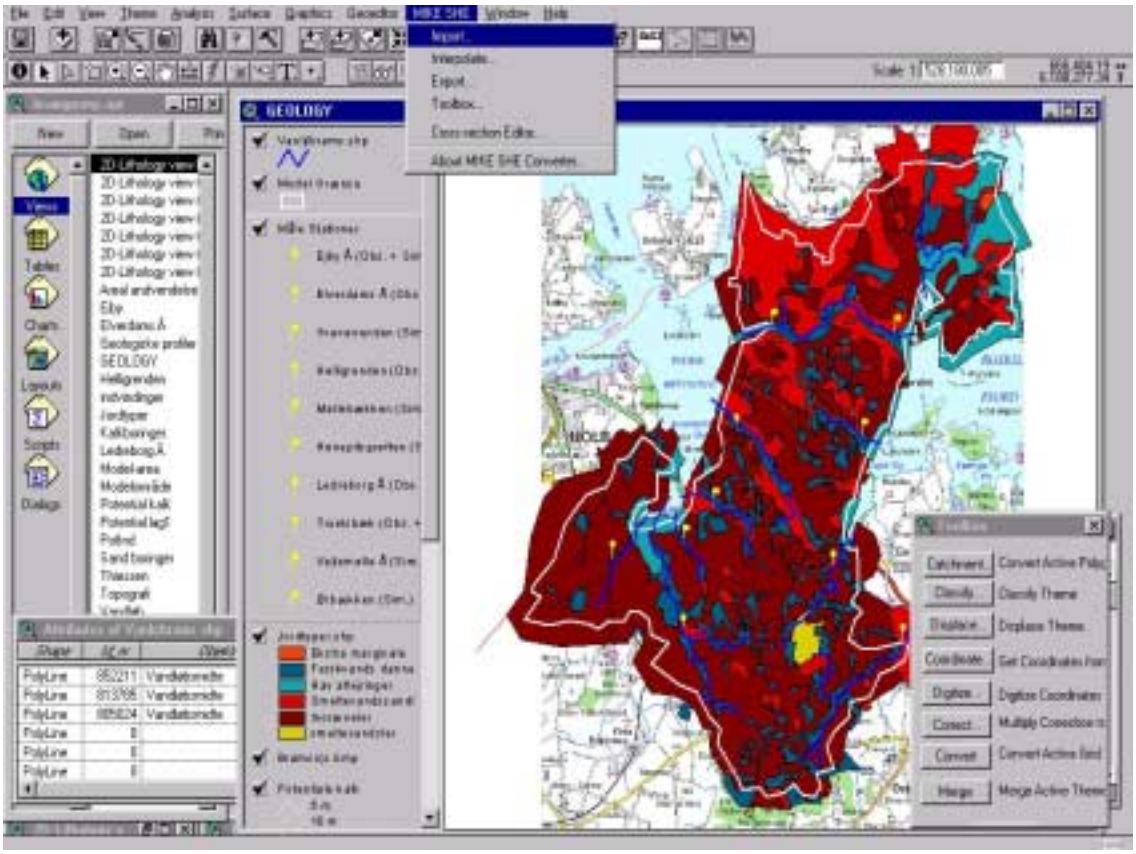
MIKE SHE Converter

User Manual



Conversion between
ArcView and MIKE SHE formats

DHI Water & Environment
May 2000

The MIKE SHE Converter (1999 B) is a product made by the DHI Water & Environment.

DHI Water & environment is a private, non-profit research and consulting organisation providing a broad spectrum of services and technology in offshore, coastal, port, river, water resources, urban drainage and environmental engineering.

The warranty given by DHI Water & Environment is limited as specified in your Software License Agreement. The following should be noted: Because programs are inherently complex and may not be completely free of errors, you are advised to validate your work. When using the program, you acknowledge that DHI Water & Environment has taken every care in the design of them. DHI Water & Environment shall not be responsible for any damages arising out of the use and application of the program and you shall satisfy yourself that the program provide satisfactory solutions by testing out sufficient examples.

The DHI Water & Environment could be contacted by:

DHI Water & Environment
Agern Alle 11
Dk- 2970 Hørsholm
Denmark

Phone: (+45) 45 16 92 00
Fax: (+45) 45 16 92 92
E-mail: software@dhi.dk
Home page: www.dhi.dk

Table of Content

ArcView – MIKE SHE Interface

Instruction for the ArcView MIKE SHE conversion extension (Version 1999 B). A tool for converting MIKE SHE files to Arc View format.

DHI Water & Environment, May 2000

1 GETTING STARTED

The user guide assumes that the user is familiar with the ESRI products Arc View version 3.1 or later and the extension Spatial Analyst 1.1. Basic Arc View or MIKE SHE functions are not addressed in this manual.

1.1 Requirements

The MIKE SHE converter requires ArcView 3.1 or later and the extension Spatial Analyst 1.1 or later.

1.2 Installation

If the MIKE SHE Converter is delivered as a setup program double clicking on the setup.exe file on the enclosed CD-ROM does the installation of the extension. During the setup the user is prompted for a shegis directory, if there is other MIKE SHE GIS products (e.g. GeoEditor) installed on the computer the same Shegis directory should be used. A Ext32 folder is created in the Shegis directory, the user will have to copy the DHI_MIKE_SHE_Converter.avx file to the Esri Ext32 folder in order to use the new extension file.

If the MIKE SHE Converter is delivered as avx files, these files should be placed in the EXT32 folder (On windows based machines this is usually c:\Esri\Av_gis30\Arcview\Ext32.), or in the userext folder, if this is created.

1.3 How to start MIKE SHE Converter

The MIKE SHE converter is designed as an Arc View Extension, the user should therefore load the extension in the Arc View. The extension menu is located in the project window file menu. In the extension menu select the MikeSheconverter. During the load procedure the extension Spatial Analyst will be automatically loaded,

if it is not already loaded. If the extension is not available the MIKE SHE Converter will fail to load.
The MIKE SHE Converter is dongle protected, if the correct dongle or licence file is not found the MIKE SHE Converter will fail to load (contact software@dhi.dk in case of any problems).

1.4 Menu

When the extension is loaded an additional option appears in the view document interface. The new menu called MIKE SHE is a drop down menu, when selected the MIKE SHE converter main menu will be shown on the screen.

Important Note: If the user is using the program Exceed, it is recommended not to use the directory c:\exceed as the working directory in ArcView, because this may cause the instabilities when the extension is run. . The working directory is changed under the File menu, in the View.

2 MAIN MENU

The extension will modify the view document interface. It will add one more menu called MIKE SHE. This menu will contain a menu item, when selecting this the main menu will appear.

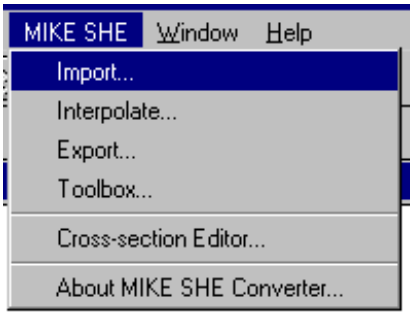


Figure 1 MIKE SHE menu

Import: Opens the Import dialog box, enabling the user to import MIKE SHE files.

Interpolate: Loads the Interpolate extension enabling the user to interpolate discrete values to surface maps.

Export: Opens the Export dialog box, enabling the user to export ArcView files to MIKE SHE format.

Toolbox: Opens the toolbox dialog.

Cross-section Editor: Loads the Cross-section Editor extension enabling the user to view and edit a MIKE11 cross-section file.

3 IMPORT

The Import menu enables the user to import T2, DIG and NWK11 files to an ArcView format.

Figure 2 Import dialog box

3.1 Import T2

The I**mport T2 file** is always enabled. Once the menu item is selected the user will be prompted the **MIKE SHE Import T2 file** dialog box. The import T2 option enables the user to import a MIKE SHE T2 file to ArcView, the T2 file will appear in ArcView as an ArcView grid theme.

3.1.1 Recommended procedure

- Select the T2 file to be imported.
- If the user wants the ArcView grid file, associated with the ArcView grid, to be saved at a user defined location specify the Grid Directory.
- Specify a xy-unit conversion if needed.
- Specify a z-multiplier to be multiplied the grid values.
- If multiple files should be imported select the **Multiple Files Import** checkbox, and use the **Add file to list** and **Delete Selected File** to add and remove files from the list.
- Press the **OK** button to start the import procedure.

3.1.2 Content of the MIKE SHE Import T2 file

Input T2 file: The user specifies the input T2 file, by clicking at the browse button.

Grid directory: If the user wants the ArcView grid file, associated with the ArcView grid to be saved at a user defined location the Grid

Directory must be specified. By default the grid file is saved in the working directory.

XY-unit conversion: To convert the xy-units from one unit to another, select From **unit** (the current unit) and To **unit** (the desired unit). The MIKE SHE Converter will then calculate a multiplier for the origin and the cellsize.

Z Multiplier: The user can specify a constant that will be multiplied on the Z-values in the entire grid.
Multiple Files Import: Enables import of multiple T2 files. When the **Multiple Files Import** is clicked on, the control associated with multiple file import will be enabled.

Add file to list: Ad a T2 file specified in **Input T2 file** to the multiple file list. The MIKE SHE Converter will check for; multiple files, extension equal to T2 and if the file exist, before adding the file to the list.

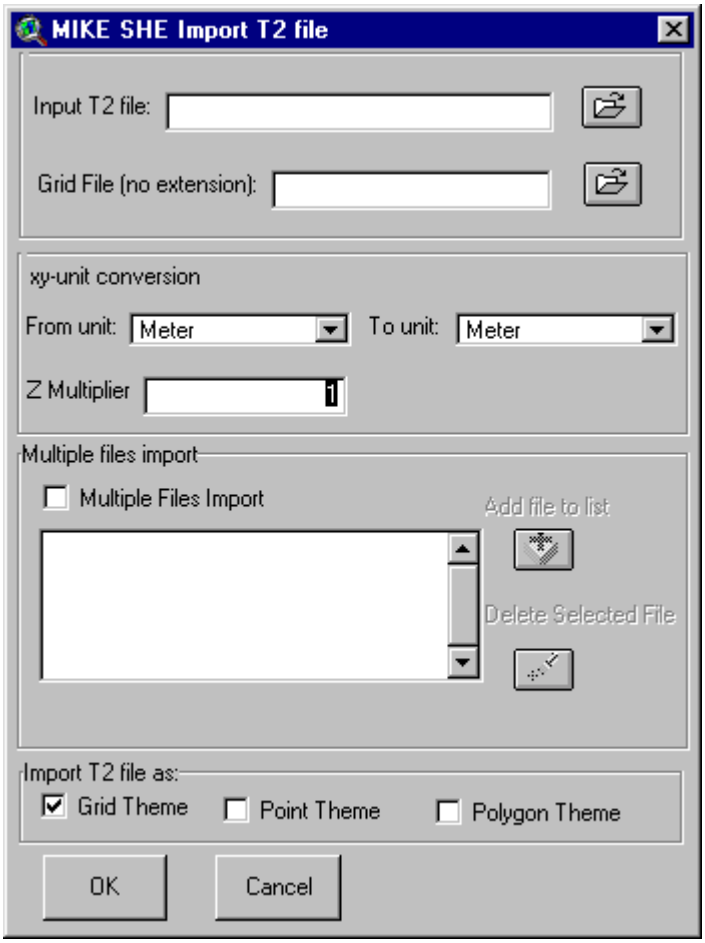


Figure 3 Import T2 file dialog

Delete Selected File: Remove a file from the list by selecting the file and clicking the **Delete Selected File** button. When the user selects a file in the list the filename will be shown in the textbox at the bottom of the dialog box.

Grid Theme: Adds the T2 file to the view as an ArcView grid file (default option).

Point Theme: Adds the T2 file to the view as a point theme, where the points represents the centre of the cell.

Polygon Theme: Adds the T2 file to the view as a polygon theme, where the polygon represents areas of equal values.

The selected file will be added as a visible grid theme to the current view.

When importing a T2 file with a direction that differs from 0 (meaning that it do not have a

Figure 4 Original MIKE SHE grid (orientation = 315°), and after converting to ArcView (orientation = 0° or N-S).

North-South direction), the MIKE SHE Converter will generate a new grid with a North-South direction, containing the old grid. During this process the extension will interpolate some new cell values using a bilinear interpolation, this could result in some small numerical errors along the grid border.

If this grid is later converted to a T2 format, the extension will save it with an N-S direction, to avoid one more interpolation, which means that the output grid probably will be bigger than the input grid. But the surplus cells will only contain No-Data values.

3.2 Import DIG

The **Import DIG** option is accessible through the main menu.
The Import DIG option enables the user to import a MIKE SHE DIG file to ArcView.

Figure 5 DIG file properties dialog

Choose the Import DIG option in the MIKE SHE menu. The user will be queried for a DIG file to import, and an output name for the output shapefile.

The user will be prompted the UTM-zone, the unit and which kind of type the DIG file belong to. The user can also specify a xy-unit converter, in order to change the units (e.g. from feet to meters), and a z-multiplier to change the unit of the values. Selecting the 'Numeric Values Only' checkbox imports all the additional data in the DIG file as numeric values. The default setting is to import the comment column in the DIG file as string values.
Pressing OK will add the DIG file to the active view as a theme, pressing cancel will stop the conversion.

3.3 Import NWK11

The **Import NWK11** option is accessible through the main menu. The Import NWK11 option enables the user to import a MIKE 11 network file to ArcView.

When the Import NWK11 option is selected the Import NWK11 file dialog box will appear.

3.3.1 Recommended procedure

- Specify the NWK11 file to be imported
- Change the output shape filename. Default the filename for the nwk11 file with the extension shp instead of NWK11.

- Specify the unit conversion.
- Press **Import** to import the NWK11 file to ArcView.

3.3.2 Content of the Import NWK11 file dialog

NWK11 file: Specify the MIKE11 network file to be imported to ArcView. Specify the filename by typing or by using the browse button.

Output shape file: The output shape file will by default be the same as the NWK11 file, only the extension will be shp instead of nwk11. Change the filename by typing or by using the browse button.

XY-unit conversion: To convert the xy-units from one unit to another, select the from unit (the current unit) and the to unit (the desired unit). The MIKE SHE Converter will then calculate a multiplier that will be multiplied on the origin and the cell size.

Import: Import the specified MIKE11 network file. The MIKE11 network file will be displayed in ArcView as a polyline theme.

Cancel: Closes the current dialog and opens the main dialog.

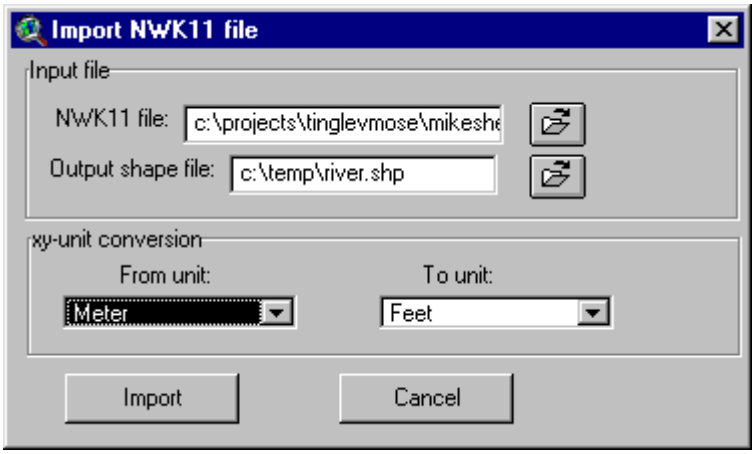


Figure 6 Import NWK11 file dialog

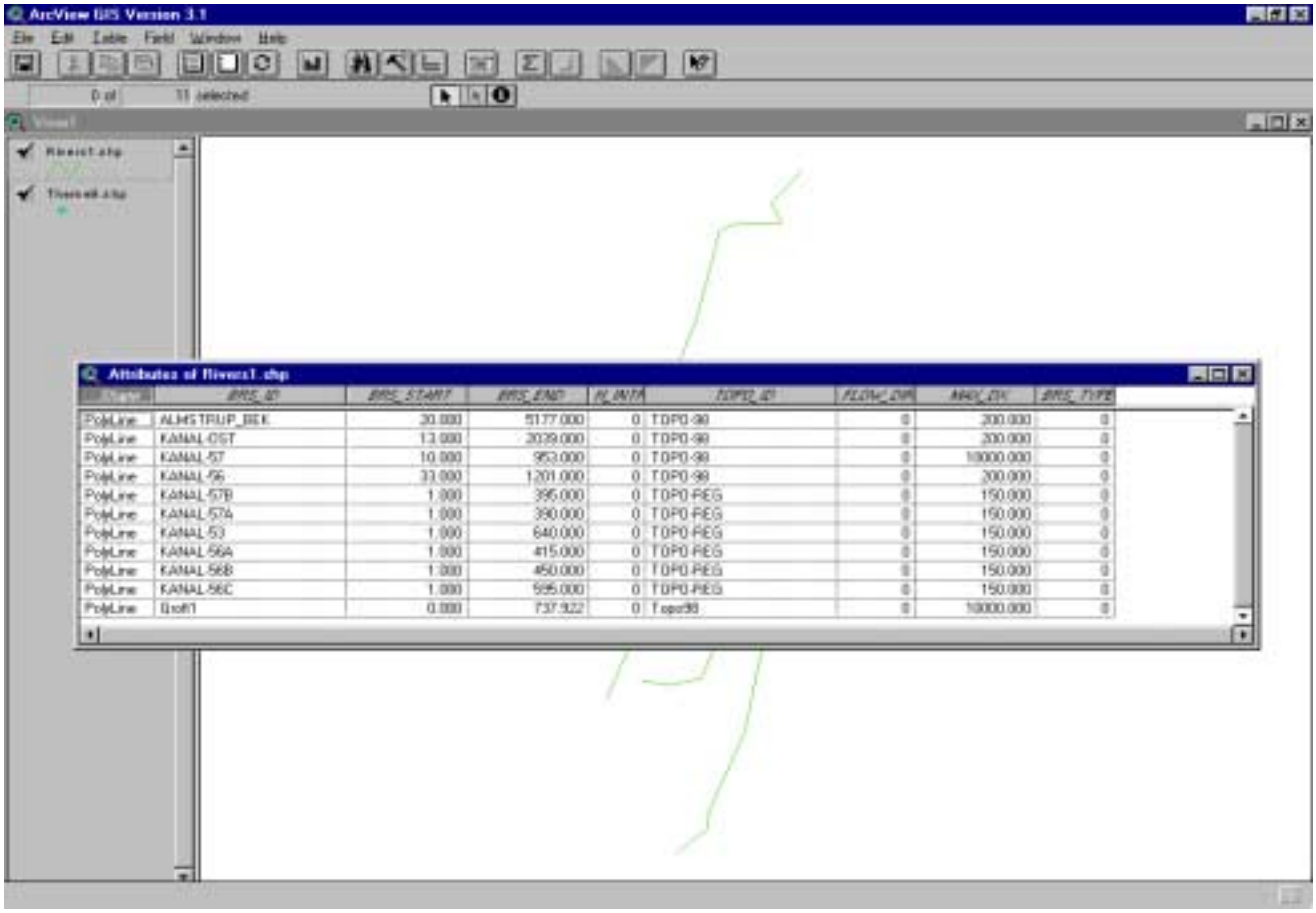


Figure 7 Format of imported NWK11 file

4 INTERPOLATE

The **Interpolate Surface** option is accessible through the main menu. This option enables the user to interpolate a surface from a point, polyline or polygon theme.

The **Interpolate Surface** part is made as a separate extension using Arc View interpolation routines. When the user selects the **Interpolate Surface** option from the main menu the MIKE SHE Converter will look for the Interpolate.avx file. If it is present it will be loaded and the **Interpolate Surface** dialog box will appear, if it is not present this option will be unavailable.

4.1 Recommended Procedure

- Select the Input theme, point, polyline or polygon theme. By default the active theme will be selected.
- Select the interpolation field.
- Define the output grid. Grid settings can be loaded from a T2 file, a grid definition ASCII file or given directly in the dialog box.
- Select a barrier theme (line theme), select none if no barrier theme is used. This theme will act as a barrier in the interpolation.
- Select the Mask Grid (grid theme), select none if no mask grid is used. When a mask grid is defined all the cells outside this grid will be given a delete value.
- Press the **Open Interpolation setting** button to set the interpolation settings, to review or change the interpolation method.
- Press the **Interpolate Surface** button to start the interpolation. The interpolated surface will be added to the active view as a grid theme.

4.1.1 Content of the Interpolation properties dialog

Input Theme: Select the input theme from the list, only point, polyline or polygon theme will be listed. By default the active theme will be selected. Polyline or polygon theme will during the conversion be converted to a point theme used in the conversion.

Interpolation Field: Select the field from the selected input theme containing the grid values.

X origin, Y origin: The x and y-coordinate of the lower left corner of the output grid.

Columns, Rows: The number of columns and rows in the output grid.

Cell Size: The output grid cell size dimensions.

Output Extent: Enables the user to select the same grid extent as the view, the display, or one of the themes in the view.

Save grid properties: Saves the set-up values to a grid definition ASCII file.

Load grid properties: Loads set-up from a grid definition ASCII file.

Load settings from T2 grid: Opens a browse dialog box for selecting a T2 file. The header values from the T2 file are extracted and imported to the dialog box.

Show grid: Displays a graphical presentation of the grid set-up.

Output grid: The name of the output grid theme.

Barrier Theme: Enables the user to select a line theme as a barrier theme. A barrier theme will act as a barrier in the interpolation, as no interpolation will occur across the line defined by the barrier theme.

Mask Grid: Enables the user to select a grid theme as mask grid. A mask grid defines the data extent of the grid. All cells outside the mask grid will be given the no-data value.

Open Interpolation settings: This will open the Interpolation settings dialog box.

Interpolate Surface: This will start the interpolation. The interpolated surface will be added to the active view as a grid theme.

Close: Closes the dialog.

The format of the T2 file and the ASCII Grid definition file are given in Appendix 1.

4.2 Open Interpolation Settings

This dialog box enables the user to select the interpolation method to use, and to set the settings for the selected method. When selected the interpolation menu will appear. This option allows the user to control the interpolation routine and selected appropriate data. Since detailed explanation of the interpolation routines are given in the Arc View

manual only a brief discussion of the different menu option are given below.

4.2.1 Recommended Procedures

- Specify the needed parameters
- Select the interpolation method to use.

4.2.2 Content of the Interpolation control dialog

Thiessen Polygons:
The Thiessen polygons are calculated using the Grid.EucAllocation request.
Euclidean allocation produces an output Grid that records for each cell the identity of the closest source cell. Each cell in an allocation receives the value of the source to which it is allocated.
In the source Grid, the sources are all of the cells with values other than delete values and all Euclidean functions are calculated from non-source cells assigned delete values. The value 0 is considered a legitimate source. Source cells that are masked with the mask of the AnalysisEnvironment will not be considered in the computations. These cell locations are assigned delete values.

Use buffered area: Defines the maximum distance that can be used to allocate a cell to a source. If a cell is further away than the defined distance to the closest cell with a value other than a delete value, the cell will be given the a delete value

Inverse Distance:
Inverse distance weighted (IDW) interpolation determines cell values using a linearly weighted combination of a set of sample points. The weight is a function of inverse distance. The surface being interpolated should be that of a locationally dependent variable.
IDW allows you to control the significance of known points upon the interpolated values, based upon their distance from the output point. The interpolation can be shifted from local to global by changing the power. Figure 8A larger power will result in less influence from surrounding points, i.e., nearby data will have the most influence, and the surface will have more detail (be less smooth). The power is a positive, real number. A common value is 2.

The barrier theme argument is used to specify the location of linear features known to interrupt the surface continuity. These features do not have z values. Cliffs, faults, or embankments are typical examples of barriers. Barriers limit the set of the input sample points used to interpolate output z values to only those samples on the same side of the barrier as the current processing cell. Input sample points that lie exactly on the barrier line are included in the sample set for both sides

of the barrier. When barriers are specified, the processing time is significantly extended.

Spline:
The basic form of the minimum-curvature Spline interpolation imposes the following two conditions on the interpolant:

- The surface must pass exactly through the data points.
- The surface must have minimum curvature-the cumulative sum of the squares of the second derivative terms of the surface, taken over each point on the surface, must be a minimum.

The **Regularized** option modifies the minimization criterion so that the third derivative terms are incorporated into the minimization criteria.

The **Weight** argument specifies the weight attached to the third derivative terms during minimization, referred to as tau in the literature. Higher values of this term lead to smoother surfaces. Values between 0 and 0.5 are suitable. Using the Regularized option ensures a smooth surface together with smooth first-derivative surfaces. This technique is useful if the second derivative of the interpolated surface needs to be computed.

The **Tension** option modifies the minimization criterion so that first-derivative terms are incorporated into the minimization criteria.

The **Weight** argument specifies the weight attached to the first-derivative terms during minimization, referred to as phi in the literature. A weight of 0 results in the basic thin plate Spline interpolation. Using a larger value of weight reduces the stiffness of the plate, and as j approaches infinity the surface approximates the shape of a membrane or rubber sheets passing through the points. The interpolated surface is smooth. First derivatives are continuous but not smooth.

The **Regularized** option usually produces more smooth surfaces than those created with the Tension option. For the Regularized option, higher values used for the weight parameter produce smoother surfaces. The values entered for this parameter must be equal to of greater than 0. The typical values that may be used are: 0, 0.001, 0.01, 0.1, and 0.5. However, for the Tension option, higher values entered for the weight parameter results in somewhat coarser surfaces, but surfaces that closely conform the control points. The values entered have to be equal to or greater than 0. The typical values are: 0, 1, 5, and 10.

Trend:
The linear trend surface interpolator creates a floating-point Grid. It uses a polynomial regression to fit a least-squares surface to the input points. It allows you to control the order of the polynomial used to fit the surface. Trend interpolation is easy to understand by considering a first-order polynomial. A first-order linear trend surface interpolation simply performs a least-squares fit of a plane to the set of input points. Trend surface interpolation creates smooth surfaces. The surface generated will seldom pass through the original data points since it performs a best fit for the entire surface. When an order higher than 1 is used, the interpolator may generate a Grid whose minimum and maximum might exceed the minimum and maximum of the input points.

As the **order** of the polynomial is increased, the surface being fitted becomes progressively more complex. A higher order polynomial will not always generate the most accurate surface, it is dependent upon the data. The lower the RMS error, the more closely the interpolated surface represents the input points. The most common order of polynomials is 1 through 3.

Kriging:
Kriging is an advanced interpolation procedure that generates an estimated surface from a scattered set of points with z values. Unlike the other interpolation methods supported by the Spatial Analyst, Kriging involves an interactive investigation of the spatial behaviour of the phenomenon represented by the z values before you select the best estimation method for generating the output surface.

Kriging is based on the regionalized variable theory that assumes that the spatial variation in the phenomenon represented by the z values is statistically homogeneous throughout the surface; that is, the same pattern of variation can be observed at all locations on the surface.

The spatial variation is quantified by the semi-variogram. The semi-variogram is estimated by the sample semi-variogram that is computed from the input point data set. The value of the sample semi-variogram for a separation distance of h (referred to as the lag) is the average squared difference in z value between pairs of input sample points separated by h.
The semi-variogram is modelled by fitting a theoretical function to the sample semi-variogram.

The Kriging request offers two types of surface estimators: **Ordinary Kriging** and **Universal**. **Ordinary Kriging** represented by the **SPHERICAL, CIRCULAR, EXPONENTIAL, GAUSSIAN, and LINEAR** methods. Kriging uses the mathematical function specified

with the method argument to fit a line or curve to the semi-variance data in the semi-variogram.

Ordinary Kriging assumes that the variation in z values is free of any structural component (drift). The five optional models are provided to ensure the necessary conditions of the variogram model are satisfied, which is not always possible with interactive "manual" variogram fitting.

Universal Kriging, represented by the **UNIVERSAL1** and **UNIVERSAL2** methods, assumes that the spatial variation across the surface also has a structural component referred to as drift. Drift is a systematic change in the cell values at a particular scale. This scale is related to the radius of the search area. The goal is to change the search radius to find the scale at which the drift can be detected and the variance is lowest. Therefore, it is not recommended that a variable Radius object be created for Universal Kriging. UNIVERSAL1 uses a first order polynomial to approximate the drift, and UNIVERSAL2 uses a second order.

Kriging is very computer-intensive process. The speed of execution is dependent on the number of points in the input data set and the size of the search window.

The **barrier** argument is used to specify the location of linear features known to interrupt the surface continuity. These features do not have z values. Cliffs, faults, or embankments are typical examples of barriers. Barriers limit the set of the input sample points used to interpolate output z values to only those samples on the same side of the barrier as the current processing cell. Input sample points that lie exactly on the barrier line are included in the sample set for both sides of the barrier. When barriers are specified, the processing time is significantly extended.

5 EXPORT

The Export menu allows the user to export ArcView files to MIKE SHE format.

Figure 9 Export dialog box

5.1 Export to T2

Choose the **Export to T2** option in the MIKE SHE menu, this option will only be enabled if there is an active theme in the view.
The **Export to T2** option enables the user to export an ArcView theme to a MIKE SHE T2 file.

5.1.1 Recommended procedure

To export an Arc View theme to a T2 file, a grid setup must be defined first and secondly the theme properties must be defined.

5.1.2 Grid set-up

When selecting the **Export to T2** option from the main menu the user will be prompted the dialog box shown in Figure 2.
The dialog box will contain the initial settings for the chosen theme. If the theme is a grid theme the initial settings will be the actual settings for the grid, but if the theme is a feature theme then the actual settings are non-existing. The extension will then use some default values (rows, columns and cell size equal to 100), the x and y origin will be determined by making a rectangle around the theme and then using the coordinates from the lower left corner.

This enables the user to define the output grid. The output grid will be displayed on the screen as a graphical presentation.

5.1.3 Recommended procedure

- Press the **Show Grid** button to se a graphical display of the grid.
- Change the grid settings by typing new values and pressing the **Show grid** button.
- Save grid settings to an ASCII file by pressing the **Save grid settings** button.
- To load a grid set-up from a file, select the **Load grid settings.**.
- Press the **OK** button to proceed.

The format of the grid definition ASCII file and the T2 file are described in Appendix 1.

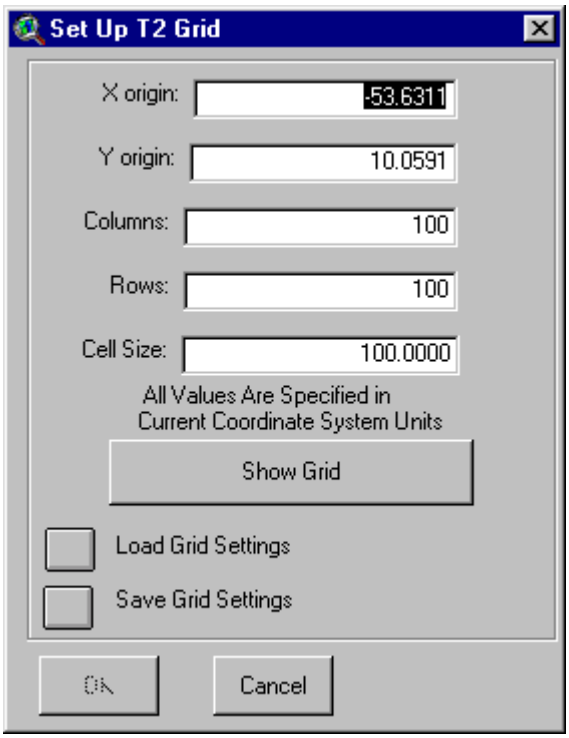


Figure 10 Set up output grid

5.1.4 Content of the Set Up T2 grid dialog

X origin, Y origin: Shows the x and y coordinate of the lower left corner.

Columns, Rows: Shows the number of columns and rows of the selected grid.

Cell size: Shows the cell size. Only grid squares are supported.

Save grid settings, Load grid settings: Save and load grid set-up from file.

Show grid: Displays a graphical presentation of the grid set-up.

OK: Closes this dialog and opens the Theme Properties dialog

Cancel: Closes the dialog box and cancel the export procedure.

5.1.5 Theme properties

When the user has defined an output grid the theme properties dialog box will appear, this will enable the user to specify the properties of the conversion.

5.1.6 Recommended procedure

- Select the output file name.
- Select the conversion option.
- Specify the grid-code field, if enabled.
- Specify a XY-unit conversion if needed.
- Specify a Z-multiplier if needed.
- Press the **OK** button to start the conversion.

5.1.7 Content of the Theme properties dialog

Current Theme: This text box will have the name of the current theme to be converted to a T2 file.

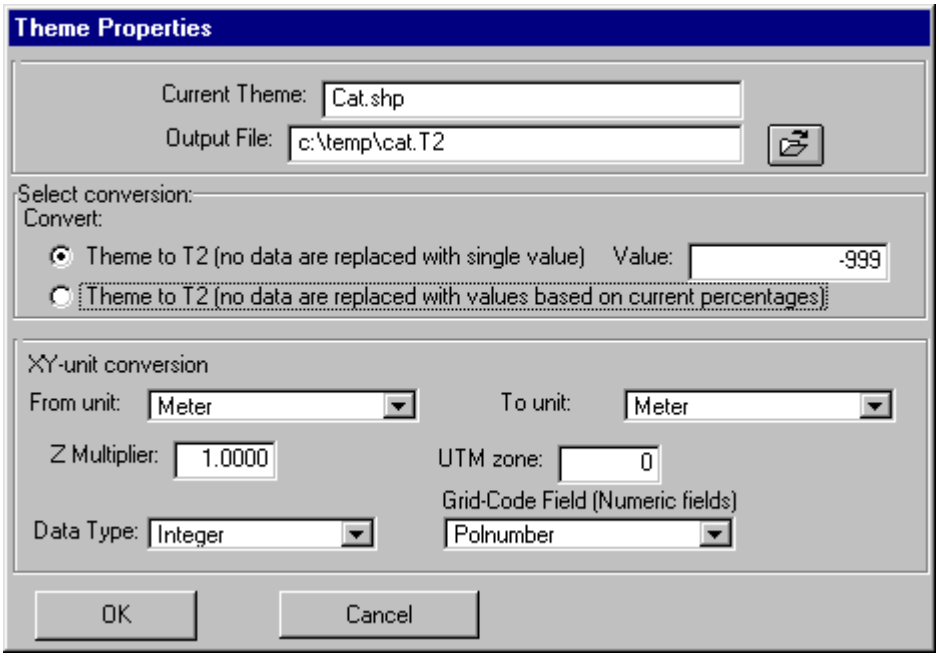


Figure 11 Theme Properties Dialog

Output File: The second text box specifies the output directory, and output filename. Use the browse button to change the directory and filename. The user don't have to type the extension T2 after the file name, although it not is shown in the menu it is automatically added by the extension.

XY-unit conversion: To convert the xy-units from one unit to another, select the from unit (the current unit) and the to unit (the desired unit). The MIKE SHE Converter will then calculate a multiplier that will be multiplied on the values in the T2 header.

Z Multiplier: The user can specify a constant that will be multiplied on the Z-values in the entire grid.

UTM zone: This value will be written in the header of the T2 file.

Grid-Code Field (Numeric fields): Specifies what field the grid should take the numeric values from.

Data Type: The user can specify if the data are float or integer data.

Theme to T2 (no data is replaced with single value): This option converts a theme to a T2 file and replaces all the no data values with a single value. This single value are optional, and could be typed in the Value text box (the default value are –1E-035).

Theme to T2 (no data is replaced with values based on current percentages): This option converts themes to T2 and fills the areas of no data with values that are based on the distribution of the existing data in the study area. If the current theme is a float raster theme, the created data will not be a random distribution of data, but instead based on the normal distribution. (It is not recommended to use this option for float data grid). This option is not optimised and the conversion could take some time.

OK: Starts the conversion.

Cancel: Closes the dialog box and abort the export procedure.

Note: The extension can handle both ArcView grid and feature-themes. If the user wants to convert an ArcView feature-theme to T2 format, the extension will do a rasterisation of the theme before converting it to a T2 format.

5.2 Export to DIG

The **Export to DIG** option is accessible through the main menu.

The Export to DIG option enables the user to export point, polyline or polygon themes to MIKE SHE DIG file.

Note: When exporting a PolyLineZ theme the z-coordinate will be used if the user selects <None> in the **Select field for z-coordinate**.

5.2.1 Recommended procedure

- Select the theme to be exported.
- Select the DIG type to export to. The available types will depend on the type of source theme.
- Specify the unit conversion, if any.
- Specify the output file.
- Select the fields for the selected Dig type.
- Press the **OK** button to export the theme.

5.2.2 Content of the Dig export dialog

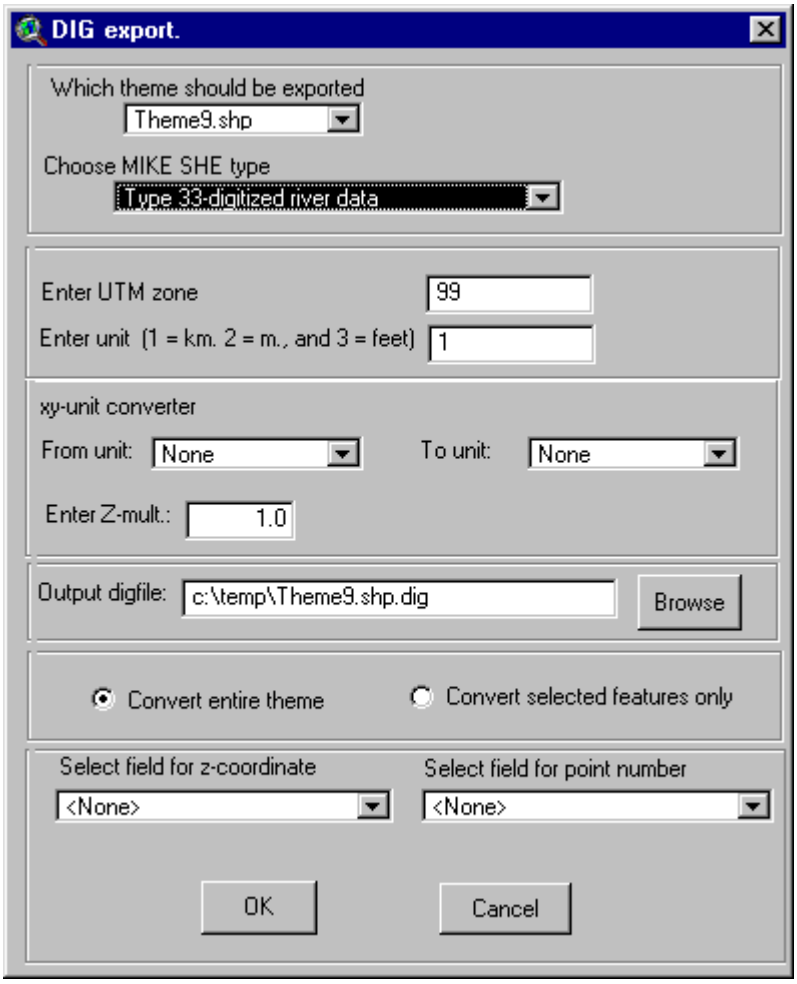


Figure 12 Export to DIG dialog box

Theme to be exported: The user should choose which theme to be exported, only the active themes are listed in the box. The name for the output DIG file will then change according to the name of the selected theme.

Output DIG file format: Will give a list of MIKE SHE DIG file types that the extension can handle:
Type 31-digitized contours (code, x, y, z)
Type 31-digitized points (code (=3), x, y, z, point number)
Type 32-digitized polygons (code, x, y, polygon number)
Type 33-digitized river data (points old format) (x, y, z, point number)

The user should then choose which type the theme belongs to.

UTM zone: Type the UTM zone for the theme. This information will only be written to the DIG header file.

Unit: Enter the appropriate unit (enter 1 for meter, 2 for kilometre and 3 for feet). This information will only be written to the DIG header file.

XY-unit conversion: To convert the xy-units from one unit to another, select the from unit (the current unit) and the to unit (the desired unit). The MIKE SHE Converter will then calculate a multiplier that will be multiplied on the origin and the cell size.

Enter z multiplier: Enter a z multiplier, this factor will be multiplied the codes, e.g. the levels for a type 31 file.

Output digfile: Specifies the output directory and filename.

Convert entire theme: If this option is activated, the entire theme will be converted to a DIG file.

Convert selected features only: If this option is activated, only selected features of the theme will be converted to a DIG file. If this button is activated when no features are selected the entire theme will be converted.

Select field for …: Here the user should select the appropriate fields for the attributes, connected to the chosen file type. This option will change according to the chosen file type (e.g. if the user chooses file type 32 (digitised polygons) then the user will only be prompted to select the field for the polygon number).

When selecting an output type the following rules are applied:

1) A point theme can only be exported to a point theme (type 31).
2) A line theme can be exported to points and to contours (type 31 and 33).
3) A polygon theme can be exported to points, contours and polygons (type 31, 32 and 33).

After pressing **OK** the DIG file will be saved at the desired location.

5.3 Export to NWK11

The **Export to NWK11** option is accessible through the main menu.

The Export to NWK11 option enables the user to export a point or polyline theme to a MIKE11 network file (NWK11 file). When the option is selected the **Export to NWK11** dialog box will appear.

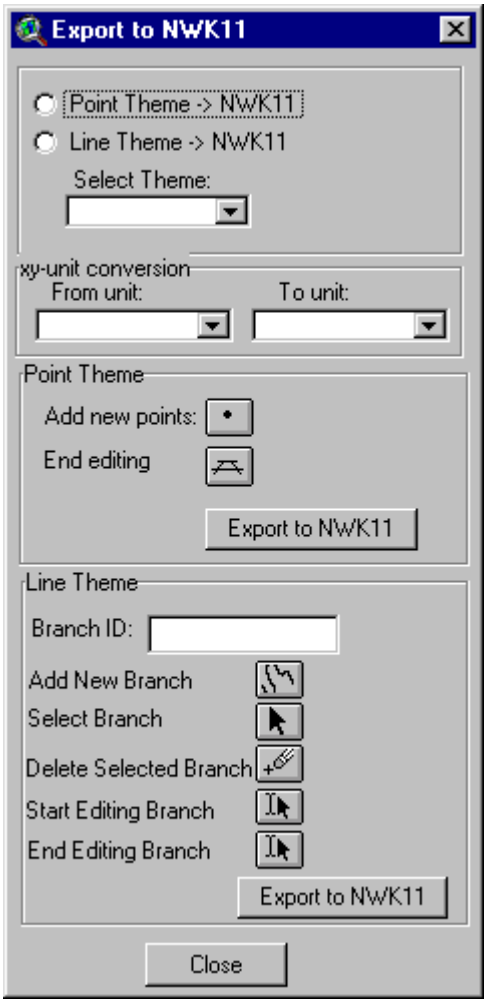


Figure 13 Export to NWK11 dialog

5.3.1 Recommended procedure

- Select point theme to NWK11 or Line theme to NWK11.
- Specify the unit conversion, if any.

Point theme -> NWK11

- Select the theme from the **Select Theme** list. Select **New** to create a new theme.
- Use the **Add new points** toll to add new points to the theme.
- Press the **End editing** button to end editing on the point theme.
- Select the **Export to NWK11** to export the point theme to a NWK11 file. The user will be prompted to specify the output file.

Line theme -> NWK11

- Select the theme from the **Select Theme** list. Select **New** to create a new theme.
- Use the **Add New Branch** button to add a new branch to the theme. The branch name will be taken from the **Branch ID** text box.
- Use the **Select Branch** tool to select a branch. The ID for the selected branch will be shown in the **Branch ID** text box.
- Use the **Delete selected Branch** tool to delete the selected branch.
- Use the **Start Editing Branch** tool to edit branches.
- Use the **End Editing Branch** tool to save the edited branch.
- Select the **Export to NWK11** to export the line theme to a NWK11 file. The user will be prompted to specify the output file.

5.3.2 Content of the Export to NWK11 dialog

Point Theme -> NWK11: Select this to export a point theme to a MIKE11 network file. The network file will only contain the points, each with a dummy ID.

Line Theme -> NWK11: Select this to export a line theme to a MIKE11 network file. The format of the line theme could be seen on figure 13.

Select Theme: Select the theme to export. Select **New** to create a new theme, the user will then be prompted to specify a filename for the new theme. The themes in the list will reflect the selection of line or point theme, e.g. if the user has selected line theme only the line themes in the view will be shown.

XY-unit conversion: To convert the xy-units from one unit to another, select the from unit (the current unit) and the to unit (the desired unit). The MIKE SHE Converter will then calculate a multiplier that will be multiplied on the origin and the cell size.

Add new points: Allows the user to add new points to the point theme. This tool is only enabled if a point theme is selected.

End editing: Stop editing of a point theme. Saves the new points added to the point theme. This tool is only enabled if a point theme is selected.

Export to NWK11: Exports the selected point theme to a MIKE11 network file. The user is prompted to specify a filename for the nwk11 file.

Branch ID: Shows the branch ID for a selected branch or for a new branch. This tool is only enabled if a line theme is selected.

Add new Branch: Adds a new branch to the selected line theme. Click on the screen to define the branch, double click to end the branch. The branch ID will be taken from the **Branch ID** text box.

Select Branch: Click on a branch to select it, and the branch ID will be shown in the **Branch ID** text box.

Delete Selected Branch: Deletes a selected branch.

Start Editing Branch: Puts the line theme in editing mode, enabling the user to edit the branches.

End Editing Branch: Saves any changes to the line theme.

Export to NWK11: Export the line theme to a NWK11 file. The user will be prompted to specify the output file.

Close: Closes the current dialog.

6 TOOLBOX

The MIKE SHE Converter Toolbox contains some different tools for use while working with MIKE SHE files in the ArcView environment.

Figure 14 Toolbox Dialog

Catchment: Convert active polygon theme to a catchment grid (MIKE SHE catchment format).

Classify Theme: Classifies a theme after a selected field. The output is a new field containing a code for each string type in the selected field, eg. selected field contains (a, b, a, c), the output field will then contain (1, 2, 1, 3)).

Displace: Displace a theme by specified x- and y-coordinates.

Coordinate: Get the coordinates for a point theme. Adds a x- and y-field to the point theme containing the coordinates for the points in the theme.

Digitise: Digitise elevation values from a inage map.

Correct: Add a correction factor to a MIKE SHE T0 file.

Convert: Converts the active grid theme to a point theme.

Merge: Merge the active theme to a merged theme.

Image: Specify the four corner coordinates for a image theme to generate the world file automatically.

Aggregate: Aggregates several polygon themes to a aggregated polygon themes

7 CROSS SECTION EDITOR

The Cross section Editor is made as an external extension. When the user selects the **Cross Section Editor** from the main menu, the MIKE SHE Converter will look in the Ext32 folder and then in the USEREXT folder for the avx file containing the Cross Section Editor. If the extension is not installed in the correct folders it will not be available for the user.

The Cross Section Extension will add a new menu called CrossSection Editing to the view, and a tool for selecting crosssections.
The cross section editor requires that the branch themes are belonging to the PolyLineM class, so if it detects a branch theme of the PolyLine class it will ask the user if the theme should be changed to the new format. If the user answers positive to that a new theme will be created and the old theme is removed from the project.

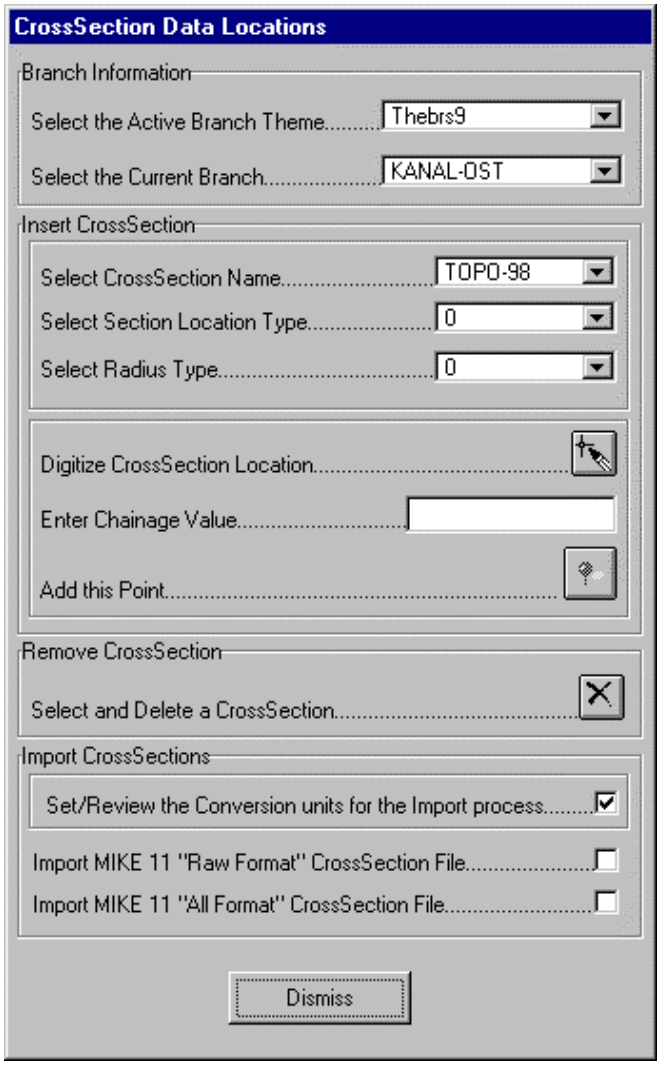


Figure 15 Data location dialog

7.1 Adding Cross-Section Interactively

To import a MIKE11 ASCII formatted cross section file select the "Import MIKE11 File" option, both the raw and the all format are implemented in the Cross Section Editor.

When adding cross sections interactively, the user begins this process by selecting the branch for the CrossSection. Then the user can enter the chainage value or the location along this branch for the CrossSection. He will then click the button for committing this point. At this time the CrossSection Editor appears with only three points representing the left most point, the point with the lowest elevation and the right most point. At this time the user needs to commit the new CrossSection by clicking on the button titled "Add this Point". This adds this new point to the theme. This process enables the "Open CrossSection View" button. When the user clicks on this button, a

CrossSection View opens with the new CrossSection displays. At this time the user can edit this CrossSection in the same manner that he might have edited previous CrossSections. The user establishes the editing environment by using the right mouse button with the cursor in the view's display area and clicking "Start Editing CrossSection". When the user is finished editing the vertices of the CrossSection, he should again use the right mouse button to access the popup menu. This time he will see a choice "Stop Editing CrossSection".

7.2 Branch Attributes Dialog

The Branch Information Dialog provides the user with an interactive method for accessing the attributes for the selected branch of the active

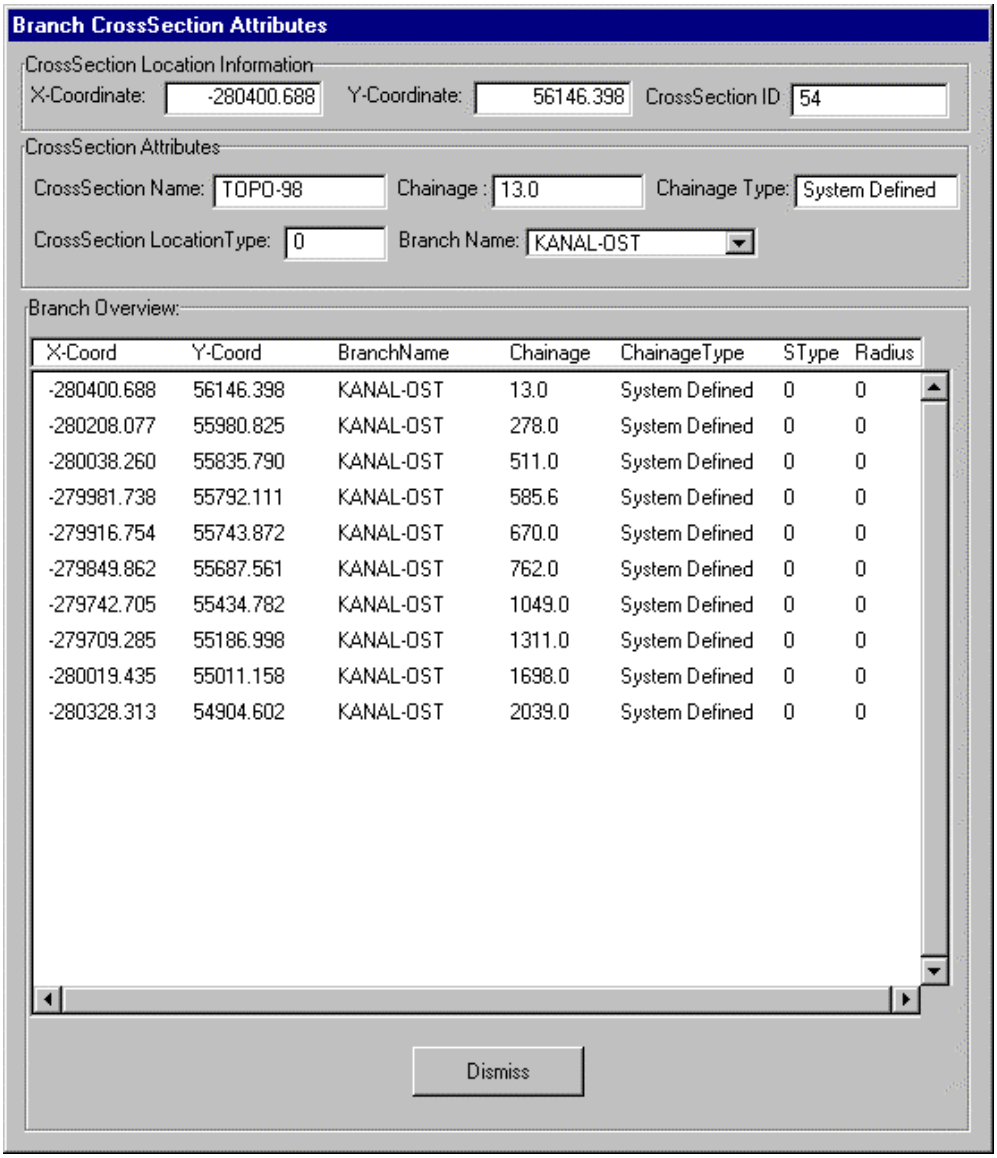


Figure 16 Branch Attributes dialog

Branch theme. The dialog is accessed by placing the cursor in the Branch view and clicking the right mouse button. Here the user selects “View Branch Tabular Data”. This requires that there is a selected branch in the Branch theme. The user will be notified if there is not. The user should use the “Branch Selecting Tool” to select the branch of interest. When the dialog opens there is no selected CrossSection. To select a CrossSection the user can either use the “CrossSection Point Selecting Tool” and points to a CrossSection point along the branch or selects the line in the overview list. Depending on the density of CrossSection points along a branch, the user might benefit by using the “Zoom to Selected” button or any of the other “Zoom” controls during the selection process. In the top panel of the dialog the user is shown the X-Coordinate and Y-Coordinate of the selected CrossSection. The Branch name, Chainage type, Chainage value and the CrossSection Location type are also shown. The Branch Name combo box allows the user to select a new branch. The Branch view then updates itself by selecting this branch and zooming to its extent. This list box at the bottom of the dialog shows the attributes for all of the CrossSections associated to this branch in a row and column format. In this box, a single mouse click will select the corresponding CrossSection and highlight it with a box in the view.

A double mouse click in either the “X-Coord”, “Y-Coord”, “BranchName”, “SectionType” or “RadiusType” columns will open a new view with the CrossSection being displayed. A double mouse click in the “Chainage” or “ChainageType” column will toggle a second panel. In this new panel the user is allowed to enter a new chainage value. The “Apply” button or a carriage return in the text line commits these changes and toggles the panel back to its original view. The chainage type is automatically updated to be “User Defined”. A second series of double clicks in these columns, on a row that is “User Defined”, toggles the chainage type back to “System Defined”. During this process, the measurements of the branch are redistributed from this location upriver and from this location down river, based upon the new value entered by the user.

7.3 The Cross-Section View

The Cross Section view(s) is a view displaying the surveyed location and elevations for a given point along the branch. In the view the user sees a polyline defining the Cross Section as well as multiple points showing the surveyed locations. The left most point, the right most point and the deepest point are coloured red which signifies them as having a Mark value of “1”, “3” or “2”, respectively. The user is allowed to display as many Cross Sections in this view as he wants. Only the “selected” Cross Section is coloured. All other “background” Cross Sections will be coloured a light grey. The user can use either

the “XSection Selection ” tool to select a Cross Section, or the “XSection Point Selecting” tool. This tool selects the CrossSection, highlights the point along the CrossSection with a box and if the CrossSection Attributes” dialog is open, the corresponding row is highlighted as well.

With the cursor in the CrossSection, a right mouse click displays a popup menu. Here the user will see a choice to display the “CrossSection Attributes” dialog. If the user clicks on this choice the “CrossSection Attributes” dialog opens to the left of the views. This dialog provides the user with an interactive view of the attributes associated to the selected CrossSection. In the top panel of the dialog, the user is shown the current Branch theme with the selected branch and the name of the selected branch. In the second panel is given a list of CrossSection view(s) and a list of all the CrossSections being displayed in the current CrossSection view. If more than one CrossSection view exists, from this list the user can select the current view. This then updates the next combo box with the list of CrossSections within the current view. The user can select the current CrossSection from here just has he could by using the “XSection Selection” tool. In the third panel, the user is given the values for SectionType and RadiusType attributes. These controls also allow the user to change these values at any time.

7.4 Editing Cross-Sections

The CrossSection can be edited in one of two ways. It can be edited interactively through the CrossSection view or through the attributes in the “CrossSection Attributes” dialog. To edit interactively in the view, the user simply needs to select the CrossSection to be edited and a click of the right mouse button displays a popup menu with a choice “Start Editing CrossSection”. This will set the editing environment for the selected CrossSection and will enable the “CrossSection Editing” tool. The user should then select this tool and then click the CrossSection to display the vertices. The vertices are displayed with a hollow box. At this time the user can “grab” any of the vertices and drag it to its new position. When the user is finished editing the CrossSection, he should again use the right mouse button to access the popup menu. This time he will see a choice “Stop Editing CrossSection”. After selecting this choice the user will be prompted for saving his edits. If the edits are committed, then the view is redrawn with all the other CrossSections plus this CrossSection and its edits. The values for the vertices in the list box will also be updated. If the edits are not committed then the view is redrawn as it was prior to starting the editing session.

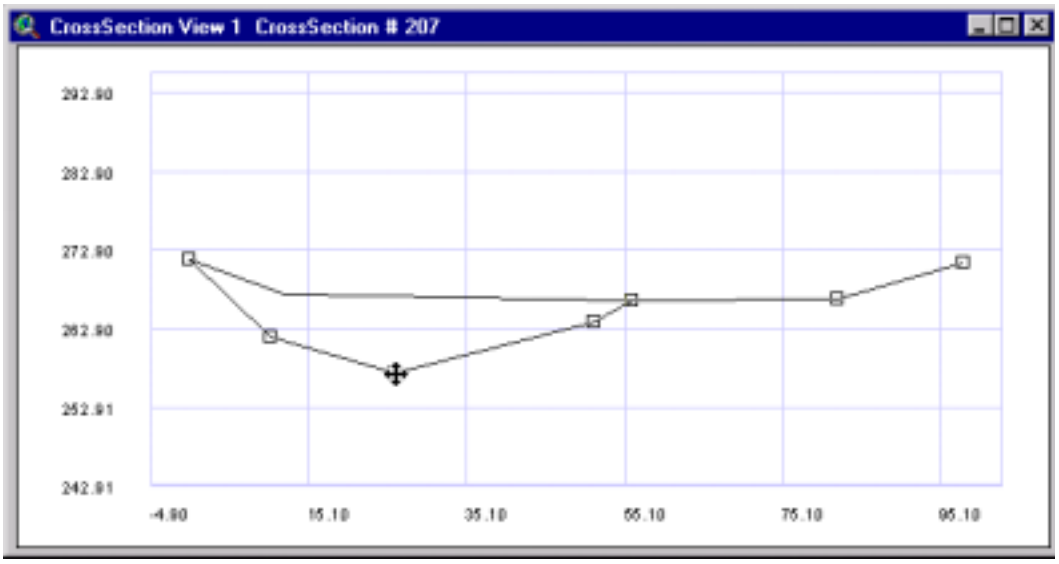


Figure 17 Cross section view

The second method of editing a CrossSection is by double clicking in the list box in the "CrossSection Attributes" dialog. The user can double click in either X-Coord column or Z-Coord column, and it is the column that determines the attribute to be edited. In either case a new popup dialog appears with a text line for the user to enter the new value. In other words, if the user double clicks in the Z-Coord column then a popup dialog appears and the user can enter a new Z-Coordinate value for the highlighted point in the CrossSection.

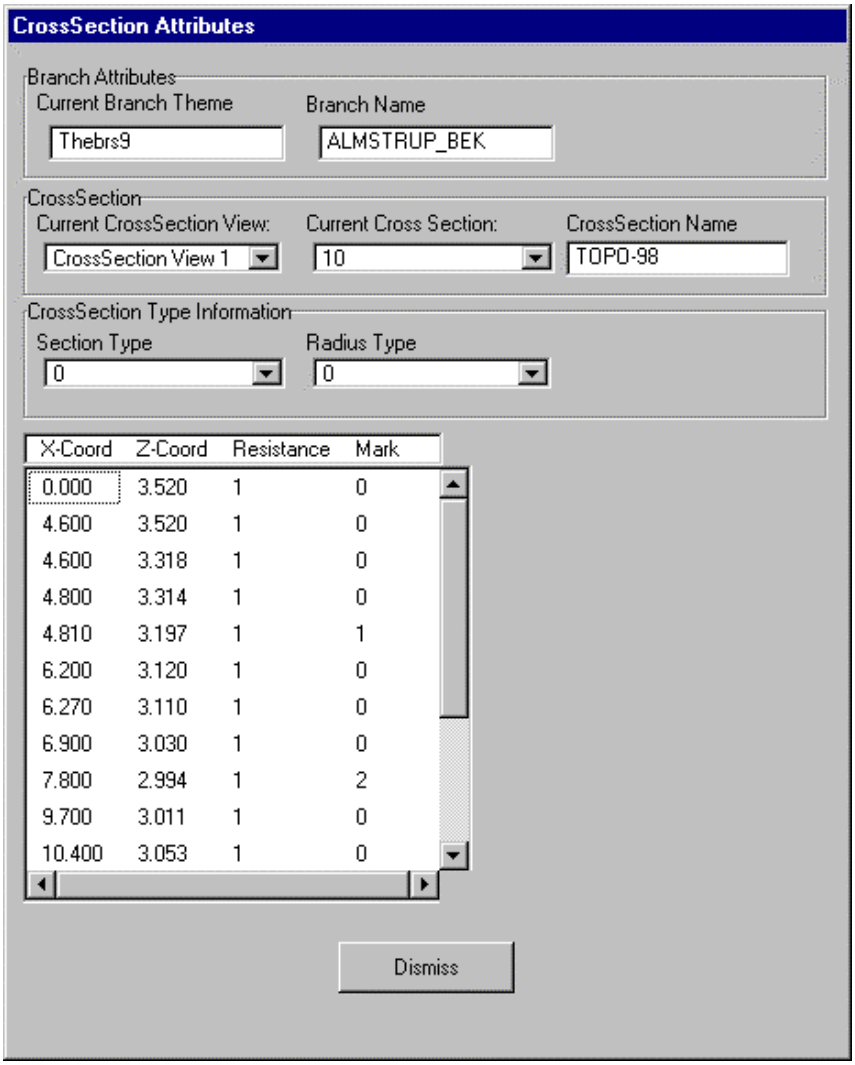


Figure 18 Cross Section Attributes dialog

APPENDIX 1

File format

T2 FILE

The T2 file is an ASCII file used for matrix data by MIKE SHE.

```
FILETYPE DATATYPE VERNO:  22 57 530
NX NY DIM XORIG YORIG  :  110 110  5.0000000E+001 -2.8347200E+005  5.2711996E+004
DELETE UTMZONE ORIENT  :  -1E-035 0 0.000000
MIN MAX MEAN ST.DEV    :  1.803138E+001 2.809112E+001 2.123936E+001 1.703041E+000
Theme: grid3
     110
        20.0172        20.04639        20.03877        20.03072     20.02321
       20.01772        20.01595         20.0356        20.05353        20.08295
```

Figure 19 The format of the T2 file

The header line (1-5) specifies the data type in the file. Each line in the header consists of 24 characters of text (which will be skipped when reading) followed by information;

Line 1: Specifies the data file type, the data type and the version number (530 for MIKE SHE version 5.3).

File type: 21- the data in the data file is considered to be integers (grid codes) even though the data is written as reals.

22-the data in the data file is considered to be reals even though the data is written as integers.

Data type: the data type is presently not used but the only allowable type is:
57: Any grid data.

Line 2: Specifies the dimensions and origin of the data;
nx, ny: dimensions of the matrix.
dim: dimension of the grid squares.
xorig, yorig: location of the origin in the coordinate system.

Line 3: Defines a delete value that defines areas with missing values or outside the catchment, the UTM zone (specify a zero if unused) and the orientation of the matrix data.
Line 4: Calculated statistical parameters for the data values.

Line 5: text line.

Grid definition file

The grid definition file is an ASCII file used to save a grid set-up. The format of the grid definition file is used for all the MIKE SHE – GIS extensions with respect to grid definition.

```
Grid Properties for DHI GIS
Grid properties Tinglev Mose 50 meter
xorg. -283471.9981
                    yorg. 52711.9960
Columns 110
Rows 110
Cellsize 50
```

Figure 20 The grid definition file

Line 1: This line identifies the file as a grid definition file. In older versions this line could be “Grid Properties for MIKE SHE GIS”.
Line 2: User comments.
Line 3: The x-coordinate for the lower left corner.
Line 4: The y-coordinate for the lower left corner.
Line 5: The number of columns.
Line 6: The number of rows.
Line 7: The dimension of the grid cells (squares).