks module by Double-clicking in [Number of defined windows]. [Use default] in the menubar for a default window layout.

31.3.4 Containers

Weightgroup
Choose an existing weightgroup or create a new one.

Default VCG %

Default spacer

Seagoing

Transverse oriented slots present
If the vessel has transverse slots, set this to Yes. If Yes, there will be an option in the Settings of the Container module to turn them on/off.

Upper side hold
Fill in the underside of slot for tier 82. You can also press space to connect this entry to a reference plane.

Bays above deck, Bays below deck
It is not necessary to fill up these lists; you can enter one, neither or both, depending on the geometry of the vessel. [Reference point] is a single coordinate, LCG of the bay. [Bay prientation] can be longitudinal or transverse (rotated 90°).

Rows above deck, Rows below deck
Enter the maximum number of rows. If the number of rows changes from odd to even (or vice versa) then set this to -1.

- Example 1 : bays 01 to 03 have 5 rows, bays 05 to 23 have 6 rows = set this to -1.
- Example 2 : bays 01 to 03 have 4 rows, bays 05 to 23 have 6 rows = set this to 6.

Note

The rotated slots are not included in this, ignore them for this entry.
If this concerns a vessel without castings, set this to -1.

30ft follow 20ft bay numbering

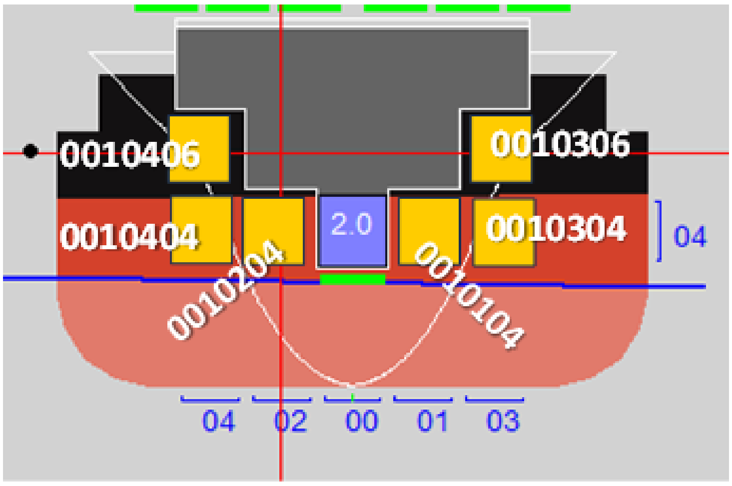
Default weights per type
Double-click to fill in the table.

Cell guides
Double-click to enter cell guides. This is currently only used in the lashing module. For the cell guides that occupy 2x20ft bays, please add the bay number on the aft (larger bay number).

Note

If you wish for the calculations to take into consideration the cell guides only if there are 40ft containers loaded in the bay, then use the even bay numbering. In the case of a mixed stack (2x20ft together with at least one 40ft) then input the odd aft bay.

Blind frames
Double-click to enter blind frames. This is currently only used in the lashing module. Please note that the bay always starts with a 0. Consider blind frames the positions where there would be imaginary containers, always 20ft normal height.



Example : Blind frames shown in yellow for bay 001.

Number of defined windows

Here you can define the window layout for the Containers module by Double-clicking in [Number of defined windows]. [Use default] in the menubar for a default window layout.

31.3.5 General cargo

Weightgroup

Choose an existing weightgroup or create a new one.

Minimum, maximum allowable weight

Set the minimum and maximum allowable weight for all general cargo.

Visible compartments in sections

Double-click to enter a list of visible compartments in sections.

Heights

Double-click to enter a name and a height for predefined-selection for free placement of cargo in the General cargo module.

Contour points

Double-click to define contour points. Please be aware that the first and the last point should be identical, indicating a closed contour.

Number of defined windows

Double-click to set the window layout for General cargo. [Use default] in the menubar for a default window layout.

31.3.6 RoRo cargo

Weightgroup

Choose an existing weightgroup or create a new one.

Landing crafts

Double-click to define any number of landing crafts.

31.3.7 Grain

To load grain graphically, it is required to define the geometry of the holds and bulkheads and possible positions. In Layout, make sure you have defined a compartment 1 metre long with the outline of the straight section in the grain hold. If there are also non-parallel sections, you need to define these as a separate compartment. This can be done from the aft/fore of the hold to where it runs parallel again. In the compartment list, you can set the tank to No so it is not counted twice.

Weightgroup grain bulkheads

Choose an existing weightgroup for grain bulkheads or create a new one.

Weightgroup grain holds
Choose an existing weightgroup for grain holds or create a new one.

Tanktop height
Lower limit of hold.

Coaming height
Upper limit of hold

Movable grain bulkheads
Double-click to define any number of movable grain bulkheads. Each grain bulkhead can be given its own name and weight.

Grain bulkhead positions
The user can define all possible positions of the grain bulkheads here, both of movable bulkheads and static hold bulkheads. For each position, the following must be entered:

- Position name: The name of the position.
- Bulkhead type: Choice of static aft, static fore or movable. In the case of a fixed bulkhead, only 1 number (rear or front respectively) needs to be entered.
- Aft: The rear of the position in metres.
- Fore: The front of the position in metres.
- Underside: The bottom of the position in metres.
- Topside: The top of the position in metres.
- Breadth: The breadth of the position in metres.
- VCG: The VCG of the movable grain bulkhead above the base in this position.
- LCG: The LCG of the movable grain bulkhead from the APP in this position.
- TCG: The TCG of the moveable grain bulkhead from the CL in this position.

'Basis' grain holds
To define grain holds, they must be available as compartments in Layout. For the part, or parts of the hold that have a simple rectangular cubic cross-section, define a rectangular compartment (hold/metre) in Layout and select this compartment here, where you set the [variable] column to "Yes" and set the length to 1 metre. This compartment is then used for all rectangular cuboid parts of the hold, for which no other compartments are set. If the hold, or parts of it, is not rectangular cuboid, then this part of the hold must be defined separately in Layout. Add these compartments in this menu, and specify the aft and forward boundary of the positions between which this grain hold is to be used.

- Variable: A variable compartment ("yes") is used between all defined positions of the grain hold, for which no other (non-rectangular cubic) hold is defined. It is then usual to define a compartment of the parallel part (hold/metre) with a length of 1 metre. The length of this defined compartment should be specified in the column [length] (typically 1 metre). A non-variable part of the hold is used between the positions specified.
- Name: Here the user can select the relevant grain hold from the compartment list by name.
- Length: If "Yes" is selected for variable, the length of the selected compartment should be specified here.
- Bulkhd. name aft: Here the user can select the name of the respective grain bulkhead position from the list as defined under [Grain bulkhead positions] which selects the rear of the hold.
- Bulkhd. name fore: Here the user can select the name of the relevant grain bulkhead position from the list as defined under [Grain bulkhead positions] which selects the front of the hold.
- Pos. aft: This is automatically filled in after selecting a position in the column "blkh. Name aft".
- Pos. fore: This is automatically filled in after selecting a position in the column "blkh. name fore".
- Startrecord: This is the line number in the grain moments table, where the table of moments for this particular part of the grain hold begins. This is useful if you want to edit or view the grain moments table manually.
- Contour points: A "0" means that no contour has been created. If a contour has been defined, the number of defined points is displayed here. You can click through on this column. You will then get an input menu, in which contour points can be defined. This contour is only used in the graphical interface to display the grain space in the side view.

Hold(s) excluded for loading of grain
It may happen that a part of the hold arises between two grain bulkhead positions where it is not desired to load grain with the graphical grain module. These parts can be specified here by specifying an aft and fore

position.

Selection and definition of grain holds
Double-click to define void spaces and tables of grain holds defined in ['Basis' grain holds]. For details, refer to section 11.1 on page 262, Selection and definition of grain holds

Note

You can import an existing graphical grain file (.gr4/.gr1) by selecting [File]–>[Import existing grain data file (.gr4/.gr1)]. gr1 is deprecated.

31.3.8 Panels

Weightgroup
Choose an existing weightgroup or create a new one.

31.4 Input of IMDG data

Transverse bulkheads
Double-click to enter (length) coordinates for transverse bulkheads.

Longitudinal bulkheads
Double-click to enter (breadth) coordinates for longitudinal bulkheads.

Decks
Double-click to enter (height) coordinates for decks.

DOC table
Double-click to fill in permissible substances per space. In order for a space to appear in this table, it has to be defined as a *DOC space* in section 31.1 on page 493, Input of spaces then [Type of space].

31.5 Create data files

Cargoquip can generate the following files:

Create tanks window definition file
Create a .tkc file

Create container definition file
Create a .cas file

Create IMDG definition file
Create a .imdg file

Create general cargo definition file
Create a .sg2 file

Create general cargo window definition file
Create a .sgc file

Create panel definition file
Create a .hch file

Create RoRo cargo definition file
Create a .ro1 file

Create RoRo landing craft definition file
Create a .ro6 file

Create damage control definition file
Create a .dmg file

Create grain definition files
Create a graphical grain definition file (.gr4), grain moments table or edit grain moments table.

- Create graphical grain definition file : create a .gr4 file.
- Create grain moments table : if a .gr4 file exists, this will create a .gr2 file.
- Edit grain moments table : Here you can edit the grain moments table. Changes are saved on exit.

It is highly advised to check these files for consistency.

31.6 File Management

Backups of all data can be made and restored here. For details, refer to section 2.9 on page 15, Data storage and backups.

Chapter 32

Photoship: measuring a ship hull by photogrammetry

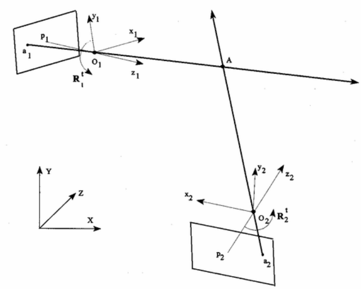
This module is designed to reconstruct existing hull shapes by calculating the 3D coordinates of measuring points by means of photography and reference points. Photoship finds itself somewhat on the edge of the natural PIAS functionality because it seldom occurs that a vessel exists, while drawing or other data are lacking. Furthermore, the photogrammetric process requires some insight and experience, so that it only will pay off when applied on a regular basis. In order to provide a notion of the operation a short introduction is presented here. A much more detailed separate manual is available on request.

32.1 The role of Photoship in the reverse engineering process

The goal of reverse engineering is to obtain an unambiguous 3D model of the hull shape. The role Photoship plays in this process consists of the construction of a wireframe model, which consists of measured points and user-defined line segments between those points. A solid model based upon such a wireframe can be generated with Fairway (see section 6.3.7.8 on page 123, Convert Wireframe to Solid). Also all other Fairway facilities can be applied on a photogrammetrically measured hull shape, notably post-processing of the hull lines (section 6.3.7 on page 113, Wireframes) and export of the 3D hull model.

32.2 The basics of photogrammetric measurement

The principle of photogrammetry is based on stereovision. The 3D coordinates of measuring points are calculated on basis of the depictions of these measuring points on multiple photos. A comparison with human vision can be made here. In order to see depth, two eyes are necessary. The principle of stereovision is illustrated in the picture below:



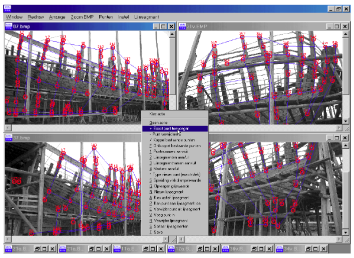
Lines of sight intersect in space

What we see here are the depictions a1 and a2 of point A on two different photos. From a1 and a2 two lines of sight are drawn through the foci O1 and O2. The point of intersection of these two lines of sight defines the 3D coordinates of A. This principle is named 'intersection'. For this principle to work, the orientation of each photo has to be known. These are calculated with the aid of reference points. Reference points are measuring points with known coordinates. They are also being used to give the model the right scale and orientation. Because we are working with measured values, the lines of sight will not intersect exactly. This is why it is impossible to find an exact solution. For this reason, the whole system of measuring points and external orientations is optimized numerically (by iteration) by means of the method of least squares. To be able to do this, initial values for the measuring points need to be known. The result is the most accurate set of values for the 3D coordinates of the measuring points. This way the hull shape is mapped completely by use of multiple photos and measuring points.

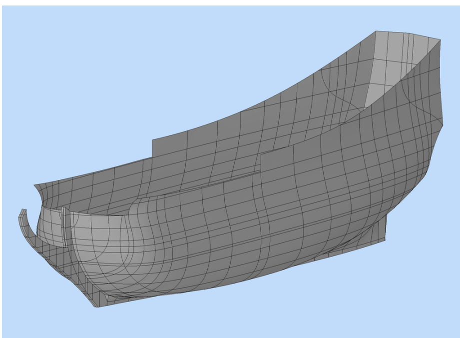
32.3 Measuring a ship hull by Photoship

1. The following steps are to be taken:
2. Place the landmarks on the hull.
3. Measure the coordinates of the reference points by conventional methods.
4. Make photos, while making sure that every landmark appears on minimal three, but preferably more shots.
5. Open the pictures in PhotoShip, give the coordinates of the reference points and de camera parameters.
6. Point out and connect the photo points.
7. Give the relations between the measuring points by defining line segments.
8. Calculate the external orientation of each photo.
9. Calculate the 3D coordinates of the measuring points
10. Save the 3D coordinates in an .SXF file, so it can further be processed with Fairway. Now the file is ready to be applied in PIAS, or e.g. to be exported to a general-purpose CAD system.

This process is elaborated step by step in www.sarc.nl/images/stories/photoship/article_photoship_en.pdf



Photo's, points and connections in the GUI of Photoship



The resulting 3D model in Fairway

Chapter 33

License terms for PIAS software in binary format

1. The software has been furnished under, and may only be used under the conditions of, a (direct or indirect) license agreement with SARC.
2. The software (and any update) in its entirety or any part of it may only be used by user on a computer furnished with a hardware or software protection device that has been provided by the licensor.
3. For archival and security purposes the software may be copied in its entirety or partly, but only for use by the user.
4. The user is not entitled to transfer the use of (any part of) the software, as well as the related documentation, to anybody else or to put it at the disposal of anybody else. Nor is the user entitled to modify, adjust, translate, counterfeit, decompile, demount, disassemble the software or make works that are based on it.
5. The software has been designed to be used on a regular Personal Computer driven by the operating system Microsoft Windows, the versions Vista, 7, 8 or 10.
6. SARC is the owner of the software and also owns its copyright.
7. Subject to an attributable failing or a wrongful act, the user cannot hold SARC liable for any damage resulting from, or related to, the use of or not being able to use the software, and indemnifies the licensor against all claims of third parties due to such damage.
8. The liability of SARC for damage suffered by the customer, being the result of an attributable failing or wrongful act, is limited to the purchase price of the software license.
9. The restriction from the previous Article does not apply in the case of foul play or serious misconduct, in which case the liability is limited to €250,000.
10. To licensors' best knowledge the software is correct. However, licensor does not warrant the correctness of the software or any part of it.
11. Dutch law is applicable to these terms and the use of PIAS. Disputes will be subjected to the judgment of a Dutch court.

Index