

Plot Composer

User Guide





PLEASE NOTE

COPYRIGHT

This document refers to proprietary computer software which is protected by copyright. All rights are reserved. Copying or other reproduction of this manual or the related programs is prohibited without prior written consent of DHI. For details please refer to your 'DHI Software Licence Agreement'.

LIMITED LIABILITY

The liability of DHI is limited as specified in your DHI Software Licence Agreement:

In no event shall DHI or its representatives (agents and suppliers) be liable for any damages whatsoever including, without limitation, special, indirect, incidental or consequential damages or damages for loss of business profits or savings, business interruption, loss of business information or other pecuniary loss arising in connection with the Agreement, e.g. out of Licensee's use of or the inability to use the Software, even if DHI has been advised of the possibility of such damages.

This limitation shall apply to claims of personal injury to the extent permitted by law. Some jurisdictions do not allow the exclusion or limitation of liability for consequential, special, indirect, incidental damages and, accordingly, some portions of these limitations may not apply.

Notwithstanding the above, DHI's total liability (whether in contract, tort, including negligence, or otherwise) under or in connection with the Agreement shall in aggregate during the term not exceed the lesser of EUR 10.000 or the fees paid by Licensee under the Agreement during the 12 months' period previous to the event giving rise to a claim.

Licensee acknowledge that the liability limitations and exclusions set out in the Agreement reflect the allocation of risk negotiated and agreed by the parties and that DHI would not enter into the Agreement without these limitations and exclusions on its liability. These limitations and exclusions will apply notwithstanding any failure of essential purpose of any limited remedy.





CONTENTS

1	Plot Composer Menu Guide	11
1.1	Introduction	11
1.2	File Menu	12
1.3	Edit Menu	13
1.4	View Menu	13
1.5	Plot Menu	14
1.5.1	Insert new plot object	14
1.5.2	Mutate plot	15
1.5.3	Size and position plot	15
1.5.4	Lock plot	15
1.5.5	To front	15
1.5.6	To back	15
1.5.7	Align	15
1.5.8	Magnification	16
1.6	Settings Menu	16
1.7	Video Generation	16
1.7.1	Video properties	17
1.7.2	Video compression	18
1.7.3	Animation setup	18
1.8	Popup Menu	18
1.9	Additional information	20
1.9.1	Marker properties	20
1.9.2	Line properties	20
1.9.3	Shortcut keys	21
1.9.4	Batch command	22
2	Dfsu Plot	23
2.1	Input Data	23
2.2	Item selection	23
2.2.1	Valid vector components	24
2.3	SubSet Selection	24
2.4	Contour Settings	24
2.4.1	Primary option	25
2.4.2	Secondary option	25
2.5	Vector Settings	26
2.6	Coordinate System	26
2.7	Points and Lines	27
2.7.1	Points	27



2.7.2	Lines	28
2.7.3	Properties	28
2.8	Output Settings	28
2.9	Popup Menu	28
2.10	Image manager	30
2.10.1	Image style	30
2.10.2	Shape style	31
2.10.3	XYZ style	31
2.10.4	Meta file style	31
2.10.5	Particle style	31
2.11	Overlay manager	32
3	Grid Plot	33
3.1	Input Data Selection	33
3.2	Item selection	34
3.2.1	General	34
3.2.2	Dfs2 result file	34
3.3	SubSet Selection	35
3.4	Contour Settings	35
3.4.1	Primary option	35
3.4.2	Secondary option	36
3.5	Vector Settings	36
3.5.1	Nested vector setup	37
3.6	Land Settings	37
3.6.1	General	37
3.6.2	Dfs2 result file	37
3.7	Coordinate System	38
3.8	Points and Lines	39
3.8.1	Points	39
3.8.2	Lines	40
3.8.3	Properties	40
3.9	Output Settings	40
3.10	Popup Menu	41
3.11	Image Manager	42
3.11.1	Image style	43
3.11.2	Shape style	44
3.11.3	XYZ style	44
3.11.4	Metafile style	44
3.11.5	Particle style	44
3.12	Overlay Manager	45
4	Profile Plot	47
4.1	Item Selection	47
4.2	Step Configuration	47
4.3	Plot Type	47
4.4	X-axis Definition	47
4.5	Y-axis Definition	48



4.6	Curve Definitions	48
4.7	Text Annotations	48
4.8	Popup Menu	48
5	Time Series Plot	51
5.1	Items	51
5.2	X-Axis Definitions	51
5.3	Y-Axis Definitions	51
5.4	Curve Definitions	52
5.5	Text Annotation	52
5.6	Popup Menu	52
6	ADCP 2D Plot	55
6.1	Input Data Selection	55
6.1.1	ADCP type 1 files	55
6.1.2	Background	56
6.2	Items	56
6.2.1	Item selection for dfs2 result files	57
6.2.2	Item selection	57
6.3	SubSet	57
6.4	Contours	58
6.5	ADCP Vectors	58
6.6	Land options	58
6.6.1	Land settings for dfs2 result files	58
6.6.2	Land settings in general	59
6.7	Coordinate System	59
6.8	Popup Menu	60
6.9	Examples	61
6.9.1	Track on bathymetry	61
6.9.2	Comparison with MIKE 21 results	62
6.9.3	Tracks with surface level	64
7	Annotation Plot	67
7.1	Annotation Plot Properties	67
8	Flow Visualisation	69
8.1	Input Specification	69
8.2	Particles	70
8.3	Land/Water Specifications.	70
8.4	SubSet Specifications	71
8.5	Coordinate System	71
8.6	Output Specification	72
8.7	Popup Menu	72
9	Probability Plot	73
9.1	Items	73
9.2	Plot Properties	73
9.3	X-Axis Definitions	74



9.4	Y-Axis Definitions	74
9.5	Curve Definitions	74
9.6	Text Annotations	74
9.7	Popup Menu	74
10	Isopleth Plot	77
10.1	Input	78
10.2	Contours	79
10.3	X-axis	79
10.4	Y-axis	79
10.5	Text Annotations	79
10.6	Popup Menu	80
11	Label Plot	81
11.1	Label Properties	81
11.2	Popup Menu	81
12	Metafile Plot	83
12.1	Metafile Properties	83
12.2	Popup Menu	83
13	Polar Plot	85
13.1	Input Data	85
13.2	Subset	85
13.3	Contour Settings	86
13.4	Type of Grid Selection	86
13.4.1	Compass grid	86
13.4.2	Wave period grid	86
13.4.3	Directional and frequency resolution	86
13.4.4	Change of wave period grid	87
13.5	Text Annotation	87
13.6	Output Settings	87
13.7	Popup Menu	87
14	Current and Wind Rose Plot	89
14.1	Item and File Selection	89
14.2	SubSeries Selection.	89
14.3	Data Classes definitions.	90
14.4	Plotting definitions.	90
15	Slice Plot	91
15.1	Input Data Selection	91
15.2	Items	91
15.3	SubSet Selection	92
15.4	Slice Selection	92
15.5	Contours	92
15.6	Vectors	93
15.7	Land Options	93



15.7.1	Land options	93
15.7.2	Land settings for dfs2 result files	93
15.7.3	Land settings for dfs3 result files	94
15.8	Coordinate system setup	94
15.9	Output Settings	94
15.10	Popup Menu	95
16	Time Series Direction Plot	97
16.1	Items	97
16.2	X-Axis Definitions	97
16.3	Y-Axis Definitions	97
16.4	Curve Definitions	98
16.5	Text Annotations	98
16.6	Popup Menu	98
17	Track Flow Visualisation	101
17.1	Input Specification	101
17.2	Particle Track Definitions	102
17.3	Land/Water Specifications	102
17.4	SubSet Specifications	102
17.5	Coordinate System	103
17.6	Popup Menu	104
18	XY Plot	105
18.1	Curve Definitions	105
18.2	Axis Definitions	105
18.3	Text Annotations	105
18.4	Popup Menu	106
Index		107





1 Plot Composer Menu Guide

1.1 Introduction

The MIKE Zero Plot Composer provides you a work environment for creating report quality hardcopy output. Furthermore, some plot controls produce animations that may be created, viewed and stored.

The heart of the plot composer is the workspace that represents the sheet of paper, which can be subsequently printed. One or more so-called containers may be placed in the workspace. The term container is used to denote an area in the workspace in which a plot or another item is placed. A specific set-up in the Plot Composer can be saved as a PFS-file (*.plc) and opened again later.

You can create a new empty Plot Composer set-up by selecting *File->New->File...* and then *Plot Composer* from the MIKE Zero Product type documents.

Any kind of plot can be assigned to a container. They are referred to as plot objects, and include a range of plotting tools for the data types in MIKE Zero.

List of plot objects

Standard graphics

- Dfsu Plot - plot 2D flexible mesh data (dfsu or mesh)
- Grid Plot - plot grid data
- Profile Plot - plot line data
- Time Series Plot - plot time series data

Advanced graphics

- ADCP 2D Plot - plotting of horizontal ADCP transect data
- Annotation Plot - plotting text data
- Flow Visualisation - float tracking plots from MIKE 21 hydrodynamic results
- Probability Plot - plot of time series as histograms
- Isoleth Plot - plotting of time variation of vertical line data
- Label Plot - plot a drawing label on the page
- Metafile Plot - plot the content of an enhanced metafile (*.emf)
- Polar Plot - plotting of directional wave spectra computed by MIKE 21 SW



- Current and Wind Rose Plot - plotting of wind/current rose from a dfs0 data file
- Slice Plot - plotting of slices of dfs2 or dfs3 data
- Time Series Direction Plot - plotting of directional dfs0 data as arrows
- Track Flow Visualisation - float tracking plots from MIKE 21 hydrodynamic results with paths of particles.
- XY Plot - plot an item against another item from a dfs0 data file

1.2 File Menu

This is where you open, save and close files, and access printer related utilities.

New

Opens the standard MIKE Zero File/New dialog where you can choose a document type to create.

Open

Opens the standard MIKE Zero File/Open dialog where you can choose an existing document to open.

Close

Closes the active document.

Save

Saves the Plot Composer PFS-file.

Save As

Saves the Plot Composer PFS-file as a new file.

Print

Produces a hardcopy of the current plot.

Batch Print

Produces a hardcopy of a time series of plots. The time series can be selected in two ways:

- selection based on time steps, e.g. from time step 2 until time step 10 by 2 time steps
- selection based on time, e.g. from 1990/01/01 00:00:00 until 1990/01/01 06:00:00 by 600 seconds

Print Set-up

Here you may select the printer and modify its set-up. It is also where you choose the layout of the paper and the paper size.



1.3 Edit Menu

Clipboard and object editing.

Cut

Presently deletes your selected data. Please note that there is no UNDO facility in the editor.

Copy

Copies your selected plot object.

Paste

Pastes your selected (copied) plot object.

If you paste to a new page, the position of the plot object will be the same as for the original plot object.

If you paste to the same page, the position of the plot object will be dislocated (10, 10) mm relative to the original plot object.

Duplicate

Duplicates your selected plot object to a position 5 mm below the original plot object at the same page.

Delete

Deletes your selected data. Please note that there is no UNDO facility in the editor.

Properties

Opens the Plot Properties window for the object in focus.

Object

The following operations may be done to the object in focus:

- Properties, the object properties may be modified
- Convert, the object type may be changed

1.4 View Menu

This is where you change margins, add toolbar views, status bar and modify the magnification of the view. You can also export to graphics.

Export graphics

The entire plot area with all plot objects within can be exported to either Clipboard, Metafile or Bitmap.

Margins

Toggles the view for margins. When margins are ON, you will see the margin of the plot area as a red dashed line.



Toolbars

Allows you to insert and remove toolbars. The Plot Menu provides a number of tools especially relevant for the Plot Composer.

Status Bar

Toggles the status bar at the bottom of the window on and off. While it provides useful information, it may be convenient to toggle the status bar off for an increased viewing area.

Magnification

The magnification of the view may be changed here.

Refresh (F5)

The refresh button (or shortcut key F5) refresh the plot with simulation data from the latest time step available. For Grid Plot objects please make sure you have forwarded the display to the last time step. During the execution of a simulation you can plot the latest results clicking on the F5 shortcut key.

1.5 Plot Menu

A number of operations related to the active plot and all plots may be performed from this menu if the 'Edit' tab in the lower left corner is selected.

The Plot Composer Toolbar show below is a mirror image of the plot menu.



1.5.1 Insert new plot object

Brings up a dialog for inserting a new plot object. There are three ways of inserting a new object:

Single Plot Maximised

Insert one single plot object with a size covering the total area inside the margins.

Single Plot Sized

Insert one single plot object with the specified size in mm plot area on the paper.

Multiple Plots Tiled

Insert a number of plots covering the total area inside the margins. A matrix of NX plots in the horizontal and NY plots in the vertical are inserted. All the inserted objects will be of the same type. You may use the Mutate plot facility to modify the individual objects later.



After having selected the way of inserting the new object(s), please select the type of object and accept.

1.5.2 Mutate plot

This dialog allows you to mutate the active plot object, e.g. change a time series plot into a grid plot. The dialog has the same appearance and functionality as the Insert New Plot Object dialog.

1.5.3 Size and position plot

Brings up the Size and Position dialog, where you may change the size of the active plot object and position it on the page.

The position on the page can be selected from a number of choices and specified in mm on the page. E.g. the plot should be placed at the bottom right, but 10 mm from the bottom margin and 5 mm from the right-hand-side margin.

1.5.4 Lock plot

The active plot object may be locked in order to prevent accidental and unintentional changes to the object. If you want to modify the plot objects, you may click on the object with the right mouse button and UNLOCK the plot again.

1.5.5 To front

Brings the active plot object to the front of a series of objects.

1.5.6 To back

Brings the active plot object to the back of a series of objects.

1.5.7 Align

Align the active objects as selected. The objects should be selected in the standard Windows way pressing the left mouse button while pressing the shift button. In order to activate this function select at least two objects. The first object, which is selected, is the master object. This is done by left clicking on the object. The slave objects are selected by pressing the "Ctrl"-key and then left clicking on the objects. This activates the Align-function. You can choose the quick-align by pressing the align-icon. The align-type default selected is then chosen. By pressing the attached drop-down button you can freely choose between 6 align types: Horizontal left, Horizontal centre, Horizontal right, Vertical top, Vertical centre, Vertical bottom. If the align-function is cho-



sen through the drop-down menu, the selected align-type becomes the new default align-type.

1.5.8 Magnification

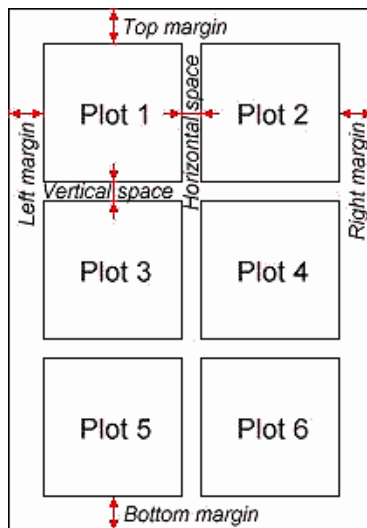
The magnification of the view may be changed here.

1.6 Settings Menu

Here the margins of the page and the units for snapping on the page may be set.

Margins

Set the margins and the units that divide the page. These margins are used when inserting "Multiple Plots Tiled". The horizontal and vertical space defines the space between adjacent containers, however they must be set in advance prior to the creation of the multi-plot in order to have an effect. See illustration below.



1.7 Video Generation

For plots with time varying data it is possible to generate a video based on the plot setup. You have the option to create a video of the single plot object alone, or you can create a video of a number of plot objects simultaneously.

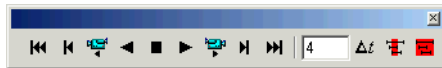


Single plot object

To record a single plot object right-click the object and select the Toolbar in the Popup Menu. The file is **recorded** by the Toolbar button with the video camera icon.

Multiple plot objects

To record multiple plot objects simultaneously, click the 'Animate' tab at the lower left hand corner. This shows the plot objects in the animate window with the toolbar shown below.



The icons to the left enables you to move forwards and backwards in time to display results from a simulation and maybe use it for Video Generation.

The icons to the right enables you to control the Animation setup with respect to the time steps that are to be used in the video.

1.7.1 Video properties

You can control various output settings for the video. Which ones and in what way is dependent on the particular plot object(s).

Double Buffering

This is a special drawing technique, that allows flicker-free animations. Double Buffering is only used if the plot control is active.

File name

The video file name is the name of the AVI-file used for generation of video output from the plot.

Frame rate

The Frame rate is the number of frames per second that is displayed in the video.

Image size

The Image size for a single plot object is defined by the layout in the plot. The Image size for multiple plot objects can be defined by one of three options:

- Minimum rectangle enclosing all plots
The image in the video is automatically set to the minimum size.
- Full paper
The image will display the full page of the plot
- Custom
Here you define the margins for the image to control the size.



1.7.2 Video compression

In order for the video file to become a reasonable size, you can choose between different compression types and compression quality.

The default available compression types depends on your windows installation.

1.7.3 Animation setup

You can control which time period that are to be displayed in the video and by which time step



This bring you to the Animation setup dialog where you can define the time period and time step in the video.



In here you specify the frequency of the time steps. Per default all time steps are used



This sets the time period of the video animation to the period that are covered by all plot objects. The time step is automatically set to the largest time step of the plot objects.



This sets the time period of the video animation to the joint period of all plot objects. The time step is automatically set to the smallest time step of the plot objects.

1.8 Popup Menu

A click on the right mouse button brings up a menu that allows for various manipulations of the active object or the object over which the cursor is positioned. Depending on the context, one or more of the following options will appear. Some options are general some options are specific to plot controls.

Properties

A plot control specific property page appears, in which the properties of the plot can be manipulated.

Add/Remove Layers

In the Image manager you can select between several file types to be included as additional layers on you plot.

In the Overlay manager you have the option to choose the order by which the drawing of the layers in your plot occur. This way you choose if the layer should be in the background or foreground on your plot.



Toolbar

You can display a toolbar that enables you to move forwards and backwards in time to display results from a simulation and maybe use it for Video Generation.

The video icon creates the file.

File Name Legend

Places a file legend in the plot container depicting the names of the data files from which the plotted data have been taken.

Scale Legend

Places a scale legend in the plot container depicting the scale of the plot.

Aspect ratio

Allows you to change the aspect ratio between the x-axis and y-axis. Per default this is fixed from file.

Zoom facilities

Zoom In allows you to zoom in on the plotted data and change the focus of the plot.

Zoom Out zooms out to default coverage.

Previous Zoom and **Next Zoom** zooms to the previous or following focus of the plot, respectively.

Draw Grid

Adds a grid to the plot.

Copy to Clipboard

Copies the selected data to the clipboard. The general idea is that the graphics from an object at any time can be copied to a standard Windows application which supports GDI-operations. An example could be to export the graphics to MS Word. Activate the "Copy to Clipboard" function, then switch to MS Word and activate the "Paste"-function or press "Ctrl" V in and active Word document, and then a copy of the plot appears in MS Word.

Save to Metafile

Saves the current plot to an enhanced meta file.

Note: This option does not support transparency.

Save to Bitmap

Saves the current snapshot plot to a bitmap file (bmp, png, jpg or tif).

Mutate Plot

This dialog allows you to mutate the active plot object, e.g. change a time series plot into a grid plot.



Size and position plot

You may change the size of the active plot object and position it on the page.

UnLock Plot

The active plot object may be locked in order to prevent accidental and unintentional changes to the object. If you want to modify the plot objects, you may click on the object with the right mouse button and UNLOCK the plot again.

1.9 Additional information

1.9.1 Marker properties

Marker symbol

There is a choice between 10 different symbols.

Marker size

The marker size is given in mm.

Marker fill style

The marker fill style can be defined as *Transparent*, *White* or *Color*.

Marker color

There is a choice between 20 different colours.

Marker spacing

The marker spacing is given in mm.

The marker spacing defines a minimum distance between markers on the plot.

Marker phase shift

The marker phase shift is given in mm.

The marker phase shift defines the start of the marker locations.

1.9.2 Line properties

Line width

The line width is given in mm.

Setting the line width to 0 will create the thinnest possible line that can be drawn to the output device (being the printer, screen etc.).

Line style

The line style can be defined as *solid*, *dashed*, *dotted*, *dashed-dotted*, or *filled*.

- For *dashed* lines the dash length is 5.0 mm and the gab is 1.8 mm wide.
- For *dotted* lines the dot length is 1.0 mm and the gab is 0.8 mm wide.



- For *dashed-dotted* lines the dash length is 3.0 mm, the dot length is 1.0 mm and the gap is 1.3 mm wide.
- Using *filled* line will cause the area between the line and the x-axis to fill with solid colour.

Line color

There is a choice between 20 different colours.

1.9.3 Shortcut keys

You can create, select and edit plot objects in the Plot Composer by using shortcut keys.

These keys provide quick access to functions listed on menus or in dialog boxes. If a function can be completed using shortcut keys, the keys are listed on the menu next to the function, as shown in Figure 1.1.

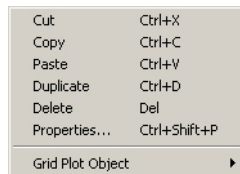


Figure 1.1 Shortcut keys for functions in Edit Menu

A list of available shortcut keys in Plot Composer is given in Table 1.1.

Table 1.1 Available shortcut keys in Plot Composer

Function	Shortcut Keys
Copy plot object	Ctrl+C
Cut plot object	Ctrl+X
Delete plot object	Del
Duplicate plot object	Ctrl+D
Activate plot object Properties dialog	Ctrl+Shift+P
Insert new plot object	Ctrl+I
New window for AVI creation	Ctrl+A
Paste plot object	Ctrl+V
Save to Metafile	Ctrl+M
Save to Bitmap	Ctrl+B



1.9.4 Batch command

It is possible to generate graphic files through a batch-command once the graphical setup file (.plc) has been generated in the Plot Composer.

Open a DOS-prompt and type one of the following commands:

- `MzPlotCompApp input.plc [-options] -printtofile output.png`
- `MzPlotCompApp input.plc [-options] -print_to_file output.png`

Alternatively you may create other file types e.g. *output.bmp*, *output.jpg*, *output.tif* or *output.emf*.

The options [-screen], [-height xxx] and [-width xxx] can be applied to set the size (pixel resolution) of the image. The option [scalefont] (alternatively [scale_font]) can be applied to scale the font size in case the height or the width of the image is user defined.

no option	generates image optimized for print
-screen	generates image by screen resolution
-screen -height 800 -width 600	generates image by 800 x 600 pixels
-screen -height 800	generates image 800 pixels high and width by screen resolution
-screen -width 600	generates image 600 pixels wide and height by screen resolution
-screen -height 3000 -width 2000 -scale_font	generates image by 3000 x 2000 pixels with font scaled relatively to screen resolution

Writing **MzPlotCompApp -?** or **MzPlotCompApp -hlp** in a DOS prompt will show the full syntax of the command.



2 Dfsu Plot

The DFSU plot control is used for plotting 2D flexible mesh data, i.e. dfsu and mesh data.

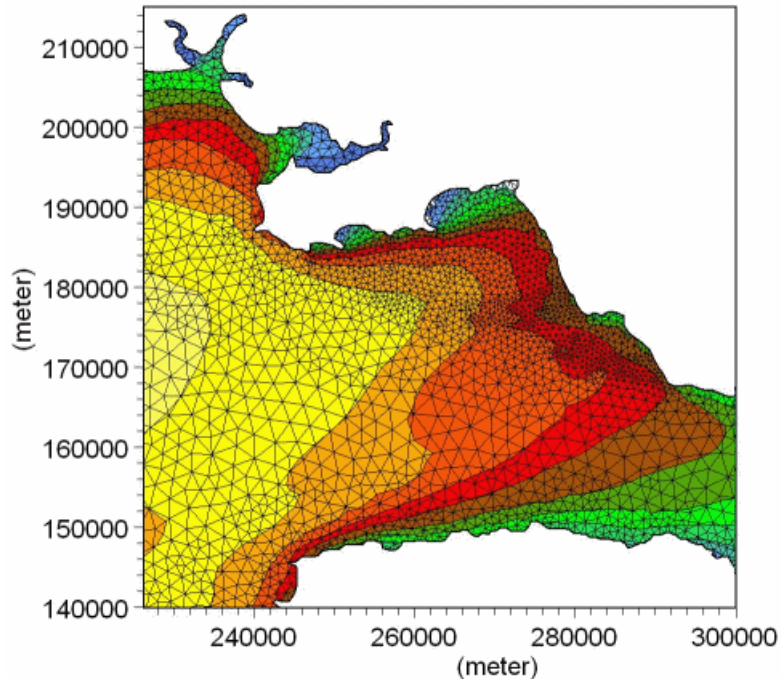


Figure 2.1 Example of DFSU plot

2.1 Input Data

The input page is used for selecting the flexible mesh data, you want to plot. You can select either a 2D DFSU data or a 2D mesh file.

2.2 Item selection

The plot control can plot data as contours. If the selected data file is a dfsu file and it contains certain flow components, e.g. current velocities or fluxes or wind velocities, the plot control can plot the data as both contours and vectors.

The appearance of the contour items is defined on the Contours page and vector items on the Vectors page.



2.2.1 Valid vector components

The items available for vector plot is pre-defined by the type of the item in the data file. The possible vector components are listed in Table 2.1 below:

Table 2.1 Valid components for vector plot

Vector name	Type of item in x-direction	Type of item in y-direction
Velocity	u-velocity component	v-velocity component
Flux	Flow Flux	Flow Flux
Wind velocity	Wind Velocity	Wind Velocity
Bed load	Bed sediment load per m	Bed sediment load per m
Suspended load	Suspended load per m	Suspended load per m
Total load	Sediment transport per m	Sediment transport per m
Accumulated load	Accumulated transport per meter	Accumulated transport per meter

Per default, the first two instances of the selected item type in the data file defines the x-component and y-component of the vector, respectively.

NOTE: You may review the item types by opening the data file using the Data Utility option.

2.3 SubSet Selection

This is where you choose a **subarea** or a **sub-timeseries** instead of plotting the whole file. A subarea is defined by the grid numbers on the spatial axes and a sub-timeseries by the time step numbers.

Instead of selecting a subarea now, you can also choose to zoom-in in the Popup Menu. Choosing it now will probably cause less memory problems for your computer, and this will be the area that is saved to the output file.

2.4 Contour Settings

You can define the way contours are displayed. The contour is defined by a Primary option that defines the contour type and a Secondary option that depends on the primary option.

You can choose to see the contours as "boxes", following the mesh elements, or you can smooth them by choosing "shading". The **"shading"** option shows different layers in different colours. If you choose "no contour", you must choose "Draw isolines" to be able to see anything at all. The isolines are lines on which the value is the same. You can see this value by choosing "With labels".



Color legend

When you choose "Shading" or "Box contours", you can also plot the color legend. The color legend can be configured using the Palette wizard. You can create, open, edit and save a palette file. You can use the dialog to calculate the limits from the data file by pressing "calculate", or you can specify the limits yourself.

If you select '<New levels/colors...>', you can define a custom palette which is either "Auto scaled" or "Fixed". On the fixed palettes, you can specify the data value ranges.

2.4.1 Primary option

Box Contour

Each grid cell is given a colour according to the value in the cell.

Secondary Options: No. 1

Box Contour with transparency

Each grid cell is given a colour according to the value in the cell.

Secondary Options: No. 1 and No. 2

Shaded Contour

The grid is covered with colours indicating the values in the cells, a smooth surface is used. The land values will always be displayed in green.

Secondary options: No. 3 or No. 4 or No. 5 + No. 2

No Contour

Used in connection with background overlays. The bathymetry will not be shown.

2.4.2 Secondary option

1. **Box contour divider**
This option can be switched on and off having selected box contour for display. A system of lines will be placed on the grid displaying the transition between the colours.
2. **Transparency**
Used in connection with background overlays the value defines the transparency of the layer.
3. **Copy Colours**
The colours indicate the values in the cell.
4. **Copy Colours with Transparency**
The colours indicate the values in the cell, one colour is fully transparent (usually white).
5. **Blend colours**
All colours are transparent with a user-defined transparency (secondary



option 2) and one colour is fully transparent (usually white).

2.5 Vector Settings

On this page you specify how the vectors should be scaled and the interval between the individual vectors.

Automatic scaling is a quick way to scale the vectors. You simply give the length (in the unit of average node distance) of the longest vector on the plot. The individual plots in a series are scaled differently. When using automatic scaling, vectors are never distorted. Therefore, when you change scaling in one direction, the scaling in the other direction is automatically changed accordingly.

User specified scaling gives you full control of the vector scaling. You specify the vector length (again in average node distance) corresponding to typically one SI unit. The individual plots in a series are therefore given the same scaling. User specified scaling can also be used for amplifying the vector length in one direction (distortion).

If the density of vectors is high, you can opt to reduce the number of drawn vectors by one of two methods:

- **Box based reduction**
You divide the whole area in a number of smaller boxes and specify the maximum number of vectors to draw within each box. The box size is specified as a percentage of the whole area.
- **Interpolation to structured mesh**
You specify the size of a structured mesh. One vector is drawn within each structured mesh element.

2.6 Coordinate System

In case you have geographical information in your input data file, you can **superimpose on the plot geographical coordinates** either as projection coordinates or as latitudes and longitudes. Furthermore, you have the opportunity to **add plotting symbols indicating true north and the convergence**, defining the angle between the longitude and the projection northing line. The indications of true north and convergence can be moved later, directly on the picture, using the Popup Menu.

When you are superimposing a coordinate system, the one specified in the Map Projection "Type" box will be used. The default projection is the one specified in your input data file.



If you select **Northing coordinates always positive**, The northing coordinates will be as 1) and 3) in Figure 2.2. If you don't select it, the southern hemisphere northing coordinates will be as 2).

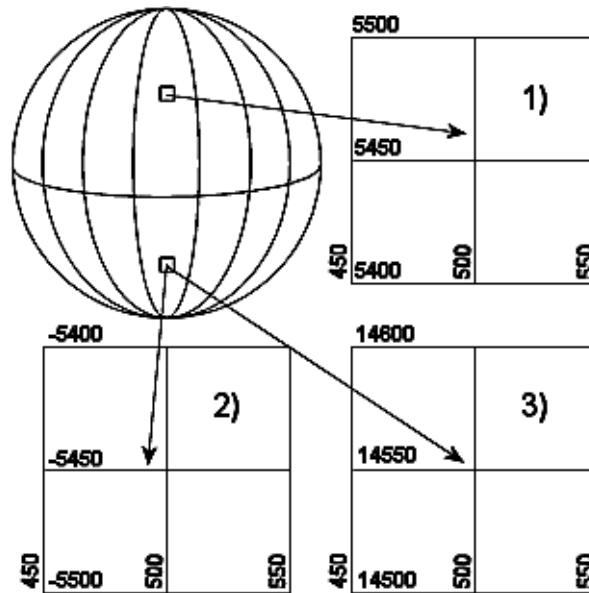


Figure 2.2 Illustration of UTM-zone

Finally, you can specify additional **axis annotation** and a **geodetic scale**. If the data is defined using LONG/LAT the scaling is not available.

Choosing automatic scaling will show the scale on the plot according to how you size the plot manually. Choosing a defined scale, ex. 1:100000, will automatically size the plot according to that scale.

2.7 Points and Lines

In case you want to add descriptive information on the plot you can **superimpose** individually defined Points and Lines. For both points and lines you may add strings to be displayed.

2.7.1 Points

You can define any number of points.

For each point you define the following:

- the coordinate (x,y)
- a text string related to the coordinate



- Properties for the point marker and text to be displayed

2.7.2 Lines

You can define any number of lines.

For each line you define the following:

- the start coordinate and end coordinate of the line
- a text related to the start coordinate
- a text related to the end coordinate
- Properties for the point markers and text to be displayed

2.7.3 Properties

Color

The colour will be the same for both markers, lines and text.

Marker and line

You define the size of the marker and if it is to be displayed at all. For lines you specify the thickness of the line.

Text

The text is defined by the size, angle, alignment and background style. For lines the text for both start point and end point can be displayed individually.

2.8 Output Settings

With this page you can control the output settings: Double Buffering and name of video file.

The file is **recorded** by the Toolbar button with the video camera icon in the Popup Menu.

Double Buffering is a special drawing technique, that allows flicker-free animations. Double Buffering is only used if the plot control is active.

The video file name is the name of the AVI-file used for generation of video output from the plot.

2.9 Popup Menu

A click on the right mouse button brings up a menu that allows for various manipulations of the active object or the object over which the cursor is positioned.



Properties

This is where you select files to plot and everything else about the picture and simulation setup.

Add/Remove layers

In the Image manager you can select between several file types to be included as additional layers on you plot.

In the Overlay manager you have the option to choose the order by which the drawing of the layers in your plot occur. This way you choose if the layer should be in the background or foreground on your plot.

Toolbar

You can display a toolbar that enables you to move forwards and backwards in time to display results from a simulation and maybe create a video file.



The video icon creates the file.

File name legend

As default, the file name and path of your input data file is shown on the plot, but you can switch this off by selecting this menu item.

Scale legend

As default, the scale is shown on the plot, but you can with this off by selecting this menu item.

Move north arrow and convergence indicator

Selecting one of these and afterwards clicking on the desired new spot will move the North Arrow or the Convergence Indicator to the new spot. But of course only if the icon is already on the plot.

Zoom facilities

Zoom In allows you to zoom in on the plotted data and change the focus of the plot.

Zoom Out zooms out to default coverage.

Previous Zoom and **Next Zoom** zooms to the previous or following focus of the plot, respectively.

Copy to clipboard

Copies the selected data to the clipboard. The general idea is that the graphics from an object at any time can be copied to a standard Windows application which supports GDI-operations. An example could be to export the graphics to MS Word. Activate the "Copy to Clipboard" function, then switch



to MS Word and activate the "Paste"-function or press "Ctrl" V in and active Word document, and then a copy of the plot appears in MS Word.

Save to metafile

Saves the current plot to an enhanced metafile.

Save to bitmap

Saves the current snapshot plot to a bitmap file. You also have the option to choose the graphics file formats .png, .jpg or .tif instead.

Font

Defines the font used in the plots.

Help

"Help" brings up the relevant on-line help pages.

2.10 Image manager

2.10.1 Image style

Here you specify how the image is displayed in the plot.

Display style

The display style decides how the colours of the images are shown.

Copy Colours

Shows the colours of the top picture only

Copy Colours with Transparency

Shows the colours of the topmost picture's colours, but you can choose one transparent colour. Areas of your picture that has this colour will be transparent, so you can see the image below.

Blend colours

The colours of the different layers will be mixed.

Stretch mode

When a pixel image is drawn in a window such that the image is smaller in at least one direction, at least some pixels in the original image will map to a single displaced pixel. The stretch mode determines the colour of those pixels.

Halftone

Maps pixels from the source rectangle into blocks of pixels in the destination rectangle. The average colour over the destination block of pixels approximates the colour of the source pixels.

Color-on-color

Deletes all eliminated lines of pixels without trying to preserve their information.



Black-on-white

Performs an operation using the colour values for the eliminated and existing pixels. If the bitmap is a monochrome bitmap, this mode preserves black pixels at the expense of white pixels.

Image origin

The default values for the origin of the image are taken from the *.dfs/*.dt data file.

Image size

The image size is defined by the pixel width and height multiplied by the number of pixels in the image for the two directions.

2.10.2 Shape style

Here you specify the style of the points and lines/polygons that may be a part of the shape file.

2.10.3 XYZ style

Here you specify the layout of the square markers that indicate each XYZ point. The colour of the markers indicate the Z-values in the XYZ file.

You may omit the XYZ data entirely, show the squares with or without black frames and include a text string showing the value of the individual point.

Finally you can set the size of the marker (in mm).

2.10.4 Meta file style

Her you specify the location and size of the picture on the plot. The default values for the Image origin are taken from the *.dfs data file and the Meta File Size is given in number of grid points.

2.10.5 Particle style

You may display the particles in one of two ways (or both):

- **Particle track**
You can define the width, style and colour of the lines, which show tracks of the particle movements.
- **Particle instantaneous**
You can define the colour and style of the marker, which shows the instantaneous position of a particle. You have the option to show the particles for one particular time step or all particles within the simulation time.



2.11 Overlay manager

Here you specify what is shown on the plot, like palettes, filename, date, time etc.

You can also specify which item goes on top, by using the arrow buttons



to put the items higher or lower in the plotting sequence.

If e.g. an item is very low on the list (high number), it will be plotted on top of an item higher on the list, thus creating a different picture than if the two were switched around.



3 Grid Plot

Plotting tool for plotting 2D contour and vector data from grid series data files (*.dfs2).

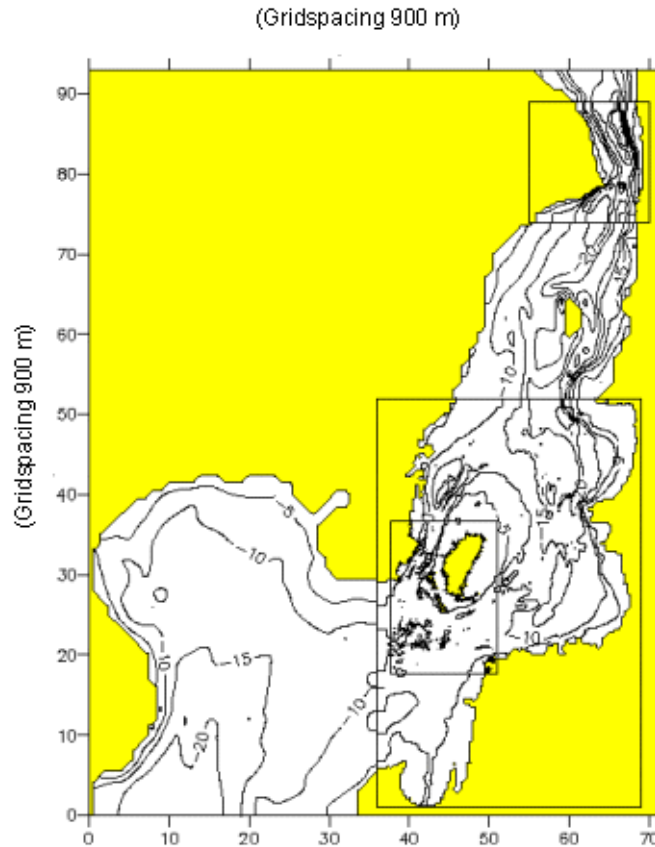


Figure 3.1 Example of grid plot

3.1 Input Data Selection

On the "Input" page, you **select the type 2 data** file(s) containing the data you wish to plot.

If you specify a MIKE 21 result file, you will also be able to plot derived items from the file, e.g. surface elevation.

The "**Master File**" is the data file covering geographically the outer-most area, i.e. the coarse grid or the main area in nested model terminology.



"Nested Files" is a list of one or more fine grid data files embedded inside the area covered by the "Master File".

Plot of data from the "Nested Files" is positioned relative to the "Master File" data using the geographical information of the data files.

3.2 Item selection

You can plot the items as **contours**. You will specify the appearance of the contours on the Contour Settings page

If the file contain fluxes (HPQ file), you can also plot some of the items as **vectors**. You will specify the layout of these vectors on the Vector Settings page.

3.2.1 General

On this page you **choose which items you want to plot**. You select the items in the list "items" and press one of the arrow buttons,



to choose if they should be plotted as contour or vector items.

If you choose to calculate the **contours** from 1 item the "standard way", you will only have to specify 1 item in the items selection's "Contour Item(s)". On the other hand, if you choose to base the contours on the **vector length**, you will have to specify 2 contour items. If the file only contains 1 item, the latter option will not be available.

On this page, you are able to choose from which items the contours should be calculated, but you will specify the appearance of the contours on the Contour Settings page.

3.2.2 Dfs2 result file

If your file has been recognized as a **Dfs2 result file** you will be prompted for the contour item and vector item that is to be plotted. You also have the possibility to plot derived items as e.g. surface elevation and velocities.

NOTE: The calculations of derived items are based on the assumption of the absolute delete value being very small, e.g. -10^{-30} . The use of other delete values may lead to incorrect derived results in grid cells next to cells holding delete values.



3.3 SubSet Selection

This is where you choose a **subarea** or a **sub-timeseries** instead of plotting the whole file. A subarea is defined by the grid numbers on the spatial axes and a sub-timeseries by the time step numbers.

Instead of selecting a subarea now, you can also choose to zoom-in in the Popup Menu. Choosing it now will probably cause less memory problems for your computer, and this will be the area that is saved to the output file.

3.4 Contour Settings

You can define the way contours are displayed. The contour is defined by a Primary option that defines the contour type and a Secondary option that depends on the primary option.

You can choose to see the contours as "boxes", following the grid cells, or you can smooth them by choosing "shading". The "**shading**" option shows different layers in different colours. If you choose "no contour", you must choose "Draw isolines" to be able to see anything at all. The isolines are lines on which the value is the same. You can see this value by choosing "With labels".

Color legend

When you choose "Shading" or "Box contours", you can also plot the color legend. The color legend can be configured using the Palette wizard. You can create, open, edit and save a palette file. You can use the dialog to calculate the limits from the data file by pressing "calculate", or you can specify the limits yourself.

If you select '<New levels/colors...>', you can define a custom palette which is either "Auto scaled" or "Fixed". On the fixed palettes, you can specify the data value ranges. (For a "land/water fixed palette" you only specify the lower limit, while the upper limit is the land value).

3.4.1 Primary option

Box Contour

Each grid cell is given a colour according to the value in the cell.

Secondary Options: No. 1

Box Contour with transparency

Each grid cell is given a colour according to the value in the cell.

Secondary Options: No. 1 and No. 2

Shaded Contour

The grid is covered with colours indicating the values in the cells, a smooth



surface is used. The land values will always be displayed in green.
Secondary options: No. 3 or No. 4 or No. 5 + No. 2

No Contour

Used in connection with background overlays. The bathymetry will not be shown.

3.4.2 Secondary option

1. **Box contour divider**
This option can be switched on and off having selected box contour for display. A system of lines will be placed on the grid displaying the transition between the colours.
2. **Transparency**
Used in connection with background overlays the value defines the transparency of the layer.
3. **Copy Colours**
The colours indicate the values in the cell.
4. **Copy Colours with Transparency**
The colours indicate the values in the cell, one colour is fully transparent (usually white).
5. **Blend colours**
All colours are transparent with a user-defined transparency (secondary option 2) and one colour is fully transparent (usually white).

3.5 Vector Settings

On this page you specify how the vectors should be scaled and the interval (in grid points) between the individual vectors.

Automatic scaling is a quick way to scale the vectors. You simply give the length (in grid points) of the longest vector on the plot. The individual plots in a series are scaled differently. When using automatic scaling, vectors are never distorted. Therefore, when you change scaling in one direction, the scaling in the other direction is automatically changed accordingly.

User specified scaling gives you full control of the vector scaling. You specify the vector length (in grid points) corresponding to typically one SI unit. The individual plots in a series are therefore given the same scaling. User specified scaling can also be used for amplifying the vector length in one direction (distortion). If e.g. 10 to the power of 3 SI-units is a more practical unit, then set 'base' equal 3, to get a more reasonable X- and Y-scale value.



3.5.1 Nested vector setup

On this dialog you specify the **offset** of and the **interval** between individual vectors for the nested, fine grid files. Both offset and interval is given in grid points.

3.6 Land Settings

3.6.1 General

On this page you define **how to represent land on the plot**.

First you specify **where to get the land information (map) from**. If you are making a bathymetry plot, then the land information can simply be taken from the plot data itself. If you are making e.g. a plot of a bed resistance map, then the land information can be taken from a bathymetry map in another data file.

Then you specify how to **distinguish land points from water points**. The value for true land is usually a minimum value. This means that all points with a value greater than or equal to the true land value (above true land) are land points.

You may also choose to specify a digitized land border to be used for land plotting, either as exported from a Digitizer file (*.bbd) or read from a XYZ ASCII file (*.xyz). Choosing the XYZ-file, you must give the projection in which the land border file data appears. This must always be one of the kind of a Universal Transverse Mercator Projection, for instance 'UTM-33' for zone 33, or 'BTM' for a local UTM zone defined in the utm.dat file in your MIKEZero\bin directory.

You specify the **land appearance** and the edge drawing of land areas. If shaded land is chosen, then the specified colour the is used.

Finally, the **edge drawing** of land areas should be defined.

3.6.2 Dfs2 result file

If your file has been recognized as a **Dfs2 result file** the land data is typically taken from the first skip record (first static item) in the plot data file.

Then you specify how to **distinguish land points from water points**. The value for true land, i.e. points that for sure never are flooded, is usually a minimum value. This means that all points with a value greater than or equal to the true land value (above true land) are land points.

You may also choose to specify a digitized land border to be used for land plotting, either as exported from a Digitizer file (*.bbd) or read from a XYZ ASCII file (*.xyz). Choosing the XYZ-file, you must give the projection in



which the land border file data appears. This must always be one of the kind of a Universal Transverse Mercator Projection, for instance 'UTM-33' for zone 33, or 'BTM' for a local UTM zone defined in the utm.dat file in your MIKEZero\bin directory.

You specify the **land appearance** and the edge drawing of land areas. If shaded land is chosen, then the specified colour the is used. Dried land can be specified to have a different appearance than true land.

3.7 Coordinate System

In case you have geographical information in your input data file, you can **superimpose on the plot geographical coordinates** either as map projection coordinates or as latitudes and longitudes. Furthermore, you have the opportunity to **add plotting symbols indicating true north and the convergence**, defining the angle between the longitude and the northing line for the map projection. The indications of true north and convergence can be moved later, directly on the picture, using the Popup Menu.

When you are superimposing a coordinate system, the one specified in the "Map projection type" box will be used. The default **Map projection** is the one specified in your background input data file.

If you select **Northing coordinates always positive**, The northing coordinates will be as 1) and 3) in Figure 3.2. If you don't select it, the southern hemisphere northing coordinates will be as 2).

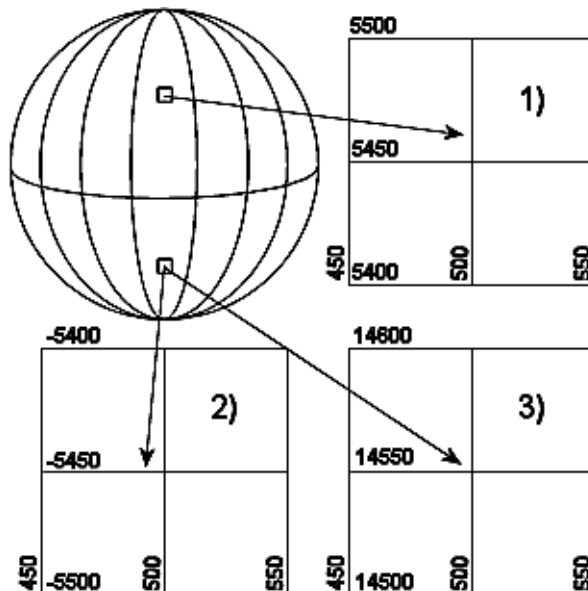


Figure 3.2 Illustration of UTM-zone



Finally, you can specify additional **axis annotation** and a **geodetic scale**. If you choose to have grid units, the scaling is not available since it doesn't make sense in this case.

Choosing automatic scaling will show the scale on the plot according to how you size the plot manually. Choosing a defined scale, ex. 1:100000, will automatically size the plot according to that scale.

3.8 Points and Lines

In case you want to add descriptive information on the plot you can **superimpose** individually defined Points and Lines. For both points and lines you may add strings to be displayed. See Figure 3.3 for an example.

First you have to define the coordinate definition, that is in which coordinate system do you define the coordinates (x,y) for the points and lines.

Next you define the Points and Lines that should be displayed on the plot.

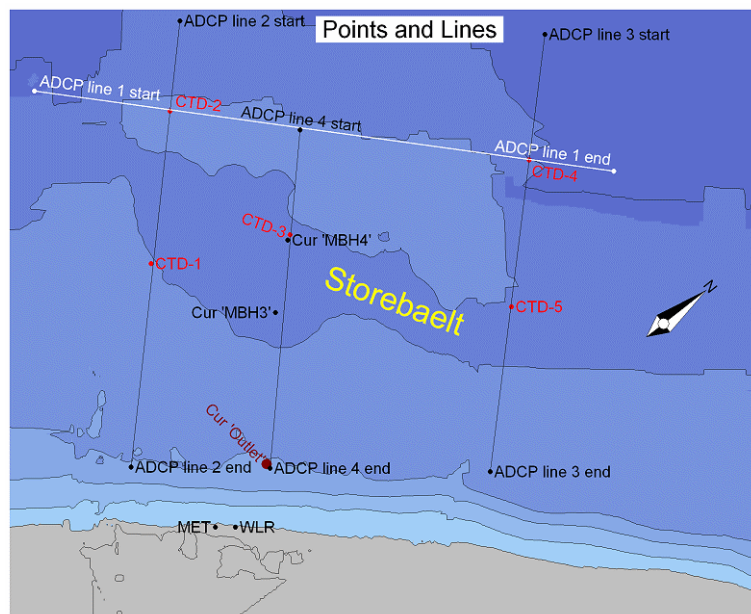


Figure 3.3 Example of grid plot with additional points and lines

3.8.1 Points

You can define any number of points.

For each point you define the following:



- the coordinate (x,y)
- a text string related to the coordinate
- Properties for the point marker and text to be displayed

3.8.2 Lines

You can define any number of lines.

For each line you define the following:

- the start coordinate (x0,y0) and end coordinate (x1,y1)
- a text string related to the start coordinate
- the end coordinate (x1,y1)
- a text string related to the end coordinate
- Properties for the point markers and text to be displayed

3.8.3 Properties

Color

The colour will be the same for both markers, lines and text.

Marker and line

You define the size of the marker and if it is to be displayed at all. For lines you specify the thickness of the line.

Text

The text is defined by the size, angle, alignment and background style. For lines the text for both start point and end point can be displayed individually.

3.9 Output Settings

With this page you can control the output settings: Double Buffering and name of video file.

The file is **recorded** by the Toolbar button with the video camera icon in the Popup Menu.

Double Buffering is a special drawing technique, that allows flicker-free animations. Double Buffering is only used if the plot control is active.

The video file name is the name of the AVI-file used for generation of video output from the plot.



3.10 Popup Menu

A click on the right mouse button brings up a menu that allows for various manipulations of the active object or the object over which the cursor is positioned.

Properties

This is where you select files to plot and everything else about the picture and simulation setup.

Add/Remove layers

In the Image Manager you can select between several file types to be included as additional layers on you plot.

In the Overlay Manager you have the option to choose the order by which the drawing of the layers in your plot occur. This way you choose if the layer should be in the background or foreground on your plot.

Toolbar

You can display a toolbar that enables you to move forwards and backwards in time to display results from a simulation and maybe create a video file.



The video icon creates the file.

File name legend

As default, the file name and path of your input data file is shown on the plot, but you can switch this off by selecting this menu item.

Scale legend

As default, the scale is shown on the plot, but you can with this off by selecting this menu item.

Aspect ratio

You have the possibility to change the aspect ratio between the x-axis and y-axis selecting this menu item.

Move north arrow and convergence indicator

Selecting one of these and afterwards clicking on the desired new spot will move the North Arrow or the Convergence Indicator to the new spot. But of course only if the icon is already on the plot.

Zoom facilities

Zoom In allows you to zoom in on the plotted data and change the focus of the plot.



Zoom Out zooms out to default coverage.

Previous Zoom and **Next Zoom** zooms to the previous or following focus of the plot, respectively.

Copy to clipboard

Copies the selected data to the clipboard. The general idea is that the graphics from an object at any time can be copied to a standard Windows application which supports GDI-operations. An example could be to export the graphics to MS Word. Activate the "Copy to Clipboard" function, then switch to MS Word and activate the "Paste"-function or press "Ctrl" V in and active Word document, and then a copy of the plot appears in MS Word.

Save to metafile

Saves the current plot to an enhanced metafile.

Save to bitmap

Saves the current snapshot plot to a bitmap file. You also have the option to choose the graphics file formats .png, .jpg or .tif instead.

Font

Defines the font used in the plots.

Note: The latest font settings will be used for all plot objects when opening the file.

Help

"Help" brings up the relevant on-line help pages.

3.11 Image Manager

Specify a new image in the list by pressing



You can choose between the following file types:

1. Image File
You can select a *.bmp, *.jpg, *.gif, *.png or *.tif image. You specify the Image style in the plot by clicking the **Edit...** button.
2. Shape File
You may select a *.shp file that is often used in GIS environments. You specify the layout of the shape file by clicking the **Edit...** button.
NOTE: As the shape files cannot be rotated in Plot Composer this only works for grid series data files (.dfs2) that are not rotated, i.e. orientation must be 0 deg.N.
3. XYZ File
You may select a *.xyz file that displays the positions and z-values for



given (x,y) coordinate positions. You have the option to import the data from various different coordinate systems. You specify the display of the points by clicking the **Edit...** button.

4. **River File**
You may select a MIKE 11 River network file (*.nwk11) that contain a river network. You have the option to import the data from various different coordinate systems. You specify the display of the river by clicking the **Edit...** button.
5. **Enhanced MetaFile**
You can select a *.emf image. You specify the Image style in the plot by clicking the **Edit...** button.
6. **Particle File**
You can select a Particle Tracking file (*.xml) that contains information about the particle tracks and the instantaneous location of each particle.

3.11.1 Image style

Here you specify how the image is displayed in the plot.

Display style

The display style decides how the colours of the images are shown.

Copy Colours

Shows the colours of the top picture only

Copy Colours with Transparency

Shows the colours of the topmost picture's colours, but you can choose one transparent colour. Areas of your picture that has this colour will be transparent, so you can see the image below.

Blend colours

The colours of the different layers will be mixed.

Stretch mode

When a pixel image is drawn in a window such that the image is smaller in at least one direction, at least some pixels in the original image will map to a single displaced pixel. The stretch mode determines the colour of those pixels.

Halftone

Maps pixels from the source rectangle into blocks of pixels in the destination rectangle. The average colour over the destination block of pixels approximates the colour of the source pixels.

Color-on-color

Deletes all eliminated lines of pixels without trying to preserve their information.



Black-on-white

Performs an operation using the colour values for the eliminated and existing pixels. If the bitmap is a monochrome bitmap, this mode preserves black pixels at the expense of white pixels.

Image origin

The default values for the origin of the image are taken from the *.dfs/*.dt data file.

Image size

The image size is defined by the pixel width and height multiplied by the number of pixels in the image for the two directions.

3.11.2 Shape style

Here you specify the style of the points and lines/polygons that may be a part of the shape file.

3.11.3 XYZ style

Here you specify the layout of the square markers that indicate each XYZ point. The colour of the markers indicate the Z-values in the XYZ file.

You may omit the XYZ data entirely, show the squares with or without black frames and include a text string showing the value of the individual point.

Finally you can set the size of the marker (in mm).

3.11.4 Metafile style

Her you specify the location and size of the picture on the plot. The default values for the Image origin are taken from the *.dfs data file and the metafile size is given in number of grid points.

3.11.5 Particle style

You may display the particles in one of two ways (or both):

- **Particle track**
You can define the width, style and colour of the lines, which show tracks of the particle movements.
- **Particle instantaneous**
You can define the colour and style of the marker, which shows the instantaneous position of a particle. You have the option to show the particles for one particular time step or all particles within the simulation time.



3.12 Overlay Manager

Here you specify what is shown on the plot, like palettes, filename, date, time, etc.

You can also specify which item goes on top, by using the arrow buttons



to put the items higher or lower in the plotting sequence.

If e.g. an item is very low on the list (high number), it will be plotted on top of an item higher on the list, thus creating a different picture than if the two were switched around.





4 Profile Plot

Plotting tool for plotting data from line series data files (*.dfs1 or *.dt1).

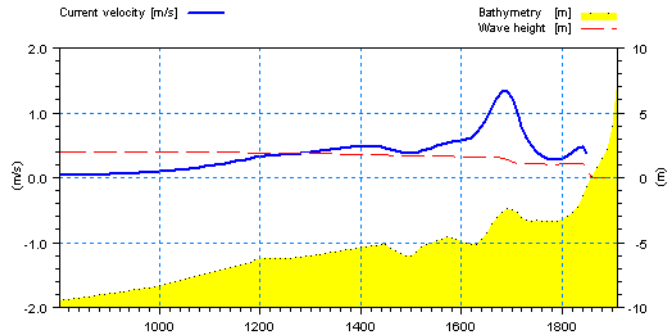


Figure 4.1 Example of profile plot showing cross-shore variation of wave height and longshore velocity

4.1 Item Selection

On this page you choose which items you want to plot.

4.2 Step Configuration

On this page you choose which time steps to plot.

4.3 Plot Type

On this page you define the the type of plot.

You can choose

- Transect Plot
- Vertical Profile Plot.

For the Vertical Profile Plot, you can invert the Y-axis.

4.4 X-axis Definition

On this page you define the limits of the x-axis.



4.5 Y-axis Definition

On this page you define the limits of the y-axis

First, you specify for each item if it should be plotted relative to the left placed or the right placed y-axis.

Then, you specify the limits of the y-axis. In case you have selected items for both the left and the right y-axis, you must enter specifications for both the axes.

4.6 Curve Definitions

On this page you define the appearance of the curves.

For each curve you define

- the line width,
- the line style as solid, dashed, dotted, dashed-dotted, or filled,
- the line colour, and
- whether or not the line should be displayed

and you may choose to display marker symbols and define

- the marker symbol,
- the marker size,
- the marker fill style, and
- the marker colour.

4.7 Text Annotations

On this page you choose which text strings that are to be put on the plot axis and header.

4.8 Popup Menu

You will get the pop-up menu if you right click with the mouse on the picture. The items in the pop-up menu are (along with some standard zoom and save function):

Properties

This is where you select files to plot and everything else about the picture and simulation setup.



Toolbar

Moving forwards and backwards in time to create a simulation and maybe a video file.

Note: It is always possible to stop an animation by pressing the <esc> key.



The video icon creates the file. The name of the video file is defined on the Step Configuration page.



The envelope icon enable the plotting of curves for several time steps simultaneously. The time step between the plotted curves is defined on the Step Configuration page.

File name legend

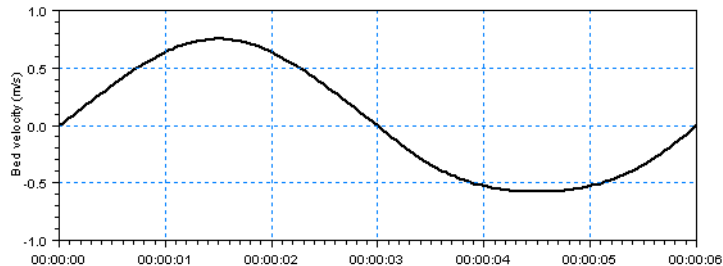
As default, the file name and path of your input data file is shown on the plot, but you can switch this off by selecting this menu item.





5 Time Series Plot

Plotting tool for plotting time series data (*.dfs0 or *.dt0 files).



5.1 Items

Use this page to select the data file(s) containing the item(s) for your time series plot.

NOTE:

Per default the values are shown in user units, however it is possible to select the unit by which the values are to be displayed.

5.2 X-Axis Definitions

On this page you define the limits of the x-axis.

5.3 Y-Axis Definitions

On this page you define the layout of the y-axis (axes).

First, you specify for each item if it should be plotted relative to the left placed or the right placed y-axis.

Then, you specify the limits and properties of the y-axis. In case you have selected items for both the left and the right y-axis, you must enter specifications for both the axes.

You have to define the axis type as one of three:

- Linear with automatic labels
- Linear with manual labels
- Logarithmic with automatic labels

**NOTE:**

For manual labels you have to define the interval settings on the axis. The axis extension will become fixed to the defined Y-values.

5.4 Curve Definitions

On this page you define the appearance of the curves.

For each curve you define

- the line width,
- the line style as solid, dashed, dotted, dashed-dotted, or filled,
- the line colour, and
- whether or not the line should be displayed

and you may choose to display marker symbols and define

- the marker symbol,
- the marker size,
- the marker fill style, and
- the marker colour.

5.5 Text Annotation

On this page you define a title for your plot.

5.6 Popup Menu

A click on the right mouse button brings up a menu that allows for various manipulations of the active object or the object over which the cursor is positioned.

Properties

A plot control specific property page appears, in which the properties of the plot can be manipulated.

Time series legend

Places a legend in the time series plot container depicting the item(s) plotted.

File name legend

Places a legend in the plot container depicting the names of the data files from which the plotted data have been taken.



Zoom facilities

Zoom In allows you to zoom in on the plotted data and change the focus of the plot.

Zoom Out zooms out to default coverage.

Previous Zoom and **Next Zoom** zooms to the previous or following focus of the plot, respectively.

Draw grid

Adds a grid to the plot.

Copy to clipboard

Copies the selected data to the clipboard. The general idea is that the graphics from an object at any time can be copied to a standard Windows application which supports GDI-operations. An example could be to export the graphics to MS Word. Activate the "Copy to Clipboard" function, then switch to MS Word and activate the "Paste"-function or press "Ctrl" V in and active Word document, and then a copy of the plot appears in MS Word.

Save to metafile

Saves the current plot to an enhanced meta file.

Save to bitmap

Saves the current snapshot plot to a bitmap file.

Font

Defines the font used in the plots.

Help

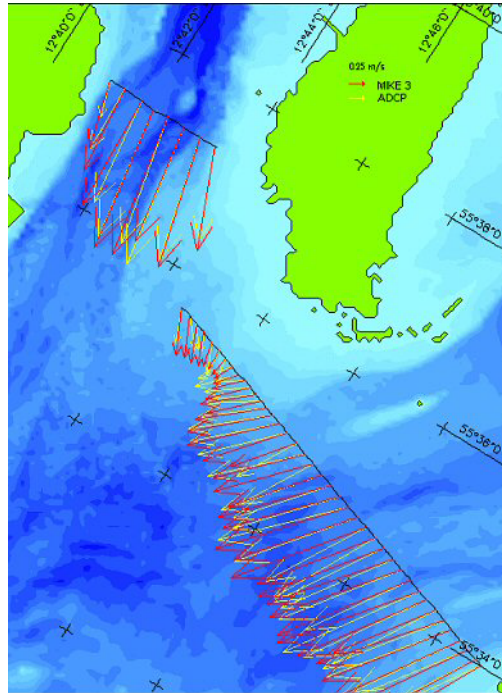
"Help" brings up the relevant on-line help pages.





6 ADCP 2D Plot

Plotting tool for plotting ADCP data.



6.1 Input Data Selection

You must define at least one dfs1 file that contains ADCP track data. Furthermore you must define a dfs2 or dfs3 file to represent the domain area. All dfs1 files must have the same start date.

6.1.1 ADCP type 1 files

An ADCP input file contains information about the temporal variation of location and related velocity vector for a given measured track. Each grid point in the dfs1 file represent a measurement.

The file can initially be generated by the template 'ADCP Vector Plot - 1 bin' in the Profile Series Editor. This file type contains 1 time step and 5 items:

1. Off-set (number of seconds after start for each measurement)
2. Easting (coordinate location of measurement)
3. Northing (coordinate location of measurement)



4. u (u-velocity component)
5. v (v-velocity component)

It is possible to include more items in the file, e.g. items with flux values. The coordinates in the ADCP file in items 2 and 3 refer to the map projection specified in the background file.

An example is shown in Figure 6.1.

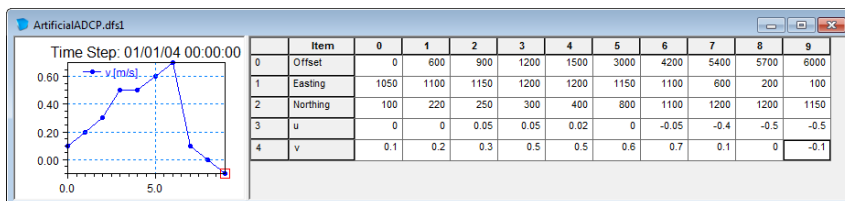


Figure 6.1 Example of ADCP type 1 file

6.1.2 Background

The background file provides the limits of the domain area, contour values and optionally comparative flow vectors. You may select one of three types of dfs files:

- Bathymetry file
(a dfs2 file containing the bathymetry)
- HD file including result components
(MIKE 21 HD and MIKE 3 HD standard dfs2/dfs3 output files, vector components can be derived and shown for comparison)
- HD file excluding result components
(MIKE 21 HD and MIKE 3 HD dfs2/dfs3 output files, vector components cannot be shown for comparison)

In case the file is a dfs3 file, you have to specify which layer number that is to be used for the background values.

NOTE: In case the background file is selected as a HD file, the time steps in the HD file must cover the time period described in the ADCP file (i.e. start time + off-set).

6.2 Items

The way of selecting items from the background file depends on the chosen type of background data.



6.2.1 Item selection for dfs2 result files

Your file has been recognized as a **MIKE 21 Hydrodynamic Module result file**. Therefore you will also have the possibility to plot derived items as e.g. surface elevation.

On this page, you are able to choose from which item the contours should be calculated, but you will specify the appearance of the contours on the Contours page.

If the file contain fluxes (HPQ file), you can also plot some of the items as **vectors** in the same position as the ADCP vectors. You will specify the layout of these vectors on the ADCP Vectors page.

6.2.2 Item selection

On this page you **choose which items you want to plot**. You select the items in the list "items" and press one of the arrow buttons,



to choose if they should be plotted as contour or vector items.

If you choose to calculate the **contours** from 1 item the "standard way", you will only have to specify 1 item in the items selection's "Contour Item(s)". On the other hand, if you choose to base the contours on the **vector length**, you will have to specify 2 contour items. If the file only contains 1 item, the latter option will not be available.

On this page, you are able to choose from which items the contours should be calculated, but you will specify the appearance of the contours on the Contours page.

6.3 SubSet

This is where you can choose a **subarea** and a specific time step (**sub-time-series**) for the background values instead of plotting the whole domain area using time step 0.

Instead of selecting a subarea now, you can also choose to zoom-in using the functionality in the Popup Menu. However, the settings of the zoom-in display will not be saved in the plot file and as will the subarea settings.

NOTE: The sub-series selection refers to the available data from the background file. The start time step specifies the time step for the shown background contours.



6.4 Contours

You can choose to see the contours as "boxes", following the grid cells, or you can smooth them by choosing "shading".

The "**shading**" option shows different layers in different colours. If you choose "no shading", you must choose "Draw isolines" to be able to see anything at all. The isolines are lines on which the value is the same. You can see this value by choosing "With labels".

Color legend

When you choose "Shading" or "Box contours", you can also plot the color legend. The color legend can be configured using the Palette wizard. You can create, open, edit and save a palette file. You can use the dialog to calculate the limits from the data file by pressing "calculate", or you can specify the limits yourself.

If you select '<New levels/colors...>', you can define a custom palette which is either "Auto scaled" or "Fixed". On the fixed palettes, you can specify the data value ranges. (For a "land/water fixed palette" you only specify the lower limit, while the upper limit is the land value).

6.5 ADCP Vectors

On this page you specify how the vectors should be scaled, the type of vector derived from the background file (if available) and the frequency between the individual vectors in the ADCP type 1 file(s).

You can also specify the colour and line width of the sailed route, the ADCP vectors and the interpolated vectors (if available) derived from the background file.

6.6 Land options

The way to represent land on a plot depends on the chosen type of background data.

6.6.1 Land settings for dfs2 result files

On this page you define how to represent land on a plot of MIKE 21 Hydrodynamic Module results.

First you specify where to get the land information from. The land data is **typically taken from the first skip record** (first static item) in the result data file.



You may also choose to specify a digitized land border to be used for land plotting, read from a XYZ ASCII file (*.xyz). You must give the map projection in which the land border file data appears.

Then you specify how to **distinguish land points from water points**. The value for true land, i.e. points that for sure never are flooded, is usually a minimum value. This means that all points with a value greater than or equal to the true land value (above true land) are land points.

You specify the **land appearance** of true and dried land areas. If shaded land is chosen, then the specified colour the is used.

Finally, the **edge drawing** of land areas should be defined.

6.6.2 Land settings in general

On this page you define **how to represent land on the plot**.

First you specify **where to get the land information (map) from**. If you are making a bathymetry plot, then the land information can simply be taken from the plot data itself. If you are making e.g. a plot using a current speed map, then the land information can be taken from a bathymetry map in another data file. With model result files, the land data is typically taken from the first skip record (first static item) in the plot data file.

You may also choose to specify a digitized land border to be used for land plotting, read from a XYZ ASCII file (*.xyz). You must give the map projection in which the land border file data appears.

Then you specify how to **distinguish land points from water points**. The value for true land, i.e. points that for sure never are flooded, is usually a minimum value. This means that all points with a value greater than or equal to the true land value (above true land) are land points.

Finally you specify the **land appearance** and the edge drawing of land areas. If shaded land is chosen, then the specified colour the is used.

6.7 Coordinate System

In case you have geographical information in your input data file, you can **superimpose on the plot geographical coordinates** either as map projection coordinates or as latitudes and longitudes. Furthermore, you have the opportunity to **add plotting symbols indicating true north and the convergence**, defining the angle between the longitude and the northing line for the map projection. The indications of true north and convergence can be moved later, directly on the picture, using the Popup Menu.

When you are superimposing a coordinate system, the one specified in the "Map projection type" box will be used. The default **Map projection** is the one specified in your background input data file.

If you select **Northing coordinates always positive**, the northing coordinates will be as 1) and 3) in Figure 6.2. If you don't select it, the southern hemisphere northing coordinates will be as 2).

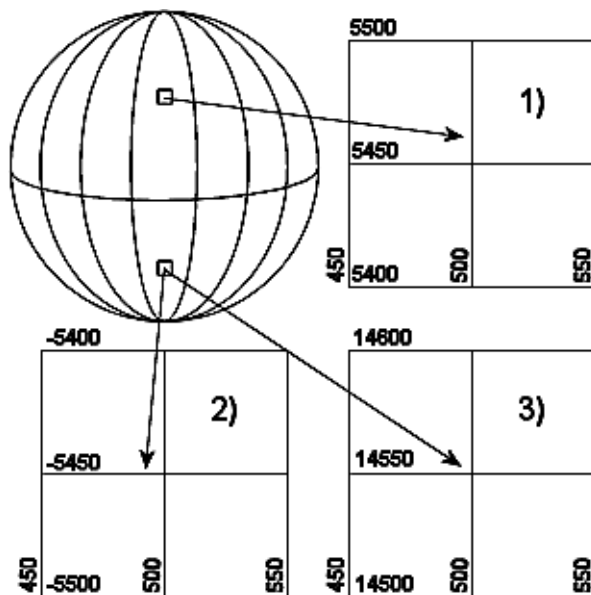


Figure 6.2 Illustration of UTM-zone

Finally, you can specify additional **axis annotation**.

6.8 Popup Menu

You will get the pop-up menu if you right click with the mouse on the picture. The items in the pop-up menu are (along with some standard zoom and save function):

Properties:

This is where you select files to plot and everything else about the picture and simulation setup.

File name legend

As default, the file name and path of your input data file is shown on the plot, but you can switch this off by selecting this menu item.



Copy or save graphics

Copies the selected data to the clipboard or saves the data in an enhanced metafile or a different graphic file format (.bmp, .png, .jpg or .tif).

Move North Arrow and Convergence Indicator

Selecting one of these and afterwards clicking on the desired new spot will move the North Arrow or the Convergence Indicator to the new spot. But of course only if the icon is already on the plot.

6.9 Examples

The following examples illustrate some of the capabilities in ADCP plot.

6.9.1 Track on bathymetry

A moving vessel has carried out measurements every minute for a few hours to illustrate the currents through a straight. The measured transect data file is shown in Figure 6.3.

The sailing path and corresponding measurements are illustrated in Figure 6.4 using a dfs2 bathymetry file as background to specify the domain.

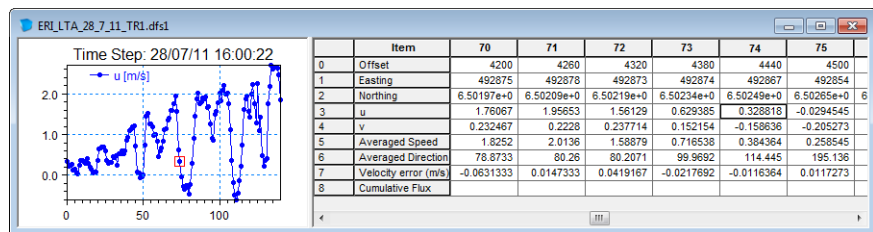


Figure 6.3 ADCP track file of measured currents

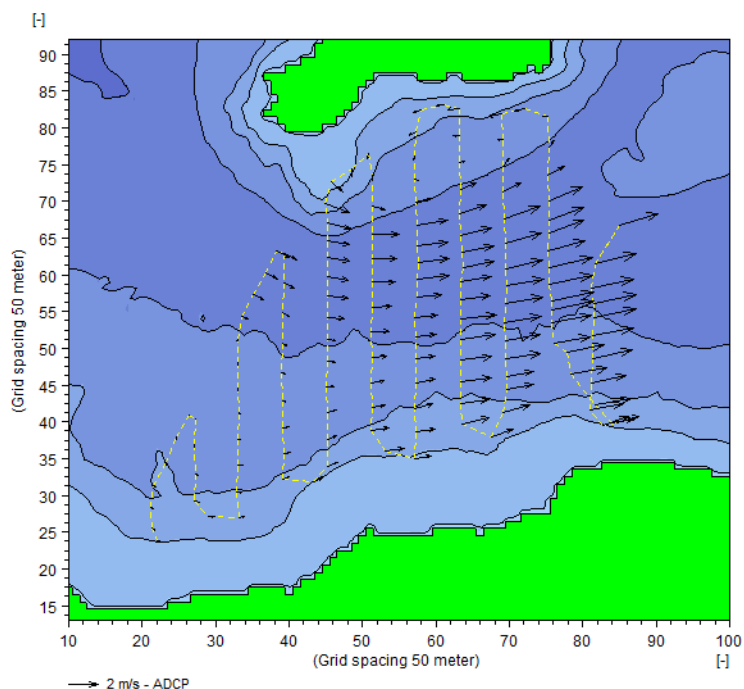


Figure 6.4 ADCP plot showing ADCP data on top of bathymetry
 Yellow stippled line: sailing path
 Black arrows: Measured current speed

6.9.2 Comparison with MIKE 21 results

Two simple ADCP track data files with artificial user defined values has been generated for a simple corner example, where currents have been generated using MIKE 21 HD by slowly raising the water level at the south boundary. The track data files are shown in Figure 6.5. The two track files are almost identical: In the first file (Track A) the initial data correspond to the start of the simulation (offset = 0), whereas in the second file (Track B) the track data has been shifted in one hour in time and 200 m to the east.

The left figure in Figure 6.6 shows the Track A data compared with the derived velocities from the background HD result file. Notice that the derived velocities for the first ADCP measurements are low, corresponding to the short time from the start of the HD simulation (offset = 0).

The right figure in Figure 6.6 shows the Track B data compared with the derived velocities from the background HD result file. Because the ADCP measurements in this case are defined as beginning with an offset of one hour from simulation start, the derived velocity vectors are larger, illustrating fully developed flow

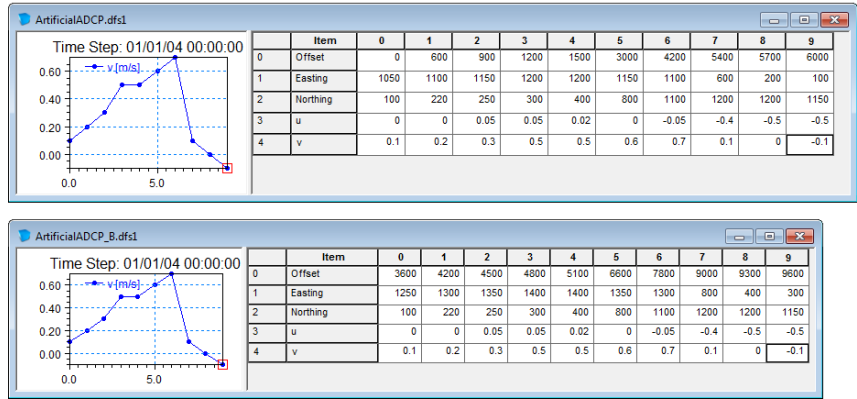


Figure 6.5 ADCP track file with user-defined data
Upper: Track data A, Lower: Track data B

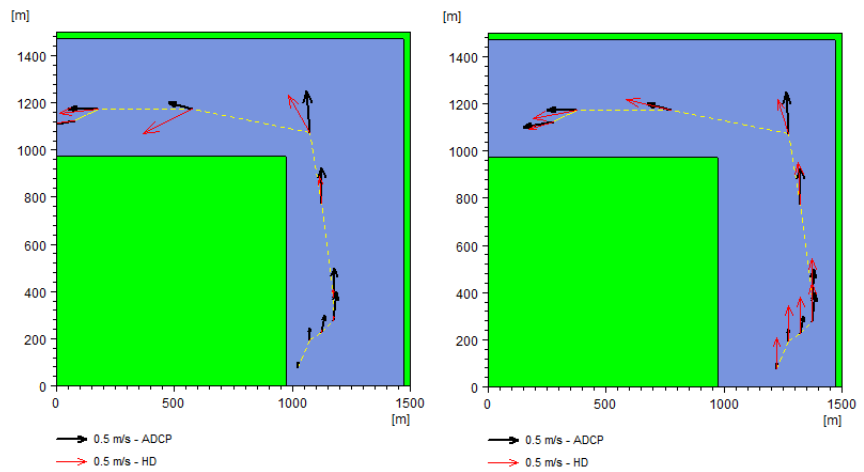


Figure 6.6 ADCP track files compared to HD simulation results
Left: ADCP track file A, Right: ADCP track file B

Figure 6.7 shows the two tracks in the same plot. Note that the derived vectors from the HD simulation are using the offset value from the ADCP data file to find the related time step values in the HD result file.

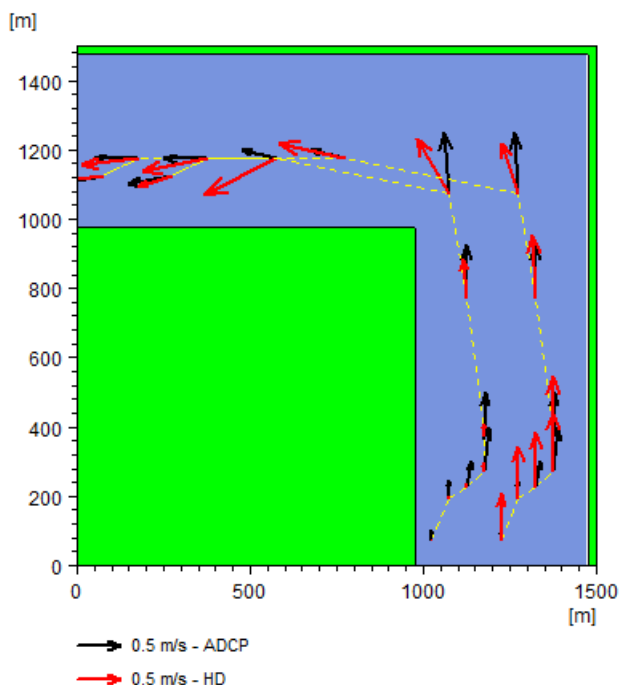


Figure 6.7 ADCP track files A and B compared to HD simulation results

6.9.3 Tracks with surface level

A moving vessel has carried out measurements approximately 5 times a minute to illustrate the currents through a straight. Measurements were carried simultaneously in three levels and the measured transect data files are shown in Figure 6.8.

The sailing path and corresponding measurements are illustrated in Figure 6.9 using a dfs2 surface level variation file as background to specify the domain area and the background values. The sub-series time step is selected to represent the conditions at the approximate time of the measurements. The land contours are defined from an ascii xyz file with land border coordinates.

In the start the vessel moved without recording velocity data. Note that when the measurements started (offset=916 s), only velocities in the upper level was recorded. At the next offset point recordings were made at all three levels. This is illustrated in Figure 6.10 where the first recording point only shows one ADCP vector whereas the next recording points shows 3 ADCP vectors each, representing the three recording levels. Notice also the slight variations in current vectors for the same recording point, but measured at different levels.

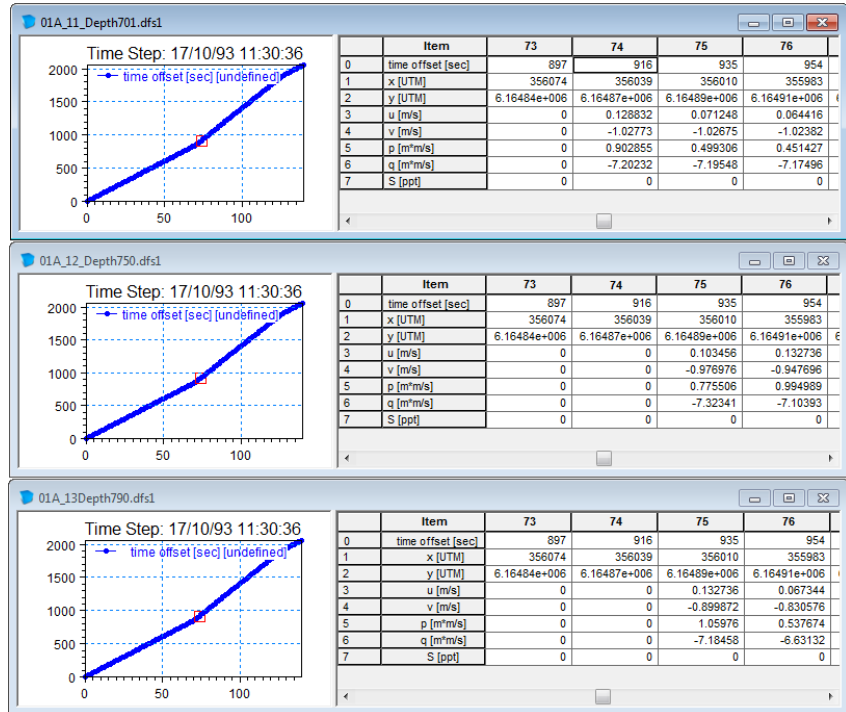


Figure 6.8 ADCP track measurements in three levels
 Upper: measurements at level -7.05 m
 Middle: measurements at level -7.50 m
 Lower: measurements at level -7.90 m

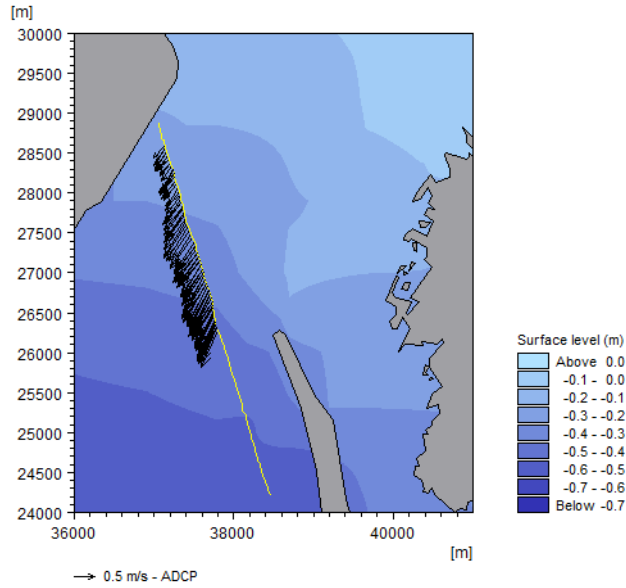


Figure 6.9 ADCP plot showing ADCP data for three levels on top of simulated surface level.

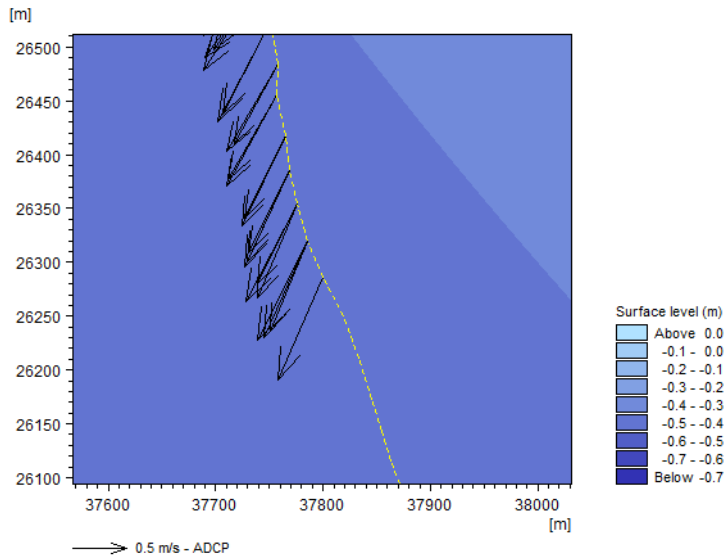


Figure 6.10 Detail of Figure 6.9. Note the difference in number of ADCP vectors at each recording point and the slight variation of velocity vectors for the different levels at a single point.



7 Annotation Plot

The Annotation Plot is a tool for adding text strings to your plot composer plot.

Annotation text in Plot Composer

7.1 Annotation Plot Properties

"Properties" is the specific property page for the current plot control, in which the properties of the plot can be manipulated.

On this property page you may:

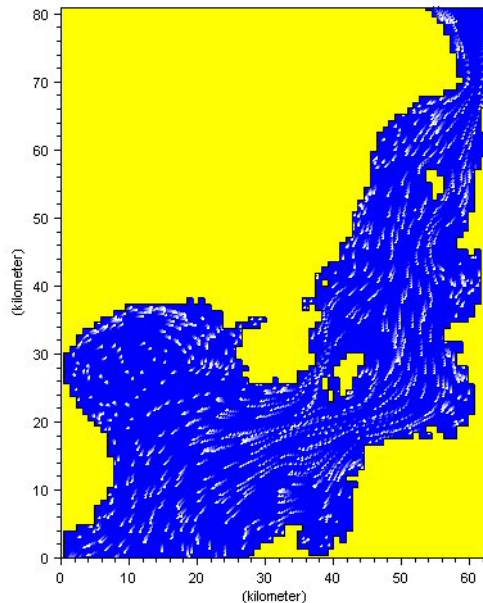
- Write a text string to be displayed
- Define which font, colour and size to be used for the text
- Define the background colour
- Define the layout of the surrounding box





8 Flow Visualisation

Flow Visualisation is a tool for making flow visualisation plots from **MIKE 21 and MIKE 3 hydrodynamic results**. The plotting tool will visualize the flow by injecting a number of particles and follow the particles as they are advected by the flow field.



For all specifications, with the exception of I/O file names, default values are supplied. You can change these specifications by modifying the corresponding dialog entries.

8.1 Input Specification

On this page, you specify the **MIKE 21 and MIKE 3 model results data** you wish to process. The tool supports following type of data:

- MIKE 21 result file (dfs2 data file)
 - Water level (H) and fluxes (P and Q)
 - Surface elevation (S) and velocities (U and V)
- MIKE 3 result file (dfs2 data file)
 - Surface elevation (S) and velocities at specific layer (U and V)

For other combinations the program will give an error message.



You can use the **“Reset” button** in cases where you change your input file. If you already made a setup with a previous file, but now you want all the default values, e.g. subarea etc., to come from the new file, you can press the reset button after selecting the new file.

You can choose the time step for your particle simulation by setting the **Interpolate to XX frames per time step**, where time step refers to time step in your input file. If you have saved your flow model result file with, say, ½ hour time step, setting the “Interpolate to XX frames per time step” value to 5, you will run your particle simulation with time steps of 5 minutes.

The **number of milliseconds per frame** is how long you need to look at each simulation step on the screen. e.g. how often the picture is updated during the simulation.

If you do not choose **double buffering when redrawing** due to memory problems, and you are saving your output to an *.avi file, please make sure to keep the full area visible on the screen and not to put any other windows on top while recording.

8.2 Particles

On this page you define **how your particles appear**. Colour, diameter and stuff like that. You can also choose whether your particles should be randomly or uniformly distributed over the area. If you want to place a single particle in the area, and calculate the trajectory, please use the “Track Flow Visualisation” plot, also provided with the Plot Composer.

The **length of the particle** defines how you see the trajectory behind the moving particle. If you define the length to be 6, you will see a tail of 6 particles, faded out at the end.

The **particle density per unit** is calculated as follows: During the simulation, the area is split up in smaller cells. The number of particles is counted within these cells. If the number of particles in a particular cell becomes smaller or larger than the minimum or maximum specified, particles will be added or removed.

8.3 Land/Water Specifications.

You must define the appearance of land areas, water areas and the land-water border.



8.4 SubSet Specifications

Here you can define the subarea of your file and the sub-timeseries you want to plot. There is also a zoom function in the Popup Menu, so you can always choose your subarea directly on the picture later.

8.5 Coordinate System

In case you have geographical information in your input data file, you can **superimpose on the plot geographical coordinates** either as map projection coordinates or as latitudes and longitudes. Furthermore, you have the opportunity to **add plotting symbols indicating true north and the convergence**, defining the angle between the longitude and the northing line for the map projection. The indications of true north and convergence can be moved later, directly on the picture, using the Popup Menu.

When you are superimposing a coordinate system, the one specified in the "Map projection type" box will be used. The default **Map projection** is the one specified in your background input data file.

If you select **Northing coordinates always positive**, The northing coordinates will be as 1) and 3) in Figure 8.1. If you don't select it, the southern hemisphere northing coordinates will be as 2).

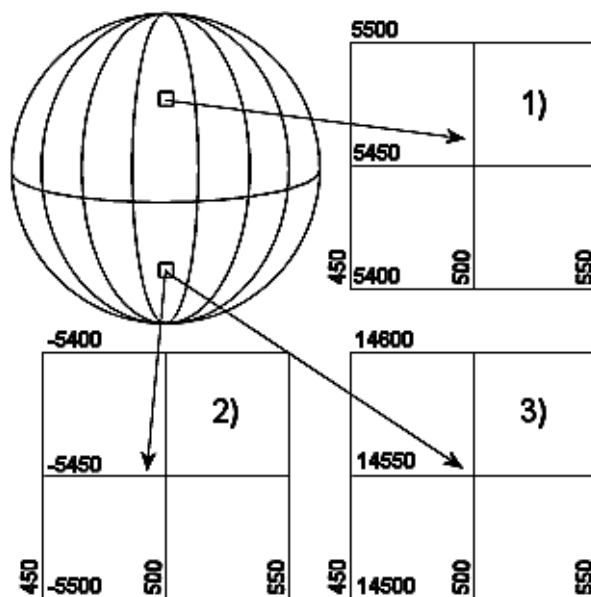


Figure 8.1 Illustration of UTM-zone



Finally, you can specify additional **axis annotation** and a **geodetic scale**. If you choose to have grid units, the scaling is not available since it doesn't make sense in this case.

Choosing automatic scaling will show the scale on the plot according to how you size the plot manually. Choosing a defined scale, ex. 1:100000, will automatically size the plot according to that scale.

8.6 Output Specification

Here you save your *.avi **animation output** file.



The file is recorded by the Toolbar button with the video camera icon in the Popup Menu.

The **Frame rate per second** is the rate of playing the video *.avi file afterwards.

8.7 Popup Menu

You will get the pop-up menu if you right click with the mouse on the picture. The items in the pop-up menu are (along with some standard zoom and save function):

Properties:

This is where you select files to plot and everything else about the picture and simulation setup.

Toolbar:

Moving forwards and backwards in time to create a simulation and maybe a video file.



The video icon creates the file.

File name legend:

As default, the file name and path of your input data file is shown on the plot, but you can switch this off by selecting this menu item.

Move North Arrow and Convergence Indicator:

Selecting one of these and afterwards clicking on the desired new spot will move the North Arrow or the Convergence Indicator to the new spot. But of course only if the icon is already on the plot.



9 Probability Plot

The probability plot control is an efficient tool for plotting histograms and cumulative distributions of any time series data. In addition user-defined fractiles can be calculated and added to the plot.

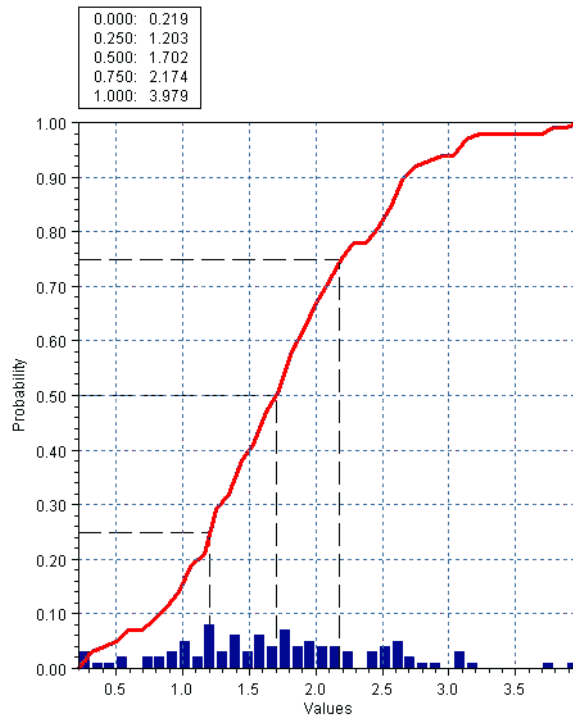


Figure 9.1 Example of probability plot showing probability distribution of wave heights and the cumulative distribution curve.

9.1 Items

Use this page to select the data file containing the item for your probability plot.

9.2 Plot Properties

Plot Type

You can define the plot type as probability distribution (bars) and/or cumulative distribution (curve).

It's possible to choose the probabilities so be showed as relative values (0-1) or as percent (0-100%).



Bin

You can specify maximum and minimum values for the actual timeseries, that should be included in the analysis. Also you can specify the number of bins which corresponds to the number of showed bars in the probability distribution. The actual number of observations/simulations included and excluded in the analysis are also showed on the page.

Fractiles

For the cumulative distribution, you can specify a number of fractiles (2 to 21), which in the default case will be regular distributed in the interval from 0-100%. The calculated values will be showed as dashed lines on the plot and they may be showed as text on the plot too.

You can also specify individual values of fractiles which then will be calculated, when the apply button of the page is pushed.

9.3 X-Axis Definitions

On this page you define the layout of the x-axis.

You can specify the limits of the x-axis as well as give the axis an annotation.

9.4 Y-Axis Definitions

On this page you define the layout of the y-axis (axes).

You can specify the limits of the y-axis as well as give the axis an annotation. You also have the option to select the axis type to be shown as linear or logarithmic.

9.5 Curve Definitions

On this page you define the appearance of the item displayed as bars and/or curve in the plot.

9.6 Text Annotations

Here you have the option to give the plot a descriptive title.

9.7 Popup Menu

A click on the right mouse button brings up a menu that allows for various manipulations of the active object or the object over which the cursor is positioned.



Properties

A plot control specific property page appears, in which the properties of the plot can be manipulated.

Fractiles

Places a text box in the probability plot container depicting the fractile values of the item plotted.

File name legend

Places a legend in the plot container depicting the names of the data files from which the plotted data have been taken.

Zoom facilities

Zoom In allows you to zoom in on the plotted data and change the focus of the plot.

Zoom Out zooms out to default coverage.

Previous Zoom and **Next Zoom** zooms to the previous or following focus of the plot, respectively.

Draw grid

Adds a grid to the plot.

Copy to clipboard

Copies the selected data to the clipboard. The general idea is that the graphics from an object at any time can be copied to a standard Windows application which supports GDI-operations. An example could be to export the graphics to MS Word. Activate the "Copy to Clipboard" function, then switch to MS Word and activate the "Paste"-function or press "Ctrl" V in and active Word document, and then a copy of the plot appears in MS Word.

Save to metafile

Saves the current plot to an enhanced meta file.

Save to bitmap

Saves the current snapshot plot to a bitmap file.

Font

Defines the font used in the plots.

Help

"Help" brings up the relevant on-line help pages.





10 Isopleth Plot

The Isopleth plotting tool can be used for plotting time variation of line series data in one plot (profile data, dfs1 files), see Figure 10.1.

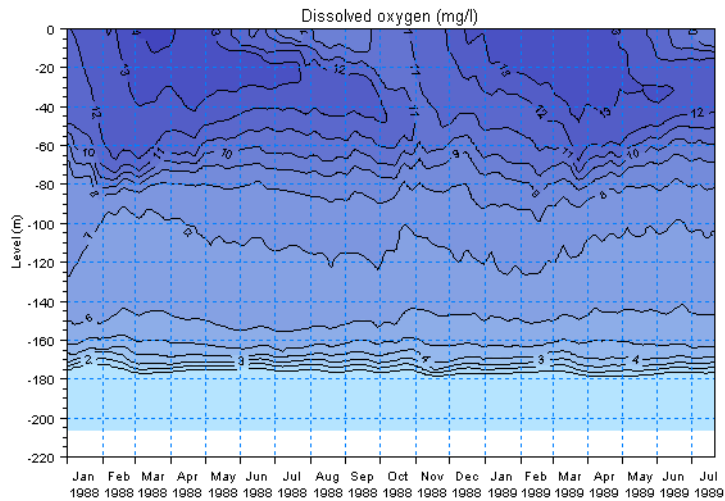


Figure 10.1 Example of Isopleth plot showing lower oxygen concentrations at the sea bed and higher concentrations near the surface. Highest levels occur during spring time.

The tool is also very useful for plotting the temporal evolution of frequency- or directional spectra computed in MIKE 21 SW (Spectral Wave Model FM). Examples are illustrated in Figure 10.2

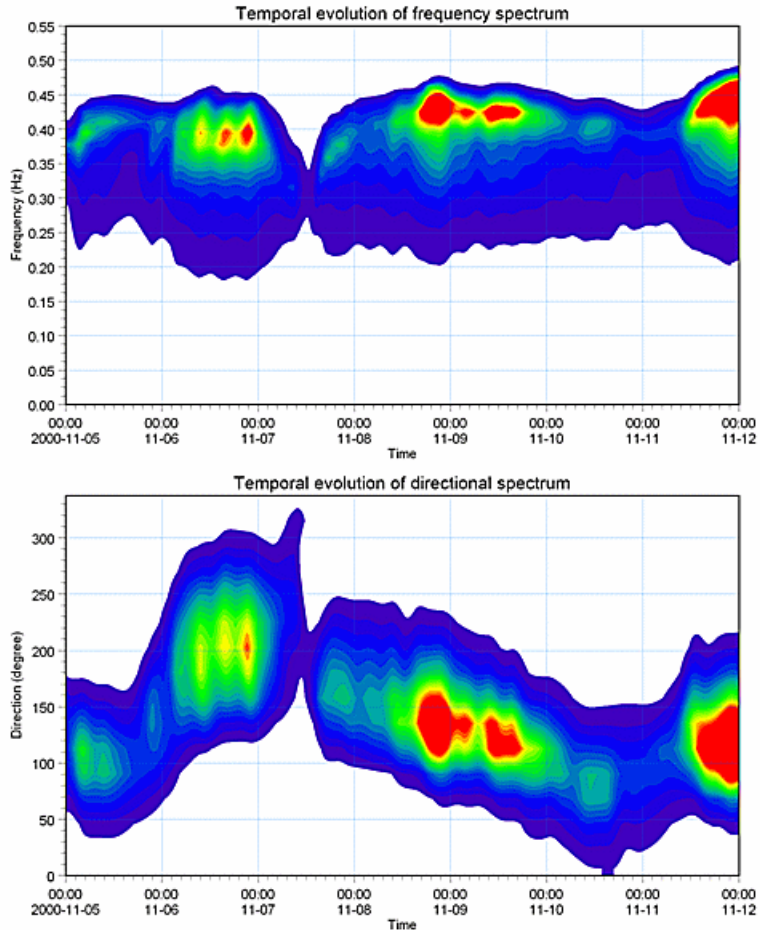


Figure 10.2 Example of Isopleth plot showing the temporal evolution of a frequency wave spectrum (upper panel) and directional wave spectrum (lower panel).

10.1 Input

Use this page to select the data file and item for your Isopleth plot.

Please Note

In case you want to plot a frequency wave spectrum (eg. output from MIKE 21 SW) please make sure the item has the correct EUM type, ie. "Directional integrated spectral density" and unit m^2s . For plotting a directional wave spectrum the correct EUM type is "Frequency integrated spectral density" and unit m^2rad .



10.2 Contours

You can choose to see the contours as "boxes", following the grid cells, or you can smooth them by choosing "shading".

The "**shading**" option shows different layers in different colours. If you choose "no shading", you must choose "Draw isolines" to be able to see anything at all. The isolines are lines on which the value is the same. You can see this value by choosing "With labels".

Color legend

When you choose "Shading" or "Box contours", you can also plot the color legend. The color legend can be configured using the Palette wizard. You can create, open, edit and save a palette file. You can use the dialog to calculate the limits from the data file by pressing "calculate", or you can specify the limits yourself.

If you select '<New levels/colors...>', you can define a custom palette which is either "Auto scaled" or "Fixed". On the fixed palettes, you can specify the data value ranges.

10.3 X-axis

On this page you define the limits of the x-axis. The x-axis represents time.

10.4 Y-axis

On this page you define the limits of the y-axis.

The definition of the y-axis depends on the type of input item.

Per default the y-axis defines depth levels. The y-axis will point downwards, with the origin of the dfs1 file as the highest value and decreasing levels. An example is shown in Figure 10.1.

In case the dfs1 file contains a wave spectrum, with an input item defined by the type 'Directional integrated spectral density' or the type 'Frequency integrated spectral density', the y-axis will define the direction or the frequency, respectively. The y-axis will point upwards with the origin of the dfs1 file as the lowest value. An example is shown in Figure 10.2.

10.5 Text Annotations

Here you have the option to give the plot a descriptive title and titles for the x-axis and y-axis.



10.6 Popup Menu

A click on the right mouse button brings up a menu that allows for various manipulations of the active object or the object over which the cursor is positioned.

Properties

A plot control specific property page appears, in which the properties of the plot can be manipulated.

File name legend

Places a legend in the plot container depicting the names of the data files from which the plotted data have been taken.

Zoom facilities

Zoom In allows you to zoom in on the plotted data and change the focus of the plot.

Zoom Out zooms out to default coverage.

Previous Zoom and **Next Zoom** zooms to the previous or following focus of the plot, respectively.

Draw grid

Adds a grid to the plot.

Copy to clipboard

Copies the selected data to the clipboard. The general idea is that the graphics from an object at any time can be copied to a standard Windows application which supports GDI-operations. An example could be to export the graphics to MS Word. Activate the "Copy to Clipboard" function, then switch to MS Word and activate the "Paste"-function or press "Ctrl" V in and active Word document, and then a copy of the plot appears in MS Word.

Save to metafile

Saves the current plot to an enhanced meta file.

Save to bitmap

Saves the current snapshot plot to a bitmap file.


Font

Defines the font used in the plots.



11 Label Plot

The Label Plot is a tool for adding labels to your plot composer plot.

 Powered by DHI	Date:	18/11/2015	Bathymetry for wave modelling Local area Alternative A	Drawing no. 2
	Init:	ajs		
Client:		Best Marine Company		MIKE2.0
Project:		New Land Project Development		

11.1 Label Properties

"Properties" is the specific property page for the current plot control, in which the properties of the plot can be manipulated.

On this property page you may:

- Include a header bitmap, e.g. a bitmap of your company logo.
- Write text into pre-defined description fields. Note: Press "ctrl enter" to achieve a line shift.
- Write text into two user-configurable fields.

11.2 Popup Menu

A click on the right mouse button brings up a menu that allows for various manipulations of the active object or the object over which the cursor is positioned.

Properties

A plot control specific property page appears, in which the properties of the plot can be manipulated.

Copy to clipboard

Copies the selected data to the clipboard. The general idea is that the graphics from an object at any time can be copied to a standard Windows application which supports GDI-operations. An example could be to export the graphics to MS Word. Activate the "Copy to Clipboard" function, then switch to MS Word and activate the "Paste"-function or press "Ctrl" V in and active Word document, and then a copy of the plot appears in MS Word.

Save to metafile

Saves the current plot to an enhanced metafile.

Save to bitmap

Saves the current snapshot plot to a bitmap file.



Help

"Help" brings up the relevant on-line help pages.



12 Metafile Plot

The Metafile Plot is a tool for adding metafiles to your plot composer plot.

12.1 Metafile Properties

"Properties" is the specific property page for the current plot control, in which the properties of the plot can be manipulated.

On this property page you may

- Select the metafile (enhanced meta file format, *.emf),
- Make the metafile plot transparent, and
- Choose a black hair-pin frame around the plot.

12.2 Popup Menu

A click on the right mouse button brings up a menu that allows for various manipulations of the active object or the object over which the cursor is positioned.

Properties

A plot control specific property page appears, in which the properties of the plot can be manipulated.

Help

"Help" brings up the relevant on-line help pages.





13 Polar Plot

MIKEZero Polar Plot tool is used to plot directional wave spectra computed by MIKE 21 SW. The wave energy density is plotted as function of the wave direction (radial direction) and wave period/frequency (circular direction).

With this tool you can create plots, raster and meta image files (BMP, PNG, JPG, TIF, EMF) and animations (AVI).

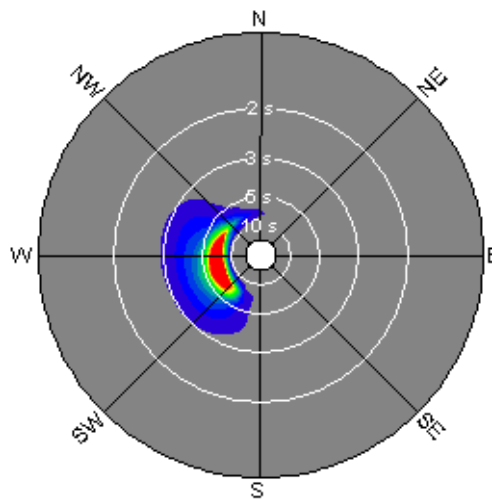


Figure 13.1 Example of directional frequency spectrum

13.1 Input Data

On the Input Data Selection dialog you **select the type 2 data** file (dfs2 or dt2) and the **item** containing the wave spectrum you wish to plot. The file is typically output from a MIKE 21 SW wave simulation.

13.2 Subset

On the Subset selection you select the first and last cell that should be plotted. Alternatively you can specify the minimum and maximum values in seconds.

By clicking "Select All", you can reset the values so the entire file will be plotted.



13.3 Contour Settings

You can choose to see the wave spectral density contours as "boxes", following the grid cells or you can smooth them by choosing "shading".

The "shading" option shows different layers in different colours. If you choose "no shading", you must choose "Draw isolines" to be able to see anything at all. The isolines are lines on which the value is the same. You can see this value by choosing "With labels".

Color legend

When you choose "Shading" or "Box contours", you can also plot the color legend. The color legend can be configured using the Palette wizard. You can create, open, edit and save a palette file. You can use the dialog to calculate the limits from the data file by pressing "calculate", or you can specify the limits yourself.

If you select '<New levels/colors...>', you can define a custom palette which is either "Auto scaled" or "Fixed". On the fixed palettes, you can specify the data value ranges.

13.4 Type of Grid Selection

The radial lines in the polar plot depict the directional resolution of the spectrum and the concentric circles depict the frequency resolution.

13.4.1 Compass grid

You can specify the directional grid type as compass directions (N, NE, W, etc.) or as compass degrees (0-360 degrees). When choosing the latter option you can define the compass degree interval. Wave energy plotted in the lower left quadrant comes from SW directions (or 225 degree) etc. The colour of the grid and labels can also be change from the default (black).

13.4.2 Wave period grid

Concentric circles represent the period/frequency grid, where the innermost circle corresponds to the maximum wave period/frequency (default) and the outermost circle corresponds to the minimum wave period/frequency (default). You can specify whether you would like to have a short or long annotation on the plot, the angle and colour.

13.4.3 Directional and frequency resolution

By including "Element Mesh" you can view the resolution in the file.

**NOTE:**

The grid spacing in the x-direction defines the type of frequency resolution: If $DX > 1$ the frequency change by logarithmic scale, otherwise the frequency distribution is equidistant.

The maximum frequency in the file must be less than or equal to 100.

13.4.4 Change of wave period grid

You can change period/frequency grid such that the innermost circle corresponds to the minimum wave period/frequency and the outermost circle corresponds to the maximum wave period/frequency.

13.5 Text Annotation

In this dialog you can specify a title of your polar plot.

13.6 Output Settings

In this dialog you can control the output settings: Double Buffering and name of video file.

The file is recorded when using the video camera button of the Polar Plot toolbar (the latter can be activated in the Popup Menu by right-clicking on the Polar Plot)

Double Buffering is a special drawing technique, that allows flicker-free animations. Double Buffering is only used if the plot control is active.

The video file name is the name of the AVI-file used for generation of video output from the plot.

13.7 Popup Menu

You will get the pop-up menu if you right click with the mouse on the picture. The items in the pop-up menu are (along with some standard zoom and save function):

Properties:

This is where you select files to plot and everything else about the picture and simulation setup.

Toolbar

Moving forwards and backwards in time to create a simulation and maybe a video file. The video icon creates the file.





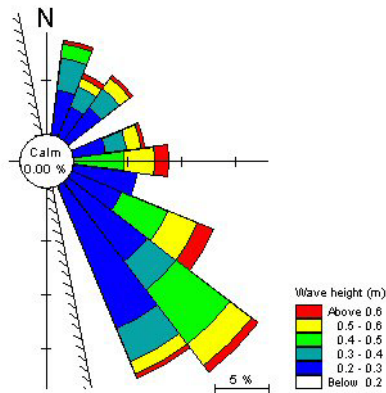
File name legend

As default, the file name and path of your input data file is shown on the plot, but you can switch this off by selecting this menu item.



14 Current and Wind Rose Plot

This tool is designed for rose plotting, e.g. wind, current, wave or sediment roses. The input is given as a time series file (type 0 file) with at least two items that can reflect the magnitude (e.g. speed or wave height) and direction.



14.1 Item and File Selection

First of all, you specify the time series file from which the rose plot is made. This is done by clicking on the selection box to the right. A dialogue will appear where the filename can be specified.

When the file has been chosen, the items to plot must be selected. You can choose between two types of description:

- u - v (these values are recalculated into magnitude and direction)
- Magnitude - Direction

Furthermore you have the option to define the weight of each time series event explicitly by selecting a third item. Per default each time series event is weighted equally.

You select and edit the individual items for plotting by clicking on the pen icon.

14.2 SubSeries Selection.

Here you specify the sub series to plot. As default, the entire time series is chosen. You can also specify whether you want to use every step in the file or only, say, every third step.



14.3 Data Classes definitions.

Here, you specify the discretisation of the rose plot.

The *Circle Sector Size* defines the directional intervals of the plot and the *Centre Angle* defines the centre angle of the first directional interval. As an example, let's say that you want to specify a rose plot with directional intervals 0° - 45° , 45° - 90° , ... etc. Then the *Circle Sector Size* must be specified as 45° and the *Centre Angle* as 22.5° .

The discretisation of the magnitude (e.g. speed or wave height) is defined by specifying the number of class delimiters and the corresponding delimiter values. To the right of the delimiter specification, the minimum and maximum value of the magnitude item in the selected time series file is given so as to make the selection easier.

The auto interpolation will automatically set the magnitude delimiters to the best fit.

In the bottom left corner, the colour palette can be chosen.

14.4 Plotting definitions.

On this dialogue, specifications to do with the plotting itself are given.

You can choose to plot the rose with a coastline. If this option is enabled, you also have to specify the angle from geographical north to the coastline normal measured clock-wise.

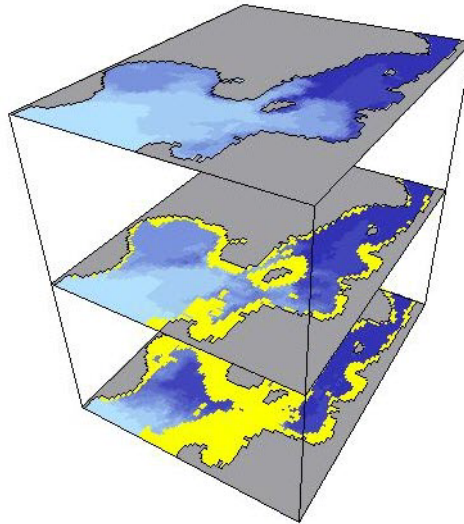
It is also possible to enable or disable the plotting of a colour legend for the magnitude and a scale legend for the tick marks on the axis.

The length of the axis can be fitted by the system or the user can define them. If the user definitions are smaller than required, the definitions are automatically updated.



15 Slice Plot

Plotting tool for plotting slices of 2D or 3D data from type 2 data files (*.dfs2 or *.dt2) or from type 3 data files (*.dfs3 or *.dt3), respectively.



15.1 Input Data Selection

You can choose to plot a .dfs3 data file or a .dfs2 data file with more than one timestep.

Plotting a .dfs2 data file with only one timestep will not make sense, since the purpose of this plotting tool is to visualize the development of a 2D area in height or time.

15.2 Items

On this page you **choose which items you want to plot**. You select the items in the list "items" and press one of the arrow buttons,



to choose if they should be plotted as contour or vector items.

If you choose to calculate the **contours** from 1 item the "standard way", you will only have to specify 1 item in the items selection's "Contour Item(s)". On the other hand, if you choose to base the contours on the **vector length**, you



will have to specify 2 contour items. If the file only contains 1 item, the latter option will not be available.

On this page, you are able to choose from which items the contours should be calculated, but you will specify the appearance of the contours on the Contours page.

15.3 SubSet Selection

This is where you choose a **subarea** or a **sub-timeseries** instead of plotting the whole file.

The subarea is a subset of the 2D area. If you only want to plot a selection of slices, you can choose this on the Slice Selection page.

15.4 Slice Selection

On this page you specify the slices to plot.

For type 3 data, the z-layers are plotted as horizontal slices and the different time steps are plotted on separate plots.

For type 2 data, the specified subseries in time is plotted as horizontal slices.

The layout is specified as the number of slices per plot. In order to see something on the plot, you have to specify at least two slices per plot.

15.5 Contours

You can choose to see the contours as "Box contour", following the grid cells, with or without a box divider.

If you choose "no contour", you must choose "Draw isolines" to be able to see anything at all. The isolines are lines on which the value is the same and correspond to a smooth version of the box contour dividers.

Color legend

When you choose "Shading" or "Box contours", you can also plot the color legend. The color legend can be configured using the Palette wizard. You can create, open, edit and save a palette file. You can use the dialog to calculate the limits from the data file by pressing "calculate", or you can specify the limits yourself.

If you select '<New levels/colors...>', you can define a custom palette which is either "Auto scaled" or "Fixed". On the fixed palettes, you can specify the data value ranges. (For a "land/water fixed palette" you only specify the lower limit, while the upper limit is the land value).



15.6 Vectors

On this page you specify how the vectors should be scaled and the interval (in grid points) between the individual vectors.

Automatic scaling is a quick way to scale the vectors. You simply give the length (in grid points) of the longest vector on the plot. The individual plots in a series are scaled differently. When using automatic scaling, vectors are never distorted. Therefore, when you change scaling in one direction, the scaling in the other direction is automatically changed accordingly.

User specified scaling gives you full control of the vector scaling. You specify the vector length (in grid points) corresponding to typically one SI unit. The individual plots in a series are therefore given the same scaling. User specified scaling can also be used for amplifying the vector length in one direction (distortion). If e.g. 10 to the power of 3 SI-units is a more practical unit, then set 'base' equal 3, to get a more reasonable X- and Y-scale value.

15.7 Land Options

The options to define land on a plot depends on the type of input file.

15.7.1 Land options

For a general dfs2 file you specify the **land appearance** and the edge drawing of land areas. If shaded land is chosen, then the specified colour the is used.

15.7.2 Land settings for dfs2 result files

On this dialog you define how to represent land on a plot of MIKE 21 module results.

First you specify where to get the land information from. The land data is **typically taken from the first skip record** (first static item) in result data file.

You may also choose to specify a digitized land border to be used for land plotting, either as exported from a Digitizer file (*.bbd) or read from a XYZ ASCII file (*.xyz). Choosing the XYZ-file, you must give the projection in which the land border file data appears. This must always be one of the kind of a Universal Transverse Mercator Projection, for instance 'UTM-33' for zone 33, or 'BTM' for a local UTM zone defined in the utm.dat file in your MIKEZero\bin directory.

Then you specify how to **distinguish land points from water points**. The value for true land, i.e. points that for sure never are flooded, is usually a minimum value. This means that all points with a value greater than or equal to the true land value (above true land) are land points.



You specify the **land appearance** of true and dried land areas. If shaded land is chosen, then the specified colour the is used.

Finally, the **edge drawing** of land areas should be defined.

15.7.3 Land settings for dfs3 result files

On this dialog you define how to represent land on a plot of MIKE 3 module results.

First you specify if you want to show land or not. The land values are taken from the prefix item in the input file.

The you specify the land **appearance** and the **edge** drawing of land areas. If shaded land is chosen, then the specified colour the is used.

15.8 Coordinate system setup

On this page you give the eye point definitions and the axis properties.

Eye Point Definition

- Twist: The twist round the Z-axis. If there is no twist, the Y-axis is pointing away from the viewer.
- Tilt: Tilt of Z-axis towards viewer.
- Zoom: Changes distance between eye point and focus point.

XY-Axis Properties

You can select either no axes, grid axes (default) or metre (and derived units) axes. Further more you can specify axes annotation.

15.9 Output Settings

With this page you can control the output settings: Double Buffering and name of video file.

The file is **recorded** by the Toolbar button with the video camera icon in the Popup Menu.

Double Buffering is a special drawing technique, that allows flicker-free animations. Double Buffering is only used if the plot control is active.

The video file name is the name of the AVI-file used for generation of video output from the plot



15.10 Popup Menu

You will get the pop-up menu if you right click with the mouse on the picture. The items in the pop-up menu are (along with some standard zoom and save function):

Properties:

This is where you select files to plot and everything else about the picture and simulation setup.

Toolbar:

The toolbar can be used for moving forwards and backwards in time to create a simulation and maybe a video file. The video icon creates the file.

The last four icons enables you to scroll up and down in the layers.



File name legend:

As default, the file name and path of your input data file is shown on the plot, but you can switch this off by selecting this menu item.

Move North Arrow and Convergence Indicator:

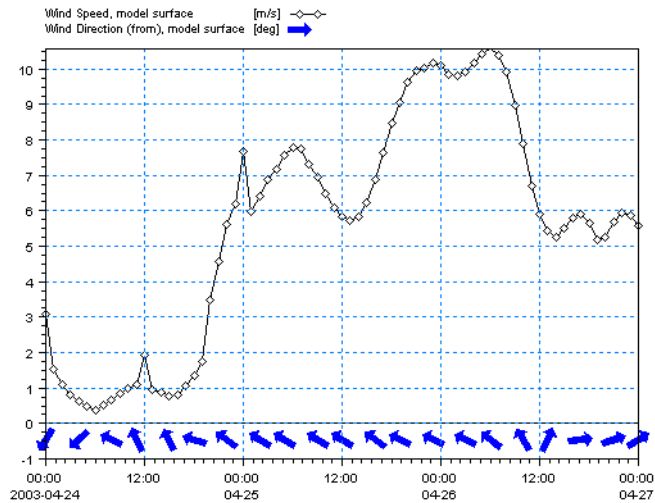
Selecting one of these and afterwards clicking on the desired new spot will move the North Arrow or the Convergence Indicator to the new spot. But of course only if the icon is already on the plot.





16 Time Series Direction Plot

Plotting tool for plotting time series data (*.dfs0 or *.dt0 files). Directional items will automatically be plotted as rotated arrows.



16.1 Items

Use this page to select the data file(s) containing the item(s) for your time series plot. If items with directional data is selected (such as wave direction or flow direction) the direction will be displayed by directional arrows rather than a curve as in the classic Time Series Plot.

NOTE:

Per default the values are shown in user units, however it is possible to select the unit by which the values are to be displayed.

16.2 X-Axis Definitions

On this page you define the limits of the x-axis.

16.3 Y-Axis Definitions

On this page you define the layout of the y-axis (axes).

First, you specify for each item if it should be plotted relative to the left placed or the right placed y-axis.



Then, you specify the limits and properties of the y-axis.
Note that the position of the directional arrows will be defined later by a distance from the x-axis line.

In case you have selected items for both the left and the right y-axis, you must enter specifications for both the axes.

16.4 Curve Definitions

On this page you define the appearance of the items displayed as curves and arrows.

For each item you define

- the marker symbol (for curves only),
- the marker size,
- the marker fill style,
- the marker colour,
- the minimum marker spacing,
- the marker phase shift and
- whether or not the marker should be displayed.

For each curve you furthermore define

- the line width,
- the line style as solid, dashed, dotted, dashed-dotted, or filled,
- the line colour, and
- whether or not the line should be displayed

For each arrow group (item) you furthermore define

- the elevation, i.e. the distance from the x-axis to the arrows in mm.

16.5 Text Annotations

Here you have the option to give the plot a descriptive title.

16.6 Popup Menu

A click on the right mouse button brings up a menu that allows for various manipulations of the active object or the object over which the cursor is positioned.



Properties

A plot control specific property page appears, in which the properties of the plot can be manipulated.

Time series legend

Places a legend in the time series plot container depicting the item(s) plotted.

File name legend

Places a legend in the plot container depicting the names of the data files from which the plotted data have been taken.

Zoom facilities

Zoom In allows you to zoom in on the plotted data and change the focus of the plot.

Zoom Out zooms out to default coverage.

Previous Zoom and **Next Zoom** zooms to the previous or following focus of the plot, respectively.

Draw grid

Adds a grid to the plot.

Copy to clipboard

Copies the selected data to the clipboard. The general idea is that the graphics from an object at any time can be copied to a standard Windows application which supports GDI-operations. An example could be to export the graphics to MS Word. Activate the "Copy to Clipboard" function, then switch to MS Word and activate the "Paste"-function or press "Ctrl" V in and active Word document, and then a copy of the plot appears in MS Word.

Save to metafile

Saves the current plot to an enhanced meta file.

Save to bitmap

Saves the current snapshot plot to a bitmap file.

Font

Defines the font used in the plots.

Help

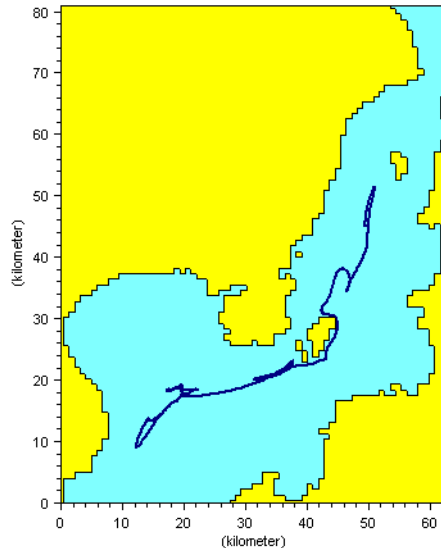
"Help" brings up the relevant on-line help pages.





17 Track Flow Visualisation

Track Flow Visualisation is a tool for making float tracking plots from **MIKE 21 and MIKE 3 hydrodynamic results**. The plotting tool will follow the paths of a number of particles injected in the flow field.



For all specifications, with the exception of I/O file names, default values are supplied. You can change these specifications by modifying the corresponding dialog entries.

17.1 Input Specification

On this page, you specify the **MIKE 21 and MIKE 3 model results data** you wish to process. The tool supports following type of data:

- MIKE 21 result file (DFS2 data file)
 - Water level (H) and fluxes (P and Q)
 - Surface elevation (S) and velocities (U and V)
- MIKE 3 result file (DFS2 data file)
 - Surface elevation (S) and velocities at specific layer (U and V)

For other combinations the program will give an error message.

You can use the **“Reset” button** in cases where you change your input file. If you already made a setup with a previous file, but now you want all the



default values, e.g. subarea etc., to come from the new file, you can press the reset button after selecting the new file.

You can choose the time step for your particle simulation by setting the **Interpolate to XX frames per time step**, where time step refers to time step in your input file. If you have saved your flow model result file with, say, ½ hour time step, setting the “Interpolate to XX frames per time step” value to 5, you will run your particle simulation with time steps of 5 minutes.

The **number of milliseconds per frame** is how long you need to look at each simulation step on the screen. e.g. how often the picture is updated during the simulation.

If you do not choose **double buffering when redrawing** due to memory problems, and you are saving your output to an *.avi file, please make sure to keep the full area visible on the screen and not to put any other windows on top while recording.

17.2 Particle Track Definitions

On this page you define the **positions of the particles** you want to follow. The procedure is the following:

- Click on the “New (Insert)” button at the list of tracks.



Choose on the left if you want grid coordinates or lat/long coordinates and specify the coordinates in the X and Y fields.

- Press the “arrow” to put the coordinate in the track list.



You can also specify the **thickness and colour** of the particle trajectories here.

17.3 Land/Water Specifications.

You must define the appearance of land areas, water areas and the land-water border.

17.4 SubSet Specifications

Here you can define the subarea of your file and the sub-timeseries you want to plot. There is also a zoom function in the Popup Menu, so you can always choose your subarea directly on the picture later.



17.5 Coordinate System

In case you have geographical information in your input data file, you can **superimpose on the plot geographical coordinates** either as map projection coordinates or as latitudes and longitudes. Furthermore, you have the opportunity to **add plotting symbols indicating true north and the convergence**, defining the angle between the longitude and the northing line for the map projection. The indications of true north and convergence can be moved later, directly on the picture, using the Popup Menu.

When you are superimposing a coordinate system, the one specified in the "Map projection type" box will be used. The default **Map projection** is the one specified in your background input data file.

If you select **Northing coordinates always positive**, The northing coordinates will be as 1) and 3) in Figure 17.1. If you don't select it, the southern hemisphere northing coordinates will be as 2).

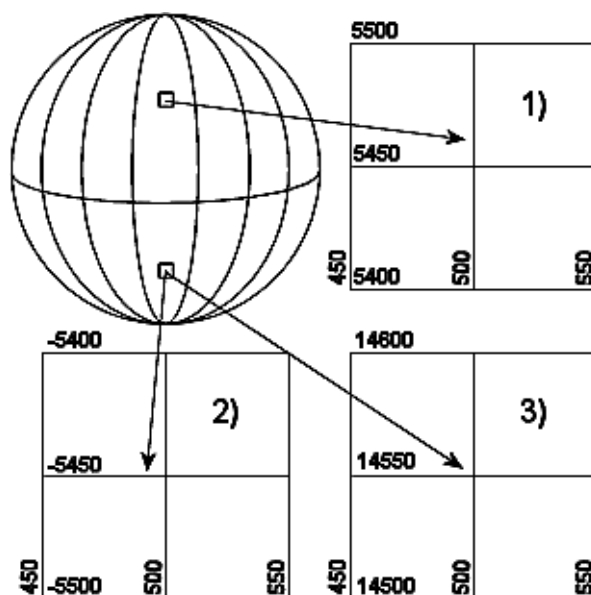


Figure 17.1 Illustration of UTM-zone

Finally, you can specify additional **axis annotation** and a **geodetic scale**. If you choose to have grid units, the scaling is not available since it doesn't make sense in this case.

Choosing automatic scaling will show the scale on the plot according how you size the plot manually. Choosing a defined scale, ex. 1:100000, will automatically size the plot according to that scale.



17.6 Popup Menu

You will get the pop-up menu if you right click with the mouse on the picture. The items in the pop-up menu are (along with some standard zoom and save function):

Properties:

This is where you select files to plot and everything else about the picture and simulation setup.

Toolbar:



With this button, you can click on the picture to visually insert the particles you want to trace.



This button starts the simulation.

File name legend:

As default, the file name and path of your input data file is shown on the plot, but you can switch this off by selecting this menu item.

Move North Arrow and Convergence Indicator:

Selecting one of these and afterwards clicking on the desired new spot will move the North Arrow or the Convergence Indicator to the new spot. But of course only if the icon is already on the plot.

Insert track starting points:

Selecting this menu item, you can click on the picture to visually insert the particles you want to trace. This works the same way as the toolbar button.



Calculate tracks:

This menu item starts the simulation in the same way as the toolbar button.





18 XY Plot

You can use this plotting facility to see the correlation between two items in a time series file.

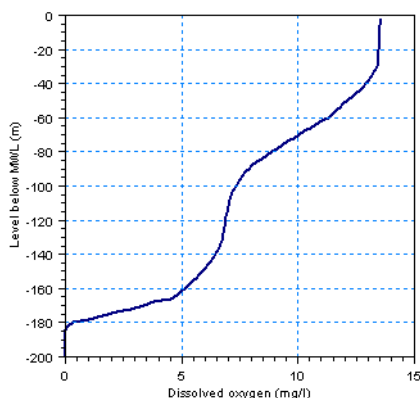


Figure 18.1 Example of XY plot

18.1 Curve Definitions

When you choose a file, you also choose which items should be the x and y axis on your plot. You will **choose the items** on the "Select items" page in the "Open" dialog.

After selecting the items, you can select line appearance or marker style.

The "Item pairs" box gives information on the items, like min/max values and units. The only thing you can choose here is whether the y axis should be on the left or right in the picture ("Axis pos" = 1 or 2 respectively).

18.2 Axis Definitions

Set the axis extremes here, either as absolute values or relative to the data.

18.3 Text Annotations

On this page you choose which text strings that are to be put on the plot axis and header.



18.4 Popup Menu

A click on the right mouse button (For some of the below mentioned menu items, you have to use the left button) brings up a menu that allows for various manipulations of the active object or the object over which the cursor is positioned.

Properties

A plot control specific property page appears, in which the properties of the plot can be manipulated. This is where you choose which file you want to plot, line appearance, labels, etc.

Toolbar

Not Available

Legend

If you choose this, you will see a plot legend in the top of your picture.

File name legend

Places a legend in the plot container depicting the names of the data files from which the plotted data have been taken.

Zoom facilities

Zoom In allows you to zoom in on the plotted data and change the focus of the plot.

Zoom Out zooms out to default coverage.

Previous Zoom and **Next Zoom** zooms to the previous or following focus of the plot, respectively.

Copy to clipboard

Copies the selected data to the clipboard. The general idea is that the graphics from an object at any time can be copied to a standard Windows application which supports GDI-operations. An example could be to export the graphics to MS Word. Activate the "Copy to Clipboard" function, then switch to MS Word and activate the "Paste"-function or press "Ctrl" V in and active Word document, and then a copy of the plot appears in MS Word.

Save to metafile

Saves the current plot to an enhanced meta file.

Save to bitmap

Saves the current snapshot plot to a bitmap file.

Help

"Help" brings up the relevant on-line help pages.



INDEX



A	
ADCP input file	55
ADCP plot	61
Animate tab	17
B	
Background	56
Batch	22
D	
Delete value	34
Derived item	34
Display style	30, 43
E	
Edit tab	14
Export graphics	13
H	
Histogram	11
I	
Image origin	31, 44
Image size	31, 44
L	
Line properties	20
M	
Marker properties	20
MzPlotCompApp	22
P	
plot object	11, 14
S	
Shape file	42
shortcut keys	21
Stretch mode	30, 43
V	
Vector component	24
Vertical Profile	47
video	16