

# Result Viewer

Presentation of results from model applications within  
**MIKE Zero**

**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 A/S (hereinafter referred to as "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.





<b>1</b>	<b>Result Viewer</b>	<b>7</b>
1.1	File Menu	7
1.2	Edit Menu	7
1.3	View Menu	8
1.3.1	Coordinate Overlays	8
1.3.2	Mouse Pointer Coordinates	8
1.3.3	Zoom facilities	8
1.3.4	Profile	9
1.3.5	Export Graphics	9
1.3.6	Aspect ratio	10
1.3.7	Toolbar	10
1.3.8	Status Bar	10
1.4	Projects Menu	10
1.4.1	Add Files to Project	10
1.4.2	Overlay Manager	12
1.4.3	Active View Settings	12
1.4.4	Work Area	12
1.4.5	Video	13
1.5	Display Properties	14
1.5.1	Image file	14
1.5.2	Grid (Curvilinear, Rectangular or Unstructured)	15
1.5.3	Shapes	15
1.5.4	Vectors	16
1.5.5	Branches	17
1.5.6	Pipes	18
1.5.7	Profiles	18
1.5.8	3D Items properties	22
1.5.9	Cross-section	24
1.5.10	UZ plot	25
1.5.11	Baseline	25
1.5.12	Spatial line	27
1.5.13	Colors	29
1.5.14	Particles	29
1.5.15	Particle tracks	32
1.5.16	XYZ data	32
1.6	Window Menu	35
1.7	Result Viewer Toolbar	35
1.7.1	Time series extractor	37
1.7.2	Profile Extractor	38
1.7.3	Cross-section extractor	40
1.7.4	UZ Specific Plots	41
	<b>Index</b>	<b>45</b>





# 1 Result Viewer

The information provided in the present manual is related, specifically, to the use of the MIKE Zero Result Viewer for the applications involving presentation of results from DHI numerical modelling software within the MIKE Zero suite.

It is organized logically following the appearance on the menu pages.

## 1.1 File Menu

This is where you open, save and close files, and access printer related utilities. The menu consists of the below Windows standard commands, plus a special Print Setup command.

### New

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

### Open

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

### Close

Closes the active document.

### Save

Saves the Animator mza-file.

### Save As

Saves the Animator mza-file as a new file.

### Print

Produces a hardcopy of the current plot.

### Print Preview

Shows a preview of the hardcopy, as it will appear on the hardcopy device.

### Print Setup

Here you may select the printer and modify its setup. This is also where you choose the layout of the paper and the paper size.

## 1.2 Edit Menu

Clipboard and object editing.

### Undo

Undo latest command.



### Cut/Copy/Paste

Cuts, copies and pastes your selected data through the Windows clipboard.

### Delete

Deletes your selected data.

## 1.3 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.

### 1.3.1 Coordinate Overlays

This menu makes it possible to overlay the workspace area with a geographical net showing the longitudes and latitudes, a map projection grid and an arrow pointing towards true North.

#### Map Projection

You can add or remove lines of constant Eastings and Northings in the display.

You can select which map projection net to overlay. Per default the displayed map projection net is defined as in the work area.

#### North Arrow

You can add or remove a north indicator in the display.

You can choose between different types of indicators as well as specify a given position for the north indicator to be placed in the display.

### 1.3.2 Mouse Pointer Coordinates

The mouse coordinates displayed can be one of the following types:

- (x,y) coordinates (grid)
- (lon,lat) coordinates
- Map projection coordinates

### 1.3.3 Zoom facilities

#### Zoom In

Allows you to zoom in on the plotted data and change the focus of the plot.





### Zoom to coordinates

Allows you to zoom in on an area specified by (x,y) coordinates.

### Zoom Out

Zooms out to default coverage.

## 1.3.4 Profile

This enables you to load, save and create a new profile plot.

If you have a profile open you can save the current profile location. This allows you to create standard profiles for comparing scenarios.

## 1.3.5 Export Graphics

The entire plot area with all plot objects within can be exported.

### 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.

### Save Vectors to Shape file

Saves the total vector field for the current time step as polylines in a shape file. The polylines will have the scale as shown in the Result Viewer.

Note: Polylines can be visualised by an arrow symbol in e.g. ArcMap.

### Batch command

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

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

- MzResultApp input.rev [-options] -printtofile output.png
- MzResultApp input.rev [-options] -print\_to\_file output.png

Alternatively you may create other file types e.g. output.bmp, output.jpg or out-put.tif.



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.

Writing MzResultApp -? in a DOS prompt will show the full syntax of the command.

### 1.3.6 Aspect ratio

This value defines how the display is scaled relatively in the x- and y-axis direction.

### 1.3.7 Toolbar

Allows you to insert and remove toolbars. The Result Viewer Toolbar provides a number of tools especially relevant for the Result Viewer.

### 1.3.8 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.

## 1.4 Projects Menu

### 1.4.1 Add Files to Project

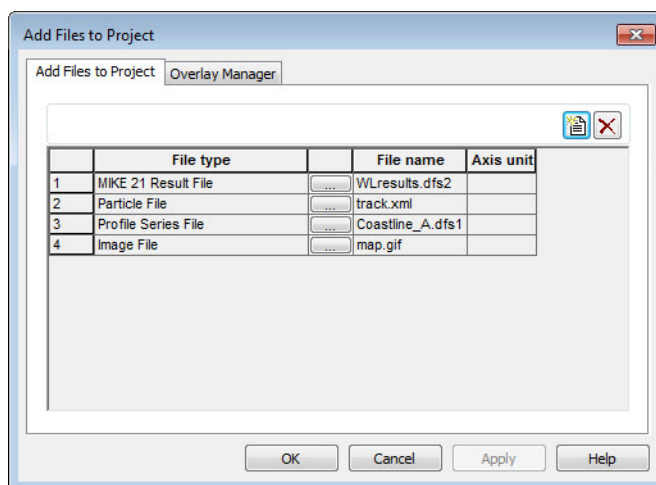


Figure 1.1 Add files to project dialog



This is where you add a new file to the project. The file can be a number of types:

- Enhanced Meta File
- Graduated Time Series File
- Image File
- Mesh File
- MIKE 11 Reservoir Result Files
- MIKE 11 Result File
- MIKE 21 C Result File
- MIKE 21 Result File
- MIKE 3 Result File
- MIKE FM Result file
- MIKE SHE Result File
- MIKE URBAN File
- MOUSE Result File
- NS 3 Result File
- Particle File
- Profile Series File
- Shape File
- XYZ File

Some of the documentation for the individual modules (e.g. MIKE SHE) contains further description and examples related to the use of the Result Viewer.

## MIKE 21 Result File

If your file has been recognized as a standard output file from MIKE 21 HD, i.e. containing the items H Water depth, P-flux and Q-flux and with Data type = 1, you get the additional option of selecting one of the derived items: Surface Elevation, Current Velocity in the X direction, Current velocity in the Y direction or Current Speed.



**Note:** The calculations of derived items are based on the assumption of the absolute delete value in the file is 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.



## 1.4.2 Overlay Manager

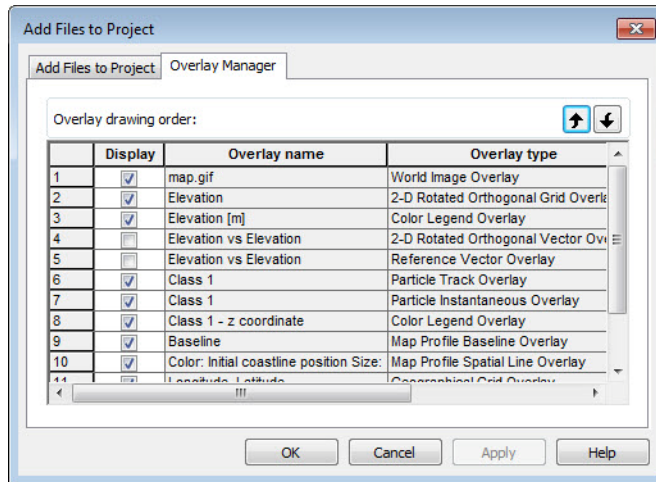


Figure 1.2 Overlay manager dialog

Here you specify what is shown on the plot.

You can also specify which item goes on top, by using the up/down arrow buttons.

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.

## 1.4.3 Active View Settings

The display of each of the layers must be defined. Dependent of the type of result file, different properties can be set.

For further description, please see Display Properties (p. 14).

## 1.4.4 Work Area

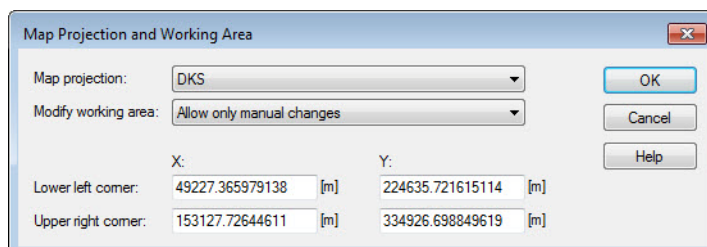


Figure 1.3 Work area definition



The work area for the presentation is set by specifying the extent of the working area and the projection to be used.

The display of the results will always be directed towards North.

## 1.4.5 Video

### 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.

### Record

Record a video in avi format. Besides the format you can select how long the video should be (no. of frames), the speed of the video (frame rate), the image size and the output name and directory.

In case you want to stop the recording before the end of the animation press 'Escape'.

### Snapshot

Make a still picture in either **\*:BMP**, **\*:JPEG**, **\*.PNG** or **\*.TIF** format. Besides the format you can select the image size and the output file name.



## 1.5 Display Properties

### 1.5.1 Image file

Styles

**Image Styles**

Draw image

Display style: Copy colors with transparency

Stretch mode: Halftone

Transparent color: [Color Picker]

Transparency: 50 [%]

**Image Area Coordinates**

	X:		Y:	
Lower left corner:	305430 [m]		6121100 [m]	
Upper right corner:	403900 [m]		6240150 [m]	

**Image Map Projection**

Type: WGS\_1984\_UTM\_Zone\_33N

Figure 1.4 Image file properties

This is where you define the display style of an image. Using transparent colours gives the possibility to view several layers.

The size and position of the image in the work area is given in the projection, such as they are defined in the Work Area.



### 1.5.2 Grid (Curvilinear, Rectangular or Unstructured)

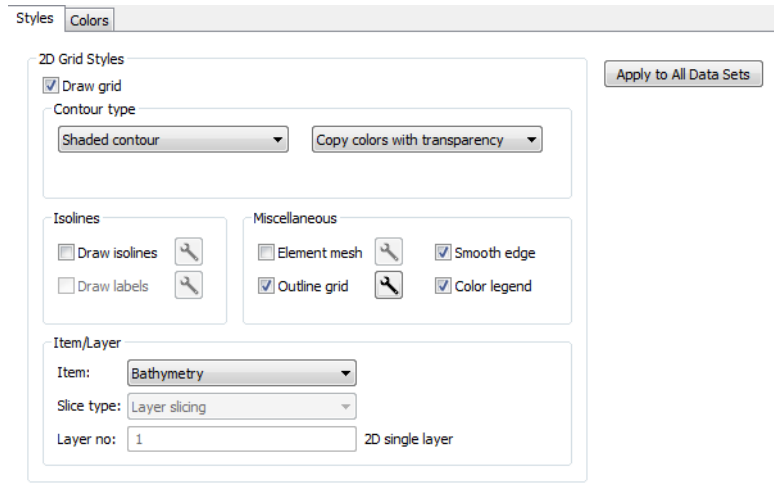


Figure 1.5 Grid properties

This is where you define the display style of a grid.

You may define contours, isolines and which item of the data file to display. The Layer no. refers to the layers in 3-dimensional data files.

### 1.5.3 Shapes

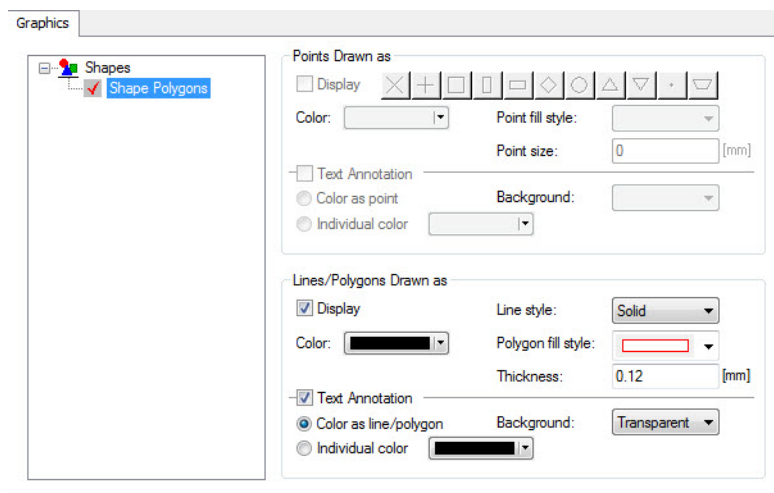


Figure 1.6 Shape file properties

This is where you define the display style of a shape file.



You define how the points, lines/polygons and text annotations are to be displayed.

The shape file must be georeferenced within the Work Area to be displayed.

## 1.5.4 Vectors

The screenshot shows the 'Vector Options' dialog box. It has a title bar with 'Vectors' and 'Vector Options'. Below the title bar, there is a '2D Vectors' section. This section contains several controls: a checked checkbox for 'Draw vectors', an unchecked checkbox for 'Draw reference vector', a dropdown menu for 'X Item' set to 'Total water depth', a dropdown menu for 'Y Item' set to 'Total water depth', a dropdown menu for 'Vector scale type' set to 'Automatic', a text input field for 'Vector scale' containing '21.7942297432739', a dropdown menu for 'Style' set to 'Vectors drawn at element centers', and a 'Mesh size' section with two input fields, both containing '50', followed by 'X' and 'vectors'. To the right of the '2D Vectors' section is a button labeled 'Apply to All Data Sets'.

Figure 1.7 Vector properties

This is where you define the display style of vectors.

You define the items for the X- and Y-coordinate, respectively. You have the option to specify the vector scale and frequency of plotting vectors.

You specify the style of the vectors in the Vector Options dialog.





Figure 1.8 Vector Options

### 1.5.5 Branches

Figure 1.9 Branch properties

This is where you define the display of branches from a MIKE 11 model setup.

You can define the branch width and the branch colour.



## 1.5.6 Pipes

Pipe Settings Graphics

Draw pipes

Pipe Width

Determined from pipe

Fixed number 100

Selected dynamic item Link Discharge Level

Maximum pipe width 100

Pipe Color

Fixed color [Red]

Selected dynamic item Link Water Level

Figure 1.10 Pipes properties

This is where you define the display of pipes from a MIKE URBAN CS or MOUSE setup.

You can define the pipes width and colour.

## 1.5.7 Profiles

This is where you define the display of e.g. a cross-section plot of a saturated zone from a MIKE SHE simulation.

See also Profile Extractor (p. 38).



## Symbol and Fonts

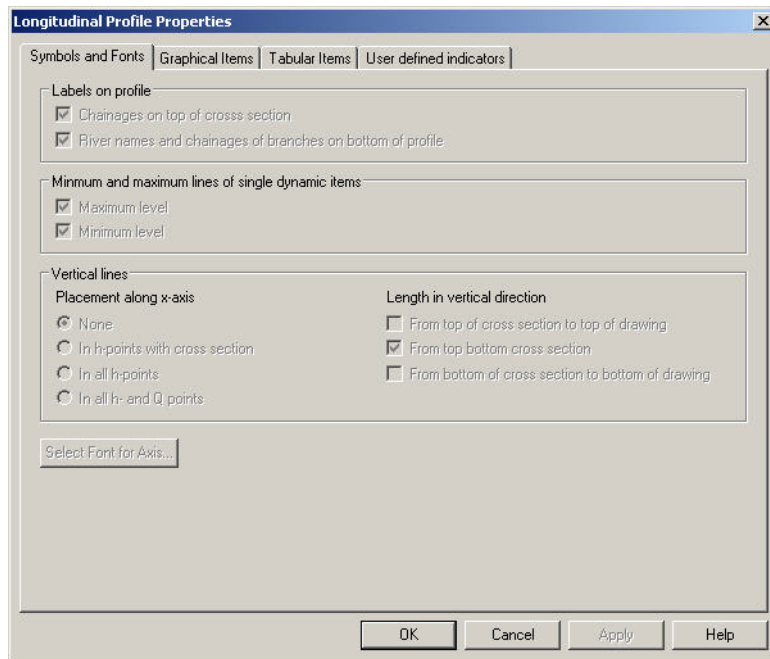


Figure 1.11 Symbol and fonts for longitudinal profile plot



## Graphical Items

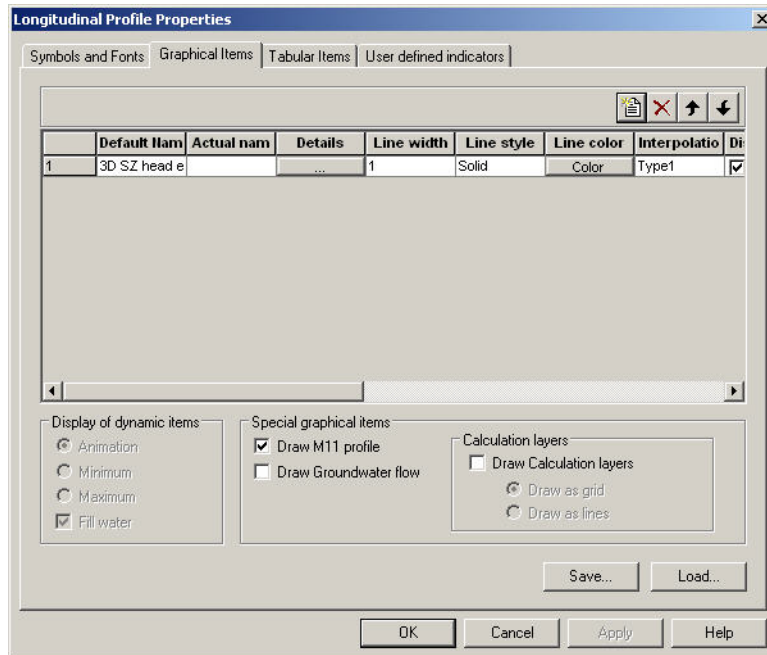


Figure 1.12 Graphical item properties in longitudinal profile plot

In this dialog you specify the layout for the individual graphical items. If you click on the Details... button for the item, you will get the 3D Items properties dialog where you can change the colour scale and plotting characteristics for the cross-section, see page 22.



## Tabular Items

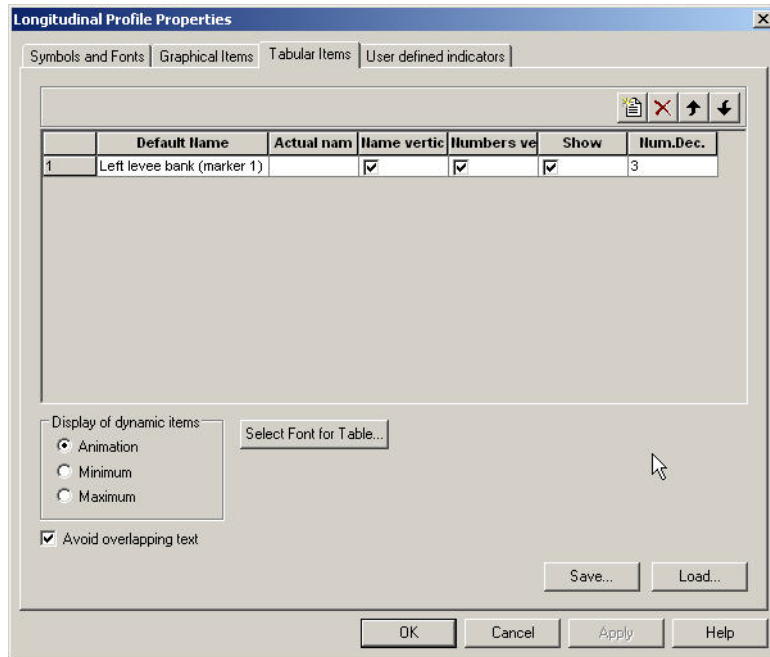


Figure 1.13 Tabular item properties in longitudinal profile plot

In this dialog you specify the layout for the individual tabular items.

When you add a tabular item you must specify whether it is a static item or a dynamic item.



Figure 1.14 Selection of tabular item



## User defined indicators

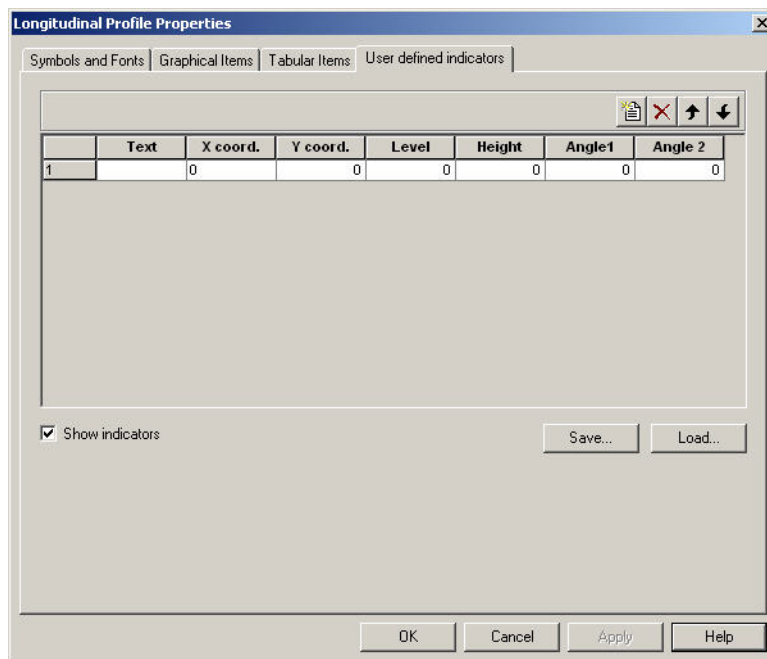


Figure 1.15 User defined indicator properties

In this dialog you specify the layout for the individual user defined indicators.

### 1.5.8 3D Items properties

In this dialog you can change the colour scale and plotting characteristics for a cross-section.



## Style

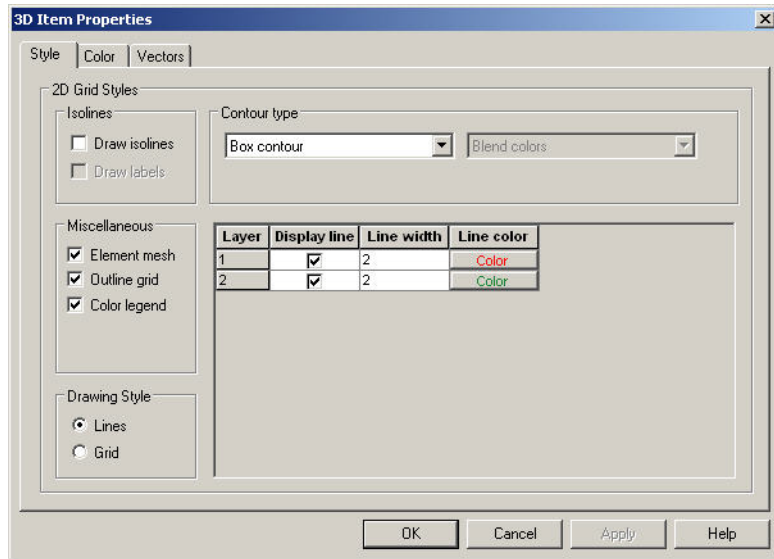


Figure 1.16 Style of contours in cross-section plots for 3D items.

## Color

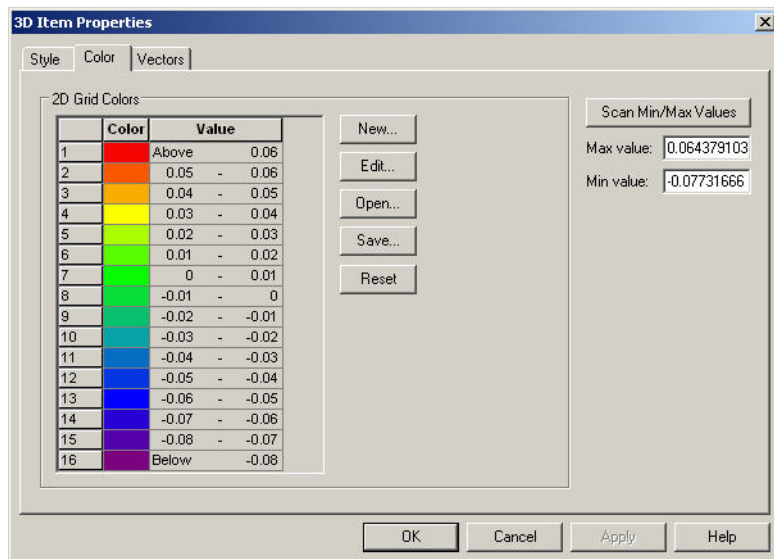


Figure 1.17 Colours for contours in cross-section plot for 3D items



## Vectors

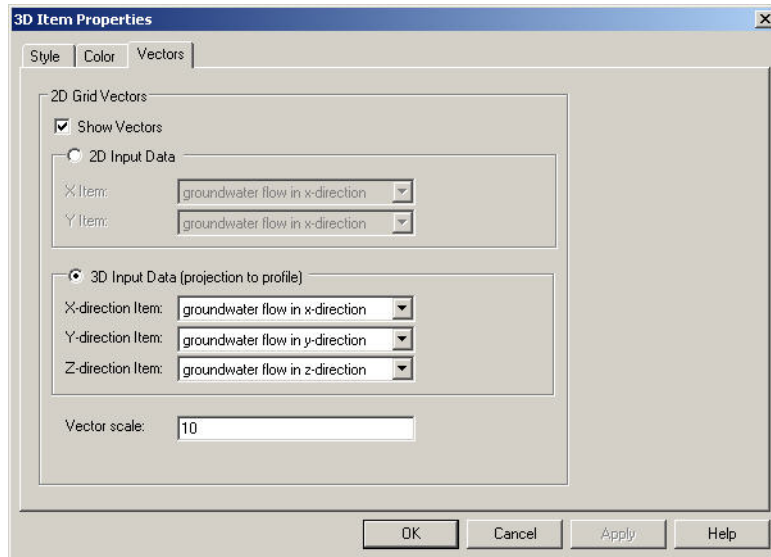


Figure 1.18 Vector properties in cross-section plot for 3D items

### 1.5.9 Cross-section

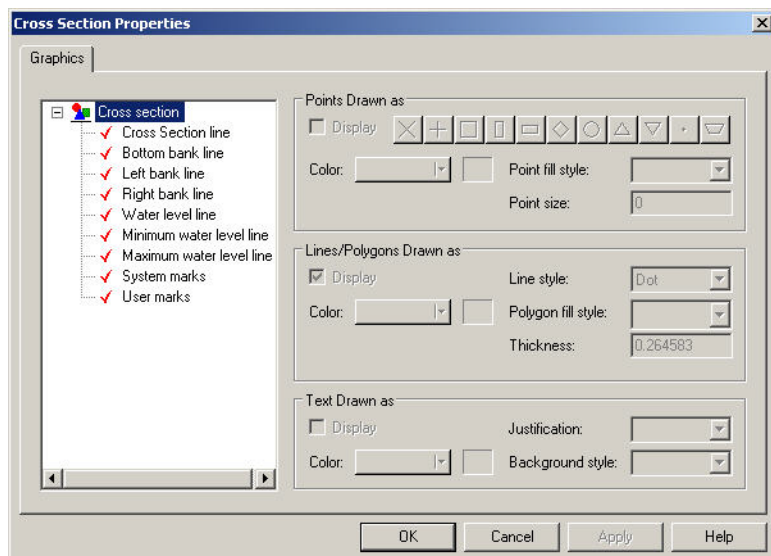


Figure 1.19 Cross-section properties

This is where you specify how a MIKE 11 cross-section can be displayed.





See also Cross-section extractor (p. 40).

### 1.5.10 UZ plot

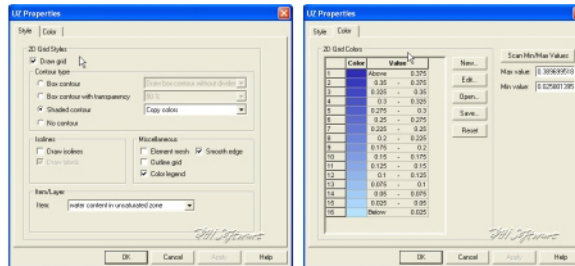


Figure 1.20 Available UZ plot properties that can be modified.

UZ properties displays unsaturated zone water contents and flow from a MIKE SHE simulation.

See also UZ Specific Plots (p. 41).

#### Style

This is where you specify the style of the contours.

#### Color

This is where you specify the colour of the contours.

### 1.5.11 Baseline

A baseline represents the spatial outline of a Profile series file defined by the geographical origin, orientation and extent.

The baseline can be shown as a combination of Lines and Points.

#### Lines

This is where you define the display of the line.

You can define the line thickness, line style and line color.

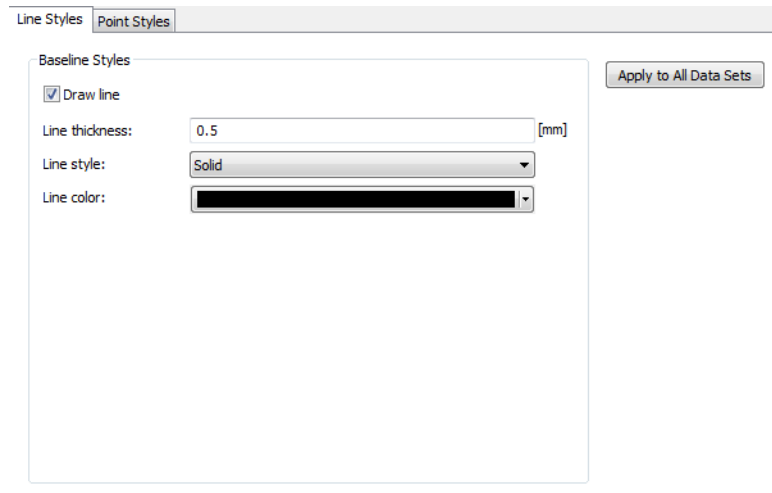


Figure 1.21 Baseline line style

## Points

This is where you define the display of the points on the line.

There are two types of point symbols:

- **Predefined vector symbols** gives you a selection of typical point symbols from which to choose. You can modify the color, point fill style and size.
- **Enhanced metafile** allows you to import and use your own point symbol in the display. You can modify the orientation and size of the metafile symbol.

Furthermore you have to option to define the drawing order.

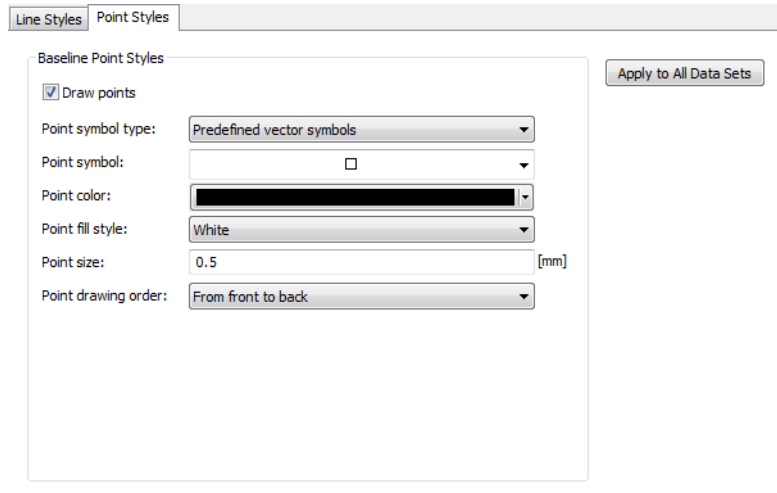


Figure 1.22 Baseline point style

### 1.5.12 Spatial line

A spatial line represents item values along a Baseline from a Profile series file, e.g. the coastline position or spatially distributed transport.

The spatial line can be shown as a combination of Lines and Points.

#### Lines

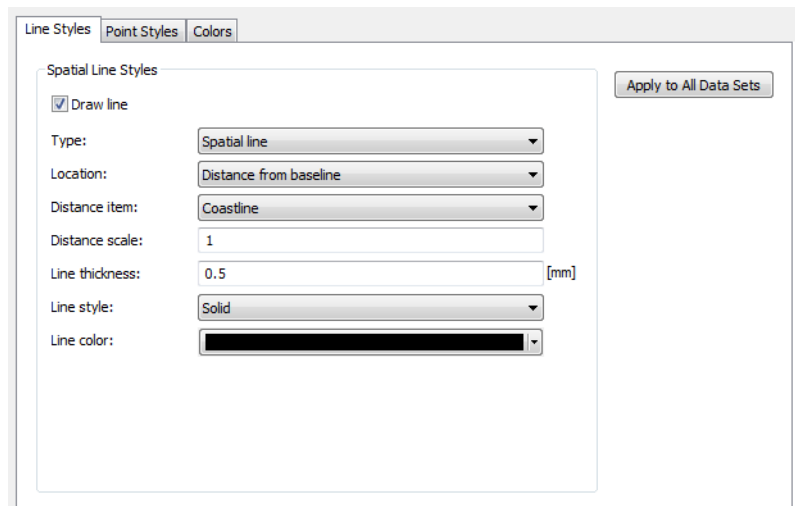


Figure 1.23 Spatial line line style properties

This is where you define the display of the spatial line.

You can display the line in two ways:

- **Spatial line**  
A line will display the item values in the file
- **Vector profile**  
The item values will be displayed as vectors extending perpendicular from the baseline. You will have the option of selecting the number of vectors to show.

You can specify the line in two ways:

- **Distance from baseline**  
The chosen item value is interpreted as a distance from the baseline. You will have the option of scaling this distance in the display.
- **Geographical position**  
You must define two items representing the (x,y) position of the grid points representing the line.

You can define the line thickness, line style and line color.

## Points

This is where you define the display of the points on the line.

The screenshot shows a software interface for configuring point styles. It includes a title bar with tabs for 'Line Styles', 'Point Styles', and 'Colors'. The main area is titled 'Spatial Line Point Styles' and contains a list of settings with checkboxes and dropdown menus. A button labeled 'Apply to All Data Sets' is located in the upper right corner of the configuration area.

Figure 1.24 Spatial line point style properties

There are two types of point symbols.

- **Predefined vector symbol** gives you a selection of typical point symbols from which to choose. You can also modify the color and point fill style.



- **Enhanced metafile** allows you to import and use your own point symbol in the display. You can modify the orientation of the metafile symbol.

The size of the symbol can be fixed or associated to a variable.

The color of a predefined symbol can be fixed or associated to a variable using a color palette.

Furthermore you have to option to define the drawing order.

### 1.5.13 Colors

The color palette specify the colors used for displaying data according to various levels. You may create a palette and save it for later use.

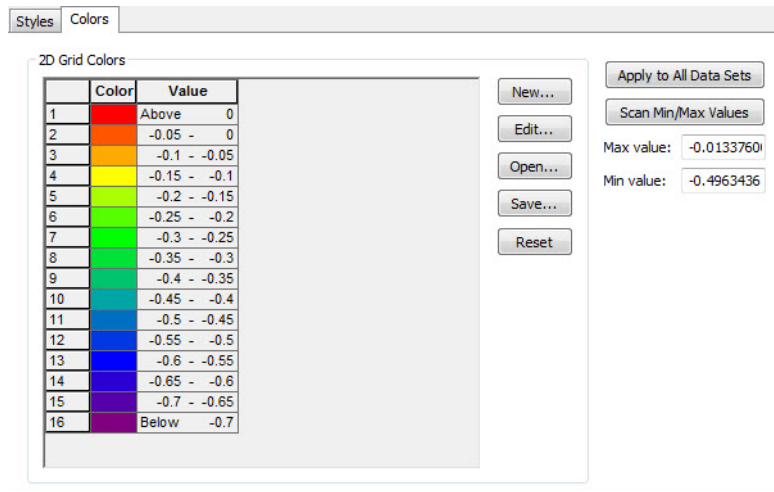


Figure 1.25 Color properties

### 1.5.14 Particles

This is where you define the display of the individual particles.

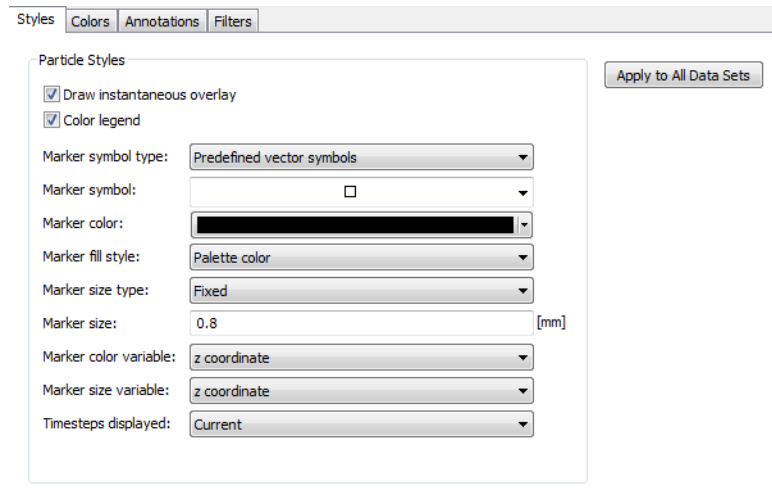


Figure 1.26 Particles properties

The particles are displayed by marker symbols.

There are two types of Marker symbols:

- **Predefined vector symbol** gives you a selection of typical point symbols from which to choose. You can also modify the color and marker fill style.
- **Enhanced metafile** allows you to import and use your own point symbol in the display. You can modify the orientation and size of the metafile symbol

You may choose to have a fixed size for all the markers or have the size vary according to the value of a selected variable. In the latter case the marker scale factor can be used to resize the marker.

You may choose to display the particles for only the current time step, a number of recent time steps or all the time steps. The tail length defines the number of recent time steps to display.

## Annotations

You can show annotations for selected particles.

You may choose to show all variables, or a selected few.

The display of annotations can also be specified by Filters.



Styles | Colors | Annotations | **Filters**

Particle Annotations

Annotation type: Values  Display unit

Annotation style: Group vertically  Align position with center

Annotation color:

	Display	Order	Variable name	Item name
1	<input type="checkbox"/>	0	Class name	Class
2	<input checked="" type="checkbox"/>	1	Particle number	Nr
3	<input type="checkbox"/>	0	x coordinate	x
4	<input type="checkbox"/>	0	y coordinate	y
5	<input type="checkbox"/>	0	z coordinate	z
6	<input type="checkbox"/>	0	Mass	Mass

Apply to All Data Sets

Figure 1.27 Annotations for Particles

### Filters

You can define filters such that only a part of the particles or annotations are displayed.

Styles | Colors | Annotations | **Filters**

Particle Filters

Apply resulting filter to: Particles and annotations

Filters:

2 | V(Nr) < 5 | AND

Apply to All Data Sets

Figure 1.28 Filters for displaying particles

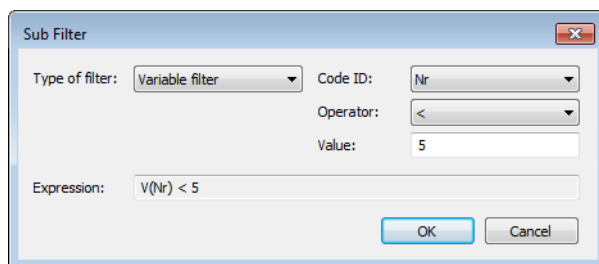


Figure 1.29 Specifying filter: this only display the first 4 particles

### 1.5.15 Particle tracks

This is where you define the display of the particle tracks.

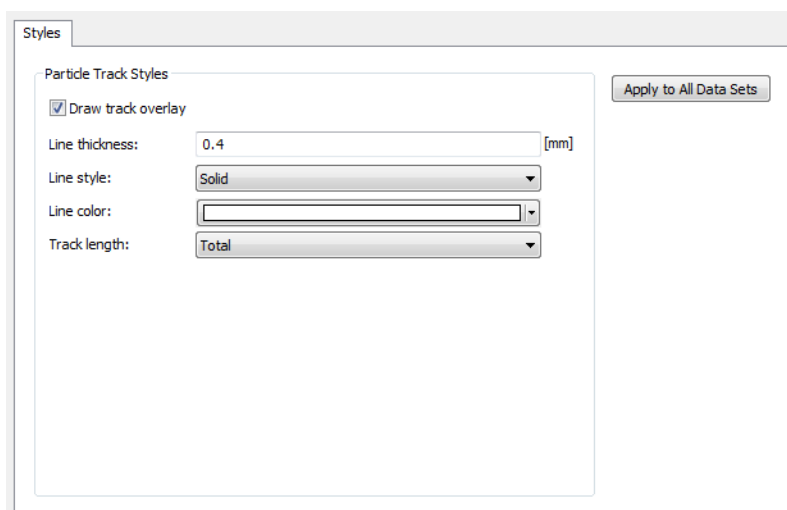


Figure 1.30 Particle Track style properties

You can define the line thickness, style and color of the lines.

The Track length may be displayed as the total track, the track from start to current time step or the recent track only (current with Tail).

### 1.5.16 XYZ data

#### Line Styles

The Line Styles dialog is an entry where you describe the column order of the xyz data to display.



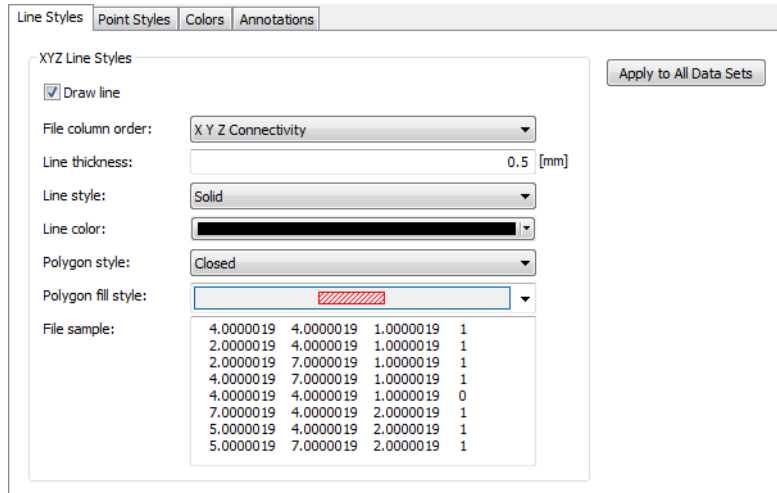


Figure 1.31 Line styles for lines connecting xyz points

Only the first three or four columns in the xyz files are considered (depending on the number of columns in the file). A sample of the contents from the selected xyz file is displayed for overview.

It is possible to connect the individual xyz points with a line. You can define the thickness, style and color for the lines. It is possible to display only closed polygons and to define polygon fill style.

In case the file contains xyz data only, the points connects as they are listed in the xyz file, otherwise the connecting lines are defined by the values in the connectivity column.

### Point Styles

This is where you define the display of the individual xyz points.



Line Styles | Point Styles | Colors | Annotations

XYZ Point Styles

Draw points

Color legend

Point symbol type: Predefined vector symbols

Point symbol: ▽

Point color: [Black]

Point fill style: Palette color

Point size type: Fixed

Point size: 0.5 [mm]

Point drawing order: From front to back

Point frequency: 1

Apply to All Data Sets

Figure 1.32 Point styles for xyz points

The points are displayed by marker symbols.

There are two types of Marker symbols:

- Predefined vector symbol gives you a selection of typical point symbols from which to choose. You can also modify the color and marker fill style.
- Enhanced metafile allows you to import and use your own point symbol in the display. You can modify the orientation and size of the metafile symbol

You may choose to have a fixed size for all the markers or have the size vary according to the value of a selected variable. In the latter case the marker scale factor can be used to resize the marker.

You may choose to display only a part of the points by specifying a point frequency > 1.

### Annotations

You can show annotations for selected points. You may choose to show all the variables, or a selected few.

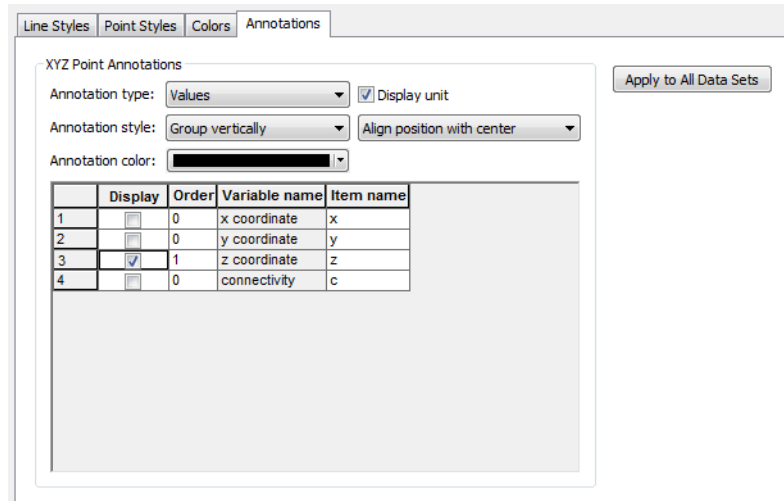


Figure 1.33 Annotation for xyz point data

## 1.6 Window Menu

This menu provides you with the standard Windows views and allows you to switch between different open windows.

## 1.7 Result Viewer Toolbar



Many of the functions in the Results Viewer are the same as those available in other DHI software tools (e.g., 2D Grid Editor). Additional tools available in the result viewer are summarized in Table 1.1 and below.

Table 1.1 Description of Result Viewer tools

Button	Name	Description
	Rewind	Rewinds result files to first time step
	Previous Step	Rewinds result files to the previous time step.
	Video Reverse	Generates an avi file from the current time step to the first time step

Table 1.1 Description of Result Viewer tools



















Button	Name	Description
	Play Reverse	Plays result files from the current time step to the first time step. Identical to Video Reverse except an avi file is not generated.
	Stop Animation	Stops forward and reverse playing of result files and creation of avi files. <b>Note:</b> It is always possible to stop an animation by pressing the <esc> key.
	Play Forward	Plays result files from the current time step to the last time step. Identical to Video Forward except an avi file is not generated.
	Video Forward	Generates an avi file from the current time step to the last time step
	Next Step	Advances result files from the current time step to the next time step
	Wind	Advances results files to the last time step
	Go to time step	Rewinds or advances result files to the specified time step
	Time step	Change the time step used by the result viewer. The time step can be less than or greater than the result file time step
	Default	Default selection tool.
	Time series extractor	Tool to extract time series data from result files. Multiple time series can be extracted by holding down the Ctrl key while left-clicking. A single extraction or the last multiple extraction is selected using a double left-click. (See also p. 37).
	Profile Extractor	Tool to extract vertical profiles (cross-sections) from 3D result files. Vertices of a profile line are specified with a single left click and the profile line is closed with a double left-click. (See also p. 38)
	Cross-section extractor	Tool to extract cross-sections of MIKE 11 results at H-points. (See also p. 40)



Table 1.1 Description of Result Viewer tools

Button	Name	Description
	UZ Plot	Tool to extract a UZ plot of the water content in the unsaturated zone. This tool generates a plot of water content versus depth with time. This tool can only be used on one cell at a time. A cell is selected by double left-click. (See also p. 42).
	UZ Scatter plot	Limits displays of results to unsaturated zone calculation cells. This button is only activated if unsaturated zone data is displayed in the result viewer. (See also p. 41).
	UZ Filled Plot	Displays interpolated unsaturated zone results in non-calculation cells and unsaturated zone results in calculation cells. This button is only activated if unsaturated zone data is displayed in the result viewer. (See also p. 41).
	Longitudinal pipe plot	Displays pipes in longitudinal plot. Consecutive nodes must be selected
	Time series pipe plot	Display time series plot for pipe network. Multiple time series can be selected

### 1.7.1 Time series extractor

The Time Series tool allows you to plot a time series of all the data available in the current view. To select display of time series data you must click on the Time series extractor icon, .

A single time series can be selected by double-clicking in the desired location. Time series can be extracted from multiple locations by holding down the Ctrl-key and left clicking on each desired location. When selecting multiple locations the Ctrl-key should be held down while double clicking on the last location.

After selecting the locations of the time series files to extract you have the option to deselect some of the selected points and to accumulate the data over the simulation period (Figure 1.34). After making the appropriate selections/deselections press the OK button to generate the time series plot. The entire extraction process can be stopped by pressing the Cancel button. An example of a time series plot generated in the Results Viewer is shown in Figure 1.35.

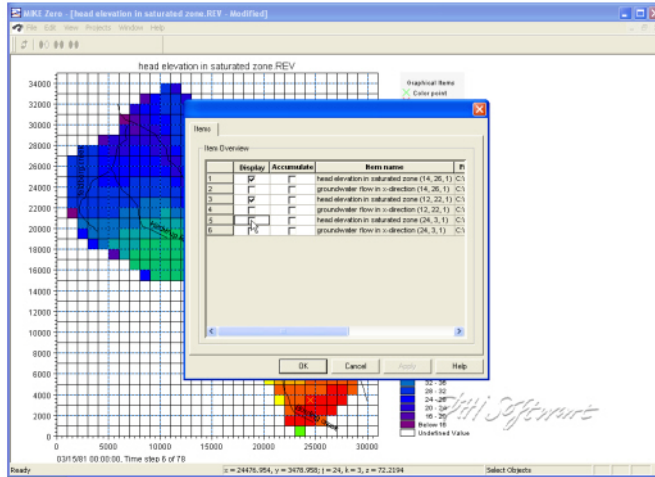


Figure 1.34 Selection of time series items to extract

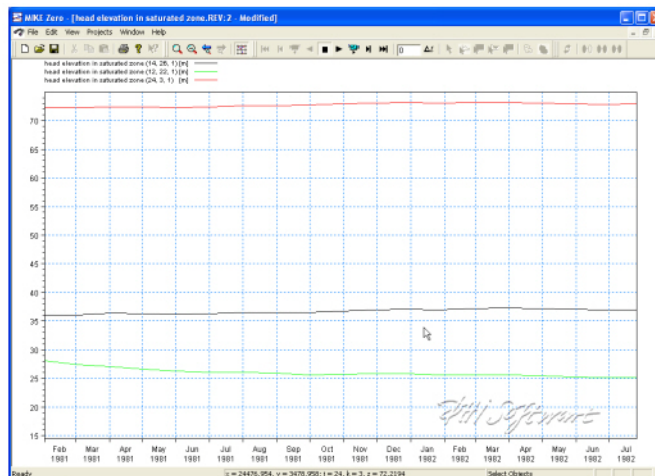


Figure 1.35 Time series plot generated using the time series extraction tool

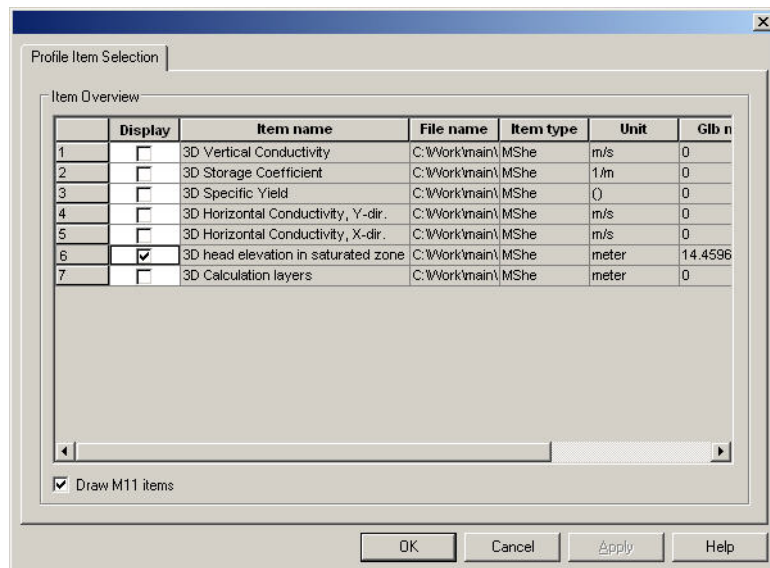
Addition graphical functions can be accessed by right-clicking in the graphical view including zooming, exporting images, exporting time series data as dfs0 files, and modification of the time series plot properties.

## 1.7.2 Profile Extractor

To display a cross section plot of a set of 3D gridded data, you must click on the Profile icon, . Clicking on this icon will allow you to interactively define a cross-section by left-clicking at each vertex of the profile line and double-clicking to close the profile.



After closing the profile, the following dialogue will be displayed listing the available output items.



Only one of these items can be selected. After selecting your item, click OK and the profile will be displayed.

The profiles extractor tool can be used to extract a cross-section through simulated MIKE SHE and MIKE 11 results. The type of cross-section created is dependent on the simulated data displayed in the result viewer. For example, if the result viewer contains simulated 3D heads and MIKE 11 results then the cross-section will have simulated water levels and simulate MIKE 11 canal stages.

After defining the profile, the items to be displayed on the profile should be selected. The resulting profile is shown in Figure 1.36. As with the other tools, extracted profiles can be animated on the screen and/or exported as avi and image files.

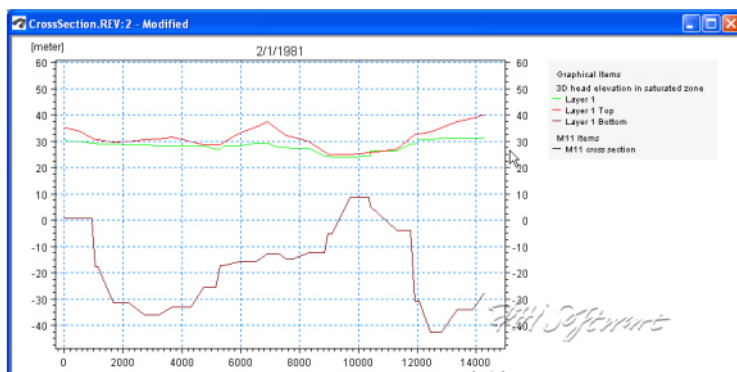


Figure 1.36 Resultant profile generated with the profile extractor tool

You can modify the plot by right clicking on the plot and selecting Properties from the pop-up menu, see Profiles (p. 18).

As with the other tools available in the result viewer, users should experiment with the available options to learn how to fully use the result viewer profile extractor.


## Saving and loading profiles

If you have a profile open, under the View/Profile item in the top menu bar (p. 9), you can save the current profile location. This allows you to create standard profiles for comparing scenarios.

To load a saved profile, make the plan view plot active, by either minimising or closing open profile plots. The View/Profile/Load option becomes active and you can load a saved profile and select the profile item normally.

### 1.7.3 Cross-section extractor

MIKE 11 results can also be added to the result viewer and simulated canal water levels can be displayed using the Cross-section extractor. The cross-section extractor shows simulated stages and the geometry of the cross-section being viewed.

To display a cross section plot from MIKE 11 results, you must click on the Cross-section icon, . After selecting the cross-section extractor tool, move the cursor over the location you want to extract the MIKE 11 results from (Figure 1.37). The simulated results are displayed along with the cross-section geometry (Figure 1.38). As with the other tools, extracted profiles can be animated on the screen and/or exported as .avi and image files.



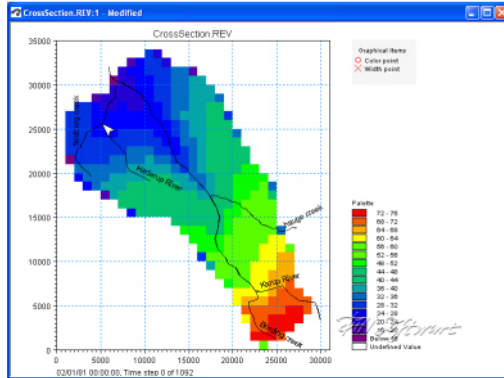


Figure 1.37 Selection of a MIKE 11 cross-section location in the result viewer

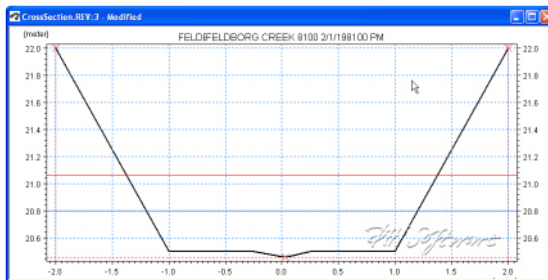


Figure 1.38 Resultant MIKE 11 cross-section plot

Addition graphical functions can be accessed by right-clicking in the graphical view. Since the cross-section plots are relatively simple, modifications are limited to changing line and marker properties, cross-section markers, etc. See also Cross-section (p. 24).

## 1.7.4 UZ Specific Plots

### UZ Scatter and Filled Plots

For unsaturated zone results, scatter or filled plots can be generated. UZ Scatter and Filled Plots are only different for simulations that do not use the “calculation in all cells” UZ module option.

UZ Scatter plots only show simulated results for UZ calculation cells. The number of UZ calculation cells may be less than the total number of active model domain used by the overland and saturated zone modules if the UZ module for the simulation is not using the “calculation in all cells” option. An example of when use of the UZ scatter plot is useful is shown in Figure 1.39.

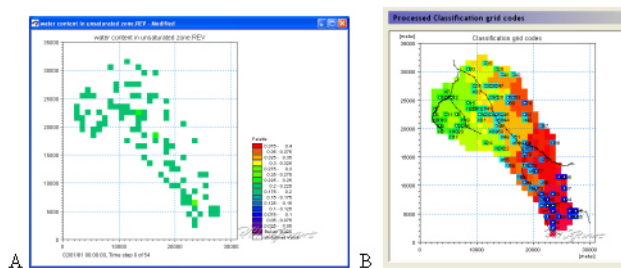


Figure 1.39 A) UZ Scatter Plot and its relationship to B) the UZ calculation cells

An example of a UZ Filled Plot is shown in Figure 1.40. In cases where the UZ module for the simulation is not using the “calculation in all cells” option the Result Viewer interpolates values from the calculation cells to adjacent inactive UZ cells.

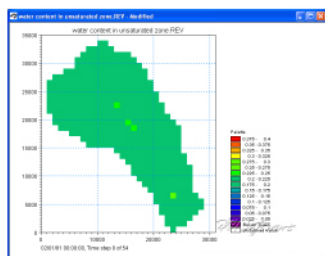



Figure 1.40 UZ Filled plot

## UZ Plot

UZ Plots can only be extracted from simulated unsaturated zone water contents and flow. This is because UZ plots display results for a single column for all of the UZ calculation nodes in the column. Other simulated UZ results show net values for the entire UZ (i.e., infiltration, recharge to the SZ, etc.).

After selecting the UZ Plot extractor tool, , move the cursor over the column you want to extract the results from and double-click (Figure 1.41). Results from multiple UZ columns cannot be displayed on the same UZ Plot.

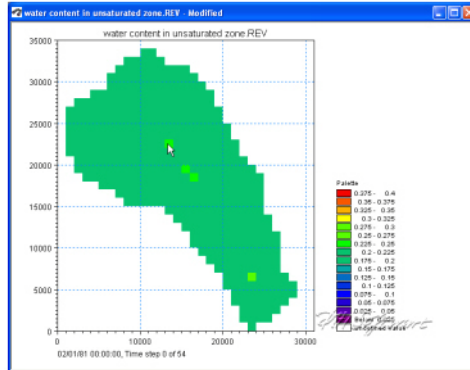


Figure 1.41 Extracting a UZ Plot from simulated unsaturated water contents and flow

The simulated water content results for the selected column are displayed in Figure 1.42. The UZ Plots show either water content or unsaturated zone flow for each node in the column (y-axis) for the entire simulation (x-axis).

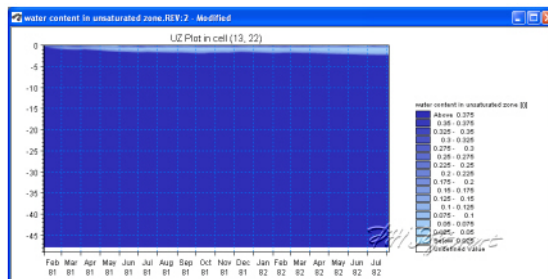


Figure 1.42 Example UZ plot of unsaturated zone water content

Addition graphical functions can be accessed by right-clicking in the graphical view. Modifications that can be made include changing the interpolation methods, adding the mesh, adding isolines, changing the colour schemes, etc. See also UZ plot (p. 25).

An example of a modified UZ plot with the mesh displayed and only showing the upper five meters of the soil column is shown in Figure 1.43.

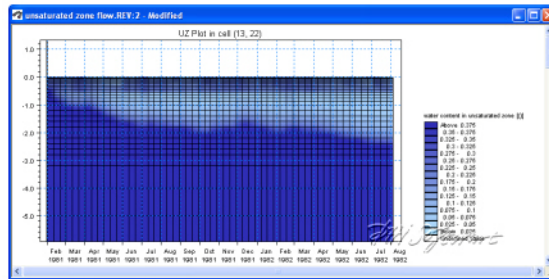


Figure 1.43 Close up of upper 5 meters of soil column with the calculation grid displayed



# INDEX



<b>B</b>	
Batch command . . . . .	9
<b>D</b>	
Delete value . . . . .	11
Derived item . . . . .	11
<b>S</b>	
Shape file . . . . .	9
<b>V</b>	
video . . . . .	13