MIKE
Powered by DHI

MIKE Zero Toolbox

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 Section III of 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 OUT OF THE USE OF OR THE INABILITY TO USE THIS DHI SOFTWARE PRODUCT, 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 COUNTRIES OR STATES 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 TO YOU. BY YOUR OPENING OF THIS SEALED PACKAGE OR INSTALLING OR USING THE SOFTWARE, YOU HAVE ACCEPTED THAT THE ABOVE LIMITATIONS OR THE MAXIMUM LEGALLY APPLICABLE SUBSET OF THESE LIMITATIONS APPLY TO YOUR PURCHASE OF THIS SOFTWARE.'

CONTENTS

MZ Tools: Concatenation . . . 11

1 Concatenation . . . 13
- 1.1 Select Data Files . . . 13
- 1.2 Subseries Selection . . . 13
- 1.3 Item Selection . . . 13
- 1.4 Options . . . 13
- 1.5 Output Specification . . . 14
- 1.6 Overview . . . 14

MZ Tools: Extraction . . . 15

2 Extraction of Time Series from 1D Data . . . 17
- 2.1 Input . . . 17
- 2.2 Subseries Selection . . . 17
- 2.3 Item Selection . . . 17
- 2.4 Point Selection . . . 17
- 2.5 Output Specification . . . 17
- 2.6 Overview . . . 18

3 Extraction of Time Series from 2D Data . . . 19
- 3.1 Input . . . 19
- 3.2 Subseries Selection . . . 19
- 3.3 Item Selection . . . 19
- 3.4 Item Selection . . . 20
- 3.5 Point Selection . . . 20
- 3.6 Output Specification . . . 21
- 3.7 Overview . . . 21

4 Extraction of Time Series from 3D Data . . . 23
- 4.1 Input . . . 23
- 4.2 Subseries Selection . . . 23
- 4.3 Item Selection . . . 23
- 4.4 Point Selection . . . 24
- 4.5 Output Specification . . . 25
- 4.6 Overview . . . 25

5 Extraction of Profile series from 2D Data . . . 27
- 5.1 Input . . . 27

5.2 Subseries Selection . . . 27
5.3 Item Selection . . . 27
5.4 Line Selection . . . 27
5.5 Output Specification . . . 28
5.6 Overview . . . 28

6 Extraction of Profile series from 3D Data . . . 29
6.1 Input . . . 29
6.2 Subseries Selection . . . 29
6.3 Item Selection . . . 29
6.4 Line Selection . . . 29
6.5 Output Specification . . . 30
6.6 Overview . . . 30

7 Extraction of 2D time series from 3D Data . . . 31
7.1 Input . . . 31
7.2 Subseries Selection . . . 31
7.3 Matrix Selection . . . 31
7.4 Output Specification . . . 31
7.5 Overview . . . 31

MZ Tools: File Converter . . . 33

8 Geo-Referencing Image File . . . 35
8.1 Select Image File . . . 35
8.2 Geo-referencing information . . . 36
8.3 Overview . . . 36

9 Mesh Converter . . . 37
9.1 Input format . . . 37
9.2 Non-MZ Bathymetry Input . . . 37
9.3 Dfs2 Bathymetry Input . . . 38
9.4 Dfsu Result File . . . 38
9.5 Units of Input Data . . . 38
9.5.1 Projected coordinates . . . 38
9.5.2 Geographical coordinates . . . 39
9.6 Element Type . . . 39
9.7 Output Data . . . 39
9.8 Overview . . . 39
9.9 Examples . . . 40
9.9.1 Rectangular grid bathymetry . . . 41
9.9.2 Curvilinear grid bathymetry . . . 42
9.9.3 ADCIRC bathymetry . . . 43
9.9.4 SMS TUFLOW bathymetry . . . 44
9.9.5 Dfsu result file . . . 45

MZ Tools: GIS . . . 47

10 GIS - Grd2Mike . . . 49
10.1 Input . . . 49
10.2 Data Specification . . . 49
10.3 Output Specification . . . 50
10.4 Overview . . . 50

11 GIS - Mike2Grd . . . 51
11.1 Input . . . 51
11.2 Data Specification . . . 52
11.3 Output Specification . . . 52
11.4 Overview . . . 52

12 GIS - Mike2Shp . . . 53
12.1 Input . . . 53
12.2 Data Specification . . . 53
12.3 Output Specification . . . 53
12.3.1 From dfs2 file . . . 54
12.3.2 From 2D dfsu file . . . 54
12.3.3 From 3D dfsu file . . . 55
12.3.4 From mesh file . . . 55
12.4 Overview . . . 55

13 GIS - Mike2Txt . . . 57
13.1 Input . . . 57
13.2 Data Specification . . . 57
13.3 Output Specification . . . 58
13.4 Overview . . . 58

14 GIS - Shp2xyz . . . 59
14.1 Input File . . . 59
14.2 Data Specification . . . 59
14.3 Output Specification . . . 59
14.3.1 Point feature . . . 59
14.3.2 Polyline feature . . . 60
14.3.3 Polygon feature . . . 60
14.4 Overview . . . 60
14.5 Example . . . 60
14.5.1 Purpose . . . 60
14.5.2 Input data . . . 61
14.5.3 Output data . . . 61

MZ Tools: Hydrology . . . 63

15 Maximum Rainfall . . . 65
15.1 Input File . . . 65
15.2 Parameters . . . 65

15.3 Output File . . . 65
15.4 Summary . . . 65

16 Mean Area Weighting Algoritm . . . 67
16.1 Weighted Average dfs File Input . . . 67
16.2 Weighted Average Weight Combination . . . 67
16.3 Temporal Distribution dfs Input File . . . 67
16.4 Temporal Distribution Weight Combinations . . . 67
16.5 Output Specification . . . 67
16.6 Overview . . . 67

17 Normalize Rainfall . . . 69
17.1 Normalize Rainfall Input Files . . . 69
17.2 Normalize Rainfall Output File . . . 69
17.3 Normalize Rainfall Summary . . . 69

18 Runoff Analysis . . . 71
18.1 Input Files . . . 71
18.2 Accumulation . . . 71
18.3 Plot . . . 71
18.4 Output File . . . 71
18.5 Summary . . . 71

MZ Tools: Statistics . . . 73

19 Statistics for DFS Data Files . . . 75
19.1 Dimension . . . 75
19.1.1 Time series . . . 75
19.1.2 Line series . . . 75
19.1.3 Matrix series . . . 75
19.1.4 Volume series . . . 75
19.2 Input . . . 75
19.2.1 Derived items . . . 76
19.3 Subseries Selection . . . 77
19.4 Calculation Parameters . . . 77
19.5 1D Line . . . 77
19.6 2D Area . . . 77
19.7 3D Area . . . 77
19.8 Statistics . . . 77
19.9 Output Specification . . . 78
19.10 Status . . . 78
19.11 Scientific Background . . . 78

MZ Tools: Time Series . . . 81

20 Interpolation of Time Series . . . 83
20.1 Input . . . 83

20.2 Interpolation Method and Subseries Selection . . . 83
20.2.1 Interpolation method . . . 83
20.3 Output Specification . . . 84
20.4 Status - Overview . . . 84
21 One Year Summary . . . 85
21.1 Welcome . . . 85
21.2 Input File . . . 85
21.3 Summary . . . 85
22 Pre-Processing Temporal Data . . . 87
22.1 Setup Name . . . 87
22.2 Station Locations . . . 87
22.3 Interpolation Method . . . 87
22.4 Output Type . . . 88
22.5 Spatially Distributed Output . . . 88
22.6 Weighted Average Output . . . 88
22.7 Temporal Extent . . . 88
22.8 Geographical Overview . . . 88
22.9 Time Series Weights . . . 89
22.10 Overview . . . 89
22.11 Scientific Background . . . 89
22.11.1 Thiessen Polygons . . . 89
22.11.2 Inverse distance weighting . . . 91
22.12 Examples . . . 92
22.12.1 Rainfall map . . . 92
23 Sorting Temporal Data . . . 95
23.1 Input Time Series File . . . 95
23.2 Subseries Selection . . . 95
23.3 Class Delimiters . . . 95
23.4 Output Data Description . . . 95
23.5 Overview . . . 96
23.6 Scientific Background . . . 96
23.7 Examples . . . 96
23.7.1 Wind measurements . . . 96
23.7.2 Wave climate . . . 101
24 Time Series Batch Conversion . . . 105
24.1 General Introduction . . . 105
24.1.1 dfs0 . . . 105
24.1.2 ASCII . . . 106
24.1.3 Excel . . . 107
24.2 Select Input Data . . . 108
24.3 Define Template . . . 108
24.4 Delimiter Description . . . 109
24.5 Time Description . . . 109

24.6 Data Description . . . 109
24.7 Item Description . . . 110
24.8 Save Template File . . . 110
24.9 Output Specification . . . 110
24.10 Overview . . . 111
24.11 Preview . . . 111
24.12 Examples . . . 111
24.12.1 ASCII to dfs0 . . . 111
24.12.2 Excel to dfs0 . . . 121

MZ Tools: Transformation . . . 127

25 Rotate Grid . . . 129
25.1 Setup Name . . . 129
25.2 Select Input File . . . 129
25.3 Subseries Selection . . . 129
25.4 Select Item . . . 129
25.4.1 Derived items . . . 129
25.5 Grid Size . . . 130
25.6 Land-Water Options and Values Outside Grid . . . 131
25.7 Output Specification . . . 132
25.8 Status - Overview . . . 132

Index . . . 133

MZ TOOLS: CONCATENATION

The CONCATENATION Tools in the MZ Toolbox include the facility:

- Concatenation

1 Concatenation

You can use this tool to concatenate files in dfs format, i.e. items from N different files will be joined in a single output file.

1.1 Select Data Files

On this page you select the number of input files and specify each.

The data must have equidistant time axis.

NOTE: It is not possible to concatenate decoupled flux files.

1.2 Subseries Selection

This page allows you to select a specific temporal subset of the input data you want to include in the concatenation.

The input files that lies in this period will be concatenated in the output. You may set a time step (in an integer number of time steps) for the concatenated output.

1.3 Item Selection

On this page you specify the number of output items and the items to be concatenated for each input files.

The item numbers must be ascending and each item number can only be used once.

1.4 Options

On this page you may specify three options.

Insert Delete if Gaps.

If the input files do not overlab in time, this option can be enabled to ensure execution by replacing missing time steps by Delete Values.

First File Defines Time

In general, all input files must have the same time step length and be aligned in time. If the time steps are not aligned, this option forces the output files to use the temporal alignment of the first input file and simply append all time steps from subsequent input files without interpolation.

Overwrite with latest

When the temporal range of the input files overlab, default is to use the data in first file in the output. If this option is enables, the data in last file is used.

1.5 Output Specification

On this page you specify the name and location of the output file, i.e. the file that will contain the resulting statistical values.

1.6 Overview

The overview page presents you a summary of all the input parameters you have specified. You should inspect that the parameters are correct and if not go back and change them.

From the page you can either *Finish* the setup which will transfer your setup definition to the current toolbox file or execute the setup.

MZ TOOLS: EXTRACTION

The EXTRACTION Tools in the MZ Toolbox include the facilities:

- Extraction of Time Series from 1D Data
- Extraction of Time Series from 2D Data
- Extraction of Time Series from 3D Data
- Extraction of Profile series from 2D Data
- Extraction of Profile series from 3D Data
- Extraction of 2D time series from 3D Data

2 Extraction of Time Series from 1D Data

You use this tool to extract time series of point values from time series of 1D data. Given an input 1D data file and a number of discrete points, the tool extracts the values from the 1D data set at the points within a given period.

The resulting time series is written to a standard DFS datafile.

2.1 Input

On this page you select the file with 1D data from which you want to extract point values.

2.2 Subseries Selection

This page allows you to select a specific temporal subset of the input data you want to include in the extraction.

Typically you probably want to process the whole period.

2.3 Item Selection

On the item selection page you specify which items you want to extract points from. From a type 1 file, you can only extract the items as scalars.

2.4 Point Selection

On the point selection page you specify the number of points and their location.

You can also specify a decimal point and the program will interpolate between values in the file.

2.5 Output Specification

On this page you specify the name and location of the output file for the extraction, i.e. the file that will contain the resulting time series.

You can also specify a data file title.

NOTE: If you want to write the output to an old format time series file (extension .dt0), you should limit the number of output items to a maximum of 16. That is the number points times the number of selected input items should be less than 16.

2.6 Overview

The overview page presents you a summary of all the input parameters you have specified. You should inspect that the parameters are correct and if not go back and change them.

From the page you can either *Finish* the setup which will transfer your setup definition to the current toolbox file or execute the setup.

3 Extraction of Time Series from 2D Data

You use this tool to extract time series of point values from time series of 2D data. Figure 3.1 illustrates the process.

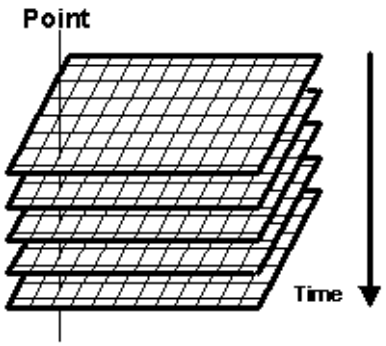


Figure 3.1 Extracting time series from grid series

Given an input 2D data file and a number of discrete points (in the figure just one), the tool extracts the values from the 2D data set at the points within a given period.

The resulting time series is written to a standard DFS data file.

3.1 Input

On this page you select the file with 2D data from which you want to extract point values.

NOTE: This functionality must be used with care for MIKE SHE files as the values from a MIKE SHE grid is grid centred.

3.2 Subseries Selection

This page allows you to select a specific temporal subset of the input data you want to include in the extraction.

Typically, you will probably want to process the whole period.

3.3 Item Selection

On the item selection page you specify which items you want to extract. You can extract the points as vectors or scalars.

If you choose to extract vectors, you can choose between the following options:

- Extract vectors directly. Choosing this option, it is not necessary to specify both X and Y components for the vector.
- Extract vectors as magnitude and direction. Here it is necessary to specify both the X and the Y component of the vector.

$$Magnitude = \sqrt{V_x^2 + V_y^2} \tag{3.1}$$

$$Direction = 90 - \tan^{-1}\left(\frac{V_y}{V_x}\right) + TrueNorth \tag{3.2}$$

- Extract vectors as projection. Here it is also necessary to specify both the X and the Y component of the vector. The program will project the magnitude of the vectors on the direction specified. The direction is specified as an angle with true north.

3.4 Item Selection

The program has now recognized your file as a Mike21 result file. Depending on whether you have Levels and Fluxes or only Levels in your result file, you can select to extract from a range of simple simulation output items and/or you can choose some "derived" items. Some items will be treated as scalars and some as vectors. This will be handled by the program.

3.5 Point Selection

On the point selection page you specify the number of points and their location.

If you chose to extract the components as scalars, the points are extracted in the coordinates (j,k) you specify in the list on this page.

If you chose to extract items as vectors, the points are not extracted in the exact coordinate that is specified in the list, but from a location placed half a grid cell down and to the left, respectively, see image below. If you for instance extract the magnitude and direction in a point with the coordinates (j,k), you will get the magnitude and direction based on model results in the points P(j,k) and Q(j,k) as illustrated below.

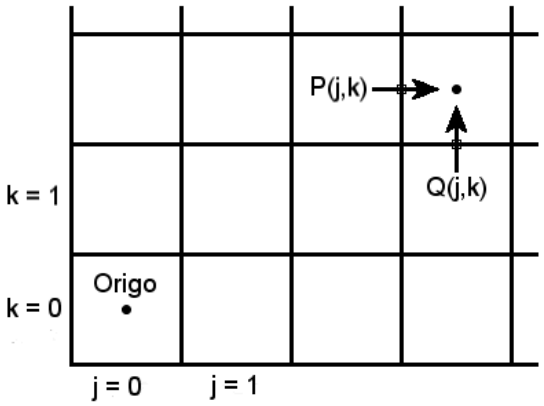


Figure 3.2 Definition of point position in extraction of vector items

3.6 Output Specification

On this page you specify the name and location of the output file for the extraction, i.e. the file that will contain the resulting time series.

You can also specify a data file title.

NOTE: If you want to write the output to an old format time series file (extension .dt0), you should limit the number of output items to a maximum of 16. That is, the number of points times the number of selected input items should be less than 17.

3.7 Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

4 Extraction of Time Series from 3D Data

You use this tool to extract time series of point values from time series of 3D data. Given an input 3D data file and a number of discrete points, the tool extracts the values from the 3D data set at the points within a given period.

The resulting time series is written to a standard DFS datafile.

4.1 Input

On this page you select the file with 3D data from which you want to extract point values.

4.2 Subseries Selection

This page allows you to select a specific temporal subset of the input data you want to include in the extraction.

Typically you probably want to process the whole period.

4.3 Item Selection

On the item selection page you specify which items you want to extract. You can extract the points as vectors or scalars.

If you choose to extract vectors, you can choose between the following options:

- Extract vectors directly. Choosing this option, it is not necessary to specify both X, Y and Z components for the vector.
- Extract vectors as magnitude and direction. Here it is necessary to specify both the X, Y and Z component of the vector.

$$Magnitude = \sqrt{V_x^2 + V_y^2 + V_z^2} \tag{4.1}$$

$$DirectionXY = \tan^{-1}\left(\frac{V_y}{V_x}\right) \tag{4.2}$$

$$DirectionZ = \tan^{-1}\left(\frac{V_z}{\sqrt{V_x^2 + V_y^2}}\right) \tag{4.3}$$

- Extract vectors as projection. Here it is also necessary to specify both the X, Y and Z component of the vector. The program will project the magnitude of the vectors on the direction specified. The direction is specified as an angle with true north and an angle with the XY plane.

4.4 Point Selection

On the point selection page you specify the number of points and their location.

If you chose to extract the components as scalars, the points are extracted in the coordinates that you specify in the list on this page.

If you chose to extract some of the items as vectors, the points are not extracted in the exact coordinate that is specified in the list on this page, but in the points in between the grid points of the original file. e.g if you specified that you want ex. the magnitude and direction in the coordinates P on Figure 4.1, you will get the magnitude and direction in the points u, v and w. On a staggered grid, that is.

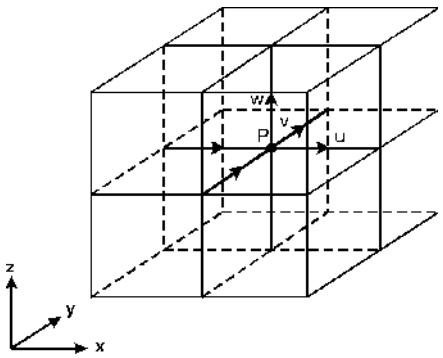


Figure 4.1 Extraction of vectors

NOTE: In the present version you can only select point at the exact grid points, i.e. the tool cannot do any spatial interpolation

4.5 Output Specification

On this page you specify the name and location of the output file for the extraction, i.e. the file that will contain the resulting time series.

You can also specify a data file title.

NOTE: If you want to write the output to an old format time series file (extension .dt0), you should limit the number of output items to a maximum of 16. That is the number points times the number of selected input items should be less than 16.

4.6 Overview

The overview page presents you a summary of all the input parameters you have specified. You should inspect that the parameters are correct and if not go back and change them.

From the page you can either *Finish* the setup which will transfer your setup definition to the current toolbox file or execute the setup.

5 Extraction of Profile series from 2D Data

This tool extracts time series of profile (line) data from 2D spatial data.

5.1 Input

On this page you select the file with 2D data from which you want to extract profile series.

NOTE: This functionality must be used with care for MIKE SHE files as the values from a MIKE SHE grid is grid centred.

5.2 Subseries Selection

This page allows you to select a specific temporal subset of the input data you want to include in the extraction.

Typically you probably want to process the whole period.

5.3 Item Selection

On the item selection page you specify which items you want to extract. You can choose the items to be vectors or scalars.

5.4 Line Selection

Here you specify the start and end coordinates for the lines within the 2D matrix, from which you want to extract profile data.

Specifying the line spacing is specifying how many points you want the line to be split in, e.g. how many points between "start point" and "end point".

If you have specified to extract some of the items as vectors, the points are not extracted in the exact coordinates on the line that you specified, but in points in between the grid points. e.g if you specified that you want a line going through the coordinates (j,k) in Figure 5.1, the program will calculate the vector components in the points P(j,k) and Q(j,k), which are afterwards projected on the line you specified.

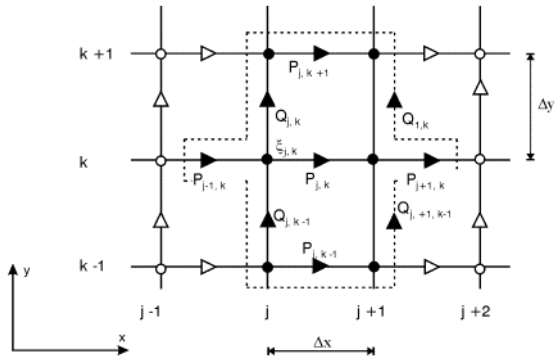


Figure 5.1 Extraction of vectors

5.5 Output Specification

On this page you specify the name and location of the output file for the extraction, i.e. the file that will contain the resulting profile series.

You can also specify a data file title.

NOTE: If you have specified one line only the geographical position of the origin and the direction of the line is saved in the dfs1 file for GIS purposes.

NOTE: If you want to write the output to an old format profile series file (extension .dt1), you should limit the number of output items to a maximum of 16. That is the number lines times the number of selected input items should be less than 16.

5.6 Overview

The overview page presents you a summary of all the input parameters you have specified. You should inspect that the parameters are correct and if not go back and change them.

From the page you can either *Finish* the setup which will transfer your setup definition to the current toolbox file or execute the setup.

6 Extraction of Profile series from 3D Data

This tool extracts time series of profile (line) data from 3D spatial data.

6.1 Input

On this page you select the file with 3D data from which you want to extract profile series.

6.2 Subseries Selection

This page allows you to select a specific temporal subset of the input data you want to include in the extraction.

Typically you probably want to process the whole period.

6.3 Item Selection

On the item selection page you specify which items you want to extract. You can extract the items as vectors or scalars.

6.4 Line Selection

Here you specify the start and end coordinates for the lines within the 3D matrix, from which you want to extract profile data.

Specifying the line spacing is specifying how many points you want the line to be split in, e.g. how many points between "start point" and "end point".

If you have specified to extract some of the items as vectors, the points are not extracted in the exact coordinates on the line that you specified, but in points in between the grid points. e.g if you specified that you want a line going through the point P on the picture below, the program will calculate the vector components in the points u, v and w, which are afterwards projected on the line you specified.

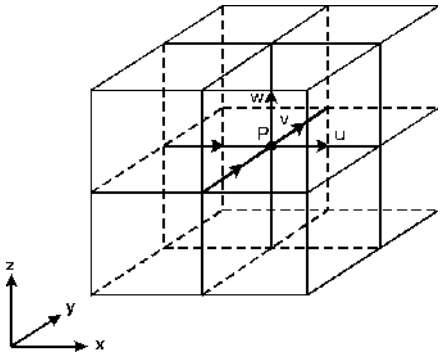


Figure 6.1 Extraction of vectors

6.5 Output Specification

On this page you specify the name and location of the output file for the extraction, i.e. the file that will contain the resulting profile series.

You can also specify a data file title.

NOTE: If you want to write the output to an old format profile series file (extension .dt1), you should limit the number of output items to a maximum of 16. That is the number lines times the number of selected input items should be less than 16.

6.6 Overview

The overview page presents you a summary of all the input parameters you have specified. You should inspect that the parameters are correct and if not go back and change them.

From the page you can either *Finish* the setup which will transfer your setup definition to the current toolbox file or execute the setup.

7 Extraction of 2D time series from 3D Data

This tool extracts time series of 2D data from 3D spatial data.

7.1 Input

On this page you select the file with 3D data from which you want to extract 2D time series.

7.2 Subseries Selection

This page allows you to select a specific temporal subset of the input data you want to include in the extraction.

Typically you probably want to process the whole period.

7.3 Matrix Selection

On this page you select the 2D slice that you want to extract.

The "Data Information" on the right hand side of the dialogue is the size of the 3D matrix in your input file.

You can define your own subarea, to be extracted to your output file, on the left-hand side.

7.4 Output Specification

On this page you specify the name and location of the output file for the extraction, i.e. the file that will contain the resulting 2D time series.

You can also specify a data file title.

NOTE: If you want to write the output to an old format 2D time series file (extension .dt2), you should limit the number of output items to a maximum of 16. That is the number lines times the number of selected input items should be less than 16.

7.5 Overview

The overview page presents you a summary of all the input parameters you have specified. You should inspect that the parameters are correct and if not go back and change them.

From the page you can either *Finish* the setup which will transfer your setup definition to the current toolbox file or execute the setup.

MZ TOOLS: FILE CONVERTER

The FILE CONVERTER Tools in the MZ Toolbox include the facilities:

- Geo-Referencing Image File
- Mesh Converter

8 Geo-Referencing Image File

An image with a related DHI world file with geo-referencing information can be included as a background layer in various MIKE Zero modules.

This tool has been implemented to create a DHI world file that contains the geo-referencing parameters for a given image file.

An example of the contents of such a DHI world file is shown below.

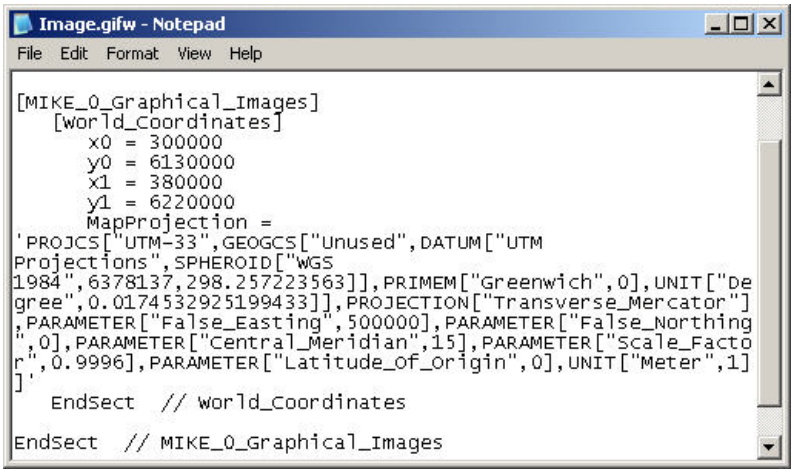


Figure 8.1 Example of DHI world file (image.gifw).

8.1 Select Image File

On this page you select the image to be geo-referenced.

The format of the image file may be one of the following:

- Bitmap (.bmp)
- JPEG (.jpg)
- GIF (.gif)
- PNG (.png)
- TIF (.tif)

The image file chosen may already be accompanied by a world file. Supported formats are:

- DHI world file format
- "older" DHI world file format without projection information
- ESRI world file format

In all cases, the tool will use the information contained in the world file to calculate the geo-referencing information.

Note that ESRI world files may contain rotation information, which is not supported by the tool.

8.2 Geo-referencing information

On this page you define the image coordinates for the lower left and upper right corner. The coordinates must be specified by the given map projection system.

NOTE: If a geo-referencing file already exists for the image, you only get the option to change the map projection.

8.3 Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either ***Finish*** the set-up, which will transfer your set-up definition to the current toolbox file, or ***Execute*** the set-up.

The created DHI World file with the geo-referencing information will have the same name as the image file with an additional ‘w’ in the end of the extension, i.e. bmpw, jpgw, gifw, pngw or tifw.

9 Mesh Converter

This tool imports bathymetry information in specific formats and converts the information into a MIKE Zero mesh file that can readily be used in the MIKE Powered by DHI software models.

9.1 Input format

Define the format of the input data. The input may be one of three types

- Non-MZ Bathymetry Input
  (data from ADCIRC or TUFLOW/SMS)
- Dfs2 Bathymetry Input
  (rectangular or curvilinear grid in dfs2 format).
- Dfsu Result File
  (flexible mesh format, bathymetry data derived from static item)

9.2 Non-MZ Bathymetry Input

Browse and select the input file.
Note that you can choose to only display a specific file extension type.

Presently you can choose between two file extension types:

- .14 (ADCIRC data)
  The z-value from ADCIRC data is assumed to describe depth, why the value is multiplied with -1 before the mesh is saved
- .2dm (SMS/TUFLOW data)
  The z-value from SMS/TUFLOW data is saved in the mesh file as is

For Non-MZ bathymetry data it is necessary to define the map projection that the (X,Y) coordinates in the input data file is referring to.

In case (X,Y) is defined in geographical coordinates, select the LONG/LAT projection found in the group 'Non UTM projections'.

NOTE: The nodes in the input file needs to be saved in increasing order. E.g. when converting a .2dm file to a .mesh file, the .2dm file's nodes need to be renumbered first in the SMS, so the Mesh Converter tool can find the correct information about the elements and nodes when the format changes.
Only the node information for nodes on external boundaries are in transferred to the resulting mesh file. The attribute for nodes within the mesh are per default set to 0.

9.3 Dfs2 Bathymetry Input

Browse and select the dfs2 input file containing the bathymetry data.

In case the bathymetry data is from a curvilinear grid, enable the Curvilinear grid check-box and browse and select the dfs2 file containing the (X,Y) coordinates relating to the bathymetry data.

The geographical information for the mesh file is derived from the input bathymetry file.

NOTE: In case the bathymetry file contains a M21_Misc custom block, only elements containing values below the land value are included in the generated mesh.

NOTE: The bathymetry data is per default derived from the first time step and the first dynamic item in the dfs2 file. In the special case where the dfs2 file contains one, and only one, static item (e.g. as for a MIKE 21 HD output file) the bathymetry data is derived from the static item.

9.4 Dfsu Result File

Browse and select the dfsu result file containing the bathymetry data.

NOTE: The bathymetry data is derived from the static item.

9.5 Units of Input Data

For Non-MZ bathymetry data it is necessary to define the units the (X,Y) coordinates and Z value is referring to in the input file.

The available units depends on the type of map projection.

9.5.1 Projected coordinates

Specify the units of the (X,Y) coordinate and the Z-value, respectively.

The projected (X,Y) coordinate must be specified in one of three units:

- Meter
- Feet
- US survey feet

The Z-level value must be specified in one of three units:

- Meter
- Feet
- US survey feet

9.5.2 Geographical coordinates

Specify the units of the (X,Y) coordinate and the Z-value, respectively.

The geographical (X,Y) coordinate must be specified using one of two units:

- Degree
- Radians

The Z-level value must be specified using one of three units:

- Meter
- Feet
- US survey feet

9.6 Element Type

For dfs2 bathymetry data it is possible to create the mesh file using one of two different element types:

- Triangular
- Quadrilateral

Note that grid cells containing land values will not be included in the resulting mesh file.

In case of triangular elements the resulting number of mesh elements will be twice the number of active grid cells in the input bathymetry.

In case of quadrangular elements the resulting number of mesh elements will be the same as the number of active grid cells in the input bathymetry.

9.7 Output Data

Browse and define the location and filename of the output mesh file.

9.8 Overview

Setup status: review the parameters. If you want to change any settings, go back; if you are satisfied with the settings click Execute, or just Finish to save the setup for later.

9.9 Examples

You can access the tool from the MIKE Zero toolbox.

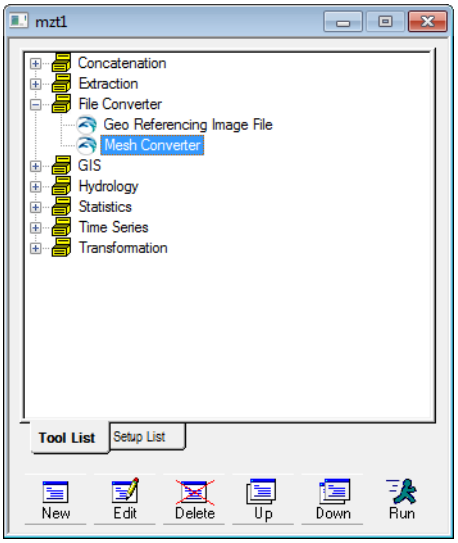


Figure 9.1 Select utility in MIKE Zero Toolbox

Setup examples of mesh conversion for various formats of input bathymetry data can be found in the default installation folder at

.\MIKE Zero\Examples\MIKE_Zero\Toolbox\FileConverter\MeshConverter. The resulting meshes are shown below.

9.9.1 Rectangular grid bathymetry

The input file is a rectangular grid bathymetry in dfs2 format.
The mesh is generated using triangular elements.

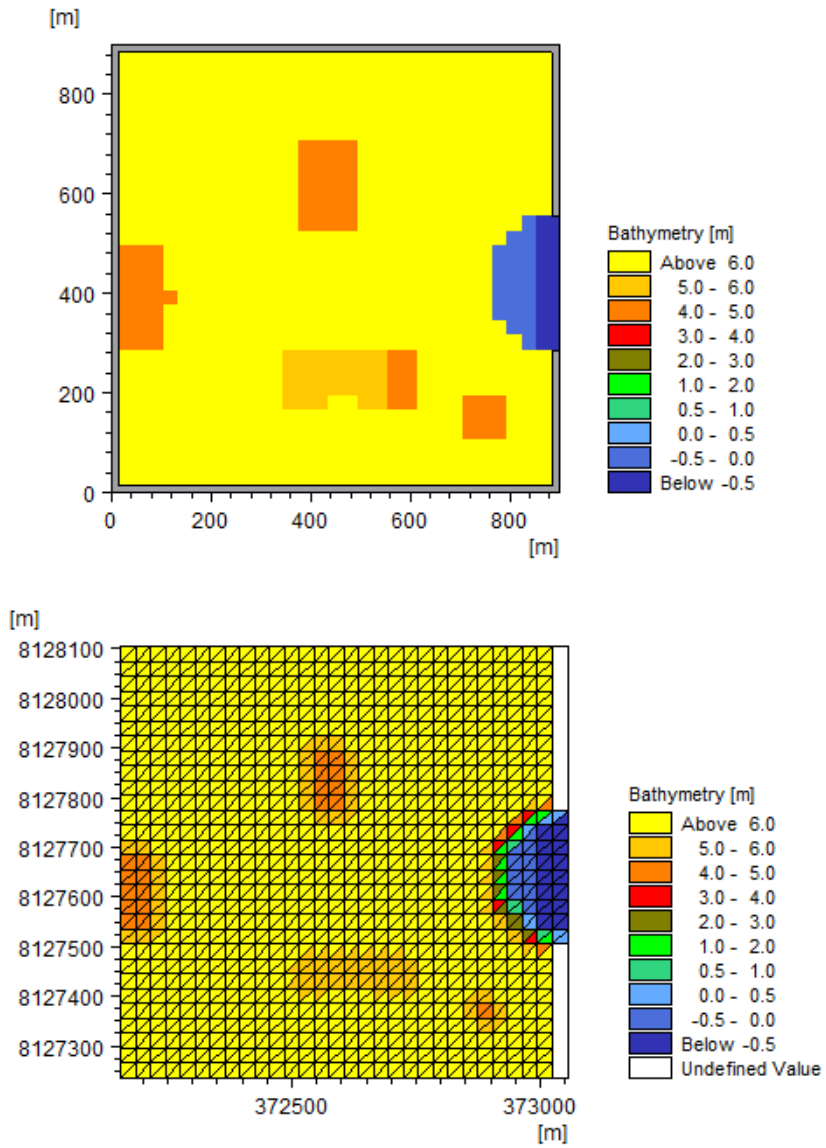


Figure 9.2 Mesh conversion of rectangular bathymetry from dfs2 file
Upper: input bathymetry
Lower: output mesh

9.9.2 Curvilinear grid bathymetry

The input file is a curvilinear grid bathymetry in dfs2 format.
The mesh is generated using quadrangular elements.

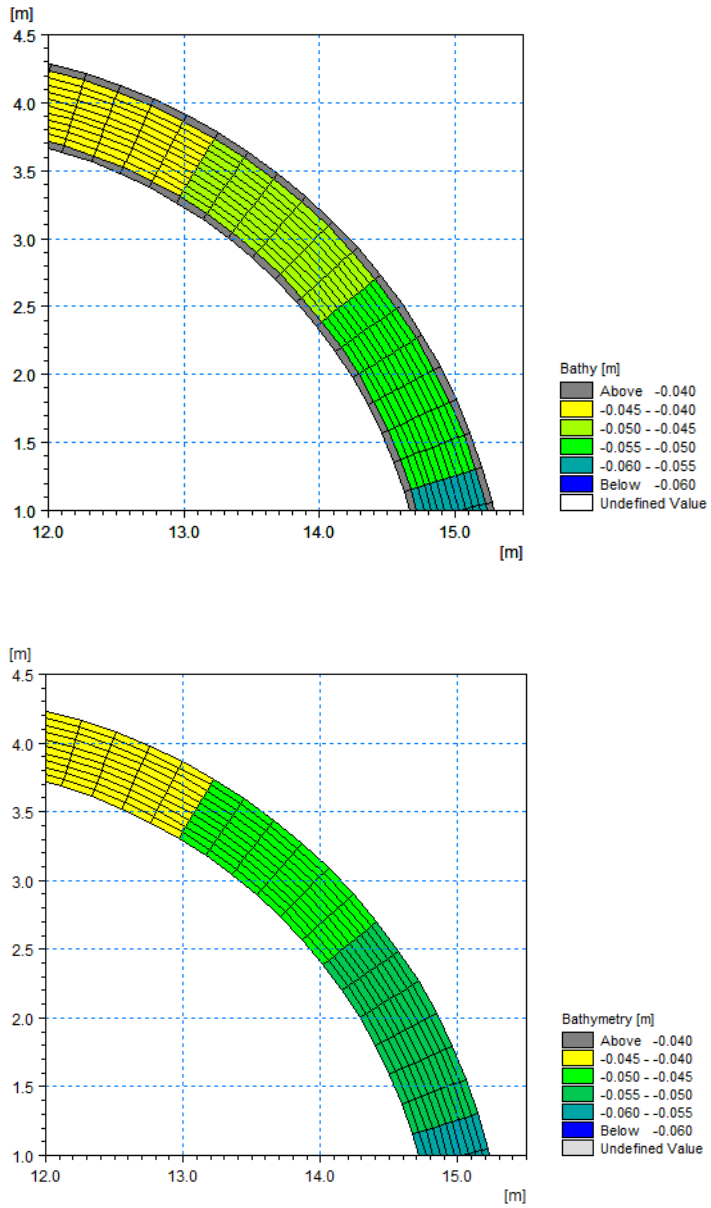


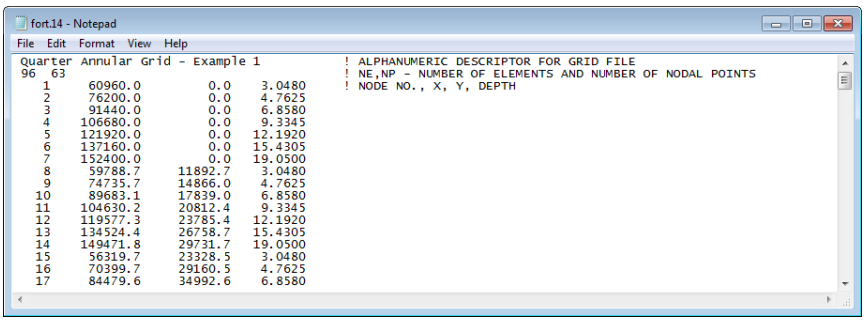
Figure 9.3 Mesh conversion of curvilinear bathymetry from dfs2 file
Upper: input bathymetry
Lower: output mesh

9.9.3 ADCIRC bathymetry

The input file is an ASCII file containing bathymetry data in ADCIRC format (.14). The file first defines the location of the individual node points, next the elements with associated node numbers.

- The mesh is created using map projection UTM-32 and the (X,Y) coordinate unit and the unit for the z-values in the input are specified as meter.

Note that for an ADCIRC bathymetry the depth values are automatically multiplied with -1 before the mesh is saved.



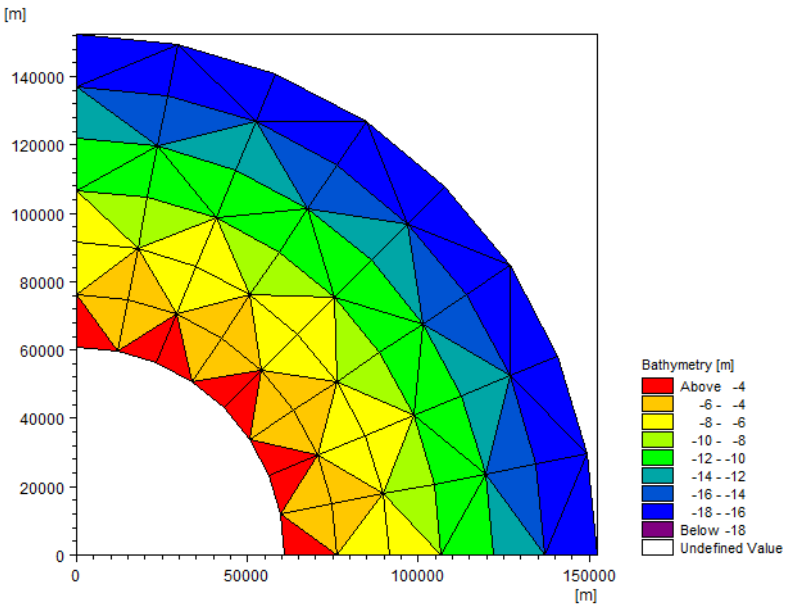


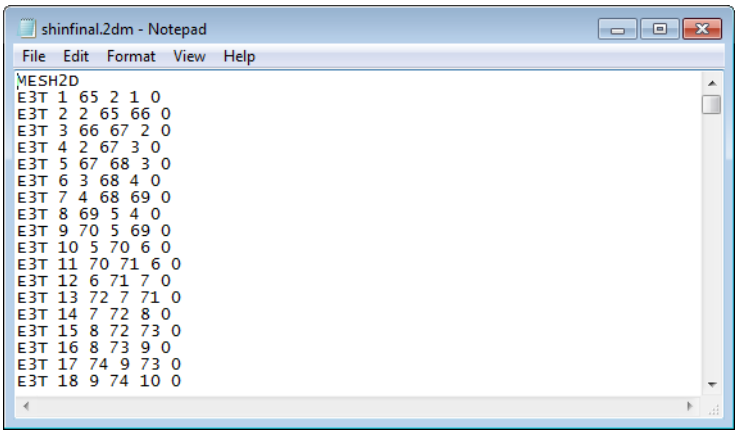
Figure 9.4 Mesh conversion of ADCIRC bathymetry data
Upper: start of input bathymetry data in ASCII file
Lower: output mesh

9.9.4 SMS TUFLOW bathymetry

The input file is an ASCII file containing bathymetry data in SMS/TuFlow format (.2dm). The file first defines the elements with associated node numbers, next the location of the individual node points.

- The mesh is created using a geographical coordinate system (LONG/LAT). The (X,Y) coordinate unit is specified as being degrees and the z-values as being defined in US survey feet.

Note that the bathymetry values in this instance should be multiplied by -1 before applying the mesh in MIKE Zero.



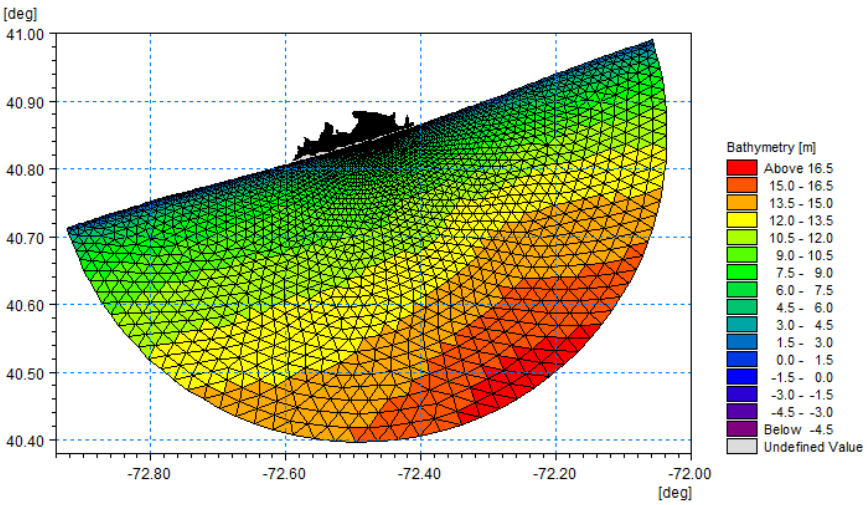


Figure 9.5 Mesh conversion of SMS bathymetry data
Upper: start of input bathymetry data in ASCII file
Lower: output mesh

9.9.5 Dfsu result file

The input file is a dfsu result file from a MIKE 21 FM simulation. The original bathymetry information is extracted from the static item along with information about code values.

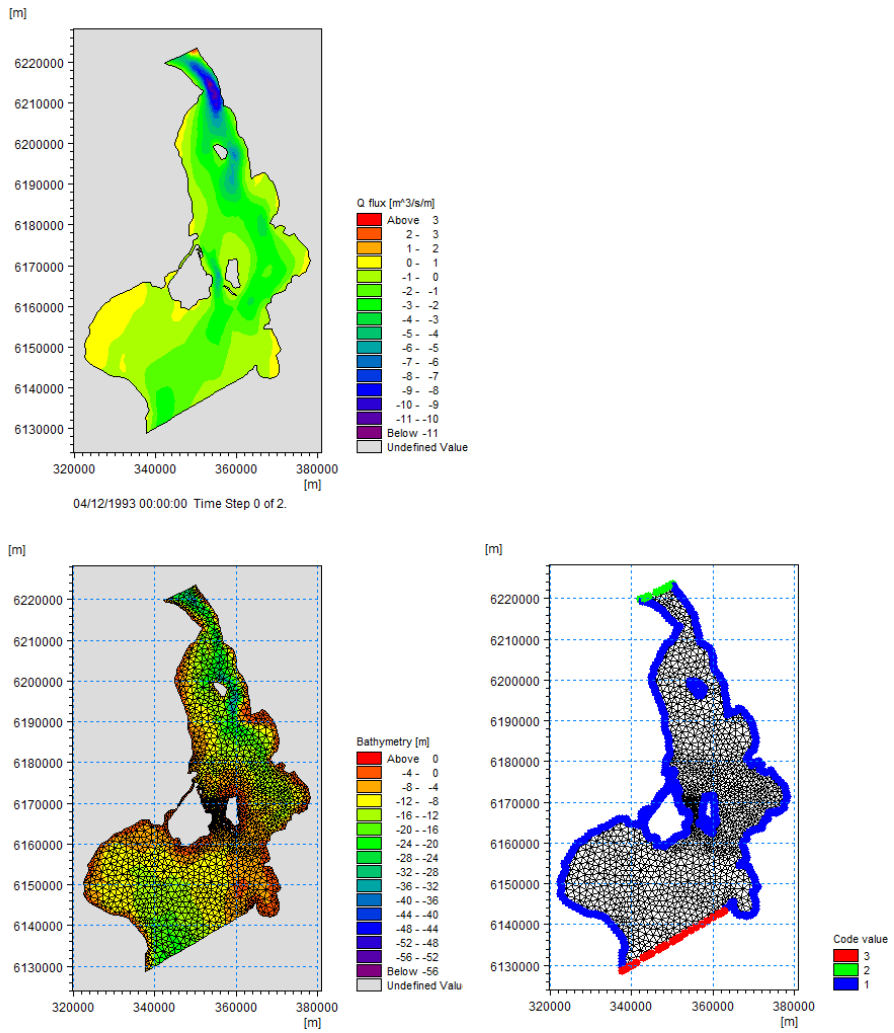


Figure 9.6 Extraction of mesh from dfsu result file
Upper: Input dfsu result file
Lower: Output mesh and related code values

MZ TOOLS: GIS

The GIS Tools in the MZ Toolbox include the facilities:

- GIS - Grd2Mike
- GIS - Mike2Grd
- GIS - Mike2Shp
- GIS - Mike2Txt
- GIS - Shp2xyz

10 GIS - Grd2Mike

This tool has been implemented to transform ESRI's Ascii Grid files (*.asc) into MIKE21 grid data files (.dfs2).

An example of ESRI's ArcView Grid Ascii format is shown below.

```
ncols         44
nrows         78
xllcorner     306219.10194
yllcorner     6054417.99390
cellsize      150.00000
NODATA_value  -9999
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 -0.2000 9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
```

Figure 10.1 Example of input asc file. Only part of file (the top) is shown)

The tool assumes that the levels and grid spacing units are in metres.

10.1 Input

On this page the ArcView Ascii Grid file (*.asc format) where data should be extracted from is selected. To select the file either type the full path or browse for the file.

NOTE: The tool assumes that the levels and grid spacing units are in metres.

10.2 Data Specification

On this page the map projection be used is specified.

10.3 Output Specification

On this page the name and title of the Output 2D Data File is specified. To specify the file either type the full path or browse for the location.

10.4 Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either ***Finish*** the set-up, which will transfer your set-up definition to the current toolbox file, or ***Execute*** the set-up.

11 GIS - Mike2Grd

This tool has been implemented to transform MIKE 21 grid data files into ASCII data in ESRI's Grid format.

Note: Import of raw ASCII data as a layer using '*Add Data*' in ArcGIS may result in a misleading color ramp. Optionally use instead ArcGIS Toolbox->Conversion->To Raster->ASCII to Raster.

Note: This tool only works correctly with a grid-orientation of 0 deg.N and a grid spacing that is the same in both X- and Y-direction.

An example of ESRI's ArcView Grid Ascii format is shown below.

```
ncols         44
nrows         78
xllcorner     306219.10194
yllcorner     6054417.99390
cellsize      150.00000
NODATA_value  -9999
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 -0.2000 9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
9.0000 9.0000 9.0000 9.0000 9.0000 9.0000 9.0000
```

Figure 11.1 Example of resulting asc file. Only part of file (the top) is shown)

11.1 Input

On this page the MIKE 21 (*.dt2 or *.dfs2 formats) file where data should be extracted from is selected. To select the file either type the full path or browse for the file.

Once the input file has been specified, the program requires the user to define only one item which will be transformed.

Note: This tool only works correctly with a grid-orientation of 0 deg.N and a grid spacing that is the same in both X- and Y-direction.

11.2 Data Specification

On this page the Time Step(s) to extract data from is specified as well as which projection system to use. The possible systems are those specified in the utm.dat file

Beware that there will be generated one output file for each selected Time Step. Each file will be postfixed with the Time Step number, e.g. having specified the output file C:\output.asc and selected time step 1 and 112 will cause the generation of the two output files C:\output001.asc and C:\output112.asc.

11.3 Output Specification

On this page the name of the Output ArcView Ascii Grid File is specified. To specify the file either type the full path or browse for the location.

11.4 Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either ***Finish*** the set-up, which will transfer your set-up definition to the current toolbox file, or ***Execute*** the set-up.

12 GIS - Mike2Shp

This tool has been implemented to transform dfs files into ESRI's ArcView shapefile format files.

12.1 Input

On this page you specify the dfs file and item(s) from where data should be extracted. The input file may be one of three types

- dfs2 file
- dfsu file
- mesh file

Once the input file and item(s) has been specified, the selected items is shown for reference.

NOTE: The field name for each item in the resulting shape file will be created using maximum 10 characters. Special characters such as '.' and ',' in the item name will be omitted.

NOTE: A dfs2 or dfsu file must be defined by an equidistant calendar axis.

NOTE: In case the 3D dfsu file is defined by a sigma-Z level grid, each layer must be extracted beforehand, e.g. by Data Extraction FM, and converted individually to shape files.

12.2 Data Specification

On this page you specify the time step(s) to extract data from. In case the input file is a dfsu file in 3D you have to specify the layer number(s) as well.

The applied domain area and map projection is shown for reference.

Note that each combination of selected time step (and layer number) will be handled individually and the output will be saved in separate output files, postfixed with the Time Step number (and layer number).

12.3 Output Specification

On this page you specify the global filename of the output files.

Output of one time step will produce the following files:

- A shape file (.shp), also called the main file, containing the geometric objects (points or polygons)

- A projection file (.prj) specifying the geographic coordinate system of the shape file
- An index file (.shx), containing file positions of the records in the main file
- A database file, (.dbf), containing the actual data values extracted from the data of the input data file

The output files will be postfixed with the Time Step number (and layer number)

The actual number of output files depends on type of input file and the selected number of time steps (and layers).

12.3.1 From dfs2 file

For each time step the following files will be generated:

1. A shape file (and accompanying index, projection and DBF files) describing the cell values by a point feature layer. The points are positioned in the centre of each grid cell.
   (naming convention: *globalname*_nodes_*timestepnumber*.*)

2. A shape file (and accompanying index, projection and DBF files) describing the cell values by a polygon feature layer. The polygons represent the extension of each grid cell.
   (naming convention: *globalname*_cells_*timestepnumber*.*)

12.3.2 From 2D dfsu file

For a 2D dfsu file the node values and element values in the dfsu file are saved in separate shape files.

The following files will be generated:

1. One shape file (and accompanying index, projection and DBF files) containing the bathymetry of the nodes by a point feature layer.
   (naming convention: *globalname*_Nodes.*)

2. A shape file (and accompanying index, projection and DBF files) for each selected time step containing the selected element values by a polygon feature layer.
   (naming convention: *globalname*_elements_*timestepnumber*.*)

12.3.3 From 3D dfsu file

For a 3D dfsu file the node values and element values in the dfsu file are saved in separate shape files.

The following files will be generated:

1. A shape file (and accompanying index, projection and DBF files) for each selected layer containing the level of the nodes by a point feature layer. (naming convention: *globalname*_Nodes_Layer*layernumber*.*)

2. A shape file (and accompanying index, projection and DBF files) for each selected time step and each selected layer containing the selected element values by a polygon feature layer. (naming convention: *globalname*_elements_Layer*layernumber_timestepnumber*.*)

12.3.4 From mesh file

For a mesh file the node values and element values in the dfsu file are saved in separate shape files.

The following files will be generated:

1. One shape file (and accompanying index, projection and DBF files) containing the bathymetry of the node points in the mesh by a point feature layer. (naming convention: *globalname*_Nodes.*)

2. One shape file (and accompanying index, projection and DBF files) containing the bathymetry of the elements by a polygon feature layer. *(naming convention: globalname*_Elements.*)

12.4 Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either ***Finish*** the set-up, which will transfer your set-up definition to the current toolbox file, or ***Execute*** the set-up.

13 GIS - Mike2Txt

This tool has been implemented to transform MIKE 21 grid data files into ESRI's ArcView txt format.

An example of ESRI's ArcView txt format is shown below.

```
x,y,Depth (m) rel to LAT
306219.093750,6054418.000000,9.000000
306310.531250,6054299.000000,4.500000
306401.968750,6054180.000000,-4.000000
306493.375000,6054061.000000,-10.000000
306584.812500,6053942.500000,-13.400000
306676.218750,6053823.500000,-15.400000
306767.656250,6053704.500000,-20.000000
306859.093750,6053585.500000,-20.500000
306950.500000,6053466.500000,-21.000000
307041.937500,6053347.500000,-21.400000
307133.375000,6053229.000000,-22.000000
307224.781250,6053110.000000,-21.900000
307316.218750,6052991.000000,-21.299999
307407.625000,6052872.000000,-21.000000
307499.062500,6052753.000000,-20.500000
307590.500000,6052634.000000,-20.000000
307681.906250,6052515.500000,-19.799999
307773.343750,6052396.500000,-18.799999
307864.781250,6052277.500000,-17.799999
```

Figure 13.1 Example of resulting txt file (derived from Donegal bay bathymetry). Only part of file (the top) is shown.

13.1 Input

On this page the MIKE 21 (*.dt2 or *.dfs2 formats) file where data should be extracted from is selected. To select the file either type the full path or browse for the file.

Once the input file has been specified, the program requires the user to define the items which will be transformed.

13.2 Data Specification

On this page the Time Step(s) and Area to extract data from is specified as well as which projection system to use. The possible systems are those specified in the utm.dat file

Beware that there will be generated one output file for each selected Time Step. Each file will be postfixed with the Time Step number, e.g. having specified the output file C:\output.txt and selected time step 1 and 112 will cause the generation of the two output files C:\output001.txt and C:\output112.txt.

13.3 Output Specification

On this page the name of the Output ArcView Ascii Text File is specified. To specify the file either type the full path or browse for the location.

13.4 Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either ***Finish*** the set-up, which will transfer your set-up definition to the current toolbox file, or ***Execute*** the set-up.

14 GIS - Shp2xyz

This tool has been implemented to extract spatial feature data from a shape file and store the values in an ascii file as xyz data.

14.1 Input File

On this page you specify the shape file from where the spatial data should be extracted.

NOTE: A “shape file" consists of a collection of files with a common filename prefix, stored in the same directory. As a minimum, the three mandatory files with extensions .shp, .shx, and .dbf should be available.

14.2 Data Specification

On this page you specify the type of geographical feature data you want to extract. You can select between three types of shape features:

- Point
- Polyline
- Polygon

Per default the z-values are derived from the chosen geographical feature. You have the option to specify the name of a field in the shape file that contains the values to be saved in the z-column of the output data.

14.3 Output Specification

On this page you specify the name of the output file.

The output will depend on the selected type of feature.

14.3.1 Point feature

The output file will contain three columns:

1. X-coordinate
2. Y-coordinate
3. Z-level

If the feature data contains a z-value (type PointZ), this value is written to the third column, otherwise the third column contains 0.
If a optional z-field has been chosen, the z-value is derived from this.

14.3.2 Polyline feature

Each polyline will be represented by the position of the connection points and the connectivity information. The output file will contain four columns:

1. X-coordinate
2. Y-coordinate
3. Z-level
4. Connectivity information (1: connected to next point, 0: end of line)

If the feature data contains a z-value (type PolylineZ), this value is written to the third column, otherwise the third column contains 0.
If a optional z-field has been chosen, the z-value is derived from this.

14.3.3 Polygon feature

Each polygon will be represented by the position of the connection points and the connectivity information. The output file will contain four columns:

1. X-coordinate
2. Y-coordinate
3. Z-level
4. Connectivity information (1: connected to next point, 0: end of line)

If the feature data contains a z-value (type PolygonZ), this value is written to the third column, otherwise the third column contains 0.
If a optional z-field has been chosen, the z-value is derived from this.

14.4 Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either ***Finish*** the set-up, which will transfer your set-up definition to the current toolbox file, or ***Execute*** the set-up.

14.5 Example

14.5.1 Purpose

For the purpose of investigating flooding in a small town the municipality has provided shape files describing various features such as houses, roads and lights.

In order to include the information in the domain description (mesh generation) the data needs to be extracted into ascii xyz files that can be read by the Mesh Generator.

14.5.2 Input data

The input data consists of three shape files:

- Houses - described by polygons
- Roads - described by polylines
- Lights - described by points
  The field 'ZValue' describes the height of the lamp post

Figure 14.1 shows how the data can be displayed in ArcMap.

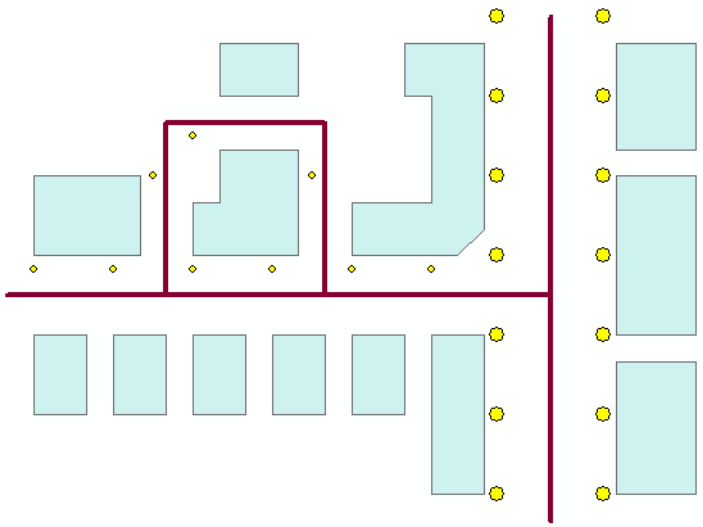


Figure 14.1 Shape file data displayed in ArcMap

14.5.3 Output data

The Shp2Xyz tool is used to extract the geographic position of the various shape file data into xyz files. The resulting xyz data files can then be imported into the Mesh Generator as scatter data or boundary arcs.

Figure 14.2 shows how the imported data may look in Mesh Generator.

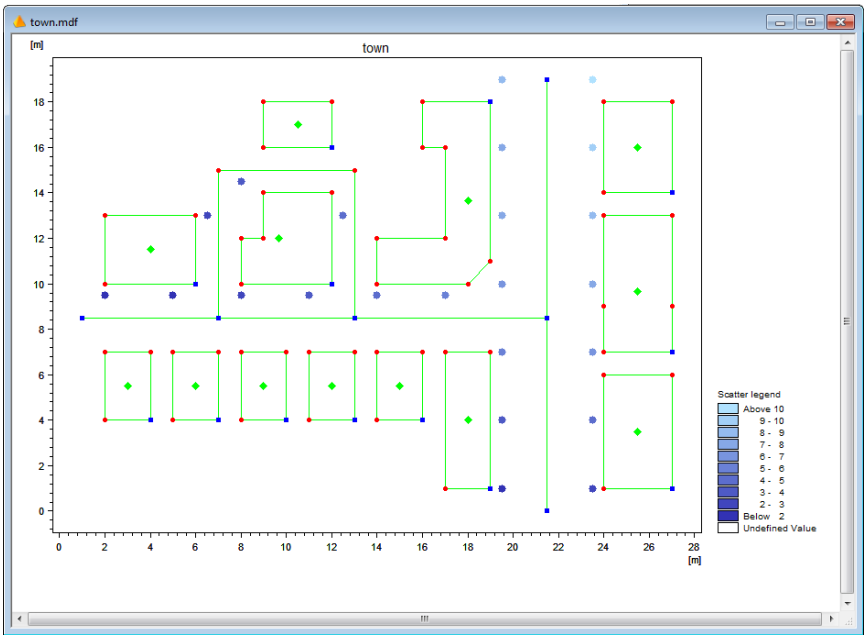


Figure 14.2 Resulting display of data in Mesh Generator
Scatter data displays internal bathymetry value of lamp posts

MZ TOOLS: HYDROLOGY

The HYDROLOGY Tools in the MZ Toolbox include the facilities:

- Maximum Rainfall
- Mean Area Weighting Algoritm
- Normalize Rainfall
- Runoff Analysis

15 Maximum Rainfall

This tool extracts maximum rainfall for a number of periods from a range of historical rainfall events.

15.1 Input File

Here the time series file for extracting maximum rainfall is selected. The time series file can either have an equidistant, or an "almost" equidistant calendar axis. The "almost" equidistant calendar axis is basically non-equidistant, but has equidistant time steps during rain events.

15.2 Parameters

- Extraction time step
- Event periods
- Extraction periods
- Extract values

15.3 Output File

Here the output time series file must be specified, full path and name. The file will contain the output from this tool.

15.4 Summary

Summary, or setup status: review parameter spec's. And if you want change any settings, go back; if you are satisfied with the settings click Execute, or just Finish to save the setup for later.

16 Mean Area Weighting Algoritm

MIKE Zero tool for mean averaging and weighting.

16.1 Weighted Average dfs File Input

Here you select input file and item in file.

16.2 Weighted Average Weight Combination

Here you select weighted average weight combination.

16.3 Temporal Distribution dfs Input File

Here you select temporal distribution input file and item in file.

16.4 Temporal Distribution Weight Combinations

Here you select temporal distribution weight combinations.

16.5 Output Specification

Here you select full path and name for the output data file.

16.6 Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

17 Normalize Rainfall

This tool normalises rainfall data measured eg every 10 min, 30 min or 1 hour with rainfall data measured over eg 24 hours. The starting time in both files must match, eg 9:00 AM, both, or 00:00, both, and both files must begin with a "delete value".

17.1 Normalize Rainfall Input Files

Number of normalisations; for each of which

- File with data to be normalised
- File with normalization data

17.2 Normalize Rainfall Output File

Here the location and name of the output time series must be entered. The file will contain the output divided over different items.

17.3 Normalize Rainfall Summary

Summary, or setup status: review parameter spec's. And if you want change any settings, go back; if you are satisfied with the settings click Execute, or just Finish to save the setup for later.

18 Runoff Analysis

Effective rainfall analysis: this tool generates as output a meta file from two time series files, one with average rainfall data and one with observed discharge data.

18.1 Input Files

Here you must select the average rainfall data file and the observed discharge data file with the data to be analysed.

18.2 Accumulation

Parameters:

- Catchment area, E, f, fsa, Rsa.
- Start, End, Duration, Base flow, Total rainfall, Total effective.

18.3 Plot

Plot page.

18.4 Output File

Here you must select a (full path and) file name for the output meta file containing the generated plot.

18.5 Summary

Summary, or setup status: review parameter spec's. And if you want change any settings, go back; if you are satisfied with the settings click Execute, or just Finish to save the setup for later.

MZ TOOLS: STATISTICS

The Statistics Tools in the MZ Toolbox include the facility:

- Statistics for DFS Data Files

19 Statistics for DFS Data Files

You can use this tool to calculate statistical values on time series (.dfs0), line series (.dfs1), matrix series (.dfs2) or volume series (.dfs3).

The output will be written to either an ascii file (in the time series case) or a standard dfs file.

19.1 Dimension

Here you select which type of file (Dimension) you want to do statistics on.

19.1.1 Time series

If you choose to do statistics on a time series file, the output will be written to an *.out file in the same directory as your input data. The name of the output file will be the setup name specified on the previous page. You will get Minimum, Maximum, Median, Mean, Root mean square, Standard deviation and Skewness for a time series file.

19.1.2 Line series

If you choose to do statistics on a line series file, you may choose up to six statistical parameters as output. You will choose these later on the Statistics dialogue.

19.1.3 Matrix series

If you choose to do statistics on a grid series file, you will have to choose one of six statistical parameters as output due to space limitations. You will choose these later on the Statistics dialogue.

19.1.4 Volume series

If you choose to do statistics on a line, matrix or volume series file, you will have to choose one of six statistical parameters as output due to space limitations. You will choose these later on the Statistics dialogue.

19.2 Input

On this page you select your input file. When you choose the file, you also choose the items that you want to do statistics on.

You can choose up to 10 items.

19.2.1 Derived items

If the data file is 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 or more of the following derived items: Surface Elevation, Current Speed, and Vx and Vy (current velocities). These items are calculated from h, p and q by the program as described below.

Figure 19.1 shows the definition of the grid items with respect to the position in the grid. The saved h, p and q values in the file are positioned in the grid as shown to the left, the derived items are all positioned in the center point of the grid element as shown to the right.

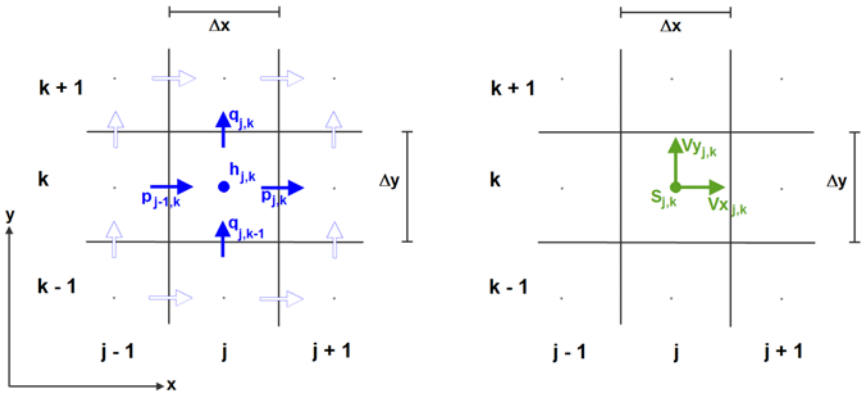


Figure 19.1 Position in grid for items.
Left: Water depth (h) and flux components (p,q)
Right: Derived current speed (S) and velocity components (Vx,Vy)

The equations used for the calculation of the derived items are as follows:

$$W_{j,k} = h_{j,k} + z_{j,k} \tag{19.1}$$

$$Vx_{j,k} = \frac{p_{j,k}}{h_{j+1,k} + h_{j,k}} + \frac{p_{j-1,k}}{h_{j,k} + h_{j-1,k}} \tag{19.2}$$

$$Vy_{j,k} = \frac{q_{j,k}}{h_{j,k+1} + h_{j,k}} + \frac{q_{j,k-1}}{h_{j,k} + h_{j,k-1}} \tag{19.3}$$

$$S_{j,k} = \sqrt{(Vx_{j,k})^2 + (Vy_{j,k})^2} \tag{19.4}$$

where *zj,k* is the bathymetry value stored in the static item.

19.3 Subseries Selection

This page allows you to select a specific temporal subset of the input data you want to include in your statistics calculation. Statistics are calculated over time, not over space, so selecting the subseries will influence the results.

Typically you probably want to process the whole period.

19.4 Calculation Parameters

As default the statistics are based directly on the time series values themselves, i.e. relative to zero.

However, for a time series file it is possible to give any constant reference level which is substracted from the time series values before the statistics are calculated.

Another possibility is to give the reference level as a time series covering the whole period. In this way it is possible to compute statistics on e.g. the difference between a simulated and a measured time series in water levels, i.e. on the error. In addition you can specify a lower and upper reference level limit. In this case only time steps for which the reference level is inside this range gives a contribution to the statistics. Thus, if only a minimum reference level is given and the time series are again water levels, statistics on the error during periods of high level can be calculated.

Note: Delete values in the file are not considered when calculating statistics. E.g. you can have an exceedance percentage of 100 percent, even if the grid point is not flooded at all time steps.

19.5 1D Line

Here you select the part of the line for which you want to calculate statistics.

19.6 2D Area

Here you select the subarea in which you want to calculate statistics.

19.7 3D Area

Here you select the subarea in which you want to calculate statistics.

19.8 Statistics

This is where you choose what type of statistics you want to calculate on your 1D, 2D or 3D data file. Due to memory constrains, you will only be able to

choose one type of statistics for your output file, when you are generating statistics for matrix or volume series.

The following statistical parameters can be calculated:

- Maximum value
  Maximum value of all data
- Minimum value
  Minimum value of all data
- Mean value
  Mean value of all data, calculated by Eq. (19.5)
- Standard deviation
  Standard deviation of all data, calculated by Eq. (19.7)
- Median value
  Value below which 50% of the data fall
- Exceedance percentage
  Percentage of exceedance given a threshold value

Note: Missing values may be excluded from the calculation. I.e. if a value is missing 50 pct of the time and otherwise constantly has a value of 5, then with a threshold of 4 the exceedance percentage will be 100% if missing values are excluded. Otherwise the exceedance percentage will be 50%.

19.9 Output Specification

On this page you specify the name and location of the 1D, 2D or 3D output data file, i.e. the file that will contain the resulting statistical values.

You can also specify a data file title.

19.10 Status

The Status page presents you a summary of all the input parameters you have specified. You should inspect that the parameters are correct and if not go back and change them.

From the page you can either *Finish* the setup which will transfer your setup definition to the current toolbox file or execute the setup.

19.11 Scientific Background

The statistical parameters are defined and calculated as follows:

The Median value: Value below which 50% of the data fall

The Mean value:

$$\bar{X} = \frac{\sum X}{N} \tag{19.5}$$

The Root Mean Square:

$$RMS = \sqrt{\frac{\sum X^2}{N}} \tag{19.6}$$

The Standard Deviation:

$$S = \sqrt{\frac{\sum (X-\bar{X})^2}{N}} \tag{19.7}$$

The Skewness:

$$Skewness = \frac{\bar{X} - mode}{S} \tag{19.8}$$

Where "mode" is the most common value.

MZ TOOLS: TIME SERIES

The TIME SERIES Tools in the MZ Toolbox include the facilities:

- Interpolation of Time Series
- One Year Summary
- Pre-Processing Temporal Data
- Sorting Temporal Data
- Time Series Batch Conversion

20 Interpolation of Time Series

This simple tool is used for changing the number of data points in a time series file. This is done by interpolating intermediate values. This gives a resulting time series with a modified time step (making the file shorter or longer, depending of the user's needs).

20.1 Input

Here one specifies the full path name of the input file, i.e. the file holding the time series data to be interpolated. The input file must be a dfs0 file. One must either type the full path name or browse for the file.

20.2 Interpolation Method and Subseries Selection

Here one selects the interpolation method (linear or piecewise cubic) and output time discretizsation: Start time, end time and length of time step for the interpolated time series output file. Upon execution, the tool writes the output time series with the requested time discretization.

20.2.1 Interpolation method

Interpolation can be performed on any time series type (e.g. instantaneous, accumulated, step accumulated, mean step accumulated, etc.), but the utility was developed having mainly in mind instantaneous time series data.

Figure 20.1 outlines the parameters needed for interpolating the value in point A.

The **linear interpolation** basically takes two consecutive points and draws a straight line between them. Furthermore, this procedure is performed to each pair of consecutive points within the output period as shown below. This set of straight lines contains all interpolated values for the output time series. The formula for linear interpolation is:

$$y = y_1 + (y_2 - y_1)\frac{x - x_1}{x_2 - x_1} \tag{20.1}$$

The **piecewise cubic interpolation** takes four consecutive points, and perform an interpolation by a cubic polynomial between the two central points (out of the four consecutive points). By performing a cubic interpolation, it is possible to smooth the interpolated function with respect to the linear interpolation. Drawing a line passing through the four interpolated points you see a curve following the trend of the data variation.

The formulas for piecewise cubic interpolation are:

$$
\begin{aligned}
s_0 &= (y_2 - y_0)\frac{x_2 - x_1}{x_2 - x_0} \\
s_1 &= (y_3 - y_1)\frac{x_2 - x_1}{x_3 - x_1} \\
a &= s_1 - 2y_2 + s_0 + 2y_1 \\
b &= y_2 - a - s_0 - y_1 \\
y &= y_1 + x(s_0 + x(b + xa))
\end{aligned}
\tag{20.2}
$$

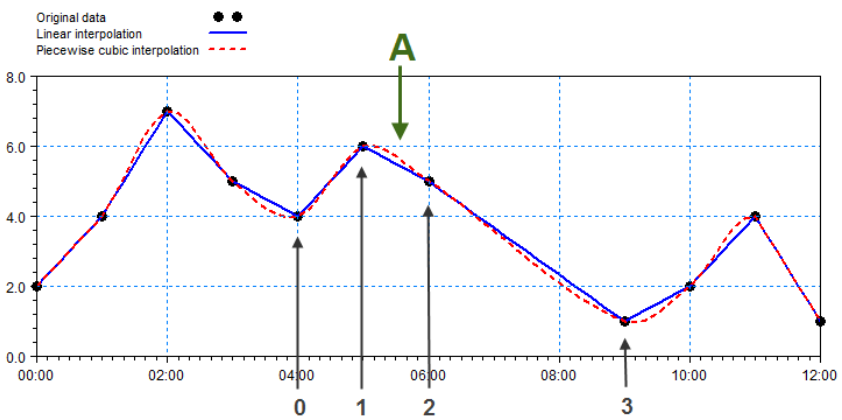


Figure 20.1 Definition of interpolation components
Solid line: result of linear interpolation
Dotted line: result of piecewise cubic interpolation

20.3 Output Specification

Here one specifies the full path, file name, and optionally the title of the output file, i.e. the file that will contain the interpolated time series. The output file is a dfs0 file.

20.4 Status - Overview

The overview page presents a summary of all the input parameters specified earlier. One should inspect the parameters to check they are correct, and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer the setup definition to the current toolbox file, or execute the set-up.

21 One Year Summary

This tool takes a precipitation or discharge time series file as input. From that a summary for each day, month and year is created.

21.1 Welcome

One year summary.

21.2 Input File

Here you select an accumulated precipitation or discharge time series file. Enter start year, months and hour. Then select if input data is precipitation or discharge. Finally, press 'View one year summary'.

21.3 Summary

Summary, or setup status: review parameter spec's. And if you want change any settings, go back; if you are satisfied with the settings click Execute, or just Finish to save the setup for later.

22 Pre-Processing Temporal Data

This tool takes as input a set of geo-referenced time series of the same scalar type e.g. rainfall, discharge, water level etc. and combines these into a area wide time series or a spatially and temporally varied data set.

22.1 Setup Name

Supply a name for the set-up so that it may be revisited.

22.2 Station Locations

Define the stations (time series) and their locations. The station locations may be loaded from a point shape file or a x, y name file. The process is in two steps the first steps involves browsing for the file and the second actually importing the station locations by clicking the 'Import stations' button. If a projection is associated with the shape file (a *.prj file is located next to the shape file) this is used to populate the projection field. If no *.prj file is present or the user has selected an xyz file then the projection is to be supplied. Please note that all subsequent geo-referenced data must use the same projection.

For each station a name is supplied if it has not been provided from the shape file along with a dfs0 file holding the data. The tool does not modify the data in the input dfs0 files and thus the data from the various input files must be compatible i.e. the same type of data and the same unit e.g. rainfall all in millimetres.

Once the table has been filled out the next button may be clicked. This action will validate the data more specifically

- dfs0 files are checked with regards to existence
- the existence of a common time span of the time series
- the locations and the stations are valid data

Please note that the x,y name file has the extension *.xyz and the values are space delimited.

22.3 Interpolation Method

The method for determining the interpolated value in the resulting output is specified here. There are two options

- Thiessen Polygons: Value assigned corresponding to nearest station
- Inverse distance weighting with user defined power parameter

The power parameter must be a positive value. A value of two corresponds to the standard distance measure.

22.4 Output Type

The output may be of two types

- Spatially distributed in either structured grid or an unstructured mesh
- A weighted time series per catchment

When selecting the former a mesh or a dfs2 bathymetry file must be provided. The file will define the grid or the mesh used for the output i.e. the file must be created prior to using the tool. Note that the file must use the same projection as selected for the stations. In the case of the weighted time series a polygon shape file must be provided. This shape file may contain multiple polygons and a weighted time series will be produced for every non empty polygon (i.e. containing at least one station).

22.5 Spatially Distributed Output

The output file for the spatially distributed output is supplied here. Note that the file extension must correspond to the type of input bathymetry that has been supplied e.g. a mesh file will correspond to a dfsu file and an input grid file will require a dfs2 file name. Along with the file name an item name must be supplied.

22.6 Weighted Average Output

For the weighted time series case an output file (dfs0 extension) along with an item name must be supplied per polygon. Note that the output files will be over written. The file names are validated when clicking next.

22.7 Temporal Extent

The temporal extent must be supplied along with the storing frequency of the output. The temporal extent must be compatible with the common time period available within the input time series.

22.8 Geographical Overview

The overview presents the locations of the stations along with the mesh, grid or polygons depending on the user selections of method. Further if Thiessen polygon calculation methods have been utilized these will also be shown.

22.9 Time Series Weights

The weighting matrix is displayed. The values may be altered or the matrix may be regenerated by activating the ‘Calculate weights’. In case of the latter a row in the matrix may be protected by use of the ‘Protect’ column. The latter ensures that user defined weights are not overridden.

22.10 Overview

Setup status: review the parameters. If you want to change any settings, go back; if you are satisfied with the settings click Execute, or just Finish to save the setup for later.

22.11 Scientific Background

The following sections give the scientific background for the interpolation methods used and for the weighting functionality.

22.11.1 Thiessen Polygons

The Thiessen polygon method assigns the value of any point to the measured value from the nearest measuring station. If this station does not contain values at the specific time step then the nearest measuring with actual data will be used instead.

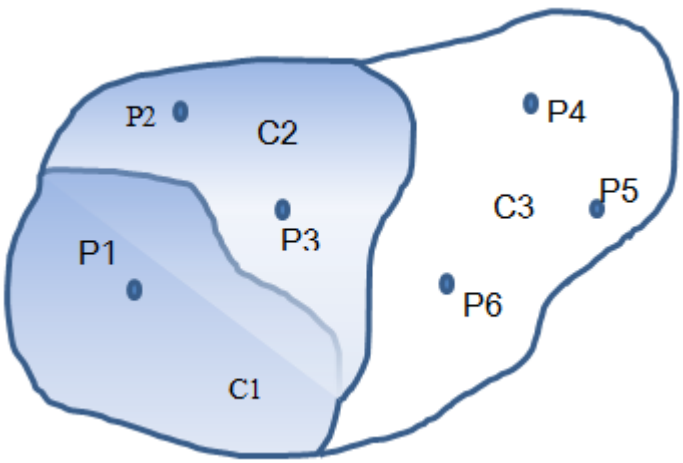


Figure 22.1 Three catchments (polygons) with a number of stations within each.

The calculation of Thiessen polygon is confined to only take place within the individual catchment. Thus in the above figure the average value within catchment C3 is only affected by the values at the stations P4, P5 and P6.

The area of interest is the non-shaded part. The Thiessen polygons for catchment C3 are illustrated below with the area of influence of station point P4 as further been shaded.

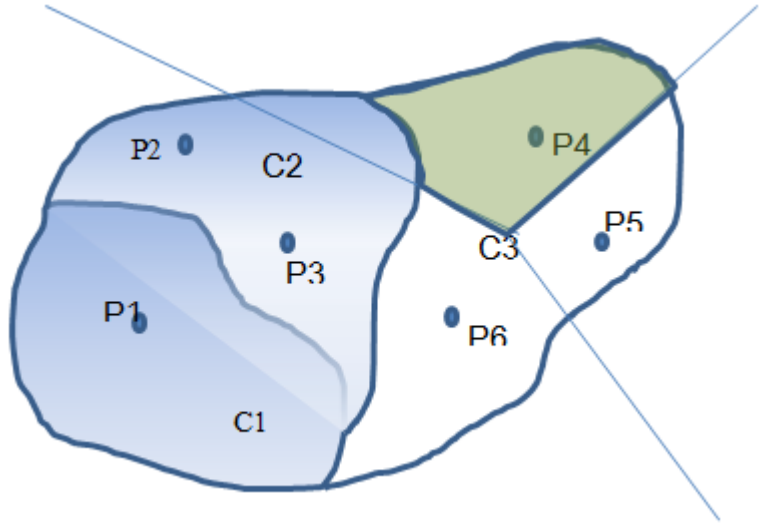


Figure 22.2 The Thiessen polygon associated with point P4 within Catchment C3.

The value *v* at any point may be written as

$$v(x,y) = \frac{1}{\sum_{j=1}^{N} w_j(x,y)} \sum_{i=1}^{N} w_i(x,y) v_i \qquad (22.1)$$

where *N* is the number of stations. The weighting values in the case of Thiessen polygons are unity or zero where unity is used for the nearest station and zero for the remaining.

To obtain an average value for the whole catchment the above equation is integrated over the whole catchment and divided by the catchment area i.e.

$$\bar{v} = \int_A \left(\frac{1}{\sum_{j=1}^{N} w_j(x,y)} \sum_{i=1}^{N} w_i(x,y) v_i \right) dA = \frac{1}{A} \sum_{i=1}^{N} A_i v_i = \sum_{i=1}^{N} W_i v_i \qquad (22.2)$$

Thus the weights for calculating the mean area weighting value per catchment is given by

$$W_i = \frac{A_i}{A} \tag{22.3}$$

where the index *i* refers to the station.

22.11.2 Inverse distance weighting

The calculation of inverse distance values is based on the equation

$$v(x,y) = \frac{1}{\sum_{j=1}^{N} w_j(x,y)} \sum_{i=1}^{N} w_i(x,y) v_i \tag{22.4}$$

the weights are in this case defined by

$$w_i(x,y) = \frac{1}{|x - x_i|^p + |y - y_i|^p} \tag{22.5}$$

Where the index *i* refers to the station index and *p* is the power parameter taking on a positive value. The larger the power parameter value the more peaky the interpolated surface will be at the station values. Note that at the measurement locations the station value is always used.

The catchment wide weights used for the inverse distance method are

$$\bar{v} = \int_A \frac{1}{\sum_{j=1}^{N} w_j(x,y)} \left(\sum_{i=1}^{N} w_i(x,y) v_i \right) dA = \sum_{i=1}^{N} \frac{1}{A} \int_A \frac{w_i(x,y)}{\sum_{j=1}^{N} w_j(x,y)} dA\, v_i = \sum_{i=1}^{N} W_i v \tag{22.6}$$

where the catchment wide weight is then given by

$$W_i = \frac{1}{A} \int_A \frac{w_i(x,y)}{\sum_{j=1}^{N} w_j(x,y)} dA \tag{22.7}$$

the evaluation of the above is done through a numerical integration using evenly distributed grid.

22.12 Examples

22.12.1 Rainfall map

The purpose of this example is to illustrate how to generate a map with spatially and temporally distributed data. The generated map may be used in connection with modelling rainfall runoff in a mountain region.

Model input

The selected area is as displayed in Figure 22.3. Rainfall data has been collected from three stations in the region and syncronised to a common equidistant calender reference.

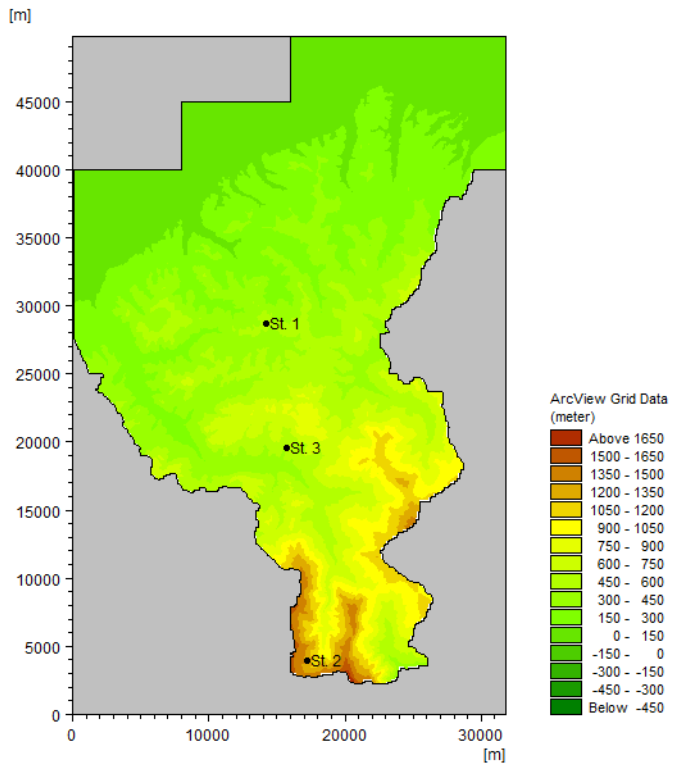


Figure 22.3 Selected area. Points indicate location of stations with rainfall data.

Parameters

Two setups are created using Thiessen polygons and Inverse distance method, respectively, to create the map. The output type is selected as spatially and temporally distributed data saved every hour during the first 50 hours of the rainfall event.

Results

The generated rainfall maps are shown in Figure 22.4.

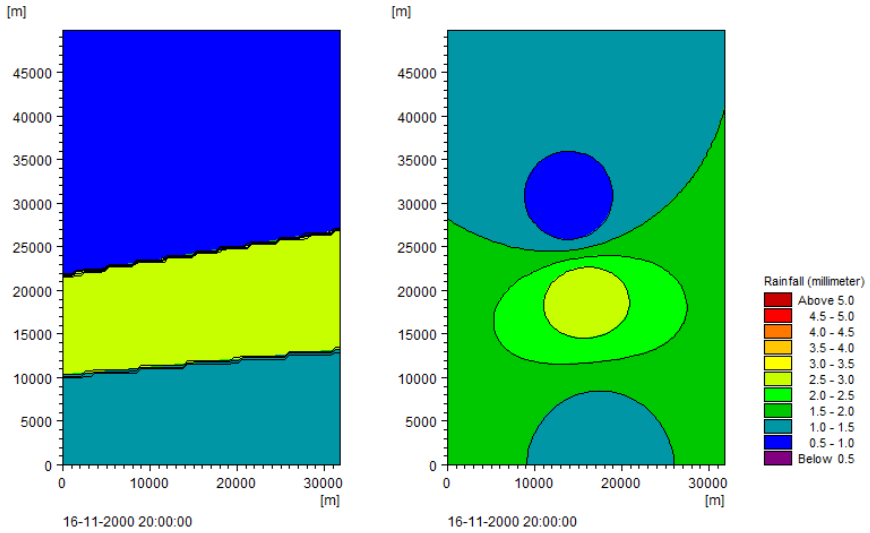


Figure 22.4 Example of resulting rainfall maps for a certain time step.
Left: Created using Thiessen polygons
Right: Created using Inverse distance method

23 Sorting Temporal Data

You use this tool to analyse time series of data and group them into various classes similar to scatter analysis.

23.1 Input Time Series File

On this page you select you input file and which items in the file that you want to use for the analysis.

You have the option to specify an additional item in the containing a duration for each event in the analysis. The resulting duration of each each class is then reflecting the number of seconds the class properties is taking place instead of the number of timesteps in the input file.

NOTE:
The time series file is assumed to have an equidistant time step interval.

23.2 Subseries Selection

On this page you specify the temporal subset of the input data you want to include in your analysis. The analyses are calculated over time, so selecting the subseries will influence the results.

Typically you probably want to process the whole period.

23.3 Class Delimiters

On this page you define the dimension of the classes that the time series data is to be sorted into.

Minimum value
The minimum value to consider for the analysis.

Maximum value
The maximum value to consider for the analysis.

No. Interval
The number of intervals defines the number of classes within each dimension.

23.4 Output Data Description

On this page you specify the name of the output file(s) you want to generate.

The default output file is a time series file containing the duration and representative value of each valid event class obtained by the analysis.

As an option you can choose to create an additional ascii file containing the results from the analysis written as scatter diagrams.

23.5 Overview

The overview page presents you a summary of all the input parameters you have specified. You should inspect that the parameters are correct and if not go back and change them.

From the page you can either *Finish* the setup which will transfer your setup definition to the current toolbox file or execute the setup.

23.6 Scientific Background

The data in the input file will be sorted as described below.

All values below the minimum value, *Min*, or above or equal to the maximum value, *Max*, will be considered as invalid.

The interval for a single class, *Dclass*, is defined by

$$\Delta class = (Max - Min)/(NoIntervals) \tag{23.1}$$

The minimum value within each class, *class(i)*, is found by

$$class(i) = Minimum + \Delta class \cdot (i - 1) \tag{23.2}$$

The event is deemed valid for the given class if

$$class(i) \leq value < class(i) + \Delta class \tag{23.3}$$

Note:
Each class is defined by a representative value, $class_{rep}(i)$, in the output:

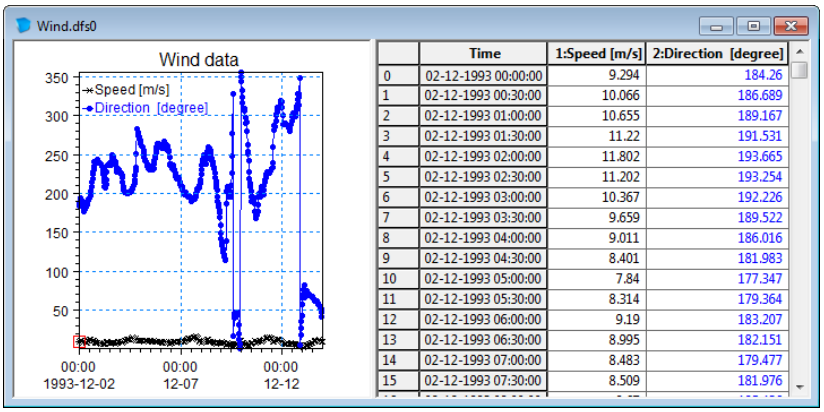
$$class_{rep}(i) = 0{,}5 \cdot (class(i) + class(i+1)) \tag{23.4}$$

23.7 Examples

23.7.1 Wind measurements

The following figures describe the process on how to sort data from wind measurements where all the events has the same duration.

The aim of the processing is to sort the data into classes.



	Time	1:Speed [m/s]	2:Direction [degree]
0	02-12-1993 00:00:00	9.294	184.26
1	02-12-1993 00:30:00	10.066	186.689
2	02-12-1993 01:00:00	10.655	189.167
3	02-12-1993 01:30:00	11.22	191.531
4	02-12-1993 02:00:00	11.802	193.665
5	02-12-1993 02:30:00	11.202	193.254
6	02-12-1993 03:00:00	10.367	192.226
7	02-12-1993 03:30:00	9.659	189.522
8	02-12-1993 04:00:00	9.011	186.016
9	02-12-1993 04:30:00	8.401	181.983
10	02-12-1993 05:00:00	7.84	177.347
11	02-12-1993 05:30:00	8.314	179.364
12	02-12-1993 06:00:00	9.19	183.207
13	02-12-1993 06:30:00	8.995	182.151
14	02-12-1993 07:00:00	8.483	179.477
15	02-12-1993 07:30:00	8.509	181.976

Figure 23.1 Input wind measurements

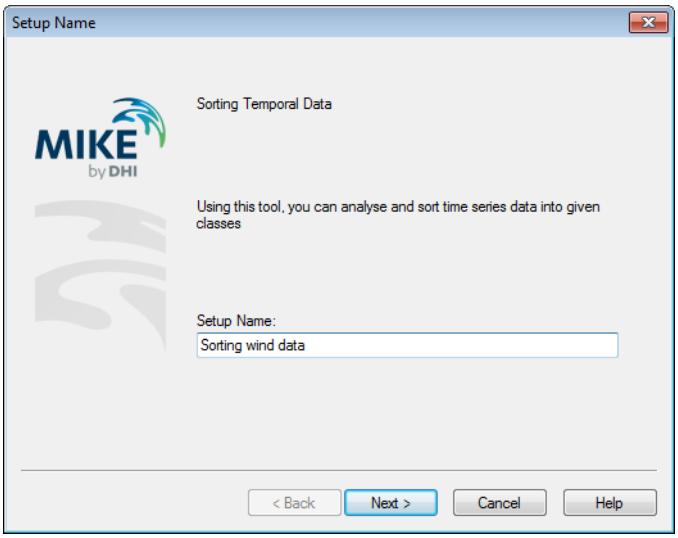


Figure 23.2 Specify name for setup

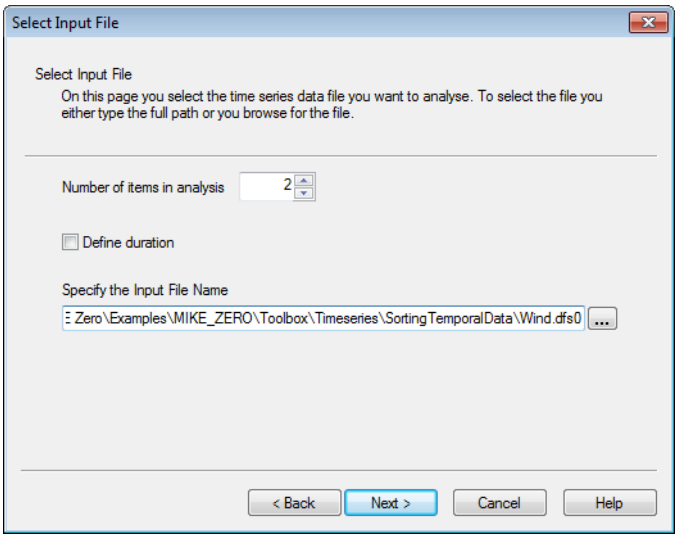


Figure 23.3 Define number of items in analysis, select input file and both input items

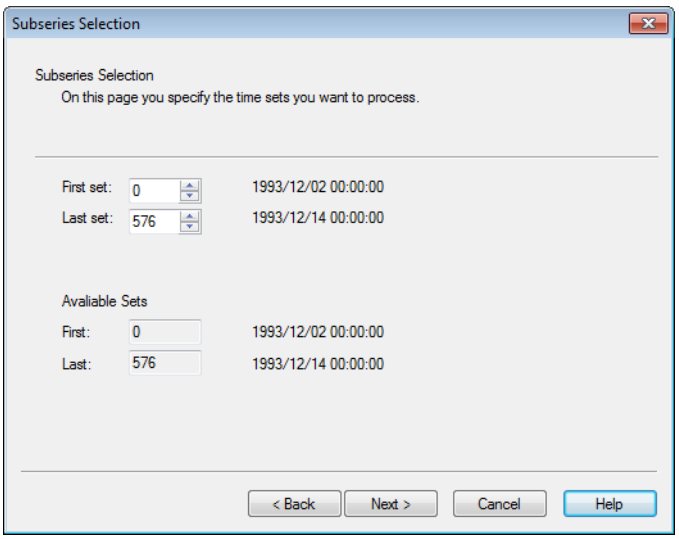


Figure 23.4 Specify subseries

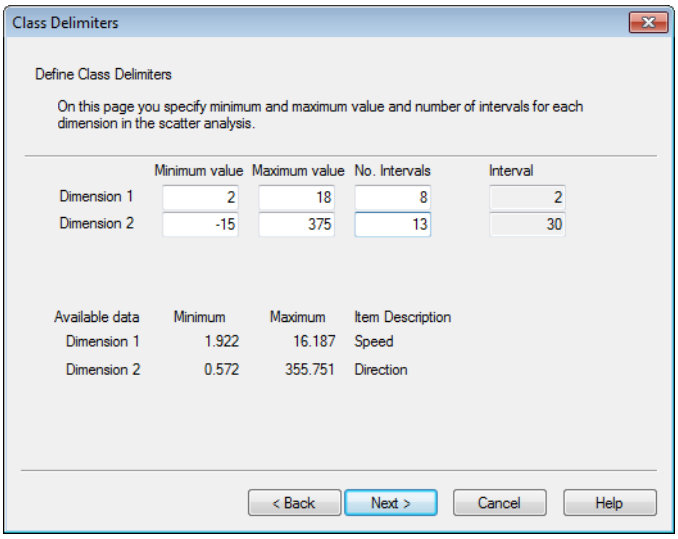


Figure 23.5 Specify class dimension and intervals for sorting

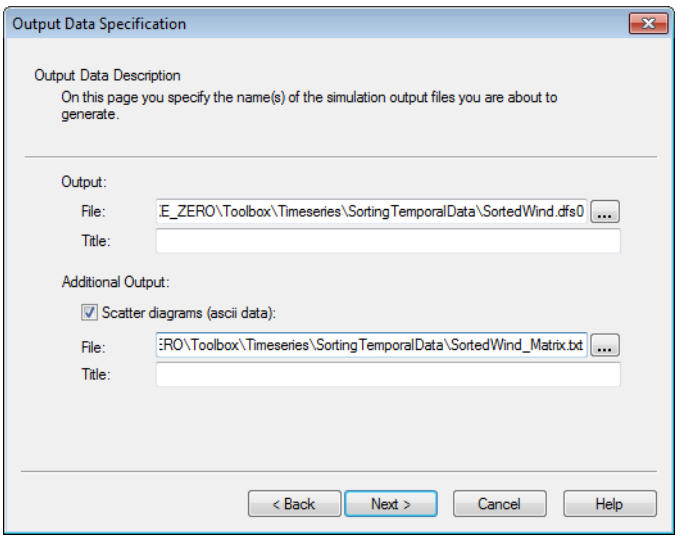
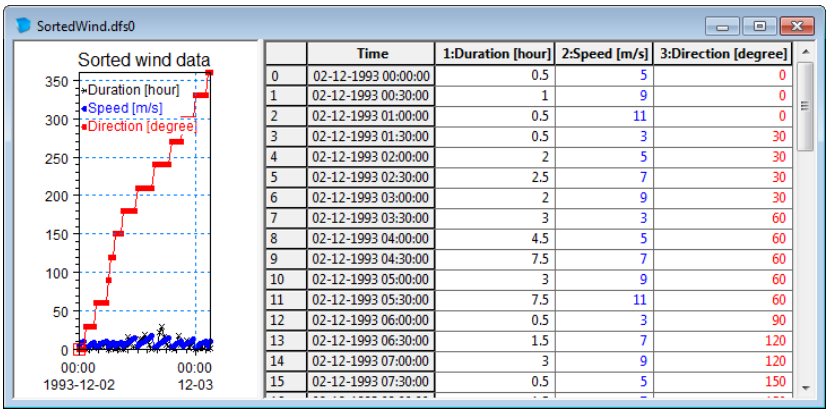


Figure 23.6 Specify output file names



	Time	1:Duration [hour]	2:Speed [m/s]	3:Direction [degree]
0	02-12-1993 00:00:00	0.5	5	0
1	02-12-1993 00:30:00	1	9	0
2	02-12-1993 01:00:00	0.5	11	0
3	02-12-1993 01:30:00	0.5	3	30
4	02-12-1993 02:00:00	2	5	30
5	02-12-1993 02:30:00	2.5	7	30
6	02-12-1993 03:00:00	2	9	30
7	02-12-1993 03:30:00	3	3	60
8	02-12-1993 04:00:00	4.5	5	60
9	02-12-1993 04:30:00	7.5	7	60
10	02-12-1993 05:00:00	3	9	60
11	02-12-1993 05:30:00	7.5	11	60
12	02-12-1993 06:00:00	0.5	3	90
13	02-12-1993 06:30:00	1.5	7	120
14	02-12-1993 07:00:00	3	9	120
15	02-12-1993 07:30:00	0.5	5	150

Figure 23.7 Resulting output time series file

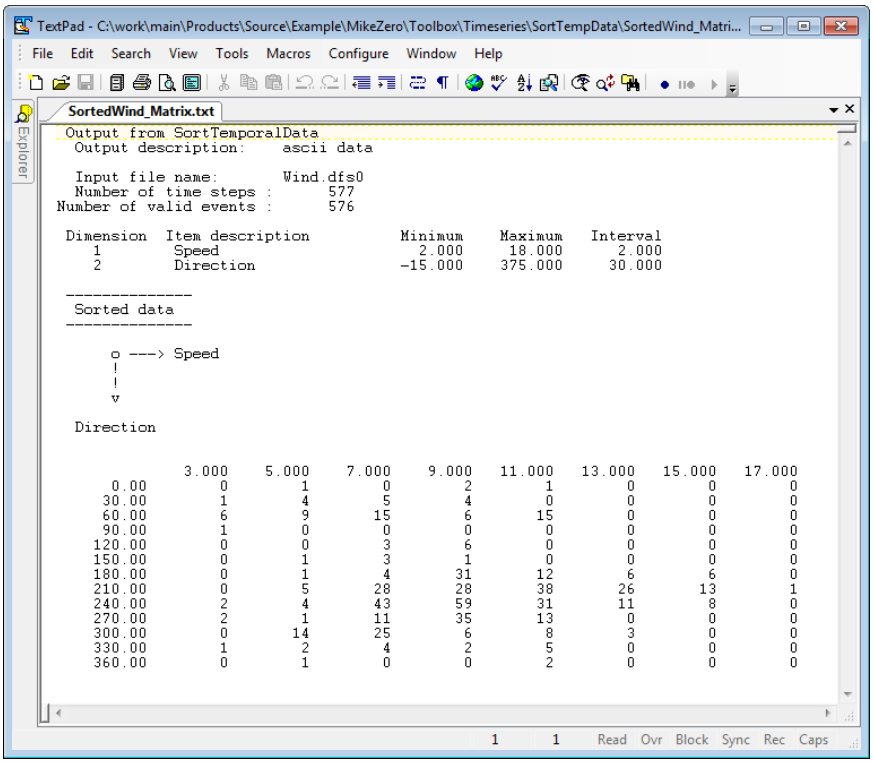


Figure 23.8 Resulting ascii output file
Note: one of the events was deemed invalid because the wind speed was below the minimum value of 2 m/s.

23.7.2 Wave climate

The following figures describe the process and result when you sort a time series file containing a wave climate where each event is defined by an individual duration.

Consider a case where a harbour experience severe seiching for waves with wave periods above 10 s that comes from SE (120 degrees -150 degrees). By sorting the available data using only 2 items and relevant class intervals it is possible to easy determine how often the seiching will occur.

The results in Figure 23.15 shows that during a total period of 35064 hours, expressed by 11333 time steps in the input time series, seiching will occur only during 7 hours, i.e. for 0.02% of a year.

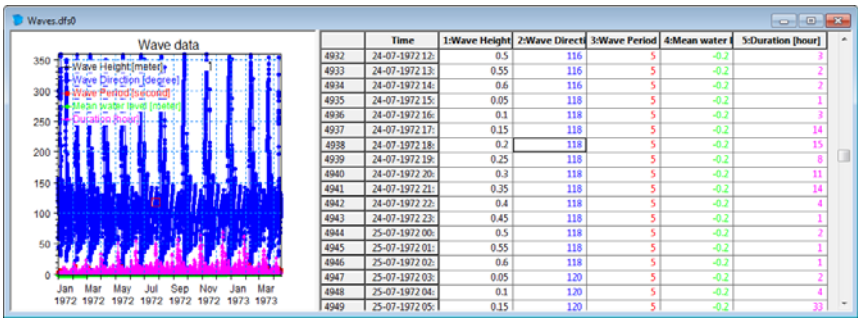


Figure 23.9 Individual wave events with varying duration

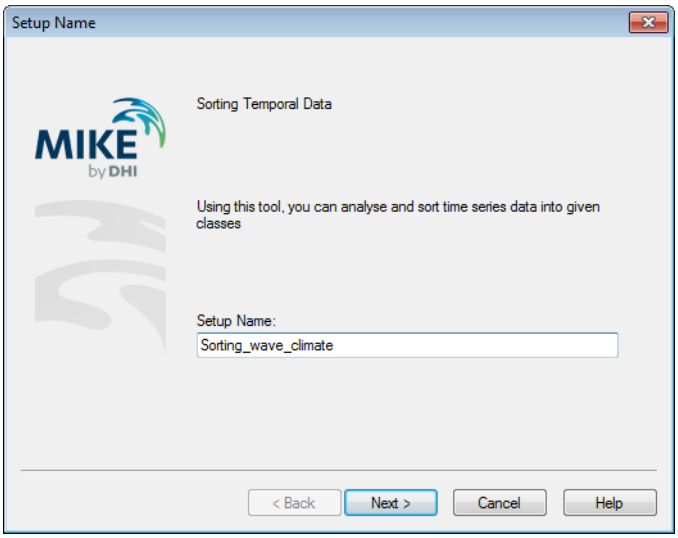


Figure 23.10 Specify name for setup

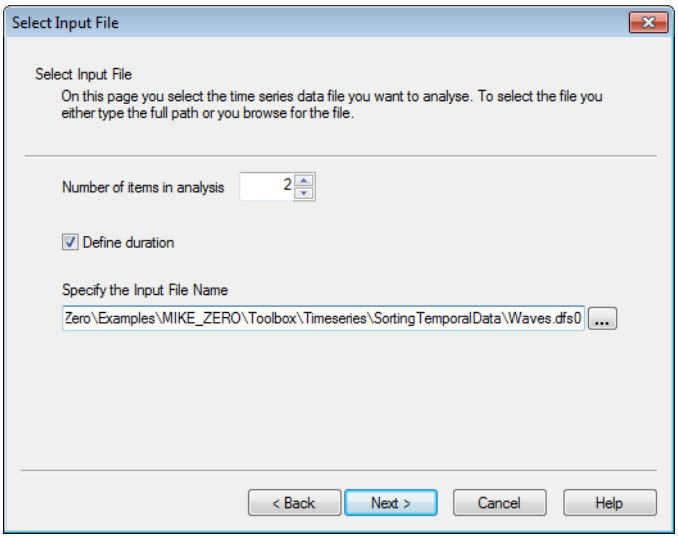


Figure 23.11 Define number of items in analysis, enable duration and select input file and input items

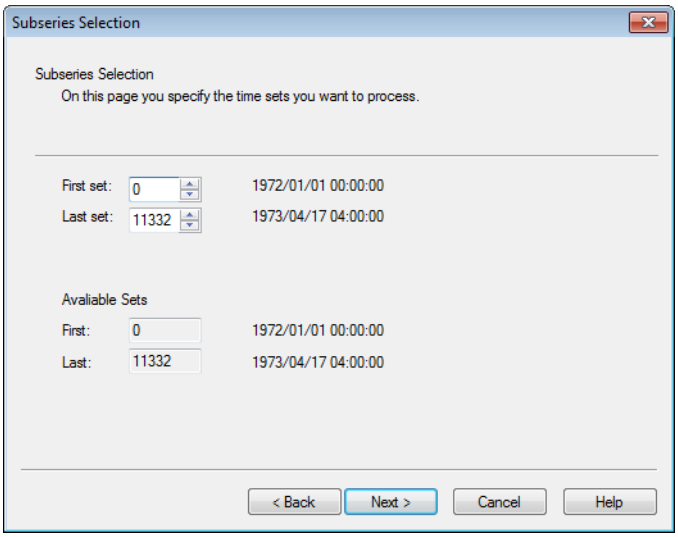


Figure 23.12 Specify subseries

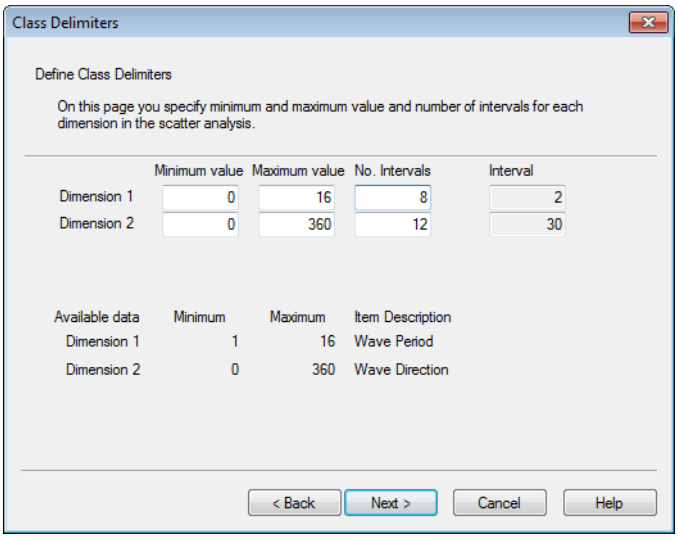


Figure 23.13 Specify class dimension and intervals for sorting

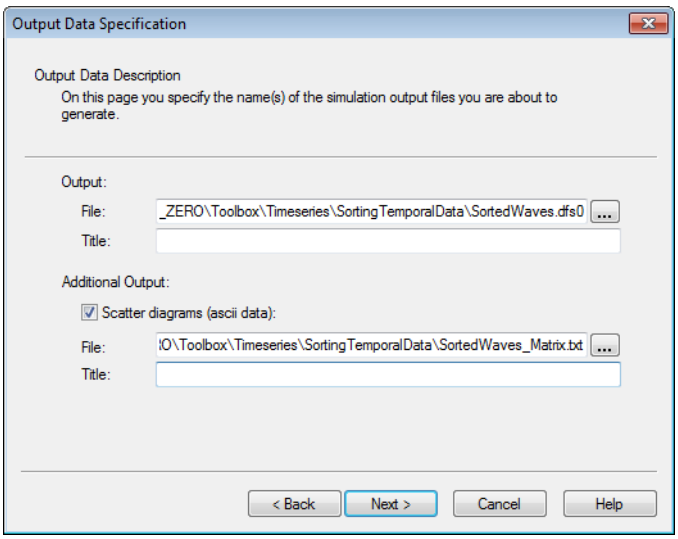
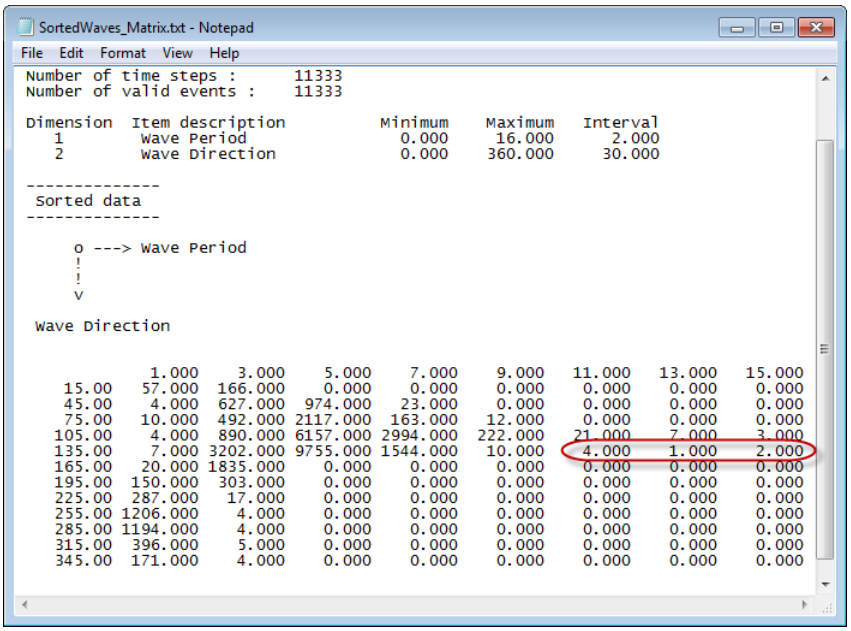


Figure 23.14 Specify output file names



SortedWaves_Matrix.txt - Notepad

File Edit Format View Help

```
Number of time steps :      11333
Number of valid events :    11333

Dimension  Item description          Minimum    Maximum   Interval
   1        Wave Period                0.000     16.000      2.000
   2        Wave Direction             0.000    360.000     30.000

--------------
 Sorted data
--------------

     o ---> Wave Period
     !
     !
     v

 Wave Direction


             1.000    3.000    5.000    7.000    9.000   11.000   13.000   15.000
 15.00      57.000  166.000    0.000    0.000    0.000    0.000    0.000    0.000
 45.00       4.000  627.000  974.000   23.000    0.000    0.000    0.000    0.000
 75.00      10.000  492.000 2117.000  163.000   12.000    0.000    0.000    0.000
105.00       4.000  890.000 6157.000 2994.000  222.000   21.000    7.000    3.000
135.00       7.000 3202.000 9755.000 1544.000   10.000    4.000    1.000    2.000
165.00      20.000 1835.000    0.000    0.000    0.000    0.000    0.000    0.000
195.00     150.000  303.000    0.000    0.000    0.000    0.000    0.000    0.000
225.00     287.000   17.000    0.000    0.000    0.000    0.000    0.000    0.000
255.00    1206.000    4.000    0.000    0.000    0.000    0.000    0.000    0.000
285.00    1194.000    4.000    0.000    0.000    0.000    0.000    0.000    0.000
315.00     396.000    5.000    0.000    0.000    0.000    0.000    0.000    0.000
345.00     171.000    4.000    0.000    0.000    0.000    0.000    0.000    0.000
```

Figure 23.15 Resulting ascii output file.
Note: emphasized values are number of hours where seiching occur.

24 Time Series Batch Conversion

The Time Series Batch Conversion tool provides a user-friendly way of converting large amounts of time series data files between dfs0, Excel and ASCII formats. The format of the input time series data in ASCII and Excel files must be defined by an user-defined template.

24.1 General Introduction

Time series data are one of the most fundamental data types in virtually the entire MIKE Powered by DHI product line. The shared format for storage of time series data is the .dfs0 file format. However, often the original time series data are given and pre-processed in other formats such as ASCII and Excel. It may be that the decimal separator is different from the commonly used and sometimes it is also required that results are exported to ASCII and Excel files for later use in 3rd-party software.

The basic time series tool, Time Series Editor, in MIKE Zero can import and export time series data in ascii format, however it can only handle one time series at a time and the format of the ASCII input file is restricted to a DHI standard for the Time Series Editor. It is thus not feasible for handling large amounts of data files.

This Time Series Batch Conversion tool can perform batch-processing of large amounts of time series data files in dfs0, ASCII and Excel format.

The resulting output data files will be formatted in a default layout.

24.1.1 dfs0

Files in dfs0 format are per default defined by an equidistant calendar axis with a number of items, see Figure 24.1.

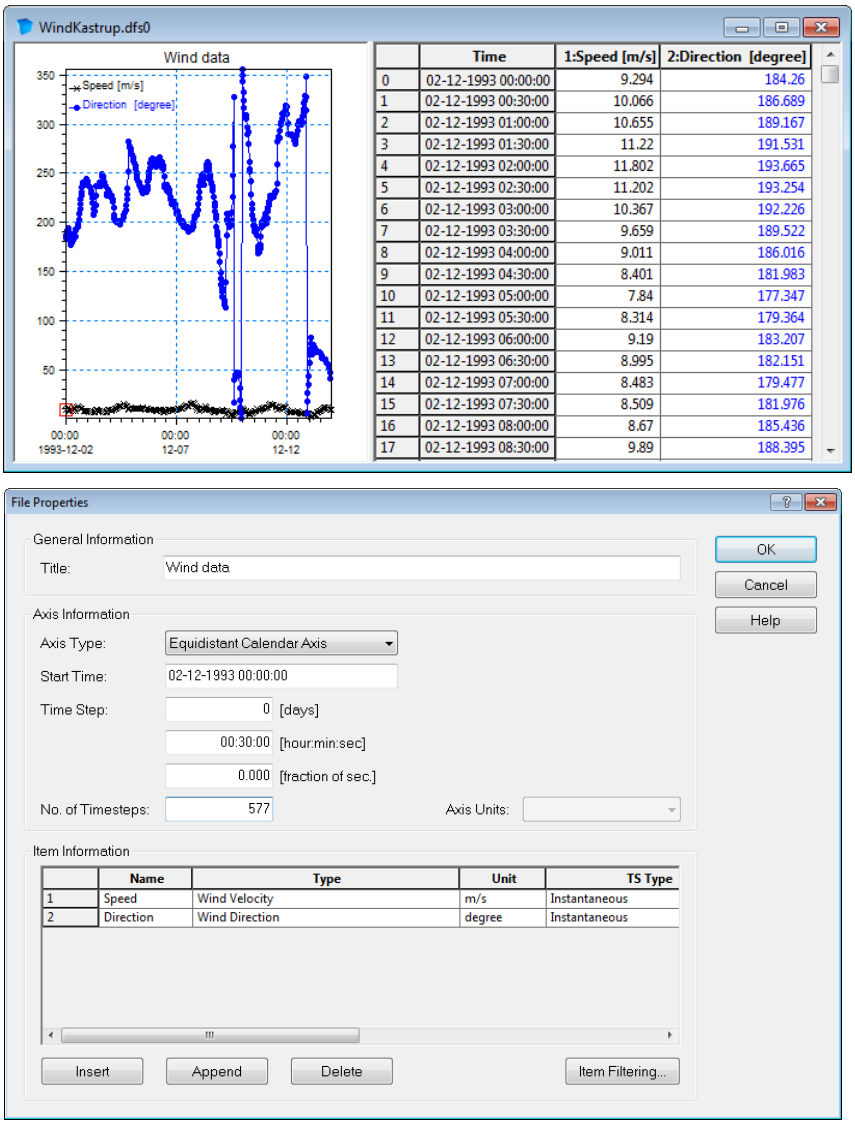
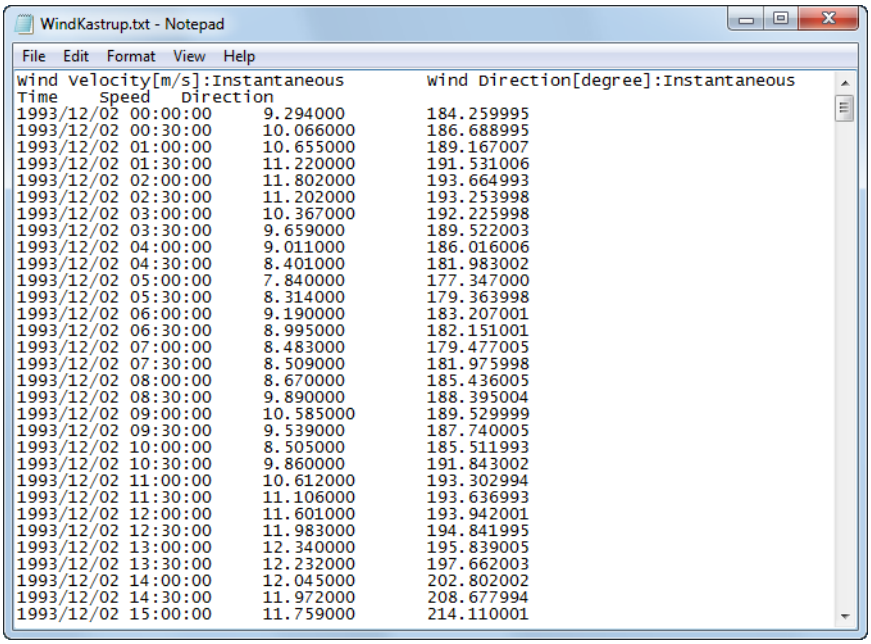


Figure 24.1 Default layout for files in dfs0 format

24.1.2 ASCII

Files in ASCII format must contain information with an increasing time record and subsequent item information.

The default ASCII format layout is as indicated in Figure 24.2, however it is possible to define templates for other layouts of ASCII files. The extension of an ASCII output file will always be .txt.



WindKastrup.txt - Notepad

File Edit Format View Help

```
Wind Velocity[m/s]:Instantaneous          Wind Direction[degree]:Instantaneous
Time     Speed    Direction
1993/12/02 00:00:00      9.294000        184.259995
1993/12/02 00:30:00      10.066000       186.688995
1993/12/02 01:00:00      10.655000       189.167007
1993/12/02 01:30:00      11.220000       191.531006
1993/12/02 02:00:00      11.802000       193.664993
1993/12/02 02:30:00      11.202000       193.253998
1993/12/02 03:00:00      10.367000       192.225998
1993/12/02 03:30:00      9.659000        189.522003
1993/12/02 04:00:00      9.011000        186.016006
1993/12/02 04:30:00      8.401000        181.983002
1993/12/02 05:00:00      7.840000        177.347000
1993/12/02 05:30:00      8.314000        179.363998
1993/12/02 06:00:00      9.190000        183.207001
1993/12/02 06:30:00      8.995000        182.151001
1993/12/02 07:00:00      8.483000        179.477005
1993/12/02 07:30:00      8.509000        181.975998
1993/12/02 08:00:00      8.670000        185.436005
1993/12/02 08:30:00      9.890000        188.395004
1993/12/02 09:00:00      10.585000       189.529999
1993/12/02 09:30:00      9.539000        187.740005
1993/12/02 10:00:00      8.505000        185.511993
1993/12/02 10:30:00      9.860000        191.843002
1993/12/02 11:00:00      10.612000       193.302994
1993/12/02 11:30:00      11.106000       193.636993
1993/12/02 12:00:00      11.601000       193.942001
1993/12/02 12:30:00      11.983000       194.841995
1993/12/02 13:00:00      12.340000       195.839005
1993/12/02 13:30:00      12.232000       197.662003
1993/12/02 14:00:00      12.045000       202.802002
1993/12/02 14:30:00      11.972000       208.677994
1993/12/02 15:00:00      11.759000       214.110001
```

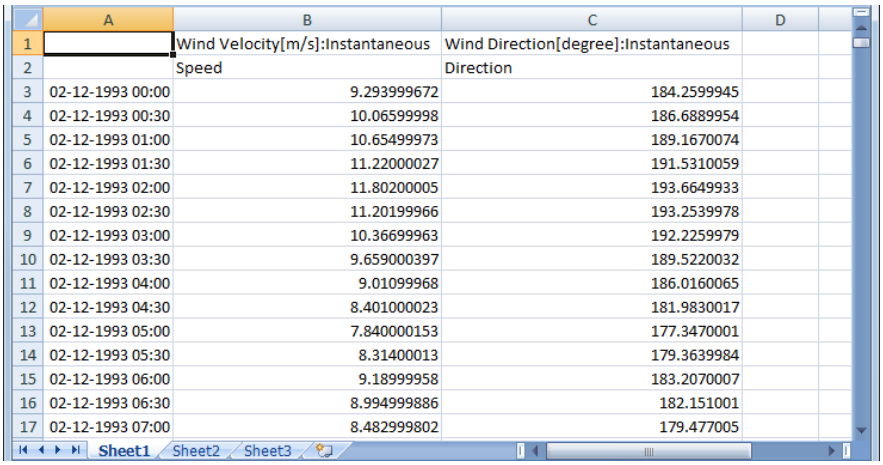
Figure 24.2 Default layout for files in ASCII format
Row 1: Type, Unit and Value type for each item
Row 2: 'Time' string + item name for each item
Row 3 and below: Date and time followed by item values

NOTE: The default layout for files in ASCII format is not the same as the layout necessary for importing ASCII files into the Time Series Editor.

24.1.3 Excel

Files in Excel format must contain information with an increasing time record and subsequent item information. The time specification may be defined in one or several columns.

The default Excel format layout is as indicated in Figure 24.3, however it is possible to define templates for other layouts of Excel files.



	A	B	C	D
1		Wind Velocity[m/s]:Instantaneous	Wind Direction[degree]:Instantaneous	
2		Speed	Direction	
3	02-12-1993 00:00	9.293999672	184.2599945	
4	02-12-1993 00:30	10.06599998	186.6889954	
5	02-12-1993 01:00	10.65499973	189.1670074	
6	02-12-1993 01:30	11.22000027	191.5310059	
7	02-12-1993 02:00	11.80200005	193.6649933	
8	02-12-1993 02:30	11.20199966	193.2539978	
9	02-12-1993 03:00	10.36699963	192.2259979	
10	02-12-1993 03:30	9.659000397	189.5220032	
11	02-12-1993 04:00	9.01099968	186.0160065	
12	02-12-1993 04:30	8.401000023	181.9830017	
13	02-12-1993 05:00	7.840000153	177.3470001	
14	02-12-1993 05:30	8.31400013	179.3639984	
15	02-12-1993 06:00	9.18999958	183.2070007	
16	02-12-1993 06:30	8.994999886	182.151001	
17	02-12-1993 07:00	8.482999802	179.477005	

Sheet1 Sheet2 Sheet3

Figure 24.3 Default layout for files in Excel format
Row 1: Type, Unit and Value type for each item
Row 2: Item name for each item
Row 3 and below: Date and time followed by item values

24.2 Select Input Data

On this page you select your input data.

You can choose to convert one file or a number of similar files contained within a folder.

The format of the data file may be one of the following:

- dfs0
- ASCII
- Excel

NOTE: If you select ASCII or Excel format you will have to define a template in the next dialog

24.3 Define Template

On this dialog you define the template to use for the conversion.

You have three options:

- Select existing template
- Modify existing template
- Create new template

If you have selected one of the first two options you must specity the template file to use.

When creating a new template the initial settings correspond to the default layout for input file type (ASCII or Excel).

NOTE: A template file has the extension .xml.

24.4 Delimiter Description

On this dialog you specify how to interpret the delimiters in the ASCII file. The delimiters may be one of the following:

- Comma
- Semi-colon
- Space
- Tab
- User defined
  In this case define the delimiter string in the box to the right

You can furthermore select to treat consecutive delimiters as one and/or ignore delimiters at the beginning of a line.

Finally you one of three ways to define the decimal separator:

- Comma
- Dot
- User locale
  I.e. the seperator is defined by the local system settings

24.5 Time Description

On this page you select how the time and date values are defined in the input file. You have to define the format and the (first) column of the specifications. You have three option to choose from:

- Date and time
- Only time
- Only date

The format is specified by year (YY), month (MM), day (DD), hour (HH), minute (mm) and second (ss).

In case the input file format is Excel the date and time may be defined over several columns and you have to specify how the data is represented in the individual columns by using the '|' symbol.

24.6 Data Description

On this page you specify how the data values are defined and stored in the input data file.

The individual items are described by a title, an item type, an item unit and a value type, selected among the available parameters in the pre-defined MIKE Zero item groups. Generally you can define the number of the row in which the specific parameter is stored or define the parameter manually, for each item individually or the same parameter for all items at once.

You have to define the row number from which the actual values are stored.

Per default an empty value denotes the delete value. You may specify the delete value otherwise by defining a specific value or the number of the row containing the delete value.

24.7 Item Description

On this page you select which items to convert and save in the output. You may select any number.

NOTE: In case you have chosen to convert a folder of data files and the input format is dfs0, the number of selectable items is determined by the number of items in the first valid dfs0 file in the folder.

24.8 Save Template File

In this dialog you save the template file that will be used in the conversion process. You may save to an existing template file or create a new template file.

NOTE: The template file will be saved when you press *Next*.

24.9 Output Specification

On this page you define the format of the converted data and the name and title for the output data file or folder.

The format of the data file may be one of the following:

- dfs0
- ASCII (default layout)
- Excel (default layout)

NOTE: If you are converting files from a folder, the names of the files will be derived from the input file names.

24.10 Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

You have the option to specify the location of an eventual log-file from the conversion. If the conversion is successful no log-file is created, otherwise a logfile with a list of the files not matching the template is saved.

From the page you can either ***Finish*** the set-up, which will transfer your set-up definition to the current toolbox file, or ***Execute*** the set-up.

24.11 Preview

The preview button shows how the program will interprete the input data on basis of the given settings. The first column shows the time whereas the remaining columns show the items(s).

In case the given settings don't allow for an interpretation of the input data a message box will appear with a informative error message before the preview is shown.

NOTE: In case the conversion is carried out on several files in a folder one of the files in the folder will be selected as representative and shown in the display.

24.12 Examples

24.12.1 ASCII to dfs0

The following figures describe the process on how to prepare a template file for converting an ASCII file to a dfs0 file and the subsequent use of this template to batch-convert a number of ascii files with the same format from the same folder.

The time series data files used in this example contains measured water levels. The files are installed as a part of the MIKE Zero Example 'Sound' located in e.g. .\Examples\MIKE_Zero\Projects\Sound\External Data\Measurements.

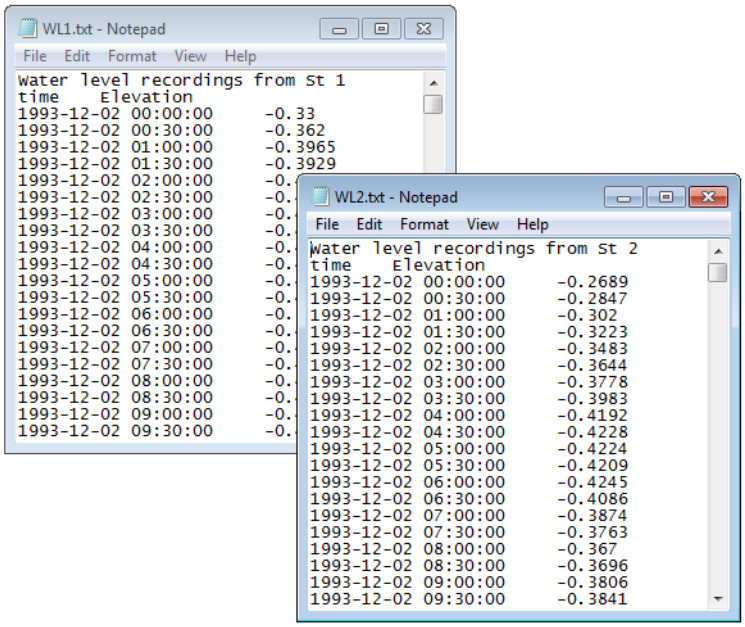


Figure 24.4 Selected input data files as displayed by notepad

Select tool and single file for input data

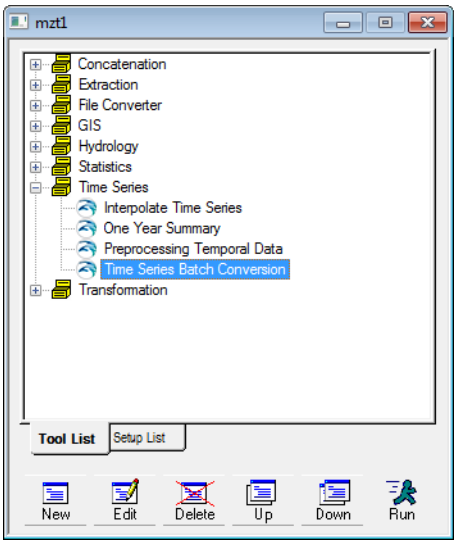


Figure 24.5 Select the utility in the MIKE Zero toolbox

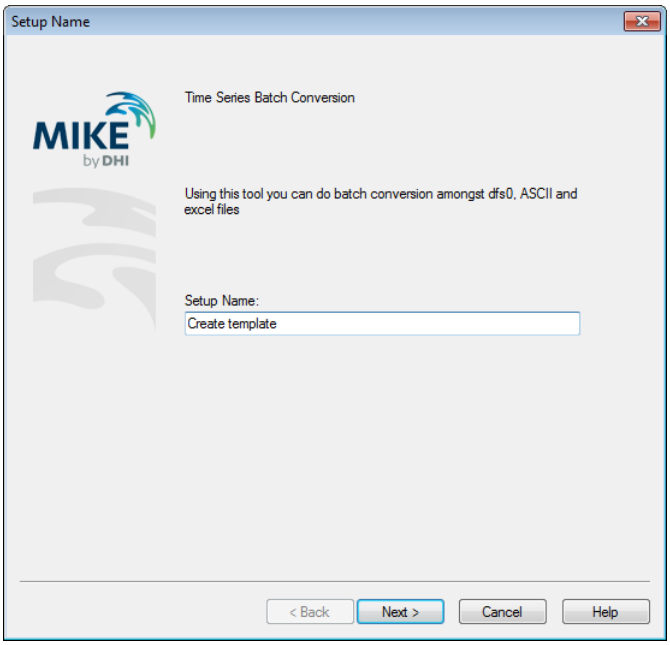


Figure 24.6 Define the name for the setup

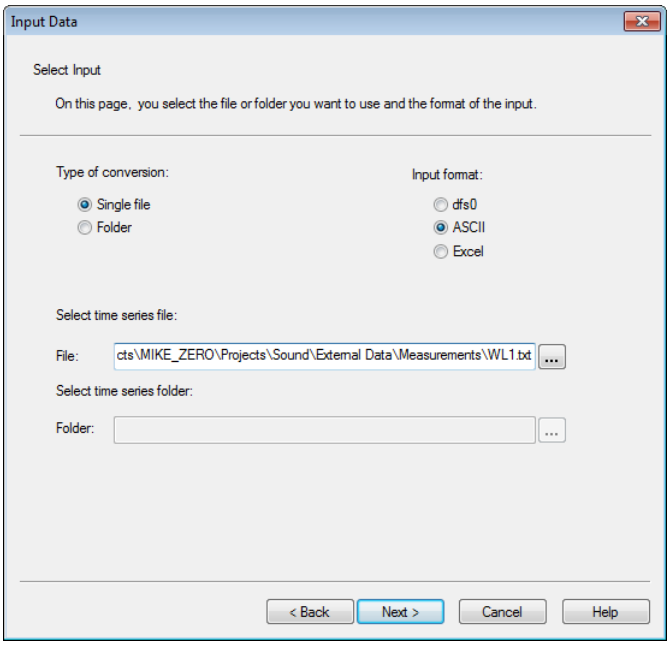


Figure 24.7 Define the ASCII input data file for template

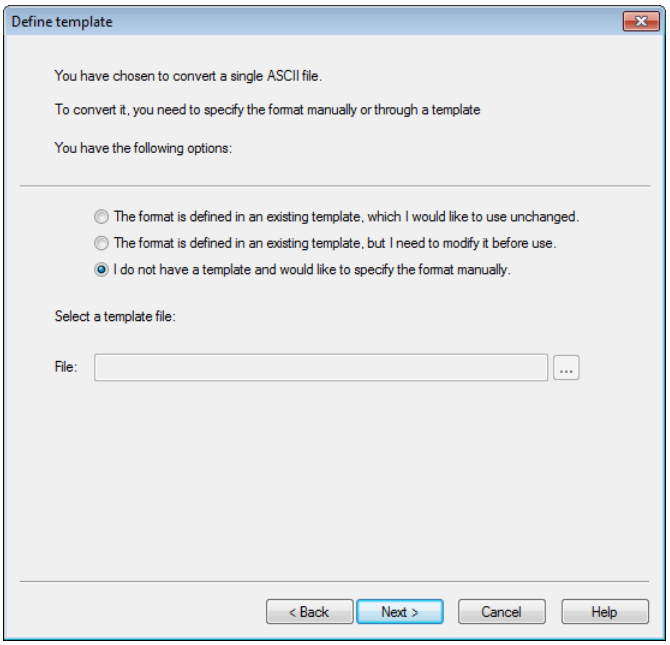


Figure 24.8 Specify that a new template is to be created

Define template for ASCII file

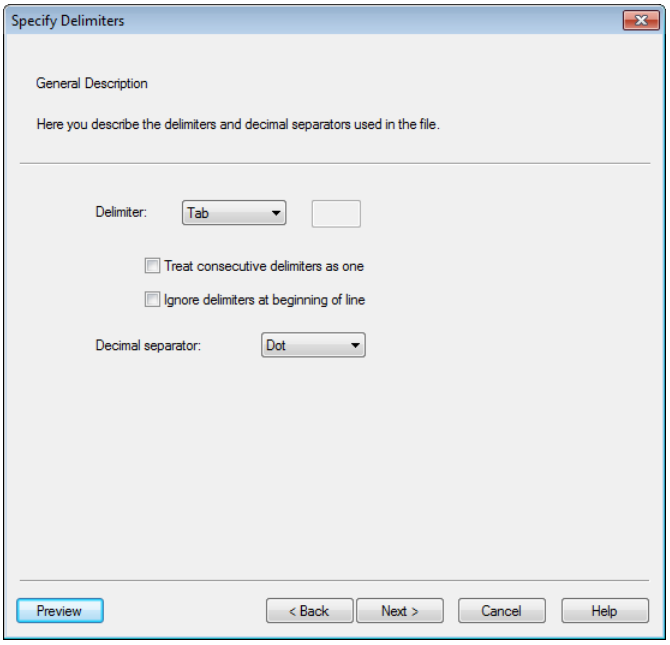


Figure 24.9 Specify delimiter and decimal separator

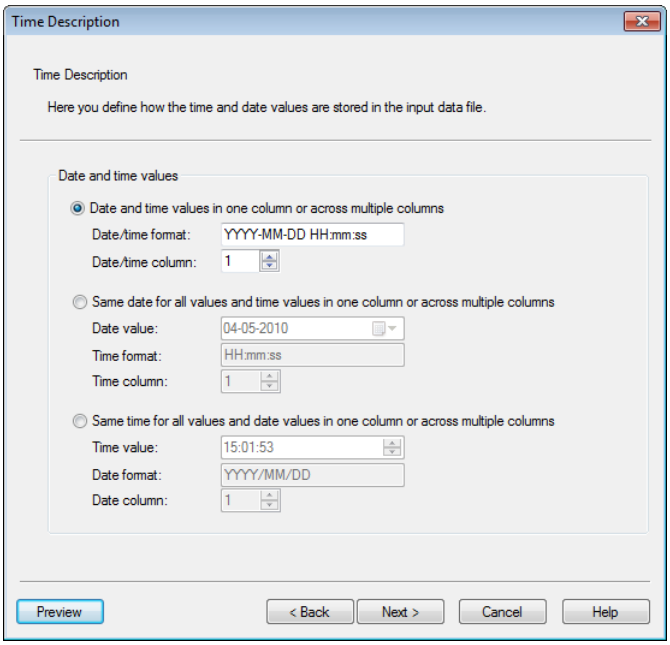


Figure 24.10 Specify time description

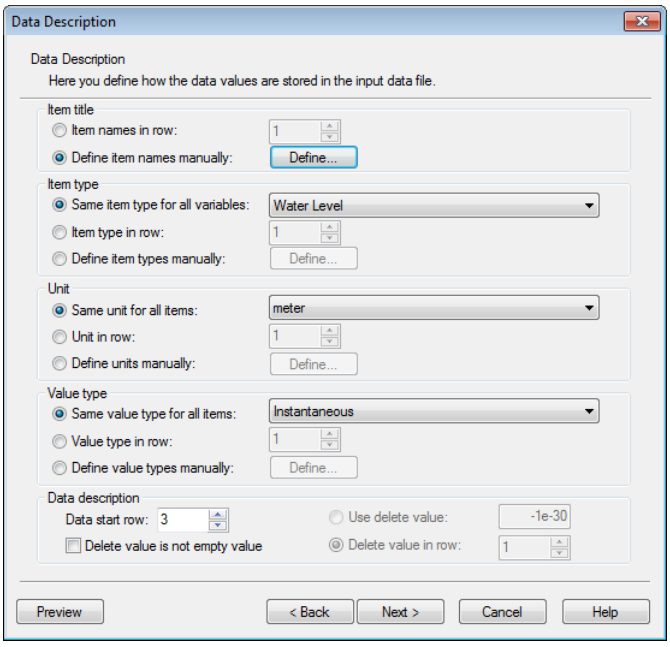


Figure 24.11 Specify item description

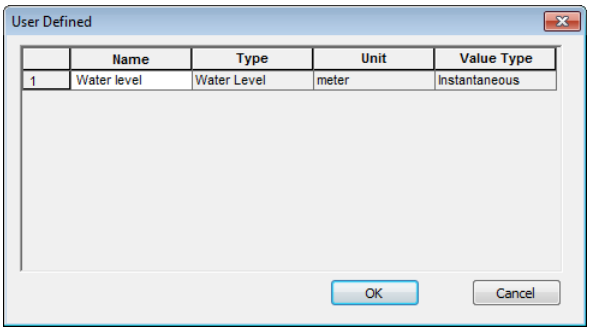


Figure 24.12 Item name specified manually

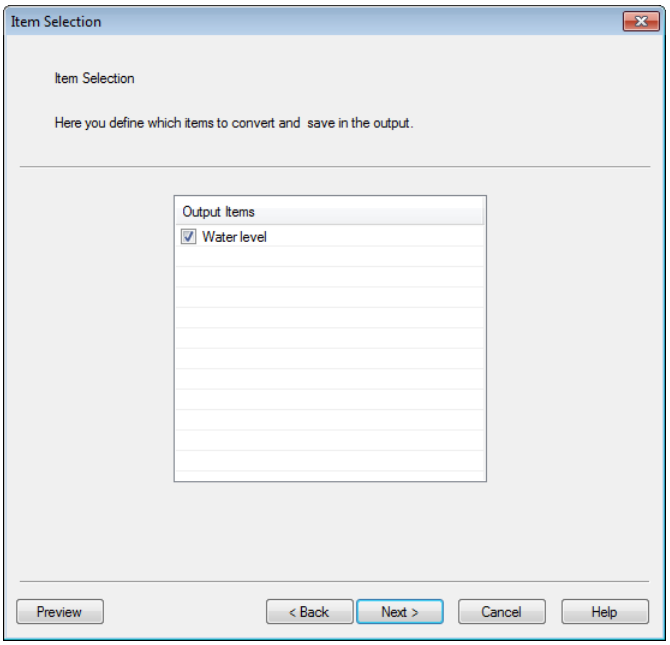


Figure 24.13 Select item to convert

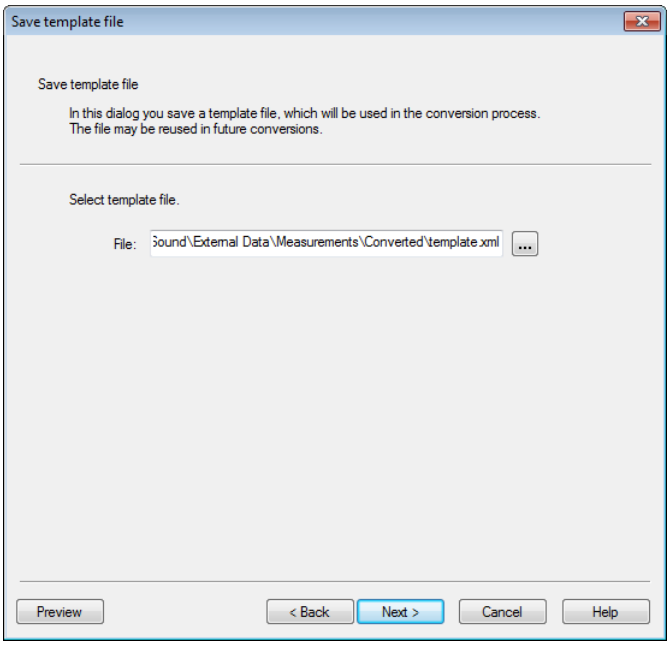


Figure 24.14 Save template file

NOTE: The template file will be saved when you press Next.

Specify output data and convert single file

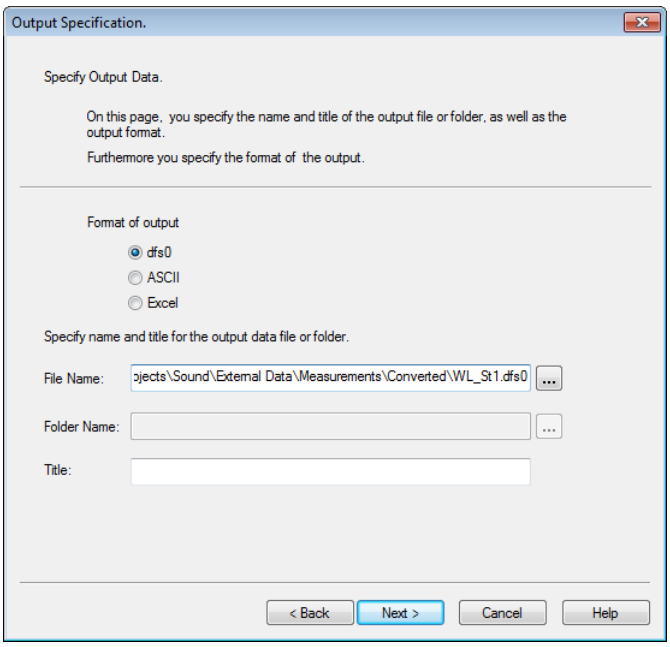
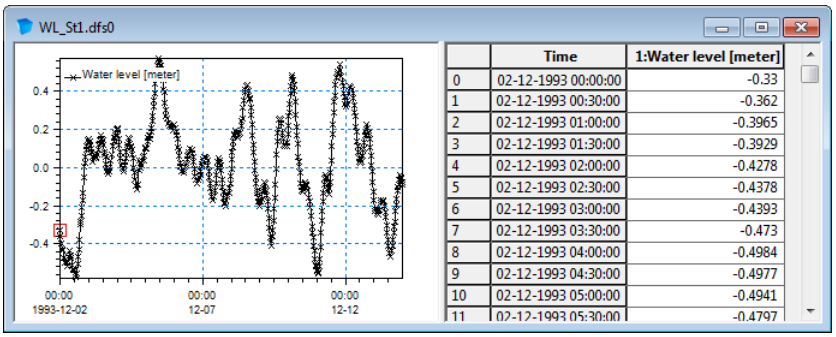


Figure 24.15 Define output format and output file name

Press Next and Execute to convert the ASCII file to a dfs0 file.



	Time	1:Water level [meter]
0	02-12-1993 00:00:00	-0.33
1	02-12-1993 00:30:00	-0.362
2	02-12-1993 01:00:00	-0.3965
3	02-12-1993 01:30:00	-0.3929
4	02-12-1993 02:00:00	-0.4278
5	02-12-1993 02:30:00	-0.4378
6	02-12-1993 03:00:00	-0.4393
7	02-12-1993 03:30:00	-0.473
8	02-12-1993 04:00:00	-0.4984
9	02-12-1993 04:30:00	-0.4977
10	02-12-1993 05:00:00	-0.4941
11	02-12-1993 05:30:00	-0.4797

Figure 24.16 Resulting file as displayed in Time Series Editor.

Convert multiple ASCII files in batch

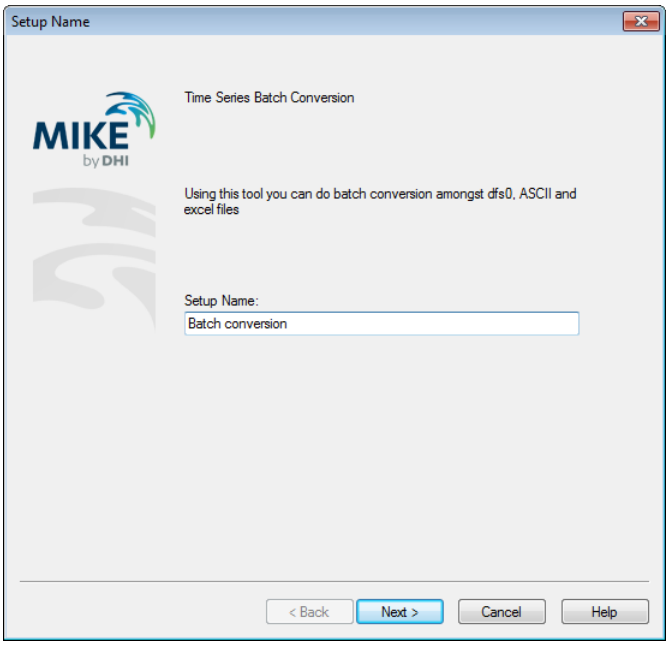


Figure 24.17 Define the name for the setup

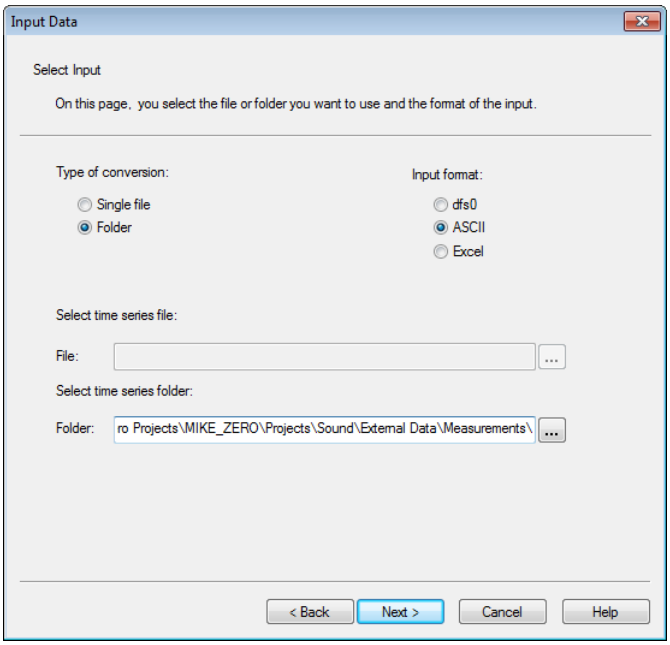


Figure 24.18 Define the folder containing the ASCII input data files

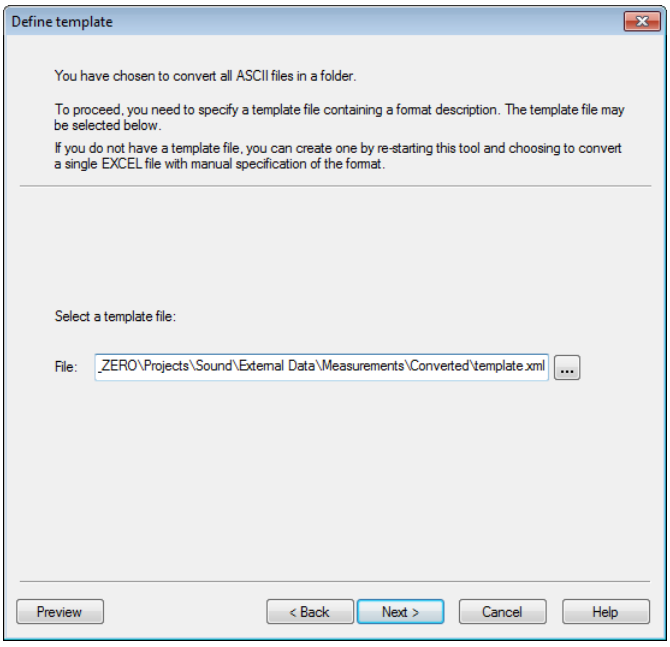


Figure 24.19 Specify the name of the existing template

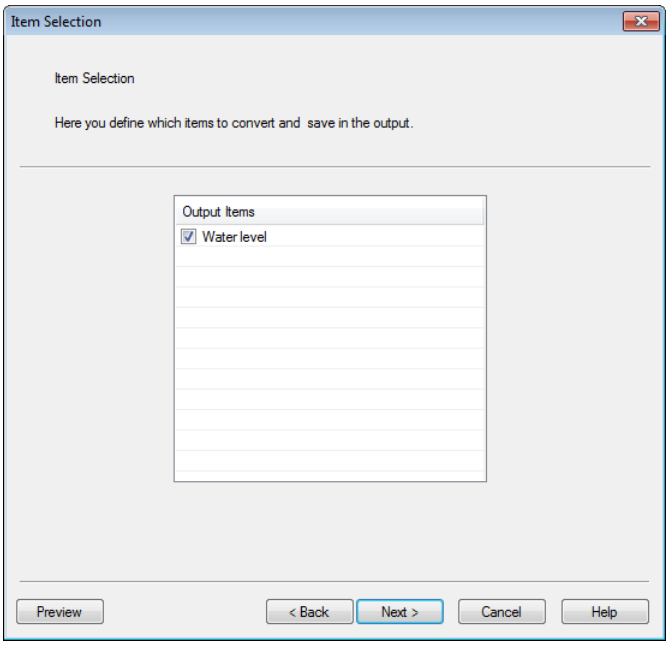


Figure 24.20 Select item to convert

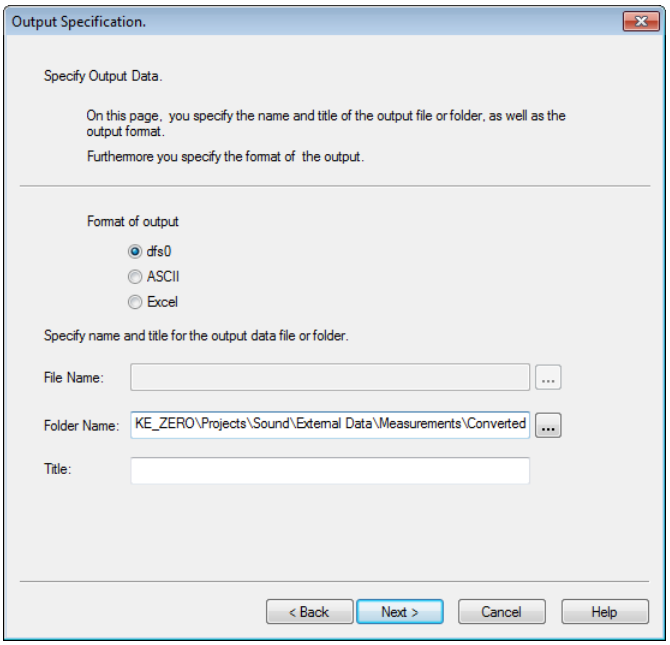


Figure 24.21 Specify format type and folder for output data files.
Press Next and Execute to convert the ASCII files to dfs0 files.

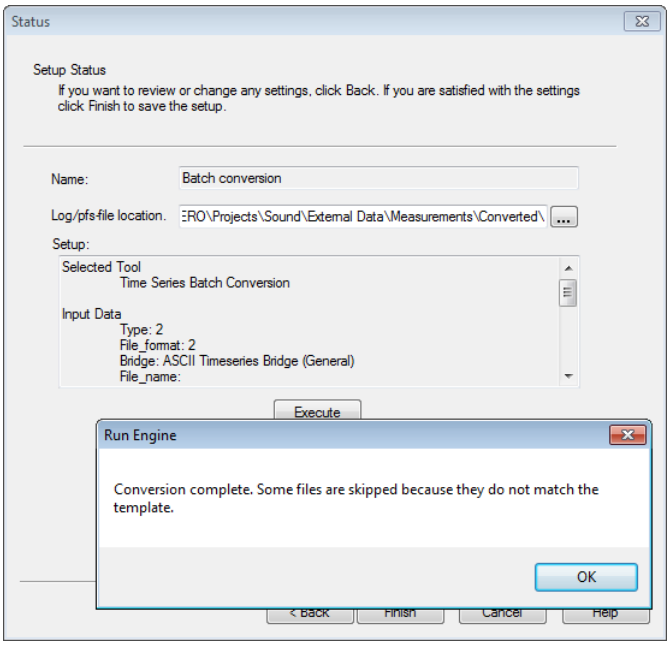
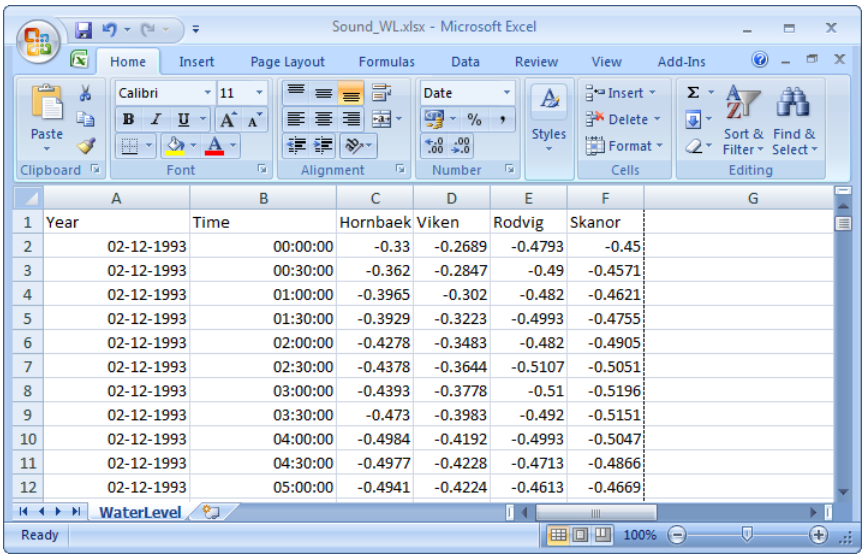


Figure 24.22 Status page.
Resulting message box revields that not all ascii files in the folder were converted due to incompatible format.

24.12.2 Excel to dfs0

The following figures describe the process on how to prepare a template file for converting an Excel file to a dfs0 file. The subsequent use of this template to batch-convert a number of Excel files work in similary way to a batch conversion of ASCII files.

The time series data file used in this example contains measured water levels in four tidal station. The file 'Sound_WL.xlsx' is installed as a part of the MIKE Zero Example 'Sound' located in e.g. .\Examples\MIKE_Zero\Projects\Sound\External Data\Measurements.



	A	B	C	D	E	F	G
1	Year	Time	Hornbaek	Viken	Rodvig	Skanor	
2	02-12-1993	00:00:00	-0.33	-0.2689	-0.4793	-0.45	
3	02-12-1993	00:30:00	-0.362	-0.2847	-0.49	-0.4571	
4	02-12-1993	01:00:00	-0.3965	-0.302	-0.482	-0.4621	
5	02-12-1993	01:30:00	-0.3929	-0.3223	-0.4993	-0.4755	
6	02-12-1993	02:00:00	-0.4278	-0.3483	-0.482	-0.4905	
7	02-12-1993	02:30:00	-0.4378	-0.3644	-0.5107	-0.5051	
8	02-12-1993	03:00:00	-0.4393	-0.3778	-0.51	-0.5196	
9	02-12-1993	03:30:00	-0.473	-0.3983	-0.492	-0.5151	
10	02-12-1993	04:00:00	-0.4984	-0.4192	-0.4993	-0.5047	
11	02-12-1993	04:30:00	-0.4977	-0.4228	-0.4713	-0.4866	
12	02-12-1993	05:00:00	-0.4941	-0.4224	-0.4613	-0.4669	

Figure 24.23 Selected input data file as displayed in Excel

Select tool and single file for input data

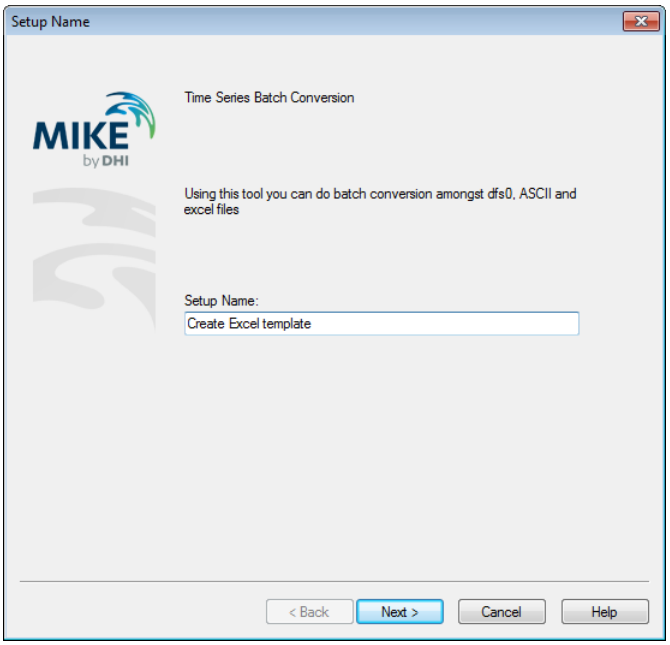


Figure 24.24 Define the name for the setup

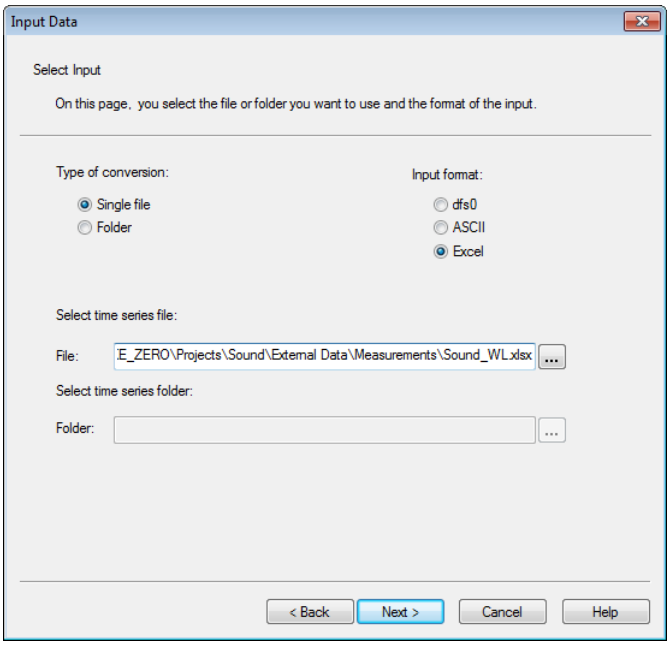


Figure 24.25 Define the ASCII input data file for template

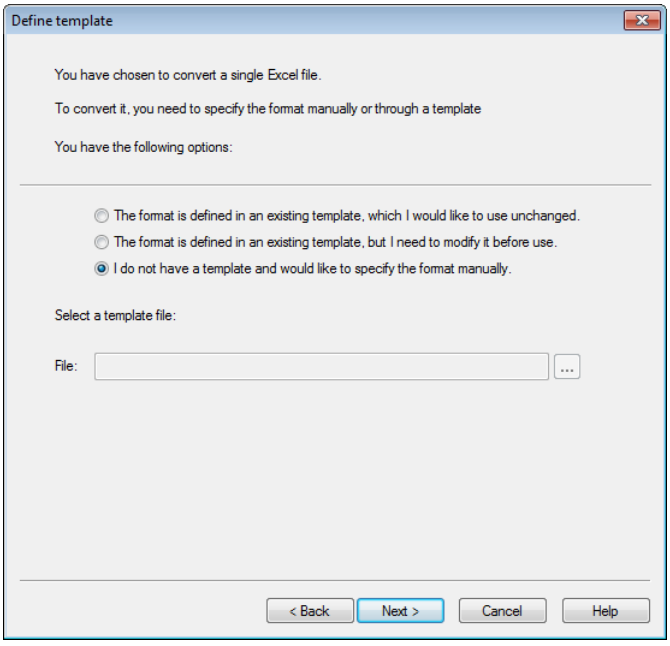


Figure 24.26 Specify that a new template is to be created

Define template for Excel file

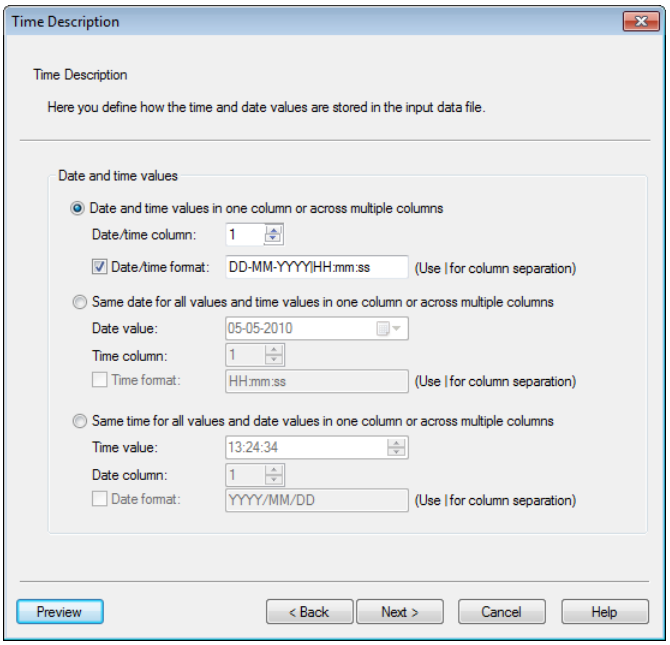


Figure 24.27 Specify time description

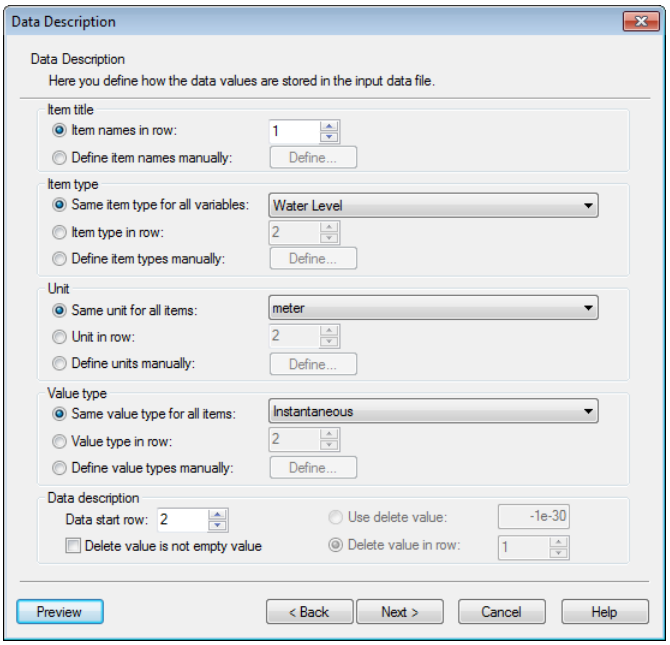


Figure 24.28 Specify item description

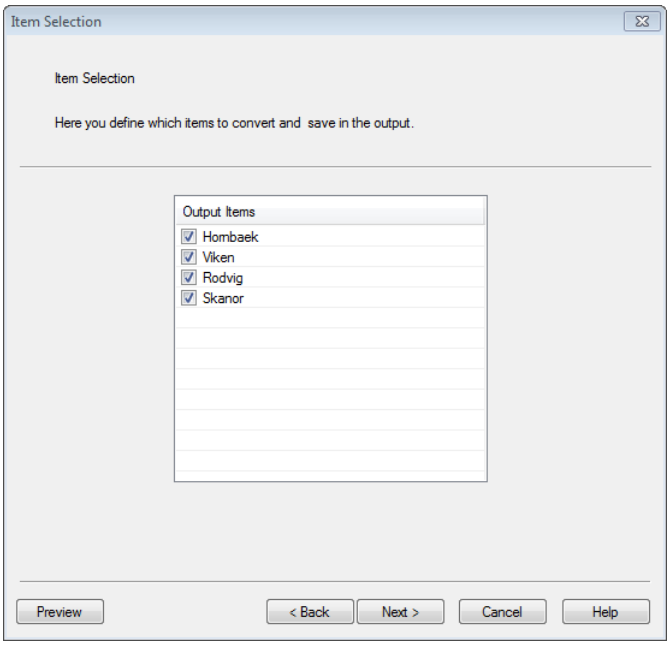


Figure 24.29 Select items to convert

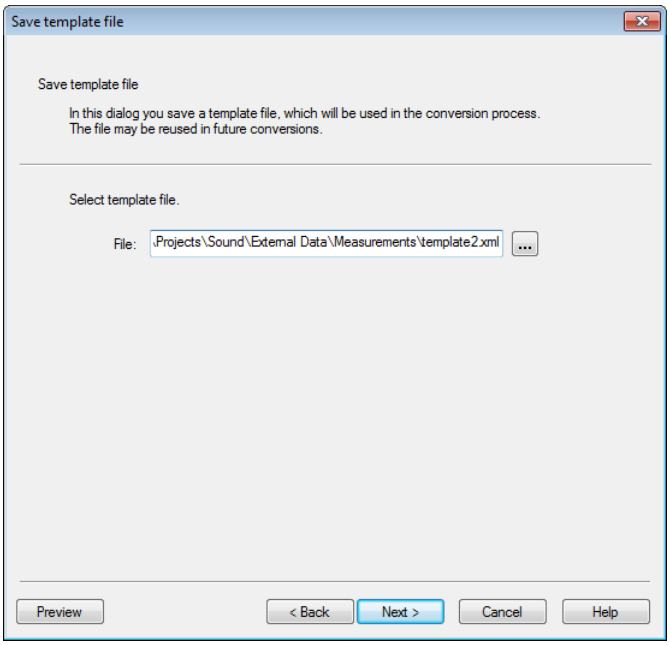


Figure 24.30 Save template file

Specify output data and convert single file

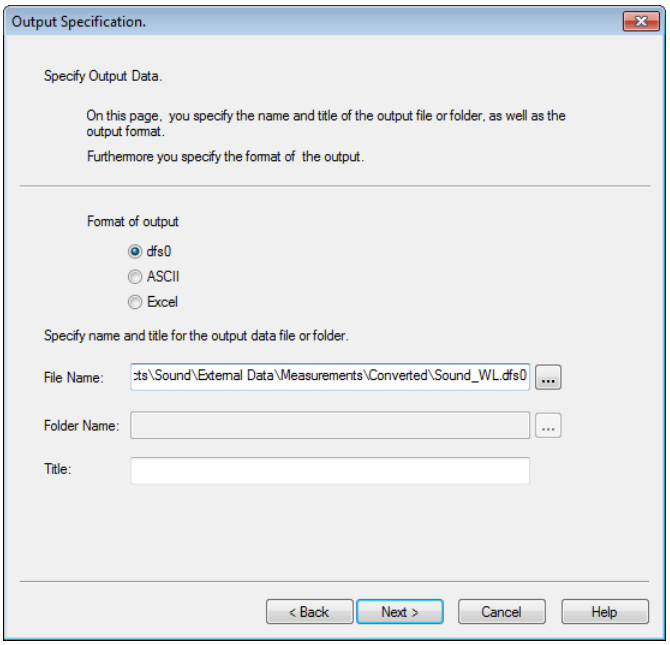


Figure 24.31 Define output format and output file name

Press Next and Execute to convert the Excel file to a dfs0 file.

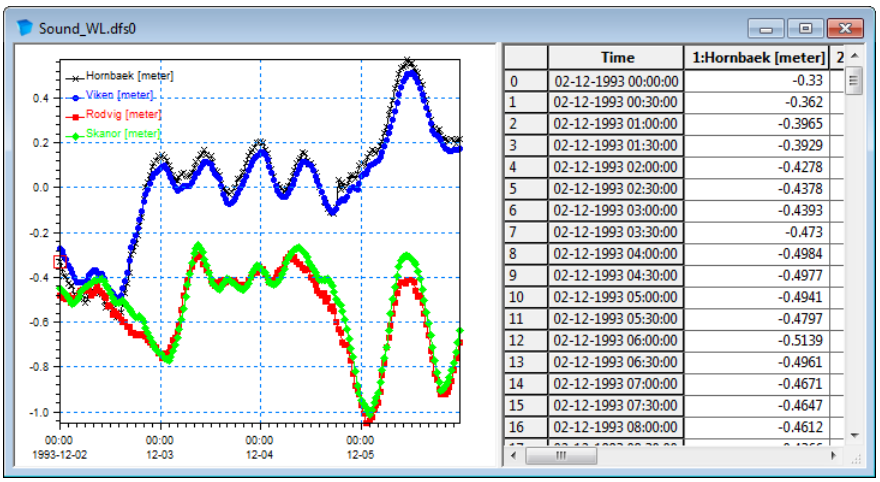


Figure 24.32 Resulting file as displayed in Time Series Editor.

MZ TOOLS: TRANSFORMATION

The TRANSFORMATION Tools in the MZ Toolbox include the facility:

- Rotate Grid

25 Rotate Grid

This transformation tool can interpolate and rotate scalar fields (e.g. bathymetries), vector fields (e.g. wind velocity components) and tensor fields (e.g. radiation stresses) stored in dfs2 data files.

The output is a new dfs2 data file with spatially interpolated values.

The interpolation is carried out using simple bilinear interpolation.

25.1 Setup Name

Here you choose your project name.

25.2 Select Input File

On this page you select the type 2 data file you want to interpolate/rotate from.

25.3 Subseries Selection

You may select a subseries instead of the whole time range.

25.4 Select Item

If the data file contains more than one item, you must specify which items you want to process.

Items are disabled by selecting 'Not Used', and scalar items are specified as 'Scalar'.

If some of the items constitute a vector field you have to specify which item is the 'Vector X'-component and which is the 'Vector Y'-component. Also, if some of the items constitute a 2. order tensor field, e.g. a stress tensor, you should specify the two normal components 'Tensor XX' and 'Tensor YY' and the shear component 'Tensor XY'.

25.4.1 Derived items

If the data file is 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 or more of the following derived items: W Surface Elevation, S Current Speed, and Vx and Vy (current velocities). These items are calculated from h, p and q by the program as described below.

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.

Figure 25.1 shows the definition of the grid items with respect to the position in the grid. The saved h, p and q values in the file are positioned in the grid as shown to the left, the derived items are all positioned in the center point of the grid element as shown to the right.

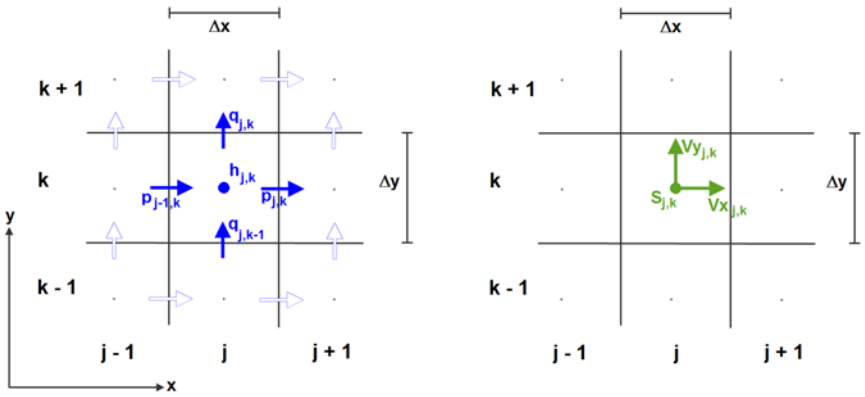


Figure 25.1 Position in grid for items.
Left: Water depth (h) and flux components (p,q)
Right: Derived current speed (S) and velocity components (Vx,Vy)

The equations used for the calculation of the derived items are as follows:

$$W_{j,k} = h_{j,k} + z_{j,k} \tag{25.1}$$

$$Vx_{j,k} = \frac{p_{j,k}}{h_{j+1,k} + h_{j,k}} + \frac{p_{j-1,k}}{h_{j,k} + h_{j-1,k}} \tag{25.2}$$

$$Vy_{j,k} = \frac{q_{j,k}}{h_{j,k+1} + h_{j,k}} + \frac{q_{j,k-1}}{h_{j,k} + h_{j,k-1}} \tag{25.3}$$

$$S_{j,k} = \sqrt{(Vx_{j,k})^2 + (Vy_{j,k})^2} \tag{25.4}$$

where *zj,k* is the bathymetry value stored in the static item.

25.5 Grid Size

On this page you define the new grid:

- You must specify the extent of the new grid, i.e. the number of grid points and the grid spacing.
- The origin of the new grid relative to the old one must be specified. The shift in the origin is expressed in grid cell units, and it can be a float number. Note that the origin of the new grid can be placed outside the old grid domain.
- The rotation angle - measured positive clockwise from new grid to old grid - must be specified.

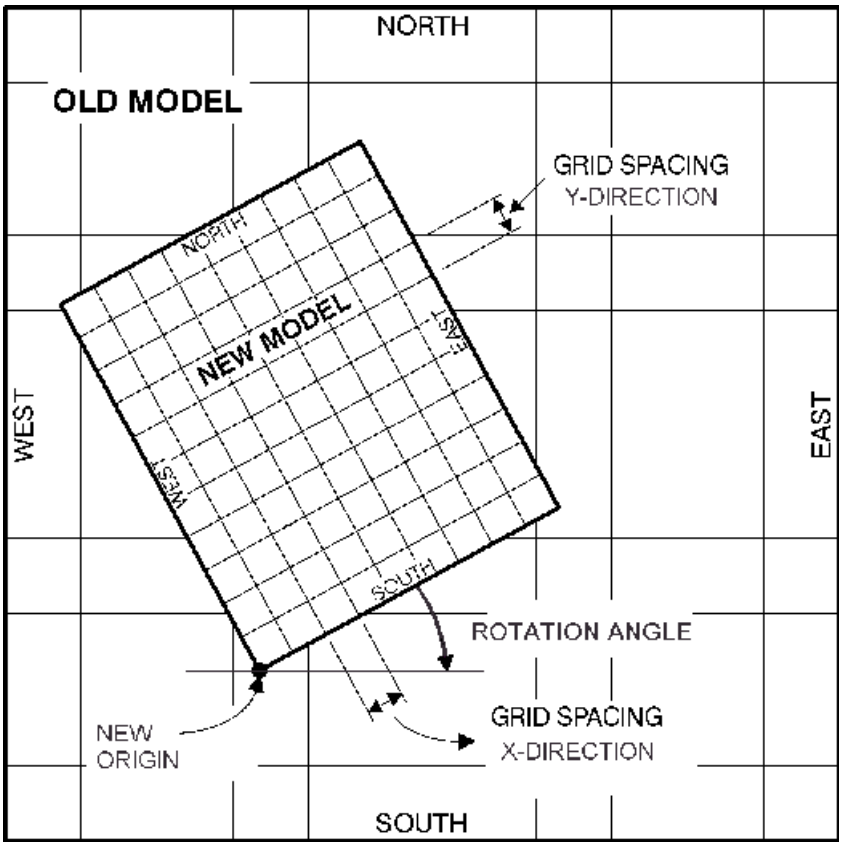


Figure 25.2 Definition of new grid

25.6 Land-Water Options and Values Outside Grid

To avoid making interpolation across land-water boundaries, i.e. interpolation that includes both water and land point values, you have to tell the tool how to distinguish water from land in your input data file:

- If your data does not have any land-water boundaries you should choose the first option,

- If you are interpolating a bathymetry you choose the second option, and
- If the data has the bathymetry written as a prefix item (e.g. simulation results) you should choose the last option.

The 'zland: Minimum value for true land' value is the value that separates land from water. The value is only relevant if land-water interpolation option is the second or third alternative. All grid points in the new grid which are placed within the land area of the old grid will be given the "zland" value.

If the new grid is not fully enclosed in the old grid, you can specify a value which will be uniformly set at all the exterior points of the old grid, in the new grid. This can either be the delete value, or a user specified value. Finally its also possible to specify values of exterior points relative to the old grid, in the new grid through a .dfs2-file. The file should in that case represent a grid having the same dimensions as the new grid which you specified earlier, but the file can contain any values for the exterior points of the old grid which you need to specify.

25.7 Output Specification

On this page you specify the name of the type 2 data file you are about to generate. A descriptive title can also be specified.

25.8 Status - Overview

The status page presents you a summary of all the input parameters you have specified. You should inspect that the parameters are correct and if not go back and change them.

From the page you can either *Finish* the setup which will transfer your setup definition to the current toolbox file or execute the setup.

INDEX

A
ADCIRC 43
ASCII format 106

D
Delete value 130
Delimiter 109
Derived item 129
Derived items 76, 129
dfs0 format 105

E
ESRI world file 35
Excel format 107

G
Geo-referencing 35

I
inverse distance weighting 91

M
M21_Misc 38

P
Polygon 59, 60
Polyline 59, 60

S
Shape file 59
SMS 44
Station location 87
Statistics 75

T
Template 108
Thiessen polygon 87, 89
Time description 109
TUFLOW 44

X
XYZ data 59