3D/2D modelling suite for integral water solutions

DELFT3D

Deltares systems

RGFGRID

Deltares

Enabling Delta Life

User Manual

RGFGRID

Generation and manipulation of curvilinear grids for Delft3D-FLOW and Delft3D-WAVE

User Manual

Hydro-Morphodynamics

Version: 4.00.30932

28 April 2014

RGFGRID, User Manual

Published and printed by:
Deltares
Boussinesqweg 1
2629 HV Delft
P.O. Box 177
2600 MH Delft
The Netherlands

telephone: +31 88 335 82 73
fax: +31 88 335 85 82
e-mail: info@deltares.nl
www: http://www.deltares.nl

For sales contact:
telephone: +31 88 335 81 88
fax: +31 88 335 81 11
e-mail: sales@deltaressystems.nl
www: http://www.deltaressystems.nl

For support contact:
telephone: +31 88 335 81 00
fax: +31 88 335 81 11
e-mail: support@deltaressystems.nl
www: http://www.deltaressystems.nl

Contents

List of Figures

List of Tables

1 Guide to this manual

1.1 Introduction

This User Manual concerns the grid generation module, RGFGRID, of the Delft3D software suite. To make this manual more accessible we will briefly describe the contents of each chapter and appendix.

If this is your first time to start working with RGFGRID module we suggest you to read and practice the getting started of Chapter 3 and the tutorial of Chapter 6. These chapters explain the user interface options and guide you through the generation of your first grid.

Chapter 2: Introduction to RGFGRID , provides specifications of RGFGRID and the areas of applications.

Chapter 3: Getting started, explains the use of the overall menu program, which gives access to all Delft3D modules and to the pre- and post-processing tools. Last but not least you will get a first introduction into the RGFGRID Graphical User Interface, used to define a grid which can be used in a hydrodynamic or wave simulation.

Chapter 4: General operation, provides practical information on the general operation of the RGFGRID module.

Chapter 5: Menu options, provides a description of all menu and toolbar options.

Chapter 6: Tutorial, emphasis at giving you some first hands-on experience in using the RGFGRID module to define the input of a simple problem and in executing a water quality simulation.

References, provides a list of publications and related material on the RGFGRID module.

Appendix A: Files of RGFGRID , gives a description of the files that can be used in RGFGRID as input or output. Generally, these files are generated by RGFGRID or other modules of the Delft3D suite and you need not to be concerned about their internal details. However, in certain cases it can be useful to know these details, for instance to generate them by means of other utility programs.

1.2 Name and specifications of the program

Title	RGFGRID
Description	RGFGRID is a program for generation and manipulation of curvilinear grids for Delft3D-FLOW and Delft3D-WAVE. The co-ordinate system may be Cartesian or spherical. Delft3D-FLOW is a simulation program for hydrodynamic flows and transports in 2 and 3 dimensions, see Delft3D-FLOW (2013). One of the wave modules in Delft3D-WAVE is SWAN which is a wave energy simulation package; see SWAN (2000).
Special facilities	sketch of coarse grid using splines smooth refinement module orthogonalisation module various grid manipulation options grid design by bathymetry or polygon control Cartesian or spherical co-ordinates dynamic memory allocation multiple grids supported

1.3 Manual version and revisions

A manual applies to a certain release of the related numerical program. This manual applies to RGFGRID, version 4.20.

1.4 Typographical conventions

Throughout this manual, the following conventions help you to distinguish between different elements of text to help you learn about RGFGRID.

Example	**Description**
Waves **Boundaries**	Title of a window or sub-window. Sub-windows are displayed in the **Module** window and cannot be moved. Windows can be moved independently from the **Module** window, such as the **Visualisation Area** window.
Save	Item from a menu, title of a push button or the name of a user interface input field. Upon selecting this item (click or in some cases double click with the left mouse button on it) a related action will be executed; in most cases it will result in displaying some other (sub-)window. In case of an input field you are supposed to enter input data of the required format and in the required domain.
<\tutorial\wave\swan-curvi> <siu.mdw>	Directory names, filenames, and path names are expressed between angle brackets, <>. For the Linux and UNIX environment a forward slash (/) is used instead of the backward slash (\) for PCs.

Example	Description
“27 08 1999”	Data to be typed by you into the input fields are displayed between double quotes. Selections of menu items, option boxes etc. are described as such: for instance ‘select *Save* and go to the next window’.
`delft3d-menu`	Commands to be typed by you are given in the font Courier New, 10 points.
➢	User actions are indicated with this arrow.
[m/s] [-]	Units are given between square brackets when used next to the formulae. Leaving them out might result in misinterpretation.

1.5 Changes with respect to previous versions

Version	Description
4.00.00	Complete new version of RGFGRID

2 Introduction to RGFGRID

2.1 Introduction

The purpose of the RGFGRID program is to create, modify and visualise orthogonal, curvilinear grids for the Delft3D-FLOW module.

Curvilinear grids are applied in finite difference models to provide a high grid resolution in the area of interest and a low resolution elsewhere, thus saving computational effort.

Grid lines may be curved along land boundaries and channels, so that the notorious 'stair case' boundaries, that may induce artificial diffusion, can be avoided.

Curvilinear grids should be smooth in order to minimise errors in the finite difference approximations. Finally, curvilinear grids for Delft3D-FLOW have to be orthogonal, which saves some computationally expensive transformation terms. Extra effort in the model set-up phase, results in faster and more accurate computations.

2.2 Co-ordinate systems

The grid system used in RGFGRID can be either Cartesian (in metres) or spherical (in decimal degrees). Cartesian co-ordinates can be displayed on a screen directly, just using a scale factor. Spherical co-ordinates can be displayed on screen as plane co-ordinates or as projected co-ordinates. Plane co-ordinates on screen give distortion in the polar direction. Depending on the type of projection, projected co-ordinates have no distortion in distance and angles. For this reason a stereographic projection is used in RGFGRID.

Starting from scratch, you have to select a co-ordinate system. The co-ordinates of all objects (land boundary, splines, grid, samples, etc.) are then in the selected co-ordinate system. When opening a grid, RGFGRID will read the co-ordinate system of the imported grid. The co-ordinates of other objects (land boundary, splines, polygons, samples and text files) are not checked; this is the responsibility of the user.

2.3 Program considerations

RGFGRID is designed to create grids with minimum effort, fulfilling the requirements of smoothness and orthogonality. The program allows for an iterative grid generation process, starting with a rough sketch of the grid by splines. Then, the splines are transformed into a grid, which can be smoothly refined by the program. Whenever necessary, you can orthogonalise the grid in order to fulfil the Delft3D-FLOW requirement of orthogonality.

Various grid manipulation options are provided in order to put the grid lines in the right position with the right resolution. For instance, a grid line can be 'snapped' to a land boundary. The surrounding grid smoothly follows. More detail is brought into the grid after every refinement step.

Existing grids may be modified or extended using this program. Grids can be locally refined by insertion of grid lines. The resulting local 'jump' in grid sizes can be smoothed by a so-called 'line smoothing'.

Bathymetry data can be displayed on the screen, so that internal gullies can be taken into account while drawing the design grid. Existing model grids can be opened and displayed on the screen, while creating new grids to be pasted later to the original. Before each modification or edit action, the grid is saved to the so-called 'previous grid'. Pressing `Esc` after an edit action, copies the previous grid back to the grid. If desired, the previous grid can be shown together with the active grid.

Grid properties such as smoothness, resolution, orthogonality etc, can be visualised to check the grid quality. Graphical output can easily be created in various formats.

3 Getting started

3.1 Overview of Delft3D

The Delft3D program suite is composed of a set of modules (components) each of which covers a certain range of aspects of a research or engineering problem. Each module can be executed independently or in combination with one or more other modules.

Delft3D is provided with a menu shell through which you can access the various modules. In this chapter we will guide you through some of the input screens to get the look-and-feel of the program. In the Tutorial, Chapter 6, you will learn to define a simple scenario.

3.2 Starting Delft3D

To start Delft3D:

◇ On an MS Windows platform: select *Delft3D* in the *Programs* menu.
◇ On Linux machines: type `delft3d-menu` on the command line.

Next the title window of Delft3D is displayed, Figure 3.1.

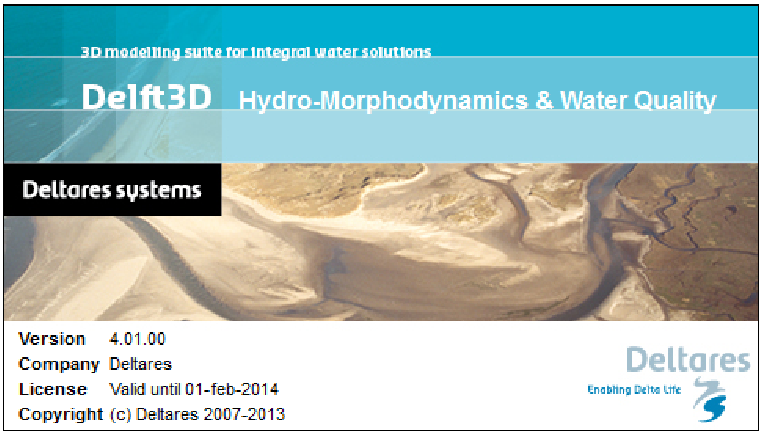


Figure 3.1: *Title window of Delft3D*

After a short while the main window of the Delft3D-MENU appears, Figure 3.2.

Several menu options are shown. For now, only concentrate on exiting Delft3D-MENU, hence:

◇ Click on the *Exit* push button.

The window will be closed and you are back in the Windows Desktop screen for PCs or on the command line for Linux workstations.

Remark:

◇ In this and the following chapters several windows are shown to illustrate the presentation of Delft3D-MENU and RGFGRID. These windows are grabbed from the PC-platform. For Linux workstation the content of the windows is the same, but the colours may be different.

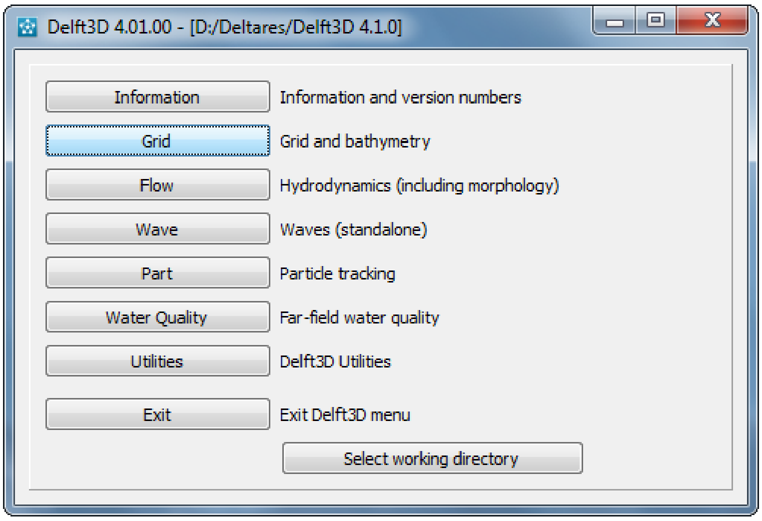


Figure 3.2: *Main window* ***Delft3D-MENU***

3.3 Getting into RGFGRID

To continue start the menu program again as indicated in Section 3.2.

◇ Click the *Grid* button, see Figure 3.2

Next the selection window for **Grid and bathymetry** is displayed for preparing a curvilinear grid, interpolate data on that grid and aggregate the hydrodynamic cells, see Figure 3.3.

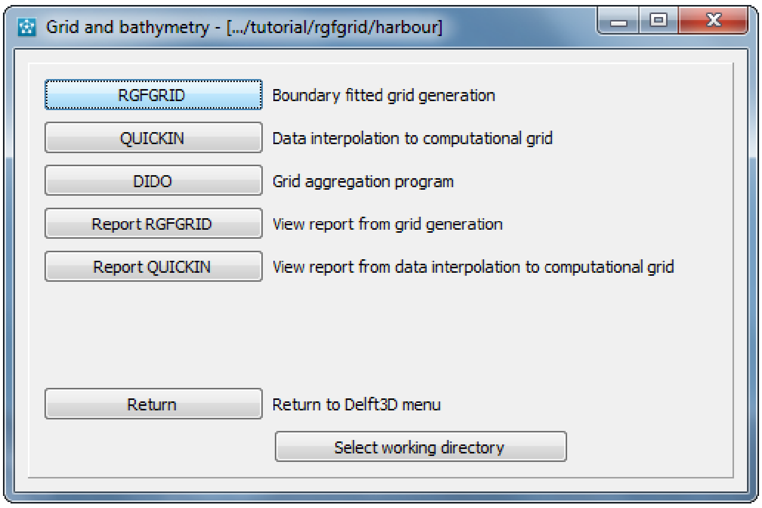


Figure 3.3: *Selection window for* ***Grid and Bathymetry***

Note that in the title bar the current directory is displayed, in our case <D:/delft3d>.

Before continuing with any of the selections of this **Grid and bathymetry** window, you select the directory in which you are going to prepare scenarios and execute computations:

◇ Click the *Select working directory* button.

Next the **Select working directory** window is displayed, see Figure 3.4 (your current directory may differ, depending on the location of your Delft3D installation).

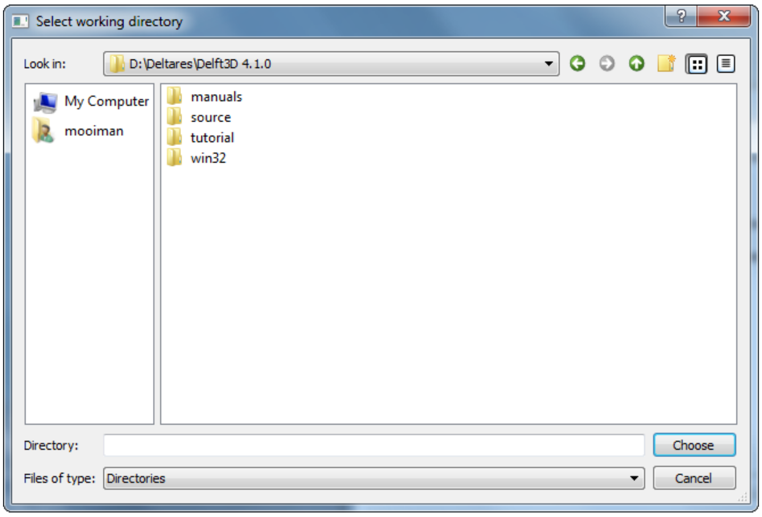


Figure 3.4: Select working directory *window*

- ◇ Browse to and open the <tutorial> sub-directory of your Delft3D Home-directory.
- ◇ Open the <rgfgrid> directory.
- ◇ Open the <harbour> directory.
- ◇ Close the **Select working directory** window by clicking button *Choose*, see Figure 3.5.

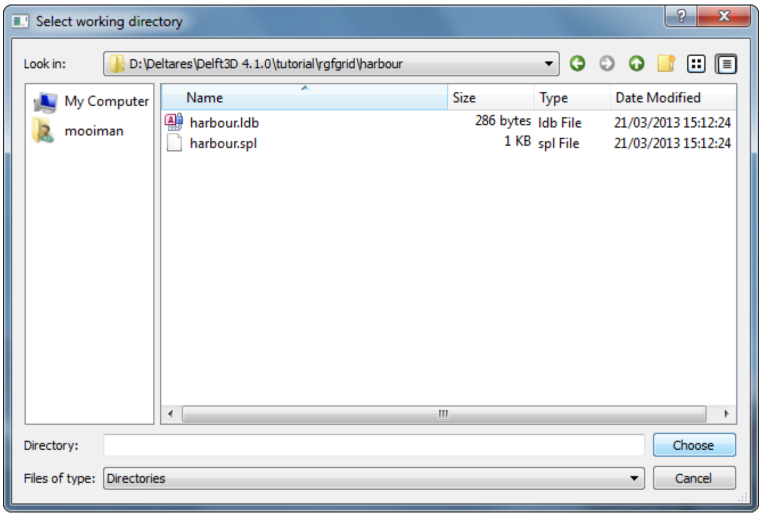


Figure 3.5: Select working directory *window to set the working directory to <rgfgrid/habour>*

Next the **Grid and bathymetry** window is re-displayed, but now the changed current working directory is displayed in the title bar, see Figure 3.6.

Figure 3.6: *A part of the current working directory is shown in the title bar due to its length*

Remark:

- ◇ In case you want to start a new project for which no directory exists yet, you can select  in the **Select working directory** window to create a new folder.

- ◇ Click on *RGFGRDID* in the **Grid and bathymetry** window, see Figure 3.3.

RGFGRID is loaded and the primary input screen is opened, Figure 3.7.

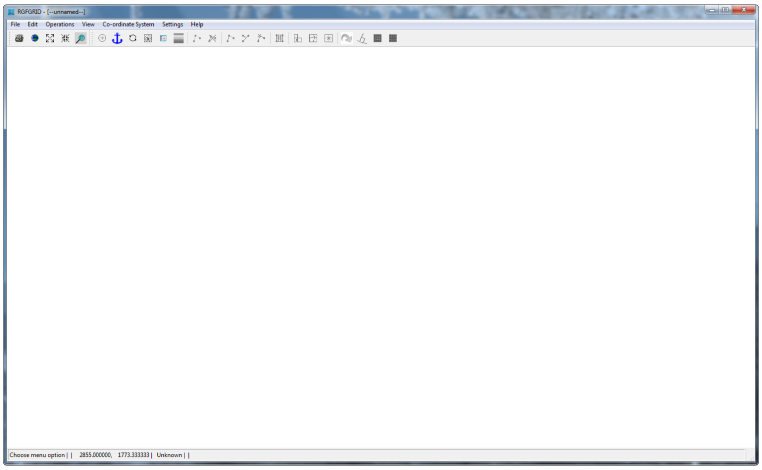


Figure 3.7: *Main window of the RGFGRID*

In the lower-left corner of the status bar RGFGRID gives additional operational information, see Figure 3.8, such as:

- User selections.
- Operational instructions (for instance Toggle anchor mode).
- x and y co-ordinates of the current cursor position.
- Co-ordinate system: Cartesian or Spherical.
- Distance (in metre) to a user-defined anchor point (only displayed when the anchor is activated).

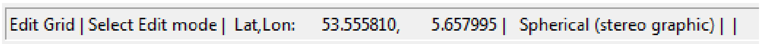


Figure 3.8: *Operational information displayed in the statusbar*

3.4 Exploring some menu options

First, set the co-ordinate system to the system you want to work in. Since we are going to work in the Cartesian co-ordinate system:

- On the *Co-ordinate System* menu click *Cartesian Co-ordinates*, see Figure 3.9

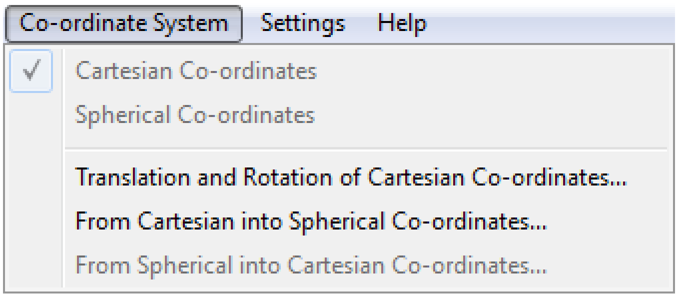


Figure 3.9: Co-ordinate System *menu, Cartesian Co-ordinates selected*

To open a land boundary:

◇ Upon selecting *File* → *Attribute Files* → *Open Land Boundary*, you can open a collection of land boundaries, see Figure 3.10. Land boundaries (or land-water marking) are in files with default mask <∗.ldb>.

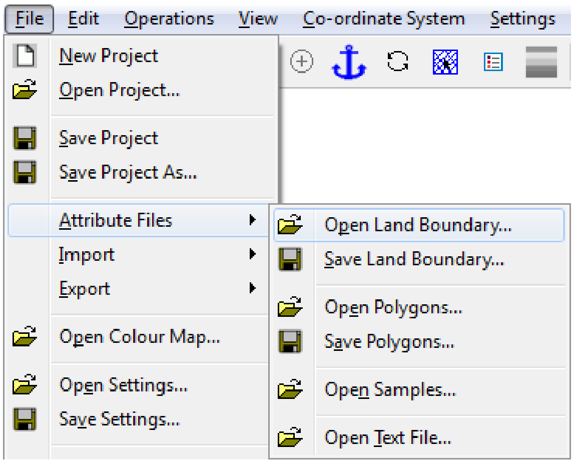


Figure 3.10: *Menu item* File → Attribute Files → Open Land Boundary

Next the **Open Land Boundary** window is displayed, see Figure 3.11.

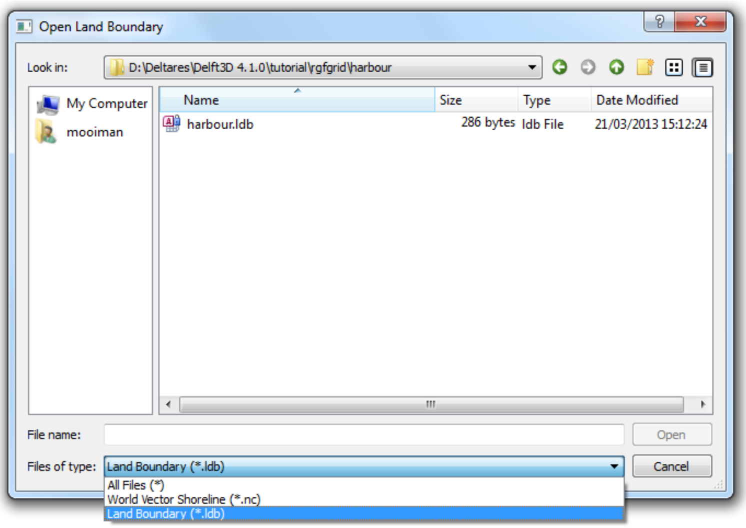


Figure 3.11: *File open window* ***Open Land Boundary***

In the current directory one land boundary file is present.

◇ Select <harbour.ldb> and click *Open* to open the land boundary file.
◇ On the *Edit* menu point to *Spline* and click *New*, or click on the toolbar.
◇ To draw a spline, click with the left-mouse to define spline-points. To finish the current spline click with the right-mouse. Click left to start with the next spline. The result may look like as in Figure 3.12

Practise with zooming in or out. To zoom in or out, either:

◇ Click on to zoom in and zoom out on the toolbar.
◇ Press the + and - key while keeping the `CTRL`-key pressed.

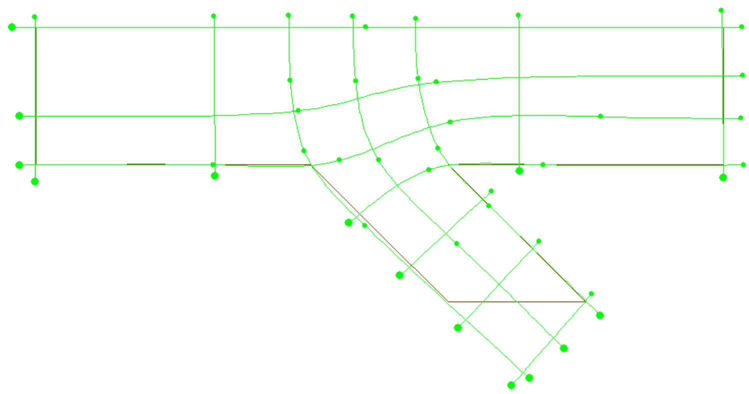


Figure 3.12: *Example of a spline grid*

◇ Use the mouse scroll wheel.

◇ To delete an entire spline, click  on the toolbar and click one of the supporting points of the spline to be deleted.

◇ To delete a single point of a spline, click  and click a spline point to delete this single point.

◇ To move a single point of a spline, click  or press R, click the point and click again at the new location.

Now we delete this spline grid:

◇ On the *Operations* menu, point to *Delete* and click *Splines*, see Figure 3.13

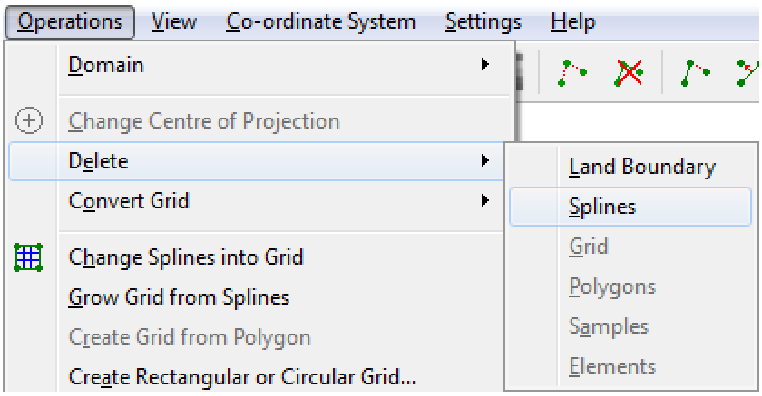


Figure 3.13: *Menu option* Operations → Delete → Splines

We will continue with an existing splines file

◇ On the *File* menu, point to *Import* and click *Splines*.

◇ Select <harbour2.spl>. After selection the file is loaded and displayed, see Figure 3.14.

◇ On the *Operations* → *Change Splines into Grid*, or click  on the toolbar.

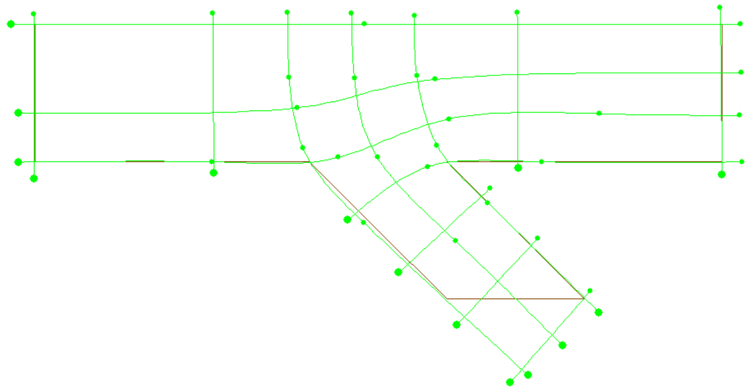


Figure 3.14: *Spline grid from tutorial file <harbour.spl>*

This operations transforms the spline grid into a grid and at the same time refines it 3 times in both directions, see Figure 3.15. The refinement factors can be set in the **General Parameters** form menu item *Settings* → *General*.

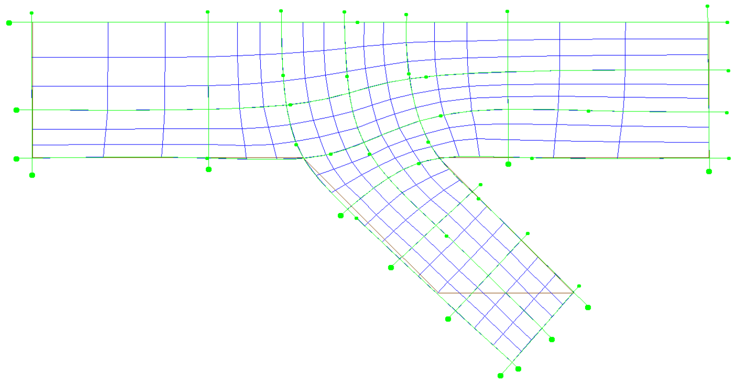


Figure 3.15: *Result of operation* OPerations → Change Splines into Grid

To save the grid

◇ On the *File* menu, point to *Export* and click *Grid*

The **Save As** window opens, see Figure 3.16.

◇ Type <harbour> and click *Save* to save your grid

You will be back in the main window of RGFGRID.

3.5 Exiting RGFGRID

To exit the RGFGRID

◇ Click *Exit* on the *File* menu.

You will be back in the **Grid and bathymetry** window, see Figure 3.3

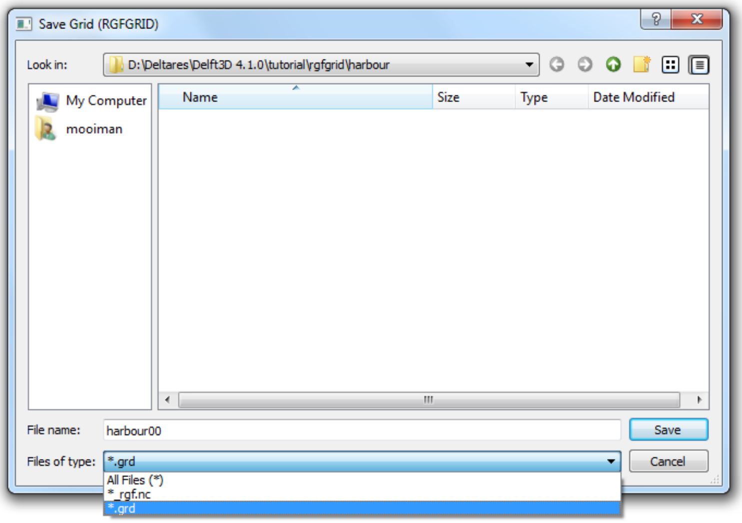


Figure 3.16: *Window **Save Grid** to save grid file*

- ◇ Click *Return* to return to the main window of Delft3D-MENU, see Figure 3.2
- ◇ Click *Exit*.

The window is closed and the control is returned to the desk top or the command line.

In this Getting Started session you have learned to access the RGFGRID and to open and and to generate and save a grid file.

We encourage new users next to run the tutorial described in Chapter 6.

4 General operation

4.1 General program operation instruction

The main menu bar is positioned at the top of the screen.

Help

Upon selecting *Help* → *User Manual*, the RGFGRID User Manual in PDF-format will be opened. Use the bookmarks in the contents to locate the subject you are interested in.

Print screen

Press `Ctrl-P` or click  on the toolbar to obtain the print window for a hardcopy of the current screen.

File menu

The file-menu is the standard **Open** and **Save As** window. The file mask depends on the type of data that you want to open or save. You can change the directory by navigating through the folders.

It is possible to specify whether to Stay on the Start-up Directory or not, in the **Settings General** form.

General cursor and keyboard functions

The left mouse button activates or confirms desired actions. The `Esc` key cancels the last edit action. The right mouse button may also confirm actions, or may put the program back into its original mode.

4.2 Key stroke functions

Key `A` = Anchor, or on toolbar 
When clicking  on the toolbar and next pressing the `A` key on the keyboard, a so-called anchor will appear, which acts as zero-distance point. The distance (in metre) of the present cursor position to this point is displayed in the status bar at the right of the co-ordinate system indicator, see Figure 4.1. Moving the cursor around and pressing `A` again will relocate the anchor. Clicking  again will de-activate the anchor.

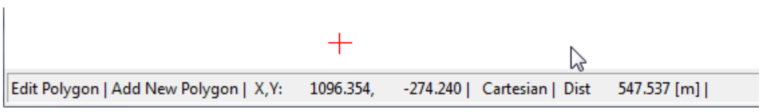


Figure 4.1: *Location of anchor + and distance between anchor and cursor at the right*

Key `D` = Delete
In the *Edit* → *Polygon* options, pressing `D` allows you to delete individual points (polygon, depth or sample).

Key `E` = Erase polygon
In *Edit* → *Polygon*, keeping `E` pressed allows you to delete the indicated polygon.

Key `I` = Insert
In *Edit* → *Polygon*, pressing `I` starts the vertex insert action depending on the first click on the screen, there are two options.

Normal	If the first click is in between two vertices of the polygon then a point will be inserted in the closest edge.
Incremental	If the first click is on a vertex of the existing polygon the the incremental insert option is started, which means all next locations click by the left mouse button are added to the polygon behind the indicated point. This mode will be cancelled when indication a existing point of the polygon and the normal insert option will be used. Hitting another key will also cancel this insert option.

Key `Ctrl-P` = Print screen
Pressing `Ctrl-P` will open the print window. The current screen will be printed to your printer or to a file.

Key `R` = Replace
In *Edit* → *Polygon*, pressing `R` allows you to replace (move) individual points.

Key `Mouse wheel`
Use the mouse wheel to zoom in and zoom out. Other ways are:

Click  on the toolbar to zoom in.

Click  on the toolbar to zoom out.

To define a zoom box, click  on the toolbar and drag a box.

Key `Ctrl +` = Zoom in
Keep the `Ctrl`-key pressed and use the + key to zoom in more.

Key `Ctrl -` = Zoom out
Keep the `Ctrl`-key pressed and use the `-` key to zoom in more.

Key `Ctrl` move cursor = move focus of screen
Keep the `Ctrl`-key pressed and move the cursor around. The current screen will move accordingly.

Key `Ctrl` arrow keys = move focus of screen left, right, up or down
Keep the `Ctrl`-key pressed and use the arrow keys to move the focus of the screen accordingly.

Key `Esc` = Undo
In various edit modes the latest action will be undone pressing `Esc` .

5 Menu options

The menu bar contains the following items, see Figure 5.1, each item is discussed in a separate section.

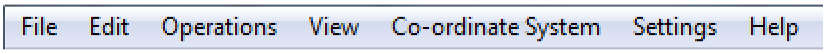


Figure 5.1: *RGFGRID menu options*

5.1 File menu

Before opening an object (land boundary, polygon, samples or splines) be sure you have set the co-ordinate system on the *Co-ordinate System* menu, see Section 5.5.

◇ When opening files, RGFGRID will not check the co-ordinate system in the file against the current co-ordinate system in RGFGRID, except when opening a grid.

On the *File* menu, see Figure 5.2, options are available to open a project (collection of grids and ddb-file), attribute files required for the definition of a grid (i.e. land boundary and samples) and to import grid related files (grids, splines and DD boundaries). The results at each stage of the grid definition process can be saved. The option to quit RGFGRID is located here also.

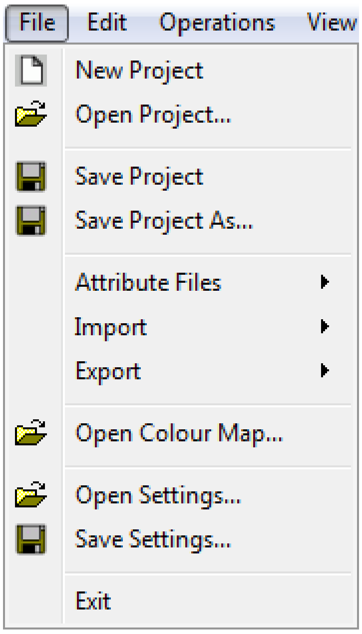


Figure 5.2: *Options on the* File *menu*

The start-up directory to open and save files can be configured in the **General Parameters** form on the menu *Settings* → *General*. As default the file menu starts at the last directory selected.

For the formats of the files you are referred to Appendix A.

5.1.1 New project

Upon selecting *File* → *New Project*, all objects (land boundaries, polygons, splines, grids, samples, etc.) will be deleted; i.e. you start from scratch.

5.1.2 Open project

Upon selecting *File* → *Open Project,* the **Open Project** window appears in which you can browse to an existing project (<∗.d3d> file).

Remark:
- ◇ A project saved by QUICKIN or D-Waq DIDO can be read by RGFGRID.

5.1.3 Save project

Upon selecting *File* → *Save Project*, the current project (grid filenames and, if applicable, DD boundaries filename) will be saved under the same name. If the project name is not known yet, the **Save Project** window appears.

Remark:
- ◇ When you started with an existing project, or when you saved the project before, saving the project will not save changes you have made to the grid(s). Either use **Save Project As** or save individual grids.

5.1.4 Save project as

Upon selecting *File* → *Save Project As*, the current project can be saved under a different name.

5.1.5 Attribute files

On the *File* → *Attribute Files* sub-menu, see Figure 5.3, options are available to open and save objects that are indirectly related to the grids.

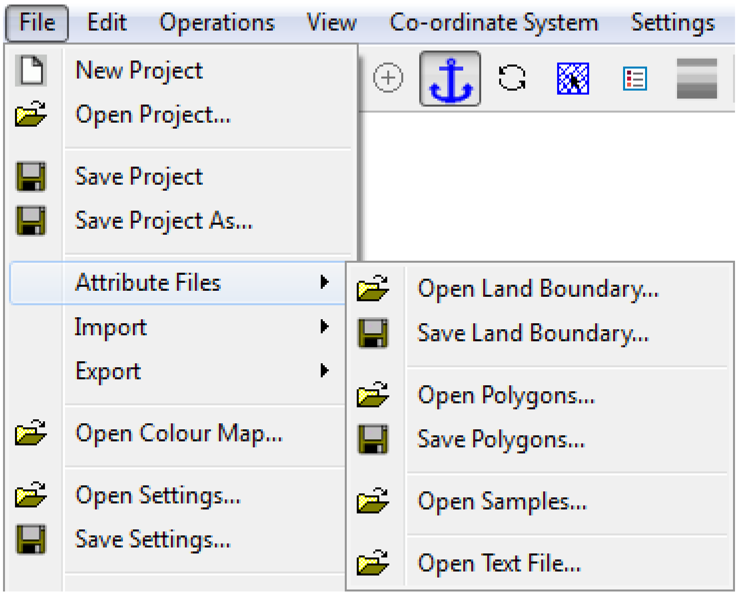


Figure 5.3: *Options on the* File→ Attribute Files *menu*

Open land boundary

Upon selecting *File → Attribute Files → Open Land Boundary*, you can open a collection of land boundaries. Land boundaries (or land-water marking) are in files with default mask <∗.ldb>. For a real application the land boundary is a guidance to define a grid for the model area.

Remark:

◇ If you open another land boundary file, it will be visualised together with the existing land boundary.

Open polygon

Upon selecting *File → Attribute Files → Polygons,* you can open a collection of polygons in a file with mask (<∗.pol>). Polygons are per definition closed. If the polygon is not closed in the file it will still be shown as closed.

Remark:

◇ If you open another polygons file, they will be visualised together with existing polygons.

Save polygons

When saving polygons, each polygon will be saved as a closed polyline. A polygon file has as default mask <∗.pol>.

Open samples

The bathymetry can be used as a guideline to determine the orientation and resolution of the required grid. This can be done visually, but also the grid design can take into account the samples. See *Settings* → *Orthogonalisation*, item *Design Method*, see Section 5.6.3.

The samples in a file with mask <∗.xyz>, may be a set of disordered x, y, z values given in a sequential list of free-formatted x, y, z values.

Remark:
◇ If you open another samples file, the samples will be visualised together with existing samples.

Open text file

Texts can be displayed in the graphics area if their position (x, y), the text and colour are defined. See an example in Appendix A.8.

5.1.6 Import

On the *Import* sub-menu, see Figure 5.4, options are available to import objects that are directly related to the grids.

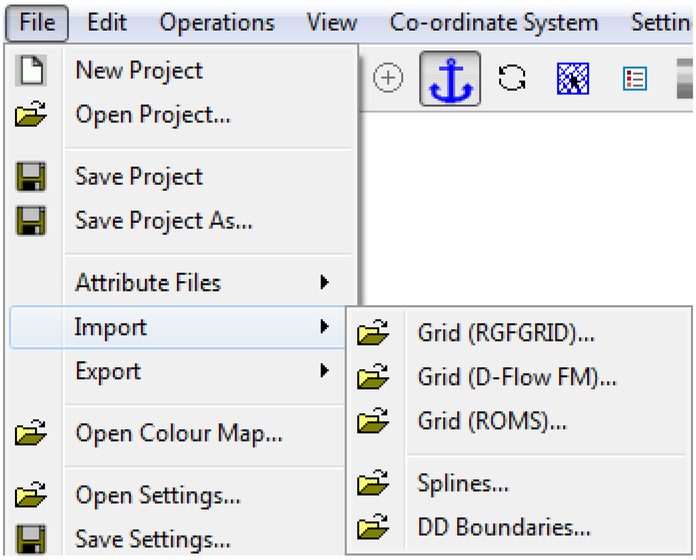


Figure 5.4: File → Import *menu options*

Grid (RGFGRID)

Upon selecting *File* → *Import* → *Grid (RGFGRID)*, you can open a collection of grids. The grid file has a default mask <∗.grd> or <∗_rgf.nc>.

Remarks:
- ◇ The co-ordinate system in RGFGRID is set accordingly to the system specified in the grid file.
- ◇ If the co-ordinate system is spherical then the co-ordinates are shown in stereographic projection.
- ◇ If no co-ordinate system is specified, Cartesian is presumed.

Grid (D-Flow FM)

Upon selecting *File* → *Import* → *Grid (D-Flow FM)*, you can open a collection of grids. The grid file has a default mask <∗_net.nc>.

Grid (ROMS)

Upon selecting *File* → *Import* → *Grid (ROMS)*, you can open a collection of regular grids in the NetCDF format off the Regional Ocean Modeling System. The grid file has a default mask <∗_roms.nc>.

Splines

The initial sketch of the grid is done by drawing splines. Splines are in files with default mask <∗.spl>.

Remark:
- ◇ If you open another splines file, the new splines will replace existing splines.

DD boundaries

In case of a domain decomposition application you will have multiple grids. How the grids are linked to each other is contained in the domain decomposition boundary file (ddb-file). The ddb-file will be made if you select *Operations* → *Compile DD Boundaries*, see Section 5.3.16.

5.1.7 Export

On the *File* → *Export* sub-menu, see Figure 5.5, options are available to export objects that are directly related to the grids.

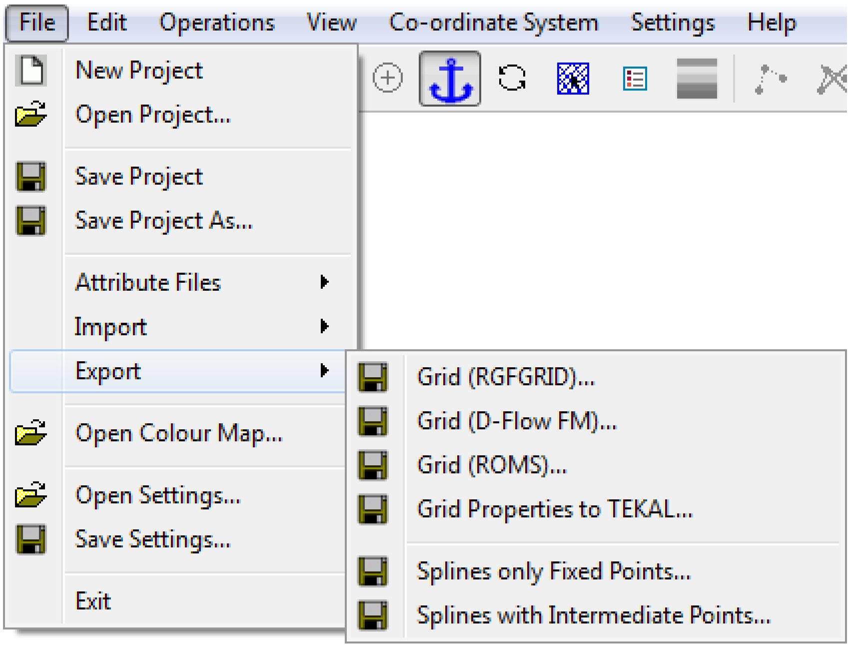


Figure 5.5: File → Export *sub-menu options*

Grid (RGFGRID)

The grid is saved in a file with mask <∗.grd> or <∗_rgf.nc>. Along with the <∗.grd> file, a second file is saved with mask <∗.enc>, containing the so-called grid enclosure, that outlines all active computational grid cells in Delft3D-FLOW.

Grid (D-Flow FM)

The grid is saved in the NetCDF file format suitable for D-Flow FM, the default mask <∗_net.nc> is used.

Grid (ROMS)

The grid is saved in the NetCDF file format suitable for the Regional Ocean Modeling System, the default mask <∗_roms.nc> is used.

Grid properties to TEKAL

The grid properties can be saved in a so-called TEKAL format, so that the properties can be visualised with Delft3D-QUICKPLOT or GPP, see QUICKPLOT (2013) and GPP (2013). The data is saved in a file with mask <∗.tek>, and contains the x, y co-ordinates, the orthogonality, the resolution, the smoothness, the curvatures, the grid sizes and the aspect ratios in columns.

Splines only fixed points

Splines are saved in a file with default mask <*.spl>. Only those points which are visualised with a dot are stored in the file.

Splines with intermediate points

The splines including the intermediate points between the points visualised with a dot, can be saved in a file with default file mask <*.spt>.

5.1.8 Open Colour map

You can choose from a number of pre-defined colour schemes (in file with masks <*.clr> or <*.clrmap>). These colour schemes have the same format as used for Delft3D-QUICKPLOT, see Appendix A.10 for the file format.

Restriction:

- ◇ Only the colour space `RGB` is supported

Remark:

- ◇ If the file <rgfgrid.clrmap> exists on the start-up directory then this file will be read, if the file does not exist on the start-up directory it will try to read the file on the installation directory <$D3D_HOME/$ARCH/plugins/default>.

5.1.9 Open Settings

If you have saved your RGFGRID settings in a previous session, you can open these settings again, see Appendix A.11 for the file format.

Remark:

- ◇ If the file <rgfgrid.ini> exists on the start-up directory then this file will be read, if the file does not exist on the start-up directory it try to read the file on the installation directory <$D3D_HOME/$ARCH/plugins/default>.

5.1.10 Save Settings

If you have made changes in one of the forms on the *Settings* menu, you can save these settings to be used later on again.

5.1.11 Exit

Exit from the RGFGRID program.

5.2 Edit menu

On the *Edit* menu, see Figure 5.6, several edit modes can be selected.

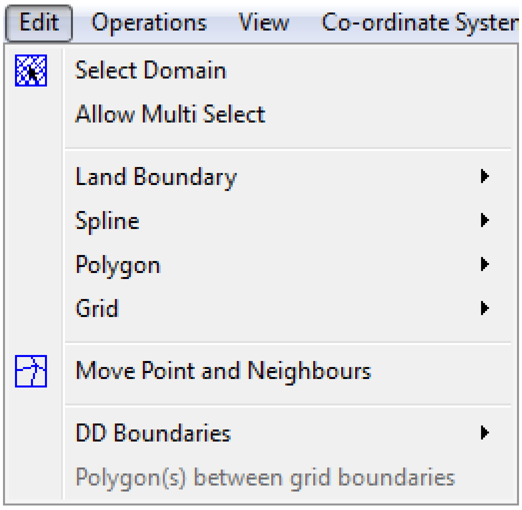


Figure 5.6: *Options on the* Edit *menu*

An edit mode is an operation mode which needs at least a mouse click, i.e. a set of operation instructions which is valid for a certain data set, and which may go with some specific display method. The following objects may be modified:

- Polygon
- Spline
- Grid
- DD Boundaries

`Esc` = Undo
In most edit modes, `Esc` will undo the latest action.

5.2.1 Select domain

If your project consists of multiple grids (so-called domain decomposition application) you can switch between the domains (grids) by clicking *Edit* → *Select Domain*, or click  on the toolbar. Next, click on the grid you want to become the active grid.

5.2.2 Delete domain

To delete a domain or grid, select *Edit* → *Delete Domain*. Next, click on an active or inactive grid. When deleting a domain, also objects (except dredge and dump areas) defined on the grid will be deleted.

5.2.3 Allow Multi Select

When selecting option *Edit* → *Allow Multi Select* you are able to select more than one polyline of the land boundary, polygon or grid. For example, to merge several irregular grids use this option to select which domains need to be merged.

5.2.4 Land Boundary

The land boundary is used to visualise the land-water interface. To edit (define or modify) a land boundary, for possible edit actions see Figure 5.7.

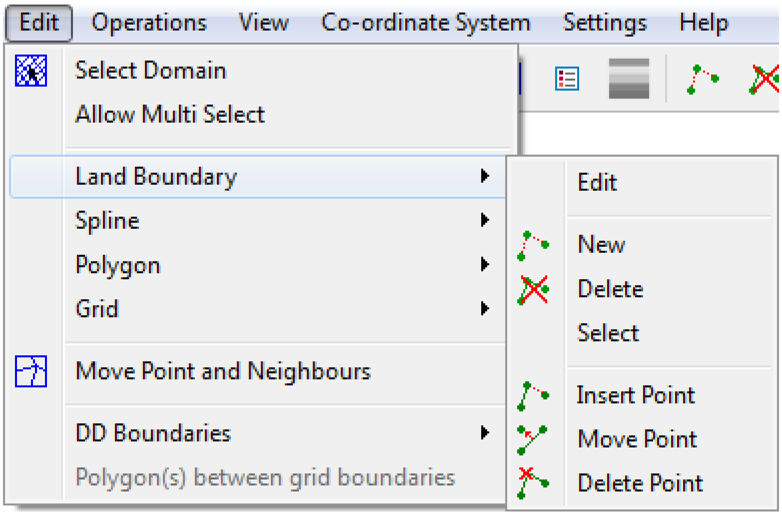


Figure 5.7: *Options on the* Edit → Land Boundary *menu*

5.2.4.1 Menu options

The key stroke to reach the menu item *Edit* → Land Boundary → *Edit* is: `CTRL+ALT+L`

Edit

Upon selecting *Edit* → *Land Boundary* → *Edit*, you can start editting a polyline that defines an Land Boundary. When there is no polyline the edit mode is set to *New*, otherwise you have to select first a polyline (from the menu *Edit* → *Land Boundary* → *Select* or press the key `s`). After you have selected the polyline you can use key-strokes, icons in the toolbar or menu items to switch the edit mode.

New

Upon selecting *Edit* → Land Boundary → *New*, you can start to define a new polyline, click on , or use the key-stroke `n` to start a new polyline.

Delete

Upon selecting *Edit* → *Land Boundary* → *Delete*, click on  , or use the key-stroke `e`, to delete (erase) the selected polyline.

Select

Upon selecting *Edit* → *Land Boundary* → *Select*, or use the key-stroke `s`, you can select a polyline by clicking on one of its edges or vertices. After that the polyline will be highlighted

Insert point

Upon selecting *Edit* → *Land Boundary* → *Insert Point*, click on , or use the key-stroke `i`, you can insert a point into the selected polyline. The point will be inserted at the nearest linear piece of the polyline.

Move point

Upon selecting *Edit* → *Land Boundary* → *Move Point*, click on , or use the key-stroke `r`, you can move (replace) a point on the selected polyline.

Delete point

Upon selecting *Edit* → *Land Boundary* → *Delete Point*, click on , or use the key-stroke `d`, you can delete a point on the selected polyline by indicating it.

5.2.4.2 Valid action keys are

The key stroke to reach the menu item *Edit* → Land Boundary → *Edit* is: `CTRL+ALT+L`

In *Edit* → *Land Boundary* mode the following keys can be used (mode is indicated in the statusbar):

- ◇ Key `a`: Add point after
  Add a point after the last point of a selected land boundary (after)
- ◇ Key `b`: Add point before
  Add a point before the first point of a selected land boundary (before)
- ◇ Key `d`: Delete single point
  delete single point in selected land boundary
- ◇ Key `i`: Insert single point
  Insert single point in the selected land boundary
- ◇ Key `r`: Move (replace) single point
  Replace single point in the select land boundary
- ◇ Key `s`: Select land boundary
  Select a spline
- ◇ Key `x`: Delete land boundary
  Delete a complete land boundary

5.2.5 Spline

On the *Edit* menu, point to *Spline*, see Figure 5.9

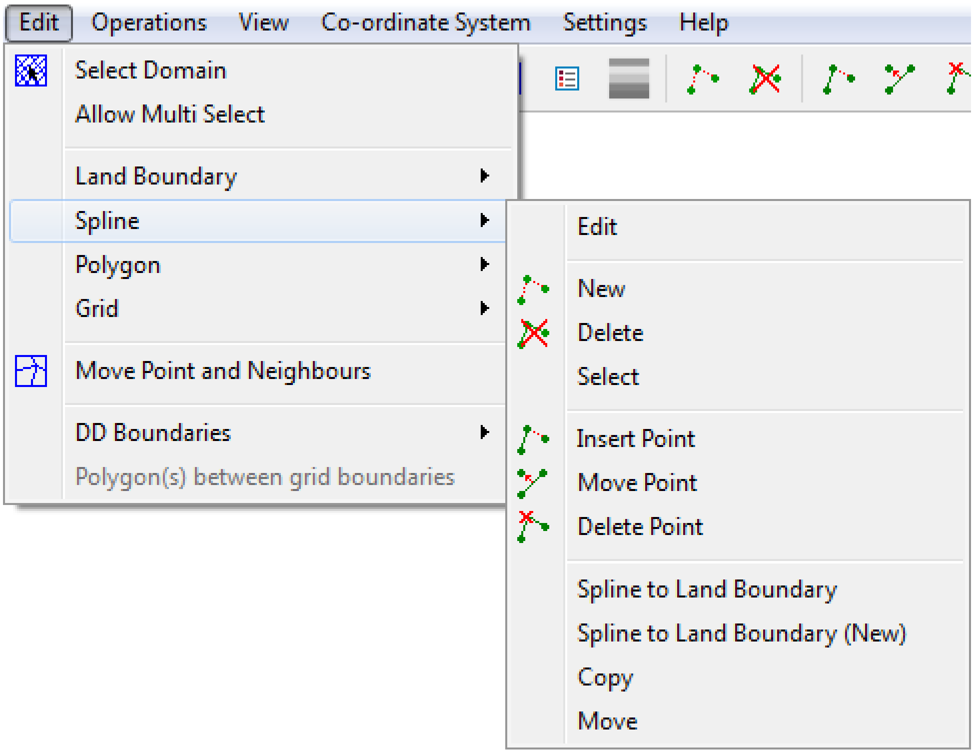


Figure 5.8: *Options on the* Edit → Spline *menu*

5.2.5.1 Menu options

The key stroke to reach the menu item *Edit* → Spline → *Edit* is: `CTRL+ALT+S`

Edit

Upon selecting *Edit* → *Spline* → *Edit*, you can start editting a spline (handled as a polyline). When there is no spline the edit mode is set to *New*, otherwise you have to select first a spline (from the menu *Edit* → *Spline* → *Select* or press the key `s`). After you have selected the spline you can use key-strokes, icons in the toolbar or menu items to switch the edit mode.

New

Up on selecting *Edit* → *Spline* → *New* you can start defining a new spline. Click the left mouse button at different positions to create a spline. To start a new spline, click the right mouse button and click the left mouse button again to create the next spline.

Delete

Up on selecting *Edit* → *Spline* → *Delete* you can delete a spline . Click with the left mouse button on a spline point, than that spline will be deleted.

Insert Point

Upon selecting *Edit* → *Spline* → *Insert Point*, you can insert a point. But first you have to select the spline in which you want to insert a point.

Move Point

Upon selecting *Edit* → *Spline* → *Move Point*, you can move a point on a spline. But first you have to select the spline in which you want to insert a point.

Delete Point

Upon selecting *Edit* → *Spline* → *Delete Point*, you can delete a point of a spline. But first you have to select the spline in which you want to insert a point.

5.2.5.2 Valid action keys are

The key stroke to reach the menu item *Edit* → Spline → *Edit* is: `CTRL+ALT+L`

In *Edit* → *Spline* mode the following keys can be used (mode is indicated in the statusbar):

- Key `a`: Add point after
  Add a point after the last point of a selected spline (after)
- Key `b`: Add point before
  Add a point before the first point of a selected spline (before)
- Key `d`: Delete single point
  delete single point in selected spline
- Key `i`: Insert single point
  Insert single point in the selected spline
- Key `r`: Move (replace) single point
  Replace single point in the select spline
- Key `s`: Select spline
  Select a spline
- Key `x`: Delete spline
  Delete a complete spline

5.2.6 Polygon

The polygon is used to limit the area of influence of operations and or edit actions. All grid points and samples that are inside the polygon are active in the subsequent interpolation or manipulation steps. The polygon is self closing.

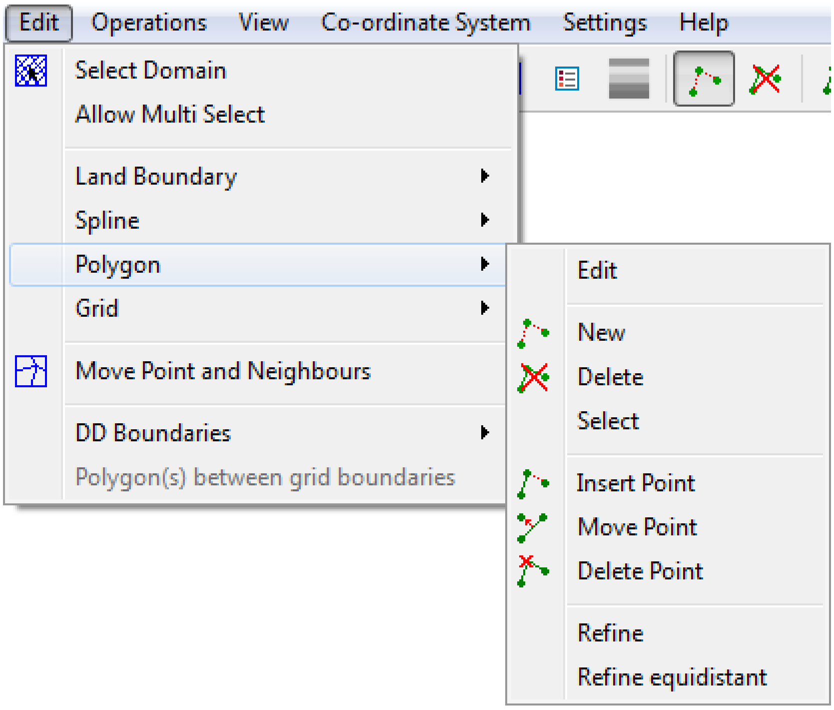


Figure 5.9: *Options on the* Edit → Polygon *menu*

5.2.6.1 Menu Options

The key stroke to reach the menu item *Edit* → Polygon → *Edit* is: `CTRL+ALT+P`

Edit

Upon selecting *Edit* → *Polygon* → *Edit*, you can start editting a polygon that defines an area of interest. When there is no polygon the edit mode is set to *New*, otherwise you have to select first a polygon (from the menu *Edit* → *Polygon* → *Select* or press the key `s`). After you have selected the polygon you can use key-strokes, icons in the toolbar or menu items to switch the edit mode.

New

Upon selecting *Edit* → Polygon → *New*, you can start to define a new polygon, click on , or use the key-stroke `n` to start a new polygon.

Delete

Upon selecting *Edit* → *Polygon* → *Delete*, click on  , or use the key-stroke `e`, to delete (erase) the selected polygon.

Select

Upon selecting *Edit* → *Polygon* → *Select*, or use the key-stroke `s`, you can select a polygon by clicking on one of its edges or vertices. AFter that the polygon will be highlighted

Insert point

Upon selecting *Edit* → *Polygon* → *Insert Point*, click on , or use the key-stroke `i`, you can insert a point into the selected polygon. The point will be inserted at the nearest linear piece of the polygon.

Move point

Upon selecting *Edit* → *Polygon* → *Move Point*, click on , or use the key-stroke `r`, you can move (replace) a point on the selected polygon.

Delete point

Upon selecting *Edit* → *Polygon* → *Delete Point*, click on , or use the key-stroke `d`, you can delete a point on the selected polygon by indicating it.

Refine

Choosing the option *Edit* → *Polygon* → *Refine* you are able to refine the selected polygon. This is done by selecting two vertices of the polygon. When selecting the two vertices the part where the refinement will be take place is marked with circles instead of dots. If you need the other part of the polygon hit the `i` to invert the selection.

The algorithm for the refinement is as follows:
The distance between the first selected point and marked neighbour, and the distance between the last selected point and its marked neighbour is determined. The refinement distance will be the linear interpolation between this two values, so a gradual change of distance along the marked part of the polygon. Exiting vertices are replace by the new ones.

Refine equidistant

To insert points to obtain an equidistant refinement on the edges of a polygon, select *Edit* → *Polygon* → *Refine equidistant*. Next, click on one or more edges from the polygon to select or deselect them, or double-click to select or deselect all edges from the polygon.

Keys actions

In *Edit* → *Polygon* → *Refine equidistant* mode the following keys can be used:

- ◇ Key `p` = Polyline refine
  Pressing `p` allows you to refine the selected polyline(s) of the polygon using a distribution ds. The message at the left of the statusbar now reads 'Press Return to accept Refinement or P to change Refinement'.
  Pressing `Return` will refine the polyline(s) within the polygon using the current value of ds, pressing `p` will allow the user to redefine ds.
- ◇ Key `l` = Snap selected polyline(s) to landboundary
  Pressing `l` allows you to snap selected polyline(s) to an existing landboundary. The message at the left of the statusbar now reads 'Press Return to accept attachment or S to (re)select segments'.
  Pressing `Return` will snap the selected polyline(s) to the landboundary, pressing `s` will allow the user to (re)select polyline(s) within the polygon.

5.2.6.2 Valid action keys are

The key stroke to reach the menu item *Edit* → Polygon → *Edit* is: `CTRL+ALT+P`

In *Edit* → *Polygon* mode the following keys can be used:

- ◇ Key `I`: Insert
  In *Edit* → *Polygon*, pressing `I` starts the vertex insert action depending on the first click on the screen, there are two options.

Normal	If the first click is in between two vertices of the polygon then a point will be inserted in the closest edge.
Incremental	If the first click is on a vertex of the existing polygon the the incremental insert option is started, which means all next locations click by the left mouse button are added to the polygon behind the indicated point. This mode will be cancelled when indication a existing point of the polygon and the normal insert option will be used. Hitting another key will also cancel this insert option.

  The message at the left of the statusbar now reads 'Insert a point', click the left mouse to insert individual points.
- ◇ Key `r`: Replace
  Pressing `r` allows you to replace (move) individual polygon points. The message at the left of the statusbar now reads 'Replace: Get a Vertex'. If you have got it by clicking the left mouse, the message will read 'Replace: Put a Vertex', and you can do so by clicking the left mouse at the new desired position.
- ◇ Key `d`: Delete
  Pressing `d` allows you to delete individual polygon points.
- ◇ Key `x`: Break open polygon
  Keep *X* pressed and move with the cursor over a polygon point to split the polygon. Effectively, the pointed co-ordinates are replaced by default 'missing' values.
- ◇ Key `e` : Erase polygon
  Entire polygon sections are deleted. Press key `e` and then click with the left mouse button on a point of the polygon which need to be deleted. Finish the operation by pressing the right mouse button.

5.2.7 Grid

On the *Edit* menu, point to *Grid* you can edit the grid, see Figure 5.10

The key stroke to reach the menu item *Edit* → Grid → *Edit* is: `CTRL+ALT+G`

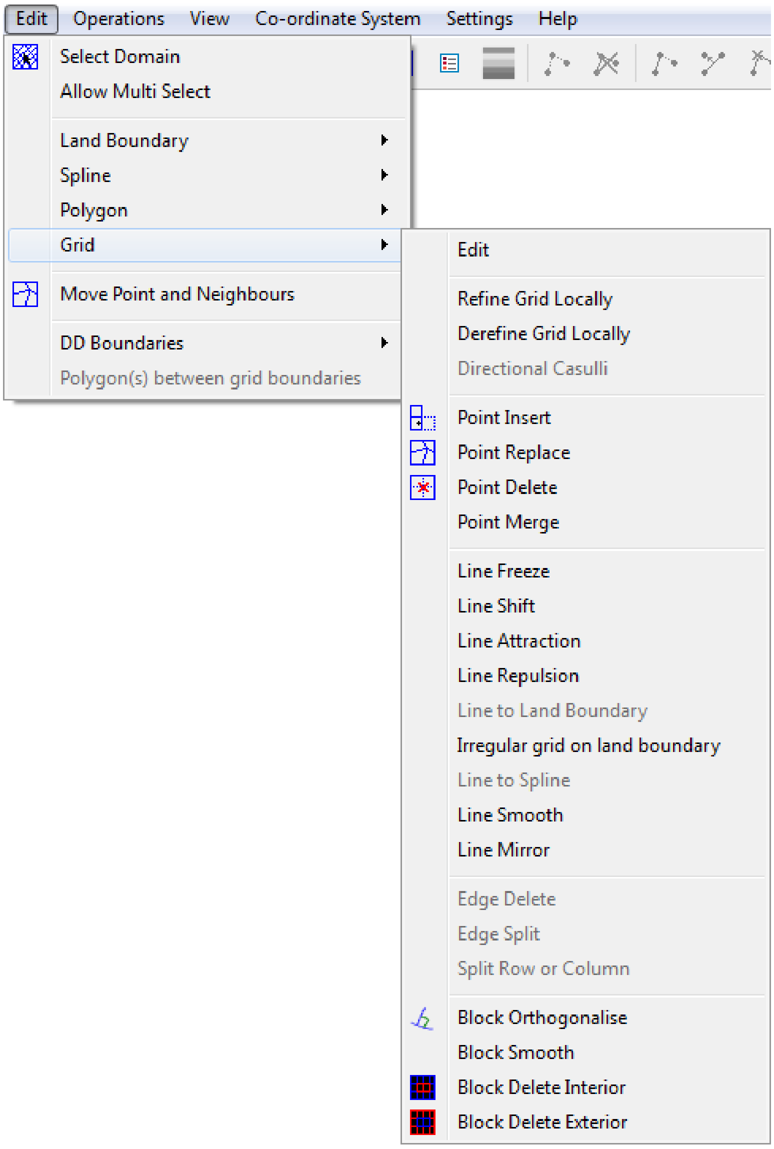


Figure 5.10: *Options on the* Edit → Grid *menu*

5.2.7.1 Menu Options

Edit

Upon selecting *Edit* → *Grid* → *Edit*, you can start editing a grid. When there is no grid the edit mode is set to *New*, which menas start editing an irregular grid. Otherwise you have to select first a grid (from the menu *Edit* → *Grid* → *Select* or press the key `s`). After you have selected the grid you can use key-strokes, icons in the toolbar or menu items to switch the edit mode.

Refine grid locally

This option operates on part of the grid and the direction depends on the grid line indicated by you.

First you specify (on the *Settings* → *General* menu) the number of times that the grid has to be refined in the M- or N-direction (see Section 5.3.10). Then you indicate 2 points on a grid line between which the refinement has to be performed.

Derefine grid locally

This option operates on part of the grid and the direction depends on the grid line indicated by you. This operation is the opposite of *Refine Grid Locally*. First you specify (menu *Settings* → *General)*, the number of times that the grid has to be de-refined in the M or N direction (see Section 5.3.10). Then you indicate on a grid line 2 points between which the de-refinement has to be performed. Next, smooth the jump in grid sizes.

Directional Casulli

TODO(??): Under construction

5.2.7.2 Point

On the *Edit* menu, point to *Grid* and click on one of the options to operate on individual grid points. To insert, delete or move grid points you can either use the menu options, the icons on the toolbar, or the keyboard to switch between these operations.

After selecting one of the options: insert, move or delete point the program is in point edit mode

Point Insert

Press the `I`-key, use the toolbar icon  or click the menu item *Edit* → *Point* → *Insert* to bring the progrma into insert mode.

If the program is in insert mode, (message 'Select Grid Cell' at the lower left side of the screen), click the left mouse inside a grid cell to create a new grid cell at the border of the grid. The indicated grid cell will be 'mirrored' to the grid cell side closest to the clicking point.

Point Replace

Press the `R`-key (Replace), use the toolbar icon  or click the menu item *Edit* → *Point* → *Move* to bring the program into replace mode.

The message at the lower left of the screen now reads 'Get a point'. Click left to indicate a grid point; the message will read 'Put a point'. Move the cursor to the desired position and click left again.

Point Delete

Press the `D`-key, use the toolbar icon  or click the menu item *Edit* → *Point* → *Delete* to bring the program into delete mode.

If the program is in delete mode, delete grid points by just clicking them.

Points Merge

Irregular grid: Upon selecting *Edit* → *Grid* → *Points Merge* two nodes can be merged. Select a node by the left mouse button and than select a node to which the first selected node is merged to.

5.2.7.3 Line

The operations line freeze, line shift, line attraction, line repulsion and line smooth operate in line mode, see Figure 5.10.

They all use the same procedure to indicate a line and an influence area.

You first indicate a line by marking its end points, using the left mouse; next you indicate the influence area by marking one or two grid-points at one or both side of the line, respectively. Pressing `Esc` enables the replacement of the last added point; pressing `Esc+Esc` cancels all the selected block points, after you may redo the selection procedure. You click the right mouse for the final selection of line and area. After the indication you perform the operation (e.g. line shifting, attraction or repulsion). The result can still be reversed (by pressing several times the `Esc` key).

Line Freeze

Frozen lines are grid lines that are kept fixed in the orthogonalisation process. That is, the end points are kept fixed and the points in between can only move in the direction along the grid line. Frozen lines can be edited by clicking 2 points that lie on the same grid line. You can unfreeze grid lines by first pressing the `D` key and click with the left mouse on one of the endpoints. You can also use `I` (insert) mode to define lines to freeze.

Line Shift

This option provides the possibility to fit the grid's edges to a land boundary. First you indicate a line and indicate the influence area. Then, you can shift the line by shifting some or all of the individual points of that line. The end points can also be shifted. After clicking the right mouse to indicate that the line has been put into the correct new position, the points on the line between the end-points will be shifted by linear interpolation between all repositioned points. Then, a field transformation will be performed in the influence area, with centre points that are now consecutive points on the shifted centre line. If you are not satisfied with the transformed result, press several times the `Esc` key. You will then be put back into *Edit* → *Line* → *Shift* mode. You can carry on shifting lines by simply repeating the same sequence of actions.

Line Attraction

Here, you have again to 'Indicate a line', by marking its end points, and to 'Indicate an influence area' (see *Edit* → *Line* → *Shift*). The grid will be attracted to the indicated line, making use of the line transformation described above, in the field indicated by the influence area.

In *Settings* → *General* the parameter *Attraction/Repulsion Parameter* can be changed, see Figure 5.41.

Line Repulsion

The reverse of *Edit* → *Line* → *Attraction*.

In *Settings* → *General* the parameter *Attraction/Repulsion Parameter* can be changed, see Figure 5.41.

Line to Land Boundary

The edge of the grid can be fitted to a land boundary by hand, using the *Edit* → *Line* → *Shift* option, or automatically, using the present option, *Edit* → *Line* → *Line to Land Boundary*. The automatic option may not always deliver exactly what you want. This can be caused by irregular shapes in the land boundary. However, we do not want to be compelled to analyse and polish up the land boundary a priori, in the digitising phase.

Therefore, both the automated and hand option are included in the program. Just indicate the first and last point of the line that you want to fit to the land boundary. Then click the right mouse. Next, all intermediate points will be translated to their nearest land boundary. Then, a line shift will be performed, equal to the one mentioned above, shifting the indicated line and the surrounding grid. Press `Esc` three times if the result is unsatisfactory. The original grid will then be restored. The algorithm which decides to which land boundary line segment the grid line should be attracted, first looks for the closest land boundary point. An error may occur here, if the closest land boundary line segment is very long, and land boundary points of other segments are more close to the indicated grid line. In that case open the land boundary as a polygon and add (insert) some points to the long land boundary segment, so that points on this segment are closest to the indicated grid line.

Irregular grid on Land Boundary

TODO(??): Under construction

Line to spline

Similar as line to land boundary. If you do not need the spline grid anymore, first delete the splines and then draw just 1 spline to which you want to attach the grid.

Line Smooth

You have to 'Indicate a line', by marking its end points, and to 'Indicate an influence area' (see *Edit* → *Line* → *Shift*). Within this area, the grid will be smoothed into the direction indicated by the line.

The smoothing process can be configured, see Section 5.6.1, parameters *Number Smoothing Iterations* and *Smoothing Parameter*.

Line Mirror

Indicate a grid line at the edge of the grid by marking its end points. Click right to execute the mirror process; grid cells will be created. After this the operation can be repeated by using the key `CTRL+M`

5.2.7.4 Block

Block delete, block cut, block orthogonalise and block smooth all operate in block mode, see Figure 5.10. An influence area (block) is indicated by clicking two, three or four points.

Block orthogonalise

Click two, three, or four points to indicate the corners of the grid block. A minimal block is selected which just contains the selected points. Press `Esc` if you want to replace the latest indicated point, press `Esc+Esc` to redo the selection of the block. Clicking right results in the orthogonalisation of the grid inside the selected block. Press `Esc+Esc+Esc` if you want to cancel the latest action, or click *Undo* on the *Operations* menu.

You can specify parameters that control the orthogonalisation in *Settings* → *Orthogonalisation*, see Figure 5.43.

Block smooth

Click two, three, or four points to indicate the corners of the grid block. A minimal block is selected which just contains the selected points. Press `Esc` if you want to replace the latest indicated point, press `Esc+Esc` to redo the selection of the block. Clicking the right mouse results in the smoothing of the grid inside the selected block. Press `Esc+Esc+Esc` if you want to cancel the latest action.

The smoothing process can be configured, see *Settings* → *General*, parameters *Number Smoothing Iterations* and *Smoothing Parameter*, see Figure 5.41.

Block delete interior

Click two points to indicate the corners of the grid block that you want to delete. A minimal block is selected which just contains the selected points. Clicking right results in the annihilation of the block area. Press `Esc` if you want to replace the latest indicated point, press `Esc+Esc` to redo the selection of the block. Press `Esc+Esc+Esc` if you want to cancel the latest action, or select *Undo* on the *Operations* menu.

Block delete exterior

Click two points to indicate the corners of the grid block. A minimal block is selected which just contains the selected points. Clicking right results in the annihilation of the grid in the area outside the selected block. Press `Esc` if you want to replace the latest indicated point, press `Esc+Esc` to redo the selection of the block. Press `Esc+Esc+Esc` if you want to cancel the latest action, or click *Undo* on the *Operations* menu.

5.2.8 Move Point and Neighbours

Modifications will be made by shifting the centre point of a field of points. The field transformation is based upon the relative shift of the centre point. For all cells in the vicinity of the centre, that shift is transformed to their local grid cell orientation and will be decreased in magnitude in proportion to the physical distance to the centre cell. In that way a quasi-orthogonal transformation is induced. The area of influence is always one sixth of the area that is currently displayed on the screen. (So, if you want to decrease or increase the area of influence, zoom in or zoom out).

5.2.9 DD Boundaries

This option is only relevant to users of the Delft3D domain-decomposition system, or if you want to keep some parts of the boundary fixed in the orthogonalisation.

At the interface between two grids of a multi-domain model (DD Boundaries) the grids should satisfy the following rules:

- ◇ At sub-domain interfaces the grids should be nicely connected (no overlap and “no holes” between sub-domains).
- ◇ In case of horizontal grid refinement, grid lines in the coarse domain should be continued in the fine sub-domain, see Figure 5.11. Thus, there should be a 1-to-N refinement, with N an integer number.

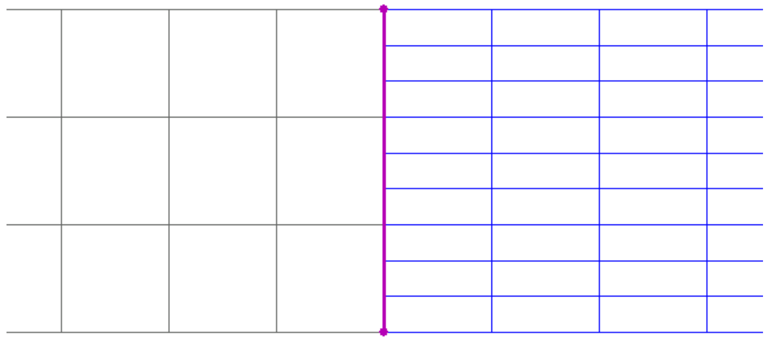


Figure 5.11: *Example of a 1-to-3 refinement along a DD boundary*

◇ Each grid line should cover or be covered by another grid line. The domain decomposition of Figure 5.12 does not fulfil this requirement. Although the DD-boudaries A and B have a correct refinement factor.

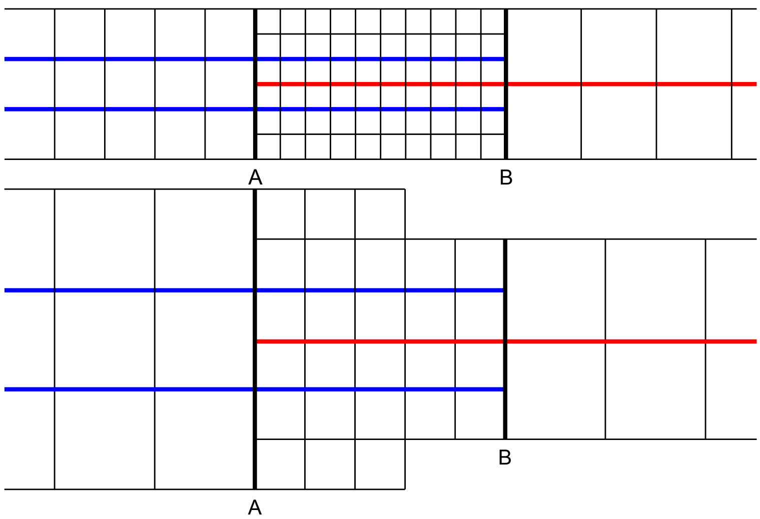


Figure 5.12: *Two examples of not allowed domain decompositions, although both DD-boundaries (A and B) satisfy the refinement condition; the red line and blue lines do not cover each other*

◇ Grids must be of the same type (thus, all in spherical co-ordinates, or all in Cartesian co-ordinates).
◇ The grid orientation should be the same (increasing M- and N-numbering in the same direction).
◇ No coupling of columns to rows or vice versa.
◇ Sub-domain interfaces should be straight lines (no stair-case interfaces).

DD boundaries can be edited by clicking boundary points that lie on the same grid line, see Figure 5.13. You can delete boundary points by first pressing the `D` key and click with the left mouse. `R` (replace) mode and `I` (insert) mode are also available. The specified boundaries are saved together with the grid in a file with mask $<$$*$.ddb$>$. This file is created when selecting *Operations* → *Compile DD Boundaries*. See Appendix A.9 for the format of this file.

DD boundaries are also used in the orthogonalisation process. Because DD boundaries can only be located on boundary points, their administration can be used to fix boundary points in the orthogonalisation process.

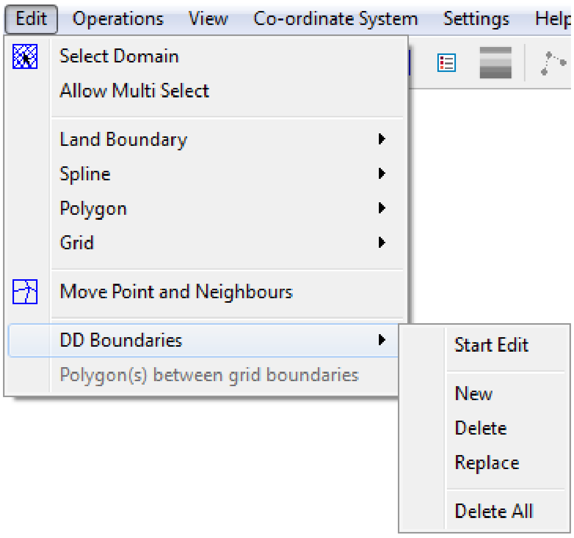


Figure 5.13: *Options on the* Edit → DD Boundaries *menu*

New

Start defining a new DD boundary.

Delete

Delete a single point of a DD boundary.

Replace

Replace a single point of a DD boundary.

Delete All

Delete all domain-decomposition boundaries.

Remark:

◇ DD Boundaries can also be defined in a single domain, see Figure 5.14.

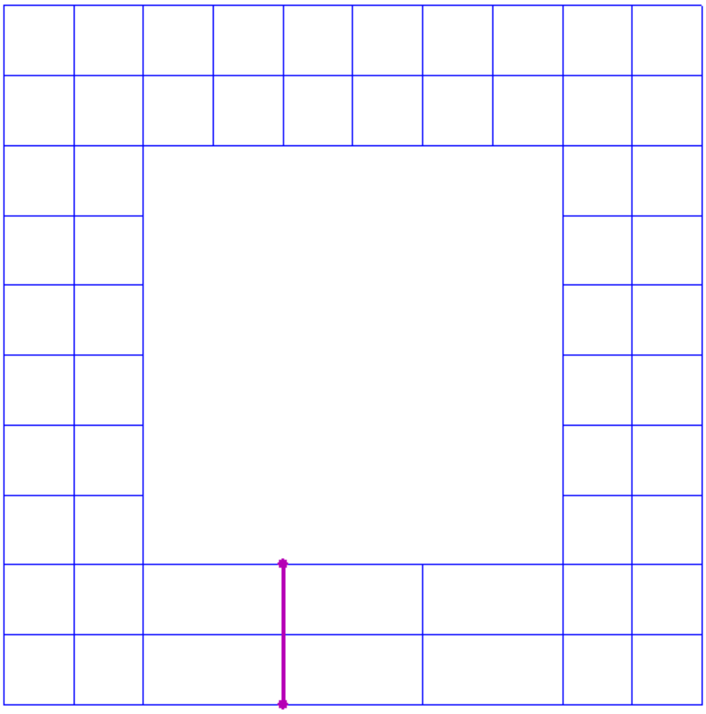


Figure 5.14: *DD Boundary in a single domain*

5.2.10 Polygon(s) between grid boundaries

The aim of this option is to define a polygon in which a irregular grid can be generated, this irregular grid can than be concatenate to the existing irregular grids.

In the *Edit* → *Polygon(s) between grid boundaries* mode a polygon can be defined which take care of the grid nodes at the existing and selected (computational) boundaries of the meshes. The polygon will consist of vertices which are attached to the boundary grid nodes of the selected grids. After wards this polygon can be handled with the normal polygon edit options to adjust the shape of the polygon. Be sure that you do not move the vertices which are locate at the grid boundaries.

The procedure is as follows: Select a point on the boundary from the first grid, select a second point aon the boundary of the same grid or another grid. If two points are selected on the boundary of the same grid then all boundary points in between these two points are selected, using a shortest node-path algorithm. To en the selection press the right mouse button or select the very first point again. Now this polygon editing behaviour will fall back to the normal polygon editing mode. If you are satisfied then generated the irregular grid by selecting *Operations* → *Create grid from polygon* from the menu bar. To merge the grids, select all grids which you want to merge and than select *Operations* → *Merge grids*. If you accidentally moved one vertex of the polygon which should lie on the grid boundary (and it was not merged) then you can merge these nodes by using the option *Edit* → *Grids* → *Merge node*

5.3 Operations menu

On the *Operations* menu, see Figure 5.16, you may choose to generate a grid from a set of splines and to perform various operations on the grid (create, de/refine, orthogonalise, compile dd, etc.). Refinement and orthogonalisation parameters must be changed on the *Settings* menu. Operations at individual grid points can be selected on the *Edit* menu.

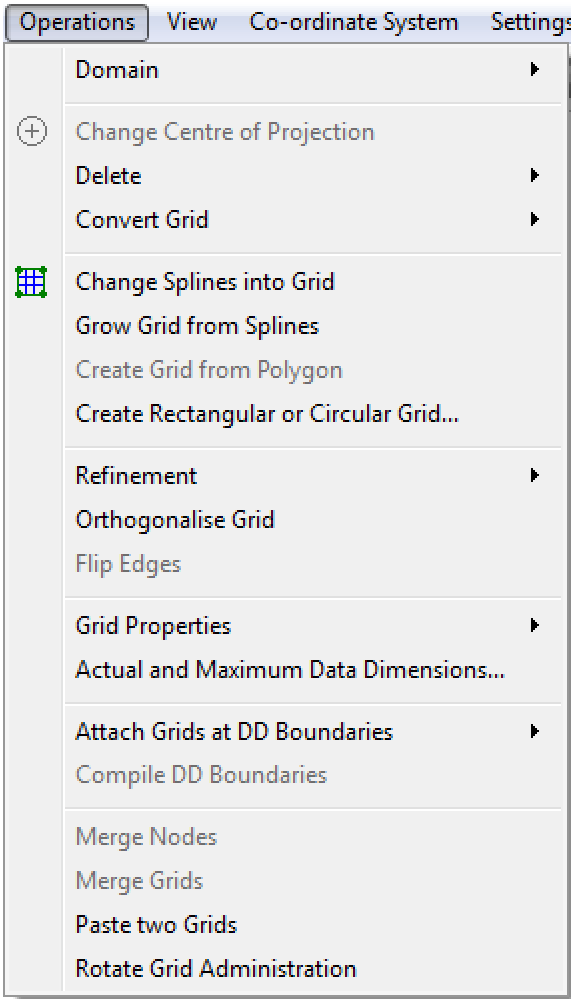


Figure 5.15: *Options on the* Operations *menu*

5.3.1 Change centre of projection

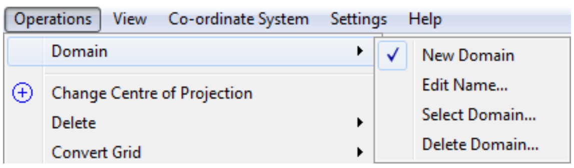


Figure 5.16: *Options on the* Operations *menu*

New domain

When selecting *Operations* → *Domain* → *New domain. . . .* A new generated (ir)regular grid will be added to the pool of grids. For regular grids this is needed to allow Domain Decomposition simulations.

Edit name

When selecting *Operations* → *Domain* → *Edit Name. . .* you are able to change the name of the domain. This is mostly needed to change the default name of the domain.

Select domain

When selecting *Operations* → *Domain* → *Select Domain. . .* you are to slect a domain from the list of domains. The menu option *Edit* → *Multi Allowed* select is should be ticked off.

Delete domain

When selecting *Operations* → *Domain* → *Delete Domain. . .* you are to delete a domain from the list of domains.

5.3.2 Change centre of projection

For spherical co-ordinates RGFGRID can use two different projections, plane projection and stereographic projection. For stereographic projection a special function is implemented to centre the computer screen to the centre of projection and the sphere. This function can be invoked by clicking the menu item *Operations* → *Change Centre of Projection* see Figure 5.16. When using this command the centre of the projection is set to the centre of the screen. This action requires recalculation of the projection and a new screen refresh. The centre of the projection does not change when using zoom in, zoom out or pan, so there is no performance drawback and a smooth screen-refresh is obtained.

5.3.3 New Domain

On the *Operations* → *New Domain* menu, create a new domain. This means that every grid create action will add a new domain to your model. If you want to adjust your grid by the options *Operations* → *Change Splines into grid* or *Operations* → *Create Rectangular or Circular Grid. . .*, deactivate this menu item first.

5.3.4 Delete

On the *Operations* → *Delete* menu, see Figure 5.18, you may choose to delete land boundary, splines, grid, polygons, samples.

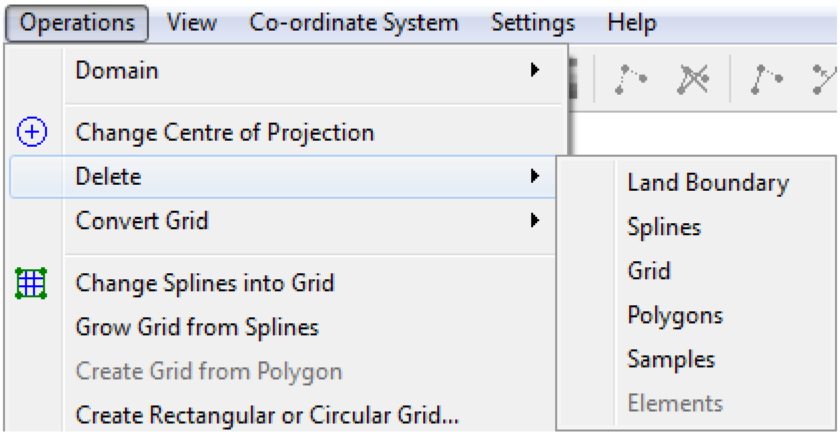


Figure 5.17: *Options on the* Operations → Delete *menu*

Land boundary

If a polygon is present the land boundary points inside the polygon will be deleted. If more polygons are defined only the first polygon will be used. If no polygon is defined you are asked if you want to delete all land boundary points.

Splines

Delete all the splines in the spline grid.

Grid

If a polygon is present the grid points inside the polygon will be deleted. If more polygons are defined only the first polygon will be used. If no polygon is defined you are asked if you want to delete all grid points.

Polygons

Delete the polygon(s).

Samples

Delete all samples within a polygon. If more polygons are defined only the first polygon will be used. If no polygon is defined, you are asked if all samples must be deleted.

5.3.5 Convert grid

Upon selecting *Operations* → *Covert Grid* you are able to convert a regular to irregular grid or the other way around.

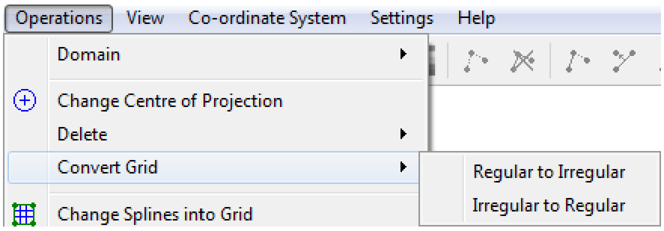


Figure 5.18: *Options on the* Operations → Convert Grid *menu*

Regular to Irregular

Convert the selected regular grid to an irregular grid.

Irregular to regular

Convert the selected irregular grid to a regular grid. Some times you need to select *Operations* → *Rotate Grid Administration* several times to get the orientation of the grid indices in the right orde.

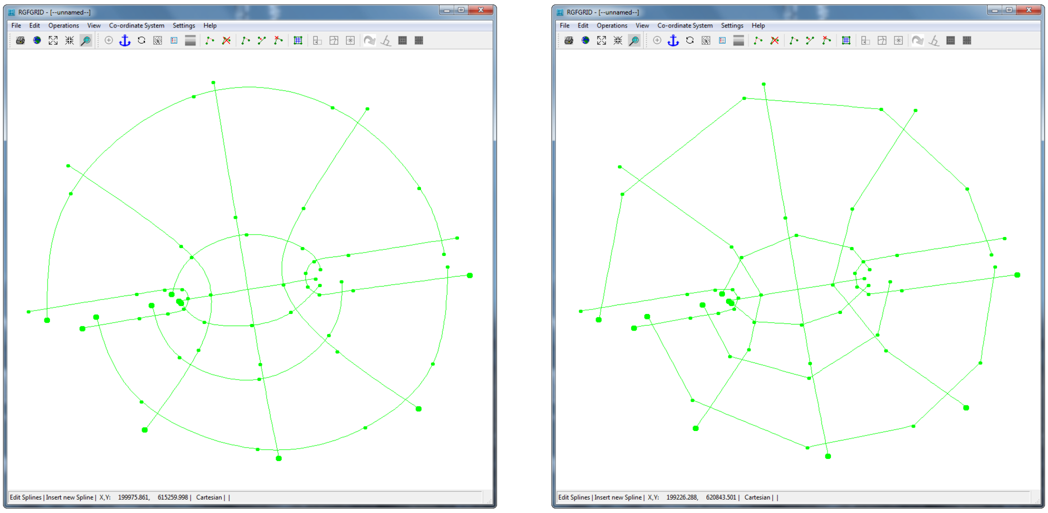
5.3.6 Change splines into grid

This operation can also be activated from the toolbar by clicking .

The splines are ordered and directly refined into a regular grid. The refinement factors can be specified by selecting *Settings* → *General* and specifying the *M-Refinement Factor* and *N-Refinement Factor*, see Figure 5.41. Spline intersection points can only be identified if the straight lines between the control points of two splines intersect. To check this visually, you can display the splines as straight lines (see Figure 5.19, this can be set in the *Settings* → *General* form, parameter *Line or Spline Representation*. The correct ordering is only possible if a consistent result-grid is feasible.

At present, the spline-grid must satisfy the following restrictions:

- ◇ The set of splines need to be topological equivalent with a rectangle.
- ◇ Splines may not intersect twice or intersect themselves
- ◇ Splines with the same orientation may not intersect



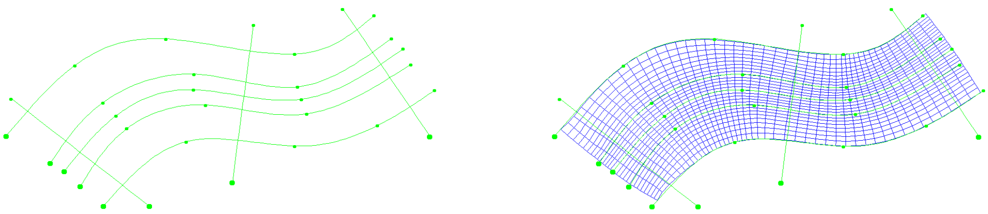
(a) Curved representation *(b) Linear representation*

Figure 5.19: *Different representation of splines*

The smoothness of the result-grid can be influenced by specifying the parameter *Equidistant or Smooth Interpolation* in *Settings* → *General*, see Figure 5.41.

5.3.7 Grow grid from splines

When selecting *operations* → *Grow Grid from Spline* a regular grid will be generated from a centre spline. This option is especially suitable to generate a grid for river simulations. The user is required to provide a spline, from which the grid is grown perpendicularly. Note that the grid can be grown from multiple centre splines simultaneously. Per centre spline, the extent of the grid and the heights of the grid layers can be controlled by supplementary splines and setting parameters (see menu option *Settings* → *Grow Grid from Splines...*). See Figure 5.45.

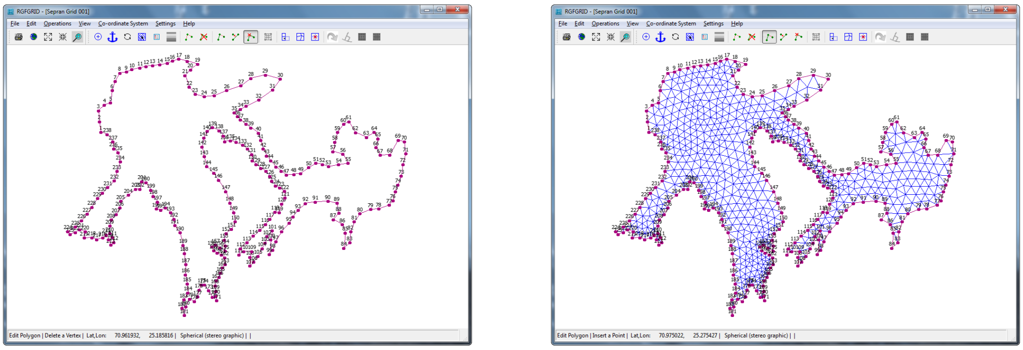


(a) Splines *(b) Generated grid from splines*

Figure 5.20: *Create grid from splines with option* Grow Grid from Spline

5.3.8 Create grid from polygon

When selecting *operations* → *Create Grid from Polygon* a irregular grid will be generated from the selected polygon (see Figure 5.21).

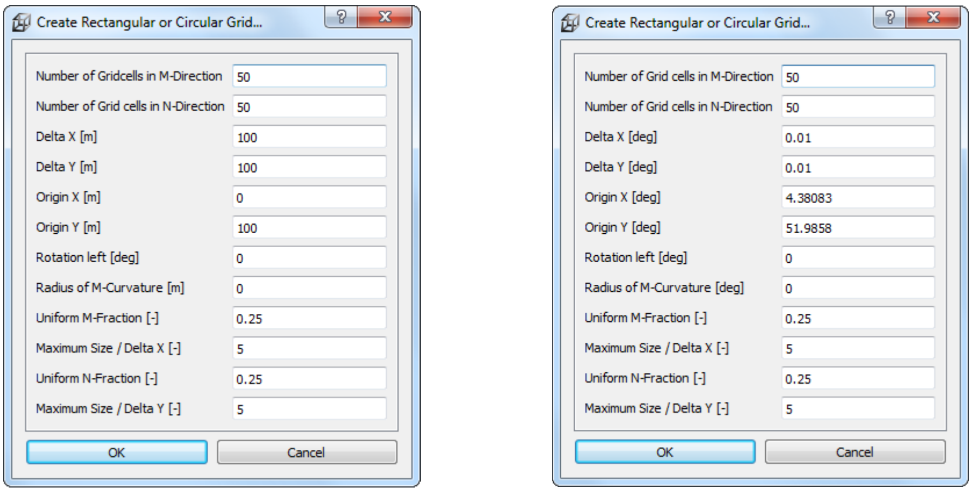


(a) *Selected polygon* **(b)** *Generated grid from polygon*

Figure 5.21: *Create grid from selected polygon*

5.3.9 Create rectangular or circular grid

Specify the grid spacing, grid origin and the number of grid cells in both directions to quickly create a rectangular grid. Grid sizes may be increased in size towards the boundaries by specifying the ration of the maximum grid-size at the boundaries relative to the size of the uniform fraction. The uniform fraction is the number of grid cells with uniform spacing vs. the total number of grid cells in a direction. A circular grid is created if the radius of curvature is non-zero. In that case, the grid origin is interpreted as its centre point. The parameters involved are, see Figure 5.22.



(a) *Cartesian grid* **(b)** *Spherical grid*

***Figure 5.22: Parameters for Rectangular or Circular Grid** form.*

The default settings are:

- ◇ *Number of Grid Cells in M-Direction* default: 50
- ◇ *Number of Grid Cells in N-Direction* default: 50
- ◇ *Delta X [m] or [deg]* default: 100.0 or 0.01

Grid cell size M-direction [m] or [deg]

◇ *Delta Y [m] or [deg]* default: 100.0 or 0.01

Grid cell size N-direction [m] or [deg]

◇ *Origin X [m] or [deg]* default: 0.0 or 4.3803

◇ *Origin Y [m] or [deg]* default: 100.0 or 51.9858

◇ *Rotation left [deg]* default: 0.0

◇ *Radius of M-Curvature [m]* default: 0.0

◇ *Uniform M-Fraction [-]* default: 0.25

Fraction of grid cells which contains the default grid size (ex. 0.25*50 = 13+1 grid cells with width size of 100 [m]))

◇ *Maximum Size / Delta X [-]* default: 5.0

◇ *Uniform N-Fraction [-]* default: 0.25

Fraction of grid cells which contains the default grid size (ex. 0.25*50 = 13+1 grid cells with width size of 100 [m]))

◇ *Maximum Size / Delta Y [-]* default: 5.0

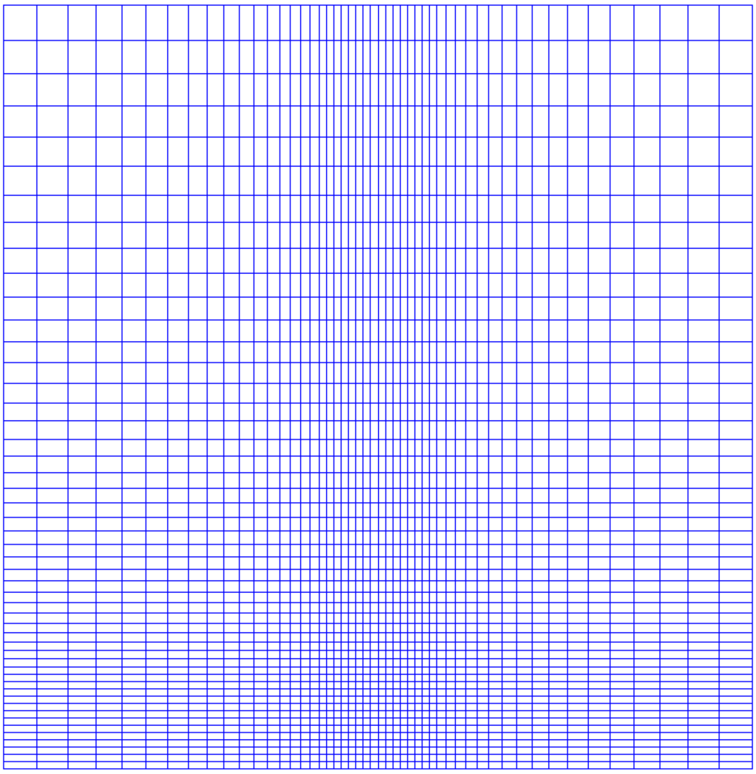


Figure 5.23: *Rectangular grid, created with* Maximum Size / Delta X = *"5" and* Maximum Size / Delta Y = *"5"*

5.3.10 Refinement

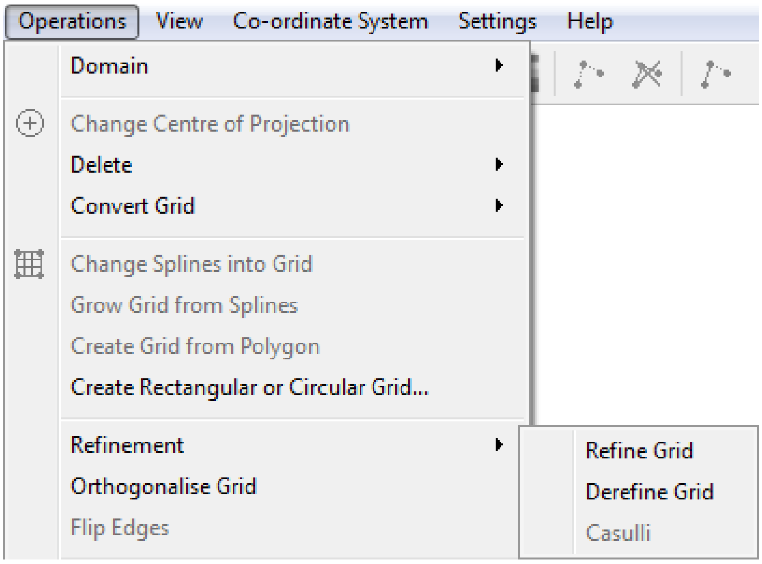


Figure 5.24: *Options on the* Operations → Refinement *menu*

Refine grid

This option operates on the whole grid and in both directions.

In *Settings* → *General* you first specify the number of times you want to refine the grid. The parameters for the M and N direction are *M-Refinement Factor* and *N-Refinement Factor*, respectively. You can identify the M and N direction by selecting *View* → *Grid* → *Lines and M, N Indices*.

Restriction:

◇ The number of refinement must be an integer number.

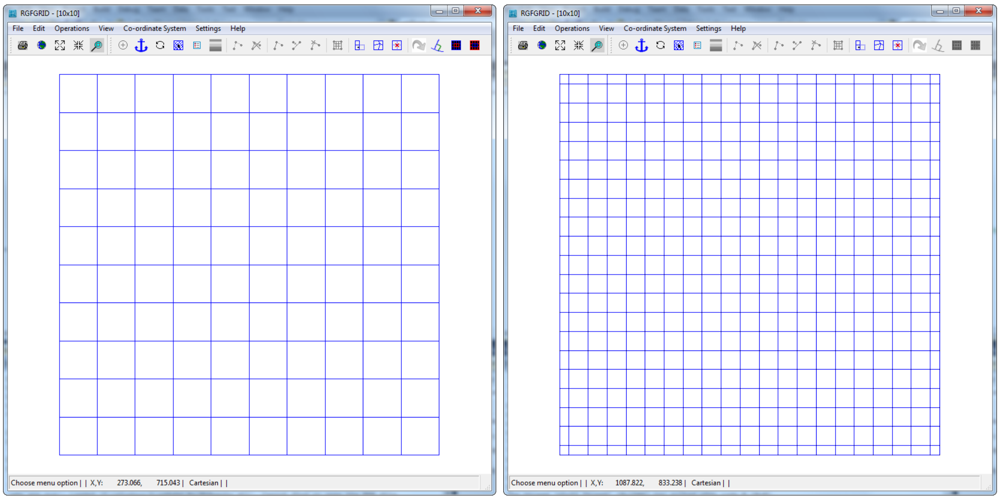
Derefine grid

This option operates on the whole grid and in both directions.

The opposite of *Refine Grid*. One limitation of the refinement procedure is that it can only refine by an integer number. The combination of refine and de-refine allows you to reach a rational number as refinement factor. e.g. You wish a refinement factor of 1.5; first refine by a factor of 3, next de-refine by a factor of 2. Next, go to *Edit* → *Line* → *Smooth* to decrease the jump in grid-sizes.

Casulli

Only applicable for whole irregular squared cell meshes.



(a) *Original square grid ($\Delta x = 100\ m$)* ***(b)*** *After Casulli refinement (most grid cells have $\Delta x = 50\ m$)*

Figure 5.25: *Example of Casulli refinement of an irregular grid*

5.3.11 Orthogonalise grid

This option operates on the whole grid or on a part of the grid. To operate on a part of the grid:

- ◇ For regular grids use *Edit* → *Block* → *Orthogonalise*. The grid will be orthogonalised in accordance with the local grid cell resolution, i.e. the overall shape will be conserved, but individual points may be shifted to get better orthogonality. You can specify parameters that control the orthogonalisation in *Settings* → *Orthogonalisation (regular)*, see Figure 5.43.
- ◇ For irregular grids you have to specify one or more polygons. The orthogonalisation will be performed only for the **selected** polygons. You can specify parameters that control the orthogonalisation in *Settings* → *Orthogonalisation (irregular)*, see Figure 5.44.

5.3.12 Flip edges

Minimalize the number of edges connected to a node. The optimal number of edges to a node is six.

Nodes that are connected to more than, say, six other nodes, are typically enclosed by cells of highly non-uniform shape and wildly varying sizes. This motivates to improve the mesh connectivity by selecting *Operation* → *Flip Edges*.

5.3.13 Grid properties

Specify the desired grid property to be shown, see Figure 5.26.

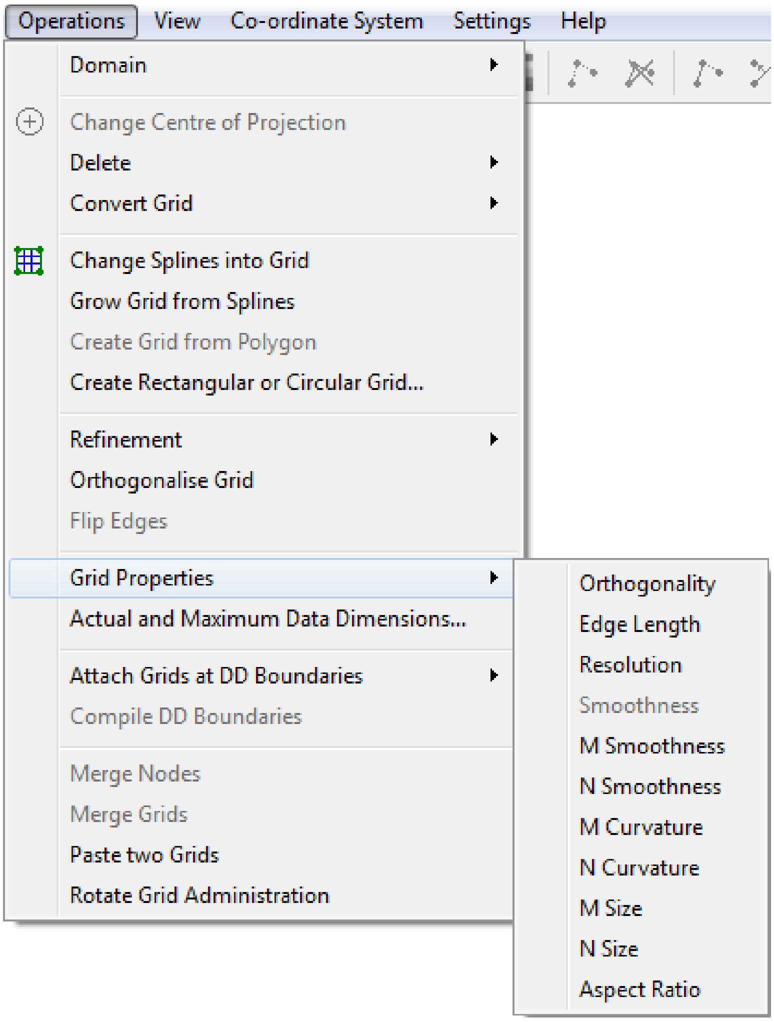


Figure 5.26: Operations → Grid Properties *options*

Orthogonality

Regular grids:
Cell centred cosine value. Keep this value low in the inner model area, e.g. 0.02-0.04. The error in the direction of the pressure gradient in Delft3D-FLOW is proportional to the deviation of the cosine value from zero. Near closed boundaries, larger values can be tolerated than in the inner model area.

Irregular grids:
Cosine value of the angle between an edge and the line between the circumcentres of the enclosing elements of that edge. Keep this value low, e.g. < 0.001.

Edge length

Show the edge length.

Resolution

Square root of grid cell area ([m]).

Smoothness

Irregular grids only. Plotting irregular grid smoothness not yet implemented.

M-Smoothness

Regular grids only. Ratio between adjacent grid cell lengths in M-direction, value ≥ 1. Preferably less than 1.2 in the area of interest.

N-Smoothness

Regular grids only. Ratio between adjacent grid cell lengths in N-direction, value ≥ 1. Preferably less than 1.2 in the area of interest.

M-Curvature

Regular grids only. Reciprocal value of radius of curvature, times 1000 ([1/m]).

N-Curvature

Regular grids only. Reciprocal value of radius of curvature, times 1000 ([1/m]).

M-Size

Regular grids only. Grid cell size in M-direction ([m]).

N-Size

Regular grids only. Grid cell size in N-direction ([m]).

Aspect-Ratio

Regular grids only. Ratio of M-size/N-size, value ≥ 1. Must be in the range [1,2] unless the flow is predominantly along one of the grid lines.

Remark:

◇ For a spherical grid the resolution, curvature and grid size are also given in the metric system.

5.3.14 Actual and maximum data dimensions

The actual and maximum dimensions of various data objects are presented in 'history', see Figure 5.27

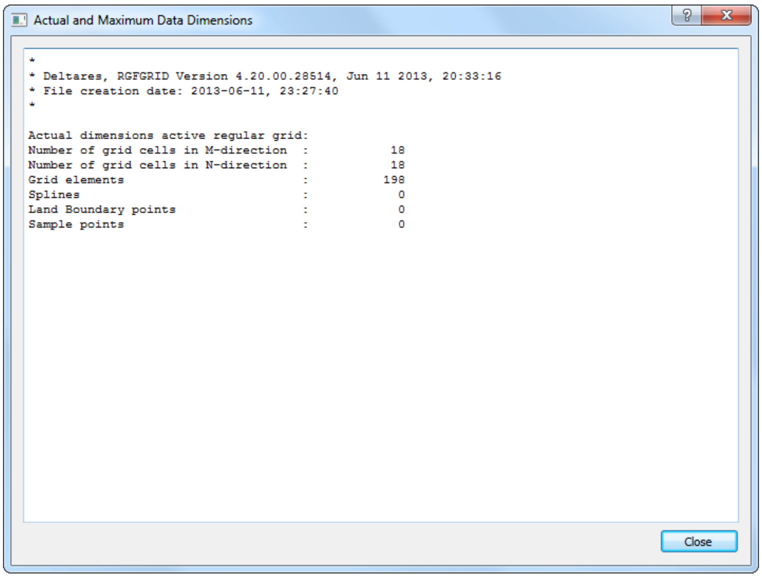


Figure 5.27: Operations *menu,* Actual and Maximum data dimensions

5.3.15 Attach grids at DD boundaries

There is a small difference between attaching regular and irregular grids. For regular grids you are able to move the DD-boundary points of one regular grid to the DD-boundary of the other regular grid, so the boundary is exactly on the same place. After that operation you have to perform the operation *Menu* →*Operations* →*Compile DD Boundaries*. For irregular grids the DD-boundaries should have the same location before the merge operation can be applied by *Operations* →*Attach Grids at DD Boundaries* →*Irregular grids*.

For the menu layout see Figure 5.30.

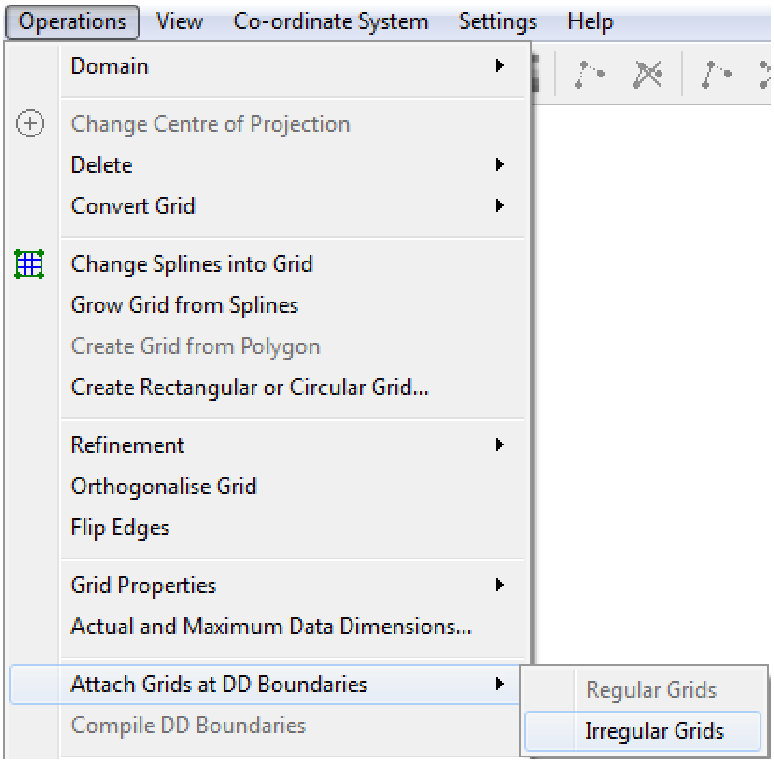
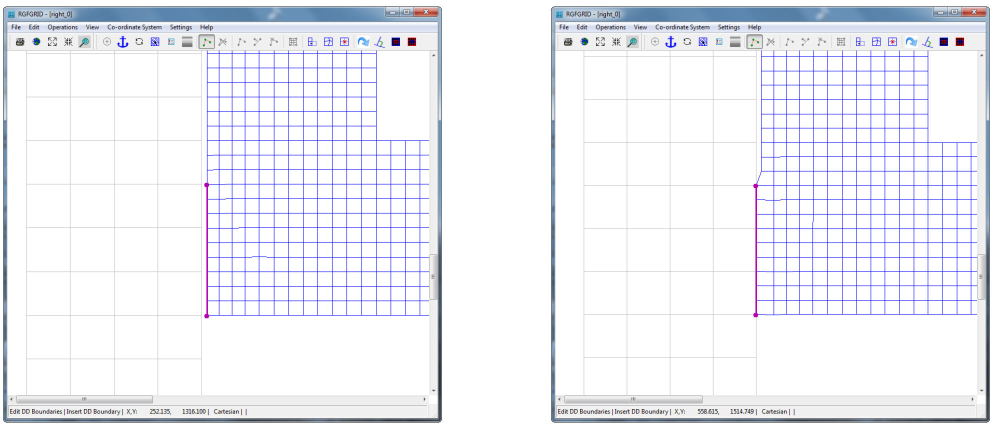


Figure 5.28: Operations →Attach Grids at DD Boundaries

5.3.15.1 Attach regular grids at DD boundaries

This option is only relevant if you want to use the multi-domain option of Delft3D-FLOW. First you have to indicate the domain decomposition boundaries in the mode *Edit → DD Boundaries*.

One of the restrictions of domain decomposition is that the domain boundary between two domains have to coincide, so there is no overlap or gap between the domains on the DD-boundary. This option attach the grid at the DD-boundary to each other, for all DD-boundaries of the current active grid. This is achieved by moving the grid points on the DD-bounday of the active grid, to the corresponding inactive grid, see Figure 5.29.



(a) *Before attach operation* ***(b)*** *After attach operation*

Figure 5.29: Operations → Attach Grids at DD Boundaries→Regular grids

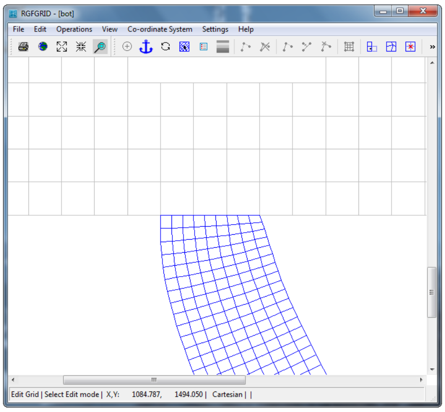
5.3.15.2 Attach irregular grids at DD boundaries

This option is most relevant if you have a multi-domain simulation model suitable for Delft3D-FLOW and you want to use D-Flow FM.

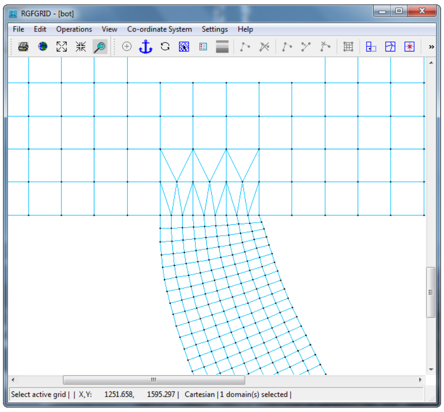
Load the regular grids which need to be merged to one irregular grid at domain decomposition boundaries (ex. Figure 5.30a).

Convert the regular grids to irregular by choosing menu option *Operations* →*Convert Grid* →*Regular to Irregular*.

To perform the merge of the irregular grids choose menu option *Operations* →*Attach Grids at DD Boundaries* →*Irregular grids* (ex. Figure 5.30b).



***(a)** Before attach operation*



***(b)** After attach operation*

Figure 5.30: Operations → Attach Grids at DD Boundaries

5.3.16 Compile DD boundaries

This option is only relevant if you want to use the multi-domain option of Delft3D-FLOW. First you have to indicate the domain decomposition (DD-)boundaries in the mode *Edit* → *DD Boundaries*.

When you have defined the DD Boundaries, the grids needed in your multi-domain application, are coupled here. Upon clicking *Operations* → *Compile DD Boundaries* a window opens, in which you can select where the DD-boundary will be saved, see Figure 5.31.

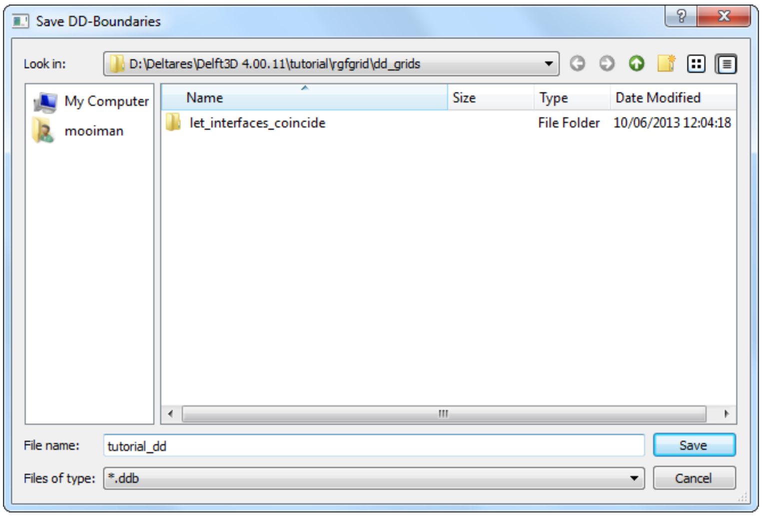


***Figure 5.31: Save DD-Boundaries** window*

The DD administration is written to a file with default mask <∗.ddb>, see Appendix A.9 for its format.

5.3.17 Merge nodes

With this option you are able to merge nodes. First you click the node which you want to merge and than you click the node to merge to. The first clicked node will move to the second node location and than the two nodes will be merged to one node.

5.3.18 Merge grids

To merge several grids you have to indicated which grids need to be merged, to select the grids use the multi selection tool (*Edit* → *Allow Multi Select*). After selecting some grids this option will merge the indicated irregular grids. Nodes from different grids with the same location will be merged.

5.3.19 Paste two grids

The second (inactive) grid is pasted to the active grid. The M, N-orientations of both grids do not have to match. The grid points on the junction line(s) should be relatively close to each other, i.e. less than one quarter of a grid cell apart. On the junction line, the grid points are a weighted average of the active grid and the inactive grid. The weighting factor can be changed in the menu option *Settings* → *General*, see Figre 5.41.

A value of 0.0 will freeze the active grid; the inactive grid will move to the active grid.

5.3.20 Rotate grid administration

The M, N orientation of the grid is rotated over 90 degrees (counter clock-wise). Maybe you need to adjust the grid administration because the grid administration over a DD-boundary should have the same grid orientation.

5.4 View menu

On the *View* menu (see Figure 5.32) options are presented how to display a spherical grid, whether or not to show the boundary and legend, to inspect the grid in 3 dimensions, how to display the objects, and to view grid properties. Display characteristics (for the legend, colours and sizes) may be changed in the *Settings* menu.

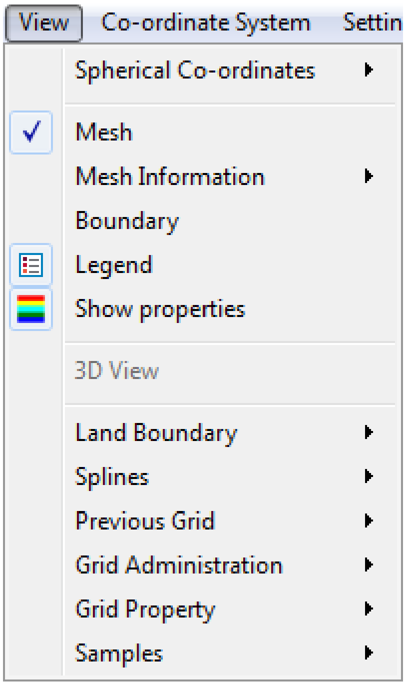


Figure 5.32: *Options on the* View *menu*

5.4.1 Spherical co-ordinates

Default: A spherical grid is shown in stereographic projected co-ordinates.

In the spherical co-ordinate system you can view the objects stereographic projected, see Figure 5.34.

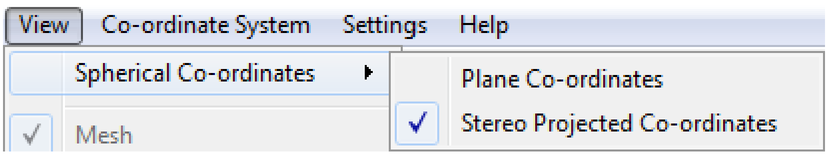


Figure 5.33: *Options on the* View → Spherical Co-ordinates *menu*

Plane co-ordinates

The co-ordinates are displayed just as they are and there is no well known projection used.

Stereographic projected co-ordinates

To display the co-ordinates a stereographic projection is used. The centre point of the stereographic projection is default the centre of the window. When zooming or scrolling the centre point is not recalculated, for recalculation the centre point you have to press menu option *Operations* $\rightarrow$ *Change Centre of Projection*.

5.4.2 Mesh

Show or hide the meshs

5.4.3 Mesh Information

Upon selecting *View* $\rightarrow$ *Mesh Information* you show or hide several numebrings of the mesh

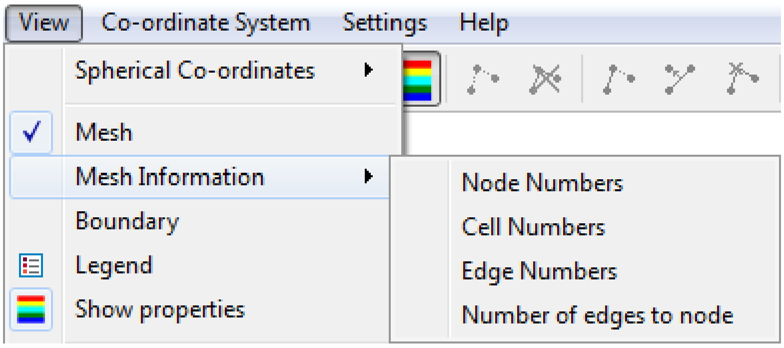


Figure 5.34: *Options on the* View $\rightarrow$ Mesh Information *menu*

Node numbers

Show or hide the node number.for regular grids it has the format (m, n) and for irregular grids it is a single integer.

Cell numbers

Show or hide the cell number.for regular grids it has the format (m, n) and for irregular grids it is a single integer.

Edge numbers

Show or hide the edge number.

Number of edges to node

Show or hide the number of edges that are connected to a node

5.4.4 Boundary

Default: Hide the boundary.

Show or hide the boundaries, open, domain decomposition as well as computational boundaries

5.4.5 Legend

Default: Show the (colour) legend.

Show or hide the colour legend on the screen.

5.4.6 Show properties

Show or hide the grid properties.

5.4.7 3D View

In the **Delft3D-3DView** window a fully 3-dimensional view of the data is shown.

`c`: Switch rendering mode
`h`: Toggle help
`i`: Inverse depth
`r`: Reste view
`s`: Toggle samples
`x`: Increase depth
`z`: Decrease depth
left mouse: Rotate in xy-plane
`CTRL`+left mouse: move origin xy-plane
middle-mouse: zoom
right mouse: rotate z-axis

5.4.8 Land boundary

Default: Show the land boundary as lines.

The following display options are available for displaying the land boundary:

◇ No Land Boundary
◇ Lines
◇ Filled

5.4.9 Splines

Default: Show the splines as lines with dots.

The following display options are available:

- ◇ No Spline
- ◇ Lines with Dots
- ◇ Lines with Dots and M, N Indices

5.4.10 Grid

Default: Show the grid as lines.

The following options are available for displaying a grid:

- ◇ No Grid
- ◇ Lines
- ◇ Lines and M, N Indices

The latter option is provided so that you can check and control the grid administrative lower left corner, i.e. the location of the (1,1) point. It gives the least confusion if this point is, the more the less, located at the lower left corner of the screen.

5.4.11 Previous grid

Default: Hide the previous grid as lines.

Sometimes, when editing the regular grid, it may be convenient to display the grid both in its present and previous state on the screen at the same time. The usual display options are available:

- ◇ No Previous Grid
- ◇ Lines
- ◇ Lines and M, N Indices

5.4.12 Grid administration

Default: Hide the grid administration.

This option allows you to visualise the grid topology in the 'computational' space (as opposed to the physical space). It helps you decide which grid extensions are allowable so that overlap is avoided. The each domain grid should always have a mono-block structure.

Select the required option:

- ◇ No Grid Administration
- ◇ Lines
- ◇ Lines and M, N Indices

5.4.13 Grid properties

Default: Show the grid property as continuous shading.

Specify how to display the desired grid property:

- ◇ No Grid Property
- ◇ Continuous Shading
- ◇ Coloured Dots
- ◇ Numbers
- ◇ Coloured edges

5.4.14 Samples

Default: Show samples as coloured dots.

Specify how to display the samples:

- ◇ No Samples
- ◇ Coloured Dots
- ◇ Coloured Numbers
- ◇ Mono Coloured Numbers

5.5 Co-ordinate System menu

On the Co-ordinate System menu you can set the desired co-ordinate system (see Figure 5.35) to Cartesian or spherical co-ordinates, see Figure 5.36. Furthermore, you can translate or rotate the objects in a Cartesian co-ordinate system and you can transform Cartesian co-ordinates to spherical co-ordinates and vice versa.

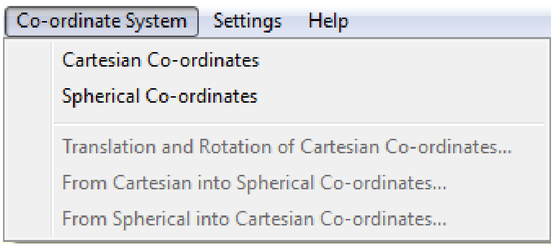
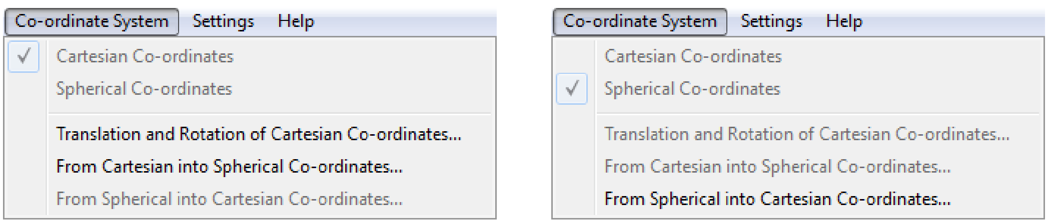


Figure 5.35: *Menu option* Co-ordinate System



(a) Cartesian co-ordinates selected *(b) Spherical co-ordinates selected*

Figure 5.36: *Menu option* Co-ordinate System.

5.5.1 Cartesian co-ordinates

In this case the co-ordinates are easting and northing in metres.

5.5.2 Spherical co-ordinates

In this case the co-ordinates are longitude and latitude, in decimal degrees.

5.5.3 Translation and rotation of Cartesian co-ordinates

This option may be applied if you are changing to a new Cartesian co-ordinate system which has a different position of the origin or another orientation. The parameters involved are, see Figure 5.37:

- ◇ Offset X direction [m] — default 0.0
- ◇ Offset Y direction [m] — default 0.0
- ◇ Rotation left [degrees] — default 0.0
- ◇ X Scale factor — default 1.0
- ◇ Y Scale factor — default 1.0

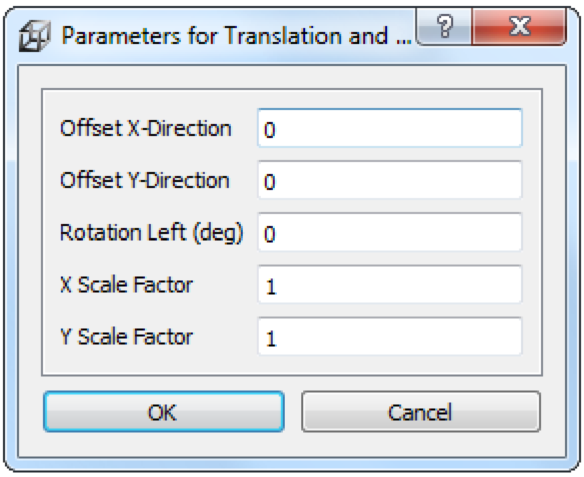


Figure 5.37: ***Parameters for translation and rotation*** *form for transformation to Cartesian co-ordinates*

Remark:

◇ A translation and rotation operates only on samples, polygon the active grid.

5.5.4 From Cartesian into spherical co-ordinates

A form will appear with the parameters for the co-ordinate conversion, see Figure 5.38. The first parameter indicates the current co-ordinate system:

1. UTM,
2. Amersfoort / RD new (EPSG:28992) or
3. RD (Parijs) (EPSG:2489).

Putting the cursor on top of this field will show the help text. If the current system is UTM you have to specify the zone number

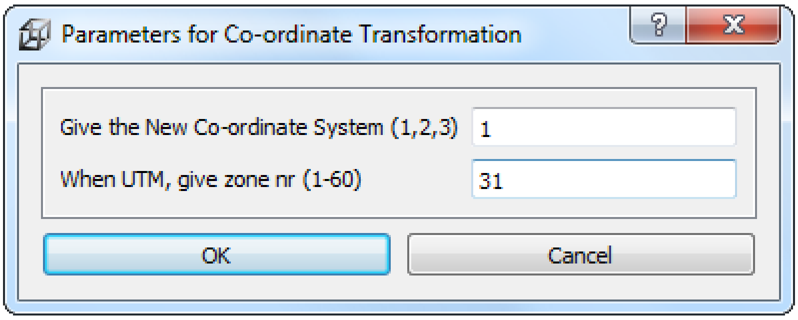


***Figure 5.38: Parameters for Co-ordinate transformation** form for transformation to spherical co-ordinates*

5.5.5 From spherical into Cartesion co-ordinates

Figure 5.39 shows the parameters involved for the conversion from spherical to Cartesian co-ordinates.

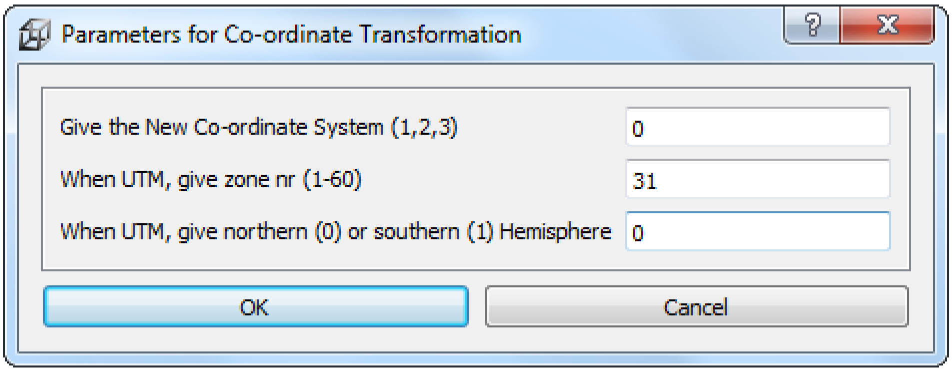


***Figure 5.39: Parameters for Co-ordinate transformation** form for transformation to Cartesian co-ordinates*

5.6 Settings menu

The following options can be accessed through the *Settings* menu, see Figure 5.40

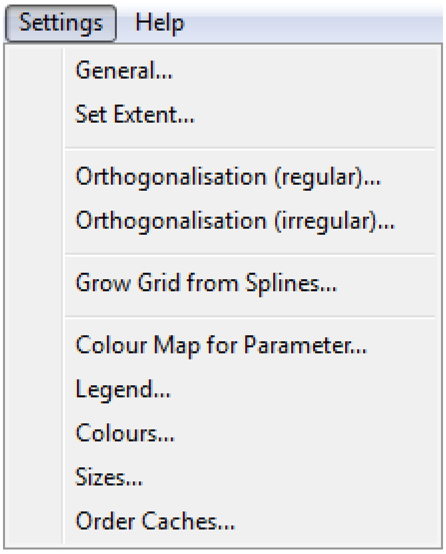


Figure 5.40: *Options on* Settings *menu*

5.6.1 General

The following parameters influence the behaviour of the operations above. They are set via the following parameter list, see Figure 5.41

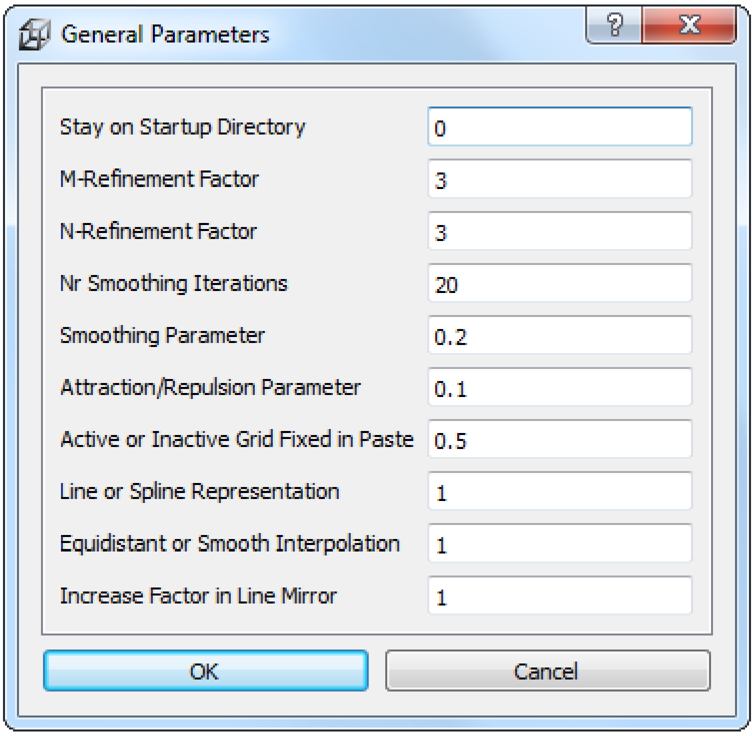


Figure 5.41: *Options on* ***Settings*** *window*

◇ Stay on Startup Directory — default: 0 (Off)
  When navigating through the directories in the file menu, you can specify whether to keep the latest visited directory (0), or always go back to the start-up directory (1).
◇ M-refinement factor — default: 3
  A value of 2 gives twice as many grid cells in m-direction. An odd value assure that the

cell centre and the mid of a cell edge are available in the coarse as well as in the refined grid.

- ◇ N-refinement factor — default: 3
  A value of 2 gives twice as many grid cells in n-direction. An odd value assure that the cell centre and the mid of a cell edge are available in the coarse as well as in the refined grid.
- ◇ Nr Smoothing Iterations — default: 20
  The smoothing in edit mode is controlled by this parameter.
- ◇ Smoothing Parameter — default: 0.2
  The smoothing in edit mode is also controlled by this parameter. A value of 0.0 results in no smoothing, a value of 1.0 in maximum smoothing.
- ◇ Attraction/Repulsion Parameter — default: 0.1
  Attraction/repulsion in edit mode is controlled by this parameter. The value is the fractional change in size of the first grid cell adjacent to the indicated line. Increase this value for more attraction or repulsion.
- ◇ Active or Inactive Grid Fixed in Paste — default: 0.5
  When pasting an active grid to an inactive grid, the grid points on the grid junction line are a weighted average between both grids. If you want to keep these points in the position of the active grid, set this parameter to 0.0. To keep the position of the inactive grid, choose 1.0; a value in between averages.
- ◇ Line or Spline Representation (0 or 1) — default: 1 (Spline)
  Splines, or grid boundaries in the orthogonalisation process, can also be represented as straight lines if this parameter is put to a zero value.
- ◇ Equidistant or Smooth Interpolation (0 or 1) — default: 1 (Smooth)
  When interpolating the splines into a grid, equidistant interpolation can be specified using a value of 0.
- ◇ Increase Factor in Line Mirror (1.0 - 10.0) — default: 1.0
  When adding grid cells using the *Edit* $\rightarrow$ *Line* $\rightarrow$ *Mirror* option, this parameter defines the size of the new grid cells.

5.6.2 Set extent

Set the horizontal extent of the canvas

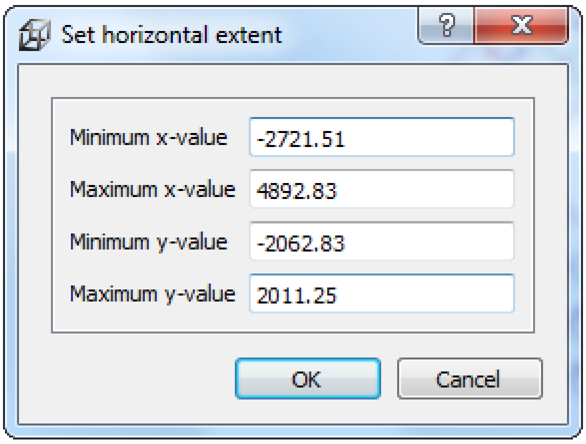


***Figure 5.42: Set horizontal extent** window*

- ◇ Minimum visible x-value
- ◇ Maximum visible x-value

- ◇ Minimum visible y-value
- ◇ Maximum visible y-value

The used extent is dependent on the current window size, keeping the x/y ratio to 1.

5.6.3 Orthogonalisation regular

With the **Orthogonalisation Parameters** form, see Figure 5.43, the orthogonalisation process can be controlled:

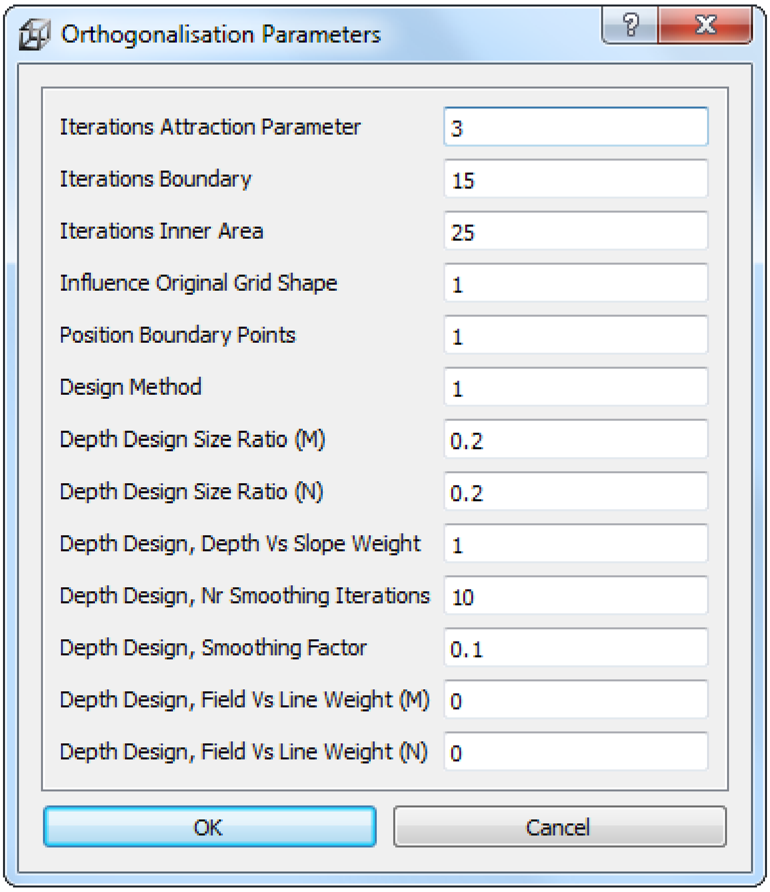


Figure 5.43: *Options on **Orthogonalisation Parameters** window*

- ◇ Iterations Attraction Parameter default: 3
  The shape of the resulting grid is based on the so-called attraction parameter, i.e. the local aspect ratio of the original grid. One complete orthogonalisation cycle consists of three loops. The outer loop is the attraction parameter loop, in which this parameter field is established. Only few of these loops are usually performed. Next, several boundary loops are performed, in each of which the inner area is solved several times. Increasing the number of attraction parameter iterations improves orthogonality, but it increases deviation from the originally designed shape.
- ◇ Iterations Boundary default: 15
  In one boundary loop, all boundary points are updated once, and all inner area points are updated as many times as specified by the next parameter. We advise values in the following range: [5 - 20].
- ◇ Iterations Inner Area default: 25

The number of inner area iterations in the orthogonalisation is advised in the range: [10 - 50].

◇ Influence Original Grid Shape default: 1.0
This parameter specifies the influence of the specified grid shape in the inner area during the orthogonalisation procedure. The grid shape in the inner area can be specified in three ways, see 'Design Method' below. With a value of 1 the specified shape is maintained as closely as possible. With a value of 0, the shape mostly depends on the shape of the boundaries and the internal corner points. Any value between 0 and 1 can be chosen.

◇ Position Boundary Points default: 1.0
This parameter specifies the freedom of movement of boundary points. These points move along splines spanned by the outer points of the grid. A value of 1 gives full freedom of movement, whereas a value of 0 keeps boundary points completely fixed. Any value between 0 and 1 may be chosen.

◇ Design Method, 1, 2, or 3 default: 1
This parameter specifies in what way the attraction parameter (local aspect ratio) field is created. The three methods are:

1 This method is based upon the aspect ratios of the original design grid (default).
2 This method uses a polygon, the polygon can be applied to control grid spacing.
3 With this method the grid resolution is controlled based upon features in a bathymetry that can be opened by means of the samples. This method is therefore called 'Depth design'.

In these methods, the attraction parameter field is based upon grid spacing functions both in the M- and N-Direction. Their local ratio forms the desired attraction parameter field. Both the M- and N-grid spacing functions can be controlled by a number of parameters, that are explained below. The same parameters also apply to method 2, that can be seen as a special case of method 3, in which the bathymetry is specified by specifying a constant 'depth' inside the polygon, different from the also constant 'depth' outside the polygon.

◇ Depth Design Size Ratio (M) default: 0.2
Both for the M- and the N-direction, the size ratio between the smallest and largest grid size in that direction can be specified. If a value of 1 is specified, a uniform distribution results. Choosing a small value will result in large grid size variation. If a value of 0 is specified, this is seen as a special case and the original grid shape is applied as the desired grid spacing function.

◇ Depth Design Size Ratio (N) default: 0.2
See *Depth Design Size Ratio (M)* above.

◇ Depth Design, Depth vs Slope Weight default: 1
Both the depth and the slope can be applied as grid spacing control functions. To obtain a high resolution in deep areas only, specify a *Small/large Size Ratio* below 1, e.g. 0.1, and specify a *Depth vs. Slope Weight* parameter of 1. To obtain small cells at steep slopes only, specify a value of 0. Any value in between 0 and 1 can be applied. In the future, the slope variation is foreseen as a controlling parameter as well.

◇ Depth Design, Number of Smoothing Iterations default: 10
To obtain smoother transitions between sloping and non-sloping areas, the grid spacing functions can be smoothed. The smoothing parameters apply both to the (M) and (N) direction. Also, the smoothing may be applied to 'spread' the grid spacing information towards grid cells that initially may lie outside the area that needs to get a high resolution. In each attraction parameter iteration, see above, the grid point spacing function is evaluated, and applied in the following orthogonalisation loop, which results in shifting the grid

points to their final position. Once getting closer to their final position, the smoothing may be decreased, so that the bathymetry features become more apparent in the grid. This process may be automated in future.

- ◇ Depth Design, Smoothing Factor — default: 0.1
  Smoothing weight of point itself to neighbours.
- ◇ Depth Design, Field vs. Line Weight (M) — default: 0
  The *Small/Large Size Ratio* parameter can either be specified to apply to the whole grid, or to every grid line. i.e. should the specified *Size Ratio* in the given direction be applied to the whole grid or to every grid line? If a value of 1 is chosen, this ratio will only occur at the maximum value of entire spacing function. If a value of 0 is chosen, this size ration will occur at every grid line.
- ◇ Depth Design, Field vs. Line Weight (N) — default: 0
  See *Depth Design, Field vs. Line Weight (N)* above.

5.6.4 Orthogonalisation irregular

With the **Orthogonalisation Parameters (irregular)** form, see Figure 5.44, the orthogonalisation process of irregular grids can be controlled:

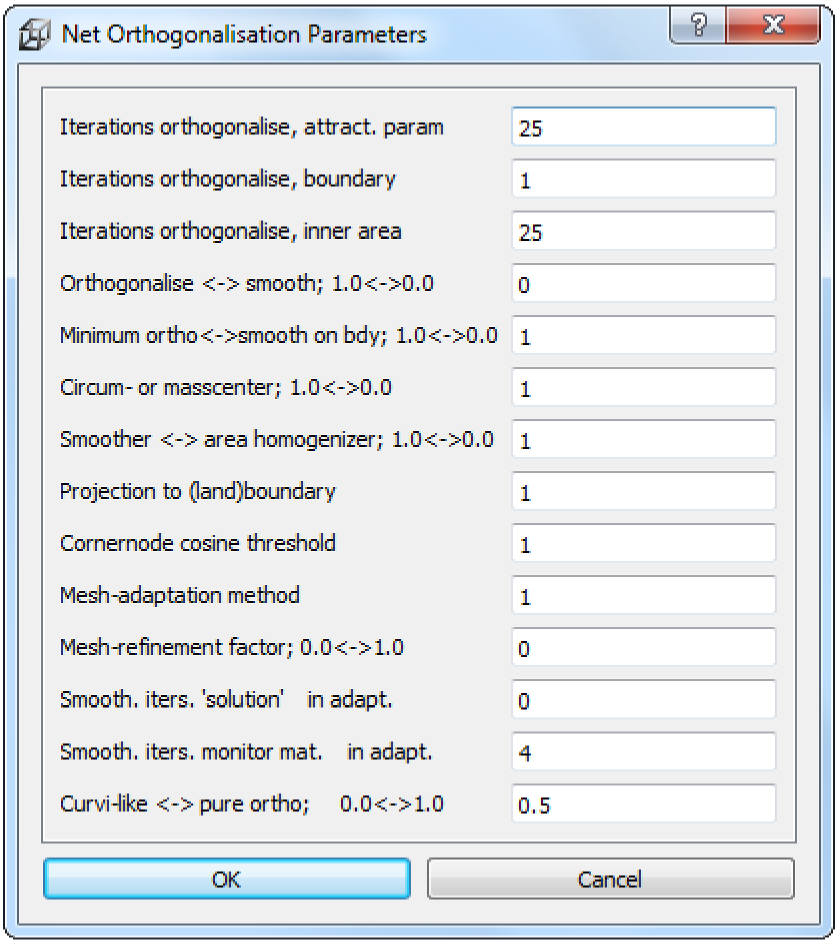


Figure 5.44: *Options on* ***Orthogonalisation Parameters (irregular)*** *window*

- ◇ Iterations orthogonalise, attraction parameters: — Default: 25
  The number of iterations in which the attraction parameters are computed for the grid. The attraction parameter is the fractional change in size of the first grid cell adjacent to the

indicated line.

◇ Iterations orthogonalise, boundary: Default: 1
This parameter can be used to prescribe the number of iterations in which the grid is moved along the boundaries to improve the orthogonality of the grid.

◇ Iterations orthogonalise, inner area: Default: 25
This parameter can be used to prescribe the number of iterations in which the grid is moved along in the interior of the domain to improve the orthogonality of the grid. The total number of iterations is the product of the three iteration values (attract. param, boundary and inner area).

◇ Orthogonalise ↔ smooth; 1.0 ↔ 0.0 Default: 0
The balance between mesh-smoothing (0.0) and mesh-orthogonalization (1.0). One has to keep in mind that mesh smoothing (ortho. param. → 0) will compromise mesh orthogonality. Sole orthogonalization (ortho. parameter=1) on the other hand, can cause highly distorted, non-smooth meshes, especially for meshes consisting of quadrilaterals. It is advised to choose a low orthogonalization parameter and repeat the process while gradually increasing the orthogonalization parameter at every repetition.

stage	1	2	3	4	5	6
ortho. parameter	0.5	0.8	0.9	0.99	0.999	0.9999

Table 5.1: *Multi-stage orthogonalization strategy*

◇ Minimum ortho ↔ smooth on boundary; 1.0 ↔ 0.0 Default: 1
This parameter can be used to prescribe the number of iterations in which the grid is moved along in the interior of the domain to improve the orthogonality of the grid. The total number of iterations is the product of the three iteration values (attract. param, boundary and inner area).

◇ Orthogonalise ↔ smooth; 1.0 ↔ 0.0 Default: 1
Balance between orthogonalisation and Laplacian smoothing. The orthogonality is defined by the angle between the line connecting water level points and the line connecting two grid cell corners. The smoothness is defined as the ratio between the areas of two adjacent grid cells.

◇ Circum- or masscentre; 1.0 ↔ 0.0: Default: 1
Define whether the orthogonality is measured based on the circumcenter of the triangle or on the mass center of a triangle.

◇ Smoother ↔ area homogenizer; 1.0 ↔ 0.0: Default: 1

◇ Projection to (land)boundary: Default: 1
0 means no projection of the grid to the landboundary while orthogonalising the grid. 1 means projection of the grid boundaries to the original grid boundaries as before orthogonalising the grid. 2 means projection of the grid boundaries to the nearest landboundary. 3 means projection of the grid boundaries as well as parts of interior part of the grid to the nearest landboundaries.

◇ Corner node cosine threshold: Default: 1
Determines whether a node is a corner on the basis of the cosine of the boundary edge angle. If a node is a corner, then the node is not moved during orthogonalisation.

◇ Mesh adaption method: Default: 1
Selection of a mesh-adaptation method. 0 means a Winslow type monitor function, 1 an arc-length monitor function and 2 a harmonic map monitor function. See Huang (2001, sect 3.3).

◇ Mesh refinement factor; 0.0 ↔ 1.0: Default: 0

Concentation of the mesh in a refined region (parameter in the mesh-adaptation method). See Huang (2001, sect 3.3).

◇ Smooth. iters. 'solution' in adapt.: Default: 0
Number of smoothing iterations of solution u in mesh-adaptation method. See Huang (2001, sect 3.3).

◇ Smooth. iters. 'monitor mat.: Default: 4
Number of smoothing iterations of monitor matrix G in mesh-adaptation method. See Huang (2001, sect 3.3).

◇ Curvi-like ↔ pure ortho; 0.0 ↔ 0.5: Default: 0
Chooses between pure orthogonalisation versus curvi-grid-like orthogonalisation in quads.

5.6.5 Grow grid from spline

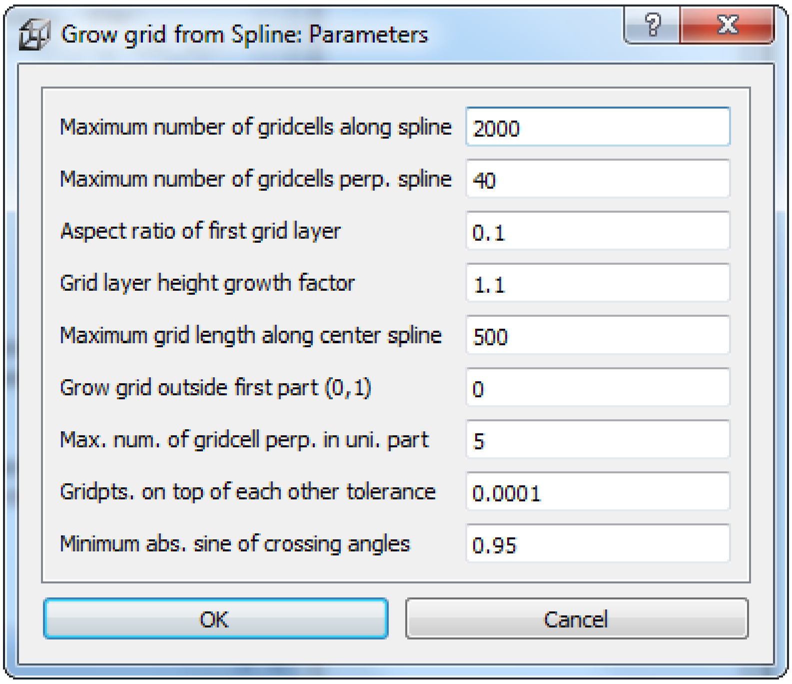


Figure 5.45: *Options on* ***Grow Grid from Spline: Parameters*** *window*

◇ Maximum number of grid cells along spline: Default: 2000
Upper bound of the number of grid cells along the centre spline. The actual number of grid cells is determined by the cell and spline lengths, but will not exceed this number. In practice one has to set this number sufficiently large.

◇ Maximum number of grid cells perp. spline: Default: 40
Upper bound of the number of grid layers that will be grown from the centre spline. The actual number of grid layer is determined by the grid height (specified by the splines) the height of the first grid layer and the grid layer growth factor, see below, but it will not exceed this number. In practice one has to set this number sufficiently large.

◇ Aspect ratio of first grid layer: Default: 0.1
The ratio of the grid cell height and length at the centre spline. If a centre spline is provided solely, the aspect ratio of the grid on either side of the centre spline is determined by this variable.

◇ Grid layer height growth factor: Default: 1.1
The fractional increase of grid layer heights in the exponentially growing part of the grid.

◇ Maximum grid length along centre spline: Default: 500

The maximum grid cell length. Note that the length decreases where the spline curvature increases

◇ Grow grid outside first part (0,1): Default: 0
create the exponentially growing grid supplementary to the uniform part (1) or not (0). This parameter has no effect if no uniform part is present, i.e. no bounding splines are provided. In that case the exponentially growing grid is the sole grid created.

◇ Maximum number of grid cell perpendicular to the center spline in the uniform part: Default: 5
The number of grid cells will not exceed this number. If necessary, the cells will be enlarged, and the aspect ratio is disregarded.

◇ Gridpoints on top of each other tolerance: Default: 0.0001
A tolerance on merged grid lines; for expert users only.

◇ Minimum absolute sine of crossing angles: Default: 0.95
Minimum value of $|\sin\alpha|$ where α is the angle between the edge and the line connecting the circumcentres of the adjacent cells to that edge.

5.6.6 Change colour map

When clicking on the *Settings* $\rightarrow$ *Change Colour Map* menu, a form opens in which you can select the relation between a parameter (i.e. Depth) and the loaded colour maps; see Figure 5.46

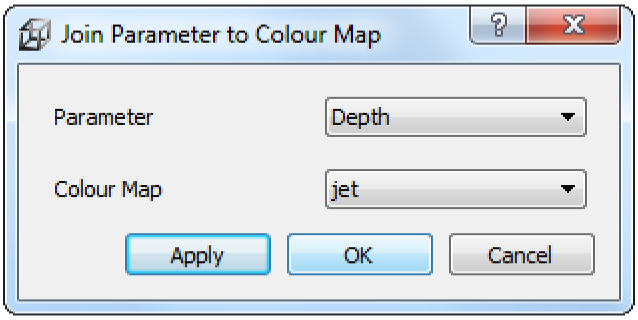


Figure 5.46: *Options on* ***Colour Map for Parameter*** *window*

5.6.7 Legend

When clicking on the *Settings* $\rightarrow$ *Legend* menu, a form opens in which you can define how the iso-colour figures should be displayed; see Figure 5.47

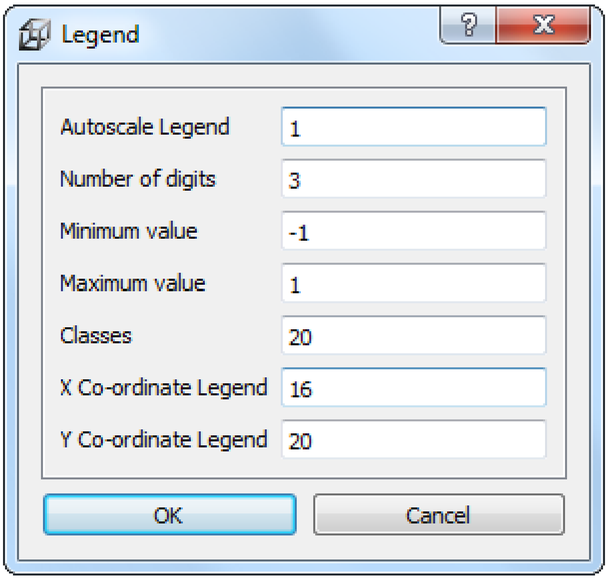


Figure 5.47: *Options on* Settings → Legend *menu*

◇ *Autoscale Legend* default: On
Specify whether the program should determine the iso-colour values automatically, or to do it yourself. If you leave it to the program, it will determine the minimum and maximum depth value within the viewing area and display the number of iso-colours specified above. Zooming in will always result in display of the same number of iso-colours. If you want to specify the iso-colour values yourself, you have to specify one of the three parameters below. When zooming in, the iso-colour values will remain fixed.

◇ *Minimum Value* default: Off
Specifying this value turns autoscale off.

◇ *Maximum Value* default: Off
Specifying this value turns autoscale off.

◇ *Classes* default: 20
The number of classes can be specified

◇ *X Co-ordinate Legend* default: 16
x Co-ordinate of lower left corner of legend in pixels

◇ *Y Co-ordinate Legend* default: 20
y Co-ordinate of lower left corner of legend in pixels

5.6.8 Colours

When clicking on the *Settings* → *Colours* menu, a form opens in which you can define the colours for background, land boundary, polygons, etc.; see Figure 5.48

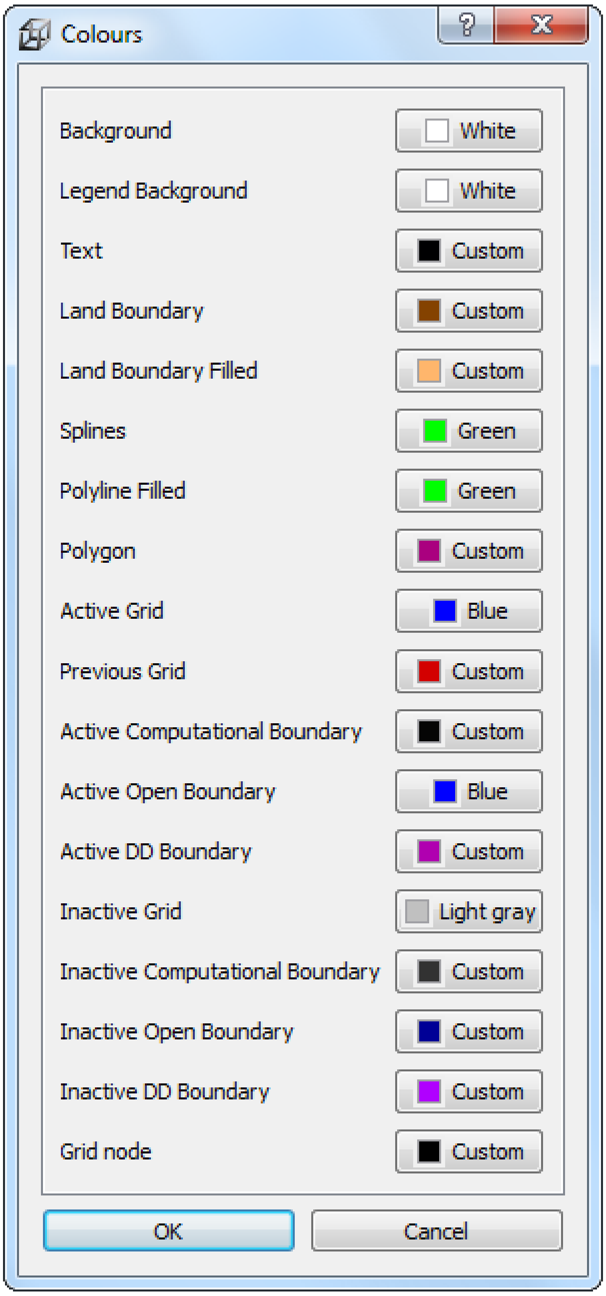


Figure 5.48: *Options on* Settings → Colours *menu*

5.6.9 Sizes

When clicking on the *Settings* → *Sizes* menu, a form opens in which you can define the linewidth and dotsize in pixels. See Figure 5.49

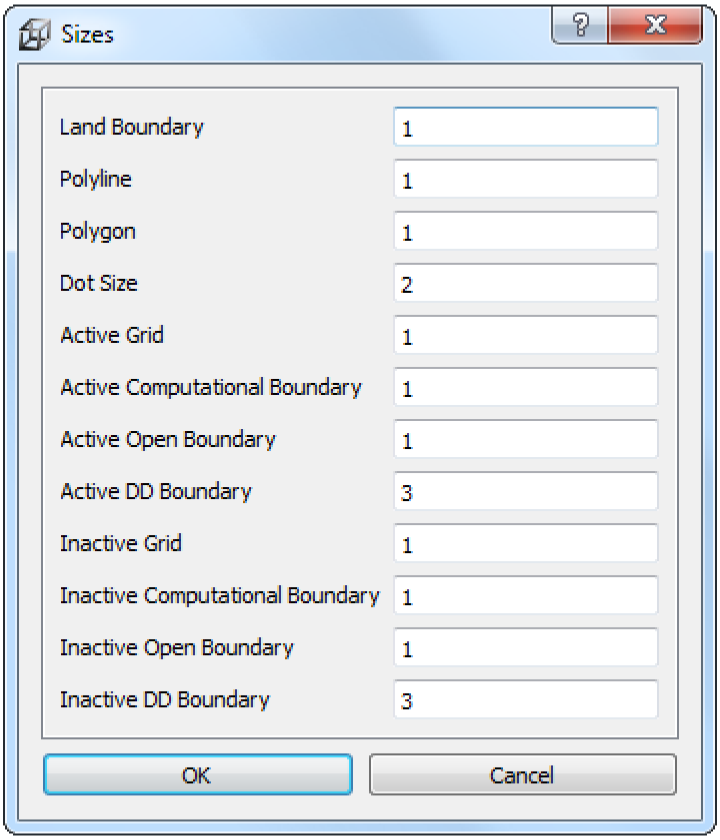


Figure 5.49: *Options on* Settings → Sizes *menu*

5.6.10 Order caches

The parameters set in the **Order caches** window, see Figure 5.50, influence the drawing order of the several items. The drawing order of the caches is: $5, 4, 3, 2, 1, 0$. Cache 5 is drawn first and cache 0 is drawn last. So the items which will drawn in cache 0 are drawn on top. If there is no need to draw a cache it will not be done, this improves the drawing performance by avoiding unnecessary drawings. Therefore, if an item is changed in cache 3 only caches $3, 2, 1$ and 0 are drawn.

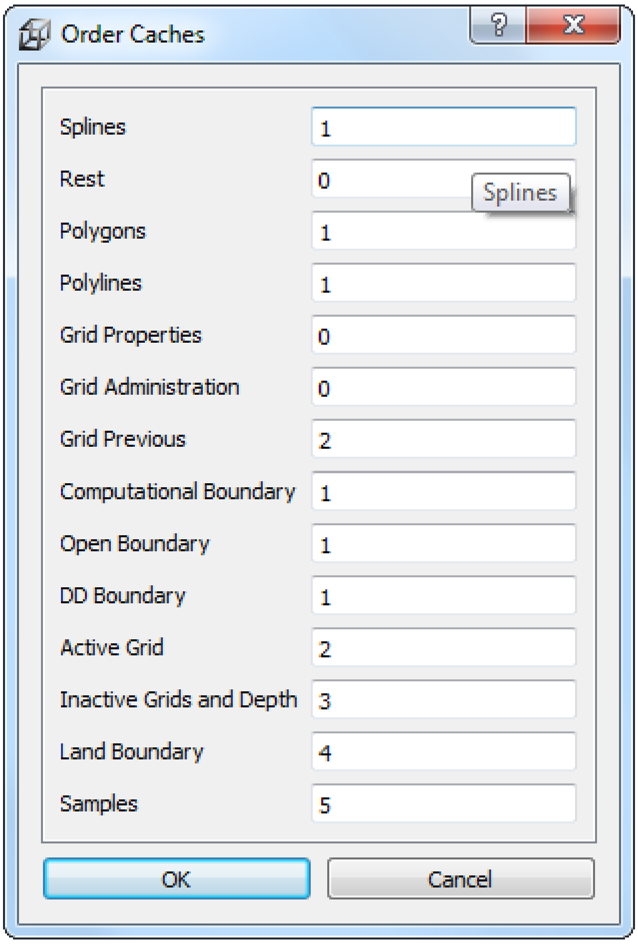


Figure 5.50: *Options on* ***Order Caches*** *window*

◇ Splines	default: 0
◇ Rest	default: 0
◇ Polygons	default: 0
◇ Grid Properties	default: 0
◇ Grid Administration	default: 0
◇ Grid Previous	default: 2
◇ Computational Boundary	default: 1
◇ Open Boundary	default: 1
◇ DD Boundary	default: 1
◇ Active Grid	default: 2
◇ Inactive Grids and Depth	default: 3
◇ Land Boundary	default: 4
◇ Samples	default: 5

5.7 Help menu

On the *Help* menu, you may choose to read the user manual or the version number of RGF-GRID ; see Figure 5.51

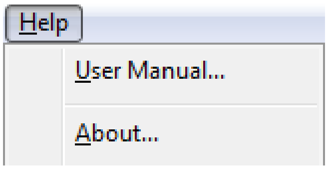


Figure 5.51: *Options on* Help *menu*

5.7.1 User manual

When clicking on the *Help* → *User Manual* the user manual of RGFGRID will be displayed (file <RGFGRID _User_manual.pdf>).

5.7.2 About

When clicking on the *Help* → *About* a window will display the current version number of RGFGRID.

6 Tutorial

6.1 Harbour

Start RGFGRID as explained in Section 3.3 with the current directory set to <$D3D_HOME/tutorial/rgfgrid/harbour>.

This tutorial uses the land boundary and spline files which are available in that directory.

6.1.1 Co-ordinate system

Before opening a land boundary and generating a grid, you have to indicate in which co-ordinate system you are going to work. This example will use Cartesian co-ordinates.

- ◇ On the *Co-ordinate System* menu, click *Cartesian Co-ordinates*.

6.1.2 Open a land boundary

Open a file which defines the land boundary.

- ◇ On the *File* menu, point to *Attribute Files* and click *Open Land Boundary*.

If you have not started RGFGRID with the current directory being
<$D3D_HOME/tutorial/rgfgrid/harbour>
then browse to this directory.

- ◇ Highlight and *Open* file <harbour.ldb>.

After opening the boundary outline as shown in Figure 6.1 is now visible on your screen.

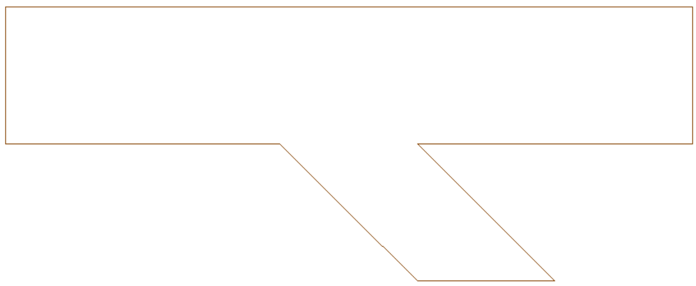


Figure 6.1: *Land boundary outline of <harbour.ldb>*

6.1.3 Zoom in and out

To zoom in or out several facilities are available:

- ◇ click  to zoom in or  to zoom out
- ◇ press the + or - key while keeping the `Ctrl`-key pressed.
- ◇ use the mouse scroll wheel.

To zoom in on a specific area:

- ◇ use  and drag a box around the area.

To zoom out to the full extent:

- ◇ click  to zoom to full extent

To pane the grid: keep the `Ctrl`-key pressed and move around with the cursor.

6.1.4 Define splines

Open a file with definition of splines.

- ◇ On the *File* menu, point to *Import* and click *Splines*.
- ◇ Open spline file <harbour.spl>.

After opening the file with spline definitions, your screen looks like Figure 6.2.

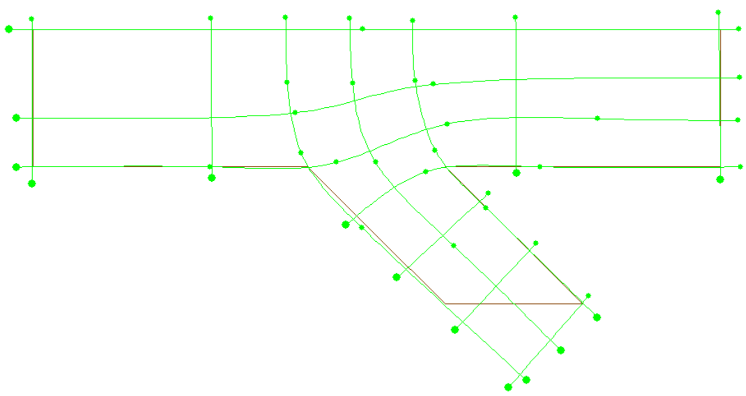


Figure 6.2: *Display of splines and land boundary in the 'harbour' tutorial*

6.1.5 Generate grid from splines

Generating a grid after opening splines.

- ◇ On the *Operations* menu, click *Change Splines into Grid*, or click  on the toolbar.

From the splines a regular grid is generated, see Figure 6.3.

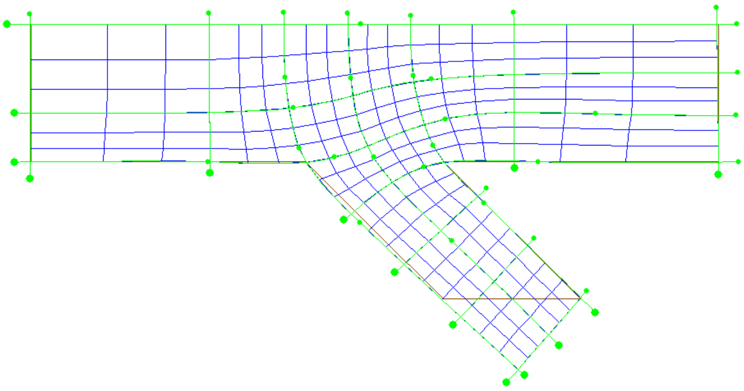


Figure 6.3: *Spline grid changed into result grid with a refinement of 3*

The refinement factors can be specified in *Settings* → *General*. The default is 3 in both directions.

Do not show the spline anymore

◇ On the *View* menu, point to *Splines*, and click *No Splines*, see Figure 6.4.

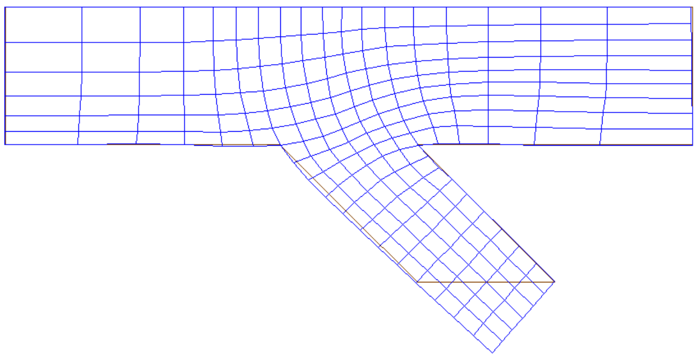


Figure 6.4: *Splines not displayed anymore*

6.1.6 Refine grid

Refine a grid.

◇ On the *Settings* menu, click *General*.
◇ Specify M and N refinement factors of "3" by "3".
◇ On the *Operations* menu, click *Refine Grid*.

The result of the refinement should look like in Figure 6.5.

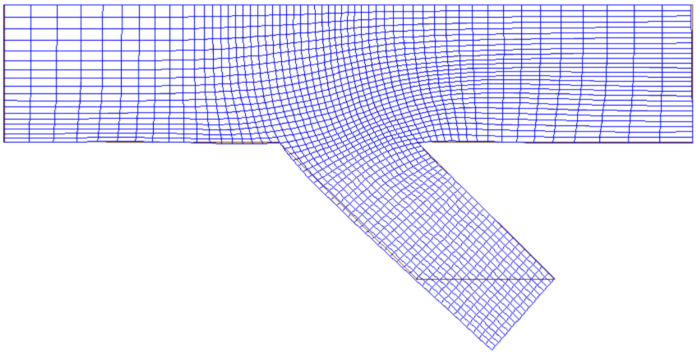


Figure 6.5: *Grid after another refinement of 3 by 3*

6.1.7 Fit grid boundary to land boundary

The grid boundaries can be fitted to the land boundary.

◇ On the *Edit* menu, point to *Line* and click *Line to Land Boundary*.

◇ You are now in "line to land boundary".

◇ Click the end points of the grid line segment that you want to attach to the land boundary. The concerning segment of the grid line will be highlighted. To expand the area of influence of the attachment transformation click one point on the grid side of the indicated line and perform the transformation, see Figure 6.6. When no influence area is indicated just the grid boundary segment is shifted to the land boundary.

For instance a point halfway in between fixed boundaries.

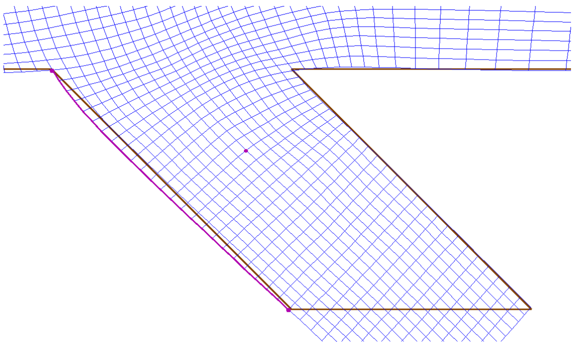


Figure 6.6: *Indicating outer grid line and influence area to be moved to land boundary*

◇ Click right to execute the *Line to Land Boundary* action, the result will look like Figure 6.7.

Remark:

◇ The previous grid can be shown using one of the options in *View* → *Previous Grid*.

This steps can be repeated until all necessary grid boundaries are fitted to the land boundary.

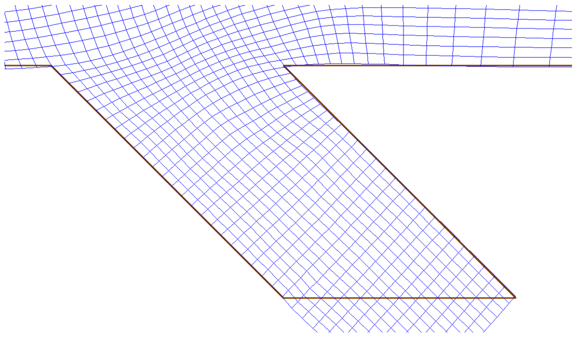


Figure 6.7: *Grid after Line to Land Boundary action*

6.1.8 Check grid orthogonality

To inspect the quality of the grid, for instance the orthogonality:

- ◇ On the *Operations* menu, point to *Grid Properties*, and click *Orthogonality*.

This will show a plot (see Figure 6.8) of the cosine values of grid corners. The cosine values should be close to zero. The error in the computed direction of the pressure term in Delft3D-FLOW is proportional to these values. In offshore areas the orthogonality should be less than 0.02. Near closed boundaries, higher values are sometimes acceptable.

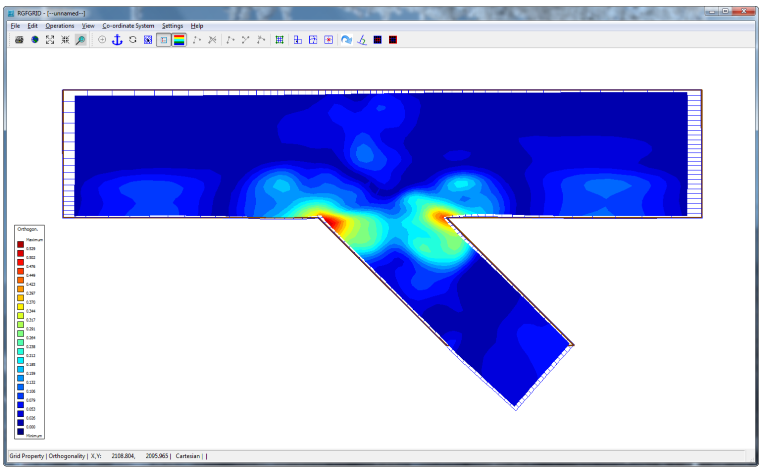


Figure 6.8: *Grid properties; orthogonality*

6.1.9 Orthogonalise grid

Now we will improve the orthogonality:

- ◇ On the *Operations* menu, click *Orthogonalise Grid* to improve the grid orthogonality.

Remarks:

- ◇ Default the legend uses auto-scaling. With respect to Figure 6.8, the colours used in Figure 6.9 are the same but the range of cosine values is different.
- ◇ To set the auto-scaling off, click on *Settings* → *Legend*, and change the appropriate parameter to 0.

To hide the grid properties.

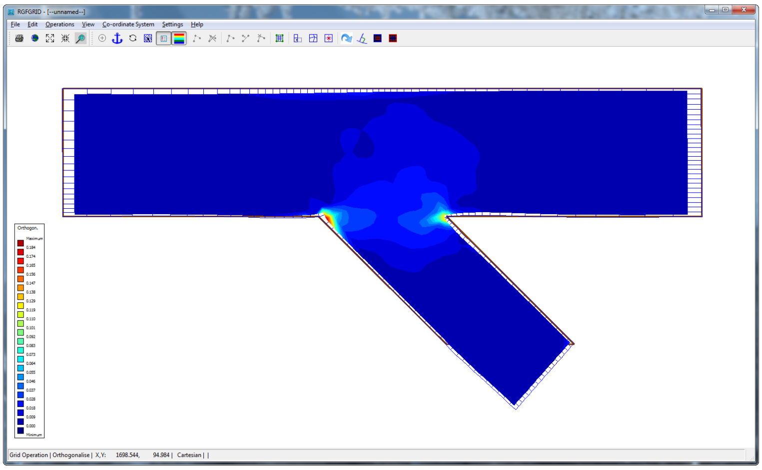


Figure 6.9: *Grid properties; orthogonality. After 1 orthogonalisation action*

◇ Select the *View* menu, point to *Grid Property* and click *No Grid Property*.

In practice, making the grid orthogonal, you will work in blocks rather than on the whole grid. To make the grid orthogonal locally:

◇ Select on the *Edit* menu, point to *Block* and click *Orthogonalise.*
◇ Click 2, 3, or 4 opposite block corners, see Figure 6.10.
◇ Click right to activate the orthogonalisation process.

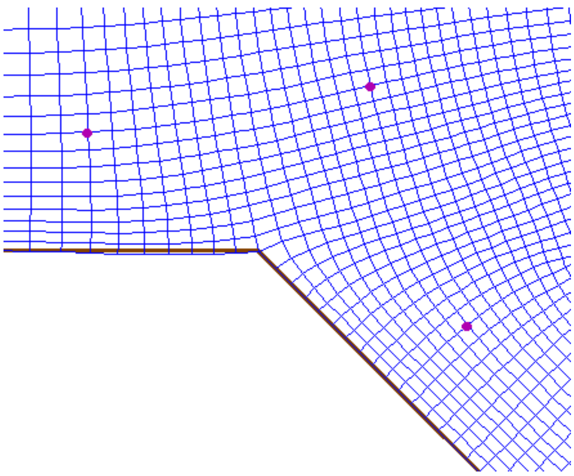


Figure 6.10: *Indicating corners for Block Orthogonalise*

6.1.10 Check other grid properties

◇ On the *Operations* menu, point to *Grid Properties* and click *Orthogonality* again

to check the result of the previous action, see Figure 6.11. Other grid properties such as grid smoothness and resolution can also be displayed. The grid should be smooth to minimise truncation errors in the finite difference scheme. Adjacent grid cell sizes should vary less than 20 percent, although locally exceptions may be acceptable.

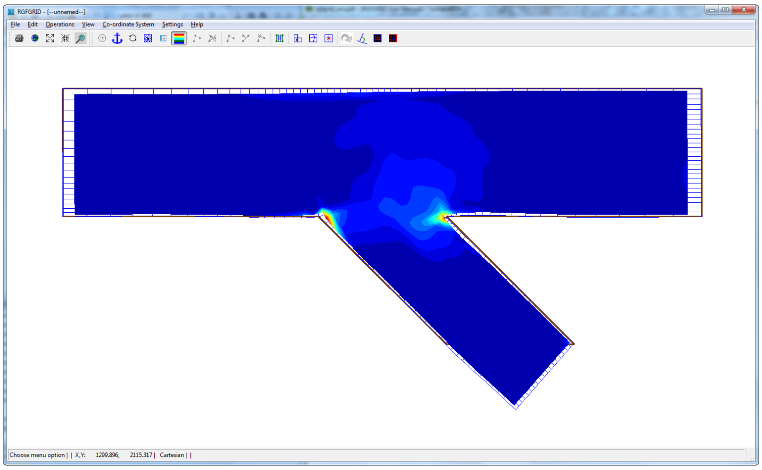


Figure 6.11: *Grid orthogonality after one block orthogonalisation operation*

6.1.11 Completion

To delete grid cells outside the land boundary:

- ◇ On the *Edit* menu, point to *Block* and click *Delete Interior*.
- ◇ Indicate a block to be deleted by clicking on opposite corners.
- ◇ Click right to activate the delete action.

You can also delete individual grid points:

- ◇ On the *Edit* menu, point to *Point* and click *Delete*.
- ◇ Press the *D*-key and click points you want to delete.

In the *Point Mode* you can also move (replace) grid points.

- ◇ First press the *R*-key, next click on a grid point, move it somewhere else, and click again.

Also in the *Point Mode*, you can add individual grid cells.

- ◇ First press the *I*-key and then click in a border cell near the concerning edge.

Do not delete grid cells outside the land boundary in earlier refinement steps, if this introduces staircase boundaries (as in the present example at the end of the harbour).

The final grid should look like Figure 6.12.

Remarks:

- ◇ Each corner point on the grid will remain fixed in the orthogonalisation procedure.
- ◇ Only internal points and points along boundaries can be shifted to improve the grid orthogonality. Creating a staircase boundary in the early stages of the iterative grid generation procedure hampers this procedure.

To shift individual grid cells:

- ◇ On the *Edit* menu, select *Point* and click *Insert*, *Move* or *Delete* or click the icons , 

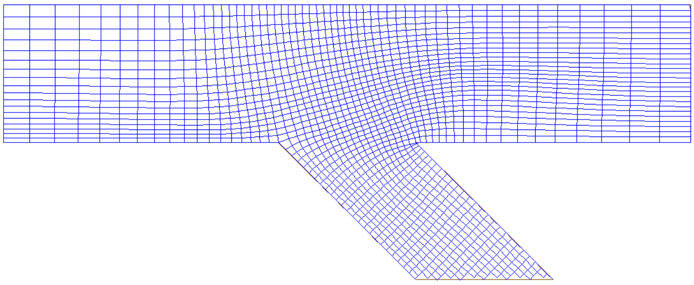


Figure 6.12: *Final result after refining, obsolete grid cells removed*

or on the toolbar.

◇ You can also switch between the move, insert and delete actions by pressing the `I`, `R` or `D` key.

Once a mode is selected, use the left mouse to let the actions take effect. Press `Esc` to undo edit actions. In the ‘spline edit mode’-mode of the program, the same keys can be used.

To exit the RGFGRID program.

◇ On the *File* menu, click *Export* $\rightarrow$ *Grid (RGFGRID)*
◇ On the *File* menu, click *Exit*.

6.2 Grid design samples

In this example we will demonstrate how to design a grid in which the shape is based on the bathymetry. This tutorial refers to the <river> directory of the RGFGRID tutorial.

◇ Start RGFGRID, and browse to the tutorial directory <river>.
◇ Open the grid file <river.grd>.
◇ Open the samples file <river.xyz>, see Figure 6.13.

◇ On the *Settings* menu, click *Orthogonalisation*:

  ◻ Set the *Design Method* to “3”.
  ◻ *Depth Design, Depth vs. Slope Weight* to “0.2”

◇ Click *OK*.
◇ On the *Operations* menu, click *Orthogonalise Grid*, the result is shown in Figure 6.14.

To exit the RGFGRID program

◇ On the *File* menu, click *Export* $rightarrow$ *Grid (RGFGRID)*.
◇ Click *Exit* on the *File* menu.

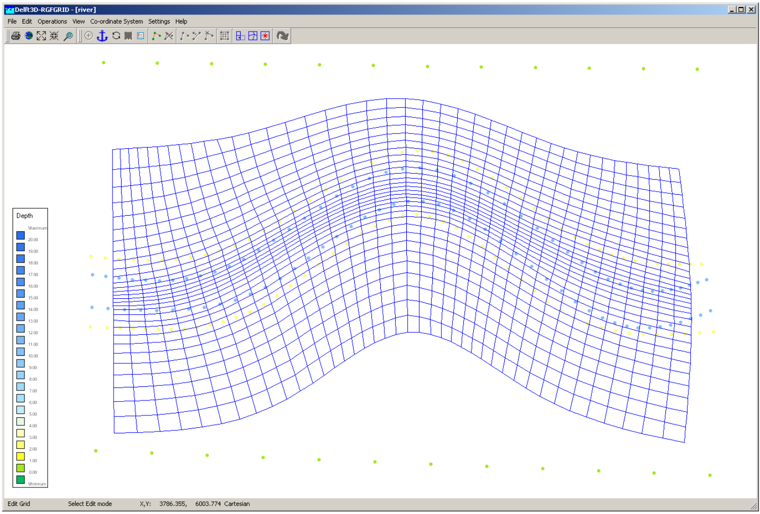


Figure 6.13: *Grid and samples for the grid design based upon bathymetry*

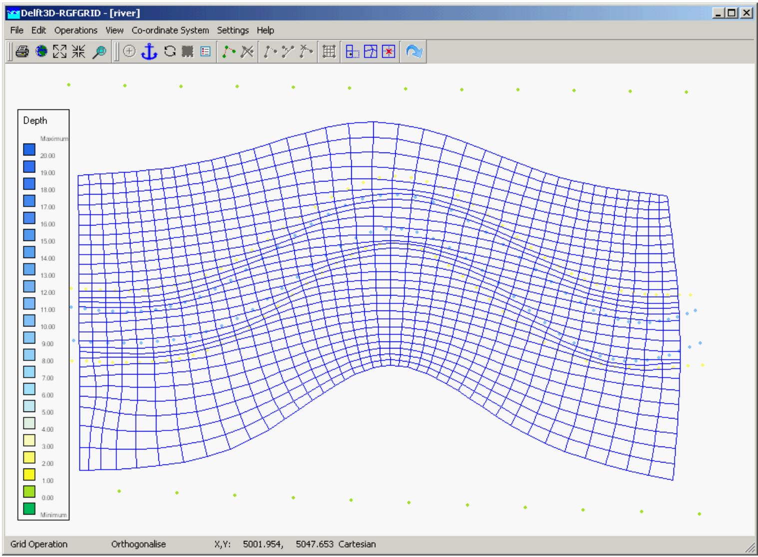


Figure 6.14: *Result grid after orthogonalisation using samples*

6.3 Paste two grids

This tutorial refers to the <Paste_Passive_Grid_to_Grid> directory of the RGFGRID tutorial.

A reason to paste two grids can be to extend an existing grid with another grid.

- ◇ Start RGFGRID, and browse to the tutorial directory <Paste_Passive_Grid_to_Grid>.
- ◇ Open the grid files <fti_02_north.grd> and <fti_02_south.grd.>
- ◇ On the *Operations* menu, click *Paste two Grids*.

to paste the two grids.

Restrictions:

- ◇ Perpendicular to the intersection line, the grid lines of both grids should be similar. Thus, refinements are not allowed.
- ◇ The grid points to paste should already be close to each other.

◇ This option only works if two grids are loaded in RGFGRID. After pasting two grids another grid can be loaded to paste.

6.4 Regular grids, irregular grids and their mutual coupling

This tutorial refers to the <dflowfm> directory of the RGFGRID tutorial.

The present section deals with four aspects of new functionalities related to the extension from curvilinear grids with unstructured, triangular grids:

1. to 'grow' a curvilinear grid (a 'regular' grid) from a geometric base line,
2. to generate unstructured, triangular grids: 'irregular' grids,
3. the coupling between regular and irregular grids,
4. the relation to existing regular grid generation options.

6.4.1 A new method to generate curvilinear grids

RGFGRID provides an improved method to generate curvilinear meshes directly from splines. In this method, a mesh is gradually developed from a base line of the channel towards the boundaries. This method requires less actions by the user and provides better orthogonality.

This approach can be illustrated as follows:

1. Load the land boundary file <scheldtharbour.ldb>.
2. Draw two cross-splines, intending to mark the inflow and outflow cross-section of the river part, through *Edit → Spline → New*.
3. Draw two additional splines, intended to loosely follow the riverbanks, in longitudinal direction.
4. Select one of the longitudinal splines and select the option *Edit → Spline → Spline to Land Boundary (new)*. The spline is now snapped to the land boundary. Repeat this action until you are satisfied with the result. For the second longitudinal spline, the actions can be repeated. The result of these actions is provided in the directory as <scheldtsplines.spl>.
5. To generate a curvilinear grid, choose *Settings → Grow grid from splines*. You will be able to set several settings of the operator. The upper 7 entries should be adapted into the values shown in Figure 6.15.

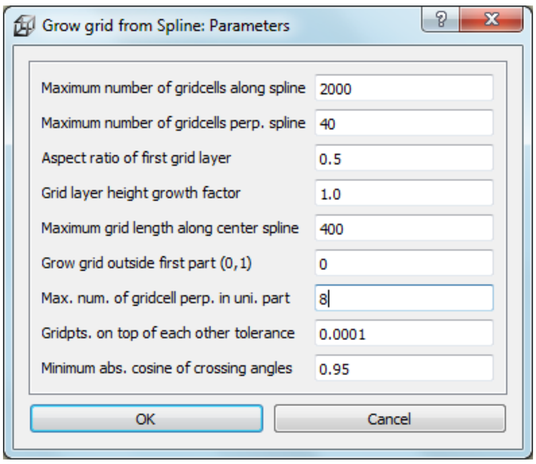


Figure 6.15: *Settings for the 'grow grid from splines' procedure.*

A brief explanation:

- ◇ Using the parameter *Max. num. of grid cells perpin uni. part*, the user can give an indication of the number of cells across the width between the longitudinal splines.
- ◇ By using the parameters *Maximum grid length along centre spline*, the user can give an indication of the length of the cells in longitudinal direction. Based on the value of the parameter *Aspect ratio of first grid layer*, the algorithm establishes a suitable mesh, under the restrictions of the prevailing maximum numbers of grid cells (first two entries).
- ◇ The option *Grid layer height growth factor* enables the user to demand for a non-equidistant mesh in cross-sectional direction. The value represents the width-ratio of two adjacent cells. Using the option *Grow grid outside first part (0/1)*, one can extend a mesh outside the longitudinal splines, for instance to capture winter bed regions of a river.

6 After entering the values of Figure 6.15, choose *Operations* → *Grow grid from splines*. This will deliver the mesh as shown in Figure 6.16.

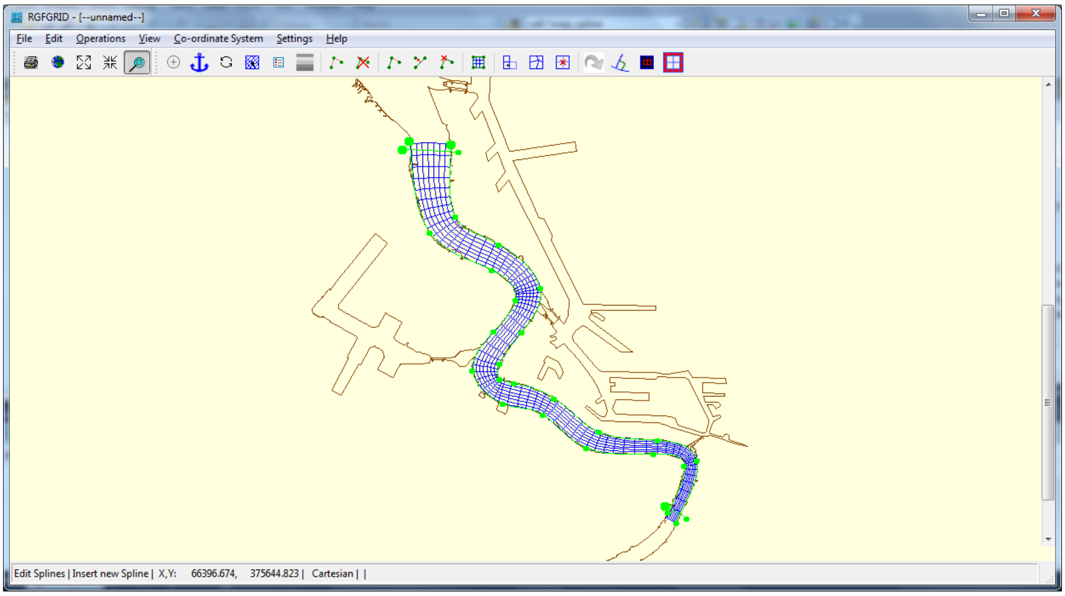


Figure 6.16: *Generated curvilinear mesh after the new 'grow grid from splines' procedure.*

7 To be able to further amend the grid, choose *Operations* → *Convert grid* → *Regular to irregular*. Strictly, the grid is now not curvilinear anymore, but unstructured.

8 Choose *View* → *Grid property* → *Coloured edge* and then *Operations* → *Grid properties* → *Orthogonality*. Now, the orthogonality of the mesh is shown. The result for the grid is also provided in the directory as <scheldtcurvi_net.nc>.

6.4.2 Irregular grids

From the previous section, a curvilinear mesh is available for the Scheldt river. The river is separated from the harbour, west of the river, by a sluice. The small area between the sluice and the Scheldt will benefit from an unstructured mesh options of RGFGRID because of its irregular geometry. This irregular geometry is meshed in this section first and afterwards connected to the existing Scheldt river mesh.

The approach is as follows:

1 Click on *Edit* → *Polygon* → *New*. The intention is to mark the area of interest (i.e. the area that should be captured by the grid) through a polygon.
2 Start drawing a polygon at a distance of the order of a grid cell away from the curvilinear mesh. Let the second point be at a relatively small distance from the first one. This distance is later used as an indication of the size of the triangular grid cells to be placed.
3 Mark the elementary locations of the area (land boundary) and place the final point again at a distance of the order of a grid cell away from the river mesh.

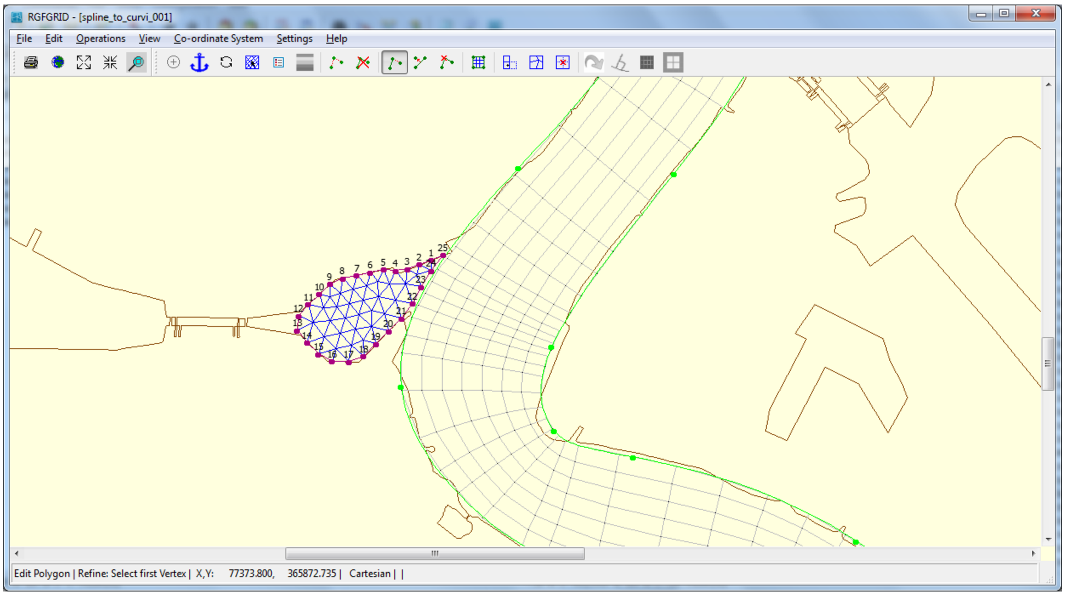


Figure 6.17: *Generated irregular grid within a polygon.*

4 Next, we choose *Edit* → *Polygon* → *Refine* and click on *two* points at the righthand-side, located close to each other, and click on the *right mouse button*. Now, the polygon is divided into a finer set of line elements. This refined polygon is also available as <scheldtpolygon.pol>. Some remarks:

  ◇ The distance between the points of the polygon is derived from the distance of the two polyline segments at both sides of the *selected* segment. The length of the polyline segments varies linearly from the segment length at the one side of the selected segment towards the segment length at the other side of the selected segment.
  ◇ You can play around to see how this works. If needed, you can add extra polyline points by choosing *Edit* → *Polygon* → *Insert point*. Choose *Edit* → *Polygon* → *Move point* if a point move would make sense.
  ◇ You can snap the refined polygon to the land boundary through *Edit* → *Edit polyline(s) within existing polygon*. The result is shown in Figure 6.17.

5 Choose *Operations* → *Create grid from polygon*. The result is shown in Figure 6.17.
6 Improve the orthogonality through *Operations* → *Orthogonalise grid*.
7 To further orthogonalise the grid by manipulating the settings, choose *Settings* → *Orthogonalisation (irregular)*, and then choosing *Operations* → *Orthogonalise grid* once again. The result for the grid is also provided in the directory as <scheldttriangle_net.nc>.

6.4.3 The coupling of regular and irregular grids

From the previous tutorial, we have ended up with two separate grids. Obviously, these two grids should properly be integrated into one single grid. Before we can couple the two grids, we should first make sure that the typical gridsize is of the same order of magnitude for both grids at the location where the connection is to be laid. Hence, basically two operations are to be done:

- ◇ Split the grid cells in the Scheldt river grid over the full width. Hence, the gridcell size in the river will match the grid cell size of the unstructured grid.
- ◇ Merge the two grids and put connections in between of these.

The splitting can be established as follows:

1. Select the river grid through *Edit* → *Select domain* and clicking the river grid.
2. Choose *Edit* → *Grid* → *Split row or column*.
3. Select the grid lines that should be split. Start at the left boundary, and apply multiple line split operations towards the other side of the Scheldt river.
4. Try to achieve the picture shown in Figure 6.18 as regards the typical grid size in the curved area.

The merging part of the coupling schedule can be done as follows:

1. Choose *Edit* → *Allow Multi Select*. By now, you enable the option to select multiple grids.
2. Choose *Edit* → *Select domain* and click on the triangular part of the grid. As soon as you have clicked on it, both meshes are highlighted blue.
3. Merge the two seperate grids through *Operations* → *Merge grids*. Now, the grids have been merged. The result of this merging operation is provided as <scheldtmerge_net.nc>.

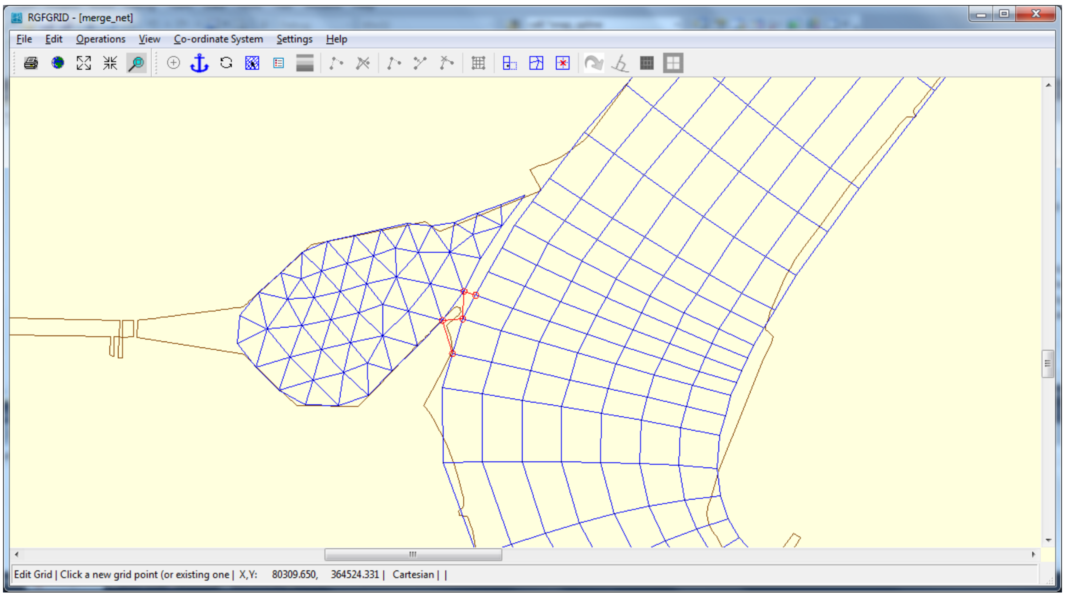


Figure 6.18: *Coupling of the two grids (regular and irregular, in blue) through manually inserting connecting grid lines (in red lines) between the two grids.*

4. As soon as the grids have been merged, new connections can be laid. Hence, choose *Edit* → *Grid* → *Edit* and then *Edit* → *Grid* → *Point insert*. Insert new gridlines in a zigzag-like style: see the red grid lines in Figure 6.18. Now, you will benefit from the (more or less) equal resolution in the river region as in the unstructured region. The integrated grid is available as <scheldtcoupling_net.nc>.

6.4.4 Relation to existing regular grid generation

The sluice area can the best be captured by a regular grid because of its rectangular shape. Thereto, you could first draw some splines in this area (see for instance Figure 6.19). Then you can establish a 3×4 grid in each block, separated by splines. To that end, choose *Operations* $\rightarrow$ *Change splines into grid*. The splines for the sluice area are available as <scheldtsluice.spl>.

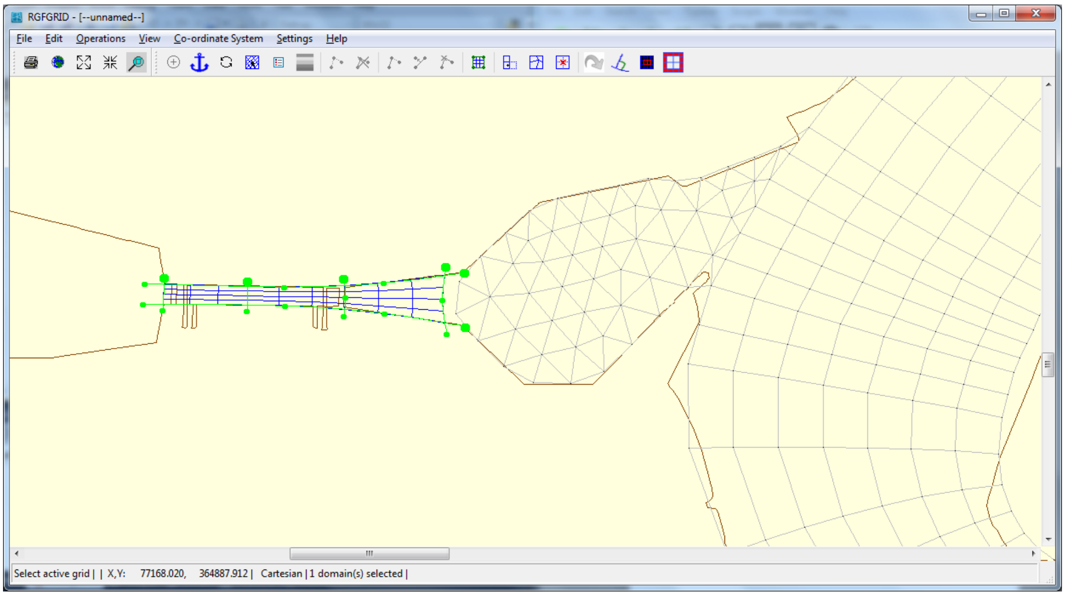


Figure 6.19: *A regular grid is suitable for the sluice area. Connections with the existing grid should further be established as well as additional orthogonalisation iterations.*

The docks of the harbour are rectangularly shaped as well. Hence, regular grids are preferred. You can also try to establish an irregular grid in this area. Thereto, draw a polygon, refine this polygon and choose *Operations* $\rightarrow$ *Create grid from polygon*.

Notice that the grid configuration as shown in Figure 6.19 needs proper connections between the sluice area and the already existing grid and, moreover, further orthogonalisation iterations before it can actually be used in computations. An example of further elaboration of the area is provided as <scheldtfinal_net.nc>.

6.5 Multi-domain grids and DD-Boundaries

This tutorial refers to the <dd_grids> directory of the RGFGRID tutorial.

The grids for a multi-domain model must satisfy the following rules:

◇ At sub-domain interfaces the grids should be nicely connected (no overlap and "no holes" between sub-domains), see Section 5.3.15.
◇ In case of horizontal, grid lines in the coarse domain should be continued in the fine sub-domain, see Figure 5.11. Thus, there should be a 1-to-N refinement, with N an integer number.
◇ In case of horizontal refinement it is advised to have an equidistant refinement.
◇ Grids must be of the same type (thus, all in Spherical co-ordinates, or all in Cartesian co-ordinates).

- ◇ The grid orientation should be the same (increasing M- or N-numbering in both sub-domains at the DD-boundary).
- ◇ No coupling of columns to rows or vice versa.
- ◇ Sub-domain interfaces should be straight lines (no stair-case interfaces).

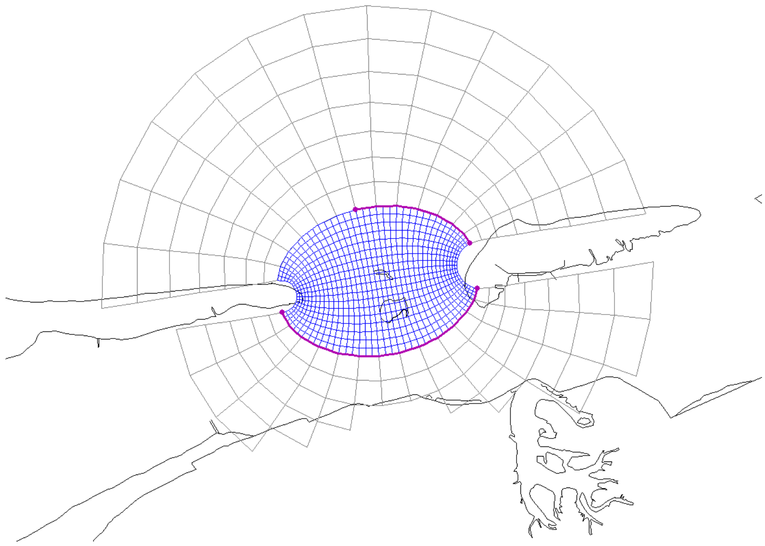


Figure 6.20: *Example of grid refinement in the horizontal direction*

RGFGRID has an option to let sub-domain interfaces coincide, see Section 5.3.15. The best moment to use this option is before refining one of the sub-domains, i.e. as long as the refinement is 1-1.

To demonstrate this functionality

- ◇ Go to directory <let_interfaces_coincide>, within the <dd_grids> directory.
- ◇ Open the grids <left_n.grd> and <right_0.grd>, zoom in on the interface between both grids.
- ◇ On the *Edit* menu, point to *DD Boundaries* → *New*.
- ◇ Click with the mouse the end points of the DD-Boundary on the interface of the active grid.
- ◇ Click *Operations* → *Attach Grids at DD Boundaries* then the whole DD-Boundary will be shifted to the interface points of the inactive grid, see Figure 6.21.

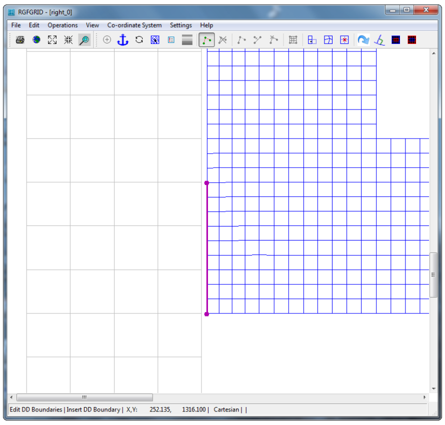
Now we have the interfaces coincide we are going to define the interfaces between the various sub-domain models.

Close and start RGFGRID in the <dd_grids> directory and load all grids into RGFGRID.

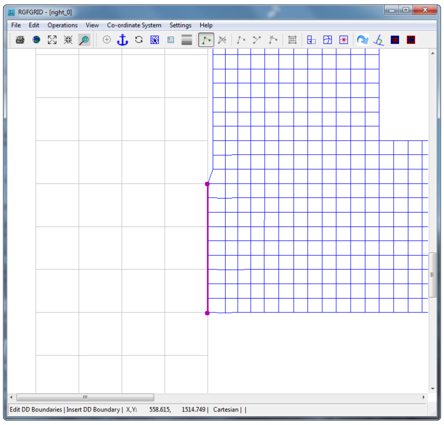
- ◇ Open the grids <bot.grd>, <left.grd>, <right.grd> and <top.grd>.

To define the DD-boundaries you select one domain as active domain, Figure 6.22:

- ◇ Make the <right.grd> the active domain
- ◇ On the *Edit* menu, point to *DD Boundaries* and click *New*.
- ◇ Specify the four DD-Boundaries as shown in Figure 6.22
- ◇ Make the <left.grd> the active domain



(a) *Before attach operation*



(b) *After attach operation*

Figure 6.21: *Let interface grid points coincide*

◇ Specify the last DD-Boundaries as shown in Figure 6.22

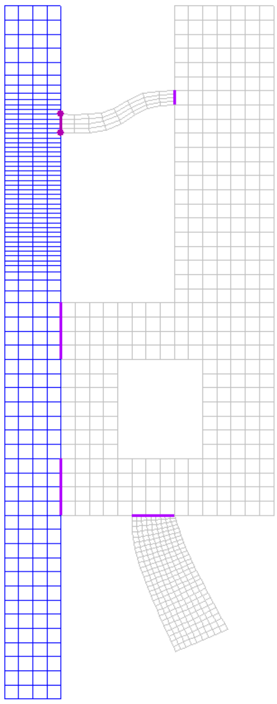


Figure 6.22: *Interfaces between sub-domains*

Now we have defined the DD-boundaries between the various domains. To gather all these information into 1 file:

◇ On the *Operations* menu, click *Compile DD Boundaries*.
◇ The **Save DD-Boundaries** dialog opens, see Figure 6.23.

◇ After pressing *OK* a message appears about the number of DD-Boundaries.
◇ Press *OK*.

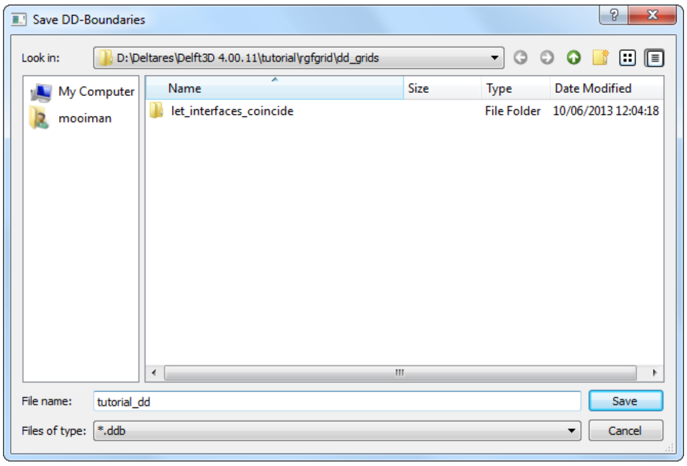


Figure 6.23: *The* ***Save DD-Boundaries*** *dialog*

The final administration will be written to a file named <tutorial_dd.ddb>. For the tutorial it looks like:

```
left.grd      5    61     5    65   top.grd      1     1     1     5
bot.grd       1    28    10    28   right.grd    6     1     9     1
left.grd      5    14     5    18   right.grd    1     1     1     5
left.grd      5    25     5    29   right.grd    1    12     1    16
top.grd       9     1     9     5   right.grd    9    30     9    31
```

Remarks:

- ◇ Before defining DD-boundaries check the orientation of each sub-domain grid.
- ◇ The orientation and order of interfaces (DD-boundaries) is free.
- ◇ It is irrelevant in which grid you define an interface, but define it once.
- ◇ If the interfacing boundaries coincide, be aware that when you orthogonalise a sub-domain grid, the grid points along these interfaces may move. To keep these points at the same place, you just re-define the DD-boundaries.

To exit the RGFGRID

- ◇ Click *Exit* on the *File* menu.

6.6 RGFGRID in the ArcMap environment

In this case you should be familiar with using co-ordinate systems in ArcMap.

ArcMap layers (e.g. shape files, SDE layers) most times have also information about the co-ordinate system (spatial reference). If not available, ArcMap marks this as unknown. On the other hand, if it is known to you, spatial reference can be added to these layers by the program ArcCatalog.

If the layers in ArcMap have a projected co-ordinate system or probably an unknown co-ordinate system, then you can use *Cartesian* in RGFGRID. It is your responsibility the co-ordinates have the unit metres. You can see the used (projected) co-ordinate system of the layers and of the data frame via the properties of the data frame. For RGFGRID the co-ordinate system of the data frame is leading. As you know, ArcMap has possibilities to set the co-ordinate system of the data frame, while layers have different co-ordinate systems.

If you want to use spherical co-ordinates in RGFGRID while using ArcMap, the co-ordinate system of the data frame must be WGS84 (in ArcMap it has the name CGS_WGS_1984). This will be the case when all layers have this co-ordinate system.

If you are familiar with ArcMap you can have one or more layers with different co-ordinate systems and select (import) the WGS84 system for the data frame. Figure 6.24 shows the properties window of de data frame, <custom> give the co-ordinate system of the data frame.

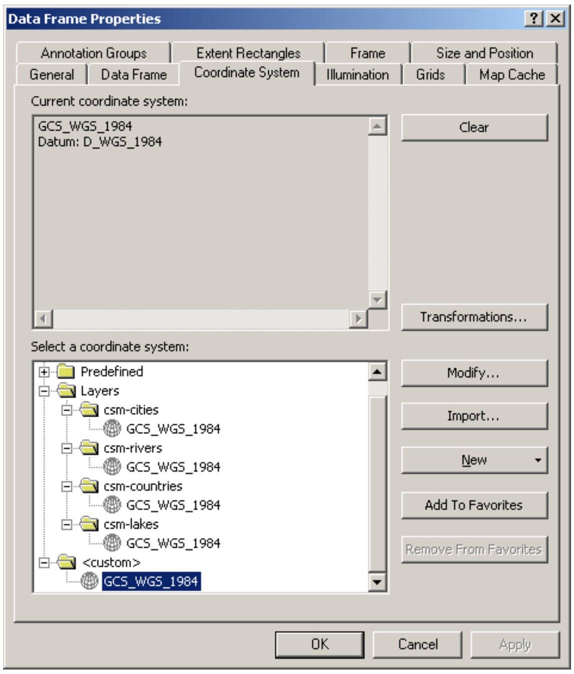


Figure 6.24: *ARC-GIS data frame properties form*

You start loading layers or an <∗.mxd > file in ArcMap. The co-ordinate system of the data frame must be as described above. You will see that ArcMap displays the values of longitude and latitude as plane co-ordinates in degrees.

Then you can start using the commands and functions of RGFGRID. The program has a menu item to change into projected co-ordinates. When using this command, the layers in ArcMap are displayed as projected and also the grid, polygons, samples of the program.

When using the menu item *View* → *Spherical Co-ordinates* → *Plane Co-ordinates*, both the layers in ArcMap and the objects of Dleft3D-RGFGRID are displayed as plane co-ordinates.

Centring the screen in stereographic mode

In the standalone version of RGFGRID, the visualisation using the stereographic projection method always uses the centre of the screen as the point where the screen touches the sphere. This is more difficult to realise when working within the GIS environment because screen handling now is governed by the GIS system. Therefore a special function has been implemented to perform this task of centring the computer screen. This function can be invoked by clicking the menu item *Operations* → *Change Centre of Projection*, see Figure 6.25. When using this command the centre of the projection is set to the centre of the screen. This action needs recalculation of the projection and a new screen refresh. By using this command you are aware of changing the screen. When using zoom in, zoom out, pan, the centre of the projection does not change. So there is more performance and a smooth screen-refresh in this case.

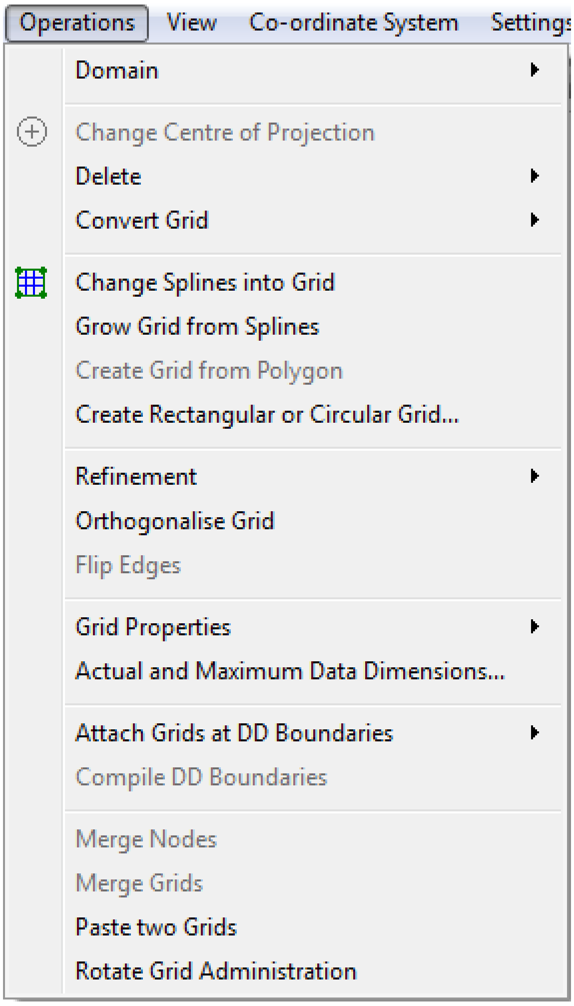


Figure 6.25: *Options on the* Operations *menu*

References

Delft3D-FLOW, 2013. *Delft3D-FLOW User Manual*. Deltares, 3.14 ed. 2

GPP, 2013. *Delft3D-GPP User Manual*. Deltares, 2.14 ed. 22

Huang, W., 2001. “Practical Aspects of Formulation and Solution of Moving Mesh Partial Differential Equations.” *Journal of Computational Physics* 171: 753–775. 68, 69

QUICKPLOT, 2013. *Delft3D-QUICKPLOT User Manual*. Deltares, 2.14 ed. 22

SWAN, 2000. *SWAN Cycle III version 40.11 User Manual (not the short version)*. Delft University of Technology, Delft, The Netherlands, 0.00 ed. 2

A Files of RGFGRID

In the following sections we describe the attribute files used in RGFGRID.

For each file which can be handled by RGFGRID we give the following information:

- ◇ File contents.
- ◇ Filetype (free formatted, fix formatted or unformatted).
- ◇ Filename and extension.
- ◇ Generated by (i.e. how to generate the file).
- ◇ Restrictions on the file contents.
- ◇ Example(s).

Remarks:

- ◇ The access mode of all attribute files is sequential.
- ◇ In the examples the file content is printed in font Courier and comment (not included in the file) between curly brackets font, unless explicitly stated differently.

A.1 Delft3D project file

File contents	Domain input for a model.
Filetype	ASCII
File format	Free formatted.
Filename	<*name*.d3d>
Generated	RGFGRID, QUICKIN, D-Waq DIDO, or manually offline

Record description:

A header block containing general information and then for each domain a detailed description.

Keyword	Description
FileInformation	
FileCreatedBy	Version string of the program who generated this file the first time
FileCreationDate	Creation date and time
FileVersion	Version number of <∗.d3d> file
Geometry	
LandBoundaryName	Name of the file with the land boundaries
LandBoundaryFormat	Format of the land boundary file, possible values are: TEKAL, NETCDF or SHAPEFILE. The NetCDF file is according the 'World Vector Shoreline' format
DDBound	
FileDDBound	Name of the file with the domain decomposition boundaries

For each grid

Keyword	Description
Grid	
Type	Format of the grid file, possible values are: RGF, RGF_NETCDF, DFLOW_FM, TELEMAC
FileName	Name of grid file with the geographical co-ordinates
FlowDepth	Name of the file containing the depth values at the cell corners of the grid
Aggregation	Name of the aggregation file

Restriction:

◇ The maximum record length in the file is 132.

Example:

The model friesian_tidal_inlet contains three different subdomains (f01, f02, f03) and the project file has the name <friesian_tidal_inlet.d3d.>

```
[FileInformation]
  FileGeneratedBy  = Deltares, Delft3D-DIDO Version 4.04.00.11836M, Jun 21 2010, 12:09:34
  FileCreationDate = 2010-06-21, 13:35:22
  FileVersion      = 0.03
[DDBound]
  FileDDBound = f34-123.ddb
```

```
[Grid]
  Type             = RGF
  FileName         = f01.grd
  Aggregation      = f34_dd-f01.dwq
  Monitoring Areas = f34_dd-f01.dmo
[Grid]
  Type             = RGF
  FileName         = f02.grd
  Aggregation      = f34_dd-f02.dwq
  Monitoring Areas = f34_dd-f02.dmo
[Grid]
  Type             = RGF
  FileName         = f03.grd
  Aggregation      = f34_dd-f03.dwq
  Monitoring Areas = f34_dd-f03.dmo
```

A.2 Land boundary file

File contents	The co-ordinates of one or more polylines. Each polyline (piecewise linear) is written in a single block of data.
Filetype	ASCII
File format	Free formatted
Filename	<*name*.ldb>
Generated	RGFGRID, QUICKIN, etc

Record description:

Record	**Record description**
	Preceding description records, starting with an asterisk ($*$), and will be ignored.
1	A non blank character string, starting in column one
2	Two integers representing the numbers of rows and number of columns for this block of data
	Two reals representing the x, y or λ, ϕ-co-ordinate

Example:

```
*
* Polyline L007
*
L007
6  2
            132400.0    549045.0
            132345.0    549030.0
            132165.0    549285.0
            131940.0    549550.0
            131820.0    549670.0
            131585.0    549520.0
*
* Polyline L008
*
L008
```

```
4  2
          131595.0    549685.0
          131750.0    549865.0
          131595.0    550025.0
          131415.0    550175.0
*
* Polyline L009
*
L009
6  2
          131595.0    549655.0
          148975.0    564595.0
          150000.0    564935.0
          152105.0    565500.0
          153150.0    566375.0
          154565.0    567735.0
```

Remark:

◇ In case this file is read as a polygon file then the polylines are closed by RGFGRID to get a polygon.

A.3 Sample file

File contents	The location and value of samples.
Filetype	ASCII
File format	Free formatted
Filename	<*name*.xyz>
Generated	Manually or Offline with QUICKIN and data from digitised charts or GIS-database.

Record description:

Filetype	Record description
Free formatted	Location and sample value per row Two reals representing the x, y or λ, ϕ-coordinate and one real representing the sample value

Example:

Sample file with 12 sample values with their location (free formatted file).

```
213813.2     603732.1     -4.053000
214686.0     607226.1     -4.522000
214891.7     610751.2     -5.000000
210330.8     601424.1     -2.169000
211798.0     604444.8     -2.499000
212460.0     607475.7     -2.760000
212436.9     610362.5     -2.865000
185535.4     606607.9      1.360000
186353.0     603789.4      1.122000
187959.2     601197.6     0.9050000
190193.0     599101.5     0.7050000
208578.7     602513.7    -0.7990000
```

A.4 Spline file

File contents	The co-ordinates of one or more polygons. Each polygon is written in a single block of data
Filetype	ASCII
File format	Free formatted
Filename	<*name*.spl>
Generated	Delft3D-RGFRID

Record description:

Record	Record description
	Preceding description records, starting with an asterisk ($*$), and will be ignored.
1	Character string of at least 1 character.
2	Two integers representing the numbers of rows and number of columns for this block of data
	Two reals representing the x, y or λ, ϕ-coordinate

Example:

```
*
* Deltares, \DRGFGRID\ Version 4.16.01.4887, Oct 18 2008, 13:26:48
* File creation date: 2008-10-19, 13:33:05
*
* Coordinate System = Cartesian
*
S001
   6   2
 -1.1520000E+02   9.9630000E+02
  1.2911200E+03   9.9878100E+02
  2.2075800E+03   1.0299500E+03
  3.0180600E+03   1.3105000E+03
  4.1090800E+03   1.3479100E+03
  5.1315300E+03   1.3354400E+03
S002
   2   2
  3.4607000E+03  -6.0347500E+02
  4.0405100E+03   5.7377700E+01
```

A.5 Polygon file

File contents	The co-ordinates of one or more polygons. Each polygon is written in a single block of data
Filetype	ASCII
File format	Free formatted
Filename	<*name*.pol>
Generated	RGFGRID, QUICKIN, D-Waq DIDO, etc

Record description:

The file may contain one or more polygons. For every polygon the file should contain a line indicating the name of the polygon, followed by a line indicating the number of points making up the polygon and the number of coordinates, i.e. 2, finally followed by the coordinate data.

Record	Record description
	Preceding description records, starting with an asterisk ($*$), and will be ignored.
1	A non blank character string, starting in column one
2	Two integers representing the numbers of rows and number of columns for this block of data
	Two reals representing the x, y or λ, ϕ-coordinate

Restriction:
◇ The first record and the last record in the block should be the same

Example:

```
*
* Deltares, Delft3D-DIDO Version 3.39.01.4423:4459, Sep 25 2008, 20:10:54
* 2008-09-25, 22:11:08
*
Observation Area 001
       5   2
   1.8768018E+05   6.1708738E+05
   1.8996981E+05   6.1001035E+05
   1.9746314E+05   6.1266423E+05
   1.9480925E+05   6.1838830E+05
   1.8768018E+05   6.1708738E+05
Observation Area 002
       5   2
   2.0011703E+05   6.1818015E+05
   1.9819166E+05   6.1063479E+05
   2.0568498E+05   6.0870942E+05
   2.0797461E+05   6.1599460E+05
   2.0011703E+05   6.1818015E+05
Observation Area 003
       5   2
   1.9340425E+05   6.1396516E+05
   2.0183425E+05   6.1365294E+05
   1.9944054E+05   6.0558720E+05
   1.9522555E+05   6.0595146E+05
   1.9340425E+05   6.1396516E+05
```

A.6 Orthogonal curvilinear grid file

File contents	The co-ordinates of the orthogonal curvilinear grid at the depth points.
Filetype	ASCII
File format	Free formatted
Filename	<*name*.grd>
Generated	RGFGRID

Record description:

Record	Record description
	Preceding description records, starting with an asterisk ($*$), will be ignored.
1	Record with `Co-ordinate System = Cartesian` or value `Spherical`
2	The number of grid *points* in m- and n-direction (2 integers).
3	Three real values (not used).
4 to K+3	A label and record number, the x-component of the world co-ordinates of all points in m-direction, starting with row 1 to row $nmax$, with as many continuation records as required by $mmax$ and the number of co-ordinates per record. The label and record number are suppressed on the continuation lines. This set of records is repeated for each row until $n = nmax$.
K+4 to 2K+3	A similar set of records for the y-component of the world co-ordinates.

K is the number of records to specify for all grid points a set of x- or y-co-ordinates.

Restrictions:

- ◇ The grid must be orthogonal.
- ◇ Input items in a record are separated by one or more blanks.

Example:

```
*
* Deltares, Delft3D-RGFGRID  Version 4.16.01.4531, Sep 30 2008, 23:32:27
* File creation date: 2008-10-01, 23:19:22
*
Coordinate System = Cartesian
       9       7
 0 0 0
 Eta=    1   0.00000000000000000E+00   1.00000000000000000E+02   2.000000...
             5.00000000000000000E+02   6.00000000000000000E+02   7.000000...
 Eta=    2   0.00000000000000000E+00   1.00000000000000000E+02   2.000000...
             5.00000000000000000E+02   6.00000000000000000E+02   7.000000...
 Eta=    3   0.00000000000000000E+00   1.00000000000000000E+02   2.000000...
             5.00000000000000000E+02   6.00000000000000000E+02   7.000000...
 Eta=    4   0.00000000000000000E+00   1.00000000000000000E+02   2.000000...
```

```
                5.00000000000000000E+02   6.00000000000000000E+02   7.000000...
 Eta=     5     0.00000000000000000E+00   1.00000000000000000E+02   2.000000...
                5.00000000000000000E+02   6.00000000000000000E+02   7.000000...
 Eta=     6     0.00000000000000000E+00   1.00000000000000000E+02   2.000000...
                5.00000000000000000E+02   6.00000000000000000E+02   7.000000...
 Eta=     7     0.00000000000000000E+00   1.00000000000000000E+02   2.000000...
                5.00000000000000000E+02   6.00000000000000000E+02   7.000000...
 Eta=     1     1.00000000000000000E+02   1.00000000000000000E+02   1.000000...
                1.00000000000000000E+02   1.00000000000000000E+02   1.000000...
 Eta=     2     2.00000000000000000E+02   2.00000000000000000E+02   2.000000...
                2.00000000000000000E+02   2.00000000000000000E+02   2.000000...
 Eta=     3     3.00000000000000000E+02   3.00000000000000000E+02   3.000000...
                3.00000000000000000E+02   3.00000000000000000E+02   3.000000...
 Eta=     4     4.00000000000000000E+02   4.00000000000000000E+02   4.000000...
                4.00000000000000000E+02   4.00000000000000000E+02   4.000000...
 Eta=     5     5.00000000000000000E+02   5.00000000000000000E+02   5.000000...
                5.00000000000000000E+02   5.00000000000000000E+02   5.000000...
 Eta=     6     6.00000000000000000E+02   6.00000000000000000E+02   6.000000...
                6.00000000000000000E+02   6.00000000000000000E+02   6.000000...
 Eta=     7     7.00000000000000000E+02   7.00000000000000000E+02   7.000000...
                7.00000000000000000E+02   7.00000000000000000E+02   7.000000...
```

A.7 Grid enclosure file

File contents	The indices of the external computational grid enclosure(s) and optionally one or more internal computational grid enclosures that outlines the active computational points in a Delft3D-FLOW computation. The file is strongly related to the curvilinear grid file.
Filetype	ASCII
File format	Free formatted
Filename	<*name*.enc>
Generated	RGFGRID

Record description:

Record	**Record description**
All	One pair of M and N indices representing the grid co-ordinates where a line segment of the computational grid enclosure (polygon) changes direction.

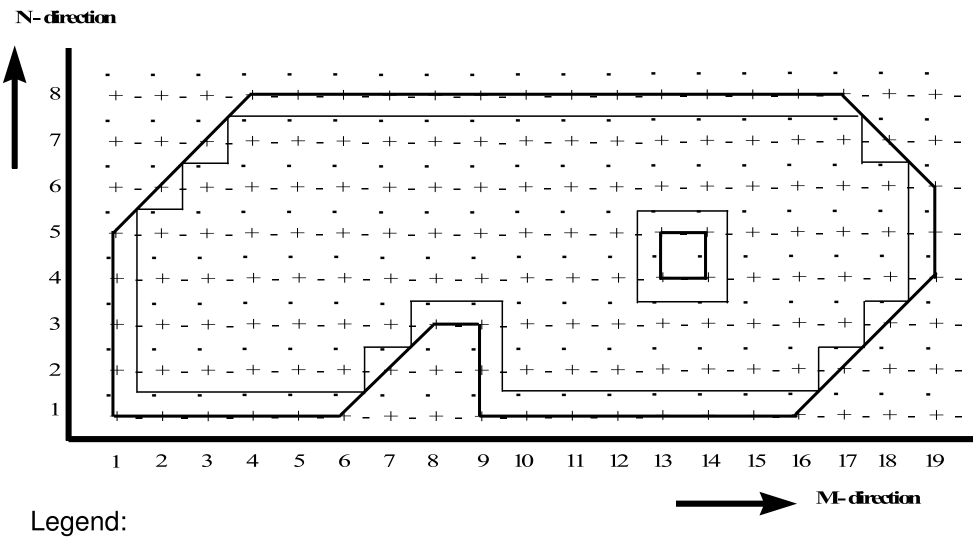
Restrictions:

- ◇ A polygon must be closed. The first point of the polygon is repeated as last point.
- ◇ A line segment may not intersect or touch any other line segment.
- ◇ The angle formed by consecutive line segments (measured counter clock-wise) can have a value of: 45, 90, 135, 225, 270 or 315 degrees, but not 0, 180 and 360 degrees.
- ◇ In a row or column there should be at least two active computational grid cells.
- ◇ Input items in a record are separated by one or more blanks.

Example:

Model area with (one) external and one internal polygon, see Figure A.1.

```
 1  1   begin external polygon
```



Legend:

$+$	water level point
$\vert$	v-velocity point
$-$	u-velocity point
Full thick line	grid enclosure and (for the external polygon only) location of water level open boundaries.
Full thin line	location for velocity or discharge open boundaries.

Figure A.1: *Example of computational grid enclosures*

```
 6  1
 8  3
 9  3
 9  1
16  1
19  4
19  6
17  8
 4  8
 1  5
 1  1   end external polygon
13  4   begin internal polygon
14  4
14  5
13  5
13  4   end internal polygon
```

A.8 Annotation file

File contents	File with x and y co-ordinates, string and rgb-colour.
Filetype	ASCII
File format	Free formatted.
Filename	<*name*.txt>
Generated	manually offline

Record description:

Record	Record Description
1	Records starting with a $*$ are comment lines Character string to define the datablock (nonblank)
2	Number of rows
3–N	real, real, string, integer: geographical co-ordinates (2 reals), text between quotes which need to be plotted (string) and rgb-colour (integer; $= 256 * 256 * r + 256 * g + b$)

Restriction:

◇ The maximum record length in the file is 132.

Example:

```
*
BL01
8 4
10.0  10.0   'string-01'  6553625
10.0  20.0   'string-02'  9830425
20.0  10.0   'string-03' 13120000
20.0  20.0   'string-04' 16724480
30.0  10.0   'string-05'    38425
30.0  20.0   'string-06'    65305
40.0  10.0   'string-07'      255
40.0  20.0   'string-08'        0
```

A.9 DD Boundary file

File contents	Domain decomposition boundaries connecting two grids for the prescribed indices.
Filetype	ASCII
File format	Fix formatted.
Filename	<*name*.ddb>
Generated	RGFGRID, or manually offline

Record description:

Record	Record Description
N	Name of the first grid, followed by four integers indicating the gridline on which the boundary lies, followed by the name of the second grid and four integers indicating the gridline on which the boundary lies.

Restrictions:

- ◇ No space allowed in grid filename.
- ◇ The maximum record length in the file is 132.

Example:

In the following example 4 sub-domains exist. Domain d01_ns is coupled to oa1_ns, ob1_ns and oc1_ns. Furthermore oa1_ns is coupled to ob1_ns, and ob1_ns to oc1_ns.

```
d01_ns.grd            5   16    5    1  oa1_ns.grd           28   35   28   20
d01_ns.grd          245    1    5    1  ob1_ns.grd           17   21    1   21
d01_ns.grd          245   52  245    1  oc1_ns.grd            1   44    1   27
ob1_ns.grd            1    4    1   21  oa1_ns.grd           28    3   28   20
ob1_ns.grd           17    4   17   21  oc1_ns.grd            1   10    1   27
```

A.10 Colour scheme file

File contents	The colour scheme
Filetype	ASCII
File format	Free formatted
Filename	<*name*.clr> or <*name*.clrmap>
Generated	manually

Record description:

Record	Record description
1	COLORMAP
2	NAME=name
3	SPACE=RGB, RGB is the only allowed space for this program
4 – N	one real and three integers.

The first column represent the relative distribution of the defined colours in column 2–4 (representing the RGB values).

Example:

```
COLORMAP
NAME=copper
SPACE=RGB
0.0000   0   0   0
0.8000 255 159 101
```

```
1.0000 255 199 127
```

A.11 Settings file

File contents	Settings of the program
Filetype	ASCII
File format	Fix formatted
Filename	<*name*.ini>
Generated	By the program

Record description:

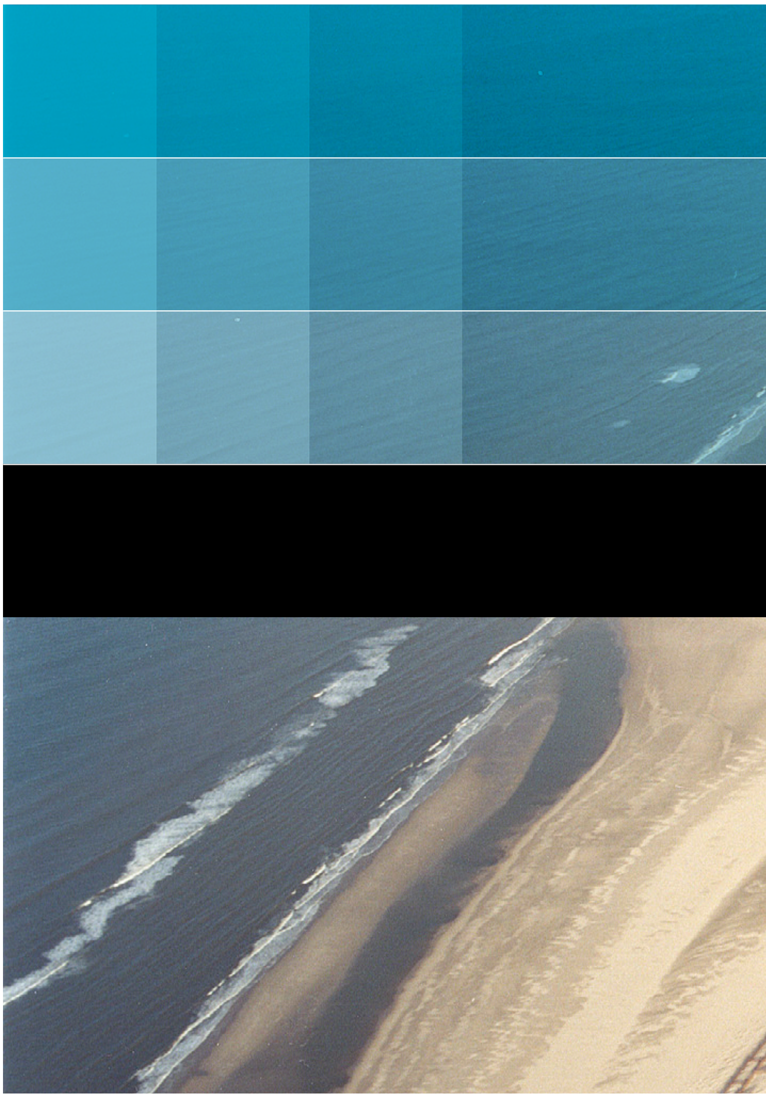
Record	Record description
FileInformation	
FileCreatedBy	RGFGRID version number
FileCreationDate	creation date and time
FileVersion	version number of <∗.ini> file
RGFParameter	
name	integer value
TextSettings	
name	value (integer or real)
RGFsettings	
name	value (integer or real)
DepthDesign	
name	value (integer or real)
Colours	
name	RGB value (3 integers) line width dots sizes

Example:

```
[FileInformation]
  FileGeneratedBy  = Deltares, Delft3D-RGFGRID Version 4.20.00.11763:11790M, Jun 16 2010, 14:21:46
  FileCreationDate = 2010-06-16, 14:23:25
  FileVersion      = 0.02
[RGFParameter]
  AutoscaleLegend                 =   1
  XCoorLegend                     =  16
```

```
  YCoorLegend                      =  20
[TextSettings]
  Line1                            =
  Line2                            =
  Line3                            =
  Fontsize                         =   3.00000000000000000E+00
  Xposition                        =   0.00000000000000000E+00
  Yposition                        =   0.00000000000000000E+00
  FontsizeTimeDate                 =   3.00000000000000000E+00
  XposTimeDate                     =   0.00000000000000000E+00
  YposTimeDate                     =   3.00000000000000000E+00
[RGFSettings]
  MGridCells                       =  50
  NGridCells                       =  50
  DeltaX                           =   1.00000000000000000E+02
  DeltaY                           =   1.00000000000000000E+02
  OriginX                          =   0.00000000000000000E+00
  OriginY                          =   1.00000000000000000E+02
  RotationLeft                     =   0.00000000000000000E+00
  RadiusCurvatureM                 =   0.00000000000000000E+00
  MFraction                        =   1.00000000000000000E+00
  MaximumSizeUniformMSize          =   1.00000000000000000E+00
  NFraction                        =   1.00000000000000000E+00
  MaximumSizeUniformNSize          =   1.00000000000000000E+00
  SphereMGridCells                 =  50
  SphereNGridCells                 =  50
  SphereDeltaX                     =   1.00000000000000002E-02
  SphereDeltaY                     =   1.00000000000000002E-02
  SphereOriginX                    =   4.38082999999999956E+00
  SphereOriginY                    =   5.19858300000000000E+01
  SphereRotationLeft               =   0.00000000000000000E+00
  SphereRadiusCurvatureM           =   0.00000000000000000E+00
  SphereMFraction                  =   1.00000000000000000E+00
  SphereMaximumSizeUniformMSize    =   1.00000000000000000E+00
  SphereNFraction                  =   1.00000000000000000E+00
  SphereMaximumSizeUniformNSize    =   1.00000000000000000E+00
  StayOnStartupDirectory           =   0
  MRefinementFactor                =   3
  NRefinementFactor                =   3
  NrSmoothingIterations            =  20
  SmoothingParameter               =   2.00000000000000011E-01
  AttractionRepulsionParameter     =   1.00000000000000006E-01
  ActiveInactivePaste              =   5.00000000000000000E-01
  LineOrSplineRepresentation       =   1.00000000000000000E+00
  EquidistantSmoothInterpolation   =   1.00000000000000000E+00
  IncreaseFactorLineMirror         =   1.00000000000000000E+00
  IterationsAttractionParameter    =   3
  IterationsBoundary               =  15
  IterationsInnerArea              =  25
  InfluenceOriginalGridShape       =   1.00000000000000000E+00
  PositionBoundaryPoints           =   1.00000000000000000E+00
  DesignMethod                     =   1
[DepthDesign]
  DepthDesignSizeRatioM            =   2.00000000000000011E-01
  DepthDesignSizeRatioN            =   2.00000000000000011E-01
  DepthDesignDepthVsSlope          =   1.00000000000000000E+00
  DepthDesignNrSmoothingIterations =  10
  DepthDesignSmoothingFactor       =   1.00000000000000006E-01
  DepthDesignFieldVsLineWeightM    =   0.00000000000000000E+00
  DepthDesignFieldVsLineWeightN    =   0.00000000000000000E+00
[Colours]
  ColourBackground                 = 255 255 200
```

```
  LegendColourBackground          = 255 255 255
  lineColourText                  = 000 255 255
  lineColourLandBoundary          = 132 066 000
  fillColourLandBoundary          = 255 182 108
  lineColourSplines               = 000 255 000
  lineColourPolyline              = 000 255 000
  fillColourPolyline              = 000 255 000
  lineColourPolygon               = 170 000 127
  lineColourActiveGrid            = 000 000 255
  lineColourPreviousGrid          = 211 000 000
  lineColourActiveCmpBnd          = 005 005 005
  lineColourActiveOpenBnd         = 000 000 255
  lineColourActiveDDBnd           = 176 000 176
  lineColourGrid                  = 192 192 192
  lineColourCmpBnd                = 050 050 050
  lineColourOpenBnd               = 000 000 150
  lineColourDDBnd                 = 176 000 255
[Width]
  lineWidthLandBoundary           =   1
  lineWidthPolyline               =   1
  lineWidthPolygon                =   1
  DotSizeSamples                  =   2
  lineWidthActiveGrid             =   1
  lineWidthActiveCmpBnd           =   1
  lineWidthActiveOpenBnd          =   1
  lineWidthActiveDDBnd            =   3
  lineWidthGrid                   =   1
  lineWidthCmpBnd                 =   1
  lineWidthOpenBnd                =   1
  lineWidthDDBnd                  =   3
[Caches]
  splines                         =   0
  rest                            =   0
  polygons                        =   0
  polylines                       =   0
  gridprop                        =   0
  gridadm                         =   0
  gridprev                        =   2
  cmpbound                        =   1
  openbound                       =   1
  ddbound                         =   1
  actgrid                         =   2
  inactgrid                       =   3
  landboundary                    =   4
  samples                         =   5
```



Deltares systems

PO Box 177
2600 MH Delft
Rotterdamseweg 185
2629 HD Delft
The Netherlands

+31 (0)88 335 81 88
sales@deltaressystems.nl
www.deltaressystems.nl

3D/2D modelling suite for integral water solutions

Delft3D

Deltares systems

QUICKIN

Deltares

Enabling Delta Life

User Manual

QUICKIN

Generation and manipulation of grid-related parameters such as bathymetry, initial conditions and roughness

User Manual

Hydro-Morphodynamics & Water Quality

Version: 4.00.30932

28 April 2014

QUICKIN, User Manual

Published and printed by:
Deltares
Boussinesqweg 1
2629 HV Delft
P.O. Box 177
2600 MH Delft
The Netherlands

telephone: +31 88 335 82 73
fax: +31 88 335 85 82
e-mail: info@deltares.nl
www: http://www.deltares.nl

For sales contact:
telephone: +31 88 335 81 88
fax: +31 88 335 81 11
e-mail: sales@deltaressystems.nl
www: http://www.deltaressystems.nl

For support contact:
telephone: +31 88 335 81 00
fax: +31 88 335 81 11
e-mail: support@deltaressystems.nl
www: http://www.deltaressystems.nl

Contents

List of Figures

1 Guide to this manual

1.1 Introduction

This User Manual concerns the data interpolation module, QUICKIN, of the Delft3D software suite. To make this manual more accessible we will briefly describe the contents of each chapter and appendix.

If this is your first time to start working with QUICKIN module we suggest you to read and practice the getting started of Chapter 3 and the tutorial of Chapter 6. These chapters explain the user interface options and guide you through the definition of your first data interpolation to a curvilinear grid.

Chapter 2: Introduction to QUICKIN, provides specifications of QUICKIN and the areas of applications.

Chapter 3: Getting started, explains the use of the overall menu program, which gives access to all Delft3D modules and to the pre- and post-processing tools. Last but not least you will get a first introduction into the QUICKIN Graphical User Interface, used to define data on of grids vertices or grid cells which can be used in a hydrodynamic, wave or water quality simulation.

Chapter 4: General operation, provides practical information on the general operation of the QUICKIN module.

Chapter 5: Menu options, provides a description of all menu and toolbar options.

Chapter 6: Tutorial, emphasis at giving you some first hands-on experience in using the QUICKIN module to define the input of a simple problem.

References, provides a list of publications and related material on the QUICKIN module.

Appendix A: Files of QUICKIN, gives a description of the files that can be used in QUICKIN as input or output. Generally, these files are generated by QUICKIN or other modules of the Delft3D suite and you need not to be concerned about their internal details. However, in certain cases it can be useful to know these details, for instance to generate them by means of other utility programs.

1.2 Name and specifications of the program

Title	QUICKIN
Description	QUICKIN is a program for the generation, interpolation or manipulation of space varying quantities such as bathymetries, initial conditions or parameter fields on Cartesian or spherical (curvilinear) grids e.g. created with RGFGRID (RGFGRID (2013)). Bathymetries can be generated for both the hydrodynamic simulation programs Delft3D-FLOW (Delft3D-FLOW (2013)) and Delft1D2D (Delft1D2D (2002)), QUICKIN can display animations of flooding files generated with Delft1D2D.
Special facilities	various interpolation methods

user selectable 'area of influence' by polygon
various viewing options
various depth manipulation options
various sample manipulation options
functionality: dredge and dump areas
new functionality: multiple grids and depths

1.3 Manual version and revisions

A manual applies to a certain release of the related numerical program. This manual applies to QUICKIN, version 4.17.00.

1.4 Typographical conventions

Throughout this manual, the following conventions help you to distinguish between different elements of text to help you learn about QUICKIN.

Example	Description
Waves **Boundaries**	Title of a window or sub-window. Sub-windows are displayed in the **Module** window and cannot be moved. Windows can be moved independently from the **Module** window, such as the **Visualisation Area** window.
Save	Item from a menu, title of a push button or the name of a user interface input field. Upon selecting this item (click or in some cases double click with the left mouse button on it) a related action will be executed; in most cases it will result in displaying some other (sub-)window. In case of an input field you are supposed to enter input data of the required format and in the required domain.
<\tutorial\wave\swan-curvi> <siu.mdw>	Directory names, filenames, and path names are expressed between angle brackets, <>. For the Linux and UNIX environment a forward slash (/) is used instead of the backward slash (\) for PCs.
"27 08 1999"	Data to be typed by you into the input fields are displayed between double quotes. Selections of menu items, option boxes etc. are described as such: for instance 'select *Save* and go to the next window'.
`delft3d-menu`	Commands to be typed by you are given in the font Courier New, 10 points.
➢	User actions are indicated with this arrow.

Example	Description
[m/s] [-]	Units are given between square brackets when used next to the formulae. Leaving them out might result in misinterpretation.

1.5 Changes with respect to previous versions

Version	Description
4.00.00	Complete new version of QUICKIN

2 Introduction to QUICKIN

2.1 Introduction

The main purpose of the QUICKIN program is to create, manipulate and visualise model bathymetries for the Delft3D modules FLOW and WAVE (2 and 3 dimensional flow and transport and 2D short wave propagation) and for the programs Delft1D2D (flood and overland flow); Delft1D2D (2002), SWAN; SWAN (2000). Initial condition fields or parameter fields can also be specified.

Bathymetries for Delft3D-WAVE and SWAN can be defined on a Cartesian or spherical (curvilinear) grid, a bathymetry for Delft1D2D is defined on a rectangular grid.

The grids can be created using the RGFGRID program.

2.2 Program considerations

One of the problems of depth interpolation is that the samples (raw data) may originate from various sources, each of different date, quality and resolution. If these samples are all copied into one large file, the 'high' quality data would be contaminated with 'low' quality data, leading to non optimal interpolation results.

In order to cope with this problem of 'data contamination', the program allows for subsequent opening and interpolation of files. Thus, starting with the best data available, the optimal bathymetry is obtained, because points that have already been interpolated with 'good' quality data, are not overwritten in subsequent steps by interpolation with 'low' quality data.

Since the Delft3D-FLOW and SWAN programs use equations that in fact are averaged over the grid dimensions, the best results are obtained if the model bathymetry approximates the real bathymetry in an averaged sense rather than in a local sense. Therefore, if the sample resolution is higher than the grid resolution, we advise to assign depth values by means of an averaging method. Instead, if there are less sample points than grid points, the depth at grid points has to be interpolated (e. g. with a triangulation interpolation method).

Since the preferred interpolation method may vary over the model area, the QUICKIN program allows you to interactively select the active interpolation area, by means of a polygon, and the desired interpolation method. Also, the classic problem of triangular interpolation over areas that should not be covered by the data, but that may be inside the triangles spanned by the triangle network, can effectively be avoided by means of a polygon. Typically, this kind of situation occurs when a tidal flat is surrounded by channels which have been measured by boat, whereas the flat itself has not been surveyed.

Furthermore, a bathymetry can be created or modified without samples data, by changing grid depth values interactively of by specifying sample data using a polygon. Typically, this is of interest when future planned channels have to be built into an existing model bathymetry. A powerful so-called 'depth diffusion' mechanism is provided, that propagates user-supplied depth information at selected grid points or lines into the model, keeping a smooth transition between the original and the constructed bathymetry. Furthermore, if desired, the bathymetry can be smoothed by the depth smoothing option.

3D viewing is implemented to provide a clear view of bathymetries.

3 Getting started

3.1 Overview of Delft3D

The Delft3D program suite is composed of a set of modules (components) each of which covers a certain range of aspects of a research or engineering problem. Each module can be executed independently or in combination with one or more other modules.

Delft3D is provided with a menu shell through which you can access the various modules. In this chapter we will guide you through some of the input screens to get the look-and-feel of the program. In the Tutorial, Chapter 6, you will learn to define a simple scenario.

3.2 Starting Delft3D

To start Delft3D:

◇ On an MS Windows platform: select *Delft3D* in the *Programs* menu.
◇ On Linux machines: type `delft3d-menu` on the command line.

Next the title window of Delft3D is displayed, Figure 3.1.

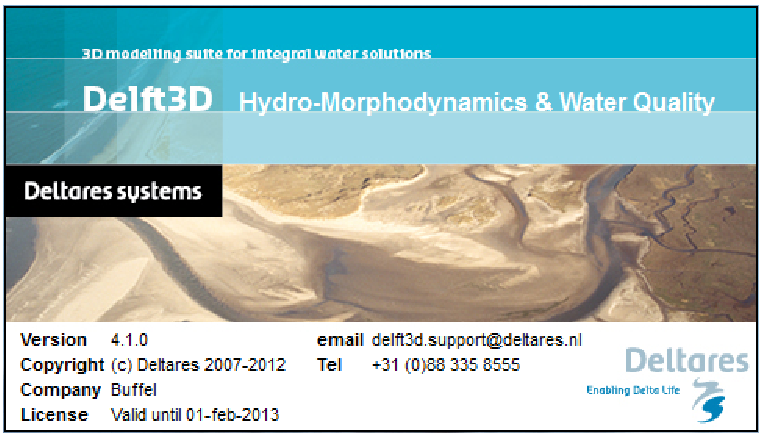


Figure 3.1: *Title window of Delft3D*

After a short while the main window of the Delft3D-MENU appears, Figure 3.2.

Several menu options are shown. For now, only concentrate on exiting Delft3D-MENU, hence:

◇ Click on the *Exit* push button.

The window will be closed and you are back in the Windows Desktop screen for PCs or on the command line for Linux workstations.

Remark:

◇ In this and the following chapters several windows are shown to illustrate the presentation of Delft3D-MENU and QUICKIN. These windows are grabbed from the PC-platform. For Linux workstation the content of the windows is the same, but the colours may be different.

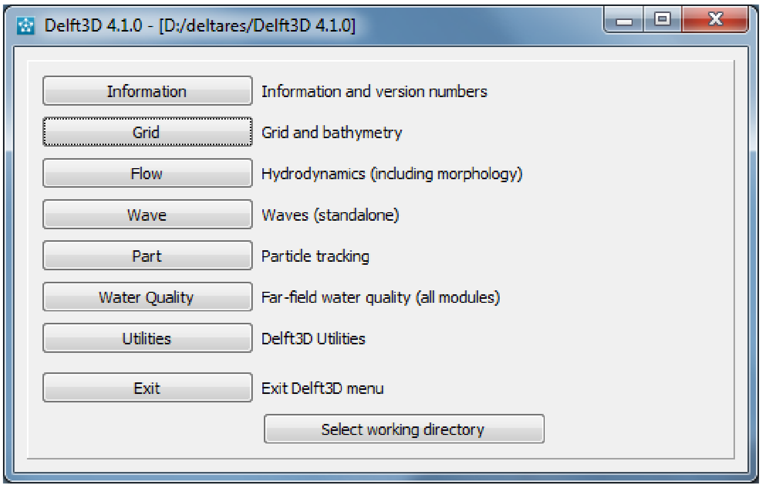


Figure 3.2: *Main window* ***Delft3D-MENU***

3.3 Getting into QUICKIN

To continue start the menu program again, as indicated in section 3.2.

◇ Click the *Grid* button, see Figure 3.2

Next the selection window for **Grid and bathymetry** is displayed for preparing a curvilinear grid, interpolate data on that grid and aggregate the hydrodynamic cells, see Figure 3.3.

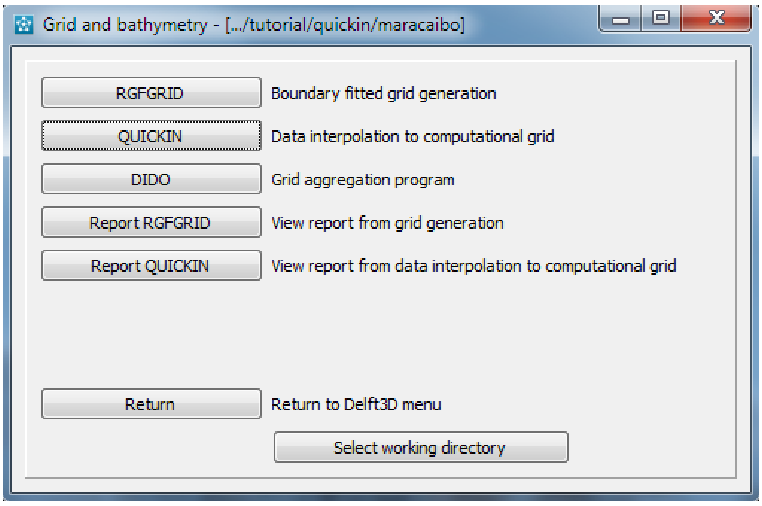


Figure 3.3: *Selection window for* ***Grid and Bathymetry***

Note that in the title bar the current directory is displayed, in our case <D:/delft3d>.

Before continuing with any of the selections of this **Grid and bathymetry** window, you select the directory in which you are going to prepare scenarios and execute computations:

◇ Click the *Select working directory* button.

Next the **Select working directory** window is displayed, see Figure 3.4 (your current directory may differ, depending on the location of your Delft3D installation).

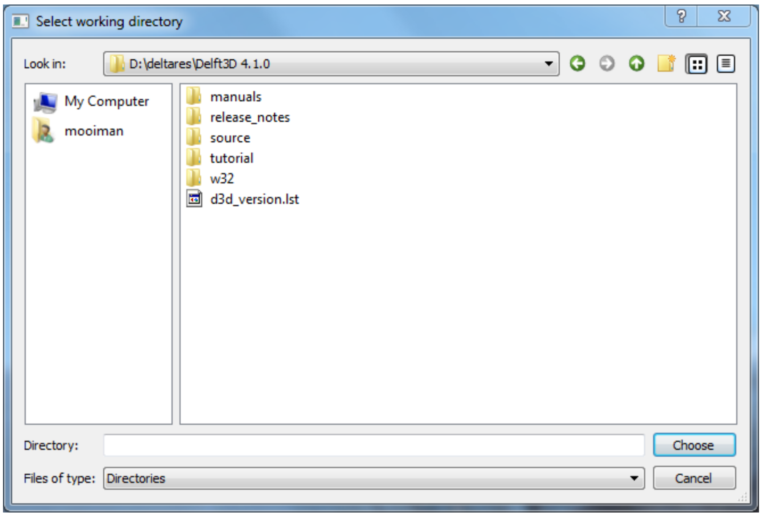


***Figure 3.4: Select working directory** window*

- ◇ Browse to and open the <tutorial> sub-directory of your Delft3D Home-directory.
- ◇ Open the <quickin> directory.
- ◇ Enter the <maracaibo> sub-directory and close the **Select working directory** window by clicking button *OK*, see Figure 3.5.

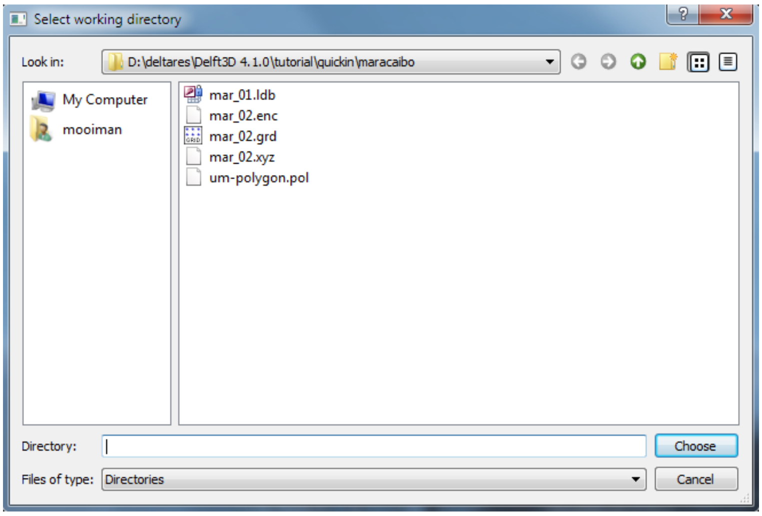


***Figure 3.5: Select working directory** window to set the working directory to <quickin/maracaibo>*

Next the **Grid and bathymetry** window is re-displayed, but now the changed current working directory is displayed in the title bar, see Figure 3.6.

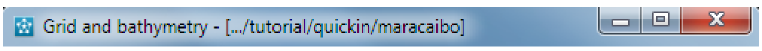


***Figure 3.6:** A part of the current working directory is shown in the title bar due to its length*

Remark:

- ◇ In case you want to start a new project for which no directory exists yet, you can select  in the **Select working directory** window to create a new folder.

- ◇ Click on *QUICKIN* in the **Grid and bathymetry** window, see Figure 3.3.

QUICKIN is loaded and the primary input screen is opened, Figure 3.7.

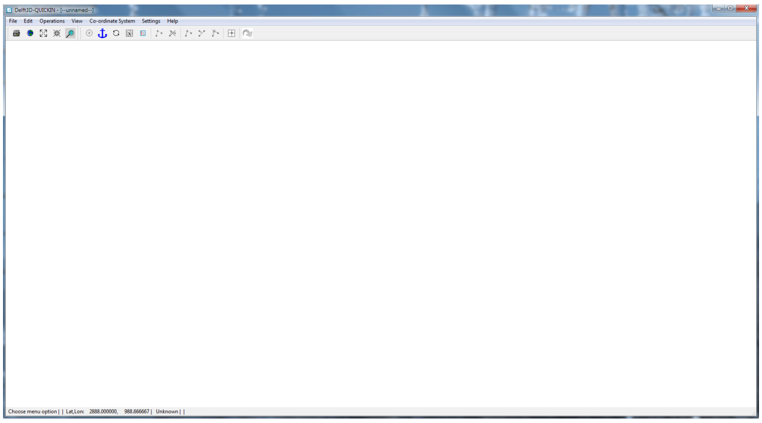


Figure 3.7: *Main window of the QUICKIN*

In the lower-left corner of the status bar QUICKIN gives additional operational information, see Figure 3.8, such as:

- ◇ User selections.
- ◇ Operational instructions (for instance Toggle anchor mode).
- ◇ x and y co-ordinates of the current cursor position.
- ◇ Co-ordinate system: Cartesian or Spherical.
- ◇ Distance (in metre) to a user-defined anchor point (only displayed when the anchor is activated).

Figure 3.8: *Operational information displayed in the statusbar*

3.4 Exploring some menu options

To open a land boundary:

- ◇ Upon selecting *File* → *Attribute Files* → *Open Land Boundary*, you can open a collection of land boundaries, see Figure 3.9. Land boundaries (or land-water marking) are in files with default mask <∗.ldb>.

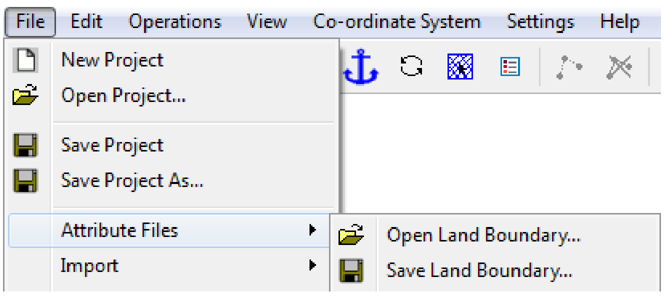


Figure 3.9: *Menu item* File → Attribute Files → Open Land Boundary

Next the **Open Land Boundary** window is displayed, see Figure 3.10.

In the current directory one land boundary file is present.

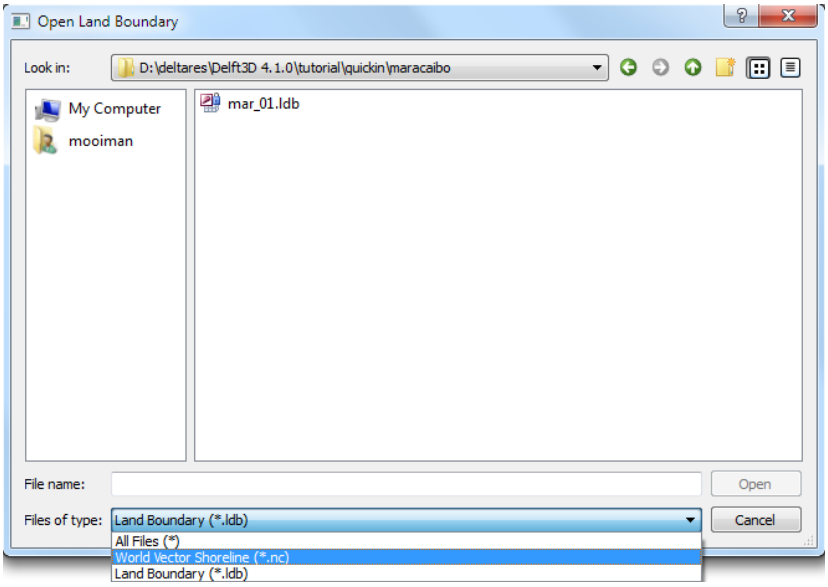


Figure 3.10: *File open window **Open Land Boundary***

◇ Select <mar_01.ldb> and click *Open* to open the land boundary file.

Similar open the grid <mar_02.grd> (on the *File* $rightarrow$ *Import* $rightarrow$ *Grid (RGF-GRID)*) and the samples in <mar_02.xyz> (on the *File* $rightarrow$ *Attribute Files* $rightarrow$ *Open Samples*). After opening these data files the screen looks like as in Figure 3.11.

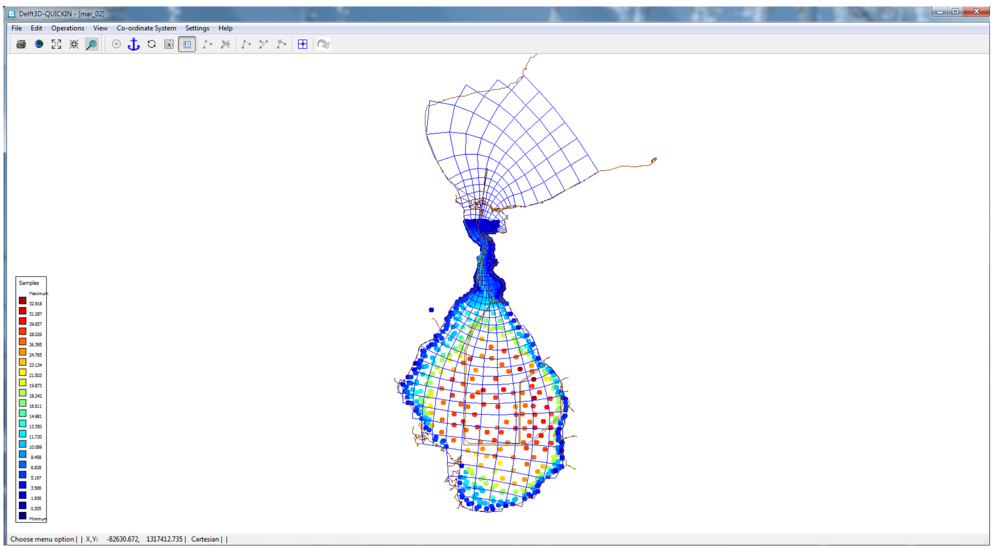


Figure 3.11: *Land boundary, grid and samples for the Maracaibo model*

Since the resolution of samples in the lake is low compared to the grid points, we will use a triangular interpolation to interpolate towards the grid points. First define a polygon

◇ On the menuEdit menu click *Polygon* or click  on the toolbar.
◇ Click with the left mouse to obtain a polygon similar as in Figure 3.12.

To activate the interpolation operation:

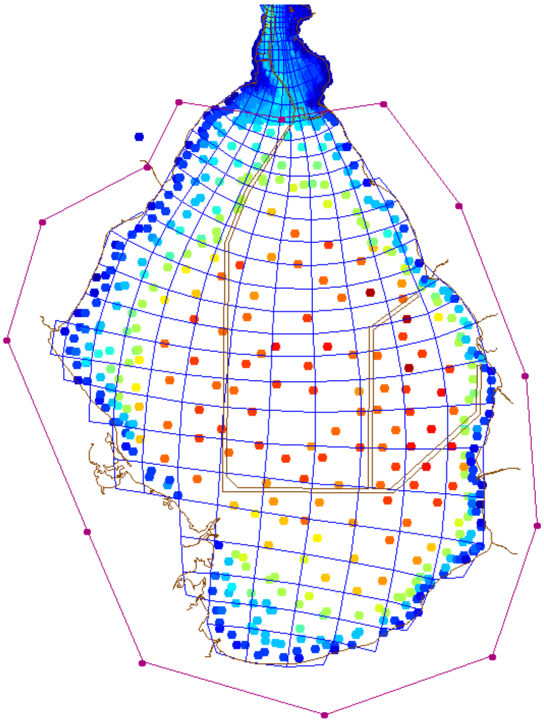


Figure 3.12: *Polygon to enclose the area for triangular interpolation*

◇ On the *Operations* menu, click *Triangular Interpolation*.

Since the samples often do not cover the shore region on land, the grid points at the land-water interface cannot get a depth value from the triangular interpolation.

◇ On the *Operations* menu, click *Grid Cell Averaging*, to obtain Figure 3.13.

We will now finish generating the depth for the Maracaibo model. Refer to Chapter 6, Tutorial, to finalise the depth.

To save the unfinished depth:

◇ On the *File* menu, point to *Export* and click *Depth*

The **Save As** window opens, see Figure 3.14.

You will be back in the main window of QUICKIN.

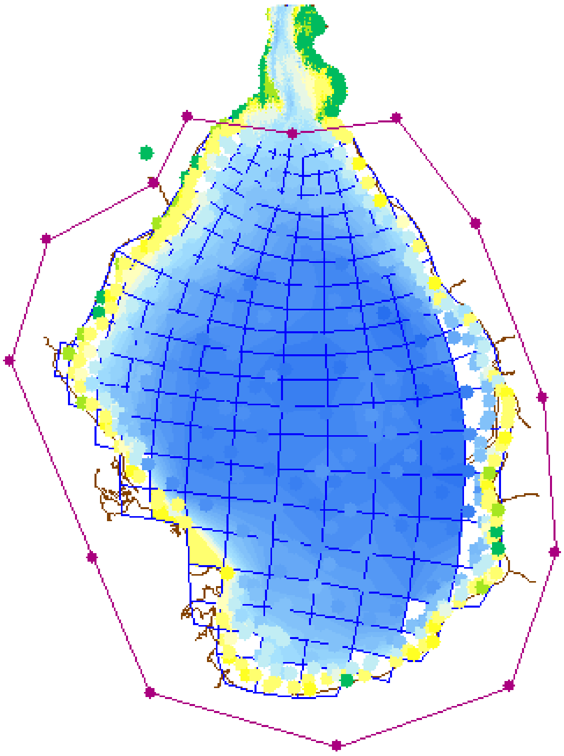


Figure 3.13: *Depth contours in the lake region after triangulation and averaging*

3.5 Exiting QUICKIN

To exit the QUICKIN

◇ Click *Exit* on the *File* menu.

You will be back in the **Grid and bathymetry** window, see Figure 3.3

◇ Click *Return* to return to the main window of Delft3D-MENU, see Figure 3.2
◇ Click *Exit*.

The window is closed and the control is returned to the desk top or the command line.

In this Getting Started session you have learned to access the QUICKIN and to open and inspect a bathymetry samples file.

We encourage new users next to run the tutorial described in Chapter 6.

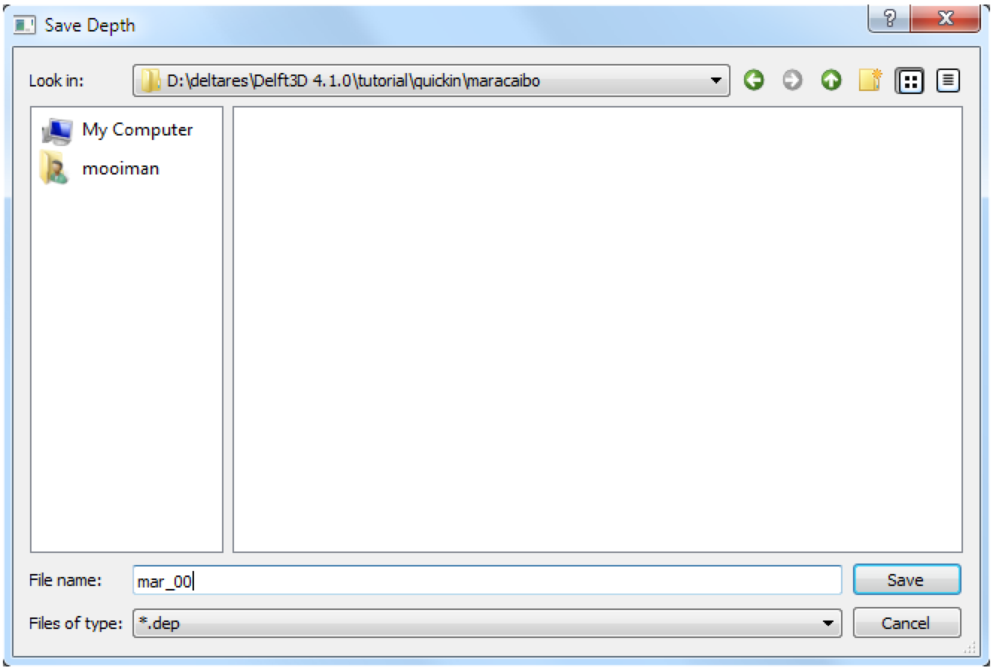


Figure 3.14: *Window* ***Save Depth*** *to save depth files*

4 General operation

4.1 General program operation instruction

The main menu bar is positioned at the top of the screen.

Help

Upon selecting *Help* → *User Manual*, the QUICKIN User Manual in PDF-format will be opened. Use the bookmarks in the contents to locate the subject you are interested in.

Print screen

Press `Ctrl-P` or click on the toolbar to obtain the print window for a hardcopy of the current screen.

File menu

The file-menu is the standard **Open** and **Save As** window. The file mask depends on the type of data that you want to open or save. You can change the directory by navigating through the folders.

It is possible to specify whether to Stay on the Start-up Directory or not, in the **Settings General** form.

General cursor and keyboard functions

The left mouse activates or confirms desired actions. The `Esc` key cancels the last edit action. The right mouse may also confirm actions, or may put the program back into its original mode. In many situations, the `Ins` and `Enter` keys on the right-hand bottom of a keyboard function in the same way as the left and right mouse, respectively.

4.2 Key stroke functions

`A` = Anchor, or on toolbar

When clicking on the toolbar and next pressing the `A` key on the keyboard, a so-called anchor will appear, which acts as zero-distance point. The distance (in metre) of the present cursor position to this point is displayed in the status bar at the right of the co-ordinate system indicator, see Figure 4.1. Moving the cursor around and pressing `A` again will relocate the anchor. Clicking again will de-activate the anchor.

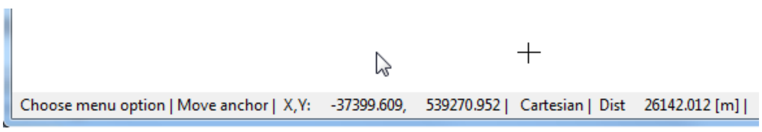


Figure 4.1: *Location of anchor + and distance between anchor and cursor at the right*

`C` = Change

In *Edit* → *Samples,* pressing `C` allows you to change the value of individual samples.

`D` = Delete
In the *Edit* → *Polygon, Depth, Samples* options, pressing `D` allows you to delete individual points (polygon, depth or sample).

`E` = Erase polygon
In *Edit* → *Polygon*, keeping `E` pressed allows you to delete the indicated polygon.

`I` = Insert
In *Edit* → *Polygon*, pressing `I` when the cursor is not on a polygon point allows you to start a new polygon. When the cursor is on a polygon point, pressing `I` allows you to insert polygon points between two points of the indicated polygon.

In *Edit* → *Samples*, pressing `I` allows you to create samples.

`Ctrl-P` = Print screen
Pressing `Ctrl-P` will open the print window. The current screen will be printed to your printer or to a file.

`R` = Replace
In *Edit* → *Polygon* or *Edit* → *Observation Area*, pressing `R` allows you to replace (move) individual points.

`Mouse wheel`
Use the mouse wheel to zoom in and zoom out. Other ways are:

Click  on the toolbar to zoom in.

Click  on the toolbar to zoom out.

To define a zoom box, click  on the toolbar and drag a box.

`Ctrl +` = Zoom in
Keep the `Ctrl`-key pressed and use the + key to zoom in more.

`Ctrl -` = Zoom out
Keep the `Ctrl`-key pressed and use the `-` key to zoom in more.

`Ctrl` move cursor = move focus of screen
Keep the `Ctrl`-key pressed and move the cursor around. The current screen will move accordingly.

`Ctrl` arrow keys = move focus of screen left, right, up or down
Keep the `Ctrl`-key pressed and use the arrow keys to move the focus of the screen accordingly.

`Esc` = Undo
In various edit modes the latest action will be undone pressing `Esc` .

`Tab` = depth cursor

In menu *Edit* → *Depth* and menu item *Depth Linear,* pressing the `Tab` key while the cursor is on a grid point, will display the depth value in the lower status bar. In normal view (not in *3D View*), you can press the `Tab` key to go into 'depth cursor mode'. This allows you to 'walk' over the depth points using the arrow keys. When in 'depth cursor mode' you can increase or decrease the depth values by using the + and - keys, respectively.

5 Menu options

The menu bar contains the following items, see Figure 5.1, each item is discussed in a separate sections

Figure 5.1: *QUICKIN menu options*

5.1 File menu

When opening a land boundary or polygon before opening the grid file be sure you choose the same co-ordinate system as the grid file you want to load, see section 5.5.

- ◇ When opening files, QUICKIN will not check the co-ordinate system in the file against the current co-ordinate system in QUICKIN, except when opening a grid.

On the *File* menu, see Figure 5.2, options are available to import land boundaries, samples, depth files and grid files. The results at each stage of the bathymetry definition process can be saved. The option to quit QUICKIN is located here also.

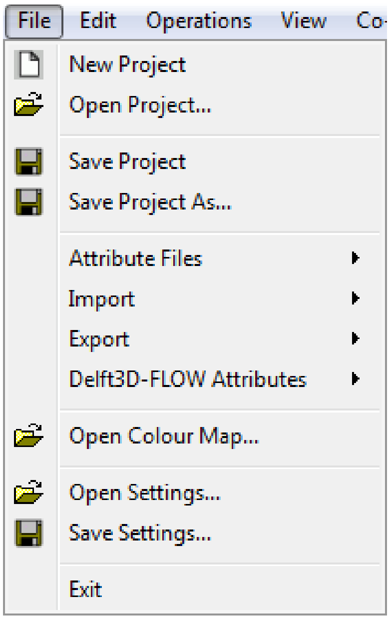


Figure 5.2: *Options on the* File *menu*

The start-up directory to open and save files can be configured in the **General Parameters** form on the menu *Settings* → *General*. As default the file menu starts at the last directory selected.

For the formats of the files you are referred to Appendix A.

5.1.1 New project

Upon selecting *File* → *New Project*, all objects (land boundaries, polygons, grids, depths, samples etc.) will be deleted; i.e. you start from scratch.

5.1.2 Open project

Upon selecting *File* → *Open Project,* the **Open Project** window appears in which you can browse to an existing project (<∗.d3d> file).

Remark:
◇ A project saved by RGFGRID or D-Waq DIDO can be read by QUICKIN.

5.1.3 Save project

Upon selecting *File* → *Save Project*, the current project (filenames for grid and depth) will be saved under the same name. If the project name is not known yet, the **Save Project** window appears.

Remark:
◇ When you started with an existing project, or when you saved the project before, saving the project will not save changes you have made to the depth(s). Either use **Save Project As** or save individual depths.

5.1.4 Save project as

Upon selecting *File* → *Save Project As*, the current project can be saved under a different name.

5.1.5 Attribute files

On the *File* → *Attribute Files* sub-menu, see Figure 5.3, options are available to open and save objects that are indirectly related to the grids.

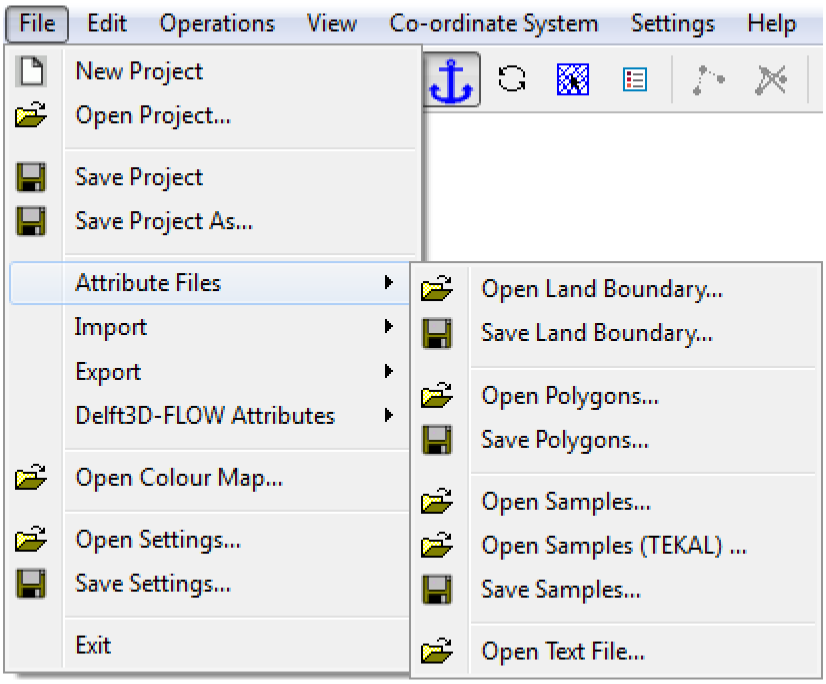


Figure 5.3: *Options on the* File→ Attribute Files *menu*

Open land boundary

Upon selecting *File* → *Attribute Files* → *Open Land Boundary*, you can open a collection of land boundaries. Land boundaries (or land-water marking) are in files with default mask <∗.ldb>.

Remark:
- ◇ If you open another land boundary file, it will be visualised together with the existing land boundary.

Save land boundary

Save the land-boundary, e.g. after it has been edited. Land boundary files have as default mask <∗.ldb>.

Open polygon

Upon selecting *File* → *Attribute Files* → *Polygons,* you can open a collection of polygons from a file with mask (<∗.pol>). Polygons are per definition closed. If the polygon is not closed in the file it will still be shown as closed.

Remark:
- ◇ If you open another polygons file, they will be visualised together with existing polygons.

Save polygon

When saving polygons, each polygon will be saved as a closed polyline. A polygon file has as default mask <∗.pol>.

Open samples

The samples in a file with mask <∗.xyz>, may be a set of disordered x, y, z values given in a sequential list of free-formatted x, y, z values.

Remark:
- ◇ If you open another samples file, the samples will be visualised together with existing samples.

Open TEKAL samples

In this file both the depth values and the grid information are contained. In order to use this option you must have saved the bathymetry in TEKAL format (on the *File* menu, point to *Export* and click *Grid and Depth (TEKAL)*.

Save samples

The program offers the facility to insert additional samples, delete or replace samples. The modified data set of samples can then be saved as a sequential list of free-formatted x, y, z values.

Open text file

Texts can be displayed in the graphics area if their position (x, y), the text and colour are defined. See an example in section A.12.

5.1.6 Import

On the *Import* sub-menu, see Figure 5.4, options are available to import objects that are directly related to the grids.

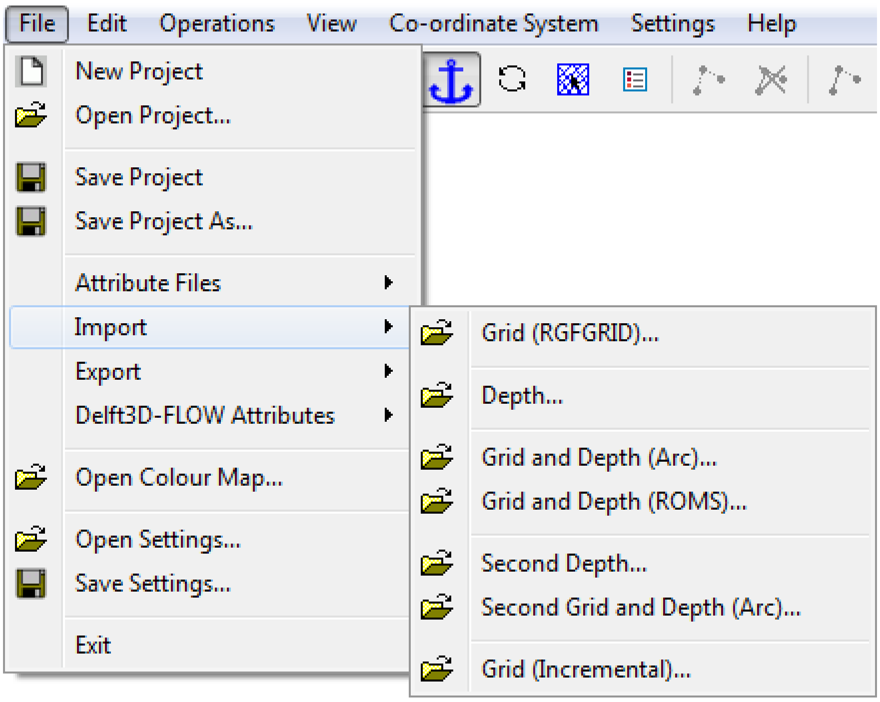


Figure 5.4: File $\rightarrow$ Import *options*

Grid (RGFGRID)

Upon selecting *File* $\rightarrow$ *Import* $\rightarrow$ *Grid (RGFGRID)*, you can open a collection of grids. The grid file has a default mask $<\ast$.grd$>$.

Remarks:

- ◇ The co-ordinate system in QUICKIN is set accordingly to the system specified in the grid file.
- ◇ If the co-ordinate system is spherical then the co-ordinates are shown in stereographic projection.
- ◇ If no co-ordinate system is specified, Cartesian is presumed.

Depth

Interpolated depth values in a file with mask <∗.dep> can be opened in the Delft3D-FLOW depth-file format.

Grid and Depth (Arc)

This option is relevant to users of Delft1D2D. Upon selecting *File* → *Import* → *Grid and Depth (ARC)*, you can open a single grid. This GIS file contains both a rectangular grid and bathymetry definition in one file. The file has mask <∗.a∗>. If the extension chosen is <amu>, QUICKIN assumes that a Delft1D2D map-file of u-velocity components has to be opened and the v-velocity components in file <∗.amv> are automatically opened as well. The resulting velocity vector field will be displayed and can be shown on a map of the Delft1D2D bathymetry. A description of this file format is given in section A.7.

Grid and Depth (ROMS)

Upon selecting *File* → *Import* → *Grid and Depth (ROMS)*, you can open a grid in the NetCDF format as used for the Regional Ocean Modelling Sytem (ROMS).

Second depth

A second file of interpolated depth values can be opened in a file with mask <∗.dep>. This second depth field can be subtracted from or added to the present depth values (e. g. adding a non-uniform mean sea level correction to the present depth values), see menu *Operations* → *Combine Depth and Second Depth*.

Second Grid and Depth (Arc)

After you have opened a bathymetry in a file with mask <∗.adp>, you can open water levels in a file with mask <∗.amz>, to be displayed together with the bathymetry.

Grid (incremental)

After you have opened a ARC-grid (*Grid and Depth (ARC)*) file with mask <∗.aht>, you can open a incremental result file <∗.inc> from Delft1D2D. This file can be used for animation in QUICKIN.

5.1.7 Export

On the *File* → *Export* sub-menu, see Figure 5.5, options are available to export objects that are directly related to the grids.

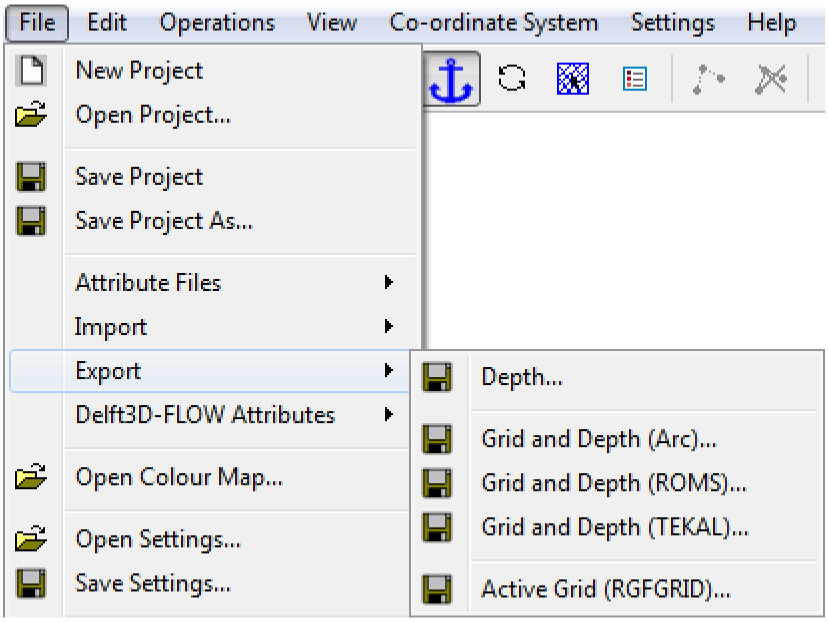


Figure 5.5: File → Export *sub-menu options*

Depth

Interpolated depth values can be saved in the Delft3D-FLOW depth-file format. The default mask is <∗.dep>. If you need to specify a different extension (initial or rougness data) enter the required filename and extension, and add as last character a dot (ex. <waterlevel.ini.> the file on disk will be named <waterlevel.ini>). Another possibility is to rename the filename after it is saved.

Grid and depth (Arc)

Save the bathymetry and rectangular grid definition.

Grid and depth (ROMS)

Save the grid and depth values in a NetCDF file which can be used with ROMS (Regional Ocean Model.System).

Grid and depth (TEKAL)

This file can be used in the Deltares | Delft Hydraulics standard plotting packages as Delft3D-QUICKPLOT and GPP, QUICKPLOT (2013) and GPP (2013).

Active grid (RGFGRID)

After you have interpolated depth values on the grid, some parts of the grid may not be covered with data, e.g. if these parts fall on land. In that case this option gives an easy way to 'cut' away those parts of the grid that are not interpolated. A new grid is written to a file with mask <∗.grd>, that only contains active grid cells, plus a so-called 'computational grid enclosure' file with mask <∗.enc>, see the Delft3D-FLOW User Manual Delft3D-FLOW (2013), that indicates the border of the computational domain in Delft3D-FLOW computations.

5.1.8 Delft3D-FLOW attributes

Dry points, thin dams and observation points for the hydrodynamic module Delft3D-FLOW can be defined in two ways: either using the **Visualisation Area** window of the FLOW Graphical User Interface, or, using QUICKIN. The (sediment) dredge and dump areas and their characteristics can only be defined in QUICKIN, see Figure 5.6.

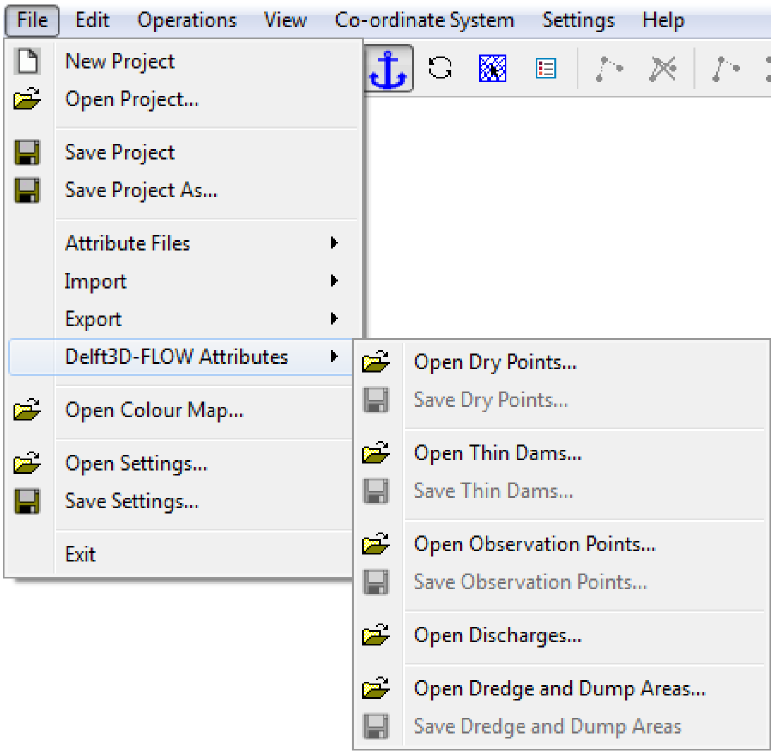


Figure 5.6: *Open and save Delft3D-FLOW attribute files*

Open dry points

Open a file with Delft3D-FLOW dry points. This file has mask <∗.dry>. See section 5.2.10 for details how to define these points.

When opening a file with dry points and dry points are already defined, then the latter will be removed.

Save dry points

Save the dry points to a file with mask <∗.dry>. A description of this file format is given in section A.9.

Open thin dams

Open a file with Delft3D-FLOW thin dams. This file has mask <∗.thd>. See section 5.2.11 for details how to define these points.

When opening a file with thin dams and thin dams are already present, then the latter will be removed.

Save thin dams

Save the thin dams to a file with mask <∗.thd>. A description of this file format is given in section A.10.

Open observation points

Open a file with Delft3D-FLOW observation points. This file has mask <∗.obs>. See section 5.2.12 for details how to define these points.

When opening a file with observation points and observation points are already defined, then the latter will be removed.

Save observation points

Save the observation points to a file with mask <∗.obs>. A description of this file format is given in section A.11.

Open discharges

Open a file with Delft3D-FLOW discharges. This file has mask <∗.src>. It is not possible to define discharge location, type and time-series, in that case you have to use FLOW-GUI.

When opening a file with discharges and discharges are already present, then the latter will be removed.

Open dredge and dump areas

Open the file that contains the dredge and dump areas (defined in a separate polygon file), the depths to be dredged and the links between dredge and dump areas. The file has mask <∗.dad>.

Save dredge and dump areas

Save the dredge and dump areas (defined in a separate polygon file), the depths to be dredged and the links between dredge and dump areas. The file has mask <∗.dad>. When asked to save the polygon file, click *Yes* when not yet saved.

5.1.9 Open Colour map

You can choose from a number of pre-defined colour schemes (in file with masks <∗.clr> or <∗.clrmap>). These colour schemes have the same format as used for Delft3D-QUICKPLOT, see section A.14 for the file format.

Restriction:
- Only the colour space `RGB` is supported

Remark:
- If the file <quickin.clrmap> exists on the start-up directory then this file will be read, if the file does not exist on the start-up directory it will try to read the file on the installation directory <$D3D_HOME/$ARCH/plugins/default>.

5.1.10 Open Settings

If you have saved your QUICKIN settings in a previous session, you can open these settings again, see section A.15 for the file format.

Remark:
- If the file <quickin.clrmap> exists on the start-up directory then this file will be read, if the file does not exist on the start-up directory it will try to read the file on the installation directory <$D3D_HOME/$ARCH/plugins/default>.

5.1.11 Save Settings

If you have made changes in one of the forms on the *Settings* menu, you can save these settings to be used later on again.

5.1.12 Exit

Exit from the QUICKIN program

5.2 Edit menu

Several edit modes are presented on the *Edit* menu, see Figure 5.7. An edit mode is an operation mode which needs at least a mouse click, i.e. a set of operation instructions which is valid for a certain data set, and which may go with some specific display method. The following objects may be modified:

- Land boundary
- Polygon
- Depth
- Samples
- Delft3D-FLOW attributes (dry points, thin dams, observation points or dredge and dump areas)

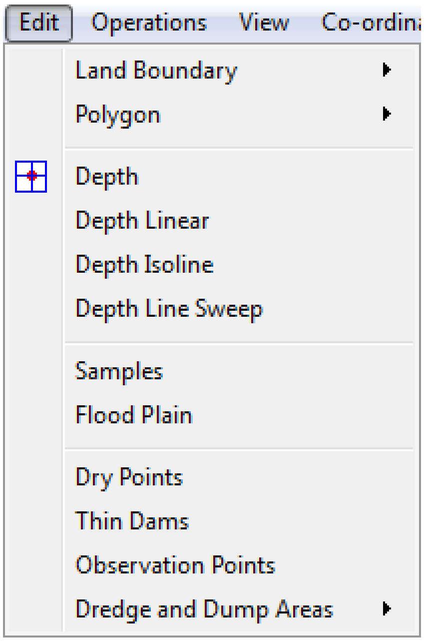


Figure 5.7: *Options on the* Edit *menu*

`Esc` = Undo
In most edit modes, `Esc` will undo the latest action.

5.2.1 Land boundary

The land boundary is used to visualise the land-water interface. To edit (define or modify) a land boundary, possible edit actions see Figure 5.8.

Edit

Upon selecting *Edit* → *Land Boundary* → *Edit*, you can start editting a polyline that defines an Land Boundary. When there is no polyline the edit mode is set to *New*, otherwise you have to select first a polyline (from the menu *Edit* → *Land Boundary* → *Select* or press the key `s`). After you have selected the polyline you can use key-strokes, icons in the toolbar or menu items to switch the edit mode.

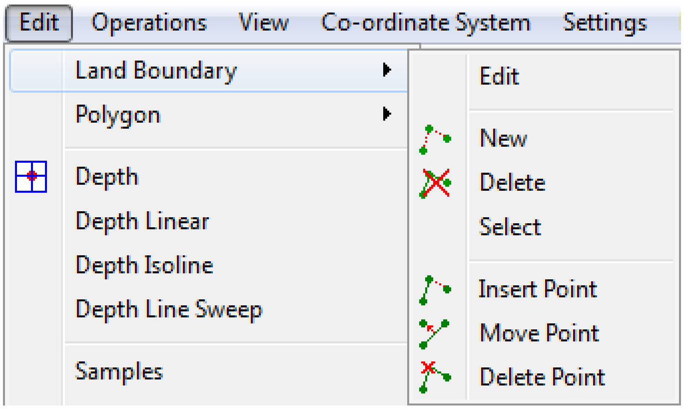


***Figure 5.8:** Options on the* Edit → Land Boundary *menu*

New

Upon selecting *Edit* → Land Boundary → *New*, you can start to define a new polyline, click on , or use the key-stroke `n` to start a new polyline.

Delete

Upon selecting *Edit* → *Land Boundary* → *Delete*, click on , or use the key-stroke `e`, to delete (erase) the selected polyline.

Select

Upon selecting *Edit* → *Land Boundary* → *Select*, or use the key-stroke `s`, you can select a polyline by clicking on one of its edges or vertices. After that the polyline will be highlighted

Insert point

Upon selecting *Edit* → *Land Boundary* → *Insert Point*, click on , or use the key-stroke `i`, you can insert a point into the selected polyline. The point will be inserted at the nearest linear piece of the polyline.

Move point

Upon selecting *Edit* → *Land Boundary* → *Move Point*, click on , or use the key-stroke `r`, you can move (replace) a point on the selected polyline.

Delete point

Upon selecting *Edit* → *Land Boundary* → *Delete Point*, click on , or use the key-stroke `d`, you can delete a point on the selected polyline by indicating it.

5.2.2 Polygon

The Area of Interest polygon is used to limit the area of influence of operations and or edit actions. All grid points and samples that are inside the polygon are active in the subsequent interpolation or manipulation steps. The polygon is self closing. see Figure 5.9.

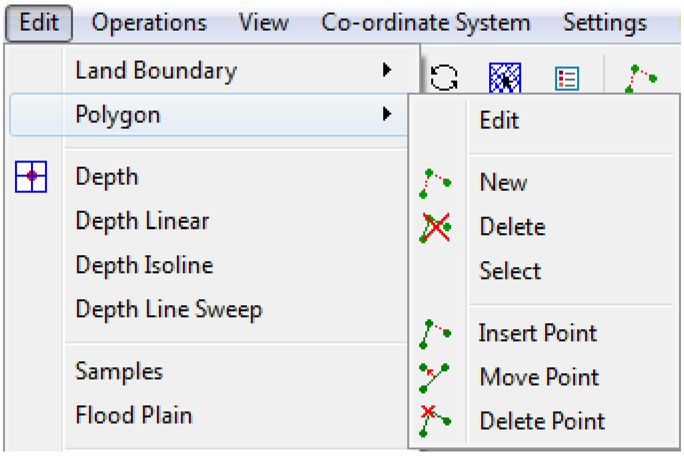


Figure 5.9: *Options on the* Edit → Area of Interest *menu*

Edit

Upon selecting *Edit → Area of Interest → Edit*, you can start editting a polygon that defines an Area of Interest. When there is no polygon the edit mode is set to *New*, otherwise you have to select first a polygon (from the menu *Edit → Area of Interset → Select* or press the key `s`). After you have selected the polygon you can use key-strokes, icons in the toolbar or menu items to switch the edit mode.

New

Upon selecting *Edit* → Area of Interest → *New*, you can start to define a new polygon, click on , or use the key-stroke `n` to start a new polygon.

Delete

Upon selecting *Edit → Area of Interest → Delete*, click on  , or use the key-stroke `e`, to delete (erase) the selected polygon.

Select

Upon selecting *Edit → Area of Interest → Select*, or use the key-stroke `s`, you can select a polygon by clicking on one of its edges or vertices. AFter that the polygon will be highlighted

Insert point

Upon selecting *Edit* → *Area of Interest* → *Insert Point*, click on , or use the key-stroke `i`, you can insert a point into the selected polygon. The point will be inserted at the nearest linear piece of the polygon.

Move point

Upon selecting *Edit* → *Area of Interest* → *Move Point*, click on , or use the key-stroke `r`, you can move (replace) a point on the selected polygon.

Delete point

Upon selecting *Edit* → *Area of Interest* → *Delete Point*, click on , or use the key-stroke `d`, you can delete a point on the selected polygon by indicating it.

To define dredge and dump areas go to the *Edit* menu, point to *Dredge and Dump Areas* and click *New Area*.

The polygon will remain valid after each operation, but it can be deleted, via menu *Edit* → *Polygon* → *Delete*. The polygon can be used as a so-called 'fault line' in the interpolation process.

Upon selecting *Edit* → *Polygon*, you can start to define a new polygon, or click on  to start a new polygon.

Valid action keys are

In *Edit* → *Polygon* mode the following keys can be used:

`I` = Insert
Pressing `I` allows you to insert individual polygon points. The message at the left of the statusbar now reads 'Insert a point', click the left mouse to insert individual points.

`R` = Replace
Pressing `R` allows you to replace (move) individual polygon points. The message at the left of the statusbar now reads 'Get a point'. If you have got it by clicking the left mouse, the message will read 'Put a point', and you can do so by clicking the left mouse at the new desired position.

`D` = Delete
Pressing `D` allows you to delete individual polygon points.

`X` = Break open polygon
Keep *X* pressed and move with the cursor over a polygon point to split the polygon. Effectively, the pointed co-ordinates are replaced by default 'missing' values.

`E` = Erase polygon
Entire polygon sections are deleted. Press key *E* and then click with the left mouse button on a point of the polygon which need to be deleted. Finish the operation by pressing the right mouse button.

5.2.3 Depth

By clicking on a grid point, a window opens in which the depth value is displayed. You can change the value manually. The depth value -999.000 indicates the grid point has no depth value (missing value).

To delete individual depth points, first press `D` and click on the grid point. Press `C` or `I` to go back in the edit mode.

`Tab` = depth cursor
In menu *Edit* $\rightarrow$ *Depth* and menu item *Depth Linear* pressing the `Tab` key while the cursor is on a grid point, will display the depth value in the lower status bar. In normal view (not in *3D View*), you can press the `Tab` key to go into 'depth cursor mode'. This allows you to 'walk' over the depth points using the arrow keys. When in 'depth cursor mode' you can increase or decrease the depth values by using the + and - keys, respectively.

Press the `D` key to delete the current depth value. Press `Esc` to undo the latest changes. Press `Tab` again to go into normal cursor mode. The depth value and the (M, N) indices of the current depth point are given in the status bar at the bottom of the screen.

5.2.4 Depth linear

Manually interpolate a line of grid depth values, by clicking two points in the grid that have valid depth values. The points in between the two grid points will be linearly interpolated in between the values of the end points. This edit option, in combination with *Operations* $\rightarrow$ *Internal Diffusion*, provides a very powerful way to build new bathymetries or modify existing ones.

See this combination like the construction of a ship. First, a ships framework is built, (lines of grid depth values) then, its plates are welded to the frame (*Internal Diffusion*).

Reject a result by pressing `Esc`. If the desired end points do not yet have valid depth values, first assign depth values by putting the cursor on top of these points and pressing `Tab`. The + or - keys will increase or decrease the depth value.

5.2.5 Depth isoline

The operation described above can also be performed in iso-line mode. The value of the iso-line is taken as the value of the FIRST point that has been indicated.

5.2.6 Depth line sweep

Indicate two points on the same grid line. The grid points on the indicated grid line will be copied onto the grid lines parallel to the indicated line. Use the polygon to limit the effective area.

5.2.7 Samples

Samples can be inserted (with a specified depth value), replaced or deleted in the same way as the points of a polygon. The value of the marked sample is displayed in the status bar at the bottom of the screen together with the x and y co-ordinates and the distance from the anchor. The value of the marked sample can then be modified by pressing the + (higher) or - (lower) key (or press D for delete or press Esc for restore). In edit mode after pressing the C-key), sample values can be modified. In edit Insert after pressing the I-key), new sample values can be inserted.

5.2.8 Check dike heights

It allows you to quickly check whether the dykes in a Delft1D2D schematisation start overtopping at the proper water level. (Typically useful in a polder area divided in different compartments). First open a Delft1D2D bathymetry <∗.adp>. Next choose this option and click some depth point from which you want to 'release' water at the level (positive up) that you specify by typing a value. Next, water is 'released' from that point and the surrounding bathymetry is 'filled' until the water is stopped either by dykes or high grounds. Increase the 'release level' until dykes start flooding. The wetted area is displayed by means of the 'second depths', see *View → Second Depth*.

5.2.9 Edit Second Depth Animate Incremental

The option allows for animation of the flooding process computed by Delft1D2D. First, open a Delft1D2D bathymetry <∗.adp> and then choose this option. Open the relevant incremental file <∗.inc> and press the space bar to start the animation. The animation can be stopped (and restarted) by pressing the space bar again, and be propagated over 1 hour intervals by pressing the +-key. The wetted area is displayed by means of the 'second depths', see *View → Second Depth*.

5.2.10 Dry points

Dry points are grid cells centred around a water level point that are permanently dry during a computation, irrespective of the local water depth and without changing the water depth as seen from the wet points. In QUICKIN dry points are specified as single points. In the Delft3D-FLOW GUI they can also be defined as a line of points.

Click inside a grid cell to define it as dry point. Click on an existing dry point to delete this point.

Remark:

◇ Dry points for the hydrodynamic module Delft3D-FLOW can be defined in two ways: either using the **Visualisation Area** window of the Delft3D-FLOW Graphical User Interface, or using QUICKIN.

5.2.11 Thin dam points

Thin dams are infinitely thin objects defined at the velocity points which prohibit flow exchange between the two adjacent computational cells without reducing the total wet surface and the volume of the model. The purpose of a thin dam is to represent small obstacles (e. g. breakwaters, dams) in the model which have sub-grid dimensions, but large enough to influence the local flow pattern. Thin dams are specified at the edges of a grid cell. In QUICKIN thin dams are specified as single grid edges. In the Delft3D-FLOW GUI they can also be defined as a line of vortices.

Click at a grid vortex to define it as thin dam. Click on an existing thin dam to delete this point.

Remark:

◇ Thin dams for the hydrodynamic module Delft3D-FLOW can be defined in two ways: either using the **Visualisation Area** window of the Delft3D-FLOW Graphical User Interface, or using QUICKIN.

5.2.12 Observation points

Observation points are used to monitor the time-dependent behaviour of one or all computed quantities as a function of time at a specific location, i.e. water elevations, velocities, fluxes, salinity, temperature and concentration of the constituents. Observation points represent an Eulerian viewpoint at the results. Observation points are located at cell centres, i.e. at water level points.

Click inside a grid cell to define it as observation point; specify a name (maximum of 20 characters) for the point. Click on an existing observation point to delete this point.

Remark:

◇ Observation points for the hydrodynamic module Delft3D-FLOW can be defined in two ways: either using the **Visualisation Area** window of the Delft3D-FLOW Graphical User Interface, or using QUICKIN.

5.2.13 Dredge and dump areas

The *Edit* → *Dredge and Dump Areas* menu item contains several items to define and edit the dredge and dump areas, see Figure 5.10, each item is discussed in a separate sections.

First you have to define the locations of dredge and dump areas by editing polygons. For each area a polygon is required. Next you give each area a meaningful name. Click the name field and specify the polygon name, reflecting for instance if it will be a dredge or a dump site, and specify a drege depth (if applicable).

The next step is to link (connect) each dredge area to a dump area. Individual links can be deleted

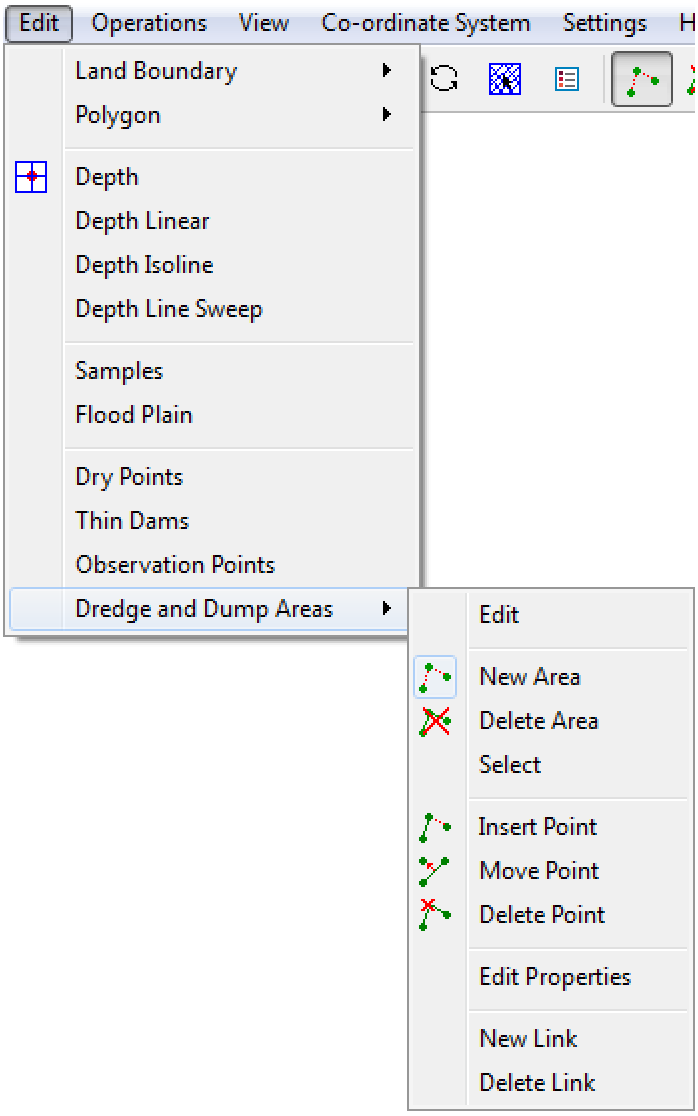


Figure 5.10: *Options on the* Edit → Dredge and Dump Areas *sub-menu*

New area

First you have to define the locations of dredge and dump areas by editting polygons. Select *Edit → Dredge and Dump Areas → New Area* and start defining a new area. Pressing the right-mouse button will stop defining the current polygon and the next left-mouse click will start a new area (polygon).

Edit properties

By pressing *Edit → Dredge and Dump Areas → Edit Properties* you can edit the properties of a dredge/dump area (i.e. name of area and the dredge depth, -999.000 means no dredging), see Figure 5.11.

If the area is a dredge area you have also to specify the amount of dredge material (percentages) which will be dumped in the linked dump areas, see Figure 5.12.

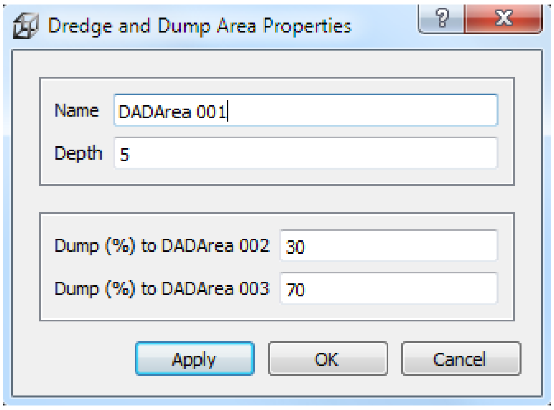


Figure 5.11: *Options on the* ***Dredge and Dump Properties*** *window for a dredge area*

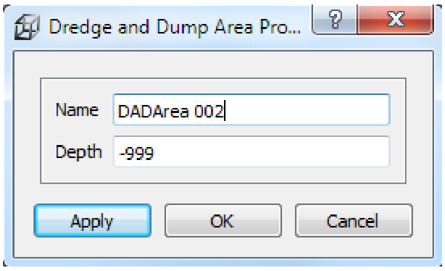


Figure 5.12: *Options on the* ***Dredge and Dump Properties*** *window for a dump area*

Delete area

Up on selecting *Edit* → *Dredge and Dump Areas* → *Delete Area* you can delete a polygon which defines a dredge or dump area. Click with the left mouse button inside a polygon, then that polygon will be deleted and all the links from or to that area.

Insert point

Upon selecting *Edit* → *Dredge and Dump Areas* → *Insert Point*, you can insert a point into the polygon. The point will be inserted at the closest linear piece of the polygon.

Move point

Upon selecting *Edit* → *Dredge and Dump Areas* → *Move Point*, you can move a point on the polygon.

Delete point

Upon selecting *Edit* → *Dredge and Dump Areas* → *Delete Point*, you can delete a point on the polygon by indicating it.

New link

Upon selecting *Edit* → *Dredge and Dump Areas* → *New Link*, you can define a link (connection) between a dredge and a dump area. First click in a dredge area, subsequently in the dump area.

Delete link

Upon selecting *Edit* → *Dredge and Dump Areas* → *Delete Link*, you can delete a link (connection) between a dredge and a dump area. After selecting the menu-item select with the left mouse button the link which need to be deleted.

5.3 Operations menu

On the *Operations* menu, see Figure 5.13, you may choose to calculate depth values at grid points from sample values using various interpolation options (grid cell averaging, triangular interpolation, internal diffusion). The area and volume of (part of) a grid can be calculated.

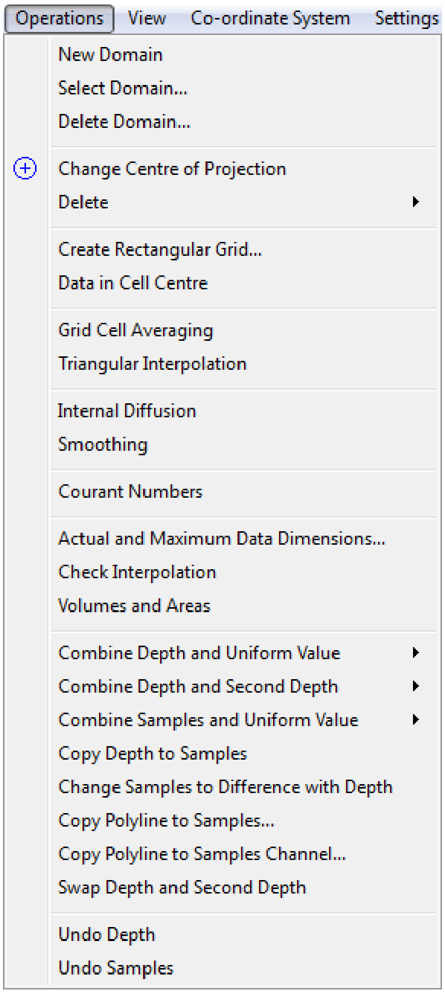


Figure 5.13: *Options on the* Operations *menu*

5.3.1 New Domain

On the *Operations* → *New Domain* menu, create a new domain. This means that every grid create action will add a new domain to your model. If you want to adjust your grid by the option *Operations* → *Create Rectangular Grid. . .*, deactivate this menu item first.

5.3.2 Select domain

If your project consists of multiple grids (so-called domain decomposition application) you can switch between the domains (grids) by clicking *Edit* → *Select Domain*, or click  on the toolbar. Next, click on the grid you want to become the active grid.

5.3.3 Delete domain

To delete a domain or grid, select *Edit* → *Delete Domain*. Next, click on an active or inactive grid. When deleting a domain, also objects (except dredge and dump areas) defined on the grid will be deleted.

5.3.4 Change centre of projection

For spherical co-ordinates QUICKIN can use two different projections, plane projection and stereographic projection. For stereographic projection a special function is implemented to centring the computer screen to the centre of projection and the sphere. This function can be invoked by clicking the menu item *Operations* → *Change Centre of Projection*, see Figure 5.13. When using this command the centre of the projection is set to the centre of the screen. This action requires recalculation of the projection and a new screen refresh. The centre of the projection does not change when using zoom in, zoom out or pan, so there is no performance drawback and a smooth screen-refresh is obtained.

5.3.5 Delete

Using this option you can (see Figure 5.14) delete the Land Boundary, Polygons, Depth, Second Depth and Samples. It is possible to indicate an area of influence, by defining a polygon. If a polygon is defined, the operation applies to points inside the polygon. When choosing *Delete* → *Polygons* all polygons will be deleted.

Land boundaries

If a polygon is defined, the land boundary points inside the polygon are removed. If no polygon exists you are asked if all land boundary points have to be removed. If you have defined two or more polygons, than only the points inside the first defined polygon will be removed.

Polygons

If polygons are present, they all will be removed.

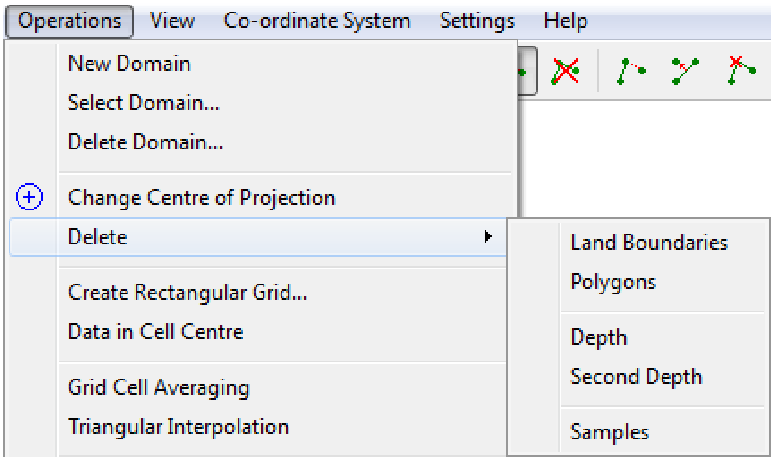


Figure 5.14: *Options on the* Operations → Delete *menu*

Depth

If a polygon is defined, the depths inside the polygon are removed. If no polygon exists you are asked if all depth points have to be removed. If you have defined two or more polygons, than only the depths inside the first defined polygon will be removed.

Second depth

If a polygon is defined, the second depths inside the polygon are removed. If no polygon exists you are asked if all second depth points have to be removed. If you have defined two or more polygons, than only the second depths inside the first defined polygon will be removed.

Samples

If a polygon is defined, the samples inside the polygon are removed. If no polygon exists you are asked if all samples have to be removed. If you have defined two or more polygons, than only the samples inside the first defined polygon will be removed.

5.3.6 Create Rectangular Grid

Specify the grid spacing, grid origin, rotation of the grid (counter clockwise) and the number of grid cells in both directions to quickly create a rectangular grid with rectangular cells. The parameters involved and their default values are, see Figure 5.15.

- *Number of Grid Cells in M-Direction* default: 50
- *Number of Grid Cells in N-Direction* default: 50
- *Delta X [m]* default: 100.0
  Grid cell size M-direction [m]
- *Delta Y [m]* default: 100.0
  Grid cell size N-direction [m]
- *Origin X [m]* default: 0.0
- *Origin Y [m]* default: 100.0
- *Rotation left [degrees]* default: 0.0

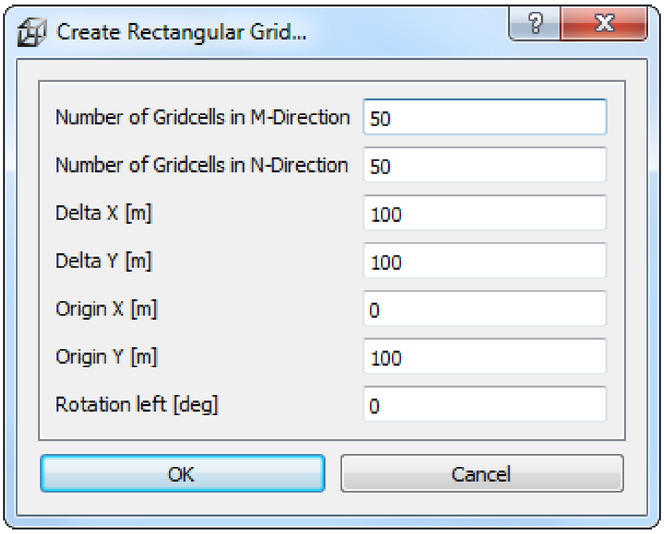


Figure 5.15: Create Rectangular Grid. . . *form*

5.3.7 Data in cell centre

QUICKIN can be applied to specify any spatially varying parameter, which may also be some initial condition or some spatially varying input parameter. Because some of these parameters may be defined at cell centres (rather than cell corners, as with Delft3D-FLOW depth values), the position of the points to interpolate (or average) to, can be changed to the cell centres. The water level is an example of a parameter defined at cell centres.

5.3.8 Grid cell averaging

This method is the preferred method if there are (many) more samples than grid points. The standard option is simple averaging. Also, the maximum or minimum value of the samples in the grid point vicinity can be chosen (maybe to guarantee a wetted gully), or the value of the closest sample within the vicinity. (See menu *Settings* → *Averaging Options*). The default vicinity is defined as the area covered by an 8 point polygon surrounding the current grid point, with its four corners as the cell centres of the four grid cells around the grid point. In between these corner points, another four points are added that lie halfway between the grid point and its neighbouring grid points. The default area can be increased or decreased in size by a linear scaling factor that can be specified in the *Settings* → *General* form.

Defaults:

- ◇ Relative Search Cell Size for Averaging: 1.0
- ◇ Minimum Number of Averaging Points: 4
- ◇ Averaging Options: Average value of near points

5.3.9 Triangular interpolation

The triangulation method is best suited for data sets that have a resolution that is about equal to or smaller than the grid resolution. The samples are first organised into a so-called Delaunay network, next grid values are interpolated. The amount of operations needed in the network creation process depends quadratically on the number of samples. So, consider working in portions rather than with the complete data sets. In that case, take special care at the border zones of adjacent interpolation areas. Maybe delete the border zones using a polygon and re-interpolate these zones using a slightly larger polygon.

Default:

◇ Default the 'interpolation process' is not shown, see the menu *View* → *Interpolation*

5.3.10 Internal diffusion

Grid points that have no depth value yet, i.e. 'blank' points with the indicator value of -999, are modified in their value by the so-called 'Internal diffusion' mechanism. Basically, this mechanism is a smoothing process which is called repeatedly, but that does not change values of already existing depth values. Therefore, a smooth transition with the existing bathymetry is inherent. This gives an easy method to fill 'gaps' in a bathymetry, or to build new bathymetries in combination with the *Edit* → *Depth Linear* mode, as explained above. The *Number of Internal Diffusion Steps* can be specified in *General* form of menu *Settings*.

Default:

◇ Number of Internal Diffusion Steps — default: 200

5.3.11 Smoothing

In some bathymetries large depth gradients may occur, that may result in non-smooth numerical results in the hydrodynamic or wave programs. Isolate these regions and treat them with this volume conserving smoothing method.

The *Number of Smoothing Steps* and the *Smoothing Factor* can be specified in the *Settings* → *General* form.

The defaults are:

◇ Number of Smoothing Steps — default: 10
◇ Smoothing Factor — default: 0.5

5.3.12 Courant numbers

The latter option copies the Courant numbers defined as

$$Courant = 2\Delta T c\sqrt{1/\Delta x^2 + 1/\Delta y^2}$$

see Delft3D-FLOW (2013), to the second depth array and sets display parameters such that the second depth is displayed as continuous shading. Δt is the computational time-step in seconds (see *Settings* → *General*), and c the local wave celerity, $\sqrt{g(h+\zeta_0)}$ (see menu item *Settings* → *General*).

The default values are:

- ◇ Time-step for Courant Number — default: 60.0 [s]
- ◇ Reference Level ζ_0 — default: 0.0 [m]

Remark:

- ◇ Note that you have to specify the time-step in seconds, but in FLOW you specify minutes.

5.3.13 Actual and maximum data dimensions

The actual and maximum dimensions of various data objects are presented in 'history'.

5.3.14 Check interpolation

If both samples and depth values are opened in the program, it is possible to check the quality of the interpolation by reverse interpolation of the samples in the created bathymetry. The reverse interpolation is performed for all grid cells within the polygon that have depth values at all four grid cell-corners. The reverse-interpolation is realised by bilinear interpolation of the samples in the grid cells. The difference between these values and the original sample values gives an indication for the quality of the interpolation. (Or an indication of the variance of your sample data in relation to the resolution of the grid. Therefore, if you see large differences, do not conclude too hastily that the interpolation itself is poor!) The mean difference, the mean absolute difference and the maximum difference are presented via menu option *Operations* $\rightarrow$ *Actual and maximum data dimensions*, together with the number of samples that have been evaluated.

5.3.15 Volumes and areas

Volumes and areas are calculated for computational grid cells within the polygon. The area of the computational cells is always smaller then the area of the polygon. If no polygon is given, all grid cells are used. The volumes and areas are computed for those cells with four valid depth values (data at cell vertices) or with one valid depth value (data at cell centre). Also the depth should be positive ($\zeta_{\text{ref}} + d^{\zeta} > 0$) for a contribution to the volumes and areas. Where d^{ζ} is the depth value at the cell centre, for data defined at the cell vertices it is defined as mean value of the four surrounding depth values.

Areas are given in the squared unit of the grid (mostly given in metres); volumes are given in squared unit of the grid multiplied by the unit of the depth. (For a Delft3D-FLOW bathymetry, both must be in metres). Volumes are given below a user definable reference level. The origin of that reference level is equal to the reference level of the depth data. The reference level is defined positive in upward direction. (Whereas depth values are defined positive in downward direction.)

Default:

- ◇ Reference Level ζ_{ref} — default: 0.0 [m]

5.3.16 Combine depth and uniform value

Using this option you can (see Figure 5.16) assign a value to not yet specified depths. For existing depths you can do some basic operations: add a constant value (for instance when you want to change the reference level), multiply with a constant value (for instance with -1 when your depths are positive upwards), or take the minimum or maximum of the depths and a specified value. Optionally, you can use a polygon to restrict the area of operation.

To use this options you first have to specify the uniform value (default: -999.0) in a pop-up window.

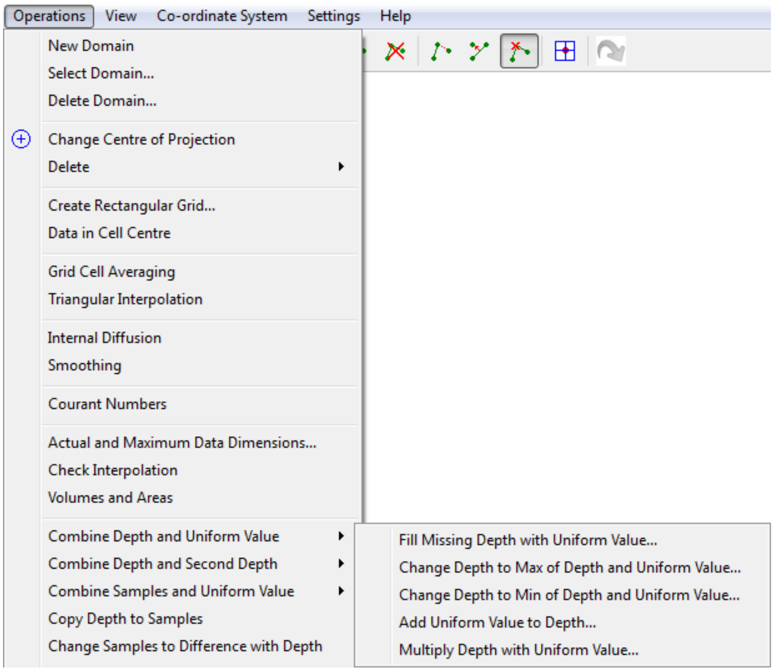


Figure 5.16: *Options on the* Combine Depth and Uniform Value *sub-menu*

Fill Missing Depths with Uniform Value

This option operates on grid points with non-existing depth values; they will be assigned the *Uniform Value*. The other options operate on grid points with existing (yet defined) depth values.

Change Depth to Max of Depth and Uniform Value

If the *Uniform Value* is larger than the depth, the depth value is replaced by the *Uniform Value*.

Change Depth to Min of Depth and Uniform Value

If the *Uniform Value* is smaller than the depth, the depth value is replaced by the *Uniform Value*.

Add Uniform Value to Depth

The *Uniform Value* is added to the depth value.

Multiply Depth with Uniform Value

The depth value is multiplied by the *Uniform Value* (for instance with -1 when your depths are positive upwards).

5.3.17 Combine depth and second depth

Using this option you can (see Figure 5.17) manipulate the depth using the second depth. The depth can be replaced by the second depth, the maximum or minimum can be taken, the second depth can be added or subtracted, the depth can be multiplied or divided by the second depth. If a polygon is defined, the operation applies to grid points inside the polygon.

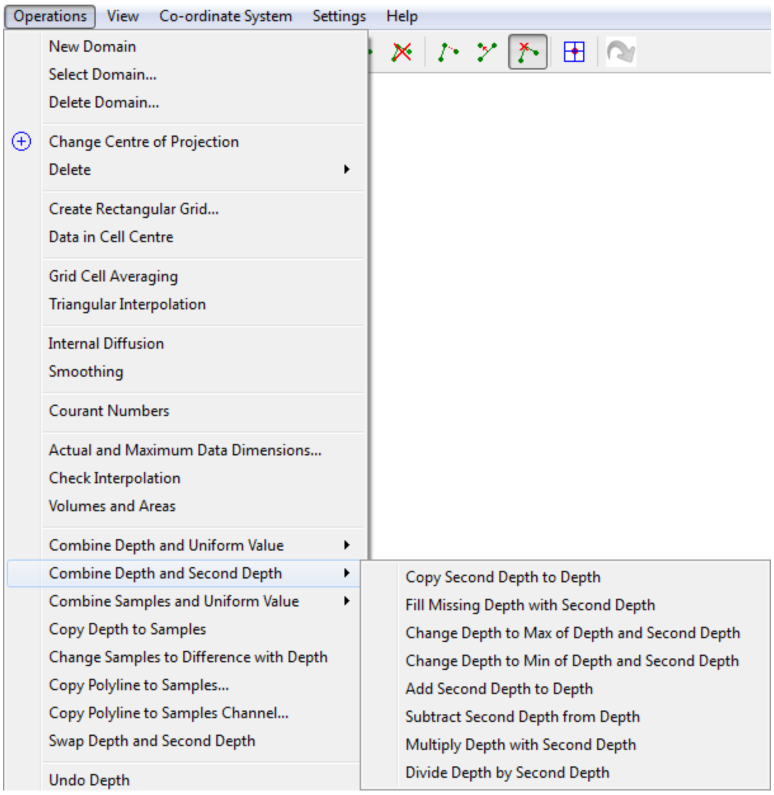


Figure 5.17: *Options on the* Combine Depth and Second Depth *sub-menu*

Copy Second Depth to Depth

If the second depth value is specified, the depth value will be replaced by the second depth value.

Fill Missing Depth with Second Depth

This option operates on grid points with non-existing depth values; they will be assigned the *Second Depth Value*.

Change Depth to Max of Depth and Second Depth

If the *Second Depth Value* is larger than the depth, the depth value is replaced by the *Second Depth Value*.

Change Depth to Min of Depth and Second Depth

If the *Second Depth Value* is smaller than the depth, the depth value is replaced by the *Second Depth Value*.

Add Second Depth to Depth

The *Second Depth Value* is added to the depth value.

Subtract Second Depth from Depth

The *Second Depth Value* is subtracted from the depth value.

Multiply Depth with Second Depth

The depth value is multiplied by the *Second Depth Value*.

Divide Depth by Second Depth

The depth value is divided by the *Second Depth Value*.

5.3.18 Combine samples and uniform value

Using this option you can (see Figure 5.18) manipulate samples compared to a *Uniform Value*. Optionally, you can use a polygon to restrict the area of operation.

To use these options you first have to specify the uniform sample value.

Change Samples to Uniform Value

Samples will get the *Uniform Value*.

Change Samples to Max of Sample and Uniform Value

Samples with a value smaller than the *Uniform Value* will get the *Uniform Value*.

Change Samples to Min of Sample and Uniform Value

Samples with a value larger than the *Uniform Value* will get the *Uniform Value*.

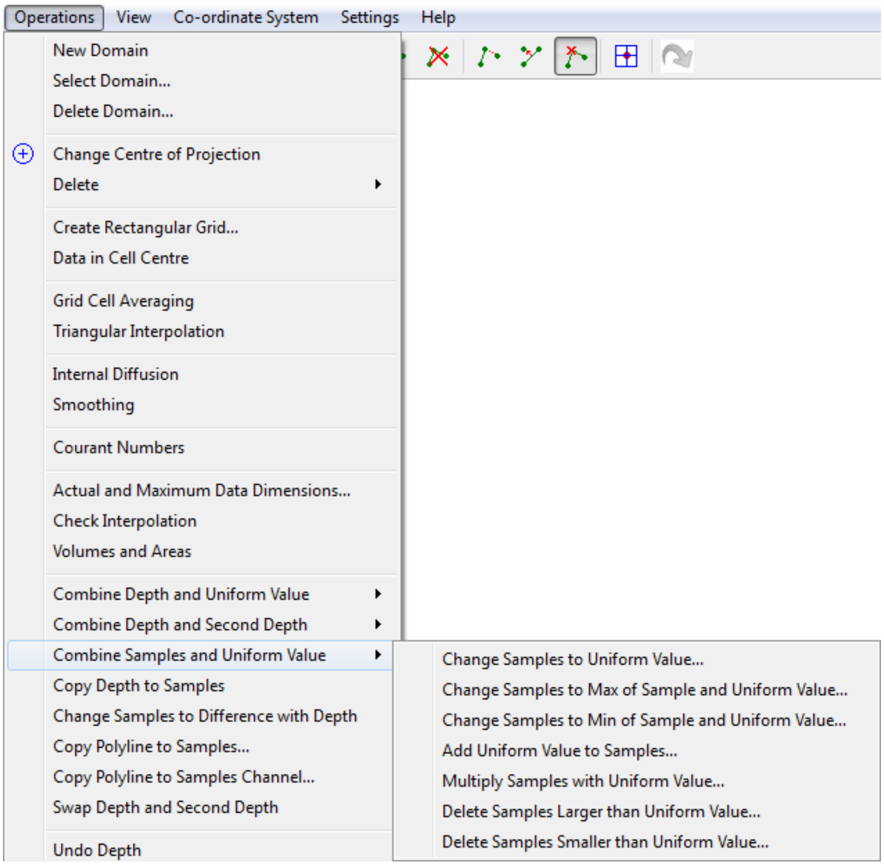


Figure 5.18: *Options on the* Combine Samples and Uniform Value *sub-menu*

Add Uniform Value to Samples

The *Uniform Value* will be added to the sample value.

Multiply Samples with Uniform Value

Samples will be multiplied by the *Uniform Value.*

Delete Samples Larger than Uniform Value

Samples larger than the *Uniform Value* will be deleted.

Delete Samples Smaller than Uniform Value

Samples smaller than the *Uniform Value* will be deleted.

5.3.19 Copy depth to samples

If you want to interpolate a detailed model based upon an existing overall model, first open the grid and the depths of the overall model. Then, copy the overall depth information to the samples and open the detailed model grid. Then, interpolate these samples to the detailed model. If a polygon is defined, the depth points inside the polygon are copied to samples.

5.3.20 Change samples to difference with depth

The depth value at sample point locations is interpolated based upon the depth values at the grid points. Next the original sample value is subtracted from this re-interpolated value.

5.3.21 Copy polyline to samples

A polyline can be changed to sample points by specifying the value at the begin and end of the polyline, see Figure 5.19. By linear interpolation the samples in-between are determined. If 'NO' linear interpolation is specified (0) the value at the begin of the polyline is used for all samples on the polyline (iso-line). The number of samples per line segment can be specified in the *Settings* → *Polyline to Samples* window, see Figure 5.19. Also, spline shape is possible for smooth isolines.

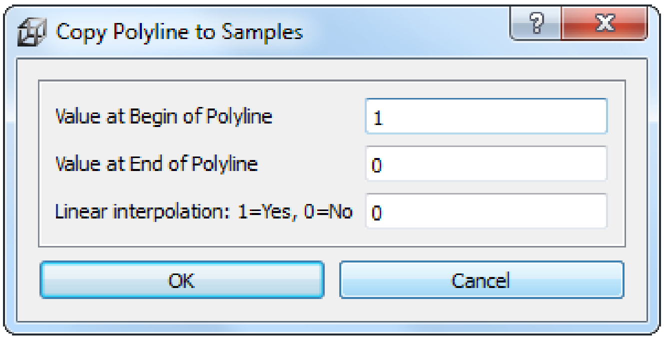


Figure 5.19: *Copy Polyline to Samples parameters*

The default values are:

◇ Value at Begin of Polyline — default: 1.0 [m]
  If no linear interpolation is used, all points will have the value at the begin.
◇ Value at End of Polyline — default: 0.0 [m]
◇ Linear Interpolation Yes or No — default: 0 (No)

Remark:
◇ If you define more than one polygon, only the first polygon will be used.

5.3.22 Copy polyline to samples channel

A polyline can be changed to sample points of a channel where the polyline is the centre line of the channel. The depth and width at the bottom have to be specified at the beginning and end of the channel as well as the channel slope and the number of samples across the channel width, see Figure 5.20. If the channel is placed in an existing bottom (presented by samples) it is possible to remove channel samples above the original bottom and remove existing samples above the channel (dredged channel). To this end a triangle network is created by the program. The position where a line of the channel slope crosses a triangle, a sample is created with a depth which is determined by linear interpolation of the depth of the triangle edges. The number of samples and the shape of the channel line can be specified in the *Settings* menu, *Polyline to Samples* window, see Figure 5.20

The default values are:

◇ Channel Width at Begin of Polyline — default: 100.0 [m]

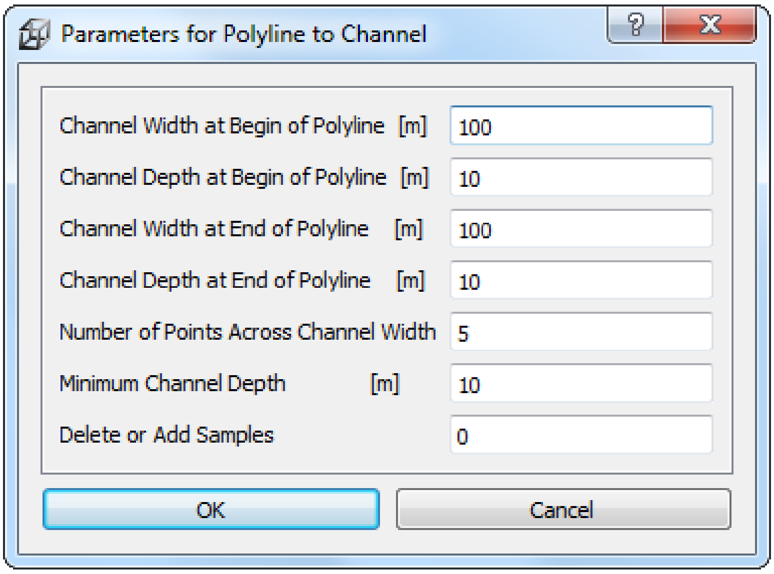


Figure 5.20: *Parameters form for Polyline to Samples Channel*

- ◇ Channel Depth at Begin of Polyline — default: 10.0 [m]
- ◇ Channel Width at End of Polyline — default: 100.0 [m]
- ◇ Channel Depth at End of Polyline — default: 10.0 [m]
- ◇ Number of Points across Channel Width — default: 5
- ◇ Channel Side Slope — default: 5
- ◇ Minimal Channel Depth — default: 10.0 [m]
- ◇ Delete or Add Samples — default: 0

Remark:

- ◇ If you define more than one polygon, only the first polygon will be used.

5.3.23 Swap depth and second depth

This will interchange the depth and second depth points. If a polygon is defined, only the depth points inside the polygon are interchanged.

5.3.24 Undo depth

Reset the depth values to values prior to the last action. When applying the edit modes, pressing `Esc` has the same effect.

5.3.25 Undo samples

Reset the sample values to values prior to the last action. When applying the edit modes, pressing `Esc` has the same effect.

5.4 View menu

On the *View* menu, you may choose to display several data sets, see Figure 5.21

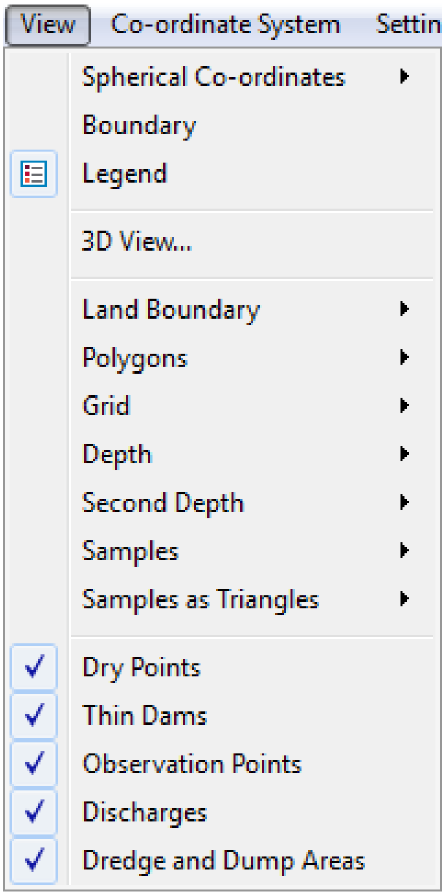


Figure 5.21: *Options on the* View *menu*

5.4.1 Spherical co-ordinates

Here you can select how to project the spherical co-ordinates onto the screen, see Figure 5.22.

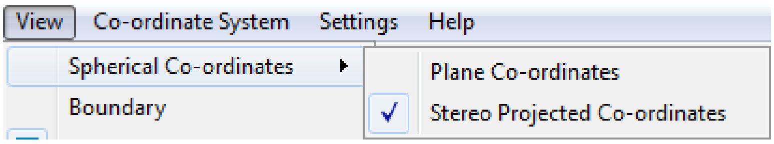


Figure 5.22: *Options on the* View → Spherical Co-ordinates *sub-menu*

Remarks:

◇ Only applicable for a spherical co-ordinate system.
◇ Default: A spherical grid is shown in stereographic projected co-ordinates.

Plane co-ordinates

Upon selecting *View → Spherical Co-ordinates → Plane Co-ordinates*, you can choose to the display the co-ordinates as they are.

Stereographic projected co-ordinates

Upon selecting *View* → *Spherical Co-ordinates* → *Stereo Projected Co-ordinates*, the co-ordinates are displayed using a stereographic projection onto the screen.

5.4.2 Boundary

Show or hide the computational boudary.

5.4.3 Legend

Show or hide the colour band on the left side of the screen.

5.4.4 3D View

Does not have the required performance, it very slow

In the **Delft3D-3DView** window a fully 3-dimensional view of the data is shown.

`c`: Switch rendering mode
`h`: Toggle help
`i`: Inverse depth
`r`: Reste view
`s`: Toggle samples
`x`: Increase depth
`z`: Decrease depth
left mouse: Rotate in xy-plane
`CTRL`+left mouse: move origin xy-plane
middle-mouse: zoom
right mouse: rotate z-axis

5.4.5 Land boundary

Upon selecting *View* → *Land Boundary*, you can show the land boundary.

The following view options are available:

- *No Land Boundary*
  Do not show the land boundaries.
- *Lines*
  Show the land boundaries as lines.
- *Lines with Dots*
  Show the land boundaries as lines with dots at the land boundary points.
- *Filled*
  Show the land boundaries as lines filled with a colour.

5.4.6 Polygons

The following view options are available:

◇ *No Polygons*
  Do not show the polygons.
◇ *Lines*
  Show the polygons as lines.
◇ *Lines with Dots*
  Show the polygons as lines with dots at the polygon points.
◇ *Filled*
  Show the polygons as lines filled with a colour.

5.4.7 Grid

The following view options are available:

◇ *No grid*
  Do not show the grid.
◇ *Lines*
  Show the grid with lines.
◇ *Node numbers*
  Show the node numbers.
◇ *Cell numbers*
  Show the grid cell numbers.

5.4.8 Depth

The following view options are available:

◇ *No depth*
  Do not show the depth.
◇ *Continuous shades*
  Show the depth as continuous shades interpolated from the values at the depth points.
◇ *Patches*
  Show the depth as average of the depth points.
◇ *Coloured Dots*
  Show the depth as coloured dots.
◇ *Coloured Numbers*
  Show depth with numbers coloured according value.
◇ *Mono coloured Numbers*
  Show depth with numbers in a single colour.
◇ *Points yet to be found*
  Show the depth points with an undefined value.

5.4.9 Second depth

The following view options are available:

- *No second depth*
  Do not show the second depth.
- *Continuous shades*
  Show the second depth as continuous shades interpolated from the values at the depth points.
- *Patches*
  Show the second depth as average of the values at the depth points.
- *Coloured Dots*
  Show the second depth as coloured dots.
- *Coloured Numbers*
  Show second depth with numbers coloured according value.
- *Mono coloured Numbers*
  Show second depth with numbers in a single colour.
- *Points yet to be found*
  Show the second depth points with an undefined value.

5.4.10 Samples

The following view options are available:

- *No Samples*
  Do not show the samples.
- *Coloured Dots*
  Show the samples as coloured dots.
- *Coloured Numbers*
  Show samples with numbers coloured according value.
- *Mono coloured Numbers*
  Show samples with numbers in a single colour.

5.4.11 Samples as triangles

The samples can be displayed as colour filled isolines using a triangle network. This is a slow visualisation method, zoom in to reduce the number of samples that must be triangulated. It is possible to display the slopes (α) of the individual triangles, or the local mild slope criterion. The slope is defined as the inner product of the triangle normal vector and the upward unit vector. The mild slope criterion is defined as α/kH, H being the total water depth, $(d + \zeta_0)$ and k being the wave number, which depends on the specified *Wave Period*. You can either display these values as numbers, or as green/red triangles indicating whether the slope or mild slope criterion is satisfied.

The following display options are available:

- *No Sample Triangles*
- *Edges*
- *Continuous Shading*
- *Edges and Continuous Shading*
- *Triangle Slopes*
  This option triggers the following sub-menu, see Figure 5.23

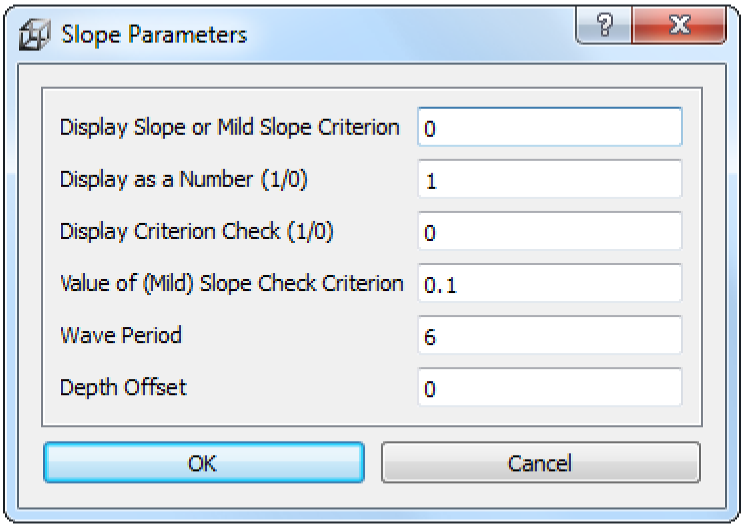


***Figure 5.23:** Options on the **Slope Parameters** window*

◇ *Display Slope or Mild Slope Criterion* default: neither
For each triangle, either the slope or the mild slope criterion can be displayed or be checked against some criterion.

◇ *Display as a Number Yes/No* default: yes

◇ *Display Criterion Check Yes/No* default: no
Shows in red/green whether the slope or mild slope criterion is satisfied

◇ *Value of (Mild) Slope Check Criterion* default: 0.1
If a criterion check is requested, then its value can be specified here.

◇ *Wave Period* default: 6.0 [s]
Needs only be specified if mild slope criterion is required

◇ *Depth Offset* default: 0.0 [m]
Is used in wave period computation, added to local depth.

5.4.12 Dry points

Show or hide the dry points.

5.4.13 Thin dams

Show or hide the thin dams.

5.4.14 Observation points

Show or hide the observation points.

5.4.15 Discharges

Show or hide the discharge locations.

5.4.16 Dredge and dump areas

Show or hide the dredge and dump areas.

5.5 Co-ordinate System menu

If you want to import some files and you known on forehand what the co-ordinate system is you have to select it here. Sometimes this will be the case if you first load a land boundary and/or polygons before you import the grid.

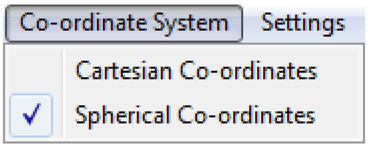


Figure 5.24: *Options on* Co-ordinate System *menu*

There are two option for the co-orduinate system

1 *Cartesian co-ordinates*.
2 *Spherical co-ordinates*
Only WGS84 is supported.

5.6 Settings menu

The following options can be accessed through the *Settings* menu, see Figure 5.25

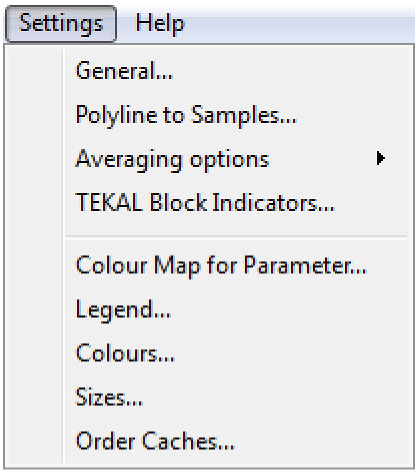


Figure 5.25: *Options on* Settings *menu*

5.6.1 General

The following parameters influence the behaviour of the operations above. They are set via the following parameter list, see Figure 5.26

◇ *Go Back to Start-up Directory Yes or No* default: 0 (No)
When navigating through the directories in the file menu, you can specify whether to always go back to the start up directory, or keep the latest directory so that you have to navigate only once.
◇ *Number of Internal Diffusion Steps* default: 200

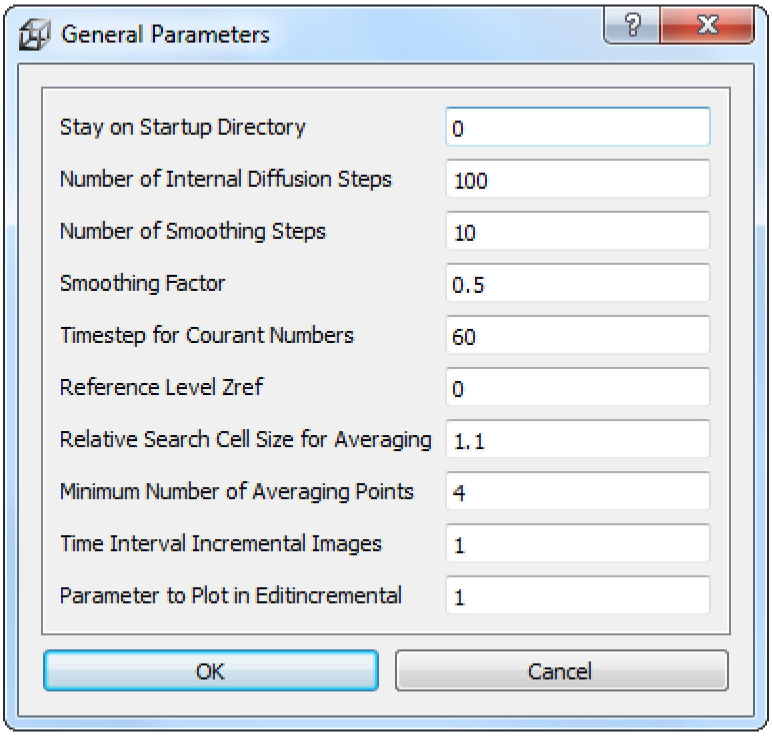


Figure 5.26: *Options on* ***Settings*** *window*

◇ *Number of Smoothing Steps* default: 10

◇ *Smoothing Factor* default: 0.05

A value of 0.0 gives no smoothing; a value of 1.0 makes a point the average of its neighbouring depth points.

◇ *Time-step for Courant Number* default: 60.0 [s]

The time step is in seconds. The Courant number is computed following the definition in the Delft3D-FLOW User Manual, valid for the ADI method in a staggered grid. The local water depth is the depth value + the Reference Level ζ_0

◇ *Reference Level Zref* ζ_0 default: 0.0 [m]

The reference level is defined as positive upward.

◇ *Relative Search Cell Size for Averaging* default: 1.1

Giving a cell search area of 1 grid cell, a value of two gives a factor 2 larger cells in both directions.

◇ *Minimum Number of Averaging Points* default: 4

◇ *Time Interval Incremental Images* default: 0.05

◇ *Parameter to Plot in Edit-incremental* default: 1

1 = water height, 2 = flow velocity, 3 = water level, 4 = U-component, 5 = V-component

◇ *Delete Polygon after Operations* default: 0 (No)

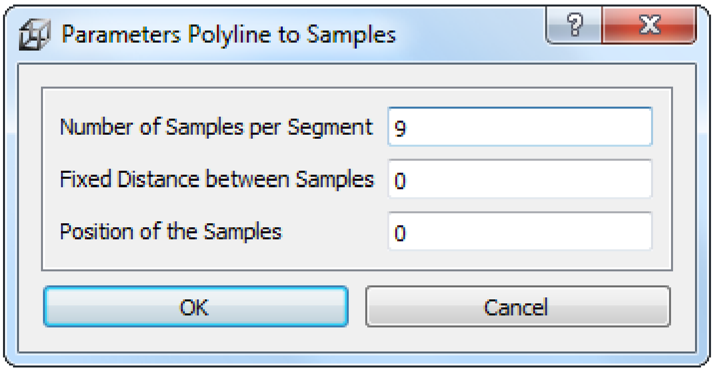


Figure 5.27: *Options on* ***Parameters Polyline to Sample*** *window*

5.6.2 Polyline to samples

In the parameter form, see Figure 5.27, you can specify the number of samples per polyline segment, the distance between the samples, and the shape of the polyline.

◇ *Number of Samples per Segment* default: 9
◇ *Fixed Distance between Samples* default: 0.0 [m]
A non-zero distance specifies the distance between the samples. If zero distance is specified, the distance depends on the segment size and the number of points per segment.
◇ *Spline Shape Yes or No* default: 0 (No)
The shape of the *Polyline to Samples* and *Polyline to Samples Channel* operations can be a straight line (0) or a spline shape (1).

5.6.3 Change colour map

When clicking on the *Settings* → *Change Colour Map* menu, a form opens in which you can select the relation between a parameter (i.e. Depth) and the loaded colour maps; see Figure 5.28

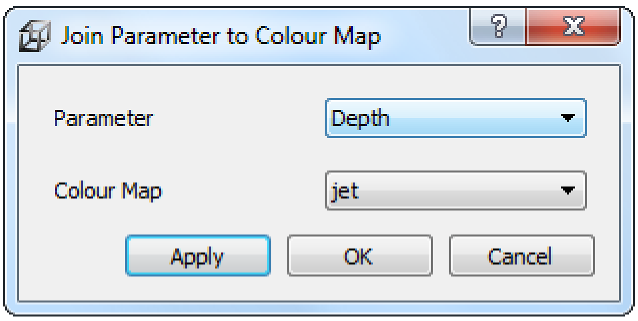


Figure 5.28: *Opions on* ***Colour Map for Parameter*** *window*

5.6.4 Legend

When clicking on the *Settings* → *Legend* menu, a form opens in which you can define how the iso-colour figures should be displayed; see Figure 5.29

◇ *Autoscale Legend* default: On
Specify whether the program should determine the isocolour values automatically, or to do it yourself. If you leave it to the program, it will determine the minimum and maximum

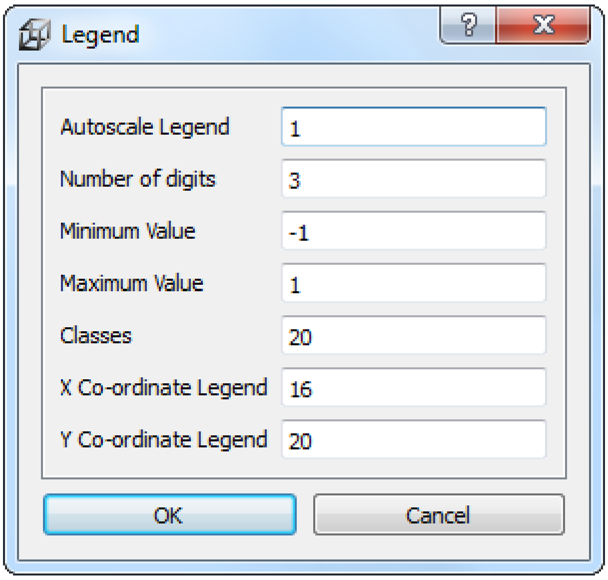


Figure 5.29: *Opions on* Settings $\rightarrow$ Colours *menu*

depth value within the viewing area and display the number of iso-colours specified above. Zooming in will always result in display of the same number of iso-colours. If you want to specify the isocolour values yourself, you have to specify one of the three parameters below. When zooming in, the iso-colour values will remain fixed.

◇ *Number of digits* default: 3
Specifying the number of digits used in the legend and on the map.

◇ *Minimum Value* default: On
Specifying this value turns autoscale off.

◇ *Maximum Value* default: On
Specifying this value turns autoscale off.

◇ *Classes* default: 20
The number of classes can be specified

◇ *X Co-ordinate Legend* default: 16
x Co-ordinate of lower left corner of legend in pixels

◇ *Y Co-ordinate Legend* default: 20
y Co-ordinate of lower left corner of legend in pixels

5.6.5 Colours

When clicking on the *Settings* $\rightarrow$ *Colours* menu, a form opens in which you can define the colours for background, land boundary, polygons, etc.; see Figure 5.30

5.6.6 Sizes

When clicking on the *Settings* $\rightarrow$ *Sizes* menu, a form opens in which you can define the linewidth and dotsize in pixels. See Figure 5.31

5.6.7 Averaging options

For all averaging methods, see Figure 5.32, the surrounding area depends on the shape of the surrounding grid cells and the parameter *Relative Search Cell Size for Averaging* (in *Settings* $\rightarrow$ *General*). The default value of 1.0 gives an area of 1.0 grid cell, a value of two gives a factor 2.0 larger cells in both directions. The shape of the area around a given grid point is defined by an eight point polygon that has the surrounding cell centres as corners and the points halfway in between the grid point and the surrounding grid points in between these corners.

- ◇ *Average Value of Near Points*
- ◇ *Value of Closest Point*
- ◇ *Maximum Value of Near Points*
- ◇ *Minimum Value of Near Points*
- ◇ *Shepard*
  The Shepard method is a weighted averaging method, with weights depending on the reciprocal of the squared distance between the grid point and the surrounding samples:

$$\overline{s} = \frac{\sum_{i=1}^{N} \frac{s_i}{d_i^2}}{\sum_{i=1}^{N} \frac{1}{d_i^2}} \tag{5.1}$$

with:

$\overline{s}$	Averaged value
N	number of samples within the polygon
d_i	distance between grid point and sample point i
s_i	value of sample point i

5.6.8 TEKAL block indicators

The next parameters (see Figure 5.33) can be specified to indicate which block of x, y and z values must be read from a TEKAL type file.

- ◇ *Block Indicator*, (4 Characters) — default: dept
- ◇ *Multiplication Factor for z Values* — default: 1.0

5.6.9 Order caches

The parameters set in the **Order caches** window, see Figure 5.34, influence the drawing order of the several items. The drawing order of the caches is: $5, 4, 3, 2, 1, 0$. Cache 5 is drawn first and cache 0 is drawn last. So the items which will drawn in cache 0 are drawn on top. If there is no need the draw a cache it will not be done, this improves the drawing performance by avoiding unnecessary drawing. So if an item is changed in cache 3 only caches $3, 2, 1$ and 0 are drawn.

- ◇ Rest — default: 0
- ◇ text — default: 0

◇ Polygons	default: 1
◇ Dredge and Dump	default: 1
◇ Dry Points, Thin Dams, Observation Points	default: 1
◇ Computational boundary	default: 1
◇ Second Depth	default: 1
◇ Active Grid	default: 2
◇ Active Depth	default: 2
◇ Samples	default: 2
◇ Land Boundary	default: 1
◇ Inactive Grids and Depths	default: 4
◇ Triangle Samples	default: 5

5.7 Help menu

On the *Help* menu, you may choose to read the user manual or the version number of QUICKIN; see Figure 5.35

5.7.1 User manual

When clicking on the *Help* → *User Manual* the user manual of QUICKIN will be displayed (file <QUICKIN _User_manual.pdf>).

5.7.2 About

When clicking on the *Help* → *About* a window will display the current version number of QUICKIN.

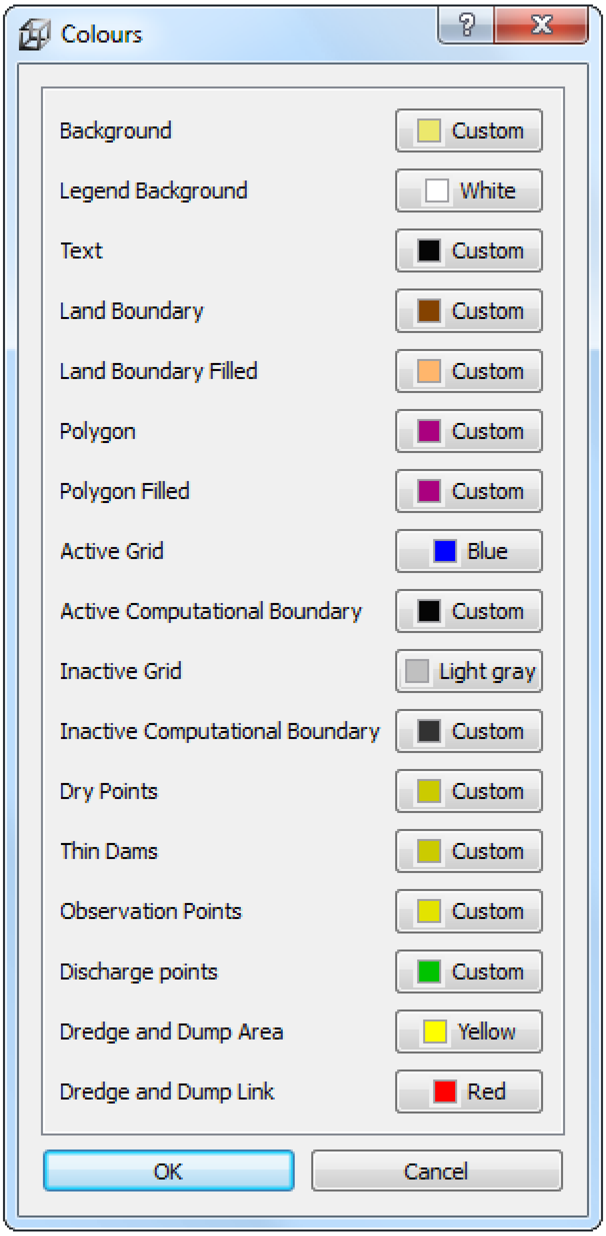


Figure 5.30: *Opions on* Settings → Colours *menu*

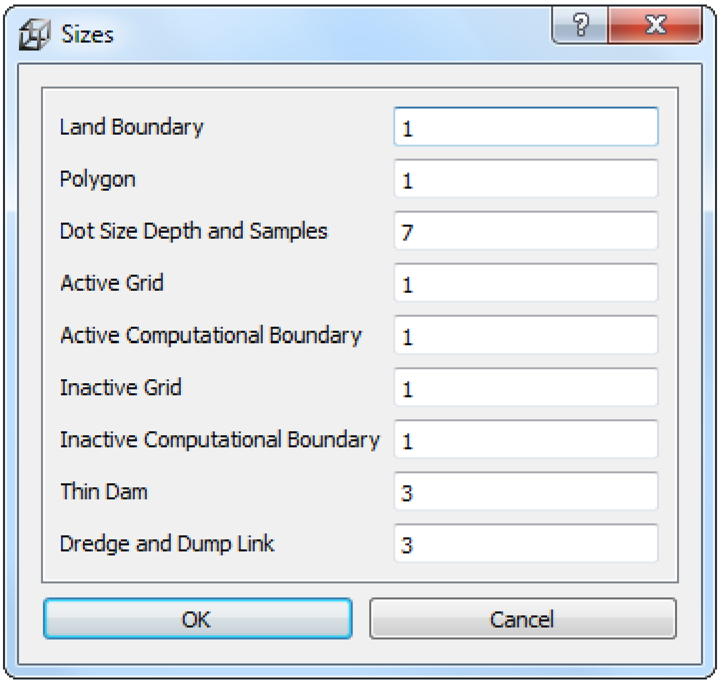


Figure 5.31: *Opions on* Settings → Sizes *menu*

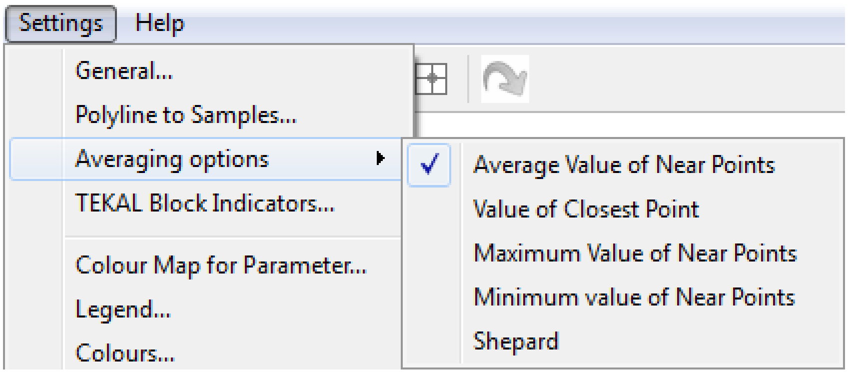


Figure 5.32: *Options on* Averaging Options *menu*

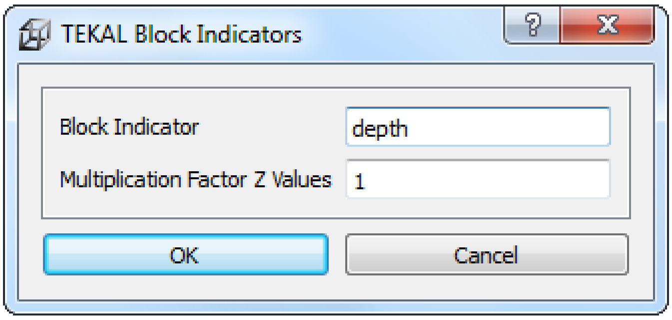


Figure 5.33: *Options on* ***TEKAL Block Indicator*** *window*

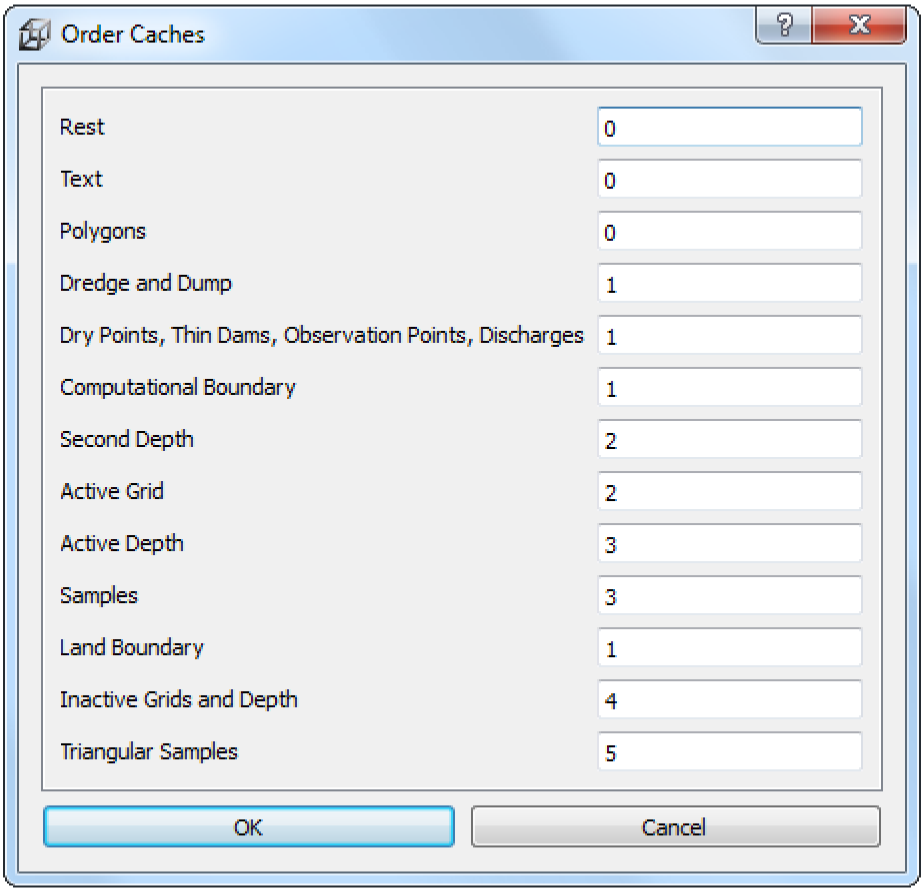


Figure 5.34: *Options on* ***Order Caches*** *window*

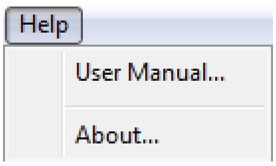


Figure 5.35: *Opions on* Help *menu*

6 Tutorial

6.1 Interpolation

6.1.1 Open grid and land boundary

For details, see Sections 3.2 and 3.3

- ◇ Start Delft3D and click *Select working directory* to enter the <tutorial\quickin\maracaibo> directory.
- ◇ Start QUICKIN.

For Lake Maracaibo we will exercise generating a bathymetry from samples for Delft3D-FLOW.

- ◇ On the *File* menu, select *Attribute Files* and click *Open Land Boundary. . . .*
- ◇ Select and open the file <mar_01.ldb> that contains an outline of the land boundary.
- ◇ On the *File* menu, select *Import* and click *Grid (RGFGRID). . .* ; open <mar_02.grd>

The land boundary and grid look like in Figure 6.1.

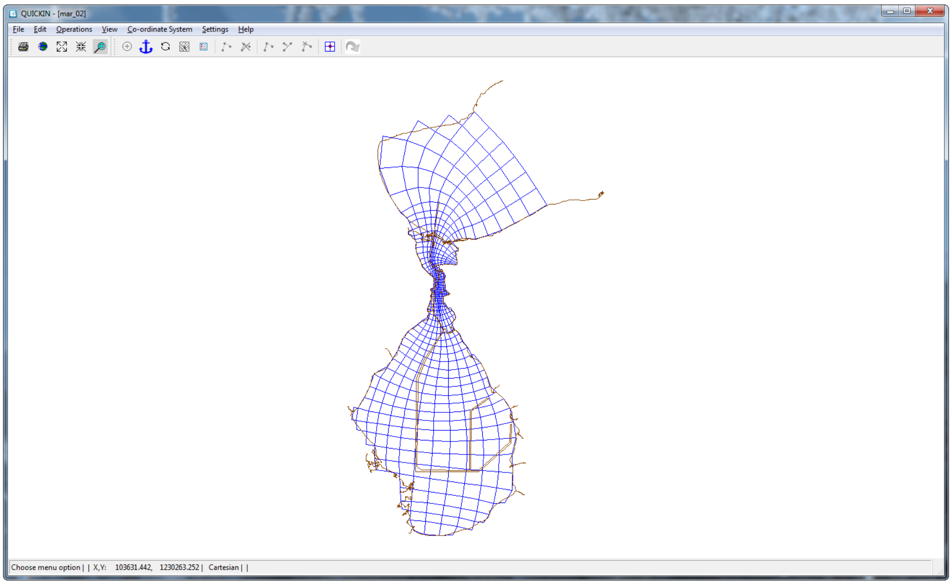


Figure 6.1: *Land boundary and grid for Lake Maracaibo*

In Delft3D-FLOW, the water level is defined in the centre of a grid cell that is surrounded by four grid points. In the next steps, the model bathymetry at the grid points will be created by interpolating the available depth samples.

6.1.2 Open samples

- ◇ On the *File* menu, select *Attribute Files* and click *Open Samples. . . .*
- ◇ Open the scattered samples file <mar_02.xyz>.

Note: that the sample density is high in the centre part and low in the lower part. In low sample density areas, triangulation is more suited, in high density areas, averaging is more suited, see Figure 6.2.

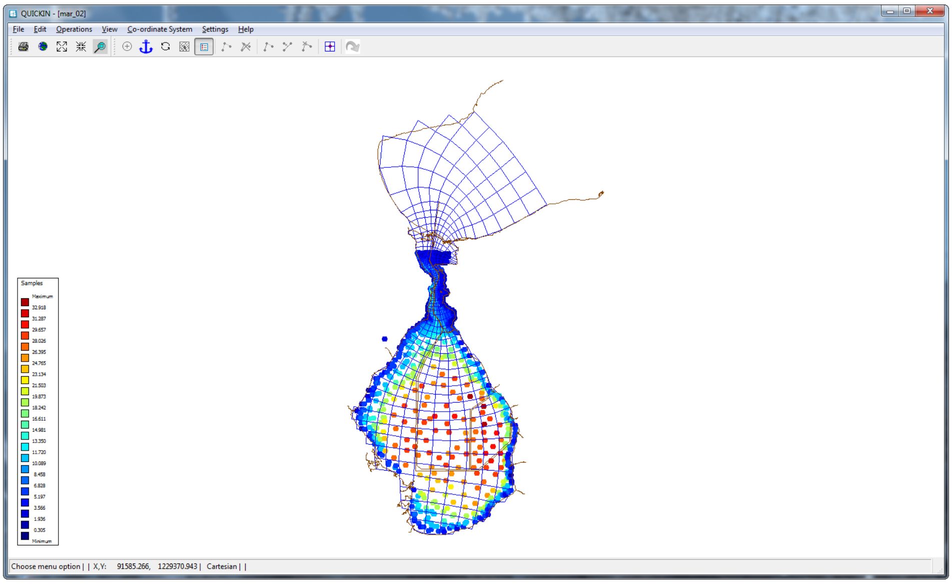


Figure 6.2: *Samples covering the Lake Maracaibo model area*

6.1.3 Zoom in and out

To zoom in or out several facilities are available:

◇ On the toolbar click  to zoom in or  to zoom out.
◇ Press the + or - key while keeping the `CTRL`-key pressed.
◇ Use the mouse scroll wheel.

To zoom in on a specific area

◇ Click  on the toolbar and drag a box around the area.
◇ To zoom out to full extent, click  on the toolbar.

To pane the grid: keep the `CTRL`-key pressed and move around with the cursor. Hereafter you will interpolate the sample data onto the grid.

6.1.4 High and low sample density

Areas with a high sample density need an other interpolation method then areas with a low sample density. In general grid cell averaging is used in areas with a high sample density, and triangulation is used in areas with a low sample density. High sample density is obtained where there are a few or more sample points in the direct neighbourhood of a grid point, low density is obtained when there is in average less then 1 sample point in the direct neighbourhood.

High sample density

The dense part is best interpolated by an averaging procedure, so that all samples within the direct surroundings of a grid point (see Figure 6.3) are taken into account to get the best volume for the grid cell.

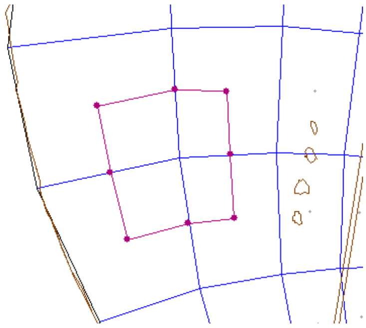


Figure 6.3: *Default relative search cell for grid cell averaging*

The default relative search area is defined by the centres of the surrounding grid cells, and the mids of the grid cell faces, see Figure 6.3. The minimum number of samples for grid cell averaging is default 4. If there are less samples in the search area, no depth value is calculated. Moreover, if there are not enough samples available, the depth should be interpolated by triangulation. The relative search area and minimum number of averaging points can be changed in *Settings* → *General*.

- ◇ Create a polygon by clicking  on the toolbar and next by clicking the left mouse around the interpolation area, see Figure 6.4.
- ◇ Click the right-mouse button to close the polygon and leave the polygon edit mode.
- ◇ On the *Operations* menu, click *Grid Cell Averaging* to perform the averaging within the previously defined polygon.

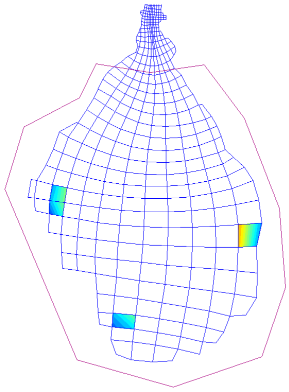


Figure 6.4: *Grid cell averaging result within a pre-defined polygon*

Low sample density

The sparse part is best interpolated by a triangulation method (*Operations* → *Triangular Interpolation*). For the moment hide the samples: *View* → *Samples* → *No Samples*. After a grid cell averaging method and a triangulation most of the depth points did get a value, see Figure 6.5.

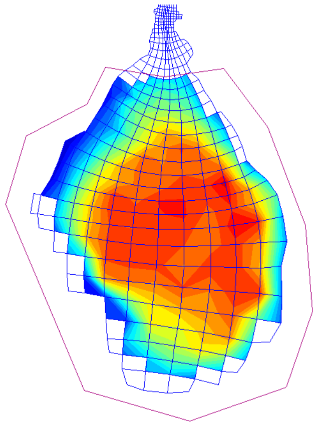


Figure 6.5: *Interpolation results after applying grid cell averaging and triangulation*

Remarks:
- ◇ Only grid points inside the polygon are taken into account for the operation.
- ◇ Only samples inside the polygon are taken into account for the operation.
- ◇ If a grid point has already a depth value assigned, the operation will not overrule this value.
- ◇ If you do not use a polygon to define a limited area, all grid points and samples will be part of the operation.

6.1.5 Points outside area of influence

The result of the previous section is shown in Figure 6.5. Note that grid points that are outside the polygon are excluded from the operation.

As long as not every grid point has a depth value, you are not finished with generating a depth file.

Points that were not interpolated yet

First remove the polygon.

- ◇ On the *Operations* menu, select *Delete* and click *Polygons*, or click on the toolbar.
- ◇ On the *View* menu, select *Depth* and click *Yet to be Found* to see which grid points still have to be assigned a depth value.
- ◇ Edit a new polygon as shown in Figure 6.6.
- ◇ On the *Operations* menu, select *Internal Diffusion* to fill in these missing points within the polygon area. After selecting *View* → *Depth* → *Continuous Shade* from the menubar you obtain Figure 6.7.

To inspect the the batymetry experiment with behaviour of the **Delft3D-3D View** window.

- ◇ Click *View* → *3D View* to obtain Figure 6.8

To change the view on the data the mouse buttons will react according which keys are hit.

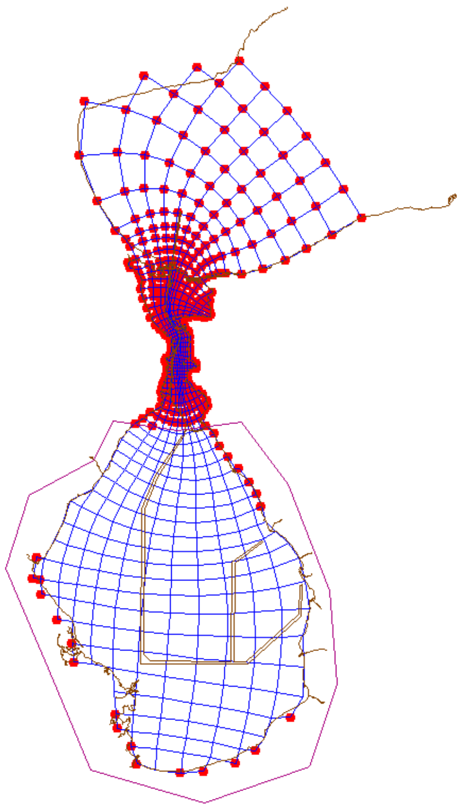


Figure 6.6: *Display of Points Yet to be Found*

The following key-strokes are available.
`c`: Switch rendering mode
`h`: Toggle help
`i`: Inverse depth
`r`: Reset view
`s`: Toggle samples
`x`: Increase depth
`z`: Decrease depth
left mouse: Rotate in xy-plane
middle mouse: zoom
right mouse: rotate z-axis
`CTRL`+left mouse: move origin xy-plane

6.2 Save depth

To save the depth values at the grid points

- ◇ On the *File* menu, select *Export* and click *Depth*.

The file window **Save Depth** opens, see Figure 6.9.

- ◇ Enter <mar_01> and click *Save* to save the depth file.

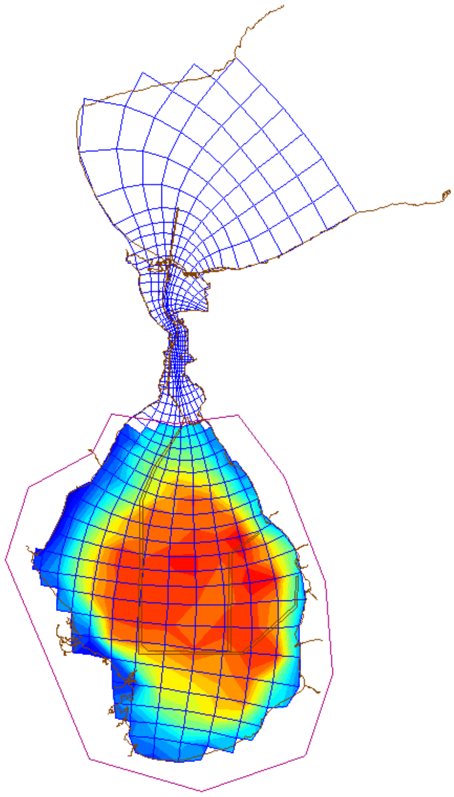


Figure 6.7: *Continuous shade display, all grid points inside the polygon have a depth value*

6.3 Dredge and dump areas

Start QUICKIN in the <tutorial\quickin\dredge_and_dump> directory.

◇ On the *File* menu, select *Attribute Files* and click *Open Land Boundary. . . .*
◇ Select and open file <manukau.ldb> that contains an outline of the land boundary.

You can fill the land boundary polygon by selecting the menu option *Settings* → *Colours* and set the appropriate colours for *Land Boundary*, *Land Boundary Filled* and select menu item *View* → *Land Boundary* → *Filled*.

Now we will define some dredge and dump areas with polygons.

◇ On the *Edit* menu, click *Dredge and Dump Areas* and *New Area*

Define the polygons like in Figure 6.10, finish a polygon by clicking the right mouse button.

Renaming the default polygon names can be done by editing the properties of an area. Select *Edit* → *Dredge and Dump Areas* → *Edit Properties*, click in an area and change the polygon names as listed below.

◇ Rename the polygon in the north-east of the harbour "dredge 1"
◇ Rename the polygon in the entrance of the harbour "dredge 2"
◇ Rename the polygon in the entrance of the harbour "dredge 3"

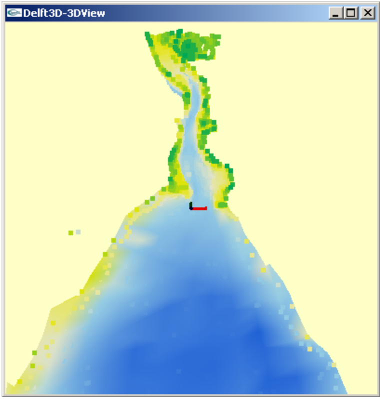


Figure 6.8: *Perspective view on the bathymetry*

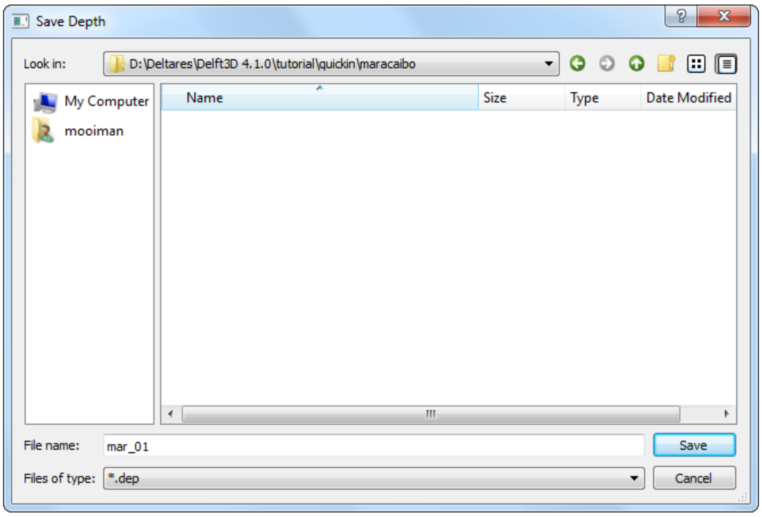


***Figure 6.9: Save Depth** window to save the depth files*

◇ Rename the northerly polygon at sea "dump 1"
◇ Rename the southerly polygon at sea "dump 2"

You will see that the the polygons are filled in and the names are shown at the mass-centre of the polygon, see Figure 6.11.

◇ When finished point to *Files* → *Delft3D-FLOW Attribute files* → *Save Dredge and Dump Areas* to save dredge and dump areas to the files <∗.pol> and <∗.dad>.

To define the links that connect dredge to dump areas, first press *Edit* → *Dredge and Dump Areas* → *New Link*. Next, click inside a dredge polygon, followed by a click inside a dump polygon.

◇ Connect 'dredge 1' to 'dump 1'.
◇ Connect 'dredge 2' to 'dump 1'.
◇ Connect 'dredge 3' to 'dump 2'.

The result is shown in Figure 6.12.

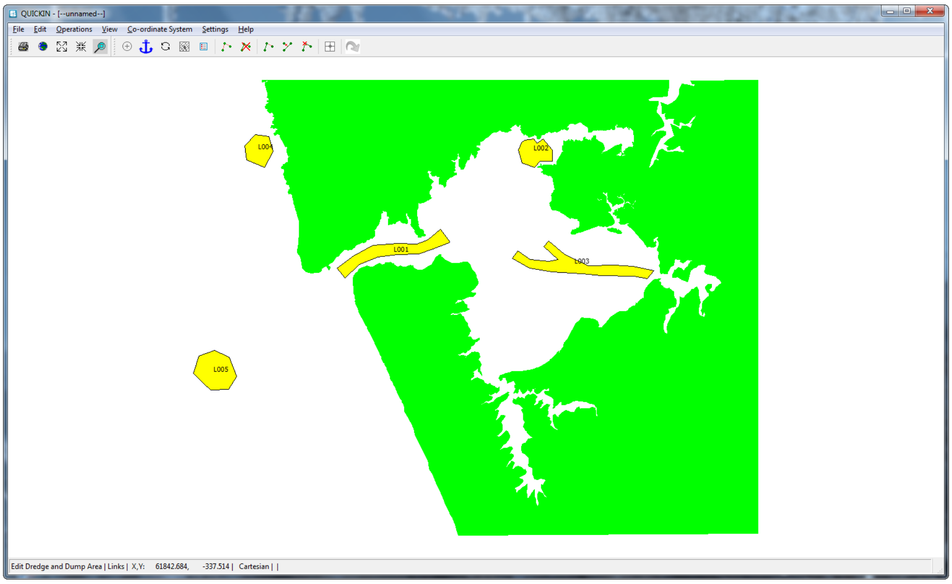


Figure 6.10: *Location of dredge and dump areas*

If you want to delete a link, first select *Edit* → *Dredge and Dump Areas* → *Delete Link*, and next click the dredge depth field of the link to be deleted.

To edit the dredging properties, select the *Edit* → *Dredge and Dump Areas* → *Edit Properties*. Now you can edit the properties of the dredge and dump area by clicking them and typing the requested fields, see Figure 6.13.

◇ Set the dredge depth in the areas 'dredge 1' and 'dredge 2' to "10.0 m", and in 'dredge 3' to "20.0 m", see Figure 6.13.

◇ On the *File* menu, point to *Delft3D-FLOW Attributes* and click *Save Dredge and Dump Areas*, and save the dredge and dump characteristics to the files <option_1_characteristics.pol> and <option_1_characteristics.dad>.

If you already have saved the polygons, ignore the save question. If not, save the polygons.

◇ Exit QUICKIN.

6.4 Flooding animation

This functionality only applies for users of Delft1D2D.

Start QUICKIN in directory <tutorial\quickin\flooding_animation>.

◇ On the *File* menu, select *Import* and click *Open Grid and Depth (Arc)* and open <atiel.aht>.
◇ On the *File* menu, select *Import* and click *Grid (Incremental)* and open the incremental file <tlt.inc>.
◇ On the *Settings* menu, click *Colour Map for Parameters*. Join the *Depth* parameter to the *bathymetry (inverse)* colour map and join the *Flooding* parameter to the *jet* colour map , see Figure 6.14.

◇ Press the spacebar to start/stop the animation. See Figure 6.15 for a certain time in the

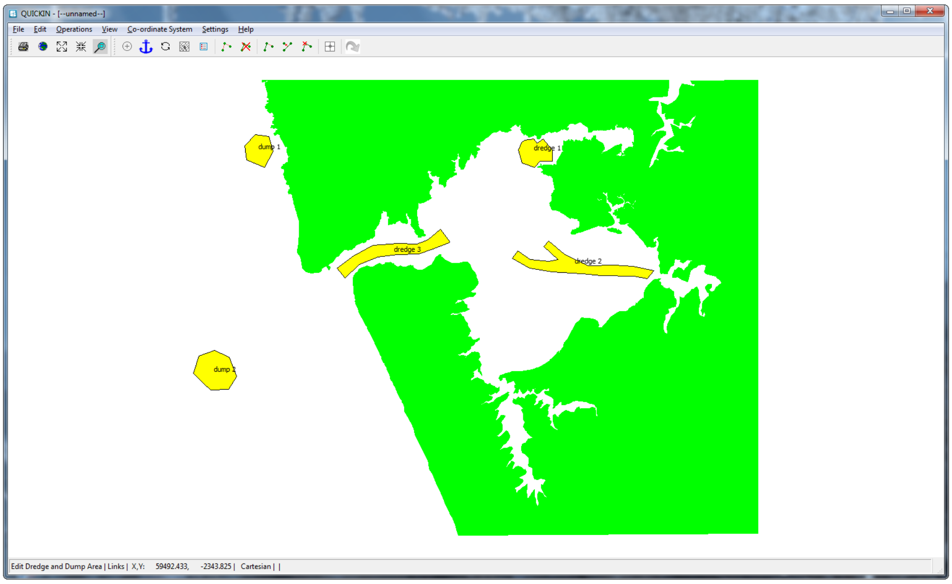


Figure 6.11: *Edit Dredge and Dump, with polygon names*

flooding animation.

◇ Exit QUICKIN.

6.5 QUICKIN in the ArcMap environment

In this case you should be familiar with using co-ordinate systems in ArcMap.

ArcMap layers (e.g. shape files, SDE layers) most times have also information about the co-ordinate system (spatial reference). If not available, ArcMap marks this as unknown. On the other hand, if it is known to you, spatial reference can be added to these layers by the program ArcCatalog.

If the layers in ArcMap have a projected co-ordinate system or probably an unknown co-ordinate system, then you can use *Cartesian* in QUICKIN. It is your responsibility the co-ordinates have the unit metres. You can see the used (projected) co-ordinate system of the layers and of the data frame via the properties of the data frame. For QUICKIN the co-ordinate system of the data frame is leading. As you know, ArcMap has possibilities to set the co-ordinate system of the data frame, while layers have different co-ordinate systems.

If you want to use spherical co-ordinates in QUICKIN while using ArcMap, the co-ordinate system of the data frame must be WGS84 (in ArcMap it has the name CGS_WGS_1984). This will be the case when all layers have this co-ordinate system.

If you are familiar with ArcMap you can have one or more layers with different co-ordinate systems and select (import) the WGS84 system for the data frame. Figure 6.16 shows the properties window of de data frame, <custom> give the co-ordinate system of the data frame.

You start loading layers or an <∗.mxd> file in ArcMap. The co-ordinate system of the data frame must be as described above. You will see that ArcMap displays the values of longitude and latitude as plane co-ordinates in degrees.

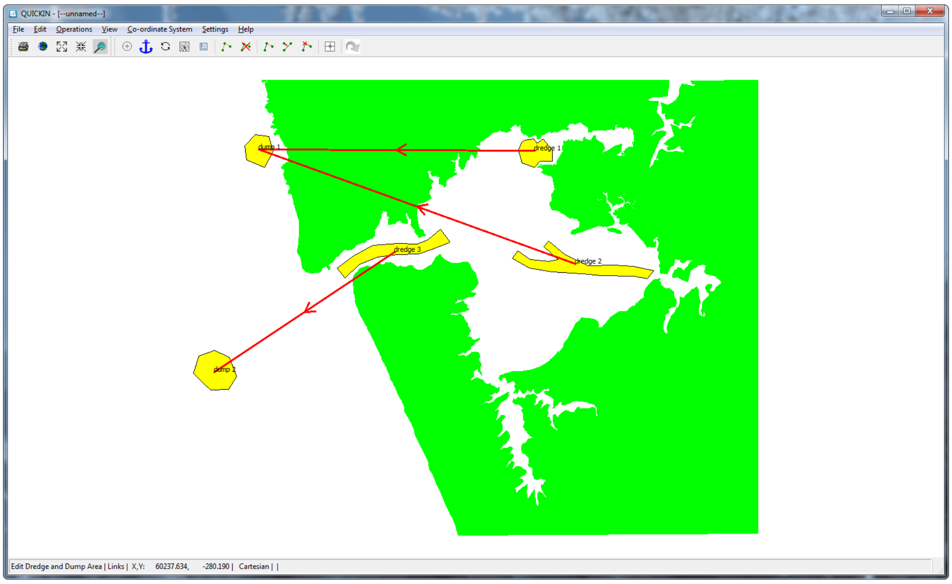


Figure 6.12: *Dredge and Dump areas with their links*

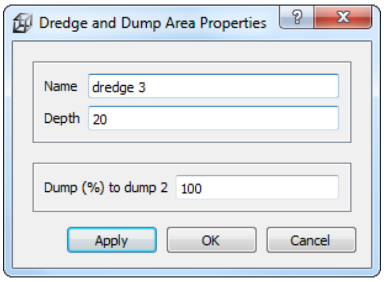


Figure 6.13: *Change the dredge and dump area properties*

Then you can start using the commands and functions of QUICKIN. The program has a menu item to change into projected co-ordinates. When using this command, the layers in ArcMap are displayed as projected and also the grid, polygons, samples of the program.

When using the menu item *View* → *Spherical Co-ordinates* → *Plane Co-ordinates*, both the layers in ArcMap and the objects of QUICKIN are displayed as plane co-ordinates.

Centring the screen in stereographic mode

In the standalone version of QUICKIN, the visualisation using the stereographic projection method always uses the centre of the screen as the point where the screen touches the sphere. This is more difficult to realise when working within the GIS environment because screen handling now is governed by the GIS system. Therefore a special function has been implemented to perform this task of centring the computer screen. This function can be invoked by clicking the menu item *Operations* → *Change Centre of Projection*, see Figure 6.17. When using this command the centre of the projection is set to the centre of the screen. This action needs recalculation of the projection and a new screen refresh. By using this command you are aware of changing the screen. When using zoom in, zoom out, pan, the centre of the projection does not change. So there is more performance and a smooth screen-refresh in this case.

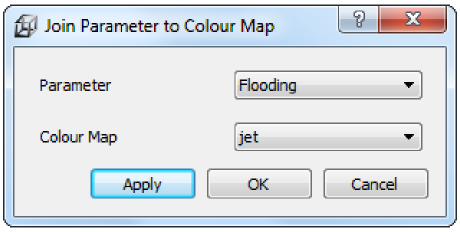


Figure 6.14: *Jooin parameter Flooding to colour map jet*

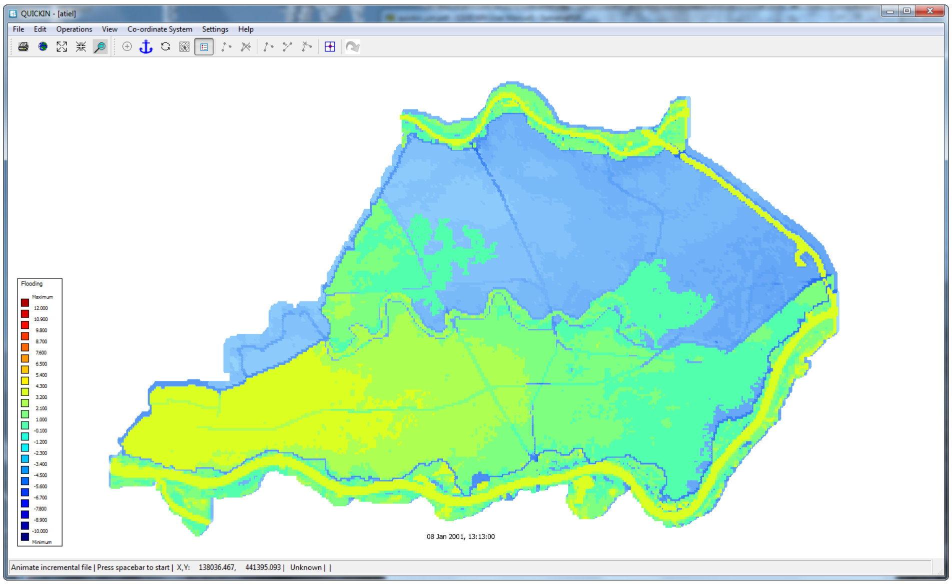


Figure 6.15: *Example of a flooding image*

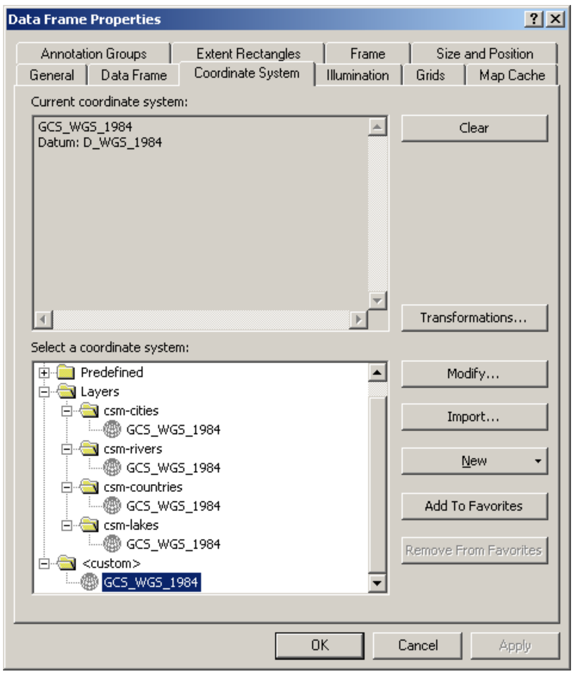


Figure 6.16: *ARC-GIS data frame properties form*

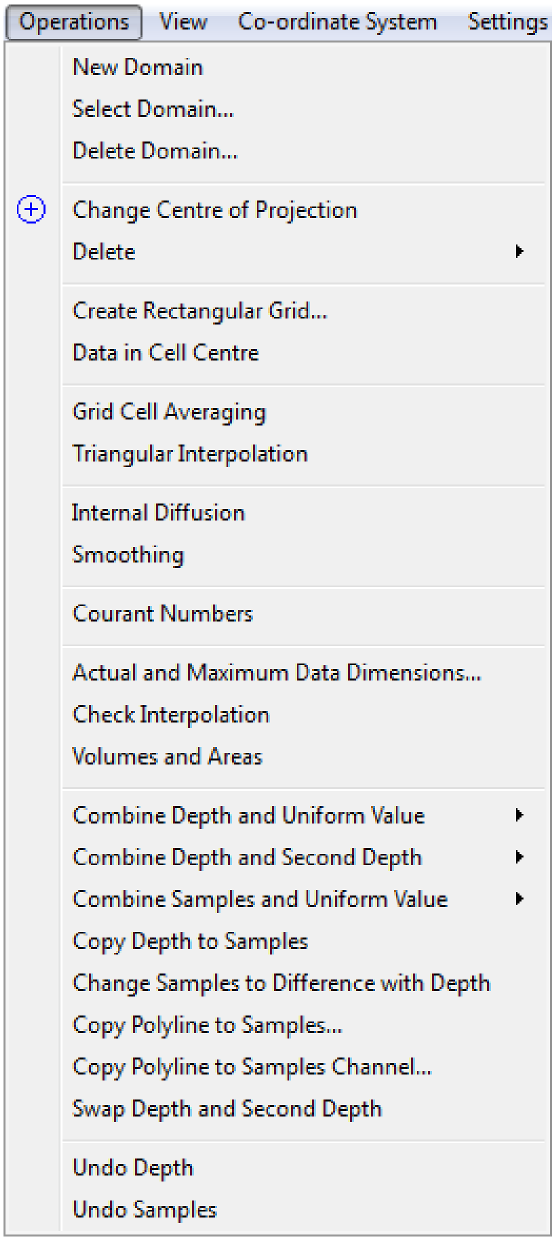


Figure 6.17: *Options on the* Operations *menu*

References

Delft1D2D, 2002. *Delft1D2D User Manual*. Deltares, 0.00 ed. 1, 5

Delft3D-FLOW, 2013. *Delft3D-FLOW User Manual*. Deltares, 3.14 ed. 1, 25, 41

GPP, 2013. *Delft3D-GPP User Manual*. Deltares, 2.14 ed. 24

QUICKPLOT, 2013. *Delft3D-QUICKPLOT User Manual*. Deltares, 2.14 ed. 24

RGFGRID, 2013. *Delft3D-RGFGRID User Manual*. Deltares, 4.00 ed. 1

SWAN, 2000. *SWAN Cycle III version 40.11 User Manual (not the short version)*. Delft University of Technology, Delft, The Netherlands, 0.00 ed. 5

A Files of QUICKIN

In the following sections we describe the attribute files used in QUICKIN.

For each file which can handled by QUICKIN we give the following information:

- File contents.
- Filetype (free formatted, fix formatted or unformatted).
- Filename and extension.
- Generated by (i. e. how to generate the file).
- Restrictions on the file contents.
- Example(s).

Remarks:

- The access mode of all attribute files is sequential.
- In the examples the file content is printed in font Courier and comment (not included in the file) between curly brackets font, unless explicitly stated differently.

A.1 Delft3D project file

File contents	Domain input for a model.
Filetype	ASCII
File format	Free formatted.
Filename	<*name*.d3d>
Generated	RGFGRID, QUICKIN, D-Waq DIDO, or manually offline

Record description:

A header block containing general information and then for each domain a detailed description.

Keyword	Description
FileInformation	
FileCreatedBy	Version string of the program who generated this file the first time
FileCreationDate	Creation date and time
FileVersion	Version number of <∗.d3d> file
Geometry	
LandBoundaryName	Name of the file with the land boundaries
LandBoundaryFormat	Format of the land boundary file, possible values are: TEKAL, NETCDF or SHAPEFILE. The NetCDF file is according the 'World Vector Shoreline' format
DDBound	
FileDDBound	Name of the file with the domain decomposition boundaries

For each grid

Keyword	Description
Grid	
Type	Format of the grid file, possible values are: RGF, RGF_NETCDF, DFLOW_FM, TELEMAC
FileName	Name of grid file with the geographical co-ordinates
FlowDepth	Name of the file containing the depth values at the cell corners of the grid
Aggregation	Name of the aggregation file

Restriction:

◇ The maximum record length in the file is 132.

Example:

The model friesian_tidal_inlet contains three different subdomains (f01, f02, f03) and the project file has the name <friesian_tidal_inlet.d3d.>

```
[FileInformation]
  FileGeneratedBy  = Deltares, Delft3D-DIDO Version 4.04.00.11836M, Jun 21 2010, 12:09:34
  FileCreationDate = 2010-06-21, 13:35:22
  FileVersion      = 0.03
[DDBound]
  FileDDBound = f34-123.ddb
```

```
[Grid]
  Type             = RGF
  FileName         = f01.grd
  Aggregation      = f34_dd-f01.dwq
  Monitoring Areas = f34_dd-f01.dmo
[Grid]
  Type             = RGF
  FileName         = f02.grd
  Aggregation      = f34_dd-f02.dwq
  Monitoring Areas = f34_dd-f02.dmo
[Grid]
  Type             = RGF
  FileName         = f03.grd
  Aggregation      = f34_dd-f03.dwq
  Monitoring Areas = f34_dd-f03.dmo
```

A.2 Land boundary file

File contents	The co-ordinates of one or more polylines. Each polyline (piecewise linear) is written in a single block of data.
Filetype	ASCII
File format	Free formatted
Filename	<*name*.ldb>
Generated	RGFGRID, QUICKIN, etc

Record description:

Record	Record description
	Preceding description records, starting with an asterisk ($*$), and will be ignored.
1	A non blank character string, starting in column one
2	Two integers representing the numbers of rows and number of columns for this block of data
	Two reals representing the x, y or λ, ϕ-co-ordinate

Example:

```
*
* Polyline L007
*
L007
6  2
            132400.0    549045.0
            132345.0    549030.0
            132165.0    549285.0
            131940.0    549550.0
            131820.0    549670.0
            131585.0    549520.0
*
* Polyline L008
*
L008
```

```
4  2
            131595.0    549685.0
            131750.0    549865.0
            131595.0    550025.0
            131415.0    550175.0
*
* Polyline L009
*
L009
6  2
            131595.0    549655.0
            148975.0    564595.0
            150000.0    564935.0
            152105.0    565500.0
            153150.0    566375.0
            154565.0    567735.0
```

Remark:

◇ In case this file is read as a polygon file then the polylines are closed by QUICKIN to get a polygon.

A.3 Sample file

File contents	The location and value of samples.
Filetype	ASCII
File format	Free formatted
Filename	<*name*.xyz>
Generated	Manually or Offline with QUICKIN and data from digitised charts or GIS-database.

Record description:

Filetype	Record description
Free formatted	Location and sample value per row Two reals representing the x, y or λ, ϕ-coordinate and one real representing the sample value

Example:

Sample file with 12 sample values with their location (free formatted file).

```
   213813.2       603732.1      -4.053000
   214686.0       607226.1      -4.522000
   214891.7       610751.2      -5.000000
   210330.8       601424.1      -2.169000
   211798.0       604444.8      -2.499000
   212460.0       607475.7      -2.760000
   212436.9       610362.5      -2.865000
   185535.4       606607.9       1.360000
   186353.0       603789.4       1.122000
   187959.2       601197.6      0.9050000
   190193.0       599101.5      0.7050000
   208578.7       602513.7     -0.7990000
```

A.4 Polygon file

File contents	The co-ordinates of one or more polygons. Each polygon is written in a single block of data
Filetype	ASCII
File format	Free formatted
Filename	<*name*.pol>
Generated	RGFGRID, QUICKIN, D-Waq DIDO, etc

Record description:

The file may contain one or more polygons. For every polygon the file should contain a line indicating the name of the polygon, followed by a line indicating the number of points making up the polygon and the number of coordinates, i.e. 2, finally followed by the coordinate data.

Record	Record description
	Preceding description records, starting with an asterisk ($*$), and will be ignored.
1	A non blank character string, starting in column one
2	Two integers representing the numbers of rows and number of columns for this block of data
	Two reals representing the x, y or λ, ϕ-coordinate

Restriction:

◇ The first record and the last record in the block should be the same

Example:

```
*
* Deltares, Delft3D-DIDO Version 3.39.01.4423:4459, Sep 25 2008, 20:10:54
* 2008-09-25, 22:11:08
*
Observation Area 001
       5   2
   1.8768018E+05   6.1708738E+05
   1.8996981E+05   6.1001035E+05
   1.9746314E+05   6.1266423E+05
   1.9480925E+05   6.1838830E+05
   1.8768018E+05   6.1708738E+05
Observation Area 002
       5   2
   2.0011703E+05   6.1818015E+05
   1.9819166E+05   6.1063479E+05
   2.0568498E+05   6.0870942E+05
   2.0797461E+05   6.1599460E+05
   2.0011703E+05   6.1818015E+05
Observation Area 003
       5   2
   1.9340425E+05   6.1396516E+05
   2.0183425E+05   6.1365294E+05
   1.9944054E+05   6.0558720E+05
   1.9522555E+05   6.0595146E+05
```

```
1.9340425E+05   6.1396516E+05
```

A.5 Orthogonal curvilinear grid file

File contents	The co-ordinates of the orthogonal curvilinear grid at the depth points.
Filetype	ASCII
File format	Free formatted
Filename	<*name*.grd>
Generated	RGFGRID

Record description:

Record	**Record description**
	Preceding description records, starting with an asterisk ($*$), will be ignored.
1	Record with `Co-ordinate System = Cartesian` or value `Spherical`
2	The number of grid *points* in m- and n-direction (2 integers).
3	Three real values (not used).
4 to K+3	A label and record number, the x-component of the world co-ordinates of all points in m-direction, starting with row 1 to row $nmax$, with as many continuation records as required by $mmax$ and the number of co-ordinates per record. The label and record number are suppressed on the continuation lines. This set of records is repeated for each row until $n = nmax$.
K+4 to 2K+3	A similar set of records for the y-component of the world co-ordinates.

K is the number of records to specify for all grid points a set of x- or y-co-ordinates.

Restrictions:
- ◇ The grid must be orthogonal.
- ◇ Input items in a record are separated by one or more blanks.

Example:

```
*
* Deltares, Delft3D-RGFGRID  Version 4.16.01.4531, Sep 30 2008, 23:32:27
* File creation date: 2008-10-01, 23:19:22
*
Coordinate System = Cartesian
       9       7
 0 0 0
 Eta=    1   0.00000000000000000E+00   1.00000000000000000E+02   2.000000...
             5.00000000000000000E+02   6.00000000000000000E+02   7.000000...
 Eta=    2   0.00000000000000000E+00   1.00000000000000000E+02   2.000000...
             5.00000000000000000E+02   6.00000000000000000E+02   7.000000...
```

```
Eta=   3   0.00000000000000000E+00   1.00000000000000000E+02   2.000000...
           5.00000000000000000E+02   6.00000000000000000E+02   7.000000...
Eta=   4   0.00000000000000000E+00   1.00000000000000000E+02   2.000000...
           5.00000000000000000E+02   6.00000000000000000E+02   7.000000...
Eta=   5   0.00000000000000000E+00   1.00000000000000000E+02   2.000000...
           5.00000000000000000E+02   6.00000000000000000E+02   7.000000...
Eta=   6   0.00000000000000000E+00   1.00000000000000000E+02   2.000000...
           5.00000000000000000E+02   6.00000000000000000E+02   7.000000...
Eta=   7   0.00000000000000000E+00   1.00000000000000000E+02   2.000000...
           5.00000000000000000E+02   6.00000000000000000E+02   7.000000...
Eta=   1   1.00000000000000000E+02   1.00000000000000000E+02   1.000000...
           1.00000000000000000E+02   1.00000000000000000E+02   1.000000...
Eta=   2   2.00000000000000000E+02   2.00000000000000000E+02   2.000000...
           2.00000000000000000E+02   2.00000000000000000E+02   2.000000...
Eta=   3   3.00000000000000000E+02   3.00000000000000000E+02   3.000000...
           3.00000000000000000E+02   3.00000000000000000E+02   3.000000...
Eta=   4   4.00000000000000000E+02   4.00000000000000000E+02   4.000000...
           4.00000000000000000E+02   4.00000000000000000E+02   4.000000...
Eta=   5   5.00000000000000000E+02   5.00000000000000000E+02   5.000000...
           5.00000000000000000E+02   5.00000000000000000E+02   5.000000...
Eta=   6   6.00000000000000000E+02   6.00000000000000000E+02   6.000000...
           6.00000000000000000E+02   6.00000000000000000E+02   6.000000...
Eta=   7   7.00000000000000000E+02   7.00000000000000000E+02   7.000000...
           7.00000000000000000E+02   7.00000000000000000E+02   7.000000...
```

A.6 Bathymetry file

File contents	The bathymetry in the model area, represented by depth values (in metres) for all grid points.
Filetype	ASCII
File format	Free formatted or unformatted
Filename	<*name*.dep>
Generated	FLOW-GUI (only for uniform depth values). Offline with QUICKIN and data from digitised charts or GIS-database.

Record description:

Filetype	**Record description**
Free formatted	Depth values per row, starting at N = 1 to N = Nmax, separated by one or more blanks. The number of continuation lines is determined by the number of grid points per row (Mmax) and the maximum record size of 132.
Unformatted	Mmax depth values per row for N = 1 to N = Nmax.

Restrictions:

- ◇ The file contains one M and N line more than the grid dimension.
- ◇ The maximum record length in the free formatted file is 132.
- ◇ Depth values from the file will not be checked against their domain.
- ◇ The input items are separated by one or more blanks (free formatted file only).
- ◇ The default missing value is: -999.0

Example:

File containing $16 * 8$ data values for a model area with $15 * 7$ grid points (free formatted file).

```
   1.0    2.0    3.0    4.0   -5.0   -5.0   -5.0    8.0    9.0   10.0   11.0
  12.0   13.0   14.0   -5.0 -999.0
   3.0    4.0    5.0    6.0    7.0   -6.0   -6.0   10.0   11.0   12.0   13.0
  14.0   15.0   16.0   17.0 -999.0
   5.0    6.0    7.0    8.0    9.0   10.0   -7.0   12.0   13.0   14.0   15.0
  16.0   17.0   18.0   19.0 -999.0
   7.0    8.0    9.0   10.0   11.0   12.0   13.0   14.0   15.0   16.0   17.0
  18.0   19.0   -7.0   19.0 -999.0
   9.0   10.0   11.0   12.0   13.0   14.0   15.0   16.0   17.0   18.0   19.0
  20.0   19.0   18.0   17.0 -999.0
  -7.0   12.0   13.0   14.0   15.0   16.0   17.0   18.0   19.0   20.0   19.0
  18.0   17.0   16.0   15.0 -999.0
  -8.0   -8.0   15.0   16.0   17.0   18.0   19.0   20.0   19.0   18.0   17.0
  16.0   15.0   14.0   13.0 -999.0
-999.0 -999.0 -999.0 -999.0 -999.0 -999.0 -999.0 -999.0 -999.0 -999.0 -999.0
-999.0 -999.0 -999.0 -999.0 -999.0
```

A.7 Grid and depth file

File contents	The rectilinear grid and bathymetry at cell centres in the model area, represented by depth values (in metres) for all grid cells.
Filetype	ASCII
File format	Free formatted
Filename	<*name*.asc>
Generated	Extern program

Header description for the Arc-files:

Keywords	**Value**	**Description**
`ncols`	*free*	number of columns used for datafield
`nrows`	*free*	number of rows used for datafield
`xllcorner`	*free*	x-co-ordinate of lower left corner of lower left grid cell
`yllcorner`	*free*	y-co-ordinate of lower left corner of lower left grid cell
`xllcenter`	*free*	x-co-ordinate of centre of lower left grid cell
`yllcenter`	*free*	y-co-ordinate of centre of lower left grid cell
`cellsize`	*free*	gridsize in x-direction and y-direction
`nodata_value`	*free*	value used for input that is to be neglected

Record description:

Filetype	**Record description**
Free formatted	`Ncols` values per row, starting at $N = 1$ to $N = Ncols$, separated by one or more blanks. The number of continuation lines is equal to `nrows`, starting from $N = Nrows$ to $N = 1$.

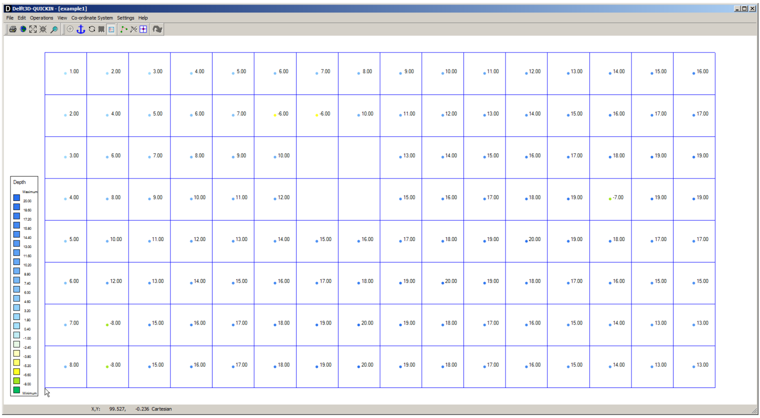


Figure A.1: *Example 1, lower left corner is located at* $(100, 0)$

Examples

Two examples are given, one example defining the lower left co-ordinate of the grid and one defining the lower left corner at the data point. Both examples has a data file containing $16 * 8$ data values. The first example has its first data value located at $(x, y) = (125, 25)$ and grid corner at $(x, y) = (100, 0)$ (see Figure A.1), and the second example has its first data value located at $(x, y) = (100, 0)$ and grid corner at $(x, y) = (75, -25)$ (see Figure A.2)

Example 1:

```
ncols           16
nrows            8
xllcorner      100
yllcorner        0
cellsize        50
nodata_value -9999
   1.0   2.0   3.0   4.0   5.0   6.0   7.0   8.0   9.0  10.0  11.0  12.0  13.0  14.0  15.0   16.0
   2.0   4.0   5.0   6.0   7.0  -6.0  -6.0  10.0  11.0  12.0  13.0  14.0  15.0  16.0  17.0   17.0
   3.0   6.0   7.0   8.0   9.0  10.0 -9999 -9999  13.0  14.0  15.0  16.0  17.0  18.0  19.0   19.0
   4.0   8.0   9.0  10.0  11.0  12.0 -9999 -9999  15.0  16.0  17.0  18.0  19.0 -7.0   19.0   19.0
   5.0  10.0  11.0  12.0  13.0  14.0  15.0  16.0  17.0  18.0  19.0  20.0  19.0  18.0  17.0   17.0
   6.0  12.0  13.0  14.0  15.0  16.0  17.0  18.0  19.0  20.0  19.0  18.0  17.0  16.0  15.0   15.0
   7.0  -8.0  15.0  16.0  17.0  18.0  19.0  20.0  19.0  18.0  17.0  16.0  15.0  14.0  13.0   13.0
   8.0  -8.0  15.0  16.0  17.0  18.0  19.0  20.0  19.0  18.0  17.0  16.0  15.0  14.0  13.0   13.0
```

Example 2:

```
ncols           16
nrows            8
xllcenter      100
yllcenter        0
cellsize        50
nodata_value -9999
   1.0   2.0   3.0   4.0   5.0   6.0   7.0   8.0   9.0  10.0  11.0  12.0  13.0  14.0  15.0   16.0
   2.0   4.0   5.0   6.0   7.0  -6.0  -6.0  10.0  11.0  12.0  13.0  14.0  15.0  16.0  17.0   17.0
   3.0   6.0   7.0   8.0   9.0  10.0 -9999 -9999  13.0  14.0  15.0  16.0  17.0  18.0  19.0   19.0
   4.0   8.0   9.0  10.0  11.0  12.0 -9999 -9999  15.0  16.0  17.0  18.0  19.0 -7.0   19.0   19.0
   5.0  10.0  11.0  12.0  13.0  14.0  15.0  16.0  17.0  18.0  19.0  20.0  19.0  18.0  17.0   17.0
   6.0  12.0  13.0  14.0  15.0  16.0  17.0  18.0  19.0  20.0  19.0  18.0  17.0  16.0  15.0   15.0
   7.0  -8.0  15.0  16.0  17.0  18.0  19.0  20.0  19.0  18.0  17.0  16.0  15.0  14.0  13.0   13.0
   8.0  -8.0  15.0  16.0  17.0  18.0  19.0  20.0  19.0  18.0  17.0  16.0  15.0  14.0  13.0   13.0
```

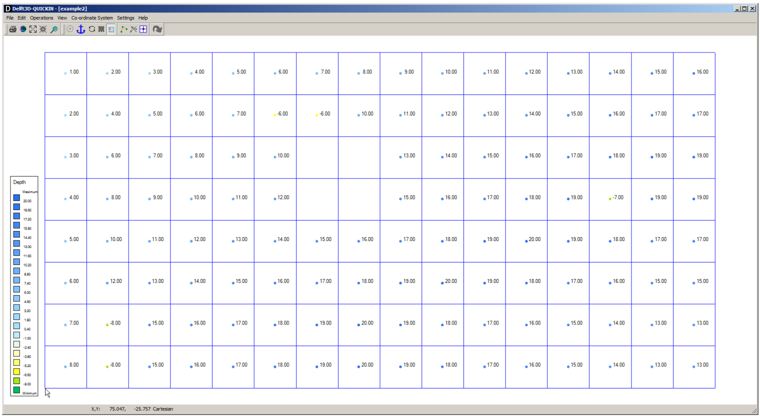


Figure A.2: *Example 2, lower left corner is located at* $(75, -25)$

A.8 Open boundary file

File contents	The location and description of open boundaries.
Filetype	ASCII
File format	Fix formatted for text variables; free formatted for real and integer values.
Filename	<*name*.bnd>
Generated	FLOW-GUI

Record description:

Record	Record description
each record	Name of the open boundary section (20 characters). Type of boundary (1 character). Z water level C current N Neumann Q discharge per grid cell T total discharge for boundary section R Riemann Type of data (1 character). A astronomic H harmonic Q QH tables (only for water level boundaries) T time-series Grid indices of the begin and end point of the boundary section (4 integers). Reflection coefficient (1 real), not for Neumann or Riemann.

Record	Record description
	Vertical profile (three strings); *only for 3D simulations and velocity type boundaries (C, Q, T and R).* Uniform Logarithmic 3D profile Two labels (each 12 characters, no blanks in the label name) referencing to the blocks in the amplitude and phase file <∗.bca>; *only if the type of data is A.*

Restrictions:

- ◇ Maximum record length in the free formatted file is 132.
- ◇ The boundary section name must start at position one in a record.
- ◇ The value of the reflection coefficient will not be checked on its domain.
- ◇ All input items in a record must be separated by one or more blanks.
- ◇ Astronomic and harmonic forced boundaries must be specified before QH-relation forced boundaries, which in turn should be specified before time-series forced boundaries.
- ◇ Astronomic and harmonic forced boundaries cannot be combined.

Example:

Two boundary sections with data type A(stronomic) and one with type T(ime series).

```
Paradise Bay 1      Q A   1   1    1   5 0.0 Uniform     Paradise_1A  Paradisee_1B
Paradise Bay 2      C A  16   3   16   6 0.0 Logarithmic  Paradise_2A  Paradisee_2B
Sea Entrance        Z T   4   8   14   8 0.0
```

Remarks:

- ◇ A label may not contain blanks between non-blank characters.
- ◇ For the labels 12 characters are read. Be sure the second label starts at least 13 positions after the start of the first.

A.9 Dry points file

File contents	Index location of (permanently) dry points.
Filetype	ASCII
File format	Free formatted
Filename	<*name*.dry>
Generated	QUICKIN or FLOW-GUI

Record description:

Record	Record description
each record	The grid indices of the begin and end point of a dry section (4 integers).

Restrictions:

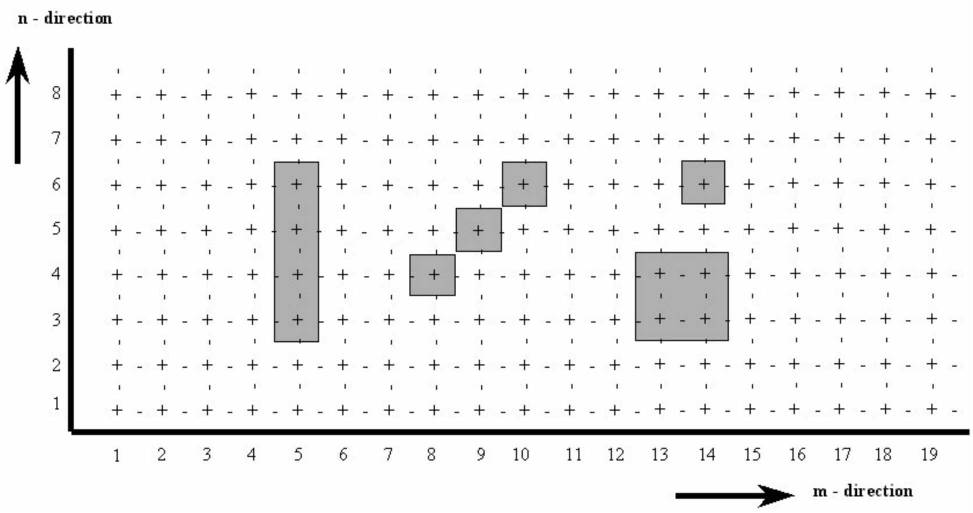


Figure A.3: *Dry points in model area*

- ◇ The angle of a line of dry points and the horizontal numerical grid axis can be an integer multiple of 45 degrees.
- ◇ Dry points may not be defined along the model boundaries (which by default lie along the lines M = 1, N = 1, M = Mmax or N = Nmax). Therefore, the indices of these points must lie between M = 2 and Mmax-1 and N = 2 and Nmax-1, respectively.
- ◇ The input items are separated by one or more blanks.
- ◇ The most lower-left dry point has indices (2, 2).

Example:

Five sets of dry points in a model area of $19 * 8$ grid points, see Figure A.3.

```
 5   3   5   6
 8   4  10   6
13   3  14   3
13   4  14   4
14   6  14   6
```

A.10 Thin dams file

File contents	Location and type of thin dams.
Filetype	ASCII
File format	Free formatted
Filename	<*name*.thd>
Generated	QUICKIN or FLOW-GUI

Record description:

Record	**Record description**
each record	The grid indices of the begin and end point of a line of thin dams (4 integers). A character indicating the type of thin dams (U or V).

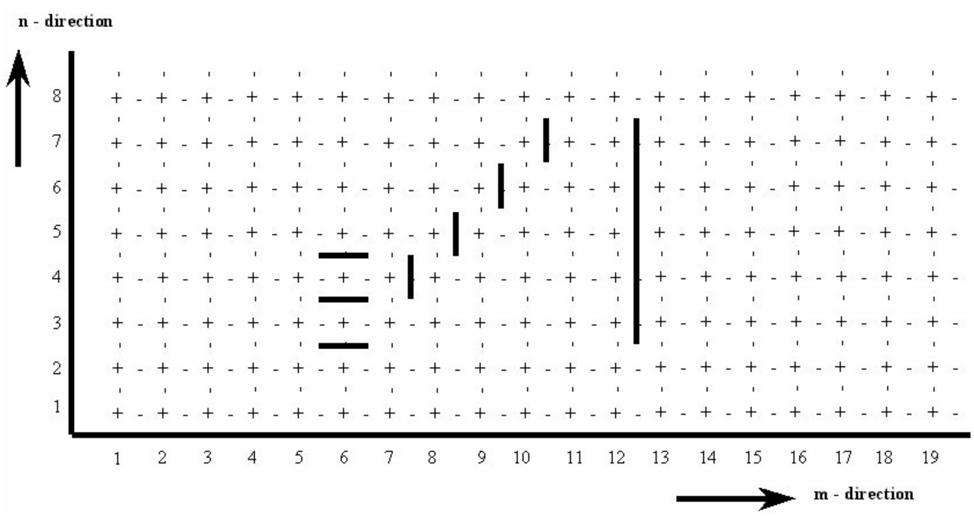


Figure A.4: *Example of thin dams in a model area*

Restrictions:

- ◇ The angle of the line segment and the horizontal numerical grid axis may be an integer multiple of 45 degrees.
- ◇ Thin dams can not be defined along the model boundaries (which by default lie along the lines M = 1, N = 1, M = Mmax or N = Nmax). Therefore, the indices of thin dams must lie between M = 2 and Mmax-1 and N = 2 and Nmax-1 respectively.
- ◇ Input items are separated by one or more blanks.
- ◇ The direction of the dam is perpendicular to the velocity direction over which the dams are superimposed!

Example:

Three (sets of) thin dams in model area of $19 * 8$ grid points, see Figure A.4.

```
 6   2   6   4 V
 7   4  10   7 U
12   3  12   7 U
```

A.11 Observation points file

File contents	Description of observation points characterised by their name and grid indices.
Filetype	ASCII
File format	Fix formatted for text variables, free formatted for integer values.
Filename	<*name*.obs>
Generated	FLOW-GUI or manually offline

Record description:

Record	**Record description**
each record	Name of observation point (20 characters), Location of the observation point in grid indices (2 integers).

Restrictions:

- ◇ One record per observation point.
- ◇ The name of the observation point must start in position one.
- ◇ The maximum record length is 132.
- ◇ Input items in a record must be separated by one or more blanks.

Example:

File containing 5 observation points:

```
Paradise Bay 1        10  11
Paradise Bay 2        11  11
Sea Entrance           4   8
Port Arthur           18  85
Deep Channel          22  44
```

A.12 Annotation file

File contents	File with x and y co-ordinates, string and rgb-colour.
Filetype	ASCII
File format	Free formatted.
Filename	<*name*.txt>
Generated	manually offline

Record description:

Record	Record Description
1	Records starting with a $*$ are comment lines Character string to define the datablock (nonblank)
2	Number of rows
3–N	real, real, string, integer: geographical co-ordinates (2 reals), text between quotes which need to be plotted (string) and rgb-colour (integer; $= 256 * 256 * r + 256 * g + b$)

Restriction:

- ◇ The maximum record length in the file is 132.

Example:

```
*
BL01
8 4
10.0  10.0   'string-01'  6553625
10.0  20.0   'string-02'  9830425
20.0  10.0   'string-03' 13120000
20.0  20.0   'string-04' 16724480
30.0  10.0   'string-05'    38425
30.0  20.0   'string-06'    65305
```

```
40.0  10.0   'string-07'      255
40.0  20.0   'string-08'        0
```

A.13 DD Boundary file

File contents	Domain decomposition boundaries connecting two grids for the prescribed indices.
Filetype	ASCII
File format	Fix formatted.
Filename	<*name*.ddb>
Generated	RGFGRID, or manually offline

Record description:

Record	Record Description
N	Name of the first grid, followed by four integers indicating the gridline on which the boundary lies, followed by the name of the second grid and four integers indicating the gridline on which the boundary lies.

Restrictions:

- ◇ No space allowed in grid filename.
- ◇ The maximum record length in the file is 132.

Example:

In the following example 4 sub-domains exist. Domain d01_ns is coupled to oa1_ns, ob1_ns and oc1_ns. Furthermore oa1_ns is coupled to ob1_ns, and ob1_ns to oc1_ns.

```
d01_ns.grd                 5    16     5     1  oa1_ns.grd             28    35    28    20
d01_ns.grd               245     1     5     1  ob1_ns.grd             17    21     1    21
d01_ns.grd               245    52   245     1  oc1_ns.grd              1    44     1    27
ob1_ns.grd                 1     4     1    21  oa1_ns.grd             28     3    28    20
ob1_ns.grd                17     4    17    21  oc1_ns.grd              1    10     1    27
```

A.14 Colour scheme file

File contents	The colour scheme
Filetype	ASCII
File format	Free formatted
Filename	<*name*.clr> or <*name*.clrmap>
Generated	manually

Record description:

Record	Record description
1	COLORMAP
2	NAME=name
3	SPACE=RGB, RGB is the only allowed space for this program
4 – N	one real and three integers.

The first column represent the relative distribution of the defined colours in column 2–4 (representing the RGB values).

Example:

```
COLORMAP
NAME=copper
SPACE=RGB
0.0000   0   0   0
0.8000 255 159 101
1.0000 255 199 127
```

A.15 Settings file

File contents	Settings of the program
Filetype	ASCII
File format	Fix formatted
Filename	<*name*.ini>
Generated	By the program

Record description:

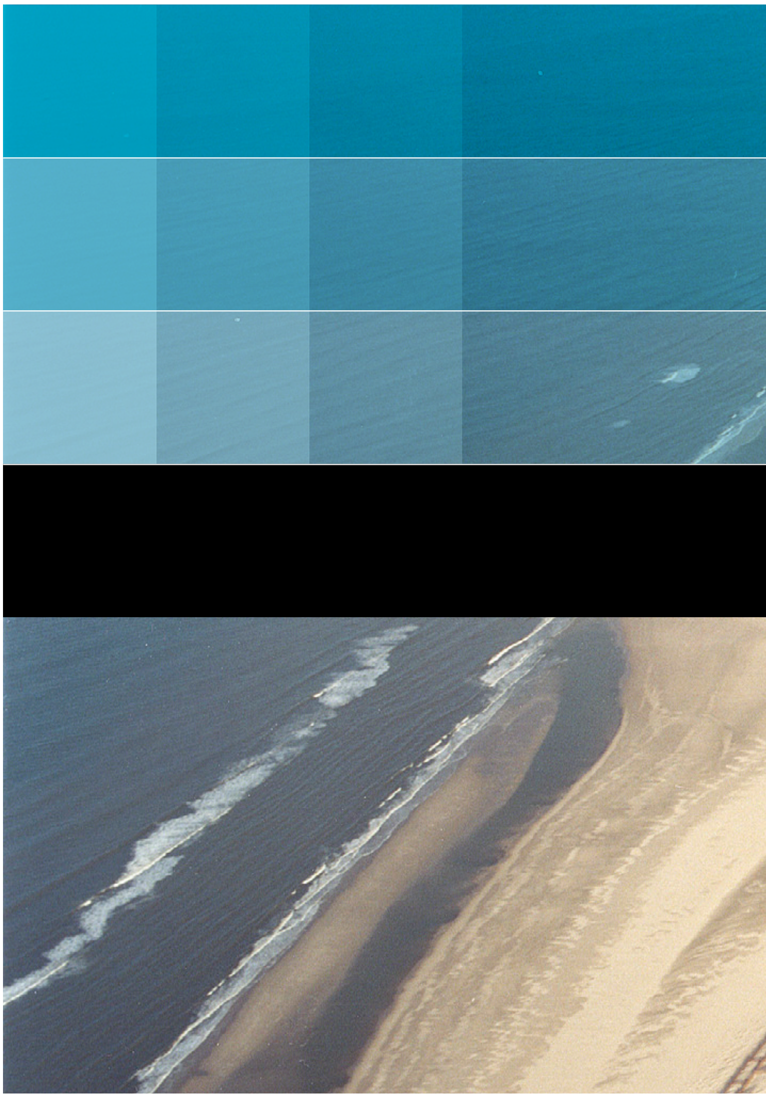
Record	Record description
FileInformation	
FileCreatedBy FileCreationDate FileVersion	QUICKIN version number creation date and time version number of <∗.ini> file
QNParameter	
name	integer value
TextSettings	
name	value (integer or real)
QNsettings	
name	value (integer or real)
Colours	
name	RGB value (3 integers) line width dots sizes

Example:

```
[FileInformation]
  FileGeneratedBy  = Deltares, Delft3D-QUICKIN Version 4.20.00.11763:11790M, Jun 16 2010, 14:26:47
  FileCreationDate = 2010-06-16, 14:27:41
  FileVersion      = 0.02
[RGFSettings]
  MGridCells                =  50
  NGridCells                =  50
  DeltaX                    =   1.00000000000000000E+02
  DeltaY                    =   1.00000000000000000E+02
  OriginX                   =   0.00000000000000000E+00
  OriginY                   =   1.00000000000000000E+02
  RotationLeft              =   0.00000000000000000E+00
  RadiusCurvatureM          =   0.00000000000000000E+00
  MFraction                 =   1.00000000000000000E+00
  MaximumSizeUniformMSize   =   1.00000000000000000E+00
  NFraction                 =   1.00000000000000000E+00
  MaximumSizeUniformNSize   =   1.00000000000000000E+00
[QNParameter]
  AutoscaleLegend           =   1
  XCoorLegend               =  16
  YCoorLegend               =  20
[TextSettings]
  Line1                     =
  Line2                     =
  Line3                     =
  Fontsize                  =   3.00000000000000000E+00
  Xposition                 =   0.00000000000000000E+00
  Yposition                 =   0.00000000000000000E+00
```

```
  FontsizeTimeDate          =   3.00000000000000000E+00
  XposTimeDate              =   5.00000000000000000E+01
  YposTimeDate              =   5.00000000000000000E+00
[QNSettings]
  StayOnStartupDirectory    =   0
  NrDiffusionStep           = 100
  NrSmoothingSteps          =  10
  SmoothingFactor           =   5.00000000000000000E-01
  DtCFLNumber               =   6.00000000000000000E+01
  ReferenceLevelZref        =   0.00000000000000000E+00
  SearchCellSizeAveraging   =   1.10000000000000009E+00
  MinAveragingPoints        =   4
  DtIncrementalImages       =   1.00000000000000000E+00
  ParameterPlotIncremental  =   1
  NrSamplePerSegment        =   9
  FixedDistanceSamplePoints =   0.00000000000000000E+00
  PositionSamples           =   0
  DisplaySlopeCriterion     =   0
  DisplayAsNumber           =   1
  DisplayCriterionCheck     =   0
  ValueSlopeCheckCriterion  =   1.00000000000000006E-01
  WavePeriod                =   6.00000000000000000E+00
  DepthOffset               =   0.00000000000000000E+00
  ChannelWidthBegin         =   1.00000000000000000E+02
  ChannelDepthBegin         =   1.00000000000000000E+01
  ChannelWidthEnd           =   1.00000000000000000E+02
  ChannelDepthEnd           =   1.00000000000000000E+01
  NrPointsAcrossChannel     =   5
  MinDepth                  =   1.00000000000000000E+01
  RemoveAddSamples          =   0
[Colours]
  ColourBackground          = 255 255 210
  LegendColourBackground    = 255 255 255
  lineColourText            = 005 005 005
  lineColourLandBoundary    = 132 066 000
  fillColourLandBoundary    = 255 182 108
  lineColourPolygon         = 170 000 127
  fillColourPolygon         = 170 000 127
  lineColourActiveGrid      = 000 000 255
  lineColourActiveCmpBnd    = 005 005 005
  lineColourGrid            = 192 192 192
  lineColourCmpBnd          = 050 050 050
  DryPoints                 = 203 203 000
  ThinDams                  = 203 203 000
  ObservationPoints         = 227 227 000
  DredgeAndDumpArea         = 255 255 000
  DredgeAndDumpLink         = 255 000 000
[Width]
  lineWidthLandBoundary     =   1
  lineWidthPolygon          =   1
  DotSizeSamples            =   7
  lineWidthActiveGrid       =   1
  lineWidthActiveCmpBnd     =   1
  lineWidthGrid             =   1
  lineWidthCmpBnd           =   1
  lineWidthThinDams         =   3
  DredgeAndDump             =   3
[Caches]
  rest                      =   0
  text                      =   0
  polygons                  =   0
  dad                       =   1
```

```
drythdobs        =   1
cmpbound         =   1
secdepth         =   2
actgrid          =   2
actdepth         =   3
samples          =   3
landboundary     =   1
inactgrid        =   4
trisample        =   5
```



Deltares systems

PO Box 177
2600 MH Delft
Rotterdamseweg 185
2629 HD Delft
The Netherlands

+31 (0)88 335 81 88
sales@deltaressystems.nl
www.deltaressystems.nl